Journal of Studies in Science and Engineering

EngiScience Publisher

Research Article

Application of CFD to Reduce the Cooling Load of a Building: An Alternative Approach for Green Building Design

Mirza Haseeb Khalid [1,*]

[1] Department of Mechanical Engineering, University of Engineering and Technology Lahore, Lahore, 54000, Pakistan

*Corresponding Author: Mirza Haseeb Khalid, E-mail: engr.mhaseebk@gmail.com

Article Info

Article History

Received Mar 01, 2022

Revised Mar 25, 2022

Accepted Mar 28, 2022

Keywords

Computer-Aided Design

CFD

Green Building Design

Abstract

Approximately 23 % of total CO_2 emissions are attributed to the construction industry each year; in 2009, the industry produced 5.7 billion tonnes of CO_2. Green building design is concerned with constructing a building in such a way that its carbon footprint is reduced throughout its life span. Lecture rooms are key components of educational institutions because they are where students spend the majority of their time. This study aims to propose a methodology to convert the existing designs of building to green buildings by optimizing the airflow. The lecture room at UET was chosen for this investigation because the air velocity distribution inside the room is not uniform due to the current design, which influences the convective heat loss from the room by creating pressure and temperature gradient. SOLIDWORKS 2021 was used to create the model of a lecture theatre. Because the majority of the rooms are naturally ventilated, a fluid dynamics analysis was performed on ANSYS Fluent 19.1 to ensure proper air ventilation. Natural airflow was improved by the addition of an extra column of windows. The total electric energy load has been computed, and the quantity of solar panels necessary to meet the demand has been recommended.

1. Introduction

The effort is being made by the scientific community to transform present construction techniques into ones that are more environmentally friendly. Engineers are attempting to develop more diversified and advanced designs for increasing efficiency, decreasing loads, and harnessing renewable energy to conserve the environment around buildings while also making them more sustainable.

As a result of the dramatic environmental and ecological changes that have occurred, a strong emphasis is being placed on lowering carbon emissions. In 2009, the construction industry was accountable for 23 % of global carbon emissions, producing 5.7 billion tonnes of CO2, with 94 % of these emissions coming from indirect sources, such as the production of building materials and the provision of electricity for a building[1] be taken to cut CO2 emissions. Green buildings were emphasized as critical components to reduce carbon emissions[2].

Designs aiming towards lowering the carbon footprint of buildings refer to green building designs. A strong collaboration between builders, designers, and engineering professionals is necessary to reduce the carbon emissions from a building. For green buildings, it is preferable to use locally created materials and sustainable energy sources whenever possible.

The lecture halls of educational institutions are where students spend the vast majority of their time. When it comes to maintaining a comfortable temperature in a room, the airflow circulation within the space becomes critically significant. Fresh air should be drawn into the room, and hot air should be expelled from it consistently through proper air ventilation. When dealing with a single room with a ceiling height of sixteen feet, it can be regulated with the aid of ceiling fans that are set in a regular pattern[3].

Mechanical or natural ventilation is used to ventilate lecture halls at Pakistani institutes depending on the climate. Most of the energy in Pakistan is derived from the burning of fossil fuels. HVAC systems utilize around two-thirds of the total energy consumed by cities each year [4]. It is critical to create a pleasant studying atmosphere inside the room by maintaining the room's ambient temperature.

To construct or improve existing rooms that have already been constructed, numerical simulations and experimental research are two methods that can be used. In recent years, computational fluid dynamics (CFD) techniques have shown to be an extremely useful tool, since they can avoid engineers from making costly mistakes during the testing process while also substantially reducing costs [5-7].

Multiple scholars had examined the performance of ventilation in an enclosed space using computational fluid dynamics (CFD). Mokhtarzadeh-Dehghan was the first to use computational fluid dynamics (CFD) to optimize airflow in a controlled volume in 1990 [8]. Then Ayad completed experiments on the distribution of airflow using computational fluid dynamics (CFD), as well as studied the effects of various air openings, such as windows and doors [9]. Analyzing the height of a naturally ventilated space with a single open side, Gan computed the height of the room using simple numerical analysis [10].

Convection is the primary mechanism through which heat is transferred out of a human body to the surrounding air. Because the temperature of fresh air is lower than that of the temperature of the body, it works as a constant sink, allowing heat to be transferred from the body to the sink through convection. The air that comes into contact with the human body will be heated and forced upward, allowing fresh air to replace it. It is possible to increase the efficiency of convective heat loss from a body by providing adequate ventilation in the surrounding area.

$$Q = HA\Delta T \tag{1}$$

In the above equation,

Q = Rate of heat transfer, ΔT = Difference in temperature and H =coefficient of heat transfer.

If the ceiling fans and windows are not appropriately located in the room, the air will be distributed unevenly, resulting in the formation of a pressure gradient in the room. The pressure in the room will fluctuate as a result of the varying air velocities present in different regions of the space. Following Bernoulli's Equation, pressure will be high in regions where velocity is low and will be low where velocity is high.

$$P_1 + \frac{1}{2}\rho v_1^2 + \rho g h_1 - P_2 - \frac{1}{2}\rho v_2^2 - \rho g h_2 = 0 \tag{2}$$

where P denotes pressure, denotes density, v denotes velocity, and h denotes elevation at two separate points in a two-dimensional flow.

