Afyon Kocatepe Üniversitesi Fen ve Mühendislik Bilimleri Dergisi
Afyon Kocatepe University – Journal of Science and Engineering
https://dergipark.org.tr/tr/pub/akufemubid

Araştırma Makalesi / Research Article
e-ISSN: 2149-3367
DOI: https://doi.org/10.35414/akufemubid.1608826
AKÜ FEMÜBİD **25** (2025) 055901 (1216-1229)
AKU J. Sci. Eng. **25** (2025) 055901 (1216-1229)

Numerical Investigation of Furniture Arrangement, Airflow Velocity, and Blowing Angle with a Wall-Mounted Air Conditioner

***Makale Bilgisi / Article Info**
Alındı/Received: 28.12.2024
Kabul/Accepted: 23.04.2025
Yayımlandı/Published: xx.xx.xxxx

Duvar Tipi Klima ile Mobilya Yerleşiminin, Üfleme Hızının ve Açısının Sayısal İncelenmesi

Alparslan ŞANLI , Hüsamettin TAN* , Tolga DEMİRCAN

Kırıkkale University, Faculty of Engineering and Natural Science, Mechanical Engineering, Kırıkkale, Turkey

Abstract
In this study, the cooling of a room using a wall-mounted air conditioner and keeping it under thermal comfort conditions are discussed. Within this scope, a standard office room model cooled by air conditioning was created and numerical analyses were conducted. In the study, four different blowing velocities, three different blowing angles and two different placements of the furniture in the room were investigated. The effects of these parameters on the velocity and temperature distribution of the air in the room, which influences thermal comfort conditions, were examined. As a result of the analysis, it was observed that the change in the blowing velocity of the air conditioner does not have much effect on the overall structure of the flow. However, it was determined that changes in the air conditioner's blowing angle and the room's layout significantly affected the velocity and temperature distribution of the air in the room. Within the range of parameters examined, it was observed that the optimum thermal comfort conditions were achieved with furniture Location-I, a blowing velocity of 0.5 m/s, and a blowing angle of 80°.

***Keywords:** Air Condition; Cfd; Thermal Comfort; Room Design*

Öz
Bu çalışmada, duvar tipi bir klima kullanılan bir odanın soğutulması ve ısıl konfor şartlarında tutulması ele alınmıştır. Bu kapsamda klima ile soğutulan standart bir ofis odası modeli oluşturulmuş ve sayısal analizler yapılmıştır. Çalışmada dört farklı üfleme hızı, üç farklı üfleme açısı ve oda içerisindeki mobilyaların iki farklı yerleşimi için incelemeler yapılmıştır. Bu parametrelerin ısıl konfor şartlarına etki eden oda içerisindeki havanın hız ve sıcaklık dağılımına etkileri incelenmiştir. Yapılan analizlerin sonucunda, klimanın üfleme hızının değişiminin akışın genel yapısı üzerinde fazla etkili olmadığı gözlemlenmiştir. Ancak klimanın üfleme açısının ve odanın yerleşim düzeninin değişiminin akış yapısını değiştirdiği, oda içerisindeki havanın hız ve sıcaklık dağılımını önemli ölçüde etkilediği belirlenmiştir. İncelenen parametre aralığında, 1 numaralı mobilya yerleşimi, 0.5 m/s üfleme hızı ve 80° üfleme açısı için optimum ısıl konfor şartlarının gerçekleştiği gözlemlenmiştir.

***Anahtar Kelimeler:** Klima; Had; Isıl Konfor; Oda Tasarımı*

1.Introduction

Nowadays, people spend most of their average life expectancy in indoor environments. Therefore, a more comfortable living space is preferred for working or resting. Air conditioners are widely used to provide thermally comfortable living spaces. Today, air conditioners are among the first preferred options for cooling or heating working and resting environments (Izgi and Kavacik, 2024). The widespread use of air conditioners reveals the importance of thermal comfort conditions and affects people's productivity in the working environment (Harrington, 1933; Ramos et al., 2021). When the studies in the literature are examined, it is seen that different methods, parameters and boundary conditions are used to optimize the air flow (Liu et al., 2016; Chung and Dunn-Rankin, 1998; Ayad, 1999). In the studies on air conditioners, the relationship between comfort and health, people's attitudes towards the use of air conditioners and thermal comfort conditions in the room are generally examined (Ramos et al., 2021; Liu et al., 2016).

Although there are many studies on air conditioners in the literature, there are relatively few studies (Yongson et al., 2007; Gosman et al., 1980; Karaşahin, 2011) on wall-mounted air conditioners located in the room. When the literature is examined, it is seen that it is not enough to focus only on the cooling system, it is more necessary to analyze the entire air flow in the cooled room (Haines, 1977). Therefore, it should be taken into account that the position of each object in the cooled space can affect the air flow. Similar studies in the literature have been examined in detail and some of them are given below.

*Sorumlu Yazar/Corresponding Author: Hüsamettin Tan
e-posta/e-mail: husamettintan@outlook.com*

Pitarma et al. (2003) numerically and experimentally investigated the air flow in an air-conditioned room. They used the k-ε turbulence model as the turbulence model for the numerical solution. In the experimental study, they designed a test environment in accordance with the actual room dimensions. With the results obtained from this test model, they performed numerical analysis by verifying the numerical study. Tripathi and Moulic (2007) investigated the velocity and temperature distribution during the cooling of a room when the inlet and outlet of the air are at a specified location and on opposite walls. As a result, they observed the formation of two vortex regions in the room. They found that the temperature distribution is more regular with increasing density. In addition, with their study, they stated that in the heating, cooling and ventilation sector, the dry bulb temperature should be 20-27°C, the relative humidity should be 30%-60%, and the air velocity should be in the range of 0.1-0.25 m/s. Pulat (2011), numerically investigated the comfort conditions in a room ventilated by an air conditioner.

For this purpose, he modeled a room prepared for thermal comfort studies in two dimensions. He determined the velocity and kinetic energy distributions for three different velocity values of the air conditioner at the same temperature. He obtained the main flow characteristics for the applied boundary conditions. He compared his results with the studies in the literature. He stated that it is possible to obtain the main characteristics of the air flow in the room with two-dimensional analysis under the applied boundary conditions, but it may not be possible to obtain reliable information about the risk of drafts. He also stated that taking the suction of the air conditioner into account in the analysis can increase the accuracy of the results.

Yongson et al. (2007) determined the temperature and velocity distribution on various planes for three different locations of the air conditioner in a room and tried to provide maximum comfort for the occupants in the room. As a result of their study, they determined that among the three different air conditioner positions they analyzed, the position of the air conditioner number 2 (the position where the air conditioner is located in the middle of the wall on the long wall of the room) is more suitable for comfort conditions. Bonefacic et al. (2007) used air conditioning for heating and cooling in their study. They determined two different room heights as another parameter. As a result of their study, they found that there is a significant relationship between room height and constant thermal comfort in the room. Başkaya and Eken (2006), in their study, examined bringing an office room to thermal comfort conditions. They examined the effects of furniture in the office room, air inlet/outlet positions, air inlet velocity, winter/summer conditions on thermal comfort. As a result, they found that the furniture in the room significantly changes the movement of air in the room and this change affects the thermal comfort conditions of a room. Youssef et al. (2018) examined the effects of cold air systems on indoor comfort. They evaluated airflow distribution and comfort levels by analyzing different supply air temperatures, air velocities, and thermal loads. The results indicated that air conditioning systems could provide high comfort under appropriate conditions. They concluded that merely lowering the supply air temperature is not sufficient to improve comfort levels, and air velocity and thermal loads must be considered. Lee et al. (2017), in their study, investigated the effect of the blades and blade angle in the air outlet region of the indoor unit of a wall-mounted (split) air conditioner on thermal comfort. In the first step of their study, they calculated the air distribution, turbulence characteristics and air flow rate depending on the blade angle. In the continuation of the study, they analyzed by changing the blade angle and air conditioner position in the CFD model to evaluate the effectiveness of the air conditioner and air exchange. As a result, they observed that the air flow rate increased by 19%-20.9% when the blade was in the middle position under high velocity air flow conditions. They also determined that the performance increases when the air conditioner is placed in the center of the wall. Harsito et al. (2021) investigated thermal comfort and energy efficiency in a mosque room with wall-mounted air conditioners. In their study, they conducted numerical analyses at different inlet angles and speeds. By examining temperature distributions and airflow patterns, they determined that increasing speed and angle had a positive impact. They emphasized that optimizing these parameters is crucial for both comfort and energy efficiency. Lu et al. (2021) investigated the use of numerical analysis to enhance thermal comfort and energy savings in low-energy smart buildings. The study evaluated how air conditioner placement affects airflow and temperature distribution, revealing that positioning the unit away from the center of the wall directed airflow over longer distances. As a result, indoor temperature distribution became more homogeneous, and energy efficiency improved. The study highlights the importance of proper window usage as a key parameter in energy-efficient building design. Wang et al. (2022) investigated the effects of multi-wing air conditioning systems, developed as an alternative to traditional AC systems, on indoor airflow and thermal comfort. Using numerical simulations and experimental tests, they analyzed different MIAC configurations and determined that the six-wing air conditioner (SIAC) model exhibited the best performance. The results showed that the design improved indoor comfort by increasing airflow distance and reducing temperature differences. Compared to traditional systems, the six-wing air conditioner had a 31% wider air supply angle, a 70% reduction in temperature differences, and an 8.1% decrease in energy consumption. Barau et al. (2024) conducted a numerical study on thermal comfort and energy savings. In their research, they compared thermal comfort conditions in terms of air velocity and average temperature by considering air conditioner placement and speed parameters. The findings indicated that the best cooling conditions were achieved when the air conditioner was mounted on the ceiling. Additionally, they emphasized that numerical analysis studies serve as a crucial tool for enhancing energy efficiency. Bojic et al. (2002) investigated the impact of air conditioning and furniture arrangement on thermal comfort in residential apartments. The airflow and temperature distribution were analyzed by placing the air conditioning unit in different locations, and the Air Distribution Performance Index (ADPI) was calculated. The results showed that positioning the air conditioner away from the wardrobe provided a more balanced airflow and a higher ADPI value. When placed near the wardrobe, airflow irregularities and temperature differences occurred, whereas better comfort was achieved in the middle and distant positions. Sabie et al. (2019) conducted a numerical analysis to investigate the impact of furniture arrangement on airflow distribution in houses. The numerical modeling was carried out for two different configurations: with and without furniture. The results showed that furniture influences airflow direction and creates low-velocity airflow zones in certain areas. However, it was determined that indoor air velocities remained within comfort standards and that the ventilation system operated effectively. Analyzing velocity contours revealed that a more homogeneous airflow was observed in the unfurnished case, whereas turbulence and flow direction changes occurred in certain areas in the furnished model.

