



An Approach for Solver Sensitivity Analysis Cfd Simulations for Natural Ventilating in a Middle-Scale Mosque

Ayşe Şeyma ARSLANTAŞ^{1,*}, İdil AYÇAM²

¹ 0000-0002-2097-2064, Gazi University, Faculty of Architecture, Department of Architecture, Eti Mah. Yükseliş sok. No.5 Maltepe, Ankara

² 0000-0001-7170-5436, Gazi University, Faculty of Architecture, Department of Architecture, Eti Mah. Yükseliş sok. No.5 Maltepe, Ankara

Article Info

Received: 03/12/2024

Accepted: 11/03/2025

Keywords

Building Physics,
Computational Methods
in Fluid Flow,
Models and Simulations
of Design

Abstract

Energy-efficient building design is a goal achieved through efficiency in the calculations of energy consumption based on building simulations during the design phase. During the design phase, measures to reduce the building's energy consumption can be taken, and in existing structures, the potential for natural ventilation can be utilized through passive design principles without compromising comfort conditions. In addition, predicting the effectiveness of natural ventilation often leads to highly uncertain results compared to mechanical systems. Unlike the extensive mechanical system analyses found in the literature, there is uncertainty in the accuracy of appropriate CFD solution steps in natural ventilation studies. Different approaches are being tested in CFD solutions to increase the accuracy of simulation-based predictions. Among the approaches to predicting ventilation effectiveness in buildings, computational fluid dynamics (CFD) calculations are common. CFD analyses show varying results depending on solver settings, discretization schemes, turbulence models, mesh sensitivity, and time methods. This study investigates the necessary steps to simulate and predict the effectiveness of natural ventilation. It explores whether a different approach is needed in terms of the CFD calculation model compared to mechanical CFD examples to simulate the indoor with natural ventilating. This study tests different analytical approaches for certain opening configurations to analyze the effect of numerous uncertain variables on comfort parameters. The extent to which the results of the Coupled and SIMPLE solvers affect cross-ventilation and perpendicular ventilation cases is questioned. Differences between parameters such as CO₂ concentration as a pollutant gas value, air velocity, and temperature have been taken into account. While the air velocity parameter shows quite similar results for both solvers, the results for CO₂ concentration and temperature parameters differ significantly. Whether the flow is in two-dimensional or 3D flow volume affects the solver settings. Since CO₂ concentrations depend on the mass increase, they were found to be slightly affected by the solvent difference. Air velocity was found to be the most important factor affecting the flow pattern and the distribution of concentrations and temperature. When evaluating cross ventilation and perpendicular ventilation in terms of indoor air quality and thermal comfort, perpendicular ventilation tends to create a more negative indoor air quality situation.

1. INTRODUCTION

Natural ventilation can be a highly efficient energy strategy as an alternative to ventilation systems that impose a 9% energy load on end users [1]. For large spaces to benefit from energy-efficient strategies, the design of openings must appropriately utilize passive ventilation strategies based on their volume. When considering mechanical systems, system loads increase in proportion to the size of the space, leading to significant energy consumption for heating, cooling, and maintaining good air quality compared to other building types. Furthermore, it's known that buildings with large volumes significantly impact operational and maintenance burdens through decisions made during the design process, especially when considering investment costs. During the design process, carefully examining the requirements of the building and the benefits that can be gained from the passive strategy of natural ventilation ensures that the goal of an energy-efficient building throughout its life cycle is achieved.

* Corresponding author: mimaseyma@gmail.com

The mosque, as a structure made up of a large single volume, is a space where air stratifications and scent accumulations occur, similar to an atrium. Among the expectations of mosque users, hygiene within the structure is paramount. [21] While thermal comfort conditions are more related to the building envelope, the characteristics of indoor air quality are directly dependent on air movement within the volume. One feature that distinguishes this space in terms of user behavior from other buildings is that it is a carpeted area where shoes are not worn. This leads to the accumulation of gases causing unwanted odors, which negatively impact comfort conditions, especially under heavy use. Utilizing simulations that will influence the design of the new mosque structure will provide data for retrofit efforts in existing mosques and will also enlighten designers in choosing energy-efficient strategies. Therefore, research is planned to explore natural ventilation conditions to reduce unnecessary ventilation loads.

To fully examine the effectiveness of natural ventilation strategies, it's essential to conduct CFD analyses that are as close to reality as possible. However, due to the challenges and uncertainties of validating these through on-site measurements or wind tunnel experiments, CFD sensitivity analyses look into how decisions made in the analysis inputs affect the results. Many studies observe changes in findings by making literature-based modifications to CFD analysis decisions (Table 1). This observation questions the accuracy of decisions based on the fraction of results. CFD simulations can vary outcomes according to the effects of fluid properties, flow viscosity, turbulence models, and discretization choices.