Geographically, Pakistan is located in the eastern hemisphere at 30.3753^{o} N and 69.3451^{o} E. A large amount of solar energy can be harvested from it. Using solar power, Pakistan is capable of producing 100,000 MWatt of energy, according to a recent report. Solar power can be used to meet the energy needs of lecture halls and other nearby buildings. As solar technology improves, it will become more widely available and less expensive. On a sunny day, a single solar panel can provide up to 350 watts. For example, in Lahore, solar energy can be harvested at 4.5 to 5.0 kW/m^2/h/day. Renewable and environmentally friendly, this energy should be used in environmentally friendly constructions. For the duration of its operational life, it will emit zero carbon dioxide. Figure 1 shows the solar radiation per annum in Pakistan.

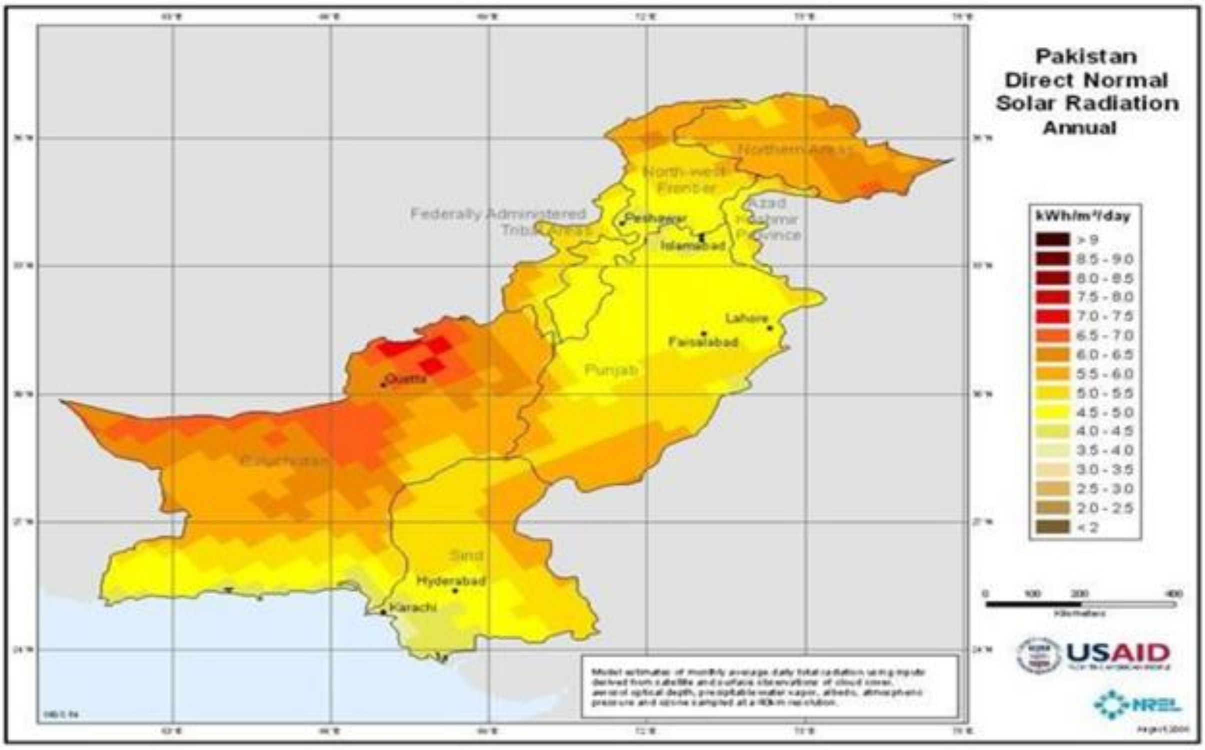

Figure 1. Solar Radiation per annum in Pakistan

To achieve the aim of converting a current building to a green building, all lighting, ceiling fans, and other electrical equipment in institutional buildings should be powered by solar energy. It takes 100 Watts to run seven ceiling fans and ten tube lights in the lecture hall at UET. An average of 1.1 kW is consumed by one room's total power consumption.

In this study, a CFD investigation of UET's lecture halls is conducted. The existing design of the lecture theatre will be improved to maintain a moderate and pleasant temperature by optimizing the natural airflow. This method can be used to change an existing building into a green and sustainable one.

2. Materials and Methods

This research was focused on a lecture hall on the first floor of the lecture theatre at the University of Engineering and Technology (UET) in Lahore. Figure 2 shows the actual lecture theatre.

Figure 2. Lecture room used in this study

2.1. Geometry of Design Using CAD

SOLIDWORKS was used to create the design of the lecture theatre. Equipment such as whiteboards, seats, and desks were not included in this edition. Windows' thicknesses were taken into account in the design process. The model is depicted in Figure 3 and Figure 4.