As a result of the literature review, it was observed that for wall-mounted air conditioners, there are generally discrete studies that examine the effects of the change of parameters such as the blowing speed of the air conditioner, blade angle, location and furniture placement on thermal comfort. In this study, a standard office room (Lee et al., 2017; Jerold, 1992) is determined and a wall-mounted air conditioner is considered to be placed inside. For 4 different blowing speeds, 3 different blowing angles and 2 different furniture positions of this air conditioner, the thermal comfort conditions in the room are numerically analyzed. In the literature, there are discrete studies in which these parameters are considered separately, but there are not many studies in which these parameters are considered together. Therefore, in this study, the effects of the variation of the three parameters on thermal comfort are evaluated together. The thermal comfort condition has been evaluated by considering the temperature distribution and airflow characteristics in the room.

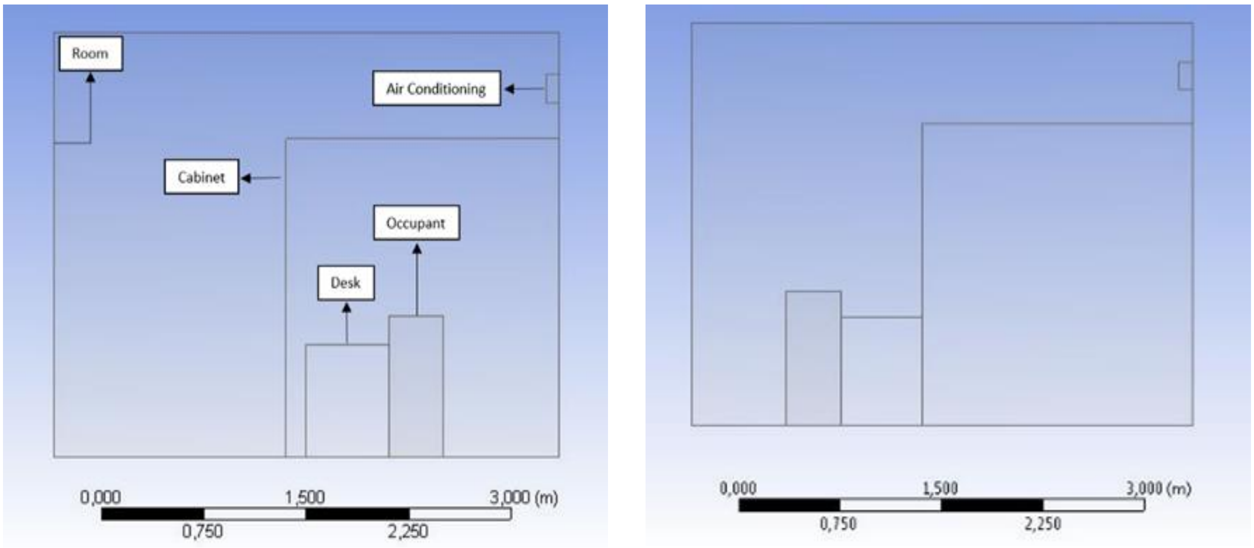
2. Materials and Methods

2.1 Definition of the problem

In this study, cooling of an office room with dimensions of 3.7x2.7x3.0 m (length x width x width) (Lee et al., 2017) with a wall-mounted air conditioner is considered. The front view of the room analyzed in the study is given in Figure 1 and the top view is given in Figure 2. In the interior design of the room, an occupant, a desk and a cupboard were considered (Jerald, 1992). In this context, a cupboard with dimensions of 2.25x0.25x2.0 m and a desk with dimensions of 0.8x0.6x1.4 m were considered in the room. In addition, a rectangular geometry with dimensions of 1.0x0.4x0.4x0.4 m was added to the room to represent an occupant sitting in front of the desk. Two different room layouts (Location-I and Location-II) were determined for different locations of the furniture in the room. In this way, the effects of the furniture and the position of the person on the air flow in the room were analyzed. For these two different room layouts, the effects of the blowing velocity and blowing angle of the air conditioner on thermal comfort were determined and the optimal comfort conditions in the room were tried to be determined.

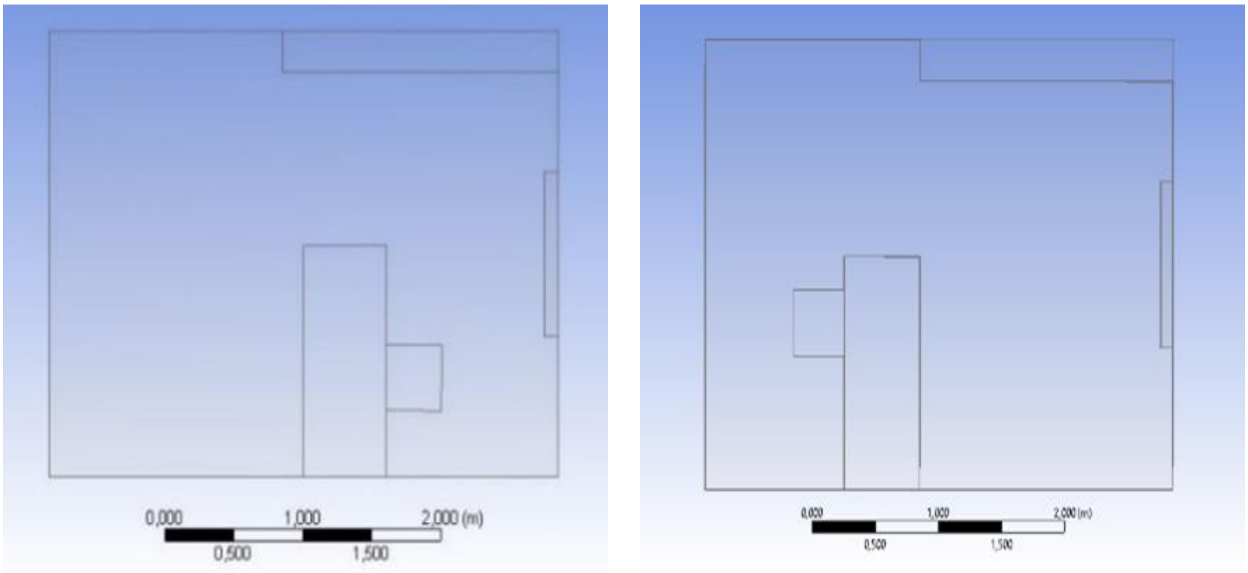
2.2 Differential equations of the problem

The air flow is assumed to be steady-state, incompressible, three-dimensional and turbulent. The k-ε turbulence model is used as the turbulence model. In line with the assumptions made, the basic differential equations used in the solution of the problem are given below. The mathematical expressions and approximation methods used in numerical models are not limited to airflow analysis but can also be applied in various engineering and scientific fields (Koca and Zabun, 2021; KOCA and ZABUN, 2022). The equations of continuity, momentum, energy, turbulent kinetic energy and dissipation rate of turbulent kinetic energy are given for Cartesian coordinates respectively. For the standard k-ε turbulence model, the Standard Wall Function wall function approximation is used.



a) b)

Figure 1. Front view a) Location-I, b) Location-II



a) b)

Figure 2. Top view a) Location-I, b) Location-II

The continuity equation, momentum equations, energy equation, turbulent kinetic energy and turbulent energy dissipation rate equations in three dimensions are presented in he below, respectively (Costa et. all, 1999).

