Characteristics of Wind Flow Around a University Campus Settlement Estimated by CFD Simulation

Yasin Furkan Gorgulu (yasingorgulu@isparta.edu.tr)
Isparta University of Applied Sciences
Aysegul Hazer
Istanbul Health and Technology University

Research Article

Keywords: Building aerodynamics, CFD, structural orientation, turbulence formation, thermal analysis, wind load

Posted Date: July 25th, 2022

DOI: https://doi.org/10.21203/rs.3.rs-1849193/v1

CHARACTERISTICS OF WIND FLOW AROUND A UNIVERSITY CAMPUS SETTLEMENT ESTIMATED BY CFD SIMULATION

Yasin Furkan Gorgulu[1], Aysegul Hazer[2]

Abstract: Performing a computational fluid dynamics (CFD) analysis of a settlement can present many parameters such as determining the wind load, predicting the temperature distribution, determining the turbulence formations about how that settlement should be located, to the scientists in the architectural and engineering departments. In this study, a CFD analysis has been done using a commercial analysis program "Ansys Fluent" and the geometry has been created using an architectural program known as "SketchUp". Istanbul Health and Technology University Merter Campus buildings have been taken as the reference models. Streamlines, velocity, pressure, temperature and turbulence kinetic energy distribution contours have been projected. The highest flow velocities and turbulence density were observed between the campus buildings, and when the temperature distributions were examined, dramatic decreases were found in the building temperatures on the windward facade. The maximum wind loads, temperature distributions and so on have been detected and according to the results, recommendations have been made.

Keywords: Building aerodynamics, CFD, structural orientation, turbulence formation, thermal analysis, wind load.

1. INTRODUCTION

With the rapid advancement of technology and great developments, looking for sustainable and durable buildings increasingly occupies the disciplines of architecture and engineering in today's world [1]. In accordance with this, climate parameters such as wind load estimation, thermal comfort, and the design of ventilation systems in the building design project should be detailed with a number of interdisciplinary studies. Therefore, numerical simulation and computer analysis are used in the most advanced approaches to microclimate predictions for building complexes [2]. It is main aim is to resolve the tasks of CFD simulation (Computational Fluid Dynamics) with the aid of software complexes. Use of the numerical simulation can reduce the cost of catastrophic mistakes and speed up problem-solving [3].

Recently, among the renewable energy sources, wind energy has become one of the most significant. Having many architectural environmental comfort attributes, wind contributes a considerable percentage of energy efficiency in terms of building sustainability [4]. Since wind can naturally ventilate

---

[1] yasingorgulu@isparta.edu.tr, Department of Machinery and Metal Technologies, Isparta University of Applied Sciences, 32700, Isparta-Turkey.

ORCID: 0000-0002-1828-2849

[2] aysegul.hazer@istun.edu.tr, Department of Architecture, Istanbul Health and Technology University Engineering and Natural Sciences Faculty,34173, Istanbul-Turkey.

ORCID: 0000-0001-5025-0743

the building and provide passive cooling, it should be taken into account in the early stages of the architectural design phase. However, this climatic parameter can not always be building-friendly. If the wind blows at too high a speed over and around a building, pedestrians can feel uncomfortable, and it can also cause energy loss. Especially given the recent high demand for tall/high buildings, there has obviously been a growing interest in calculating wind load impacts on buildings. In high-rise buildings, in order to satisfy structural safety, one of the dominant design requirements is the wind load calculation. Because the velocity of the wind is directly proportional to the height of the building, the horizontal vibration is felt more on the upper floors of tall buildings. This condition can result in adverse effects on user comfort. Moreover, human nature can respond to that motion with psychological discomfort. Excessive building motion can also cause noise and damage non-structural elements like curtain walls, which can predispose them to breakage and reduce their life expectancy [5].

A building design project is a difficult task with high risk, and it is significant to get accurate analysis results before project construction. In these circumstances, CFD analysis has shown to be an excellent instrument for architects and engineers to anticipate the performance of their structures under various situations, reducing uncertainty and allowing them to make logical decisions during the early stages of the design process. Project costs are determined very early in the design process, it is critical to make consistent decisions. A well-tested and well-studied design may result in reduced energy usage and longer-lasting results, as well as reduced failure risk [6]. These critical considerations involve a variety of internal and outside design concerns, such as wind load calculation, safety and pollution management, and providing thermal comfort within the structure as well as pedestrian comfort [7].

Wind-related concerns increase as tall structures and skyscrapers grow more complicated in terms of overall design and size. Even ordinary building designs must account for wind loads in particular high-wind regions, such as coastal sites. Wind analysis is the process of determining the influences of wind on a building or structure and creating designs to minimize these effects. By using wind engineering research and considering building aerodynamics, design engineers and architects may assure a cost-effective, safe, and sustainable design. When analysing the impacts of wind load on building designs, architects and engineers take into account two key factors [8]:

1.1. Aerodynamic Modifications to the Shape of the Buildings

Many factors influence the wind conditions around a building, including ambient wind statistics, local topography, building mass, nearby foliage, and closeness of similarly tall structures. All of these factors can have a significant impact on the winds that blow around the base of a new constructing building. However, the most important of these factors may be the building mass. The volume has a critical role in wind flow surrounding construction because its aspect ratio changes the pressure distribution around these structures. Also, the geometry of the building is inducted by buffering and torsional vibration, which has great importance for the pressure zones and levels that are created by the airflow around the building. This also affects the aerodynamics of the building. Skirting vortices are caused by air molecules that are pushed by each other and have different wind velocity values. Different regions occur around a building due to the variety of wind directions and velocity. This condition can be enhanced with the studies to be done at the design stage [9].

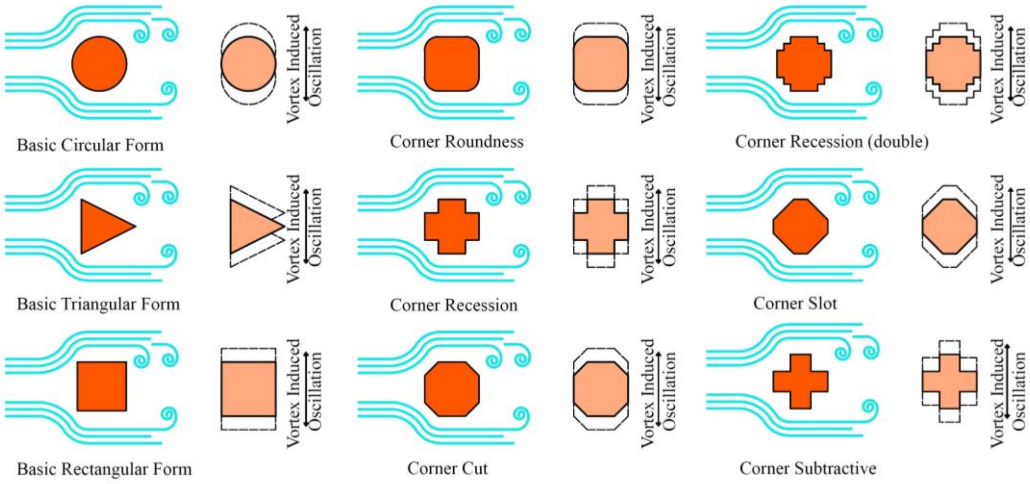

Figure 1.1. Comparison of vortex-shedding performance of buildings cross-sections [10].

Conventional cross-sections of buildings are generally derived from a circle, triangle, and square shapes sequentially. Figure 1.1. shows other geometric plan shapes derived from these shapes and the vortex-shedding formation surrounding them. Also, Figure 1.1. demonstrates openings or through openings, which prevent the formation of large vortices. These help stabilize the across-wind load and decrease the drag force by reducing the wake area. Modifications to the building's basic plan form or a more three-dimensional design on the top level can lessen self-induced vortex shedding pressures. Daemei and Eghbali [11] used Autodesk Flow Design software to do a numerical simulation to see how different alterations affected the lowering of the wake region behind high-rise buildings. Rounded, chamfered, and recessed corners were among the alterations made to triangle and square structures. In comparison to the original building, the study suggested that the wake region might be decreased by 30–50%. In that context, buildings must be designed with a more streamlined shape and less decoration to reduce buffeting loads [8].

At present, vertical architecture designs have been commonly adopted to fulfil the demands of the continually growing urban population. As buildings become taller and with advances in construction techniques resulting in more flexible buildings and sculptural forms, they become more susceptible to large-amplitude wind-induced vibrations [8]. According to the Council of Tall Buildings and Urban Habitat (CTBUH), an organization used for collecting and storing tall building databases, the number of tall buildings has an increasing tendency, with an addition of 175 tall buildings in 2020 [12]. Due to this rapid increase in the number of tall buildings, the main wind directions can change at the urban scale. Therefore, the geometry of buildings in urban areas is far more complicated than in open rural settings, and it has a substantial impact on wind flow at the micro-meteorological scale [13].

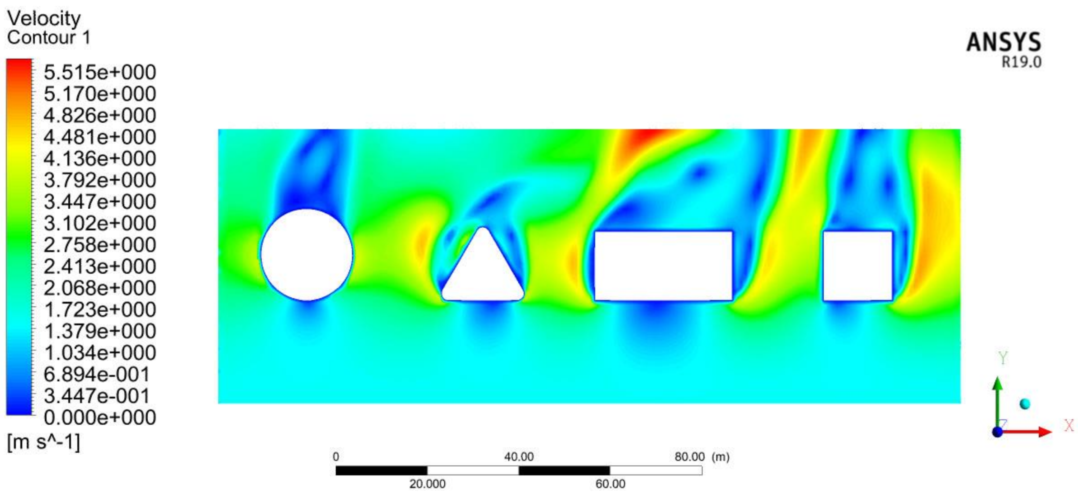

Figure 1.2. Air flows from buildings with different geometric cross-sections acting on each other.