The airflow distribution within a space was evaluated numerically through experimental data decades ago, but today, computational fluid dynamics tools are used. With the tools provided by advancing technology, ventilation simulations are conducted during the design phase. The study of fluid movement is based on the Eulerian control volume method in internal flows. As a control volume, the iterative calculations of continuity, velocity in the x, y, and z directions, energy, turbulence, and species equations are dependent on each cell of the flow volume. Additionally, the discretization schemes in the solution system of the equations and the calculation order (first order-second order) in each grid cell affect the accuracy of the results.

Literature-based natural ventilation CFD simulations include various approaches from calculating fluid movement based on the geometry of the space to the decisions involved (Table 1). Many of these studies focus on openings to assess ventilation under wind effects. Depending on the direction of the wind approaching the building facade, the pressure coefficient (C_p) increases as the distance between the openings in the wind direction increases [3]. To evaluate the effectiveness of the openings, air change ratio and pollution concentration values are assessed according to the ASHRAE 62.1 standard. Chang et al. (2020) studied CO concentration based on wind speed within a tunnel and reduced the air change duration by increasing the eddy events in their simulation model where they added a jet fan[4].

In the method of finite volumes, CFD analyses provide scientifically reliable solutions due to their precise results for determining air movement within a space. In this computational model, which considers all parameters of flow during the calculation steps, the primary factors that dictate fluid motion are flow velocity and, consequently, acceleration. The thermal distribution caused by fluid movement, concentrations of pollutant gases, and values of comfort parameters are secondary parameters that need to be evaluated based on the mutual effects of the fluid's speed and the sources of these values.

Table 1. CFD calculation decisions in large-volume geometries in the literature. (comp: computational, expr: experimental, num: numerical, meas: measurement)

research	building type	method	Strategies				induced by		scale		details of simulation
			stack effect	cross ventilation	single sided ventilation	two sided ventilation	buoyancy	wind	small	full	
2018, Qinzi, L.	domain	expr. and comp.	✓	✓				✓	✓		RNG κ - ϵ
2018, Gong, J., Hang, J.	atrium	expr. and num.	✓				✓		✓		
2011, Gao, C.F., Lee, W.L.	residential	comp.	✓					✓			κ - ϵ
2005, Hunt, G., Linden, P.F.	domain	expr. and comp.	✓	✓	✓	✓	✓	✓	✓		RNG κ - ϵ
2005, Cook, M., Li, Y., Hunt, G.	domain	expr. and num.	✓			✓	✓	✓			
2008, Fitzgerald, S.D., Woods, A.W.	domain	expr. and num.	✓	✓							
2013, Stavridou, A.D., Prinos, E.P.	domain	expr. and comp.		✓			✓	✓	1		RNG κ - ϵ
2020, Chang, X. et al.	tunnel	num. and comp.		✓		✓		✓		✓	RNG κ - ϵ
2016, Daish, N.C. et al.	domain	expr. and comp.			✓			✓	1/25		
2011, Walker, C. et al.	model	expr. and num.	✓				✓		1/12		κ - ϵ
2021, Varela-Boydo, C.A. et al.	windcatcher	comp.				✓		✓	✓		κ - ω
2010, Gan, G.	model	comp.		✓			✓	✓		✓	RNG κ - ϵ
2020, Sultansu, S. Ayhan, O.,	model	comp. and meas.									
2006, Asfour, O	religious	comp.		✓		✓		✓	✓	✓	RNG κ - ϵ
2018, Mohammadmirzaei, M.	model	comp. and meas.	✓	✓				✓			RNG κ - ϵ
2019, Gürhan, Ö.	historic	comp.		✓				✓		✓	κ - ϵ
2019, Sapuram, N.,	tall building	comp.		✓				✓		✓	
2009, Mouriki, E.	atrium	comp. and anly.	✓					✓			

In the literature, large structures with atriums, semi-open stadiums, indoor sports halls, and mosques have been the subject of experimental studies. For the atrium volume, when the openings are arranged vertically opposite each other, cross ventilation occurs and stratification can be observed at the levels above the air outlet opening [5].

The mosque architecture has been studied in terms of the effects of dome shapes and openings, as well as the impact of minarets on ventilation within the context of computational fluid dynamics. Different dome shapes and all wind angles (0°, 45°, and 90°) were analyzed at wind speeds of 1 and 3 m/s, revealing that 87.5% of the openings facilitate outward airflow [2]. It was also observed that outward flows increased at 0° and 90° wind angles, while inward flows increased at a 45° wind direction. When the minaret was positioned towards the side of the wind direction, it was noted to increase indoor air speeds [2]. Reda (2022) has analyzed a model involving users in his study. In the simulation studies of mechanical ventilation, they solved the steady-state condition analysis using a second-order discretization scheme [20].