Continuity Equation

$$\frac{\partial(u)}{\partial x}+\frac{\partial(v)}{\partial y}+\frac{\partial(w)}{\partial z}=0 \quad (1)$$

Momentum Equations

$$\frac{\rho\,\partial(uu)}{\partial x}+\frac{\rho\,\partial(uv)}{\partial y}+\frac{\rho\,\partial(uw)}{\partial z}=\frac{\partial}{\partial x}\left(\mu_{ef}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{ef}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{ef}\frac{\partial u}{\partial z}\right)+S_u \quad (2a)$$

$$\frac{\rho\,\partial(uv)}{\partial x}+\frac{\rho\,\partial(vv)}{\partial y}+\frac{\rho\,\partial(wv)}{\partial z}=\frac{\partial}{\partial x}\left(\mu_{ef}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{ef}\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{ef}\frac{\partial v}{\partial z}\right)+S_v \quad (2b)$$

$$\frac{\rho\,\partial(uw)}{\partial x}+\frac{\rho\,\partial(vw)}{\partial y}+\frac{\rho\,\partial(ww)}{\partial z}=\frac{\partial}{\partial x}\left(\mu_{ef}\frac{\partial w}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{ef}\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{ef}\frac{\partial w}{\partial z}\right)+S_w \quad (2c)$$

Energy Equation

$$\frac{\rho\,\partial(uT)}{\partial x}+\frac{\rho\,\partial(vT)}{\partial y}+\frac{\rho\,\partial(wT)}{\partial z} = \frac{\partial}{\partial x}\left(\Gamma ef\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma ef\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma ef\frac{\partial T}{\partial z}\right) \quad (3)$$

Where "T" is the temperature and "Γ_{ef}" is the effective heat transfer coefficient. The effective heat transfer coefficient is equal to the sum of the laminar and turbulent heat transfer coefficients and it is given in the Equation 4.

$$\Gamma ef = \frac{k}{c_p}+\frac{\mu_t}{\sigma_t} \quad (4)$$

Turbulent Kinetic Energy

$$\rho\frac{\partial}{\partial x}(uk)+\rho\frac{\partial}{\partial y}(vk)+\rho\frac{\partial}{\partial z}(wk) = \frac{\partial}{\partial x}\left(\frac{\mu_{ef}}{\sigma_k}\frac{\partial k}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu_{ef}}{\sigma_k}\frac{\partial k}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu_{ef}}{\sigma_k}\frac{\partial k}{\partial z}\right)+G_K + G_B-\rho\varepsilon \quad (5)$$

Where the term "k" represents the kinetic energy of turbulence, "G_K" represents in the Equation 6a that is the rate of generation of kinetic energy due to frictional forces and "G_B" represents in the Equation 6b that is the rate of buoyancy generation of turbulent kinetic energy due to buoyancy forces (Yüce, 2019).

$$G_K = \mu_t\left\{2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial u}{\partial y}\right)^2+\left(\frac{\partial u}{\partial z}\right)^2\right] + \left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}+\frac{\partial v}{\partial y}\right)^2\right\} \quad (6a)$$

$$G_B = -g\frac{\mu_t}{\sigma_t}\frac{1}{\rho}\frac{\partial p}{\partial y} \quad (6b)$$

Turbulent energy dissipation rate

$$\rho\frac{\partial}{\partial y}(v\varepsilon)+\rho\frac{\partial}{\partial y}(w\varepsilon) = \frac{\partial}{\partial x}\left(\frac{\mu_{ef}}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\mu_{ef}}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}\right) + \frac{\partial}{\partial z}\left(\frac{\mu_{ef}}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial z}\right) + \frac{\varepsilon}{k}[C_{1\varepsilon}G_K + C_{3\varepsilon}G_B - \rho C_{2\varepsilon}\varepsilon] \quad (7)$$

The terms "σ_ε", "$C_{1\varepsilon}$", "$C_{2\varepsilon}$" and "C_μ" represent the constants of the turbulence model and are taken as σ_ε=1.34, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, $C_{3\varepsilon}$=1.00 and C_μ=0.09 respectively (Çalışkan et. all, 2011).

2.3 Boundary conditions

The air is blown into the office at a constant temperature T=283 K, with four different velocities (V=1.4, 1, 0.7 and 0.5 m/s) and three different blowing angles (θ=45°, 60° and 80°). At the location of the air conditioner suction vent, the outlet pressure is taken as equal to atmospheric pressure $P_ç = P_{atm}$). The representation of boundary conditions is shown in Figure 3. It is assumed that the walls of the room, the remaining surfaces of the air conditioner, the surfaces of the furniture and the body surfaces of the person have a non-slip boundary condition and therefore all velocity component values on these surfaces are zero. It is also assumed that no heat transfer takes place from these surfaces and that the surfaces are adiabatic ($\frac{\partial T}{\partial n} = 0$). Heat transfer by natural convection is neglected.

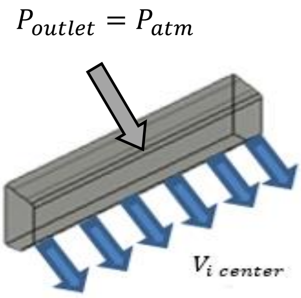


Figure 3. Representation of Boundary Conditions

2.4. Numerical Method

The continuity, momentum, energy and turbulence equations of the problem considered in this study are solved numerically using the boundary conditions of the problem. ANSYS Fluent software package, which is based on computational fluid dynamics (CFD), is used for the analysis. In order to test the reliability of the numerical method and to determine the turbulence model to be used, a validation study was carried out for the results of a reference experimental study (Nielsen et. all, 1990) and turbulence model used in this study was selected because it is the most widely applied method in the literature for indoor airflow analysis (Bamadu et. all, 2017, Lu et. all, 2021). In this context, it was decided to use the standard k-epsilon turbulence model as the turbulence model. As a discretization method, the "Second Order Upwind" method was preferred for the momentum and energy equations and the "First Order Upwind" method for the turbulence equations. The details of the analysis settings are given in Table 1.

Table 1. Solver settings

Turbulence Model	k-epsilon
Momentum discretization	Second order upwind
Energy discretization	Second order upwind
Turbulence discretization	First order upwind
Pressure discretization	Standard
Solver Method	Simple
İteration	5000

The convergence criterion for the numerical analysis was set as 10^{-3} for continuity, momentum, k and epsilon values and 10^{-6} for the energy equation. For the control volume of the room, which is the solution volume of the problem, the mesh structure shown in Figure 4 was created. Tetrahedral mesh was used in the study because it provides better adaptation in areas with complex geometry, allowing for a more precise modeling of the flow field. While constructing this mesh structure, a higher density mesh structure was created in the regions where the air conditioner blowing nozzle and the outlet vent are located. In order to see whether the numerical analysis results are independent of the mesh structure, different values were determined that is given in the Figure 5.

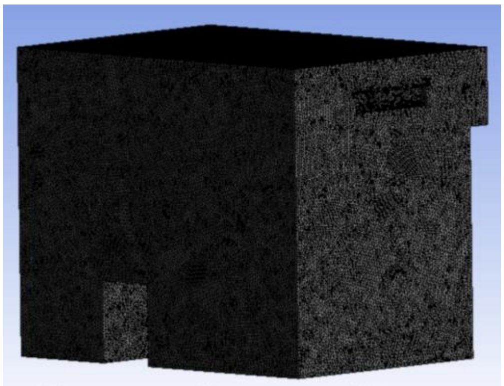


Figure 4. Mesh structure

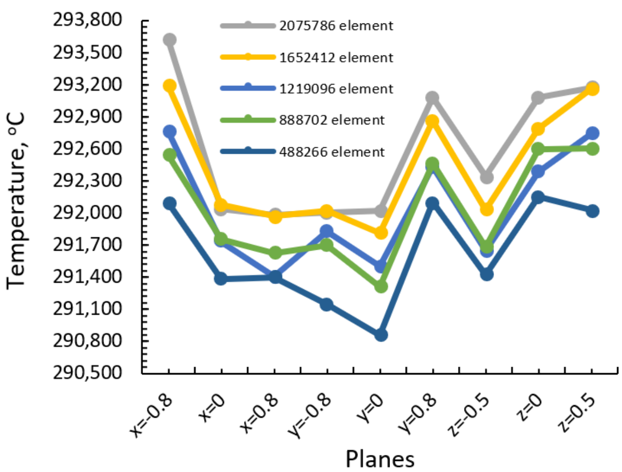


Figure 5. Mesh independence test

For these mesh structures, the degree of convergence, processing time and in-room temperature distribution were obtained. The results showed that increasing the number of elements beyond 1,652,412 did not significantly change the average temperature (all planes) in room, while computational time increased considerably. The optimum mesh structure was used in all analyses performed in this study. The mesh quality is high, with well-balanced element quality, orthogonality, low skewness and results are given in the Table 2.