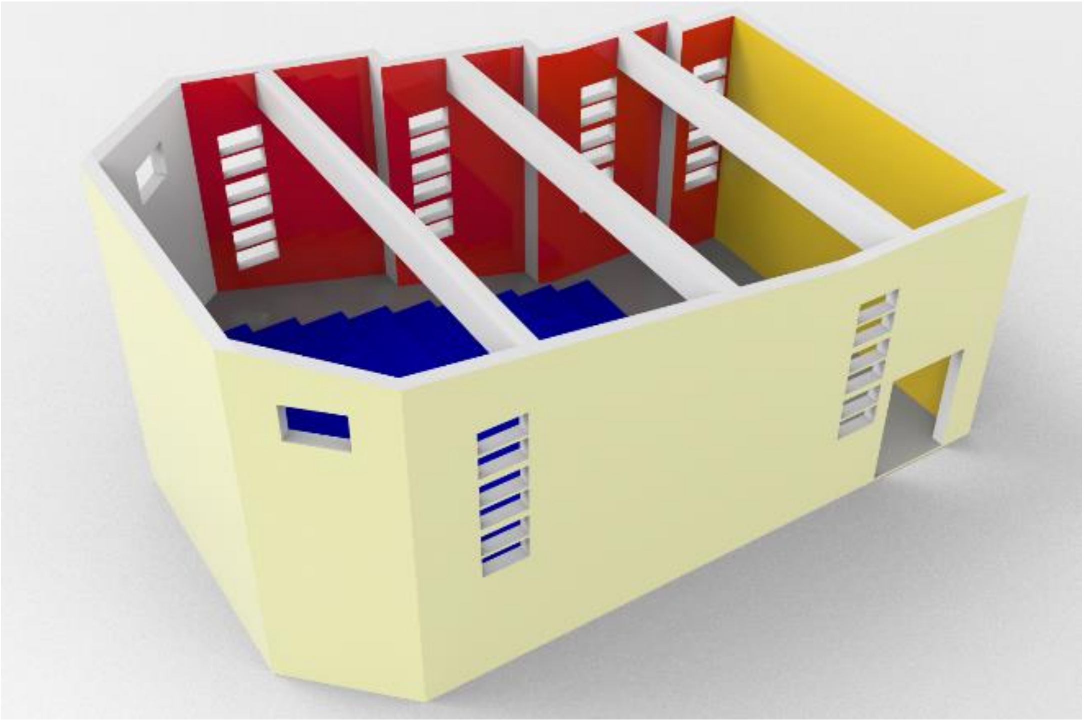

Figure 3. Geometry isometric view (side)

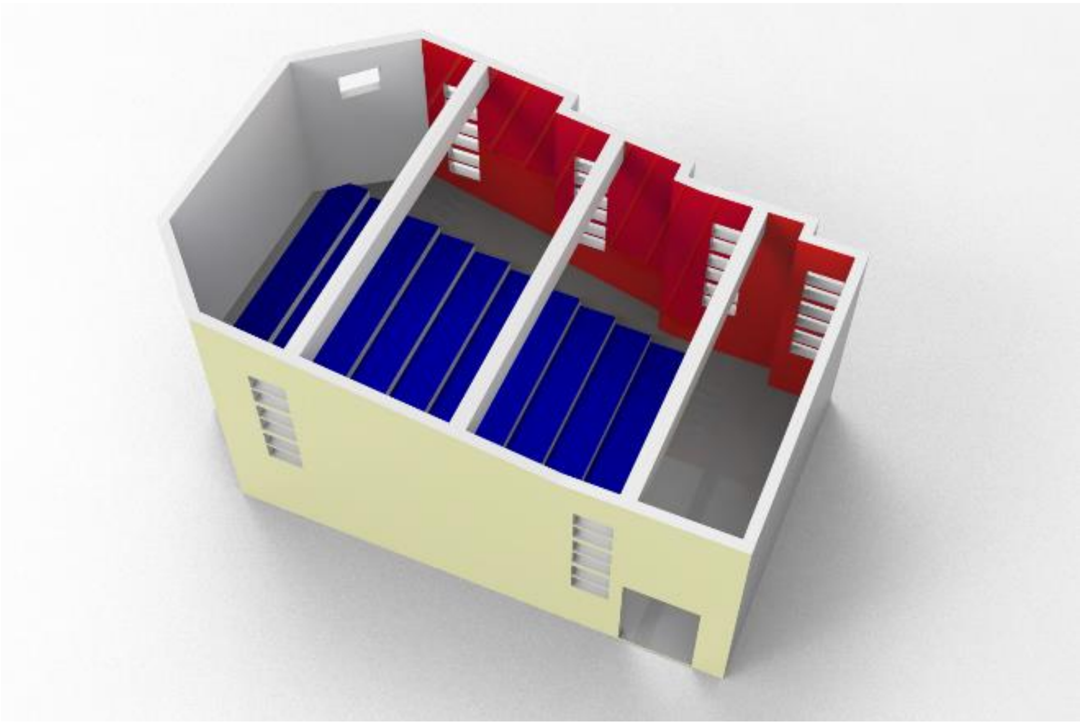

Figure 4. Geometry isometric view (top)

Major dimensions of the lecture theatre are depicted in figure 5. The unit of measurement used in this study is meter.