In the literature, experimental, computational, and measurement studies have been conducted on variables like spatial formation, the positioning of openings, the impact of heat sources, the effect of wind towers,

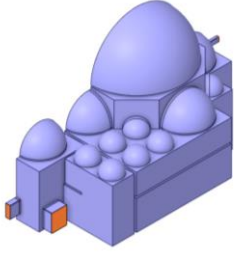
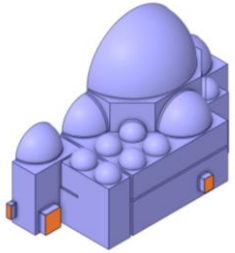
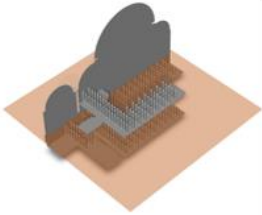
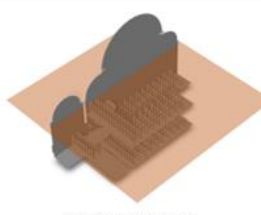
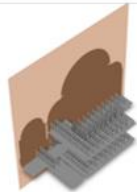
wind and buoyancy effects, the influence of atriums on ventilation, and the connections between rooms. The aim of these studies is to investigate how architectural formation affects energy-efficient ventilation, and they have provided results in this direction. In the studies carried out, the turbulence calculation model used in the analysis approach varies from one to another. Essentially, when simulating the ventilation of a space, the results are entirely dependent on the calculation model. The calculation model, which expresses the mathematical computation of fluid movement, is based on a series of decisions, and at this stage, decisions are made based on properties like fluid, domain, and scale. The accuracy of the conducted studies requires a validated approach in architectural CFD analyses. The accuracy of the approach can be ensured by measurements, or it can be evaluated by looking at the systematic variation of parameters, convergence, and residual values. In this study, results were assessed by trying out two different solvers on two different geometries.

In this study, the differences in the results of analyses with various computational approaches were examined to determine air movement in mosque structures that are similar to atrium buildings in terms of volumetric properties. The aim of this study is to investigate whether a different approach is needed in terms of the CFD calculation model when simulating architectural spaces, compared to mechanical CFD examples. Defining the fluid domain and determining the calculation model in building simulations are crucial for the accuracy of the results. The findings of the SIMPLE and Coupled solver simulations conducted for cross geometry and perpendicular ventilation were compared.

2. METHOD

This study examines large-scale, heavily used religious buildings. Among these structures, mosques were selected as the type with the least area allocated per person (0.72-1.00 m²). Considering the mosque forms applied in our country, the traditional style of mosque typology is commonly seen. Among the projects of the Directorate of Religious Affairs, the Type-7 project, known for its ease of construction and capacity, is chosen for the CFD study in this work. The Type-7 mosque project is a medium-sized mosque with a capacity of 473 people, featuring a half dome and a U-shaped balcony plan, combining the ground floor and balcony level.

Table 2. CFD models and evaluation planes and lines.