Table 2. Mesh quality

Mesh Metric	Average Value
Element Quality	0.83246
Orthogonal Quality	0.766662
Skewness	0.232

The computational domain was defined in a Cartesian coordinate system. The origin (0,0,0) is defined at the exact center of the room. The **x-axis** represents the horizontal direction, the **y-axis** represents the depth, and the **z-axis** represents the height of the room. The coordinate system and axes are clearly depicted according to in Figure 6.

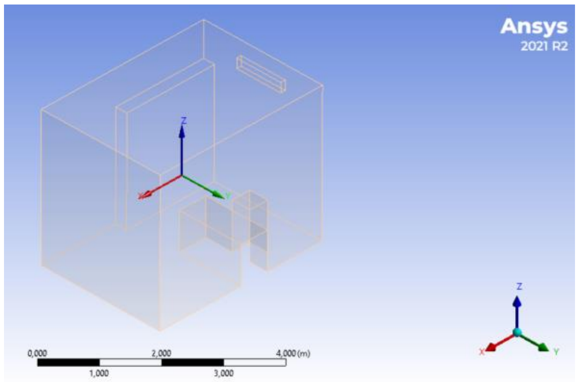


Figure 6. The representation of the origin point

2.4 Validation of the numerical method

Before starting the numerical analysis, a validation study was performed for the results of an experimental study (Nielsen et. All, 1990) taken as a reference from the literature to test the reliability of the numerical method used and to determine the turbulence model to be used. In that study, Nielsen et. al (1990) experimentally investigated the velocity and temperature distributions for an IEA (International Energy Agency) Annex20 test room. Using the same geometry and boundary conditions of this experimental study, numerical analyses were performed for different turbulence models. In the steady-state analysis, all walls are assumed to be adiabatic. Air enters the system through the supply vent located at the top left corner at a velocity of 0.455 m/s and exits the room through the exhaust vent at the bottom right corner. The Reynolds number, defined based on the height (H) of the supply vent, is 5000. The agreement of different numerical methods with the experimental results from the referenced study has been examined. In this context, Standard k-ε swf, Realizable k-ε, RNG k-ε, Standard k-w and SST k-w turbulence models and wall approaches are discussed. The geometry and boundary conditions used in the validation are shown in Figure 7.

In the analysis performed in the steady state, the side walls and ceiling are assumed to be adiabatic. A heat flux of 63.08 W/m^2 is given at the floor. Air enters the system at a speed of 0.455 m/s from the blowing grille in the upper left corner and leaves the room through the suction grille in the lower right corner. The value of the Reynolds number defined according to the height (H) of the blowing vent is 5000. The comparison of the numerical results obtained as a result of the analysis with the literature results is given in Figure 5. When Figure 8 is examined, the experimental results of the study used as a reference and the numerical results obtained for different turbulence models within the scope of this study are seen together. As can be seen from the figure, the results obtained for the k-epsilon turbulence equation and the literature results are quite consistent with each other. Therefore, it can be said that the numerical method used in this study and the numerical results obtained from the analysis are reliable.

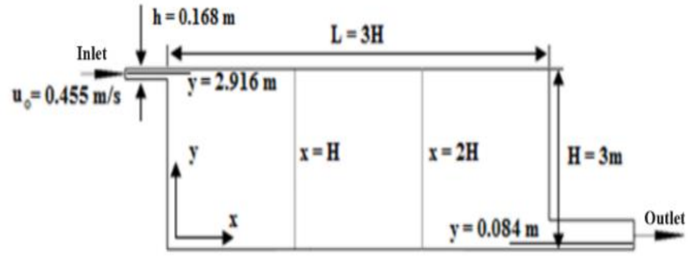
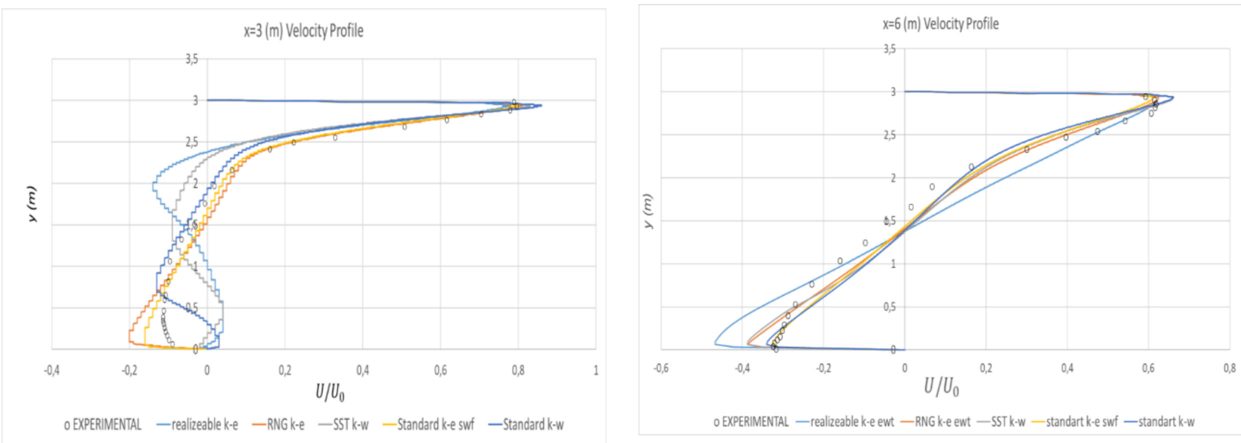


Figure 7. Boundary conditions on Annex20 room



a) b)

Figure 8. a) x=3 m Velocity Profile, b) x=6 m Velocity Profile (Nielsen et. all, 1990)

In another validation study, Bonefacic et al. (2007) designed two three-dimensional rooms with different heights set at two values (2.5m and 3m) to examine the effects of room height on thermal comfort. To analyze the results at specific velocity and temperature conditions, they observed the vertical temperature distribution at the center of the room. In Figure 6, Bonefacic et al. (2007) conducted their analysis with a room height of 2.5m and presented the temperature distributions along the vertical line passing through the center of the room. When examining the temperature distribution graph in Figure 9, it is observed that the results are consistent with those obtained by Bonefacic et al.

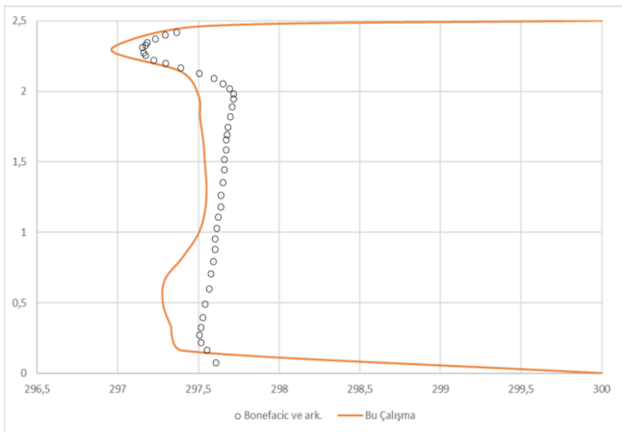


Figure 9. Temperature distribution at different heights (Bonefacic et. all, 2007)

3. Results and Discussions

In this study, the velocity and temperature distributions of an office room cooled by a wall-mounted air conditioner were investigated. In this context, the analyses were repeated for four different values of the blowing velocity (1.4 m/s, 1.0 m/s, 0.7 m/s and 0.5 m/s), three different values of the blowing angle (45°, 60°, 80°) and two different placements of the furniture in the room (Location-I and Location-II). The temperature of the air blown from the air conditioner was kept constant at 283 K. As a result of the analyzes, the effects of the air velocity blown from the air conditioner inlet, blowing angle and furniture placement on the thermal comfort in the room were tried to be determined. Three different room planes (y=0, y=0.4 and y=0.8) were selected to better examine the results. y=0 is the vertical plane passing through the center of the room and was chosen to examine the direction and characteristics of the air flow from the air conditioning nozzle. The y=0.4 (for Location-II) and y=0.8 (for Location-I) planes are the planes that vertically intersect the desk and the occupant from the center and are the planes that cover the working area and the thermal comfort in this area is examined. In the numerical analysis phase, numerical analyses were first performed for four different velocity values and the most appropriate velocity value was determined according to the results obtained. Then, numerical analyses were performed for three different blowing angles and the optimum blowing angle was determined according to the results obtained. Finally, the optimal velocity and blowing angle were considered as boundary conditions and analyzed for two different furniture positions and the results were presented visually. In this context, 8 different cases given in Table 3 were determined and the results obtained for these cases are discussed in detail through the figures given below.