For instance, in Figure 1.2., having different geometrical plan shapes for buildings on an urban scale, the effects of airflow around the buildings and wind velocity on each other are shown with the Ansys Fluent software. Because of their aerodynamic, architectural, and structural advantages over other shapes, square and triangle cross-sections are two of the most prevalent cross-sections used in tall building design. The figure 1.2. shows a few well-known cross-section; square, rectangle, triangle, and circle shapes. In previous research, aerodynamic adjustments in the shape of squares and triangles have been demonstrated to diminish wind-induced responses in both along and across-wind directions. These adjustments show that the corners are the most important portions in improving a tall building's aerodynamic performance. Aside from the alterations already discussed, several studies show that the aspect ratio of tall buildings is another factor that changes wind characteristics by changing the pressure distribution around these structures [8]. As a result, the wind should not be considered on a single-building scale. Built environment, building geometry, urban climate, and thermal comfort conditions should all be considered when developing the project.

In the field of wind engineering, many scholars have studied airflow around high-rise/tall buildings, but it is relatively deficient in low-rise buildings investigated [14]. In the case of low-rise buildings, generally roof shape is mentioned. Because low-rise buildings always have a blunt body characteristic, leading to highly complex airflow around them. On the rooftops, for example, vortices of various sizes will emerge, such as destructive separation bubbles and conical vortices. Low-rise building aerodynamic difficulties have gotten attention and scientific investigation in recent decades. However, further research is needed to see how changes in roof design impact the airflow field surrounding low-rise buildings [15].

1.2. Pressure Loads and Facade Design

Wind pressure is an important design output parameter for analyzing the reaction of all facades to wind loads. Because of the exponential rise in wind speed, wind pressure over the windward facade increases with building height. Wind pressure is affected by a variety of factors, including building size and shape, the built environment, and wind characteristics [7]. This mostly entails a steady analysis to identify areas of maximum and minimum pressure values that would be subjected to greater forces and may require strengthening for safety. While basic coding methods may typically be used to determine pressure loads for simple structures, complicated geometries require comprehensive numerical analysis or wind tunnel testing to provide reliable results.

Building envelopes are currently built to resist severe load scenarios, which necessitates the use of bigger and bulkier constructions. This modifies the facade design based on the local conditions. The result is a textured, dimpled building surface, with the ultimate shape determined by the wind direction and force [16]. Wind analysis may be seamlessly integrated into the design process thanks to ever-improving digital design and analysis technologies. Fast decision-making is critical in the early stages of conceptual design. When it comes to assessing a building's wind performance, CFD models may assist in evaluating wind-architecture interactions and, as a result, generating wind-formed designs. Due to the negative pressure, intense suction forces develop on wide facades when the narrow facade of the structure is positioned towards the direction of the wind. Strong pushing effects occur on the wide facade of the building when it is positioned in the direction of the wind. Negative pressure, on the other hand, causes powerful suction effects on the opposite side [9]. Some of the most common design modifications that may be used to mitigate wind impacts operate by minimising or suppressing vortices [17–19]. Some designs in building facade mass to decrease wind loads are illustrated in Figure 1.3.

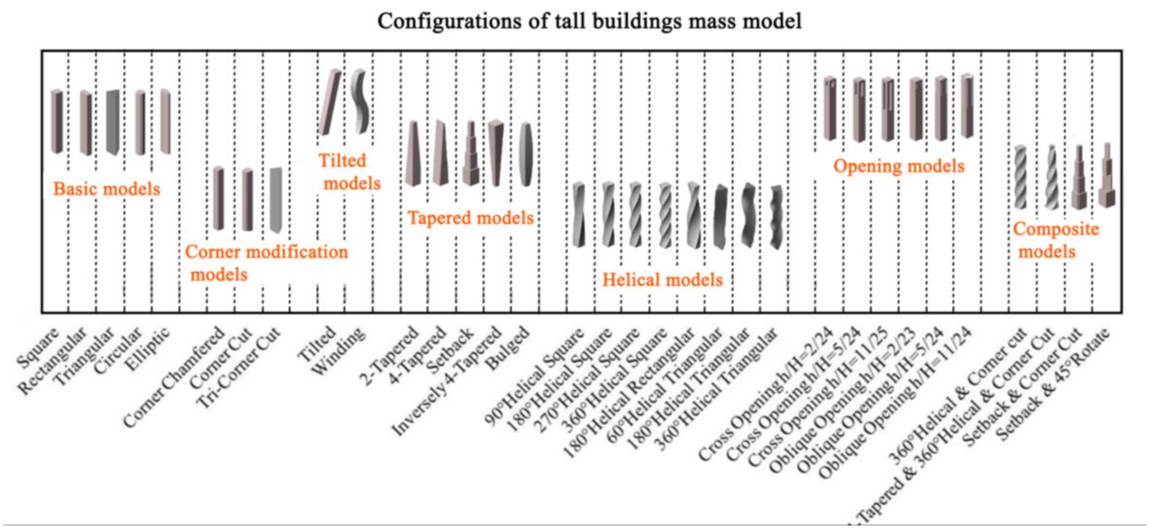

Figure 1.3. Typical configurations of tall buildings mass model [17].

In recent years, with high performance criteria, curtain wall systems have been installed in almost every major city. One of these performances is already resistance to wind pressure and vibration. To suppress this undesired vibration a number of mitigation approaches such as application in damping and double facade systems have proposed. Especially, the application of double facade has gained considerable attention.

According to the was tested by Yuan et al [20], with assemble of vertical splitter plates in attached on building facade, negative peak pressure on curtain wall can mitigate by %42.

The first structural element to be subjected to wind loads is curtain walls. Wind loads can influence the structural design of curtain walls in a variety of ways, including bluff body aerodynamics analysis, application of appropriate regulations, judicious assessment of material geometry and composition, and consideration of manufacturing and positioning constraints. Differential column shortening, lateral story drift, building racking, slab and beam edge deflections, and building vibrations are all movements and deformations that should be addressed while constructing the curtain wall systems. To guarantee that the facade functions as planned, strong coordination between the facade engineer and the structural engineer becomes a priority. Panel fallout, connection failure, excessive glass rotation inside the frame, and uneven joints are all examples of failure modes caused by a lack of coordination. To prevent these faults from occurring, the delicate assembly of curtain wall elements called brackets is required [21].

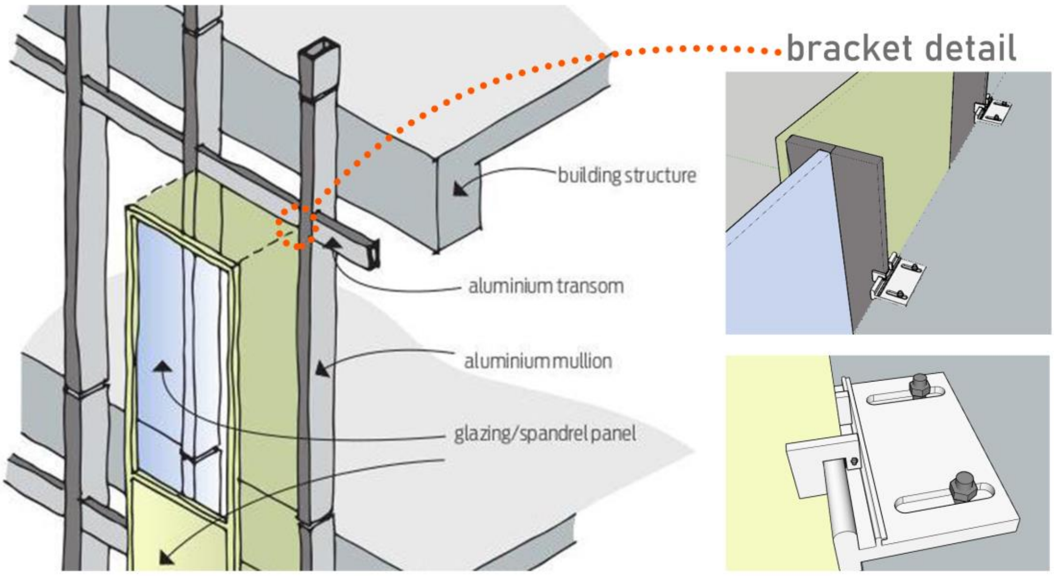

Figure 1.4. Stick system curtain wall and unitized curtain wall panel assembly detail [22].

2. MATERIALS AND METHODS

In this study, the simulation was carried out with two different software packages. The first of these is architectural three-dimensional (3D) modelling program Sketch Up with V-Ray rendering plug-in. By means of this program, the geometry of the building was created in coherence with the real dimensions. Afterwards, the designed mass was transferred to Ansys Fluent software, which is a fluid simulation program. With the help of CFD method, the program can be used to make more accurate forecasts about air flow, which is also the subject of this study.

Based on CFD computation, part one of this project was fine-tuned according to the actual components and material specifications of aluminium spandrel panels. Part two of this project mainly investigated curtain wall wind load with the help of models developed.

Flow velocity measurements have been taken with the help of an anemometer at a height of 5 meters from the ground throughout 2021, the annual average air velocity was taken into account to be used in the analysis and this value was taken as approximately 1.5 m/s. The received data are in agreement with the data from meteorology. In addition to these data, flow velocities, temperatures and wind directions of Istanbul for 2021 were taken. The wind directions of Istanbul Merter on month and hour basis are given in Figure 2.1. Each color represents different compass directions. The shaded overlays indicate night and civil twilight.

