Numerical Investigation for the Distribution of Air Velocity and Temperature in an Occupied Zone by using Displacement Ventilation

Dr.Israa Ali Abdulghafor1* [1]Institute of Technology, Middle Technical University, Iraq

ABSTRACT

The current investigation presents a numerical analysis to distribute the temperatures and velocities in an occupied zone under different provided air temperatures and velocities. The steady-state and incompressible flow conditions are assumed by CFD simulation. The provided air velocities are investigated of (0.75,1,1.25and1.45)m/s with 1600 was a fixed internal load and a fixed provided air temperature of 18 °C. The provided air temperatures are investigated at (18,20,22 and 24) °C with 1600W as a fixed internal load and fixed provided -air velocity of 0.75m/s. The results show the best velocities and temperatures distribution at the provided -air velocity of 0.75m/s and air temperature of 18°C. While best rate air temperature for an occupied zone is achieved at provided-air velocity and temperature of (1.45m\s and 18 °C) respectively. The air distribution becomes more turbulence as provided air velocity and temperature increase.

Keywords: Numerical investigation; Displacement ventilation; Velocity distribution; Tempertaure Disrbution ; Fluent 14.5

1. Introduction (11 pt, Sentence case)

The operation of displacement ventilation (DV) is based on the variation in an air density , the room air is divided for two stratum, a unclean top region, and a clean bottom region, this is done due to supply cold air at low velocity in the bottom region and drawing a hot air from the top region. By free convection from a heat source, a vertical air motion in the room is produced . Due to free convection, the hot unclean air rises to the top region . The flow rate of convection relative to the flow rate of ventialion calculate the altitude of the boundary between the two region. The flow rate of ventilaion in the room is equal to the sum of hot flow rates of convection at the top region minus downward direction moves from cool surfaces to the bottom region , moves the boundary upwards by increased flow rate of ventilation and moves the boundary downwards when decreased flow rate at constant flow rates of convection [1]. During recent decades, many studies have been achieved to enhance the performance of DV systems and facilitate the design steps of DV systems. Also, enhancement of thermal comfort conditions via using a DV system has been studied. The performance of the displacement ventilation system in an office in terms of thermal energy using the simulations model is evaluated by Hensen and Hamelinck. (1995),[2] the simulations are achieved on a typical office module located in the Netherlands. The casual gain with simulation is 30,35,40 and 50 W/m2. The results show that energy savings of up to 14% when casual gains are 30W/m2 during summer months while at casual gains higher than 30W/m2 displacement ventilation benefits are hidden. The displacement ventilation needs to combine with the chilled ceiling at casual gains of more than 35 W/m2. Kobayashi and Chen. (2003),[3] produced a validated CFD program to study the floor displacement ventilation system behavior with various furniture regulation , internal load ,and air atlrate rates and.CFD program is validated by experimental data. The experimental study was achieved by an environmental chamber well insulated (R = 5 K m2/W), the chamber was divided into the test chamber and environmental chamber by using movable wall. The results show from a validated program that the rate of air alterate must be chosen carfully to prevent a high (Percentage of dissatisfied people due to draft) PD distribution. The thermal condition and energy saving of building when using displacement ventilation was studied by Madureira. (2008),[4] using EP (Energy Plus) simulation with a simple model for 400m2 area and 4m in high with 32m2 window area chosen as an office building. A comparison of the results obtained from ES with the results obtained from CFD. Also compared the performance of displacement ventilation and mixing ventilation using a real model located in Lisbon . The results show that the difference when comparing ES and CFD simulations results was 5% considered an acceptable limit. 80% of energy-saving when using DV compared with using equivalent MV. DV consider more suitable with a simple model. The real building model was built depending on the real building information. Chitaru et al. (2018),[5] presented a numerical study for comparison of displacement ventilation with a mixed flow ventilation approach. Various aspects of the two systems as the distribution of air temperatures and velocities in the occupied region , and the effect of a draft in defined places. Overcomes showed that the displacement ventilation system appeared features by comparison with mixed ventilation, this is because the buoyancy-driven flows was rised produced due to the indoor heat behooves. A draft impact , to case of the displacement ventilation, was low slop for exiting in the occupied region however the emissions of heat from the indoor behooves have raised the temperature gradient in the occupied region. Andersson et al. (2022),[6] presented numerical and experimental investigated for confluent jets ventilation (CJV) apparatus has three various nozzle arrangements (1 × 19, 2 × 19, 3 × 19) at two various airflows and temperature of the supplied air. The performance of the CJV supply apparatus was studied focused on the thermal comfort conditions, heat removal effectiveness, and, indoor air quality (IAQ) in convention hall . Between an experimental and a numerical resuts a comparison was made and the overcomes appeared the θ^(2^-) -f sample possessed the good correspond to the studied samples of turbulence . A numerical overcomes appeared that the arrangement size possessed the most impact on both the occupied region conditions and near-field produced. In a major arrangement consisted of many rows and less momentum conserved, the mean age and temperature of the air were better than in a single-row arrangement that had more momentum. A major arrangement consisted of many rows had a more IAQ and a more heat removal efficiency in the occupied rgion. The aim of this study, analysis numerically the distribution of air temperatures and velocities in the occupied zone under various conditions by using certain designs of displacement ventilation.

2. Numerical Analysis

In this study, the CFD is used to simulate three-dimensional airflow and temperatures in the occupied zone by using DV. The principle of simulation is to solve the mechanical equations which depend on foundational conservation laws of physics that expressed in form of nonlinear partial differential equations. These equations are solved numerically using the CFD model and produce domain values to the velocity, and temperature [7]. In the present work, the model of K-ε turbulence was used because it is simple to run , causes to the stable computation, that converge relatively easily, and accountable predictions to many flows [8].In following the basic steps for a numerical analysis:

2.1.Geometry of Tested Room

Figure 1 shows the geometry used in the present work, drawn by using Gambit 2.4.6 [9]. The room surface and other accessories of the room were built by Gambit 2.4.6 and then a three-dimensional solid model was generated. Displacement ventilation (DV) was studied with different provide air velocities and different provide air temperatures with a constant internal load and assuming an insulation room. The DV system is used consists of two grills, one for the inlet of supplied air and the other one for the outlet of the return air, the inlet grill locates at floor level with a diameter of 31 cm and the outlet grill locates at 260 cm height from floor with 31 cm in diameter. This DV system is installed in a room with dimensions of (300×400×270)cm in width, length, and height respectively. To produce the wanted cooling load, two heaters are used with a circular shape and each heater produces 800W.

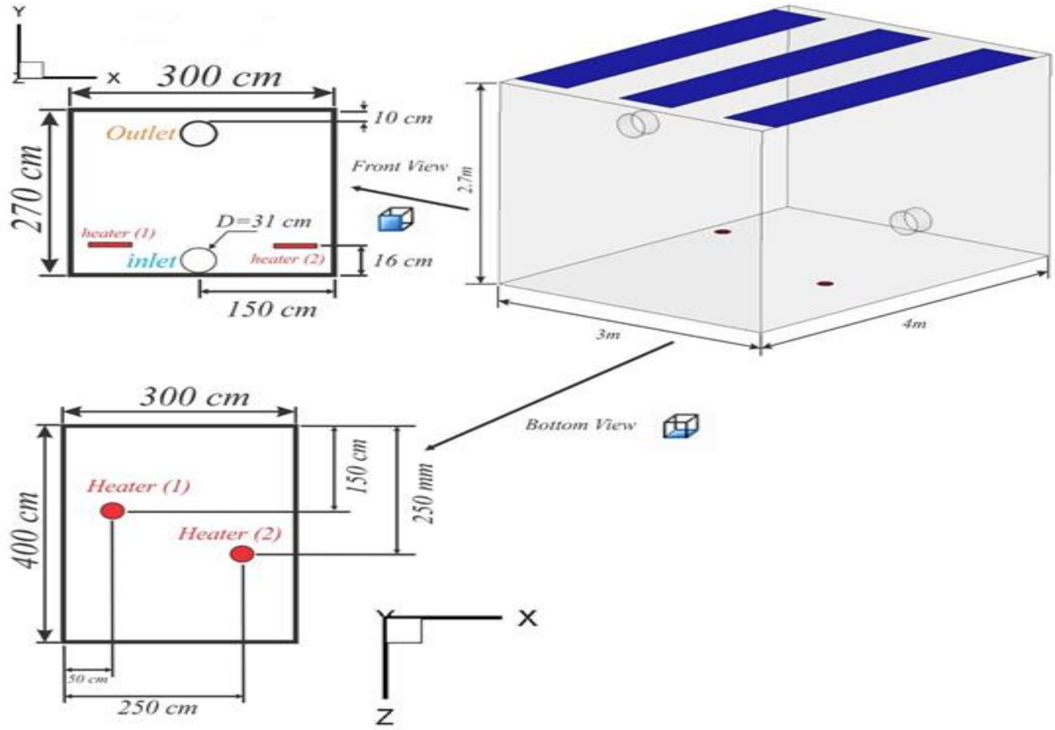

Figure 1.Geometry of tested room

2.2.Mesh Generation

The mesh generation is an important step to fit the boundaries of the computational domain required by standard CFD methods. Also solution of three-dimensional Navier-Stokes, energy, and continuity equations was described by mesh. This kind of problem covers a wide range of engineering applications. The three-dimensional mesh generation included surface mesh generation and volume mesh generation. For surface mesh generation, the two-dimensional surface mesh was generated. The element shape used was Triangular with Pave shape type because the geometry is very complex have different shapes circular for the heater and rectangular for the room. For volume mesh generation, volume mesh was generated in the building. Near the heaters, the fine mesh was used while coarse mesh was used as the distance from the surface increased. Telrahydral was the element shape used also the type of mesh was T-grid. Figure 2 shows the meshed models and megascopic section from them. Figure 3 show the element type used in surface and volume meshing in this work.