The change in the velocity with the change in the blowing velocity is shown together in Figure 10 for the y=0 m plane. The velocity distributions in the y = 0 m plane indicate variations in airflow behavior for the four investigated cases, with noticeable differences in jet trajectory and velocity magnitude. Compared to other cases, some configurations exhibit wider airflow dispersion, which may impact thermal comfort. Figure 10c exhibits higher maximum velocity and a more pronounced jet flow penetration, whereas Figure 10d shows lower velocity with a wider distribution of low-speed regions.

Table 3. Investigated computational cases

Case	Velocity (m/s)	Blowing Angle (°)	Blowing Temp. (°C)	Furniture Location
I	1.4	45	10	LOCATION-I
II	1	45	10	LOCATION-I
III	0.7	45	10	LOCATION-I
IV	0.5	45	10	LOCATION-I
V	1.4	60	10	LOCATION-I
VI	1.4	80	10	LOCATION-I
VII	0.5	80	10	LOCATION-I
VIII	0.5	80	10	LOCATION-II

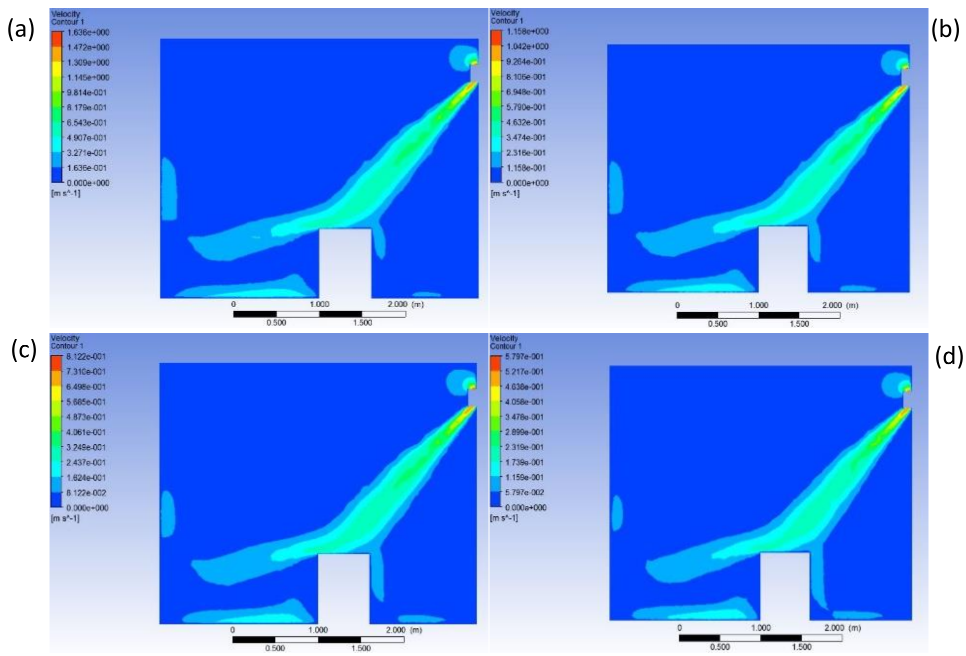


Figure 10. Velocity distributions on y=0m plane for investigated cases a) I, b) II, c) III and d) IV

In all cases, the streamlines in the figure 11 indicate strong recirculation zones in the lower-left region, but the intensity and size of the vortices vary between configurations. The velocity magnitudes along the jet path are highest in case b, as indicated by the extended yellow region, whereas case d demonstrates lower jet velocity and a more diffused flow pattern. Overall, cases 11a and 11c maintain stronger circulation patterns, while case 11d experiences the weakest vortex formation, suggesting variations in outlet conditions significantly influence the flow field.

The change in the temperature distribution with the change in the blowing velocity is shown together in Figure 12 for the y=0.8m plane. When Figure 12 is examined, it is seen that the average temperature in the room increases with the decrease in the blowing velocity and higher temperature layers are formed in the room. It can be said that the increase in the air conditioner blowing velocity provides better cooling in the room. There was a temperature difference of approximately 2°C between the highest blowing velocity and the lowest blowing velocity. However, it should be kept in mind that the energy consumption of the air conditioner will increase with the increase in the blowing velocity of the air conditioner. Blowing speed directly affects the energy consumption of the air conditioner. Higher blowing speeds enhance cooling efficiency by distributing cold air more rapidly throughout the room. However, this also increases the power consumption of the fan and compressor, leading to higher energy usage.

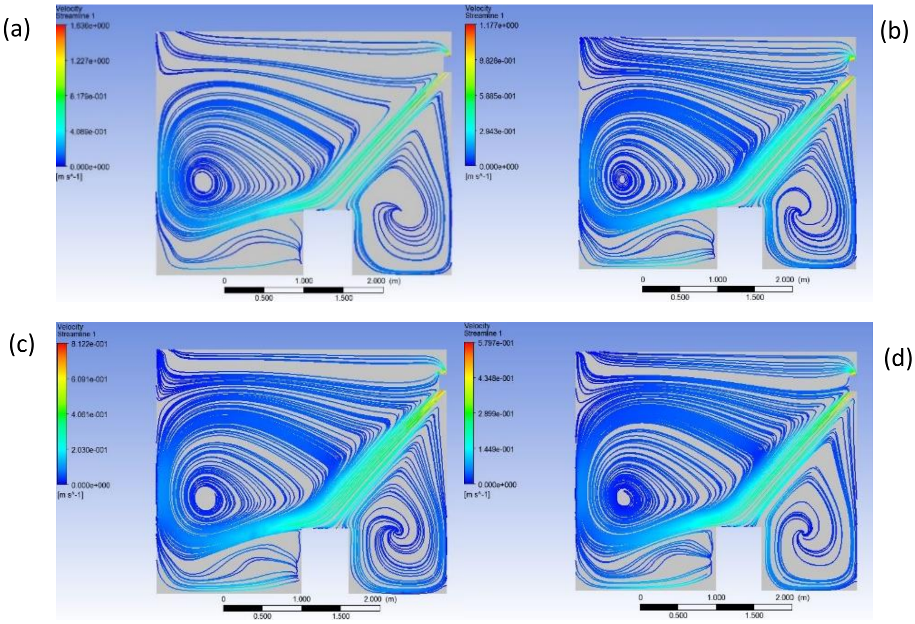


Figure 11. Streamlines on y=0m plane for investigated cases a) I, b) II, c) III and d) IV