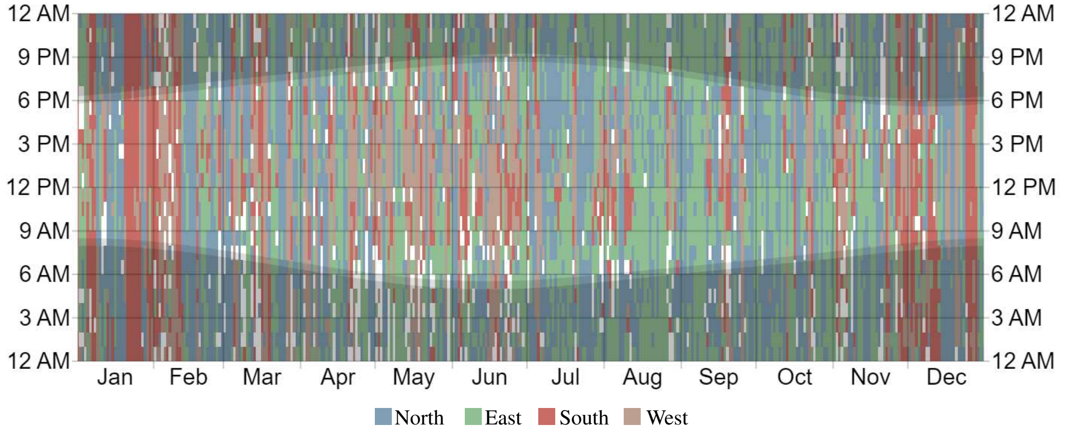

Figure 2.1. Istanbul's hourly wind directions in 2021 [23].

The flow velocity taken in the simulation is the average of the data taken during the year 2021. Figure 2.2. depicts the flow velocities of Istanbul according to the months. The maximum flow velocities reached during the day are shown with red bars, while the remaining flow velocities are shown with gray bars. As can be seen, the values obtained with the anemometer and the data obtained from the meteorology overlap.

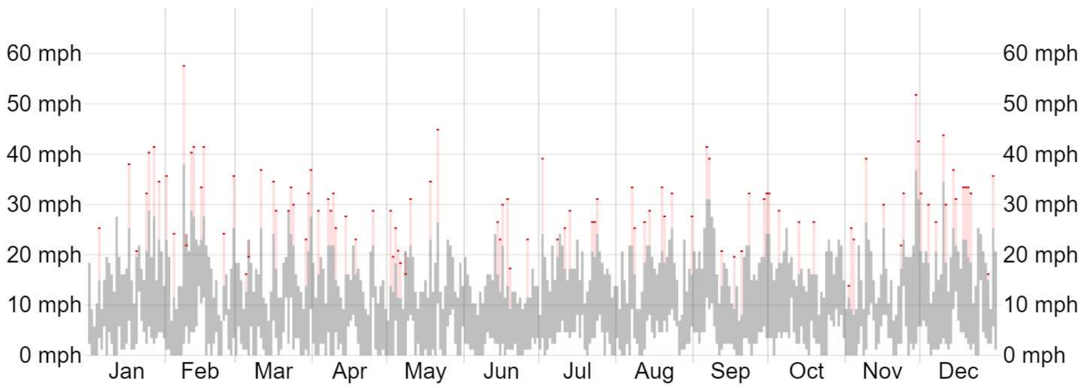

Figure 2.2. Istanbul's wind speeds in 2021 [23].

The temperature value of the simulation air flow was taken as approximately 16 degrees Celsius. This value has been recieved by taking the average of the data taken throughout the year. Figure 2.3. shows the distribution of taken air temperature data by month. When the temperature data and meteorological data are compared, it is seen that the data also match with each other [24].

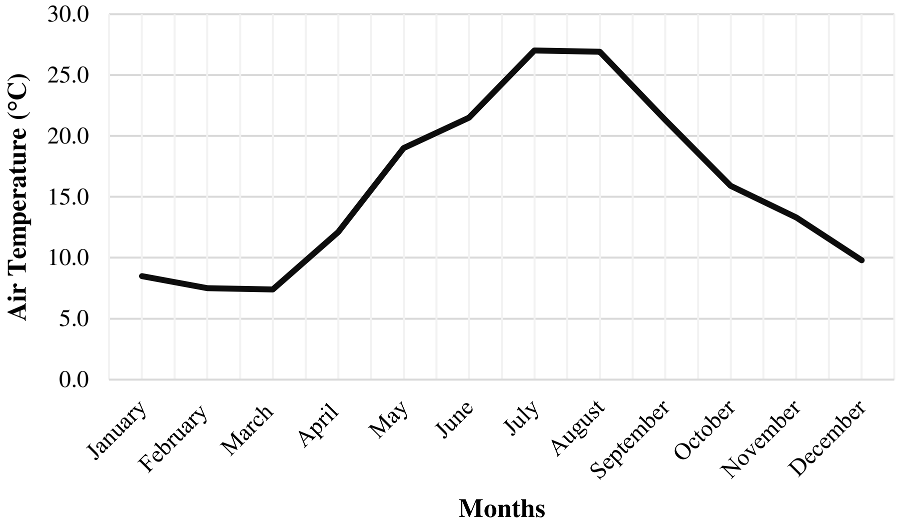

Figure 2.3. Average air temperatures of Merter Campus.

2.1. Campus Settlement Design

The subject of this study is the Merter Campus of the Istanbul Health and Technology University, located in the Gungoren district of Istanbul. As it is shown in the Figure 2.4., the campus has two separate buildings located on an area of 2,186 square meters. Having almost identical architectural floor plans, the buildings are a distance of approximately 16 meters apart, and prevailing winter winds blow from the northeast between them.

Figure 2.4. Campus settlement and satellite view.

In this connection, the analysis of the wind flow to determine its effect on the courtyard microclimate is important in terms of ensuring user comfort. In addition to this, calculating the airflow characteristics (wind pressure distributions, velocity distributions and turbulent dissipation) in and around the building can help minimise energy losses. For this purpose, SketchUp software was used for creating 3D building geometry. İnitially, the contours of the 3D models of buildings were obtained from AutoCAD files. Because CAD objects are not directly supported by SketchUp, the CAD file was cleaned up and the appropriate file format imported into SketchUp. Subsequently, the built environment construction was taken from the master zoning plan and was included in the model by volume with the help of aerial photographs (see Figure 2.4.).

Buildings which are located within the campus boundary are roughly rectangular shape in plan. The volume of buildings is roughly cuboid. They are cantilever structures, 1 meter each from the second floor. The envelope of buildings is a stick curtain wall system that transmits light planar. The building in the northwest is known as the block E, which located health sciences faculty, it has 8 floors and is approximately 27 m high. The ground floor of block E serves as a commercial area, so this floor has a glass-clad facade together with an upper floor. The other upper floors of the building make up an aluminium composite curtain wall system. A storey of curtain wall frames has been divided at half rate into vision glass and spandrel panels. The vision panels function as windows and can open transom. They have been used to hide the parapet height in our case study buildings also. The building in the southwest is known as the block D, which located engineering and natural sciences faculty, it has 7 floors and is approximately 24 m high. A part of the ground floor in the block D is used for commercial purposes, the rest of all architectural characteristics, such as floor plans and facade system, are similar as in the block E. In addition to aerial and real photographs of the blocks, three-dimensional models of the buildings have also been designed and rendered in Figure 2.5. for the readers to understand more clearly.

Figure 2.5. Campus settlement 3D model design with V-Ray rendering plug-in and illusturateded with Adobe Photosop software.

It is relevant to use at the present time numerical simulation to determine wind loads on buildings and to investigate wind flows time in the built area. This method provides for a large reduction in the expense of correcting mistakes while also speeding up the problem-solving process. Therefore, modern methods of real-time Computational Fluid Dynamics (CFD) data visualisation were chosen in order to ease the interpretation of simulation results.

2.2. Computational Fluid Mechanics (CFD)

In order to create a model exposed to the changing wind conditions requires a sequential analysis from the local conditions to the global building area, assessing their behaviour in the wind. First, live physics simulations have to be employed to obtain the shape change caused by the acting wind. Second, building mass is tested in the CFD plug-in. CFD simulations consist of three main parts. These are the computational domain, meshing and boundary conditions for inlet, outlet and other surfaces [9].

In this paper, initial a volumetric mesh of the geometry of interest and its surroundings was created. When modeling the flow domain, that is, the computational domain, it was created at a height of 30 meters from the sides of the buildings and 20 meters from the highest ceiling.In this domain, merely two buildings of Istanbul Health and Technology University was located. The prevailing wind direction was coded as northeast and 1.5 m/s. After the domain was created, building mass was transferred from the file drawn in SketchUp software. Secondly, the meshing process on the building geometry was started. In meshing, because of the grid quality and quantity ascertaining computation time and results, a hexahedral grid type was chosen in the prediction of results. The average skewness rate range recommended by Ansys is known to be less than 0.95 [25–27]. The mean skewness value for the simulation is 0.23, which is described as "Excellent" for Ansys. For orthogonal quality, 0.7-0.95 is "Very Good" quality, and this value obtained in simulation is 0.76. Mesh views are given in Figure 2.6. Model dimensions were generated to 1/1 scale. The overall count of nodes and elements are 248,147 and 1,324,710.

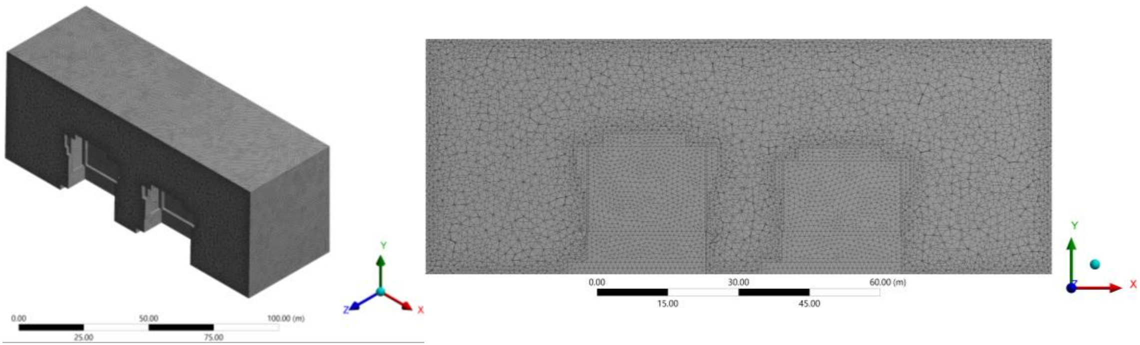

Figure 2.6. Three-dimensional mesh structure and front view.