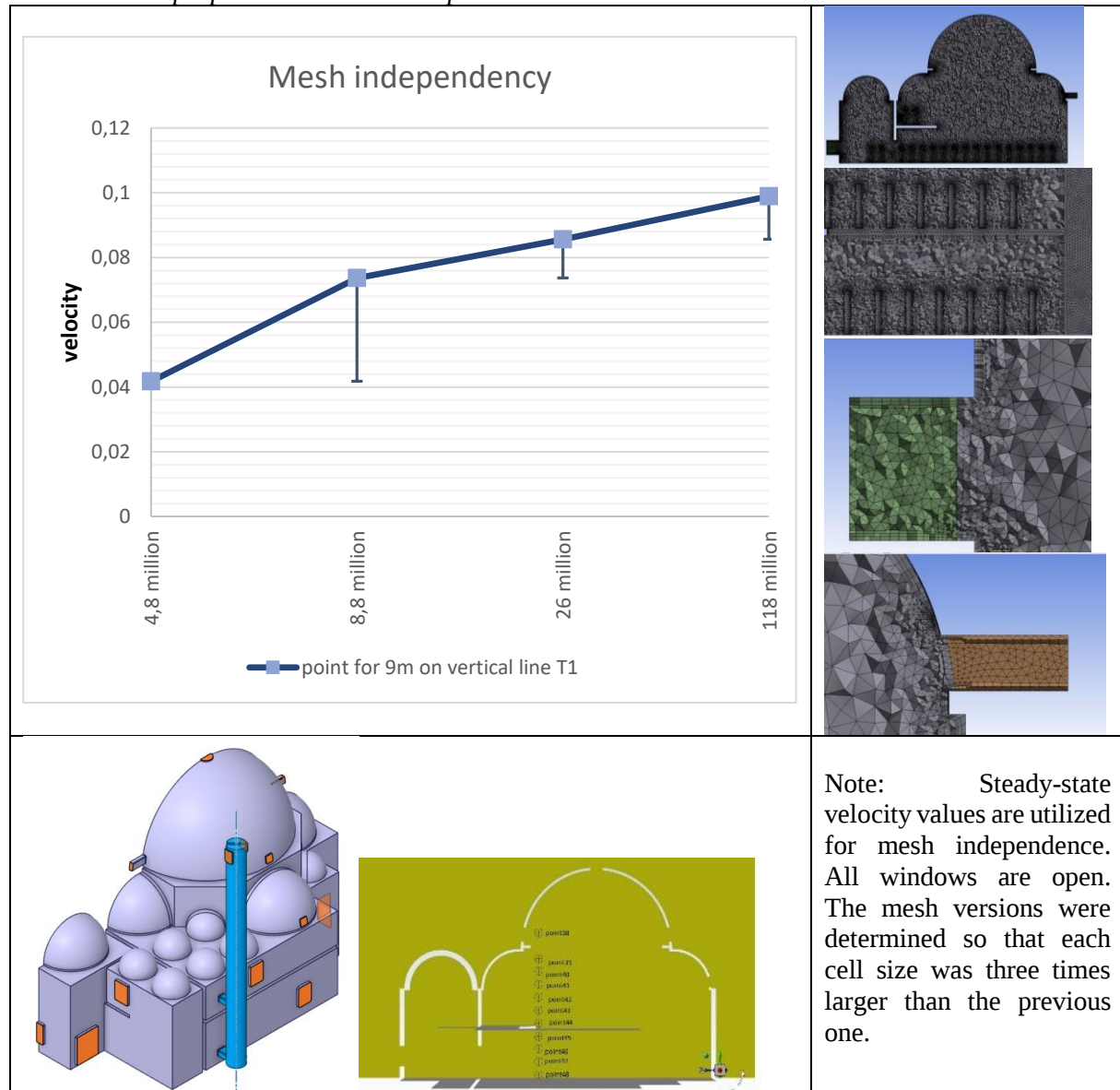
CV2D0S0 model		PV12D0S0 model	
			
Inlet: vent1 (120 x 160 cm)		Inlet: vent1 (120 x 160 cm)	
Outlet: vent3 (80 x 100 cm)		Outlet: vent7 (120 x 160 cm)	
PARAMETER (grafic limits) konfor limits			
	HAİRİM PLAN	MAHFİL PLAN	DD SECTION
			
	AA SECTION	BB SECTION	CC SECTION

The geometric model created for the CFD analysis is symmetrical. Users and shoe racks have been calculated as fully loaded to capacity. A cylindrical model represents the user, and for CO₂ entry, a circular opening has been made, while a square prism represents the cabinets.

The ANSYS Fluent 2022 program was used for CFD analysis. A mesh grid was created using tetrahedral elements, suitable for CFD analysis of complex geometries. The minimum element height was set to 5 cm for the boundary layer. To ensure the flow reaches a stable regime before the vents, an external volume was created at a distance of 1 m outside the window.

For cross ventilation and perpendicular ventilation, the inlet dimension (vent1) is 1.2 x 1.6 m, and the outlet dimension (vent3) is 0.8 x 1 m. For perpendicular ventilation, the inlet dimension (vent1) is 1.2 x 1.6 m, and the outlet dimension (vent 7) is 1.2 x 1.6 m.

The parameters for evaluating CFD cases are temperature, ambient velocity, CO₂ concentration, PMV, PPD, and Draught rate. While PMV and PPD are indicators related to thermal conditions, according to the ISO 7730 standard, the Draught rate is a comfort indicator that depends on air velocity and thermal conditions.

Tablo 3. Mesh properties and mesh independence.

To ensure that the analysis results are independent of mesh size, a mesh grid was selected based on the steady analyses performed with 4.8 million, 8.8 million, and 26 million mesh counts, as there was no significant difference found between the results with 8.8 million and 4.8 million mesh cells.

2.1. Boundary Conditions

Despite the selected project type having been implemented in many parts of the country, this study considers the climatic and geographical features of Ankara. The average outdoor air conditions for the winter months are taken from the TMY (1991) data for Ankara. The ambient pressure is based on the pressure at an elevation of 800 m above sea level, and the fluid behaves like an ideal gas. The outdoor CO₂ concentration is assumed to be 400 ppm. Considering that the building is in a fully occupied state, heat input, and CO₂ inlet have been defined from the modeled users. The building is accepted to be adiabatic, and heat losses/gains from the building envelope and radiation sources are neglected.

Due to the type of mosque layout, the building has been modeled symmetrically. On the entrance facade, a window is defined as an inlet, and on another facade, an opening is marked as an outlet, allowing for the analysis of cross and perpendicular natural ventilation. It has also been assumed that the door is continuously open. All the walls are made of gypsum material, and the transparency of other windows is

ignored. A full occupancy situation has been analyzed in the building, and CO₂ and heat sources have been identified from the users. Heat sources have been defined from the floor and belongings. To evaluate odor problems that affect the perception of hygiene in mosques, ammonia (NH₃) gas is used as a representative gas, considered as an emission source from shoe cups. Fluid materials are assumed to behave as ideal gases. The turbulence intensity for inlets and outlets is set at 20%. Turbulence kinetic energy and turbulence dissipation rate are referred to as K and epsilon, respectively.

Table 4. Boundary condition of cases.