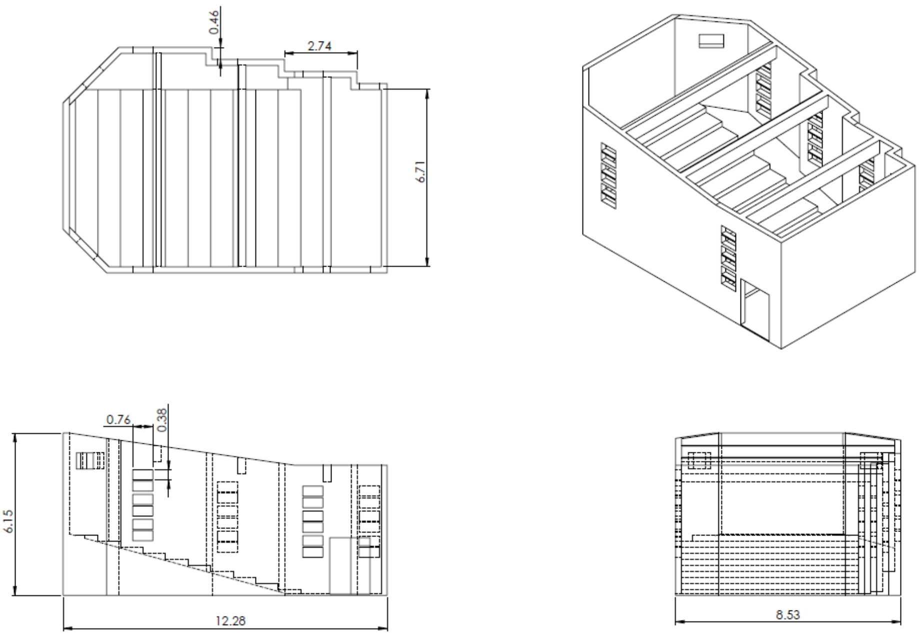

Figure 5. Major dimensions of geometry

2.2. Design of Fluid Domain

Before moving on to fluid modeling, it was necessary to precisely model the geometry of the fluid domain. Using SOLIDWORKS, the model of a lecture room was created. A solid block was modeled from which the actual model of lecture theatre was subtracted by using the 'Combine' command. The final fluid domain is displayed in figure 6.

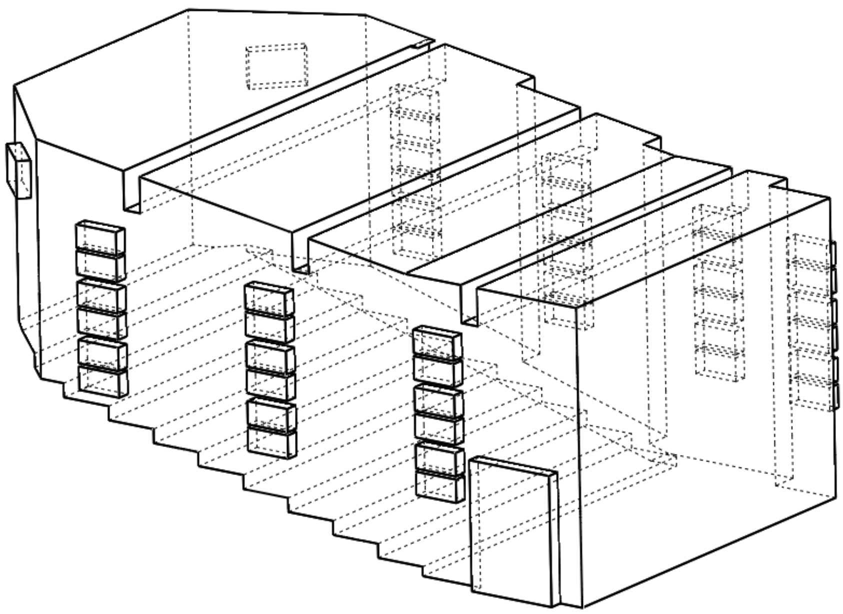

Figure 6. Lecture theatre fluid domain design

2.3. Details of Meshing

Figure 7 shows the meshed geometry after the generation of the fluid domain. Certain refinements were added to make sure that the mesh is refined enough in the critical regions.

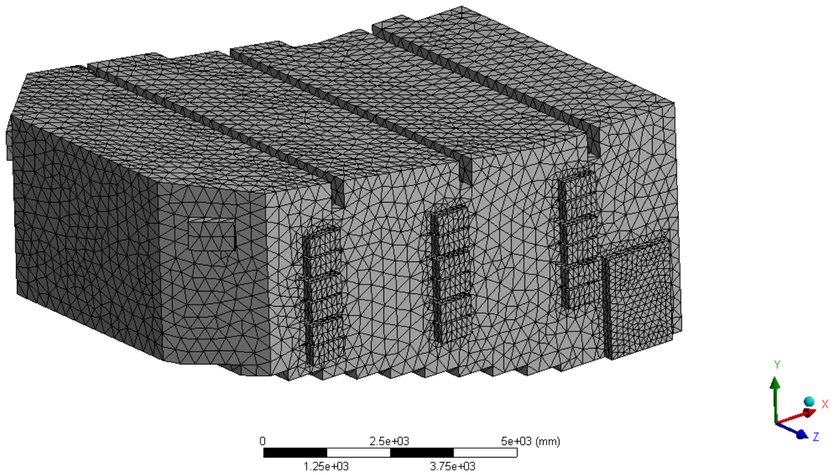

Figure 7. Mesh representation of geometry

2.4. Boundary Conditions

The following assumptions were made during this study.