Finally, the third requirement has to be satisfied to impose. The need to incorporate empirical information for the exterior wall boundary conditions. Within the scope of this study, aluminium spandrel curtain wall panel was considered as a facade cladding material for the simulation. The thermophysical properties of material components of the aluminium spandrel system and boundary conditions for numerical simulations which are thermal conductivity, density and specific heat were inserted in the simulation. Thermal conductivity is 160 W/mK, density is 2800 kg/m^3 and specific heat is 880 J/kgK [28].

While getting started on wind-oriented architectural design, not only prevailing local climate to evaluate but also purpose of use of the building, indoor moisture access and ventilation status necessery to considerate [28]. Thus, this simulation was carried out with materials that were pretended to be under room conditions (temperature of 21–23 °C) [29–32].

3. RESULTS AND DISCUSSION

CFD analysis of Istanbul Health and Technology University Merter Campus Settlement is simulated and velocity, pressure, temperature, turbulent kinetic energy contours and streamlines are illustrated in figures. There are totally two buildings in the campus settlement (see Fig. 2.1). Temperature and air velocity measurements made once a week throughout 2021 in the campus settlement have been validated by the simulation results. In the simulation; annual average air flow velocity, air temperature and wind direction data are used. Airflow in the northeast direction, which is the prevailing wind direction, is simulated. In the simulation, the flow direction is defined as the -z direction. Buildings, building exterior materials (coverings), pavement were simulated in the analysis, and the heat transfer between them and the accompanying flow events were examined. Streamlines related to flow velocity are given in figures 3.1.-3.5. in isometric and top view of the analysis. Streamlines are projected through the planes created at 5-meter intervals starting from the ground floor.

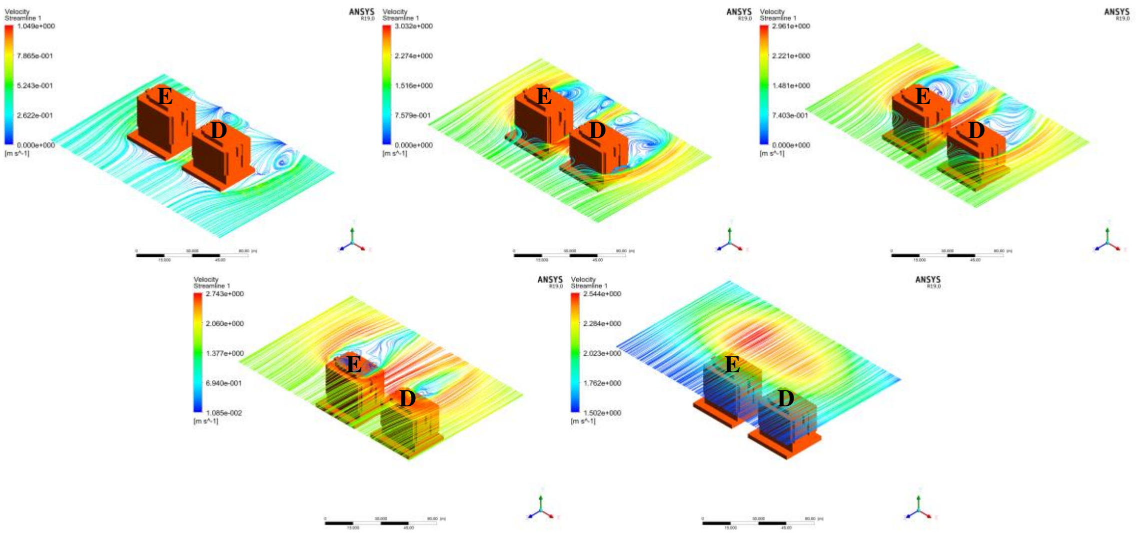

Figure 3.1. Isometric view of streamlines (ground floor-40 m).

As can be seen from the streamlines starting with the ground floor and going up to 40 meters in the Figures 3.1. and 3.2., the air flow velocity between the two buildings increases to approximately twice the initial level. After passing the buildings, separations occur in the flow, which creates a high degree of turbulence at the rear of the buildings. As a result of the air velocity measurements taken with the help of anemometer at a height of 5 meters from the ground throughout 2021, the annual average air velocity was taken into account to be used in the analysis and this value was taken as approximately 1.5 m/s.

It is seen that turbulences occur at almost every height in the back parts of the health sciences (block E) and engineering and natural sciences (block D) faculties buildings, which have different heights, cross-sections and asymmetrical structures.

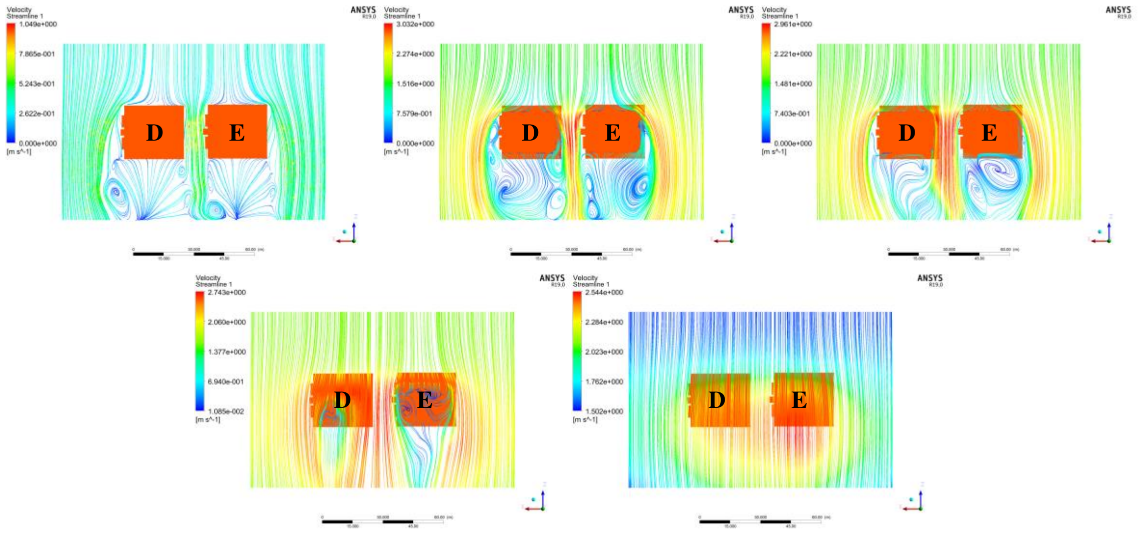

Figure 3.2. Top view of streamlines (ground floor-40 m).

It is seen that behind the health sciences faculty building, turbulences with higher intensity and even more than one turbulence form at some heights. In the streamlines at 30 meters, the flows on the roofs of both buildings on the roofs are visible. Turbulence has been observed around and above the structures on the roofs.

Figure 3.3. shows three-dimensional air velocity streamlines and streamlines that travel through the entire flow volume. The area shown in grey is defined as the concrete floor. The walls of the buildings and the ground floor were defined as walls in the simulation and their temperatures were inserted.

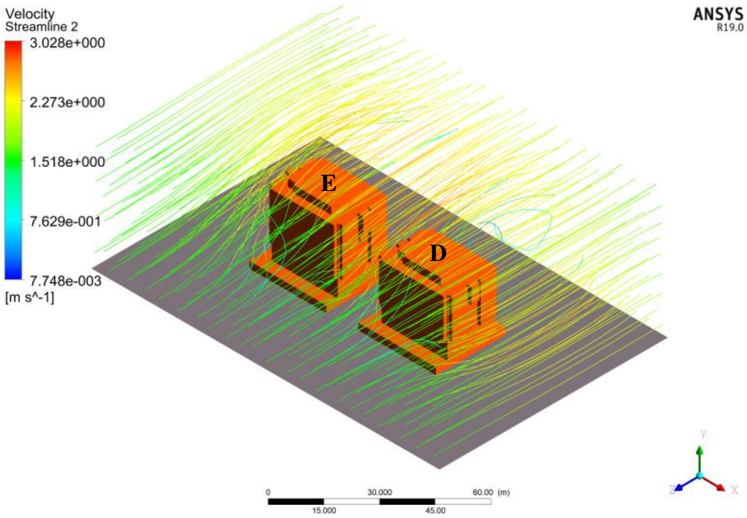

Figure 3.3. Three-dimensional streamline view.

Figure 3.4. and 3.5. isometric and top views of the speed contours up to 40 meters with a height difference of 10 meters from the ground floor. It can be seen from the velocity contours that the initial flow velocity increases at the between the buildings and mostly around them.

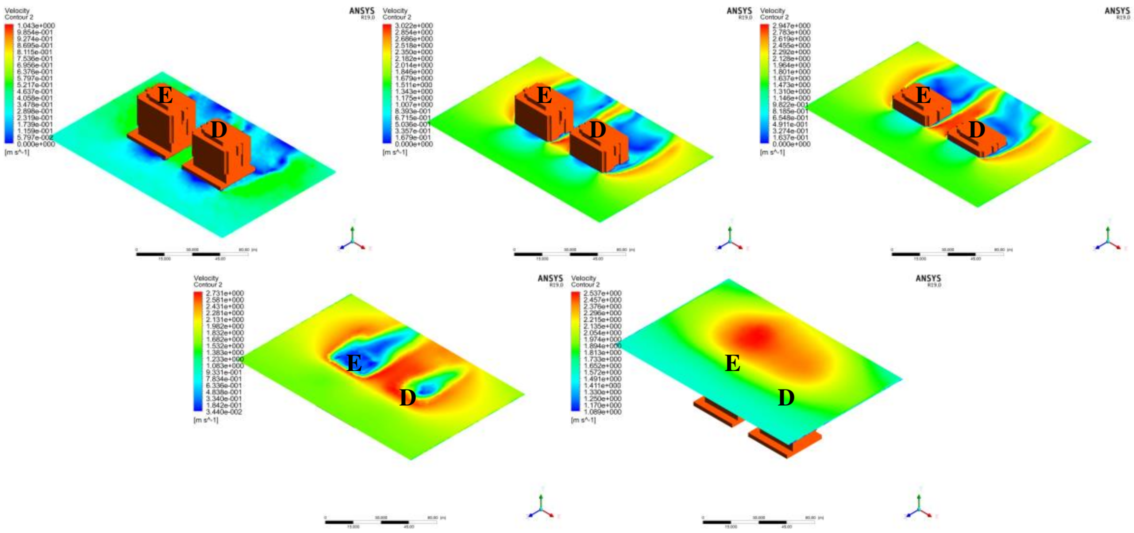

Figure 3.4. Isometric view of velocity contours (ground floor-40 m).