	CV2D0S0-SIMPLE	CV2D0S0-Coupled	PV12D0S0-SIMPLE	PV12D0S0-Coupled
vent3 (60 x 80)	91723 Pa	91723 Pa		
vent7(120 x 160)			91863 Pa	91863 Pa
solver	SIMPLE	Coupled	SIMPLE	Coupled
door (200 x 230)		0,50 m/s		
vent1 (120 x 160)		0,52 m/s		
heat-source (floor)		120 W/m ² , gypsum		
heat-source (body)		48,83 W/m ² , wood		
outdoor temperature		1.6 °C		
indoor initial temperature		21 °C		
pollution source body		0.002553435 kg/s CO ₂		
pollution source shoe cup		4.19075e-6 kg/m ³ -s NH ₃		

In this study, the results of two measurements were analyzed using ANSYS Fluent software. CFD calculations require careful definition of boundary conditions when simulating natural ventilation conditions. In the literature, studies on ventilation influenced by wind have examined the inputs for large building samples, which change depending on wind speed and opening size, despite the selected solver and energy dissipation. (Table 1) Due to the flow being natural, turbulent flow equations were chosen. A relaxation factor of 1 was set for the convergence of the solution. All pressure, velocity, and turbulence discretization schemes are second-order discretization.

The model used tetrahedral cells to create a mesh network, which provides the highest quality mesh depending on the complexity of the geometry. Convergence was achieved at a level of 10⁻³ for continuity and x-y-z velocity and at a level of 10⁻⁷ for energy residual. Time integration uses the explicit method, with a time step of 2 seconds. Fluent analyses ran for 5 minutes.

3. RESULTS

Cross-ventilation and perpendicular-ventilation situations are simulated with SIMPLE and Coupled solvers. Cross-ventilation cases occur through a fluid but perpendicular-ventilation cases occur rotational fluid pattern. According to the coupled solver, three components of the velocity vector are taken into consideration. Consequently, perpendicular cases differ from others for the coupled solver.

Table 5. Velocity, temperature, and CO₂ concentration on central vertical three lines, for CV2D0S0 and PV12D0S0 cases.