- The wind is blowing from the west side windows at a velocity of 1.34 meters per second.
- The temperature outside is 300^{o}C.
- The windows and entrance on the east side of the room serve as an outlet for the room.
- All of the windows and doors are open entirely.
- The openings for the air conditioning system are treated as walls, and no air can enter or depart via them at any time.
- The fluid is of inviscid nature

3. Results and Discussions

3.1. Base Design

3.1.1. Distribution of velocity inside the room

Figure 8. shows the CFD findings of the current lecture room design. On the right, it can be observed that air enters through three columns of windows facing west and flows through the building towards windows on the east and doors. As it is assumed that the door is open, airflow is expected to pass through the door as well. As a result of this airflow, the volume near the whiteboard will be conditioned, and heat loss

will occur as a result of convection, resulting in a reduction in the cooling demand. There is an exit available because of windows on the back rows and the front rows, allowing the air to pass through them and so allowing uninterrupted ventilation in the front and back rows as well. However, in the center rows, there is no east-facing window; as a result, the flow of air is trapped within the building due to the absence of windows, resulting in the low-velocity zone shown in figure 8. When the air coming via the west windows approaches the east side wall, the air molecules strike the wall and bounce back, causing an interruption in the flow of air. The low-velocity regions are indicated by blue color, while the high-velocity parts are indicated by red color. The pattern of velocity within the room is depicted in Figure 8.

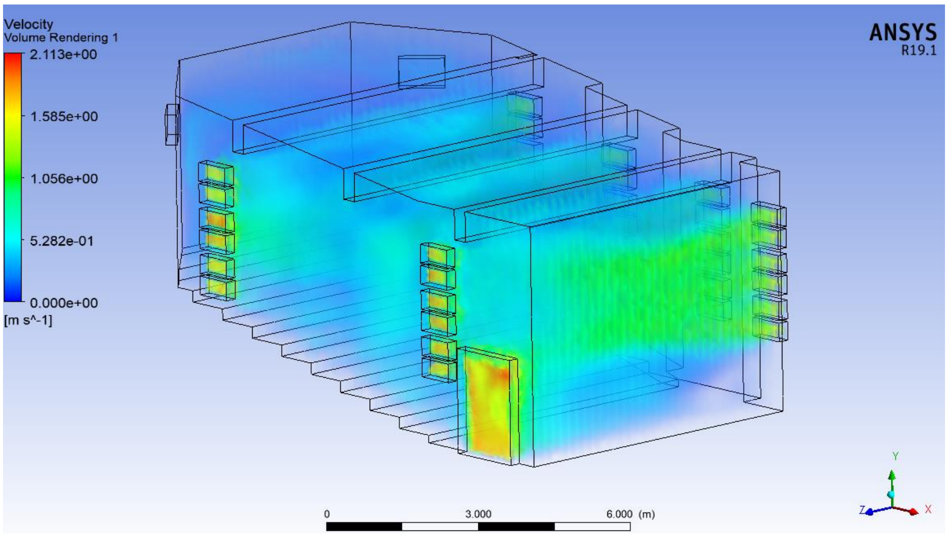

Figure 8. Velocity distribution of present design

3.1.2. Contours of pressure

Because it is the center rows that are of concern, a distinct plane has been included. This plane is used to investigate the characteristics of the flow. In figure 9, the findings of pressure are produced on the additional plane. The inlet of air is indicated on the right side, while the exit is indicated on the left side. The presence of small regions of low pressure around the entry can be detected as a result of fluid expansion. The bluish greenish region in the center of the room denotes the areas of moderately high pressure. However, on the left-hand side, there is a reddish region that shows the presence of areas of high pressure. The most pressure is concentrated in the middle, and pressure decreases as the streams of air are directed towards the ceiling or floor. It is clear from the use of Bernoulli's equation that the pressure in the low-velocity regions is greater than in the high-velocity sections. The same is true for high pressures on the left side of the body.