Looking at the first images in Figures 3.4. and 3.5., it is seen that the flow velocity is quite low. This is due to the boundary layer on the floor and the walls of the building. It is observed that high air velocities are high on the roofs as well as in and around buildings. It is seen that the flow velocity has doubled in the terrace parts.

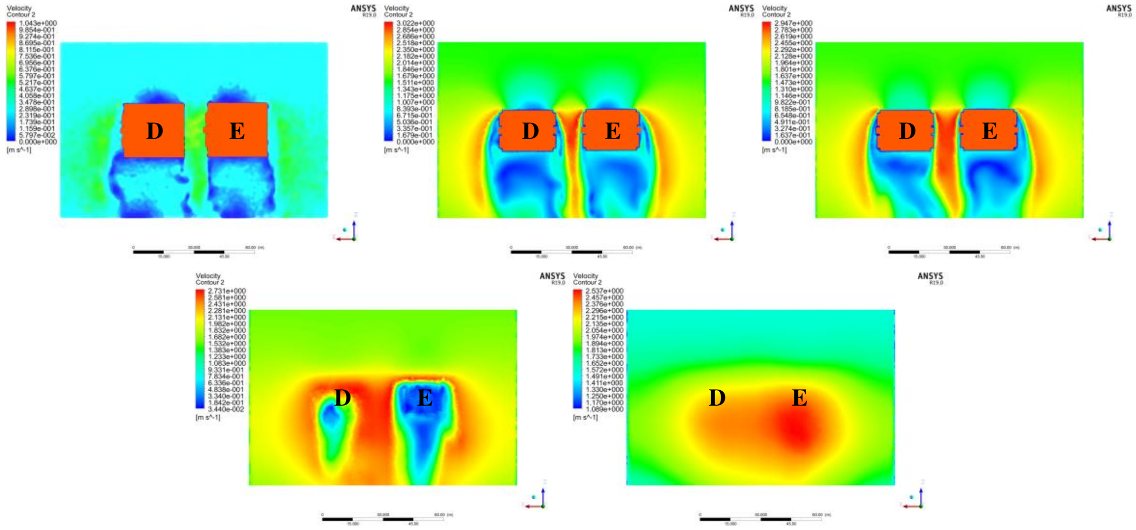

Figure 3.5. Top view of velocity contours (ground floor-40 m).

In Figure 3.6., velocity contours are illustrated in the plane formed perpendicular to the direction of the wind and passing through the middle section of the two buildings. It is observed that the flow velocity increases as moved away from the buildings, hence the boundary layer. At the speed contours in the middle of the buildings, it is seen that the wind speed reaches 3 m/s with dark red colour.

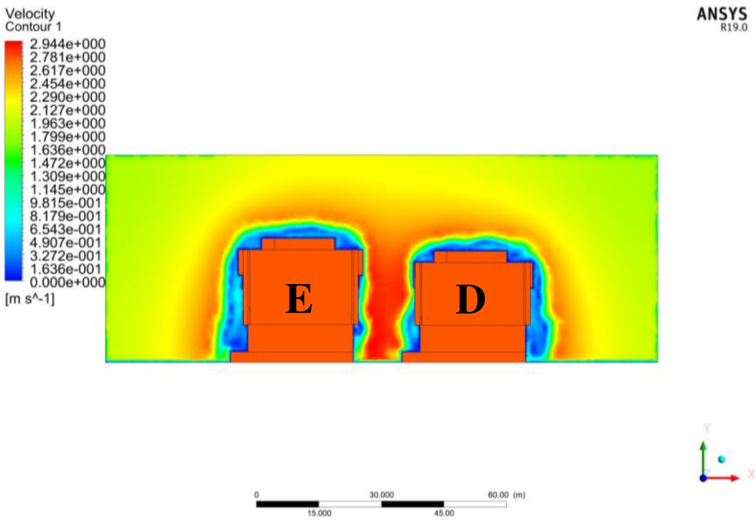

Figure 3.6. Air flow velocity contours of buildings from the wind direction.

Figure 3.7.-3.9. shows the pressure contours obtained as a result of the simulation. Here is the Figure 3.7. showing the results of the pressure taken every 10 meters isometrically while the Figure 3.8. shows the top view. While the red colour on the scale represents the highest values and therefore the positive pressure, the parts that appear in blue indicate the minimum pressure values and even negative pressure and therefore vacuum formation in some places.

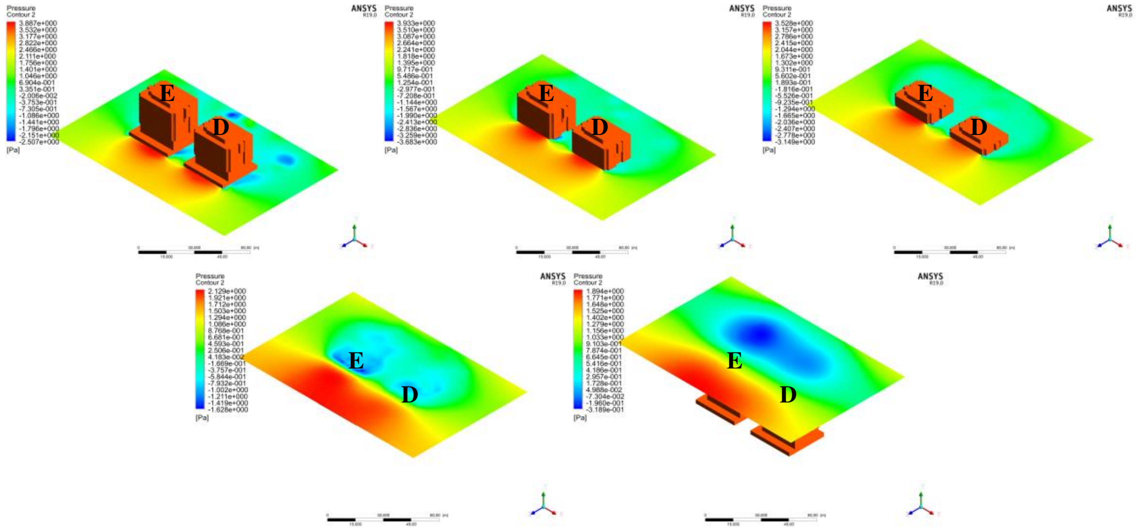

Figure 3.7. Isometric view of pressure contours (ground floor-40 m).

Depending on the kinetic energy of the air molecules, it is seen that the pressure values reach the maximum in the parts where the wind hits the walls of the buildings. In general, it can be said that while positive pressure is observed at the front of the buildings, negative pressure is observed in the parts where the flow is separated and turbulence occurs.

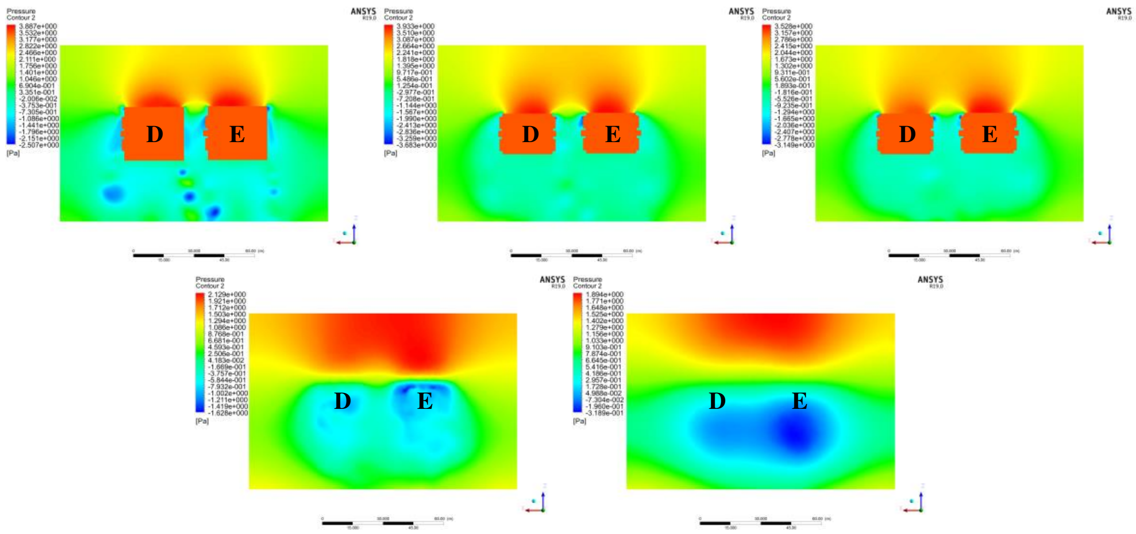

Figure 3.8. Top view of pressure contours (ground floor-40 m).

In Figure 3.8., the pressure contour section in the plane passing through the middle axis of the buildings from the direction of the wind is illustrated. It is observed that the turbulent zones formed between and around the buildings on the ground floor cause negative pressure.

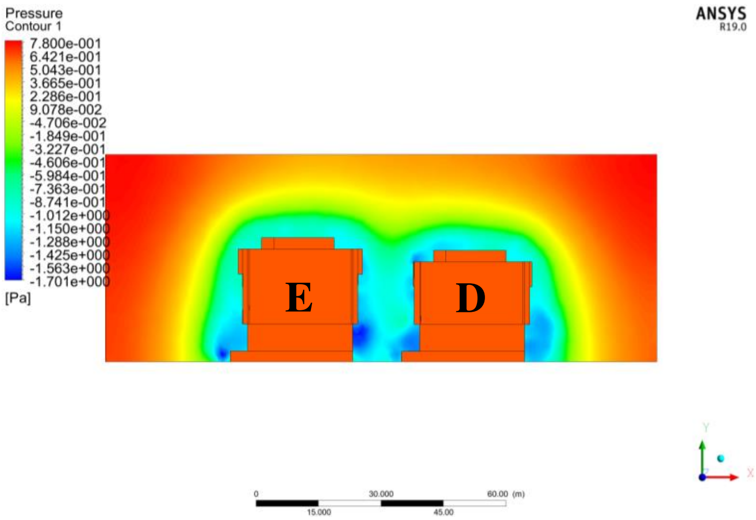

Figure 3.9. Air pressure contours of buildings from the wind direction.