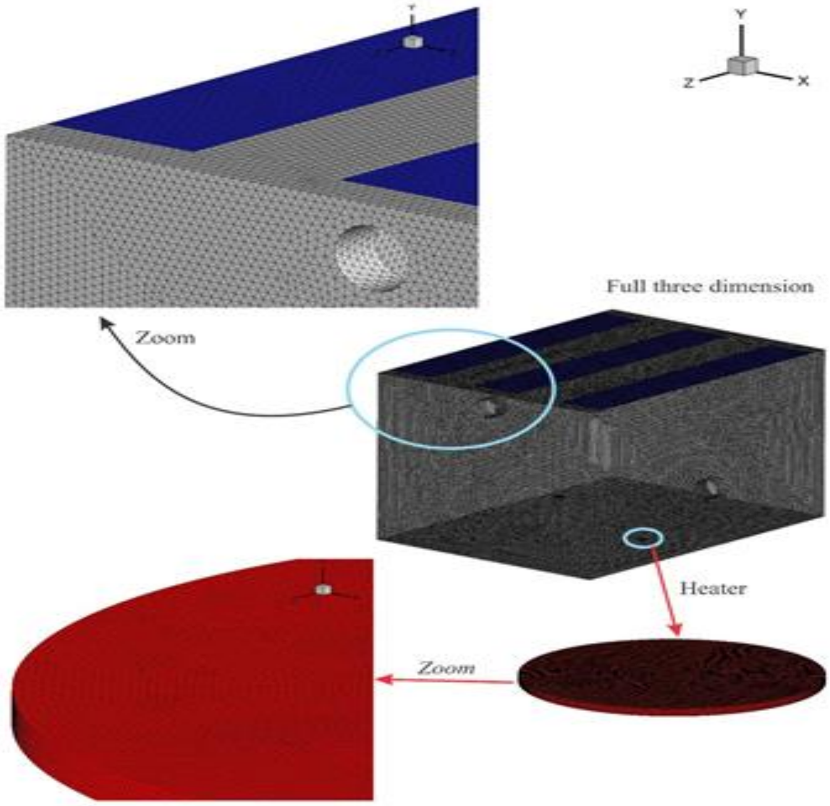

Figure 2. The mesh models and megascopic section for the displacement ventilation

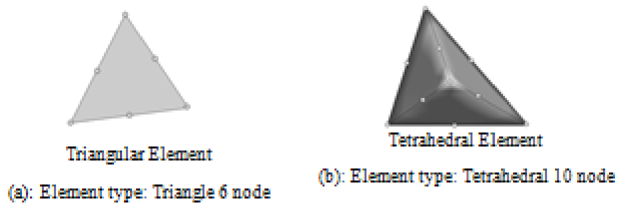

Figure 3. The element type

2.3 .Solution

By using ANSYS FLUENT 14.5 **[10]**. The solution of three-dimensional Navier-Stokes, energy, and continuity equations was solved. The steps of the solution can show in Figure 4.

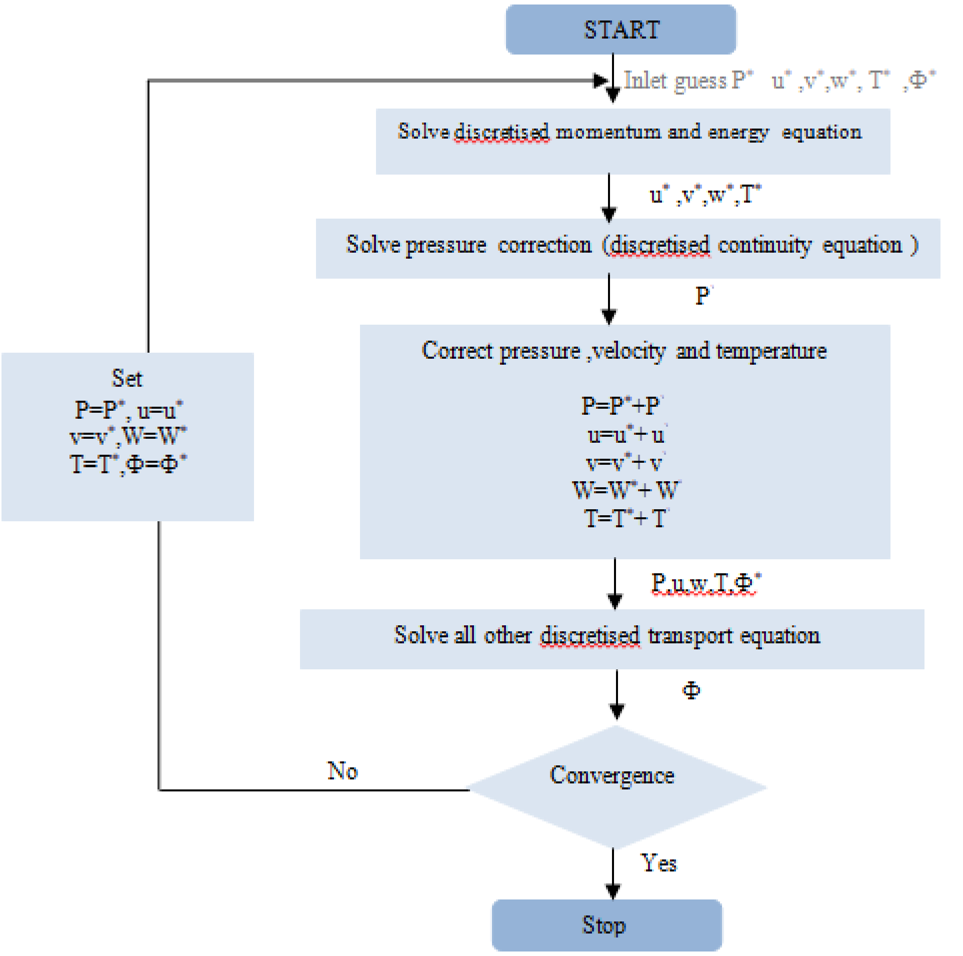

Figure 4. The steps of equations solution

2.4. Numerical Cases Setup in Ansys Fluent 14.5

2.4.1. General Setup

- Pressure –Based solver.
- Absolute velocity formulation.
- Steady-state for time consideration.

2.4.2. Materials Properties

In properties, the item required the input properties of the working fluid, in the present work is air, and also properties of materials that are used in the construction model. The properties of air and materials can be seen in Table 1, and Table 2 respectively.

Table (1): Air Properties

Properties	Unite	Value
Density (ρ)	kg/m^3	1.225
Viscosity (μ)	Kg/m.s	$1.7894x10^{-5}$
Conductivity (k)	W/m.K	0.0242
Specific Heat (C_p)	J/kg.K	1006.43
Thermal Expansion coefficient (β)	/K1	$.47x10^{-3}3$

Table 2. Properties of Material

Part	Type of Material	Density (ρ) Kg/m^3	Specific Heat (C_p) J/kg. K	Conductivity(k) W/m.K
Ceiling and Other side of room insulator	Glass-Wool	64	670	0.04

2.5. Boundary Conditions:

To get an accurate solution must be input boundary conditions for the model room. Figure 5 can see these conditions for displacement ventilation and Table 3 shows the conditions used

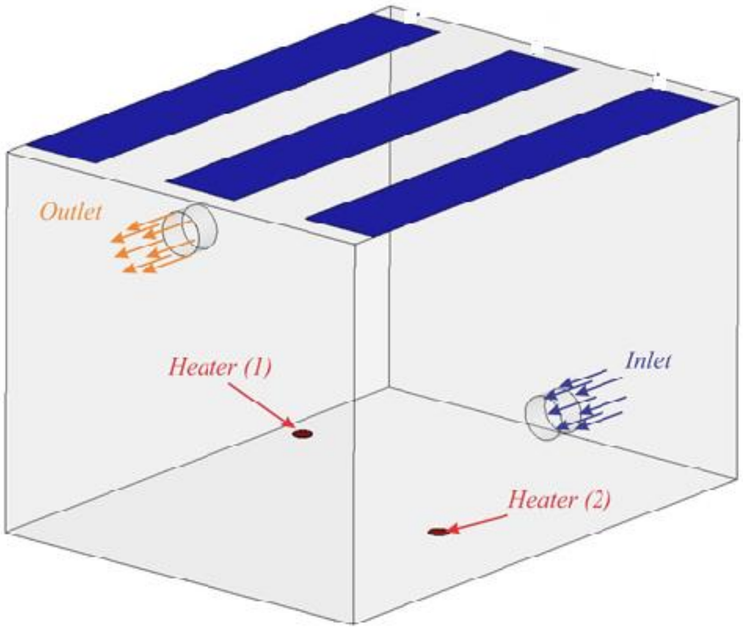

Figure5. The boundary conditions used for DV

Table3. Boundary Conditions for Cases studied

Number of case	Velocity of supplied air (m/s)	Temperture of supplied air (°C)
1	0.75	18
2	1	18
3	1.25	18
4	1.45	18
5	0.75	18