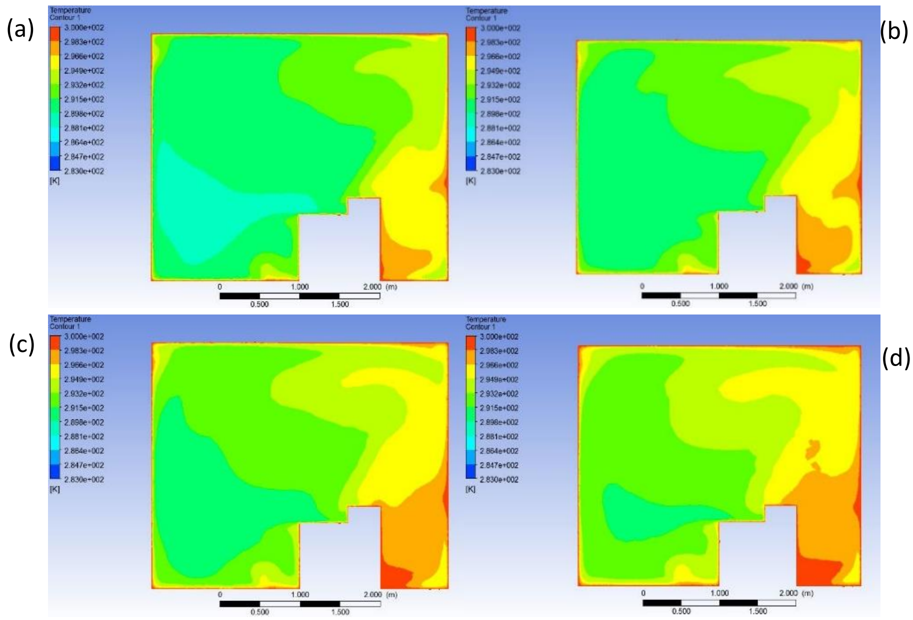
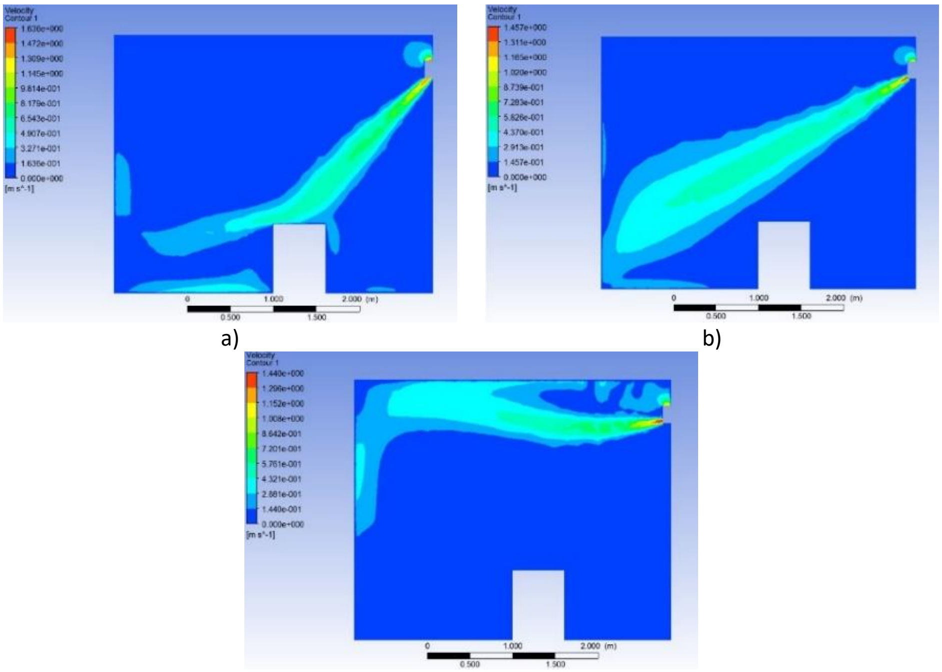


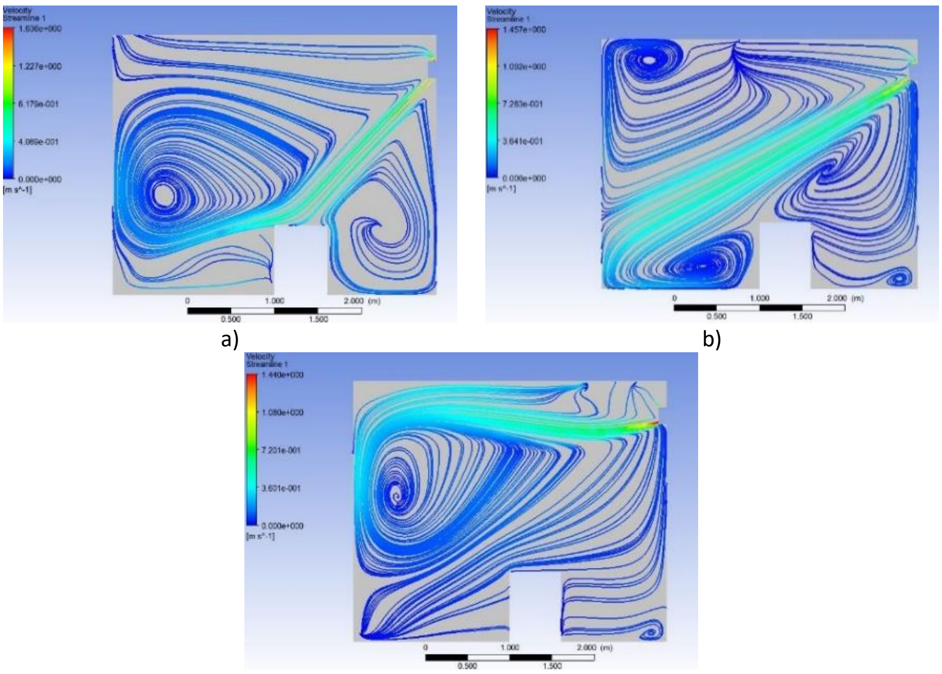
Figure 12. Temperature distributions on y=0.8m plane for investigated cases a) I, b) II, c) III and d) IV

For an air conditioner blowing speed of 1.4 m/s, the velocity distributions obtained by varying the blowing nozzle angle (45°, 60° and 80°) are shown in Figure 13 and the streamlines are shown in Figure 14. When Figure 13 is analyzed, it is clearly seen that the change of the air conditioner blowing angle significantly changes the velocity distribution of the air in the room. With the increase in the blowing angle, the flow direction changes and the region where the cold air is effective increases in the vertical direction. When Figure 14 is analyzed, it is seen that the number and size of the vortices formed in the room also change with the change of the air conditioner blowing angle. In this context, it is seen that one vortex is formed in the room when the blowing angle is 80°, two vortices are formed for 45° and three vortices are formed for 60°. It can be predicted that as the number of vortices in the room increases, thermal comfort may be negatively affected and the thermal comfort for the person in the room may increase.



a) b) c)

Figure 13. Velocity distributions on y=0.8m plane for investigated cases a) I, b) V and c) VI



a) b) c)

Figure 14. Streamlines on y=0.8m plane for investigated cases a) I, b) V and c) VI

The temperature distributions are given together in Figure 15 for different blowing angles at the same velocity values. When the air is blown from the air conditioner at an angle of 80 degrees, it is seen that the air moves parallel to the ceiling surface and cools the room air by spreading over the opposite wall surface. Therefore, for this blowing angle, people in the room do not directly encounter cold air. For the other two blowing angles, it is seen that the cold air is directed directly towards the center of the room and is effective in the zone where people are located. This situation may cause the occupant in the room to directly encounter cold and fast air and cause uncomfortable condition. Therefore, it is seen that the 80° blowing angle can provide more comfortable cooling than the other two blowing angles under the given conditions.

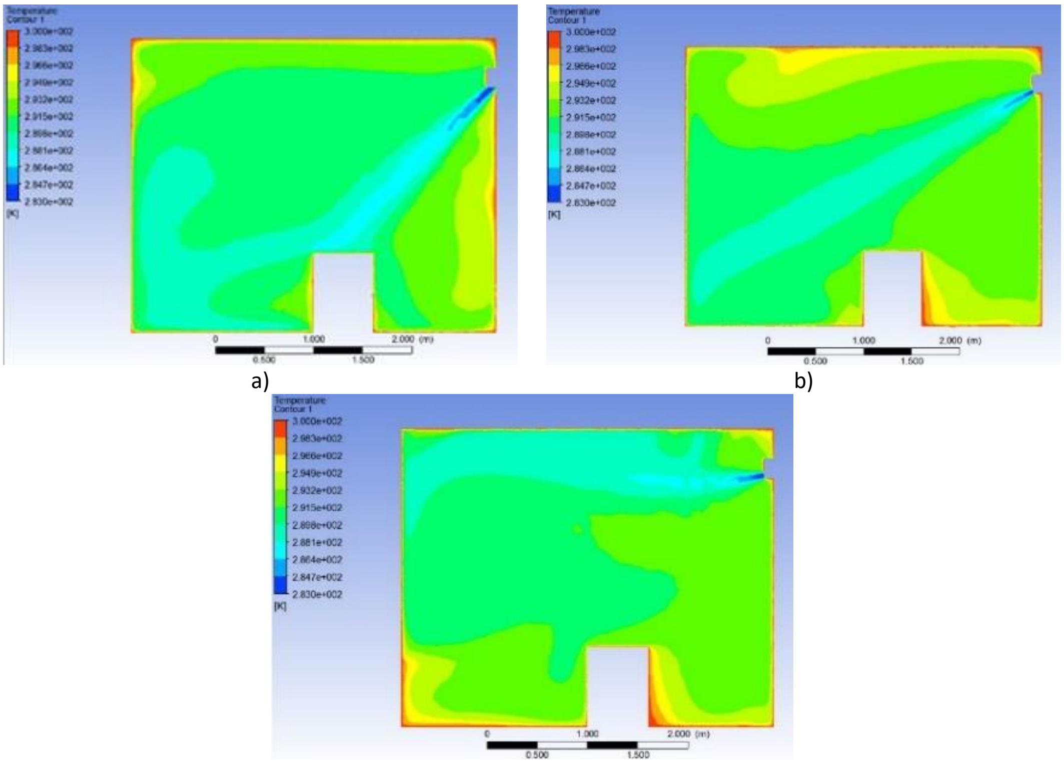


Figure 15. Temperature distributions on y=0m plane for investigated cases a) I, b) V and c) VI