Figure 3.10. shows the heat transfer occurring on the building surfaces and thus the temperature contours. The coolest areas are shown in dark blue, appearing in the direction the wind is coming from and where it hits the walls of the buildings. The temperature difference between the hottest and coolest point appears to be about 12 degrees Celsius.

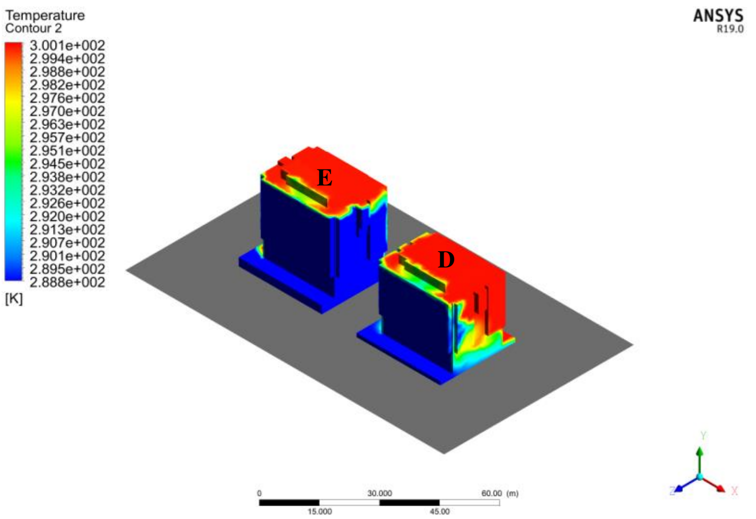

Figure 3.10. Isometric view of temperature contours on the walls of buildings.

It can be seen that the sides and backs of the buildings are warmer and less affected by the cold wind compared to the front surfaces. It is seen that the aluminium flooring, which is the outer covering of the building wall, transfers the heat very quickly and approaches the temperature of the wind as much as possible.

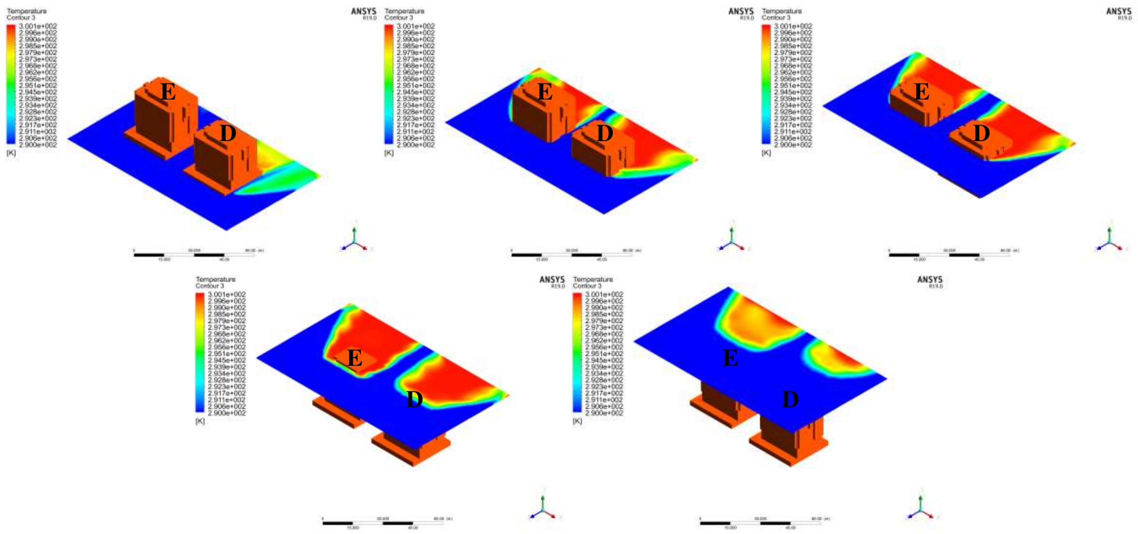

Figure 3.11. Isometric view of temperature contours (ground floor-40 m).

Figures 3.11.-3.13. depict the change in air temperature. It is clearly seen that the heat transfer takes place rapidly in the regions where the speed increases and the wall temperatures in those regions reach the air temperature values.

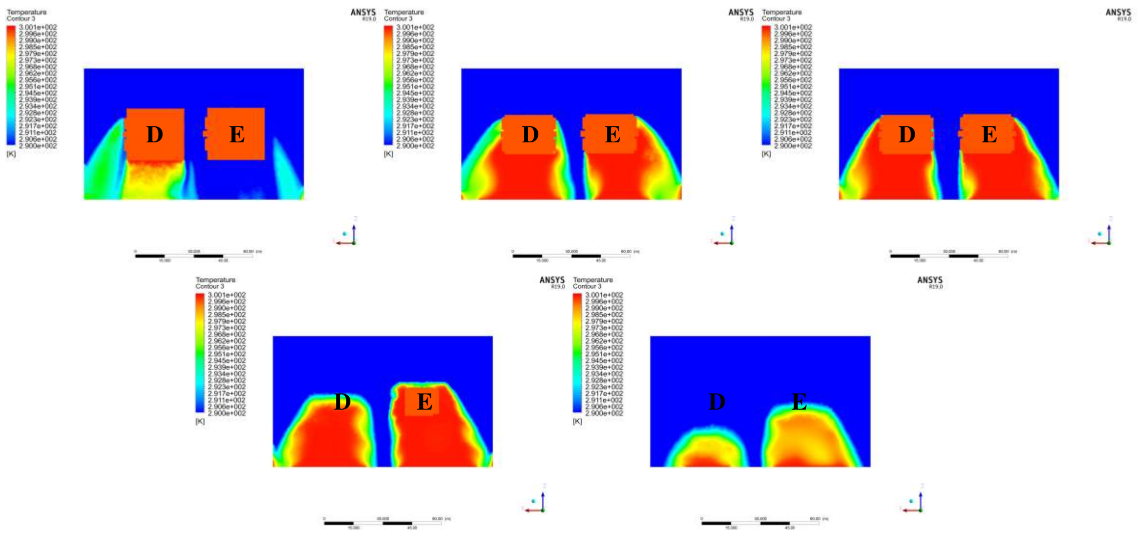

Figure 3.12. Top view of temperature contours (ground floor-40 m).

It is observed that the heat wave spreads behind the buildings with the effect of the wind hitting the buildings. As got closer to the boundary layer, the temperature increases, and as moved away, the decrease in air temperatures can be seen with the inclusion of air velocity.

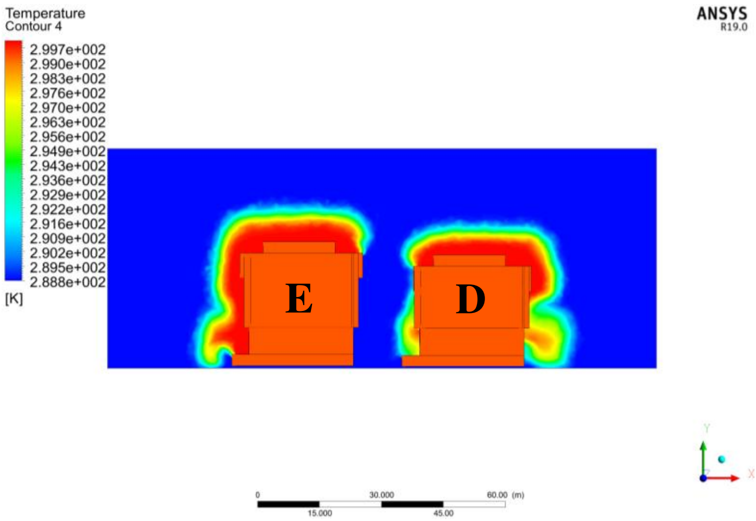

Figure 3.13. Air temperature contours of buildings from the wind direction.

When viewed from the front surfaces of the buildings in Figure 3.13., the heat dissipation of the buildings from the building walls can be seen. The distribution of cold air temperature between two buildings is thought to be due to air velocity. In places where turbulence occurs, the air velocity drops very low and even goes to zero in some places, there is a temperature diffusion.

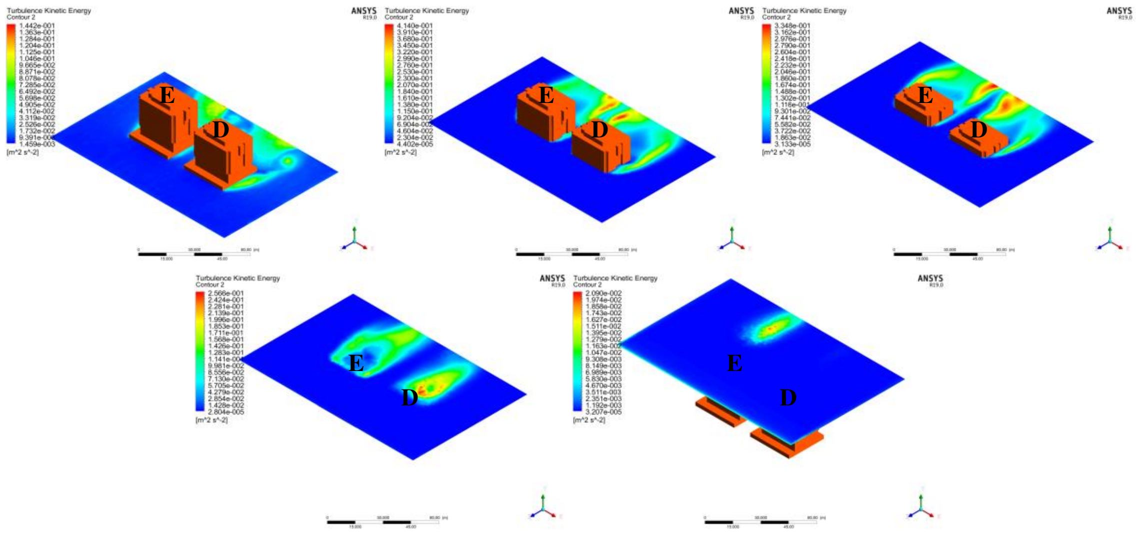

Figure 3.14. Isometric view of turbulence kinetic energy contours (ground floor-40 m).