6	0.75	20
7	0.75	22
8	0.75	24

2.6. Solution Methods and Solution Controls

The SIMPLE algorithm with a Second-Order Upwind scheme for energy and momentum equations was utilized for solving the coupling of pressure –velocity. The control of change of primary variable is necessary, this is because the nonlinearity equation groupand coupling of pressure –velocity to be solved by FLUENT14.5.This is fulfilled by under-relaxation, which diminishes the variations of the primary variable produced during iterations. Table 4can see the value of under–relaxation factors applied in the present work to solve momentum equations and other scalar quantities

Table 4.Show the Under -Relaxation Factor

Property	Under –Relaxation Factor
Pressure	0.3
Density	1
Body Force	1
Momentum	0.4
Turbulent Kinetic Energy	0.8
Turbulent Viscosity	1
Turbulent Dissipation Rate	0.8
Energy	1

5. Vaildation of The Present Study

The current results of the CFD model for predicting the average room air temperature under different supplied-air velocities and temperatures are validated by the experimental data in our previous study (AbdulGhafor et al, 2018) **[11]**. Figure 6 and Figure7 show the comparison between the overcomes of this study and our overcomes of experimental previous study. Figure 6 and Figure 7 appear good agreement between the present results and the previous study and the error for both figures are not exceed 6%.

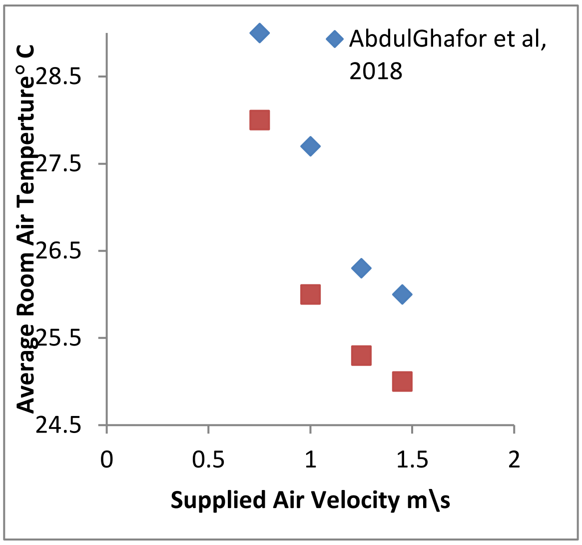

Figure 6.The validation of present work for average room temperature and supplied air velocity

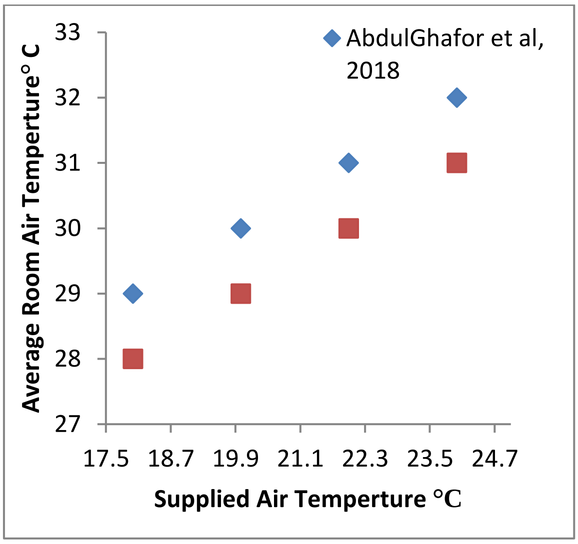

Figure7.The validation of present work for average room temperature and supplied air velocity

4. Results and Discussion

Figures from 8 to 11 show the vectors of velocity at e (X-Z) plane at various places of Y(10,110,170 and 265)cm for supplied air velocity of (0.75,1,1.25 and 1. 45)m/s respectively. From figure 8 can be concluded that for Y is 10cm, the velocity increases in some areas posited above and alignment the heat sources also the velocity increases near supplied air grill outlet. For Y is 110cm, the heat is more distributed to the remaining layers and the velocity increases compared with Y of 10cm and resulting in the turbulent flow and vortices being noticed clearly. For Y is 170cm, the air velocity increase, and vortices appear. At Y is 265 cm can be concluded that vortices increase because an increase the velocity of air with elevation, the warm air is accumulated near the wall to escape by the opening of the return grill. The same manner can be seen in figures9, 10, and 11. By comparing figures 8 to 11 for the various supplied-air velocity of (0.75,1,1.25 and 1.45) m/s can be noticed that as the velocity of supplied air increases, the flow becomes more turbulent, and the average room air temperature becomes less and reaches about 25 °C for supplied air velocity of 1.45 m/s. Figures 12 to15 show the cotours of temperature at (X-Z) planes for various y posited of Y(10,110,170 and 265)cm for each velocity of supplied air. From figures 12 to 15 can be concluded that generally speaking the temperature increases with height due to heat exchange. At Y is 10cm, it's appeare that the coldest air is accumulated near the floor level since the cold air is heavier. At Y is 110cm, the temperature increases more than that at Y of 10cm due to the effect of the heaters. When Y is 170cm, the temperature of air increases, and the highest temperature can be seen at Y is 265cm where the air becomes in contact with the ceiling.

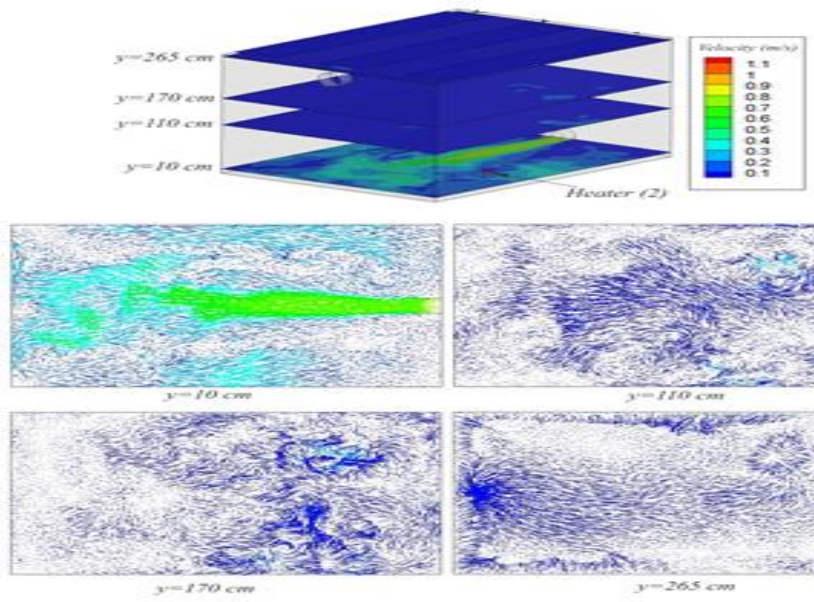

Figure8.Velocity vector at different Y for velocity of supplied of 0.75(m\s)