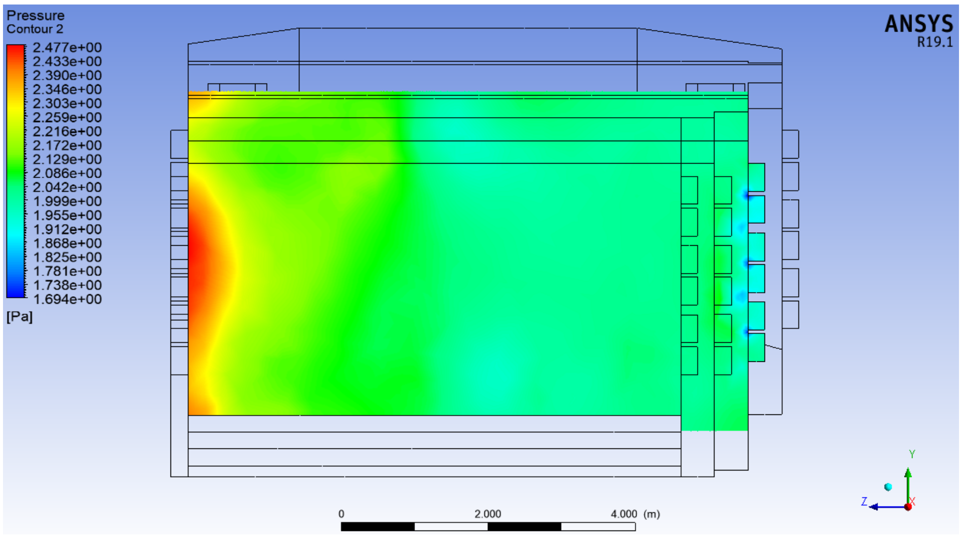

Figure 9. Base Design Pressure Contour

3.1.3. Contours of Velocity

The results of the velocity experiment are displayed in a contour plot. It can be seen that the largest velocity occurs at the entrance region, which is comprised of windows, and that the magnitude of the velocity decreases gradually as the airflow flows away from the inlet. Zero-velocity zones are denoted by a dark blue region around the windows and along the walls. Because of the no-slip condition, a layer of zero velocity is generated near the walls, which implies that the flow stays to the walls and that the velocity progressively increases as the flow goes away from the walls. As there are no windows, air strikes the walls and shifts away from the wall, moving towards the roof or the floor. Due to the absence of windows to serve as exit regions, the airflow is disrupted, and ventilation is affected. As there is no flow in that region, the amount of heat lost through convection would decrease. Resultantly, the amount of cooling load and electricity required to cool the middle rows would increase. Figure 10 depicts the results of the velocity contours of the initial design.

3.2. Design Optimization

According to the CFD results of the current design, it is necessary to improve the flow in the central zone of the room. To do this, a minor modification to the design is recommended, which is the addition of windows of the same size and height on the east side. That would allow air to escape through the windows in the center rows.

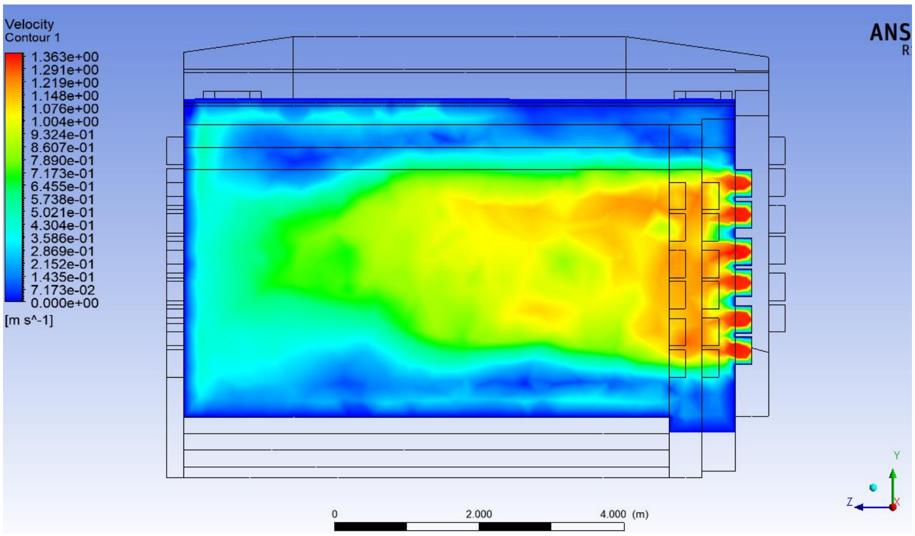

Figure 10. Base Design Velocity Contours

3.2.1. Distribution of velocity inside the room

The findings of the velocity distribution in the suggested design are shown in figure 11. It can be seen that the velocity has been increased in the middle region of the lecture theatre due to the addition of windows. The air molecules do not bounce back off the walls, and the air that comes from the west side can now easily depart through the windows on the opposite side, resulting in an undisrupted airflow and better heat removal due to convection than in the previous case. Figure 11 shows the results of the velocity distribution for optimized design.

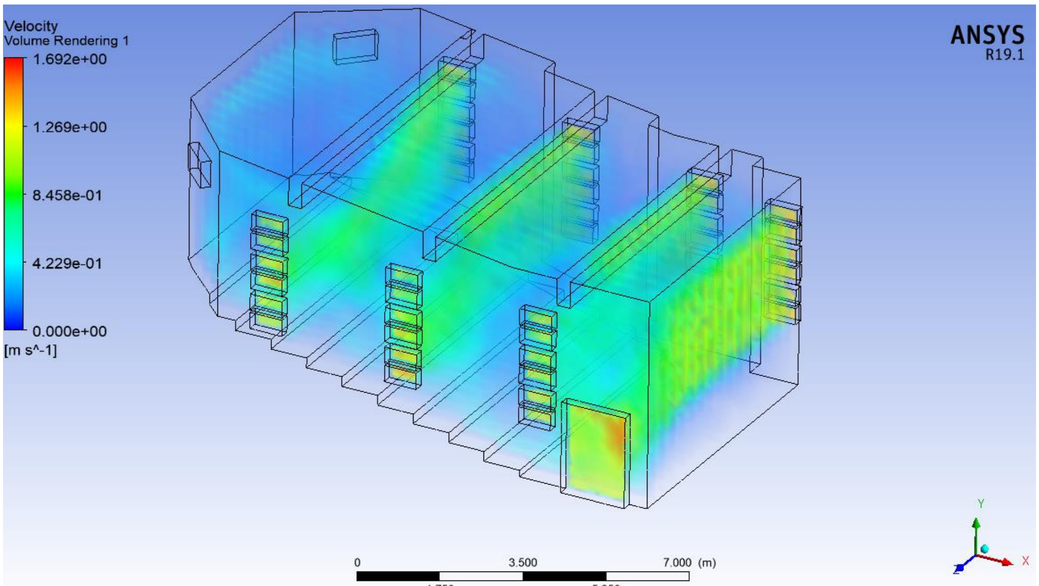

Figure 11. Optimized Design Velocity Distribution

3.2.2. Contours of Pressure inside optimized design

The pressure contours of the suggested design were generated. The airflow is not disrupted, as can be seen on the left, and as a result, the regions of high pressure have been reduced on the left side. Though high-pressure areas remain toward the roof, the area where students will be seated has a more uniform distribution of pressure. Figure 12 shows the results of the pressure distribution for optimized design.