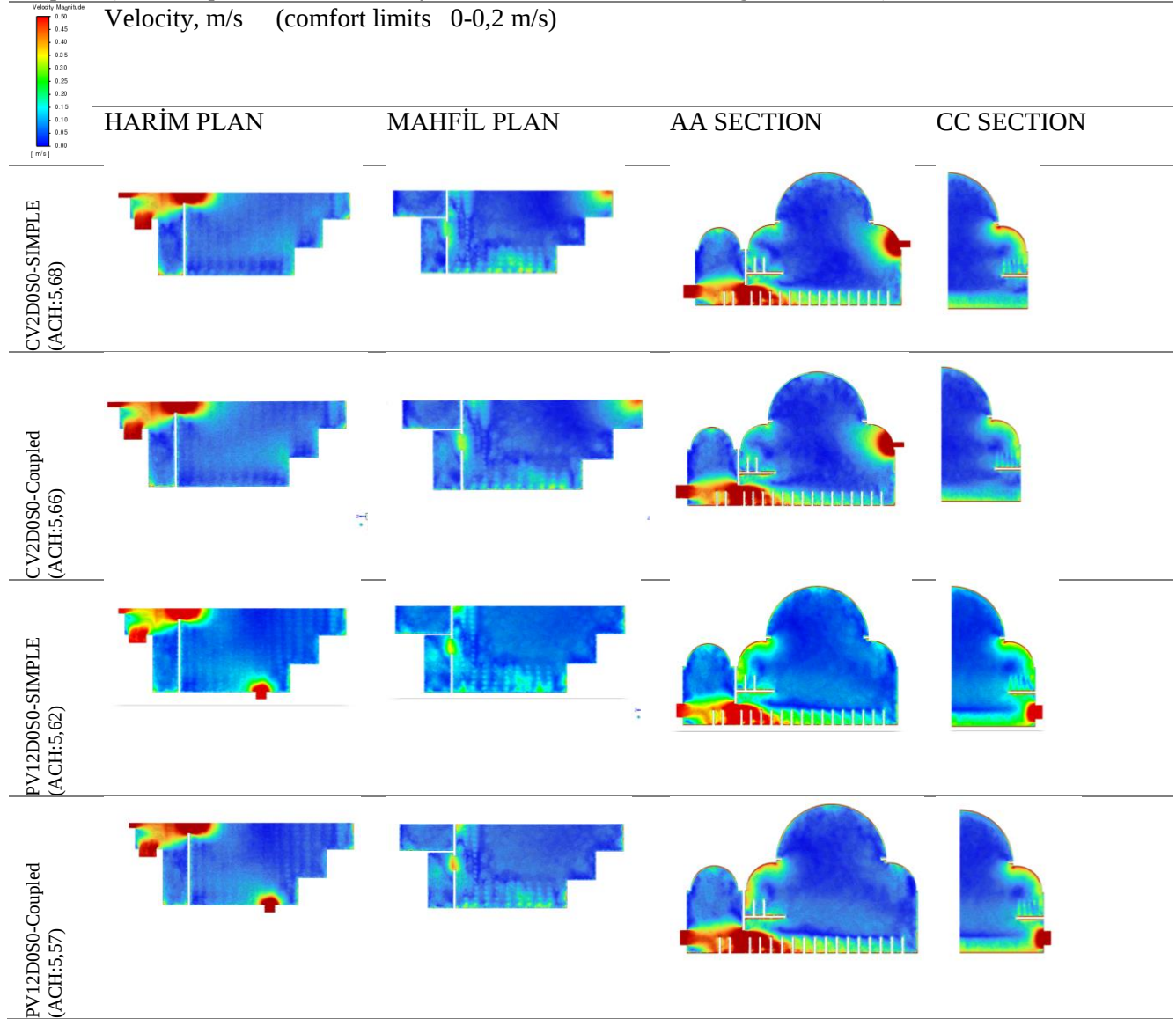


For both geometries, it's expected that the trends of the evaluation parameters on the central vertical lines show parallelism and have similar values to a certain extent. On the central three vertical lines, the CV2D0S0 velocity trendline is anti-parallel to the PV12D0S0 velocity trendline. Similarly, the CV2D0S0 temperature values differ from the PV12D0S0 temperature values over time. In the occupant zone, the CO₂

concentration trendline is the same, but the CO₂ concentration values are distinct from the PV12D0S CO₂ concentration values over time.

Coupled solver cases and SIMPLE solver cases are distinguished from each other because the geometries are 3D in both cases. The velocity value, which is the most important parameter that ensures the flow, differs between the coupled and SIMPLE solvers, and this is crucial for the accuracy of the results.

Table 6. Velocity for cases CV2D0S0-SIMPLE, CVD0S0-Coupled, PV12D0S0-SIMPLE, PV12D0S0-Coupled. (Colormap 0-0,5 m/s and comfort max limit 0,5 m/s according to ISO 7730)



In the SIMPLE solver results for both geometries, the air velocity value is observed at high speeds in a wider region within the volume. In terms of ACH (Air change per hour), it is seen that the SIMPLE solver has a higher value depending on the air velocity values in the vents. The fact that the opening is close to the areas where the users are located creates unfavorable conditions in terms of comfort conditions.

Table 7. CO₂ concentrations for cases CV2D0S0-SIMPLE, CV2D0S0-Coupled, PV12D0S0-SIMPLE, PV12D0S0-Coupled. (Colormap: 400 ppm – 1500 ppm and comfort max limit 1000ppm according to ASHRAE and TS 16798-1)



In the SIMPLE solver results where the air velocity values are higher, it is understood that in the CV2D0S0 analysis, the air flow exits from the outlet without reducing the concentration in the space due to the mutual location of the openings. However, for PV12D0S0 case, CO₂ concentration is lower in SIMPLE solver results where ACH and velocity values are higher. The fact that the airflow is perpendicular, not linear, and has a perpendicular direction has led to a lower CO₂ concentration inside the structure.

For CO₂ concentrations, the difference between the results for the two solvents increases over time, and for longer periods of use, such as 20 minutes, the final CO₂ concentration value will be quite different.

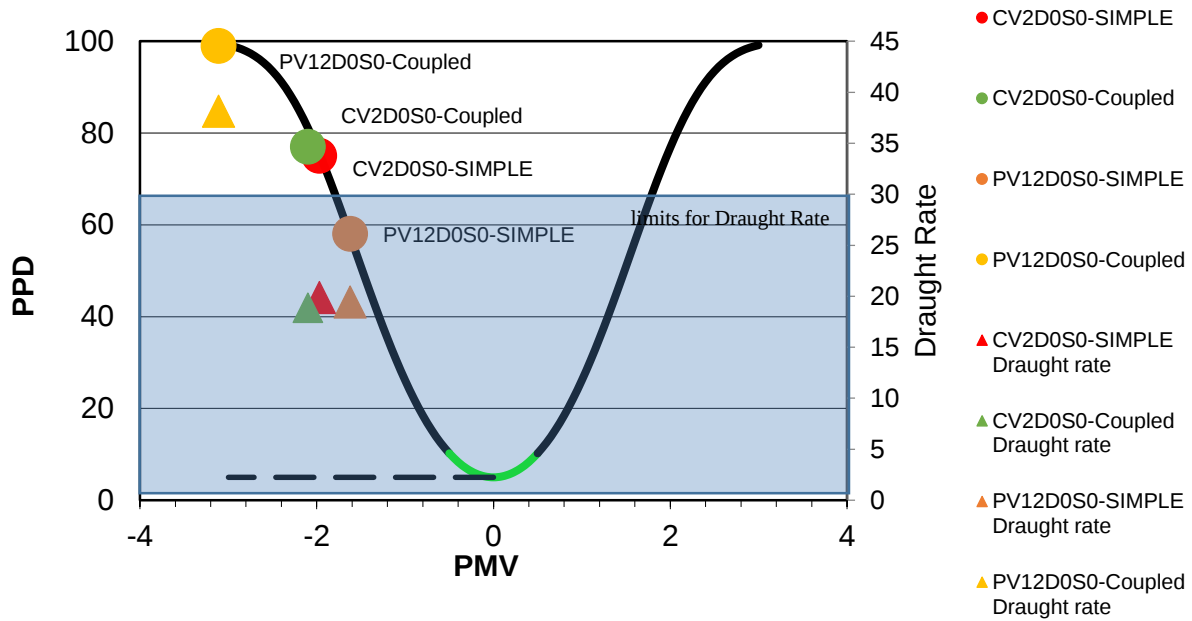


Figure 1. PMV, PPD, and Draught rate (According to ISO 7730)