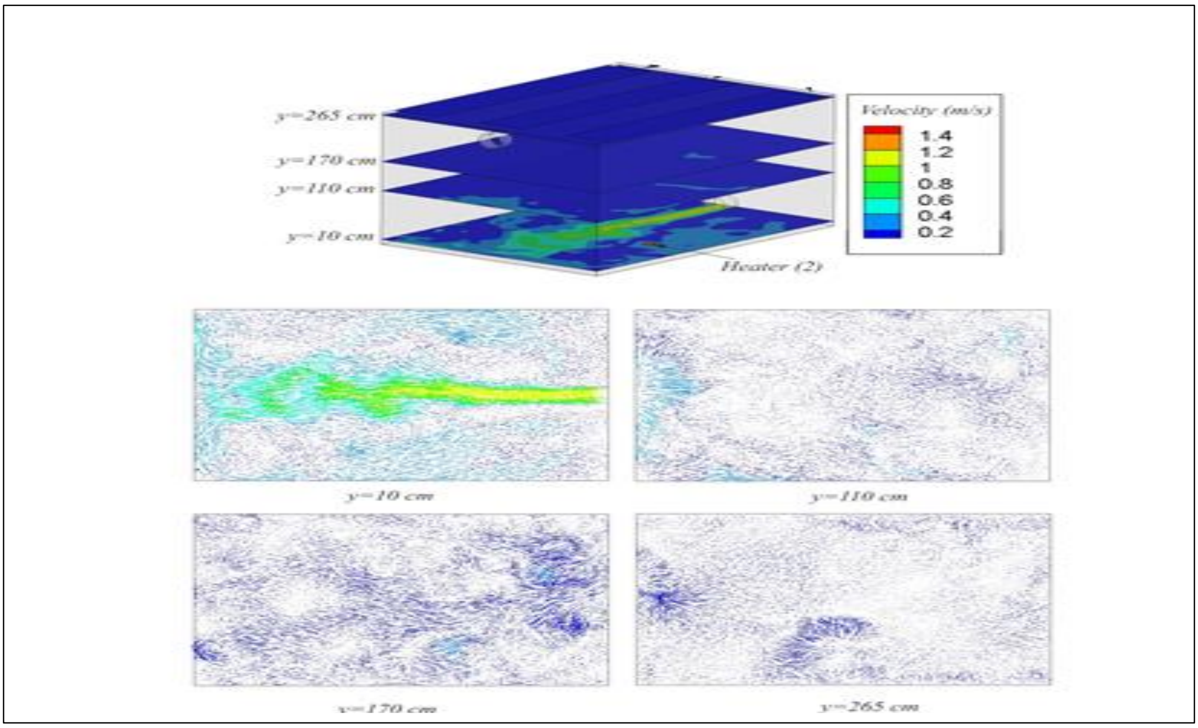

Figure9.Velocity vector at different Y for velocity of supplied air of 1 (m\s)

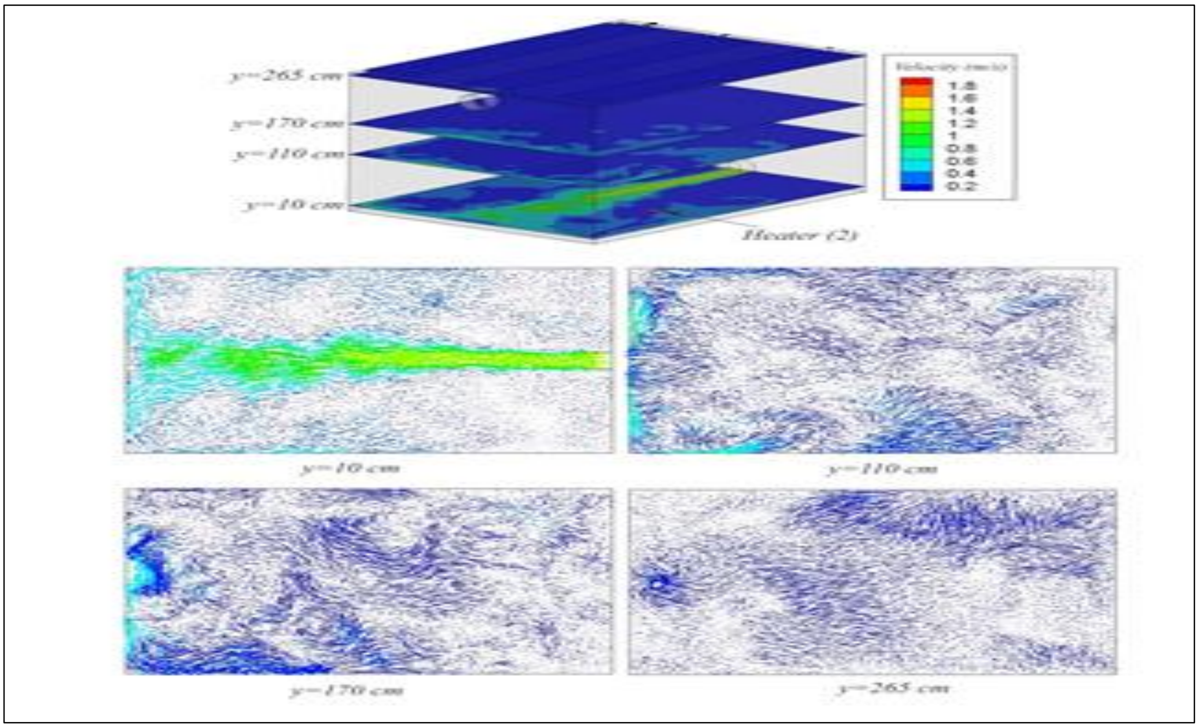

Figure10.Velocity vector at different Y for velocity of supplied air of 1.25(m\s)

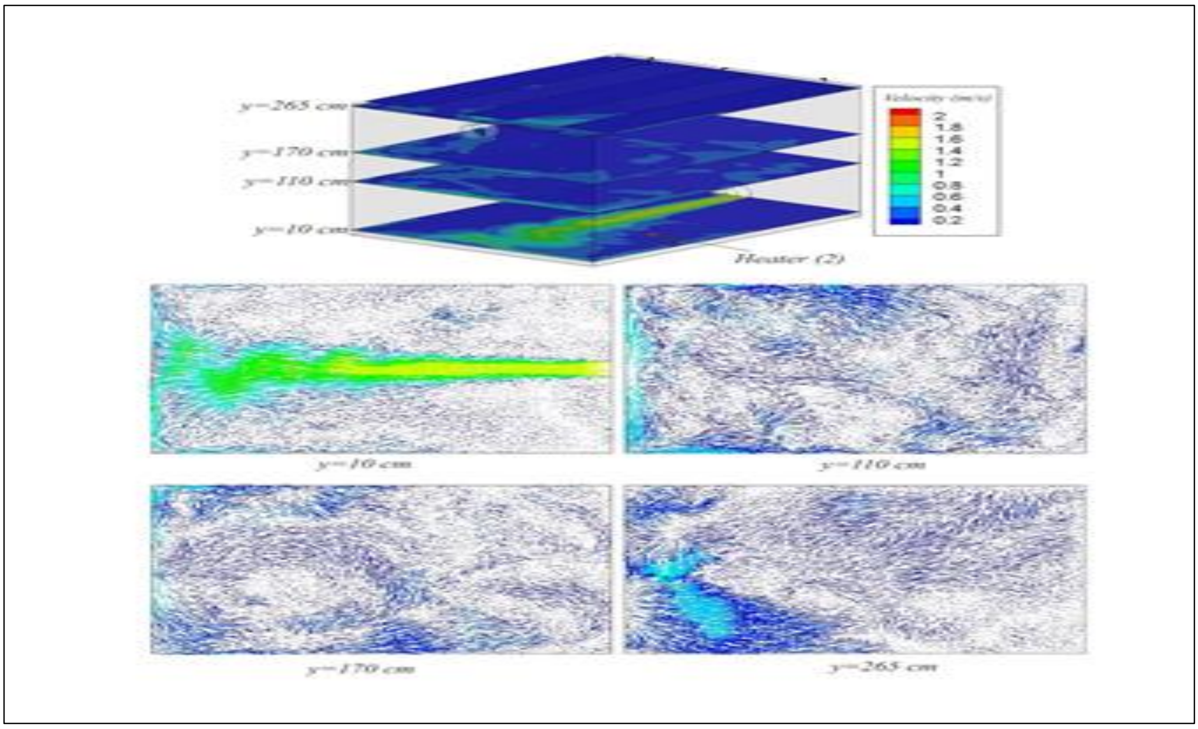

Figure11.Velocity vector at different Y for velocity of supplied air of 1.45(m\s)

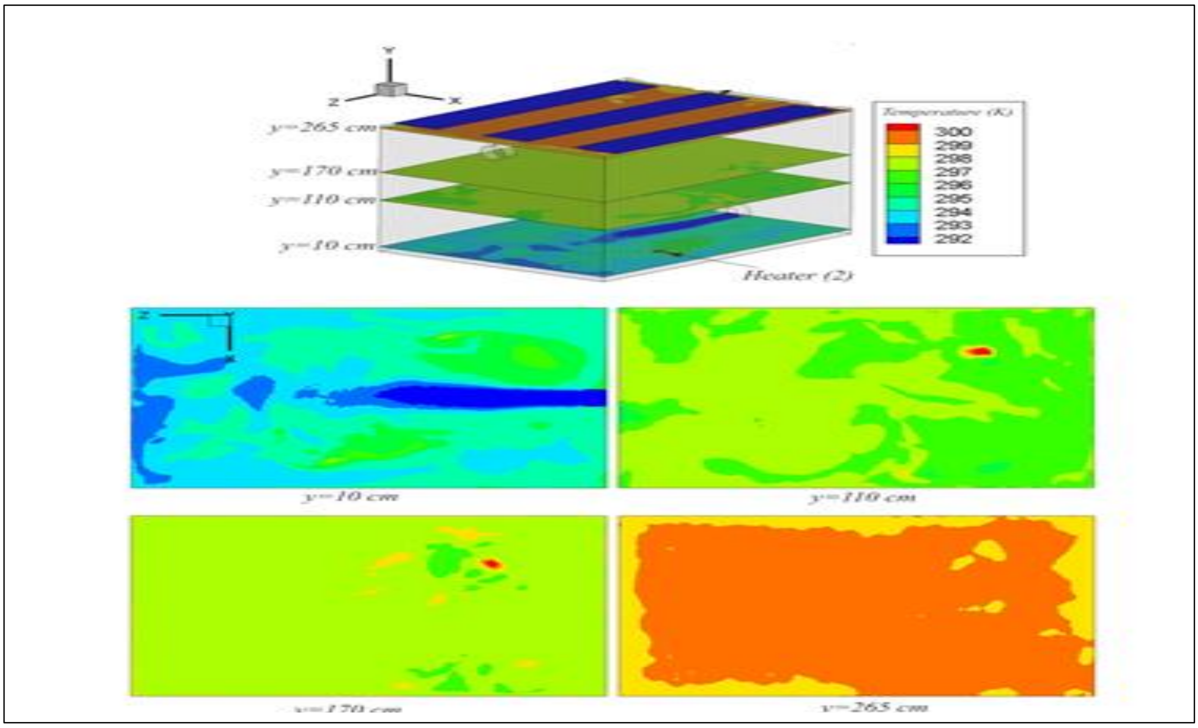

Figure12.Temperature contour at different Y for velocity of supplied air of 0.75(m\s)