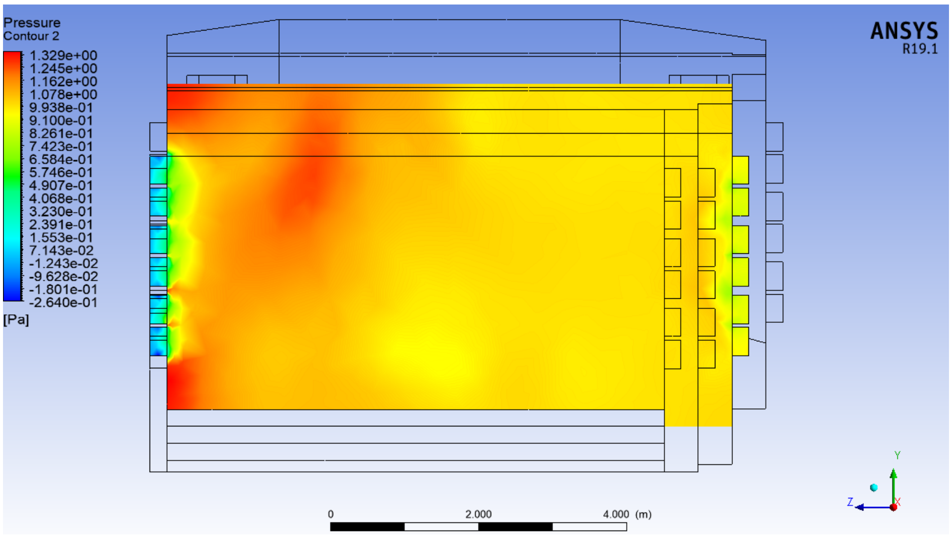

Figure 12. Optimized Design Pressure Contours

3.2.3. Contours of Velocity inside optimized design

The findings of the velocity contours of the suggested design show that the flow is uniform from the input to the output. It can be seen that there is no stagnation of air in the middle rows of the lecture theatre for the proposed design. Figure 13 shows the results of the velocity distribution for optimized design.

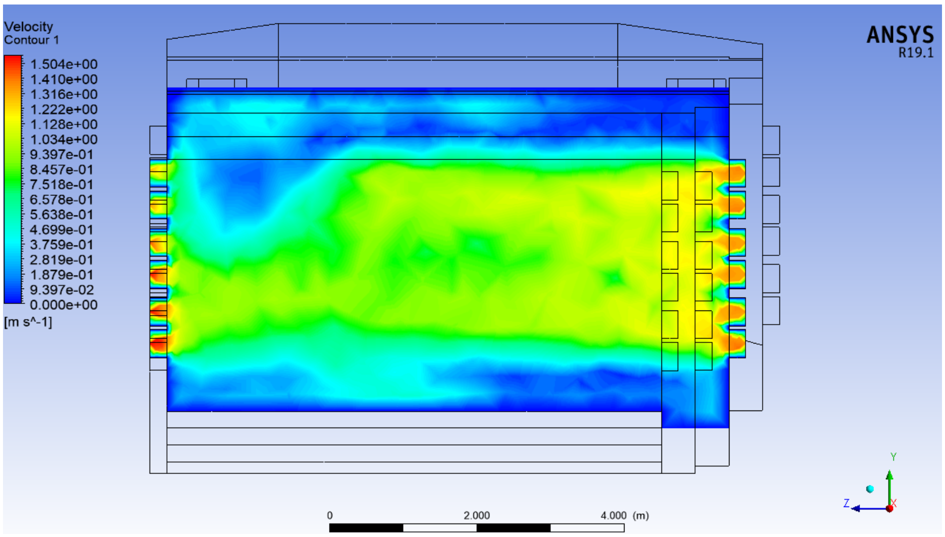

Figure 13. Optimized Design Velocity Contour

3.3. Results Comparison

To study the distribution of velocity from the west side to the east side, a line was added as a reference in the middle region. When there is no window present on the east wall, the air velocity decreases to zero. However, when a column of windows was added, the air velocity doesn't become zero and shows a small rise in magnitude. In the initial design, the slope of the curve was on the downward side, whereas the proposed design has a curve that is flat in the room area and gradually increases as it nears the exit. Because of the improved airflow, greater heat loss can occur as a result of convection, which would result in a reduction in cooling loads. A reduction in cooling load means a reduction in power consumption, which is a step forward in the direction of green building design.