In the analysis conducted for winter conditions in Ankara, using a boundary condition of 1.6°C as the average for the winter months from TMY (1991) resulted in PMV and PPD values being far from comfort conditions. Additionally, the Draught rate value for the CV2D0S0 Coupled and SIMPLE cases meets comfort conditions, whereas the PV12D0S0-Coupled case has a high draught rate. The high draught rate in the PV12D0S0-Coupled case is due to the parameters it depends on temperature and air speed.

Draught rate (DR) indicates the airflow rate in the environment and is a type of air movement that is particularly bothersome for people standing near windows in communal areas. Defined in ISO 7730, this parameter is calculated based on the air speed, turbulence intensity, and ambient temperature, and it tends to have a value that contrasts with thermal comfort (PMV) during winter conditions. As the natural airflow increases, the ambient temperature decreases. The PV12D0S0-Coupled case represents the most unfavorable example in terms of DR and PMV. Other examples have a DR value below the 30% limit.

4. DISCUSSION

The varying appearance of results dependent on the solver variable highlights the need for careful decision-making regarding the computational model in CFD analyses. When looking at the average values of three vertical lines within the space in Table 5, both the Coupled solver and the SIMPLE solver are consistent in terms of the trend of CO₂ concentration. The difference in trend between the velocity and temperature values, as well as the fluctuation in the velocity value at the 2nd minute, indicates that the SIMPLE solver is not stable. In the case of CV2D0S0, an irregular graph for the velocity value is observed, while in the PV12D0S0 case, fluctuations are noted in the temperature value. In CFD literature, achieving convergence and stabilization throughout repeated solutions reflects the appropriateness of the computational settings. However, the SIMPLE solver does not show stable results in the examples presented in this study.

In Table 6, the case with the lowest air change rate is the PV12D0S0-Coupled case. Comparing the air change rate with CO₂ concentration values in Table 5, the lowest example of ACH corresponds to the highest CO₂ concentration in Table 7.

Stavridou and Prinos, in their 2013 study, demonstrated that air layering occurs at the elevations above the outlet. The results of this study show that in areas where the airflow is minimal, meaning where the air

speed is low, there is a buildup of CO₂. In this study, the CV0D0S0 case outlet is located above the 1st floor, while the PV12D0S0 case outlet is at the ground floor. For the CV0D0S case, when the inlet and outlet are chosen at different levels and in opposing locations, the flow pattern within the volume becomes linear. In the PV12D0S0 case, since the inlet and outlet are on the same floor but on perpendicular facades, the flow cannot facilitate the exchange of air in the upper levels, resulting in a high CO₂ concentration. Therefore, alongside the heights of the inlet and outlet, the flow pattern is significant. Especially in large volumes, areas that the flow cannot reach may create unfavorable environments regarding indoor air quality (IAQ) and thermal comfort.

In Figure 1, there isn't a relationship between the DR and PMV values, but both are indoor comfort parameters and are conversely positioned regarding comfort limits for samples. Under winter conditions, when the airspeed and turbulence are increased, as seen in the example PV12D0S0, the DR rate rises significantly while the PMV thermal comfort drops drastically.

5. CONCLUSION

The results showed that Solver settings play an important role in the prediction of the fluid patterns. As seen in the two examples where the fluid motion is linear and rotates in the 3D volume, there are significant differences, especially in the air velocity parameter. Although the parallelism of the results for temperature and CO₂ concentration is understood as the similarity of these two solvents, the most important parameter in indoor flow pattern determination is the velocity value. In CFD analysis, solver settings depend on the properties of the geometric model. Whether the flow is in two-dimensional or 3D flow volume affects the solver settings.

The reason for these differences is that in 3D geometries, the vectors of the flow vector in three directions in the Cartesian coordinate system affect the acceleration in all three directions. The use of the Coupled solver, which takes into account the effects of these vectorial values on each other in the calculations, will affect the accuracy of the results. With the Coupled solver, the residual graphs show a more stable trend.

Depending on the temperature, both PMV and PPD values and draught rate values are outside the comfort limits at the end of 20 minutes. It is seen that natural ventilation does not provide comfort limits in winter conditions, but it is thought that the expected standards can be met when the outdoor air temperature is close to the comfort temperature.

The CO₂ concentration was found to be slightly affected by the solvent difference, as it increases in the space due to each user, as a constituent in the fluid, due to the mass increase.

In each case in this study, the air velocity in the area close to the opening is higher than the comfort level and users in this area may experience negative space comfort due to drafts.

Air velocity was found to be the most important factor affecting the flow pattern and the distribution of concentrations and temperature.