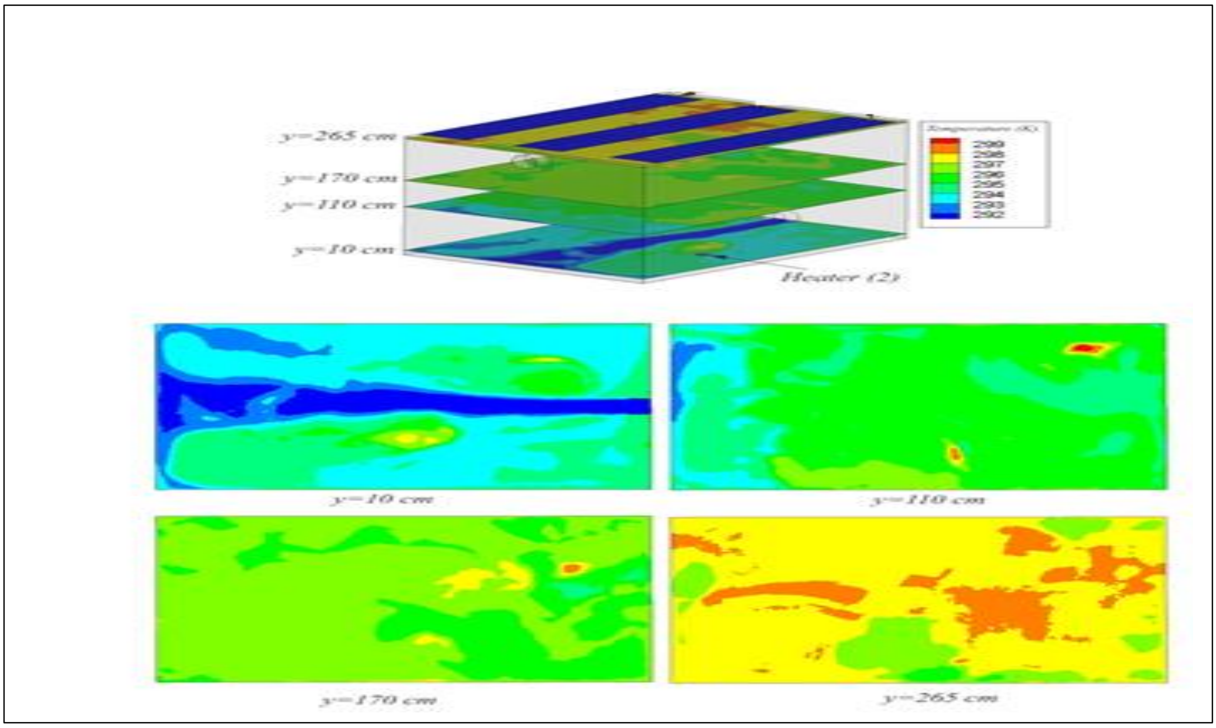

Figure13.Temperature contour at different Y for velocityof supplied air of 1 (m\s)

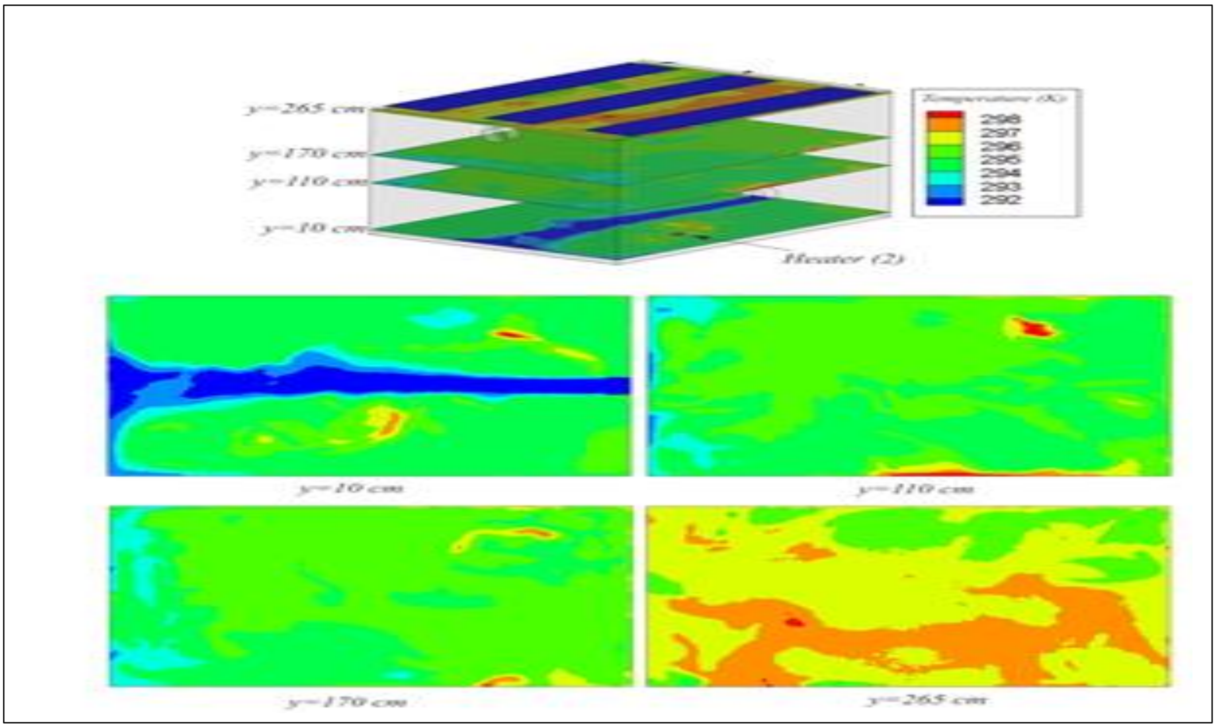

Figure14.Temperature contour at different Y for velocity of supplied air of 1.25(m\s)

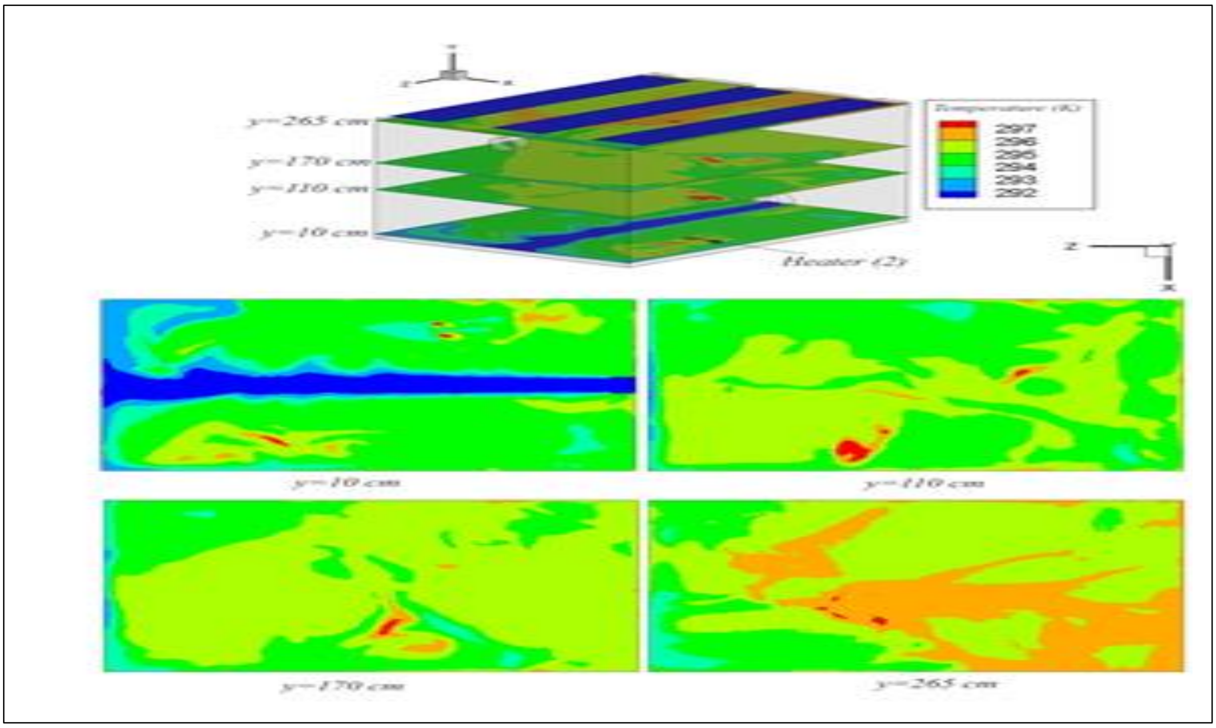

Figure15.Temperature contour at different Y for velocity of supplied air of 1.45(m\s)