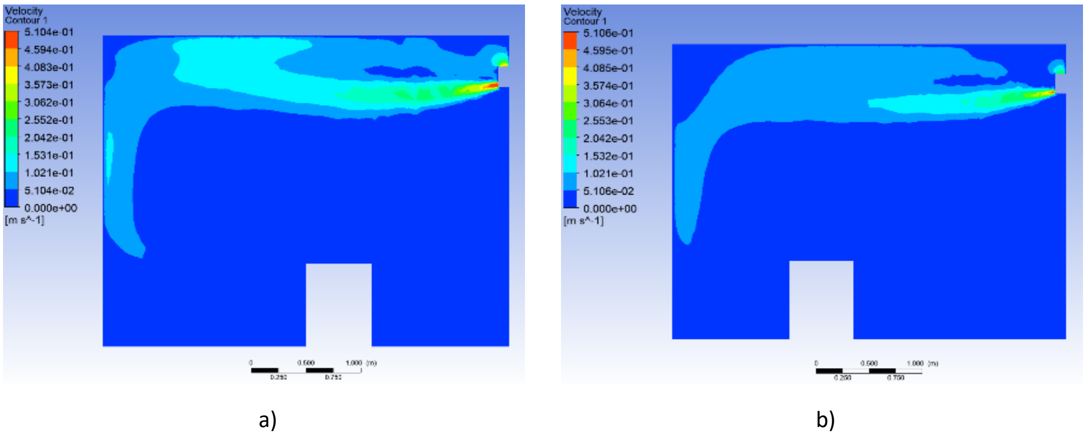
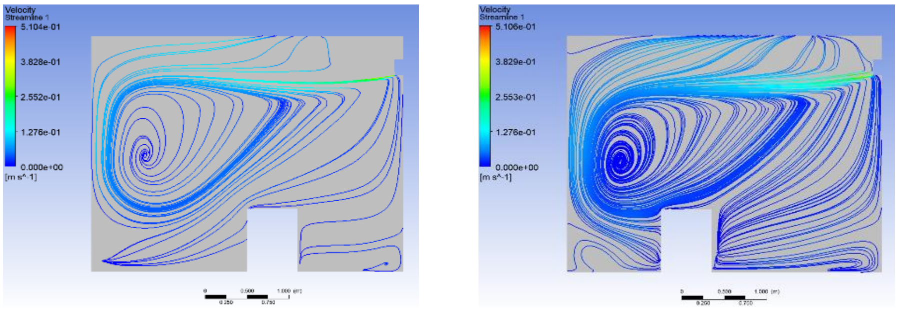


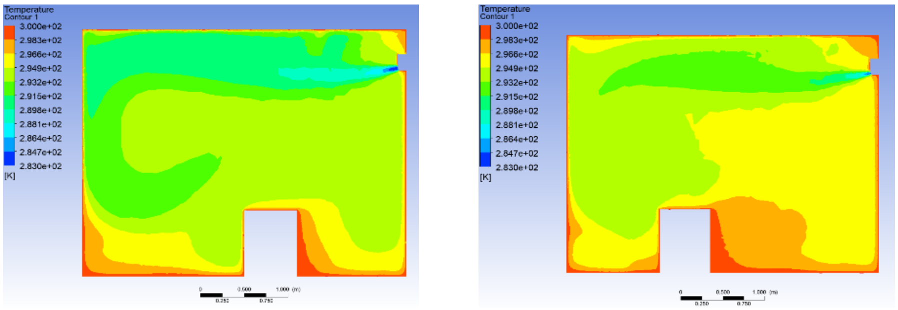
Figure 16. Velocity distributions on y=0m plane for investigated cases a) VII and b) VIII

Figure 16 shows the velocity distributions and Figure 17 shows the streamlines obtained for two different room furniture layouts when the air conditioner blowing velocity is 0.5 m/s and the blowing angle is 80°. Figure 16 shows that the air flow shows similar characteristics and follows the same way for both cases considered. Similar velocity distributions were realized for both furniture placement configurations. Figure 17 shows that large vortices have formed in both cases, but their structures and distributions differ. In Case VII (a), a single dominant vortex covers a broader region, leading to a more widespread airflow pattern. In Case VIII (b), the vortex appears more compact and intense, with additional smaller flow structures forming due to the presence of obstacles, creating a more complex flow field.



a) b)

Figure 17. Streamlines on y=0m plane for investigated cases a) VII and b) VIII

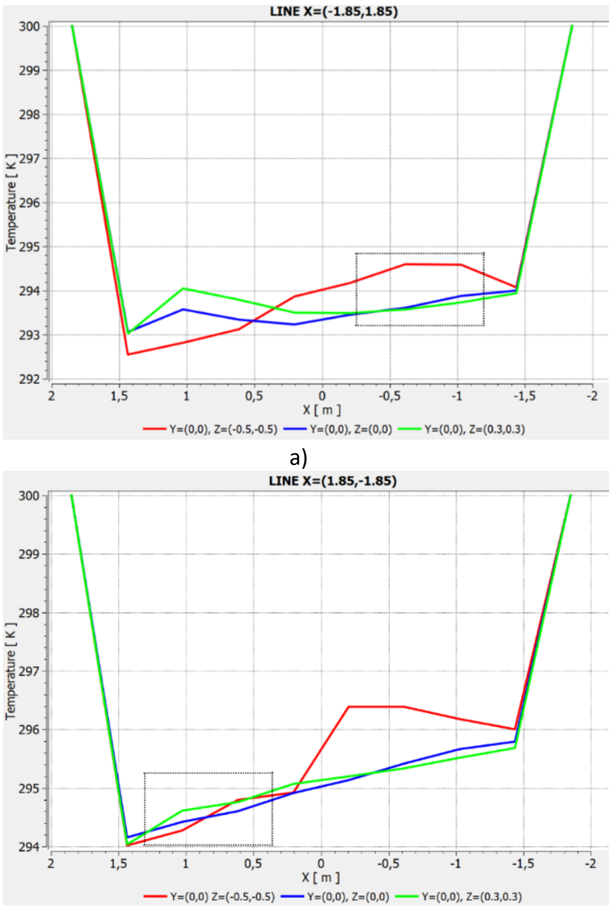


a) b)

Figure 18. Temperature distributions on y=0 m plane for investigated cases a) VII and b) VIII

Figure 18 shows the temperature distributions obtained for two different room furniture layouts when the air conditioner blowing speed is 0.5 m/s and the blowing angle is 80°. When the figure is analyzed, it is seen that changing the furniture layout significantly affects the temperature distribution inside the room. Therefore, it can be said that furniture layout is important for the cooled room to reach thermal comfort conditions and to increase the thermal satisfaction of the occupant working in the room.

Three separate lines were determined at heights of 1, 1.5, and 1.8 meters from the floor of the room, based on the Z-axis coordinates. These lines correspond to the following points: (X=-1.85, 1.85, Y=0, 0, Z=-0.5, -0.5), (X=-1.85, 1.85, Y=0, 0, Z=0.0), and (X=-1.85, 1.85, Y=0, 0, Z=0.3, 0.3). The temperature values obtained for two different room furniture layouts for the case where the air conditioner blowing velocity is 0.5 m/s and the blowing angle is 80 ° are given in Figure 19 using the three lines generated. As can be seen from the figure, lower temperatures occur in the areas where the occupant is working in the office room examined for case VII. A temperature difference of approximately 2 °C was observed between the two cases shown in the figure. In the parameter range examined, it is seen that the furniture layout conditions considered in Case VII are more suitable in terms of thermal comfort.



b)

Figure 19. Temperature distribution along 3 different lines for cases a) VII and b) VIII

4. Conclusions

In this study, cooling of an office room with a wall-mounted air conditioner is considered. In this context, four different air conditioner blowing velocities, three different air conditioner blowing angles and two different furniture layouts are analyzed. In this context, firstly, a comparison was made with an experimental study in the literature and the turbulence model to be used was decided. As a result of the analyzes, the velocity and temperature distributions in the room were determined and the thermal comfort condition of the room was examined. Although the velocity magnitudes vary with different blowing velocities, the overall flow structure and circulation patterns remain largely similar. However, localized variations in velocity distribution, especially near the jet outlet and along the walls, indicate that the blowing velocity has a noticeable. Increasing blowing velocities have led to the formation of jets with higher kinetic energy in the room, allowing the flow to penetrate deeper. However, it was observed that the blowing angle of the air conditioner significantly affected the flow profile. With the change of the blowing angle, it was determined that different locations and different numbers of vortices were formed in the air flow in the room. The interaction of the airflow with obstacles, walls, and the exhaust vent in the room varies. The blowing angle has affected the number and location of vortices by changing the direction of the air jet and the distribution of airflow in the room. Since these vortices change air flow and temperature distribution, they have a direct impact on thermal comfort. An incorrect blowing angle has caused unbalanced airflow, reducing comfort, while an optimal angle has provided a more homogeneous air distribution, enhancing comfort. In addition, it was determined that changing the furniture layout in the room also significantly affects the temperature distribution in the room. The furniture placed in the room obstructed the airflow, changing its direction. In different arrangements, the airflow followed different paths, creating turbulence regions and vortices. These vortices increased air mixing, affecting the temperature distribution and causing the airflow to be trapped in certain areas. It was observed that the most favorable situation in terms of thermal comfort in the parameter range examined was realized under conditions where the air conditioner blowing velocity was 0.5 m/s, the blowing angle was 80^{o} and the furniture layout was Location-I.