In Figures 3.14.-3.16., there are contours showing turbulence density, also known as turbulence kinetic energy. Turbulence intensity is quite high in red tones and very low in dark blue tones. The highest turbulence kinetic energy density is seen at 10 meters above the ground and between two buildings, in areas where flow diverges and high intensity turbulence occurs.

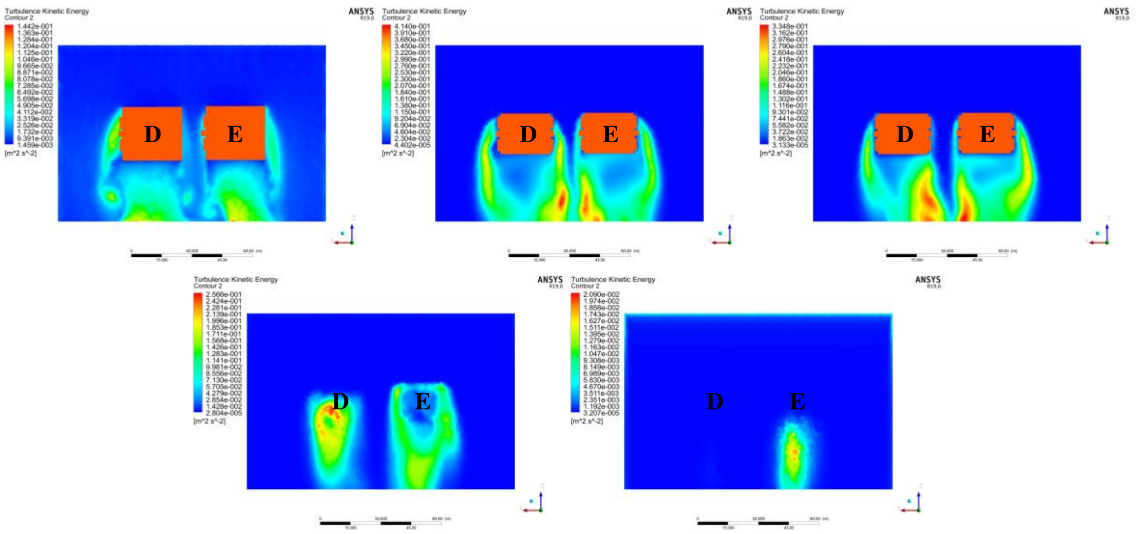

Figure 3.15. Top view of turbulence kinetic energy contours (ground floor-40 m).

It can be seen that the turbulence density and intensity increase between and behind the buildings. When the buildings are viewed from the front, the maximum turbulence kinetic energy is seen on the surface facing the engineering and natural sciences faculty building of the faculty of health sciences and where it is slightly higher than the ground. It is noteworthy that turbulent kinetic energy occurs near the building walls and exactly where the flow diverges.

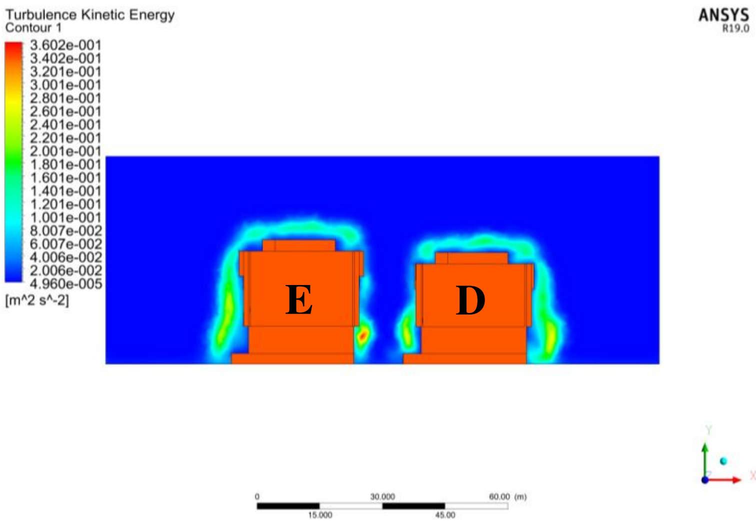

Figure 3.16. Air turbulence kinetic energy contours of buildings from the wind direction.

Figures 3.17.-3.20. was created based on the data received on a line of 100 meters in the z-axis direction, passing through the middle of the two buildings. In Figure 3.17., the velocity values on the mentioned line were taken from the ground floor to a height of up to 40 meters. Ground floor velocity values generally show a fluctuating profile and reach their maximum value at approximately 50 meters, that is, in the middle of two buildings. The maximum value read here stands out as the highest value when compared to all heights. Values at other heights draw almost consecutive profiles. While flow velocities draw a curve profile inversely proportional to height, this situation is reversed after 60 meters.

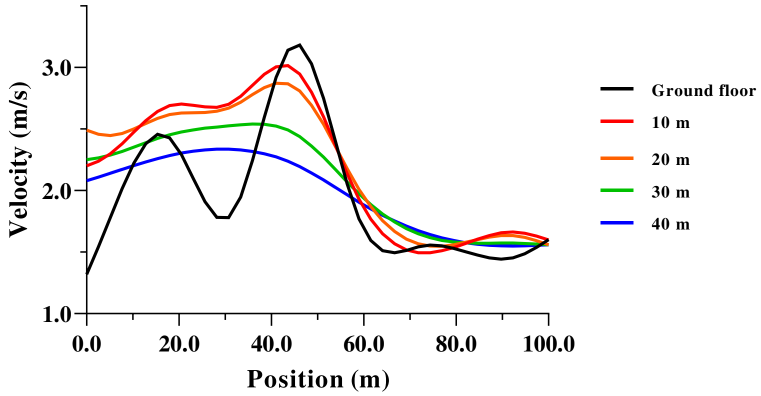

Figure 3.17. The air flow velocities at different altitudes.

The break in the velocity curves at the 60th meter is also clearly seen in the pressure curves in the Figure 3.18. The fluctuating structure in the velocity values taken on the ground floor also shows itself in the pressure curves. As it rises from the ground, it is seen that the negative pressure values turn into positive. After the 60th meter, this situation reverses as in the velocity curves.

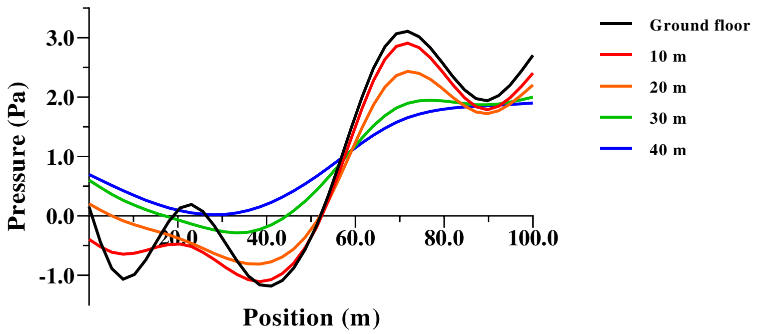

Figure 3.18. The air pressures at different altitudes.

As can be predicted from the temperature contours, the changes in the temperature curves in the Figure 3.19. occur after the buildings pass and reach the heat waves. The peak occurring in the temperature curve at a height of 10 meters is even higher than the other curves. A certain correlation with altitude and temperature cannot be established from the curves.

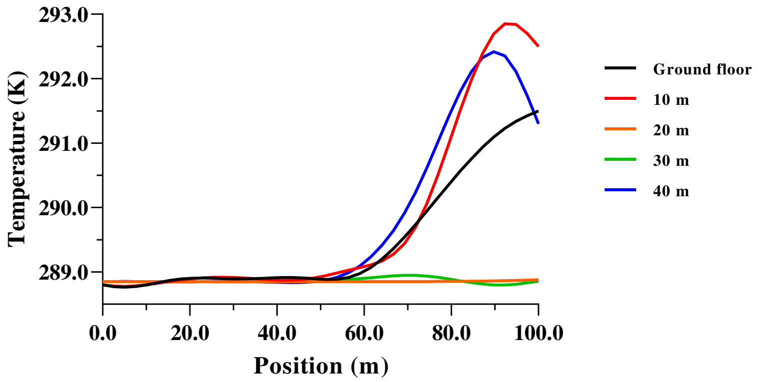

Figure 3.19. The air temperature at different altitudes.

Looking at the turbulent kinetic energy curves in Figure 3.20., it can be said that the coloured turbulence kinetic energy contours indicate the same parallel results. Turbulences occur in regions where the flow leaves the surfaces. From this point of view, on the line where the data is taken, the turbulence kinetic energy decreases as the height increases. As can be seen from the turbulence kinetic energy figures, turbulence mostly occurs near the walls of the buildings. Therefore, looking at the curves in the data taken from the middle of the buildings, it is thought that the decrease is due to this.

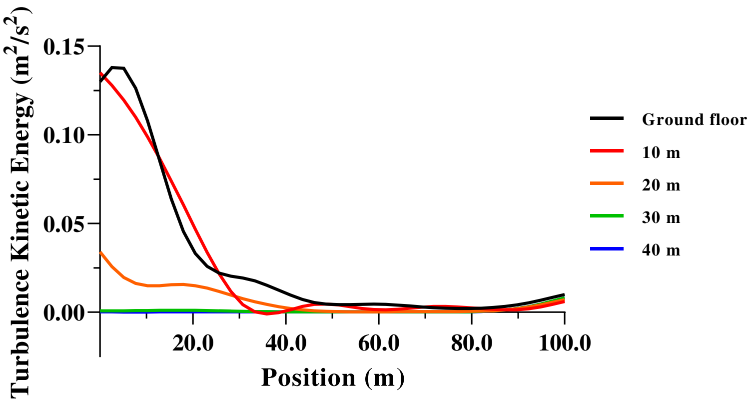

Figure 3.20. The air turbulence kinetic energy at different altitudes.