Figures 16 to 19 describe the vectors of velocity at the (X-Z) plane at various y of (10, 110, 170, and 265) cm for each temperature of supplied air of (18,20,22and24)°C. A fixed supplied-air velocity of 0.75m/s and an internal load of 1600W with steady stay condition are assumed . From the above-mentioned figures can be concluded that the velocity vector has a similar behavior as explained in figures8 to 11. Comparing are made from figures 16 to 19 between tempertures of supplied-air are (18, 20,22, and 24) °C for the plane (X-Z) and be can be noticed that as the temperature of supplied air increases, the cooling capacity of displacement ventilation decreases so that the air room becomes warmer with less density. It's clear that at a temperature of supplied air of 24°C the air room has less density than that at the temperature of supplied air of 18 °C and as a result the room air velocity increases and that leads to an increase in Re so the turbulent flow and vortices can be noticed clearly.

Figures 20 to23 show the temperature contours at the (X-Z) plane at different y of (10,110,170, and 265)cm for different temperatures of supplied air. From figure (20) can be concluded that the room air temperature increases with height and air become lighter so that the air velocity increases with height. The same manner can be noticed in figures 21 , 22, and 23. the comparison is made from the above-mentioned figures between supplied-air temperatures of (18, 20,22, and 24) °C for the plane (X-Z) can be noticed that the temperature of supplied air increases the cooling capacity of air decreases so it's clear that at a temperature of supplied air of 18°C the temperatures of air room are less than a temperature of supply air of 20,22, and 24°C. Also can be noticed that the amount of warm air near the ceiling zone at the temperature of supplied air is 18°C is less than that at temperature air of 20,22, and 24 °C. With a plane of (X-Z) for each temperature of supplied air (18,20,22, and 24)°C, it's clear that the air temperature of the room increases with the room height. The temperature in the occupied zone does not exceed (28, 29,30, and 31)°C for the temperature of supplied air of (18,20,22, and 24)°C respectively. This is because that air capacity decreases as the temperature of supplied air increases.

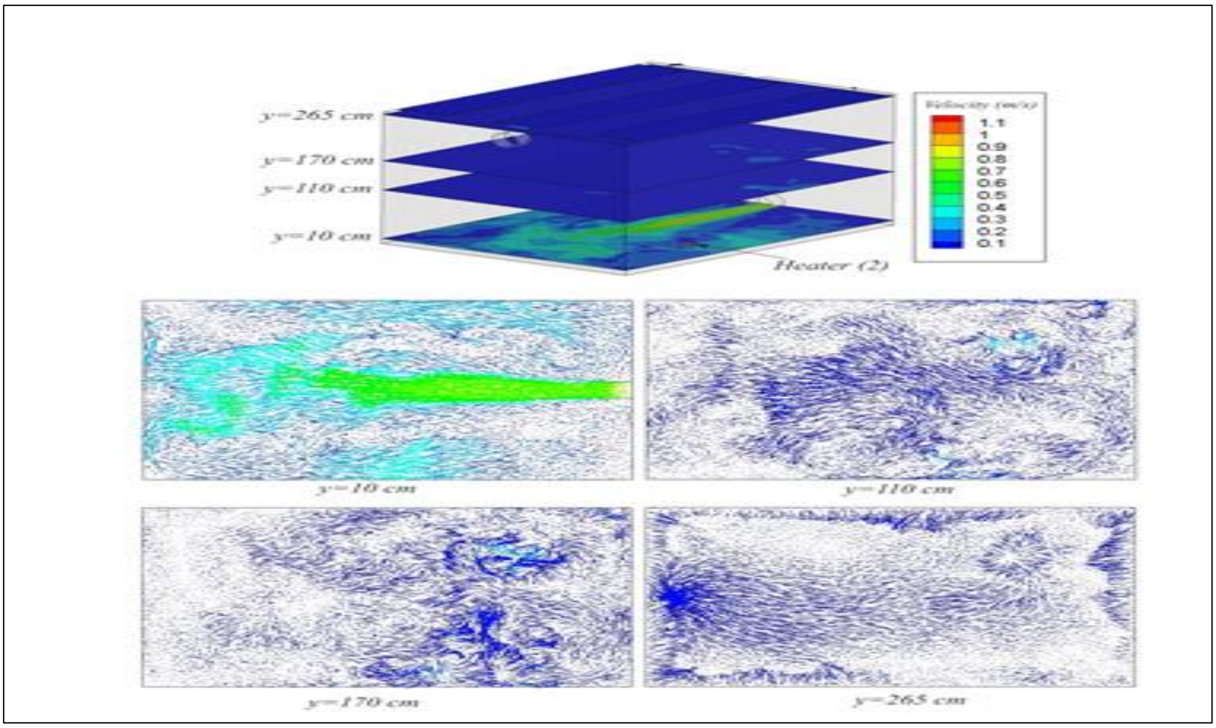

Figure16.Velocity vector at different Y for supplied temperature of air is 18(°C)

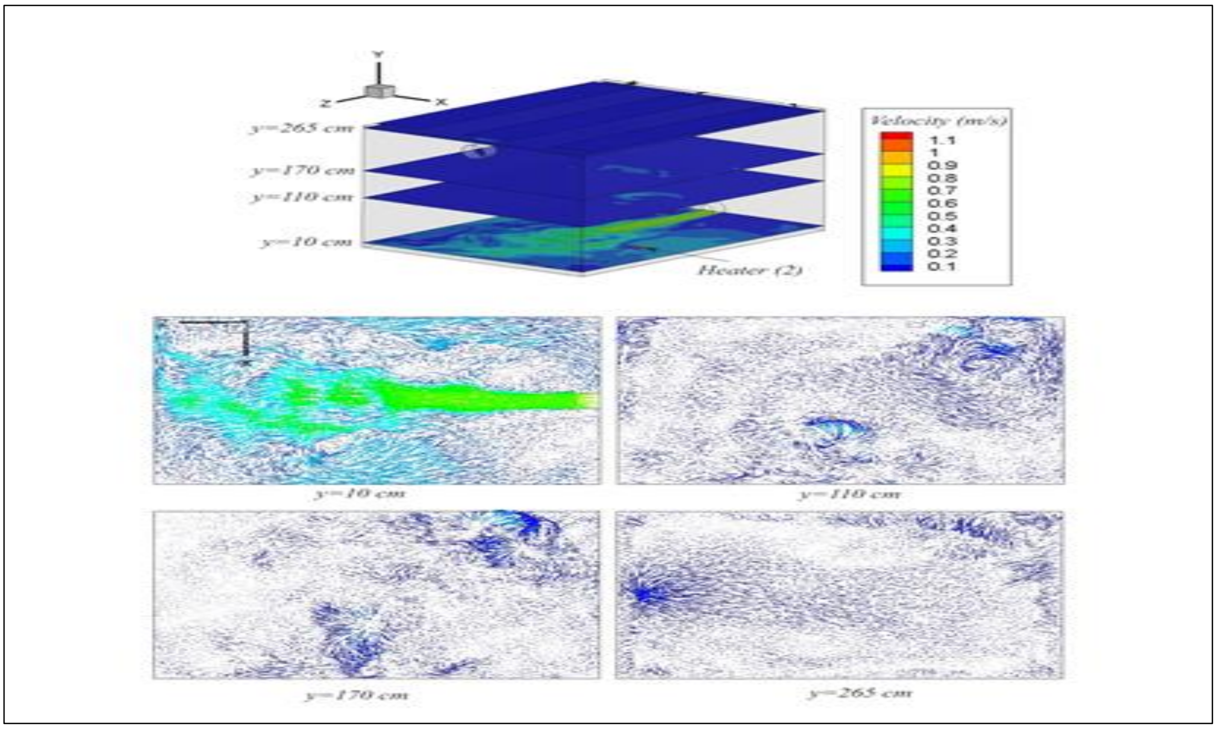

Figure17.Velocity vector at different Y for supplied temperature of air of 20(°C)