The fact that the evaluation parameters have opposing effects necessitates conducting a significantly higher number of simulations and diversifying the variables to achieve optimal conditions. In this study, in order to further advance the investigations related to opening position and the solver, it is essential to determine suitable and accurate approaches for CFD solutions by conducting research based on variations in opening sizes, air velocities, and climate temperatures. Additionally, turbulence approaches, which have been extensively studied in the literature, need to be evaluated in the context of large volumes and differing opening positions. As a result, if CFD analyses are applied carefully in spaces, with a thorough examination of computational models and accurate interpretation according to the characteristics of the problem, they can be considered a reliable prediction for natural ventilation.

REFERENCES

- [1] URL-1. DOE Report. High Efficiency HVAC Systems, URL: <https://www.psoklahoma.com/savings/newsletter/story?StoryID=1727> Last Accessed: 01.12.2024
- [2] Asfour,O.S. (2006) Using CFD to Investigate Ventilation Characteristics of Domes as Wind-Inducing Devices in Buildings, *The University of Nottingham*, PhD Thesis, United Kingdom.
- [3] Daish, N.C., Carrilho da Graça, G., Linden, P.F., Banks, D. (2016) Impact of aperture separation on wind-driven single-sided natural ventilation, *Building and Environment*, 108, 122-134.
- [4] Chang,X., Chai,J., Liu,Z., Qin, Y., Xu, Z. (2020) Comparison of ventilation methods used during tunnel construction, *Engineering Applications of Computational Fluid Mechanics*, 14:1, 107-121.
- [5] Stavridou, A.D., Prinos,E.P. (2013), Natural ventilation of buildings due to buoyancy assisted by wind: Investigating cross ventilation with computational and laboratory simulation, *Building and Environment* 66, 104-119.
- [6] Qinzi, L. (2018) Modeling of Opening Characteristics of an Atrium in Natural Ventilation, Master Thesis, *Massachusetts Institute Of Technology*.
- [7] Gong,J., Han,J.(2018) Buoyancy-driven natural ventilation in one storey connected with an atrium, *International Journal of Ventilation*, 18:4, 281-302.
- [8] Gao, C.F., Lee, W.L.(2011). Evaluating the influence of openings configuration on natural ventilation performance of residential units in Hong Kong, *Building and Environment*, 46, 961-969.
- [9] Fitzgerald, S.D., Woods, A.W. (2008) The influence of stacks on flow patterns and stratification associated with natural ventilation, *Building and Environment*, 43, 1719–1733.
- [10] Hunt,G., Linden, P.F.(2005) Displacement and mixing ventilation driven by opposing wind and buoyancy, *Fluid Mechanic*, 527, pp. 27–55
- [11] Gao, F., Wang,H., Wang,H. (2017) Comparison of different turbulence models in simulating unsteady flow, *Procedia Engineering*, 205, 3970–3977.
- [12] Gan, G., (2010) Interaction Between Wind and Buoyancy Effects in Natural Ventilation of Buildings, *The Open Construction and Building Technology Journal*, 4, 134-145
- [13] Sultansu,S., Onat,A. (2020) The CFD Analysis of Ventilation and Smoke Control System with Jet Fan in A Parking Garage, *International Journal of Advances in Engineering and Pure Sciences*, 1: 89-95.
- [14] Varela-Boydo,C.A., Moya, S.,L., Watkins,R. (2021) Analysis of traditional windcatchers and the effects produced by changing the size, shape, and position of the outlet opening, *Journal of Building Engineering*, 33:101-828.
- [15] Walker, C., Tan,G., Glicksman,L.(2011) Reduced-scale building model and numerical investigations to buoyancy-driven natural ventilation, *Energy and Buildings*, 43, 2404–2413.
- [16] Mouriki,E.(2009) A Thesis in the Department of Building, Civil, and Environmental Engineering, Thesis, *Concordia University*, Canada.
- [17] Gülhan,Ö. (2019) Natural Ventilation Design For Historic Libraries With Cfd (Computational Fluid Dynamics) Simulation, Master Thesis, *İzmir Instutie of technology*, İzmir, Türkiye

- [18] Sapuram,N.(2019) Natural Ventilation In Tall Buildings Development of Design Guidelines Based on Climate and Building Height, Master Of Building Science, *University Of Southern California*.
- [19] Mohammadmirzaei, M. (2018) Numerical studies of turbulence effects in cross-flow Ventilation, Master Of Degree, *San Jose State University*, California.
- [20] Reda, İ. , AbdelMessih, R.N., Steit M. , Mina E.M.,(2024) Thermal performance of domed roof in air-conditioned spaces, *Energy and Built Environment* 5 (2024) 270–287.
- [21] Topraklı,A.Y., Işıklar Bengi,S.(2019) Camilerin İskân Sürecinde Değerlendirmesine Yönelik Bir Yaklaşım An Approach To Post-Occupancy Evaluation Of Mosques, *Uluslararası Sosyal Araştırmalar Dergisi*, 12;64.
- [22] Cook, M., Li,Y., Hunt, G. (2005) CFD Modelling of buoyancy-driven natural ventilation opposed by wind, *Ninth International IBPSA Conference*, Montréal, Canada.