4. CONCLUSIONS AND RECOMMENDATIONS

The 2019 version of Ansys Fluent, a computational fluid dynamics (CFD) simulation program, was used in the study. Here, the dominant wind direction, the northeast direction, was used, and the wind data taken during the year 2021 was transferred to the simulation. In the simulation, all the parameters that can be entered are considered in order to simulate the real conditions as much as possible. Parameters such as flow velocity, air temperature, temperature of the building wall material, type of building wall cladding and heat transfer coefficient, floor temperature and type were entered into the simulation and tried to approach the real conditions. Air temperatures and flow velocities were taken with the help of the anemometer weekly during 2021, and the averages of these data were used in the simulation. As a result of the simulation; coloured contours were taken as isometric, top and front views. In addition to these, the data taken from the middle of the two buildings were visualised and curves were drawn in order to better evaluate the subject. It has been studied to examine the flow and weather events occurring in the Istanbul Health and Technology University Merter Campus, to improve it in the next applications and to understand the comfort conditions. In this campus, two buildings belonging to the faculty of health sciences and the faculty of engineering and natural sciences were examined. Streamlines, flow velocity contours, pressure contours, temperature contours and turbulent kinetic energy contours are viewed and projected. In general, it is seen that turbulences occur in many places and this may affect the students and staff there. It has been observed that turbulences mainly occur between two buildings, behind them and near the ground floor. It is known that turbulence also affects the comfort conditions of the people in the university, and it can be said the simulation is validated the status when these data are taken into account. In addition, heat transfer analysis was also carried out and it was observed that dramatic transitions occurred on the front facades. This study thus sheds light on new studies on reducing adverse conditions.

In addition to the ones made in future studies, time dependent heat transfer can be observed. Since the building is constantly heated, heat production values can be added to the simulations as another parameter. On more powerful computers, meshes with larger area or with smaller mesh elements can be discarded and analysed. By entering the temperature and material properties of all wall layers, thermal analyses can be performed as well as flow analysis.

Declarations

Ethical Approval

Not applicable.

Competing Interests

The authors declare that they have no conflict of interest.

Author's Contributions

All authors contributed equally to this work.

Funding

This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Availability of data and materials

The data that support the findings of this study are available from Istanbul Health and Technology University. Restrictions apply to the availability of these data, which were used under license for this study. Data are available from the authors upon reasonable request and with the permission of Istanbul Health and Technology University.

REFERENCES

1. Dogan, T., Kastner, P.: Streamlined CFD simulation framework to generate wind-pressure coefficients on building facades for airflow network simulations. Build. Simul. 14, 1189–1200 (2021). https://doi.org/10.1007/s12273-020-0727-x

2. Toparlar, Y., Blocken, B., Maiheu, B., van Heijst, G.J.F.: A review on the CFD analysis of urban microclimate. Renew. Sustain. Energy Rev. 80, 1613–1640 (2017). https://doi.org/10.1016/j.rser.2017.05.248

3. Calautit, J.K., Hughes, B.R., Chaudhry, H.N., Ghani, S.A.: CFD analysis of a heat transfer device integrated wind tower system for hot and dry climate. Appl. Energy. 112, 576–591 (2013). https://doi.org/10.1016/j.apenergy.2013.01.021

4. Olenkov, V.D., Lazareva, I. V., Biryukov, A.D.: Numerical simulation of wind flow around building complex with different software approaches. IOP Conf. Ser. Mater. Sci. Eng. 687, (2019). https://doi.org/10.1088/1757-899X/687/5/055066

5. Chen, Q.: Using computational tools to factor wind into architectural environment design. (2004)

6. Churazova, A.: Wind Load Calculation on a Building with CFD, https://www.simscale.com/blog/2017/05/wind-loads-buildings/

7. Zhao, D.X., He, B.J.: Effects of architectural shapes on surface wind pressure distribution: Case studies of oval-shaped tall buildings. J. Build. Eng. 12, 219–228 (2017). https://doi.org/10.1016/j.jobe.2017.06.009

8. Jafari, M., Alipour, A.: Methodologies to mitigate wind-induced vibration of tall buildings: A state-of-the-art review. J. Build. Eng. 33, 1–60 (2021). https://doi.org/10.1016/j.jobe.2020.101582

9. Günaydın, T.İ.: Wind Flow Analysis on Simple plan-shaped Buildings. MEGARON / Yıldız Tech. Univ. Fac. Archit. E-Journal. 16, 286–305 (2021). https://doi.org/10.14744/megaron.2021.99975

10. Elshaer, A., Bitsuamlak, G.: Multiobjective Aerodynamic Optimization of Tall Building Openings for Wind-Induced Load Reduction. J. Struct. Eng. 144, 04018198 (2018). https://doi.org/10.1061/(asce)st.1943-541x.0002199

11. Daemei, A.B., Eghbali, S.R.: Study on aerodynamic shape optimization of tall buildings using architectural modifications in order to reduce wake region. Wind Struct. 29, 139–147 (2019)

12. CTBUH Staff: Tall Buildings in 2019 : Another Record Year for Supertall Completions. CTBUH J. 42–49 (2020)

13. Kalmikov, A., Dupont, G., Dykes, K., Chan, C.: Wind power resource assessment in complex urban environments : MIT campus case-study using CFD Analysis. In: Awea 2010 Windpower Conference. p. 2010 (2010)

14. Liu, Z., Yu, Z., Chen, X., Cao, R., Zhu, F.: An investigation on external airflow around low-rise

building with various roof types: PIV measurements and LES simulations. Build. Environ. 169, 106583 (2020). https://doi.org/10.1016/j.buildenv.2019.106583

15. Akon, A.F., Kopp, G.A.: Turbulence structure and similarity in the separated flow above a low building in the atmospheric boundary layer. J. Wind Eng. Ind. Aerodyn. 182, 87–100 (2018). https://doi.org/10.1016/j.jweia.2018.09.016

16. Kabošová, L., Kmeť, S., Katunský, D.: Wind flow around buildings of basic shapes with and without a wind-adaptive envelope. Sel. Sci. Pap. - J. Civ. Eng. 15, 59–75 (2020). https://doi.org/10.1515/sspjce-2020-0007

17. Bandi, E.K., Tamura, Y., Yoshida, A., Chul Kim, Y., Yang, Q.: Experimental investigation on aerodynamic characteristics of various triangular-section high-rise buildings. J. Wind Eng. Ind. Aerodyn. 122, 60–68 (2013). https://doi.org/10.1016/j.jweia.2013.07.002

18. Kim, Y.C., Tamura, Y., Yoon, S.-W.: Effect of taper on fundamental aeroelastic behaviors of super-tall buildings. Wind Struct. 20, 527–548 (2015). https://doi.org/10.12989/was.2015.20.4.527

19. Tamura, Y., Tanaka, H., Ohtake, K., Nakai, M., Kim, Y.: Aerodynamic characteristics of tall building models with various unconventional configurations. Struct. Congr. 2010. 3104–3113 (2010). https://doi.org/10.1061/41130(369)278

20. Yuan, K., Hui, Y., Chen, Z.: Effects of facade appurtenances on the local pressure of high-rise building. J. Wind Eng. Ind. Aerodyn. 178, 26–37 (2018). https://doi.org/10.1016/j.jweia.2018.05.004

21. Hamakareem, M.I.: Design of Curtain Walls for Wind Loads -Details and Calculations, https://theconstructor.org/structural-engg/design-curtain-walls-wind-loads/17683/

22. Pringle, T.: Curtain wall cladding | BRANZ Build, https://www.buildmagazine.org.nz/index.php/articles/show/curtain-wall-cladding

23. İstanbul Bölgesinde 2021 Yılında Geçmiş Hava Durumu (Türkiye) - Weather Spark, https://tr.weatherspark.com/h/y/95434/2021/2021-yılı-için-İstanbul-Türkiye-Tarihi-Hava-Durumu#Figures-Temperature

24. Sıcaklık - İstanbul - İklim Robotu Türkiye, https://www.havaturkiye.com/weather/maps/city?LANG=tr&PLZ=_____&PLZN=_____&WMO=17060&CONT=trtr&R=0&LEVEL=162&REGION=0005&LAND=TU&MOD=tab&ART=TEM&NOREGION=0&FMM=1&FYY=2021&LMM=12&LYY=2021

25. GORGULU, Y.F., OZGUR, M.A., KOSE, R.: CFD analysis of a NACA 0009 aerofoil at a low reynolds number. J. Polytech. 0900, 0–1 (2021). https://doi.org/10.2339/politeknik.877391

26. CAMBAZ, A., GÖRGÜLÜ, Y.F., ARAT, H.: Two-Phase Numerical Modelling of a Wet Exhaust System in a Catamaran Motor Yacht Diesel Engine. Eur. J. Sci. Technol. 31, 165–170 (2021). https://doi.org/10.31590/ejosat.1007351

27. Cambaz, A., Gorgulu, Y.F., Arat, H.: Analysing fluid-structure interaction with CFD and FEA on a marine double-wall LNG piping system. Multidiscip. Sci. J. Marit. Res. 36, 51–60 (2022). https://doi.org/10.31217/p.36.1.6

28. Boafo, F.E., Kim, J.H., Ahn, J.G., Kim, S.M., Kim, J.T.: In Situ Experimental Investigation of Slim Curtain Wall Spandrel Integrated with Vacuum Insulation Panel. Buildings. 12, (2022). https://doi.org/10.3390/buildings12020199

29. Blumberg, M.S.: Body Heat: temperature and life on earth. (2002)

30. Childs, G.E., Ericks, L.J., Powell, R.L.: Thermal conductivity of solids at room temperature and below; a review and compilation of the literature. , Washington D.C. (1973)

31. Room temperature - New World Encyclopedia, https://www.newworldencyclopedia.org/entry/Room_temperature

32. Kittel, C., Kroemer, H.: Thermal Physics. W. H. Freeman and Company (1980)

Supplementary Files

This is a list of supplementary files associated with this preprint. Click to download.

- GraphicAbstract.jpg