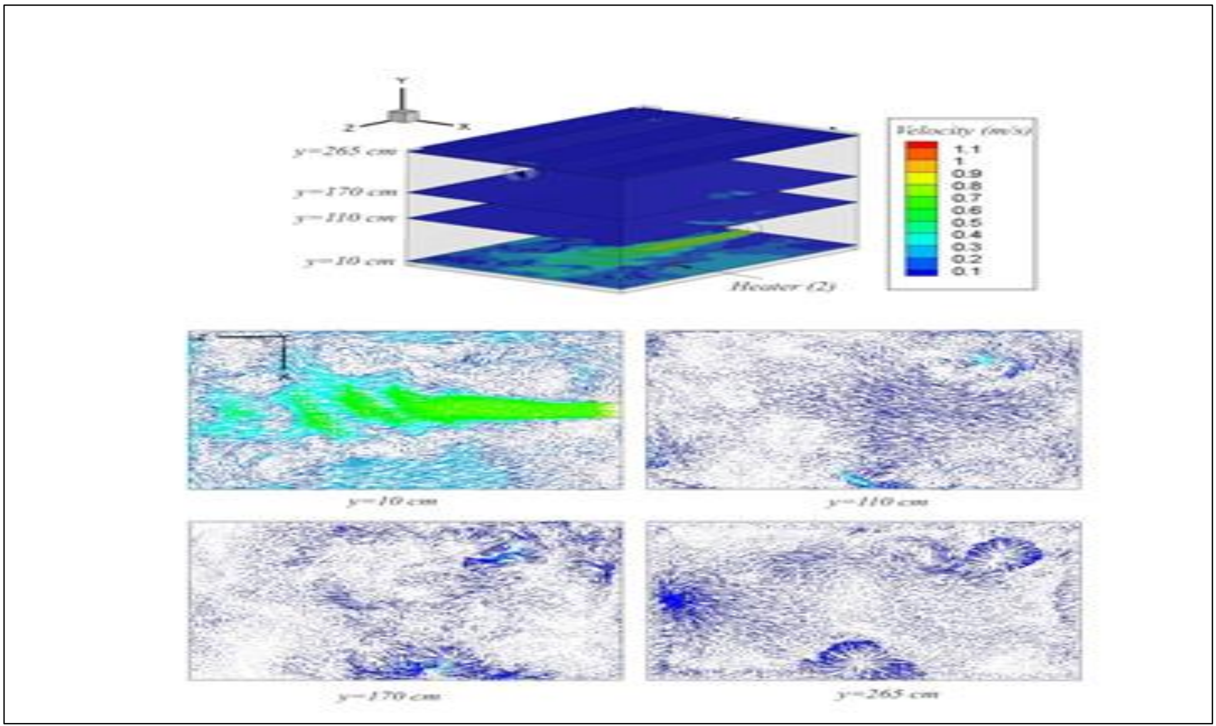

Figure18.Velocity vector at different Y for supplied temperature of air of 22(°C)

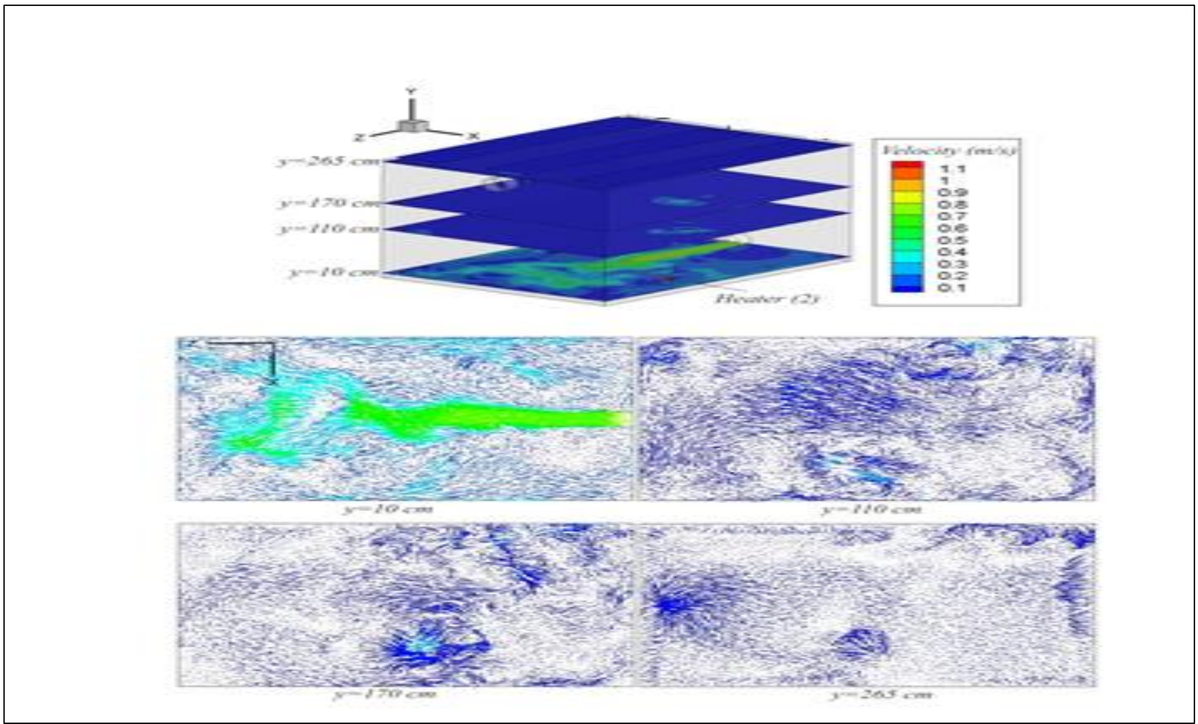

Figure19.Velocity vector at different Y for supplied temperature of air of 24(°C)

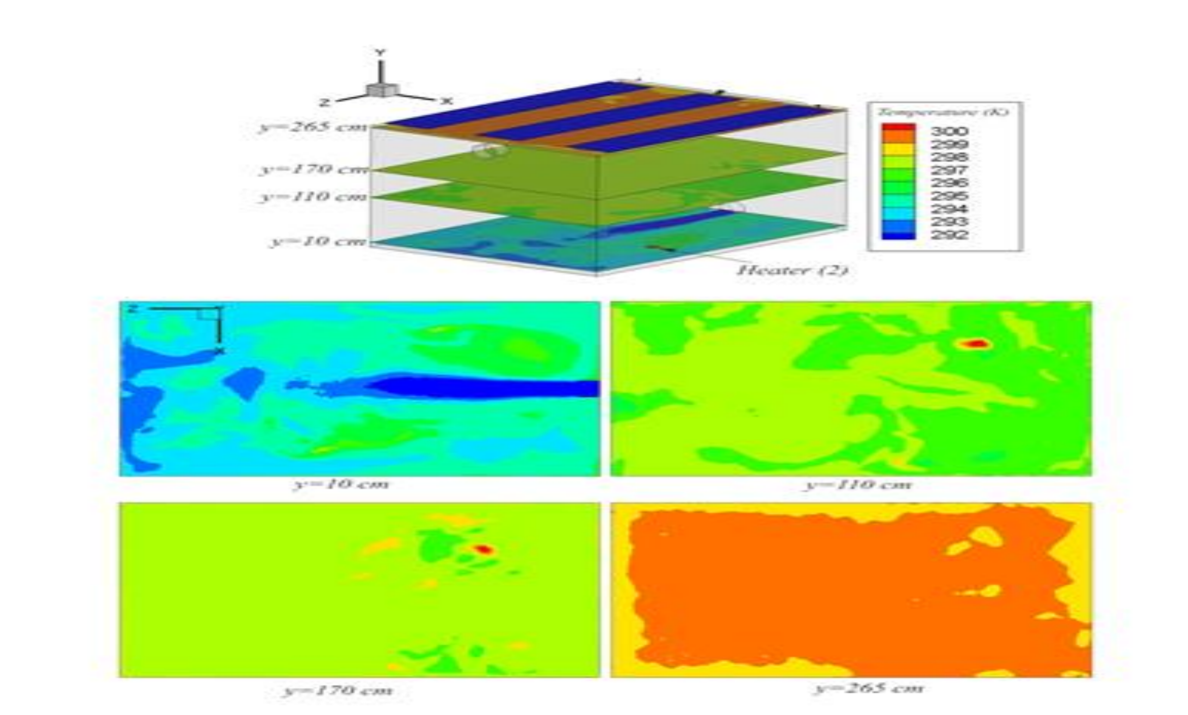

Figure20. Temperature contour at different Y for supplied temperature of air of 18(°C)