Declaration of Ethical Standards
The authors declare that they comply with all ethical standards.

Credit Authorship Contribution Statement
Author-1: Conceptualization, Validation Methodology, Analysis
Author-2: Writing-Original draft preparation, Methodology, Analysis
Author-3: Visualization, Investigation, Writing-Reviewing and Editing.

Declaration of Competing Interest
The authors have no conflicts of interest to declare regarding the content of this article.

Data Availability
Statement all data generated or analyzed during this study are included in this published article.

5. References

Anak A., Barau B., Zambri AN., 2024. Flow Observation on Air Flow Distribution in Room by HVAC System using Ansys Fluent CFD Simulation. *Journal of Complex Flow*, **6(1)**,29-37.

Avara, A. and Daneshgar, E., 2008. Optimum placement of condensing units of split-type air-conditioners by numerical simulation. *Energy and Buildings*, 40**(7)**, 1268-1272. https://doi.org/10.1016/j.enbuild.2007.11.011

Ayad, S. S., 1999. Computational study of natural ventilation. *Journal of Wind Engineering and Industrial Aerodynamics*, **82**, 49-68. https://doi.org/10.1016/S0167-6105(98)00210-4

Başkaya, Ş. and Eken, E., 2006. Numerical investigation of air flow inside an office room under various ventilation conditions. *Pamukkale University Engineering College Journal of Engineering Science*, 12**(1)**, 87-95.

Bamodu O., Xia L., Tang L., 2017. A Numerical Simulation of Air Distribution in an Office Room Ventilated by 4-Way Cassette Air-conditioner. *Energy Procedia*, **105**:2506-2511. https://doi.org/10.1016/j.egypro.2017.03.7221

Bojic M., Yik F., Lo TY., 2002. Locating air-conditioners and furniture inside residential flats to obtain good thermal comfort. *Energy and Buildings*, 34**(7)**, 745-751. https://doi.org/10.1016/S0378-7788(01)00143-8

Bonefacic, I., Frankovic, B., Vilicic, I., and Glazar, V., 2007. 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*, **Vol II**, 1055-1062.

Chung, I. P. and Dunn-Rankin, D., 1998. Using numerical simulation to predict ventilation efficiency in a model room. *Energy and Buildings*, 28**(1)**, 43–50. https://doi.org/10.1016/S0378-7788(97)00061-3

Costa, J. J., Oliveira, L. A., and Blay, D., 1999. Test of several versions for the k-ε type turbulence modelling of internal mixed convection flows. *International Journal of Heat and Mass Transfer*, **42**, 4391-4409. https://doi.org/10.1016/S0017-9310(99)00075-7

Çalışkan, S., Altunok, T., Başkaya, Ş., and Güngüneş, H. M., 2011. Numerical analysis of a commercial display cabinet with air curtain. *Journal Faculty Engineering and Architecture Gazi University*, 26**(2)**, 415-425.

Haines, R. W., 1977. Control system for heating, ventilating and air conditioning. 2nd ed. New York: Van Nostrand Reinhold.

Harrington, E., 1933. Air conditioning for comfort and health, past-present-future. *Journal of the Franklin Institute*, 215**(6)**, 655-696. https://doi.org/10.1016/S0016-0032(33)90679-1

Harsito C., and Permata ANS., 2021. Investigation of air distribution in mosque rooms with different angles of supply and inlet velocity. *Int J Heat Technol.,* 39**(4)**, 1383-1388. https://doi.org/10.18280/ijht.390439

Izgi, F., and Kavacık, M., 2024. Analyzing global competitiveness of Turkish air conditioning industry. *Turkish Journal of Engineering*, 8**(2)**, 209-234. https://doi.org/10.31127/tuje.1372141

Jerold, L. A., 1992. Architectural plan for adding on or remodeling. TAB Books.

Karaşahin, F., 2011. Numerical investigation of air conditioning of a furnished office room. Master's Thesis, Gazi University, Ankara.

Koca, F., and Zabun, M., 2021. The effect of outlet location on heat transfer performance in micro pin-fin cooling used for a CPU. *European Physical Journal Plus*, 136, 1115. https://doi.org/10.1140/epjp/s13360-021-02113-4.

Koca, F., and Zabun, M., 2022. Nozul Mesafesinin Çarpan Jet Isı Transferi ve Akışkan Akışı Üzerindeki Etkileri. *Osmaniye Korkut Ata Üniversitesi Fen Bilimleri Enstitüsü Dergisi*,5**(2)**,1008–1021.

Lee, S., Lee, J., and Kato, S., 2017. Influence of vane angle on the effectiveness of air of wall-mounted split-type air conditioners in residential buildings. *Science and Technology for the Built Environment*, 23**(5)**, 761-775. https://doi.org/10.1080/23744731.2016.1260410

Liu, N., Lai, X. L., Yan, K., and Zhang, H., 2016. Investigation of flow and heat transfer characteristics on different heat exchangers of air conditioner. *Applied Thermal Engineering*, **103**, 428–43. https://doi.org/10.1016/j.applthermaleng.2016.04.084

Lu YC., Wei Chen C., Gong XJ., Rong Lee K., Tseng CJ., 2021 CFD Simulations for Thermal Comfort and Energy Saving. *Proc - 22nd IEEE/ACIS Int Conf Softw Eng Artif Intell Netw Parallel/Distributed Comput SNPD*, 167-170. https://doi.org/10.1109/SNPD51163.2021.9704978

Nielsen, P. V., 1990. Specification of a two-dimensional test case. Institute for Bygningsteknik, Aalborg Universitet.

Pitarma, R. A., Ramos, J. E., and Carvalho, M. G., 2003. *Computational and experimental reduced scale modelling of air-conditioned rooms*. Building Simulation 2003: Eighth International IBPSA Conference, 1041-1045, Eindhoven, Netherlands.

Pulat, E., 2011. *Numerical investigation of air distribution in a room ventilated by an air conditioning unit*. Ulusal İklimlendirme Kongresi, 389-400, Antalya, Turkey.

Ramos, G., Lamberts, R., Abrahão, K. C. F. J., Bandeira, F. B., Teixeira, C. F. B., De Lima, M. B., Broday, E. E., Castro, A. P. A. S., Leal, L. De Q., De Vecchi, R., De Zorzi, L. De M., Dornelles, K. A., Duarte, S., Faisca, R. G., Fontenelle, M., Freitas, T., Atem, C. G., Grigoletti, G., Maciel, L., De Moraes, C. M., Mueller, C. M., Sartori, T., Silva, B., Silva, C., Da Silva, M. V., Silveira, A. L., Souza, R. V. G., and Xavier, A. A., 2021. Adaptive behaviour and air conditioning use in Brazilian residential buildings. *Building Research & Information*, 49**(5)**, 496-511. https://doi.org/10.1080/09613218.2020.1804314

Sabie DD., Fatu V., Ghiaus AG., 2019. Local analysis of airflow distribution in open concept passive houses. *E3S Web Conference*, **111**, 1-7. https://doi.org/10.1051/e3sconf/201911101019

Tripathi, B. and Moulic, S. G., 2007. Investigation of the airflow patterns inside a room through the low Re k-ε *Model. Heat and Technology*, 25**(1)**, 25-31.

Wang J., Liu X., Tian C., Wu L., Xi G., 2022. Indoor Environment Control Effect Based on Multi-Impeller Air Conditioner. *J Therm Sci.*, 31**(6),** 2137-2154. https://doi.org/10.1007/s11630-022-1537-5

Yongson, O., Badruddin, I. A., Zainal, Z. A., and Narayana, P. A. A., 2007. Airflow analysis in an air conditioning room. *Building and Environment*, **42**, 1531-1537. https://doi.org/10.1016/j.buildenv.2006.01.002

Youssef AA., Mina EM., ElBaz AR., AbdelMessih RN., 2018 Studying comfort in a room with cold air system using computational fluid dynamics. *Ain Shams Eng J*., 9**(4)**, 1753-1762. https://doi.org/10.1016/j.asej.2016.07.005

Yüce, B. E., 2019. Comparison of turbulent flow models in office medium and numerical simulation of various heating systems by using thermal comfort indices and Gagge Model. PhD Thesis, Bursa Uludağ University, Bursa.