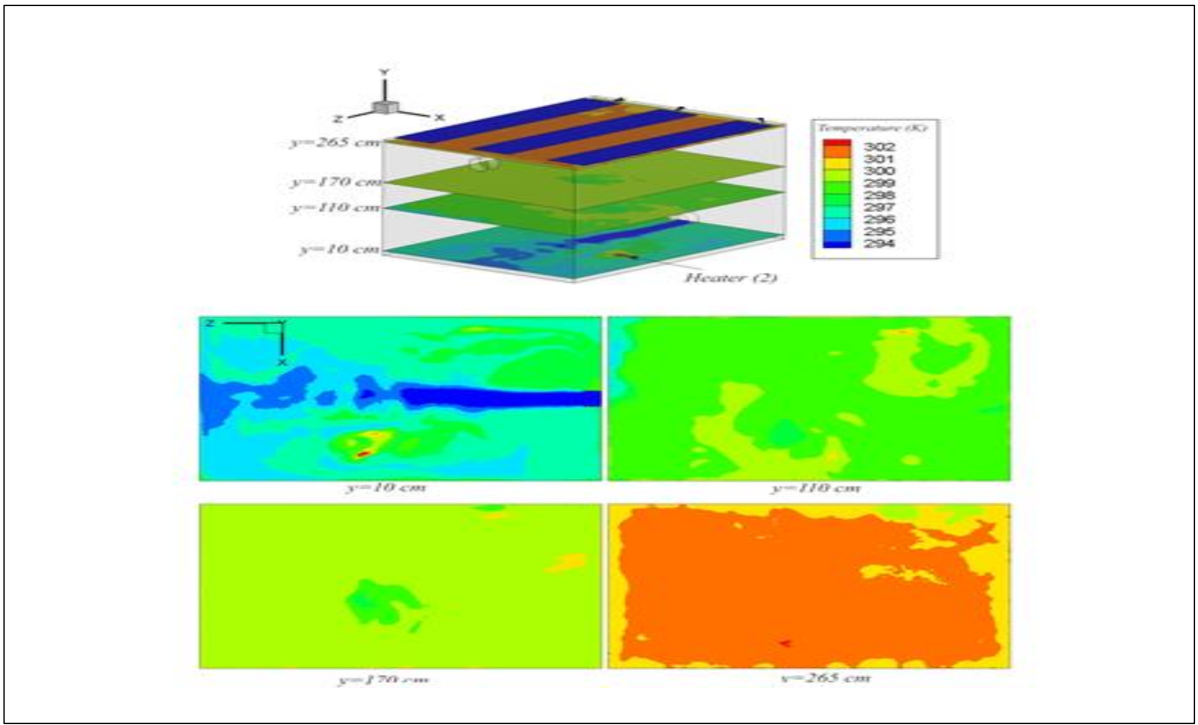

Figure21.Temperature contour at different Y for supplied temperature of air of 20(°C)

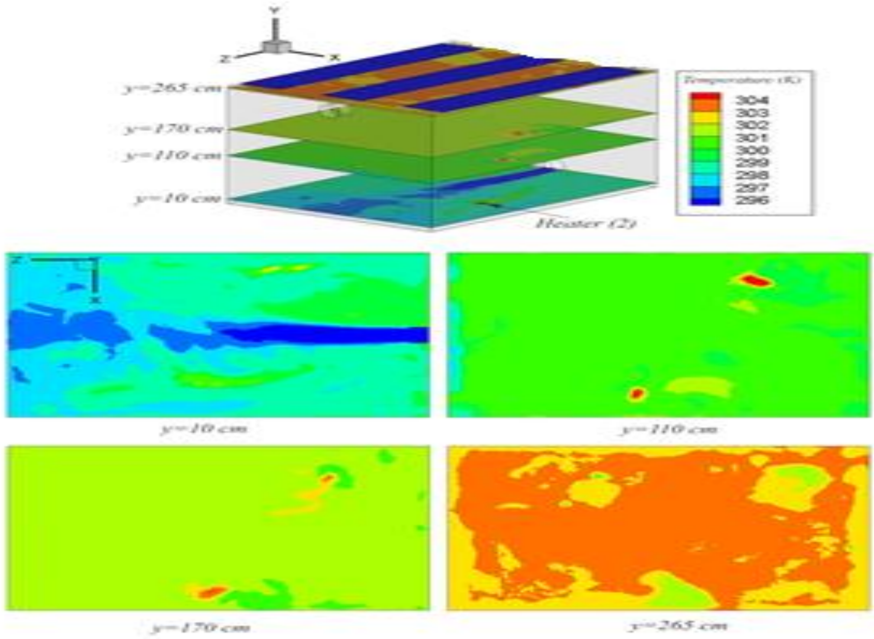

Figure22.Temperature contour at different Y for supplied temperature of air of 22(°C)

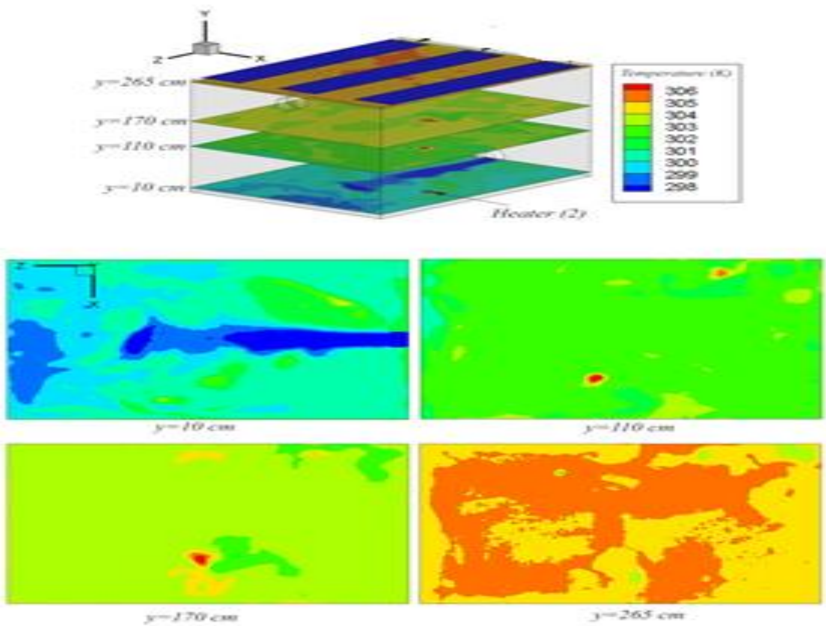

Figure23.Temperature contour at different Y for supplied temperature of air of 24(°C)

5.Conclusion

The velocity and temperature distribution in an occupied zone of a room are studied numerically under different conditions of supplied air of displacement ventilation. The numerical study is achieved by using Fluent 14.5 in a room has dimensions of (300×400×270)cm in width, length, and height respectively. The displacement ventilation is installed in a room with a 31 cm diameter for inlet and outlet grills. The distribution of velocity and temperature are studied at a different supplied-air velocity of displacement ventilation of (0.75,1,1.25.and1.45)m\s with a constant load of 1600 W and constant supply air temperature of 18 °C. Also, The distribution of velocity and temperature are studied at a different supplied air temperature of displacement ventilation of (18,20,22.and 24) °C with a constant load of 1600 W and constant supplied-air velocity of 0.75 m\s. The results show that, under different supplied-air velocities and temperatures, the airflow is tented to be more turbulence as supplied air velocity and temperature increase caused uncomfortable conditions. The best airflow can be obtained at supplied-air velocity and temperature of (1.45 m\s and 18°C) respectively.

Declaration of competing interest

The authors declare that they have no any known financial or non-financial competing interests in any material discussed in this paper.

References

1. M.Road, Aylesford, and Kent, “ An Introduction to chilled beams and chilled ceilings,”.Waterloo
2. Air Products plc,pp.1-36,2012.
3. J .L. M Hensen and M. J .H Hamelinck, “ Energy simulation of displacement ventilation in offices”. Building Services Engineering Research and Technology,pp. 77-8,1995.
4. Y.Yin, W.Xu, J.K.Gupta, A.Guity, P.Marmion, A.Manning, B.Gulick, X.Zhanga and Q.Chen, “Experimental study on displacement and mixing ventilation systems for patient ward”, HVAC&R Research, pp.1175-1191,2009
5. N. M.Mateus and G. C Da Graca , “ Simulation and measured performance of displacement ventilation systems in large rooms “ Building and Environment ,pp.470-482,2017.
6. G. Chitaru , C. Berville and A. Dogeanu, “Numerical simulation and comparison of two ventilation methods for a restaurant – displacement vs mixed flow ventilation,”. E3S Web of Conferences 32, pp.1-7, 2018
7. H. Andersson , M. Cehlin and B. Moshfegh, “ A Numerical and Experimental Investigation of a Confluent Jets Ventilation Supply Device in a Conference Room,”.Energies,pp.1-30 ,2022.
8. L. Jalil, A.H.Taki and D.L. Loveday , “COMPUTATION OF AIRFLOW IN A DISPLACEMENT VENTILATION/CHILLED CEILING ENVIRONMENT,”. Proceedings: Indoor Air,pp272-277, 2002.
    A. Bakker, “ Applied Computational Fluid Dynamics,”.USA,pp.1-612,2008.
9. Gambit 2.4.6 program,2015.
10. ANSYS FLUENT 14.5 program,2015.
11. I.A AbdulGhafor1 , A. A. Abdulrasool and Q. SMehdi, “EXPERIMENTAL STUDY TO THE PERFORMANCE OF THE DISPLACEMENT VENTILATION UNDER DIFFERENT OPERATION CONDITIONS,”. Int. J. Adv. Res,pp. 485-494,2018