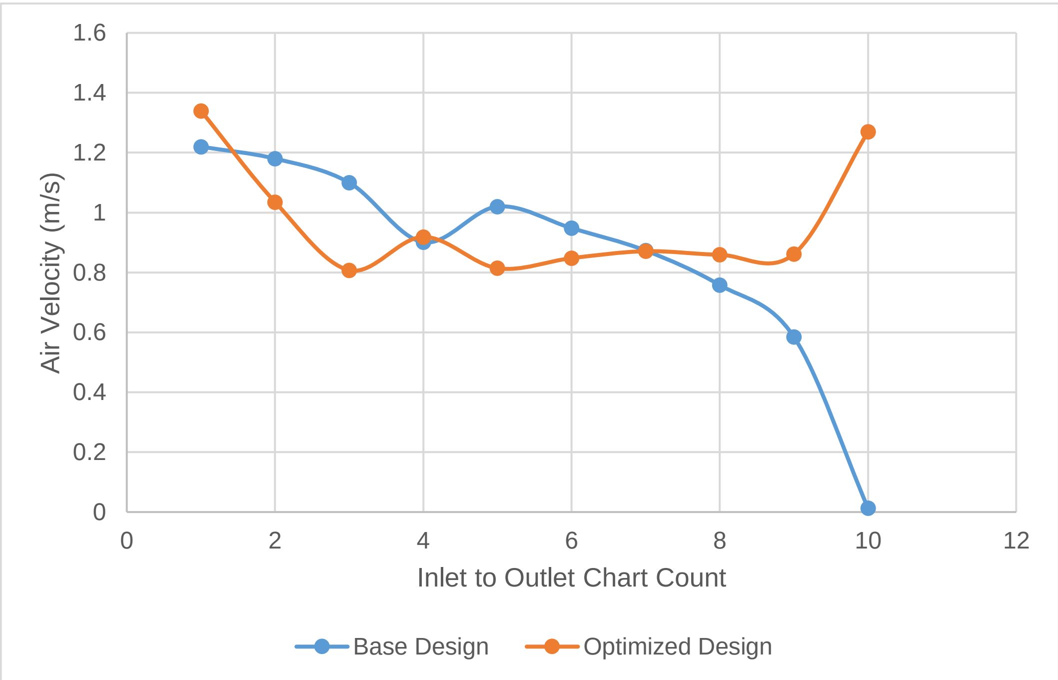

Figure 14. Comparison of both designs

4. Solar Power Calculations

4.1. Single Panel Energy

On a bright sunny day, a single solar panel may generate 300 watts of power. The power output is estimated to be 335 Watts, taking into account weather changes and the efficiency loss.

In Pakistan, the annual sunlight hours range between 2,900 and 3,300 hours. The amount of energy generated by a single solar panel in 8 hours is calculated as:

$$Energy_{KWh} = \frac{Power_{(Watts)} \times time_{hours}}{1000\ w/kw}$$

$$Energy_{KWh} = \frac{335 \times 8}{1000}$$

$$Energy_{KWh} = 2680\ Watt.Hours$$

4.2. Consumption of Power

The lecture hall is equipped with seven fans and ten tube lights.

Consumption of single fan= 100 Watts

Consumption of single light= 40 Watts

For eight hours each day, the daily energy consumption is 8.8-kilowatt-hours (KWh).

4.3. Required Solar Panels

The required amount of energy = 8.8 Kilo Watt Hours

Single Panel Energy Producing Capacity = 2 Kilo Watt Hours

Required Panels = 8.8 / 2.7 = 3.2

So, from the above approximation, the required number would be 4.

5. Conclusions

In this study, an alternative approach to design the green building was presented. A 3D model of the existing lecture theatre was modeled using SOLIDWORKS and a CFD analysis was conducted to study the air distribution inside the room. It was observed that stagnation of air takes place on the door side of the lecture theatre because of the absence of windows. Due to stagnation of air, heat loss due to convection reduces in that region, and the cooling load required to remove the heat from this region increases. An additional column of windows was added in the second design and CFD analysis was conducted on it. The problem of stagnation of air was reduced by the addition of a new column. Hence, it can be concluded that the current buildings can also be converted into green buildings by optimizing the natural airflow and reducing the cooling load required.

Declaration of Competing Interest: The author declares no conflict of interest.

References

[1] L. Huang, G. Krigsvoll, F. Johansen, Y. Liu, and X. Zhang, "Carbon emission of global construction sector," *Renewable and Sustainable Energy Reviews,* vol. 81, pp. 1906-1916, 2018.

[2] R. Falkner, "The Paris Agreement and the new logic of international climate politics," *International Affairs,* vol. 92, no. 5, pp. 1107-1125, 2016.

[3] A. S. Bachrun, T. Z. Ming, and A. Cinthya, "Building Envelope Component to Control Thermal Indoor Environment in Sustainable Building: A Review," *Sinergi,* vol. 23, no. 2, pp. 79-98, 2019.

[4] G. Elshafei, A. Negm, M. Bady, M. Suzuki, and M. G. Ibrahim, "Numerical and experimental investigations of the impacts of window parameters on indoor natural ventilation in a residential building," *Energy and Buildings,* vol. 141, pp. 321-332, 2017.

[5] J. L. S. Chong, A. Husain, and T. B. Tuan, "Simulation of airflow in lecture rooms," in *Proceedings of the AEESAP International Conference*, 2005, pp. 7-8.

[6] K. Cheong *et al.*, "Thermal comfort study of an air-conditioned lecture theatre in the tropics," *Building and environment,* vol. 38, no. 1, pp. 63-73, 2003.

[7] S. Lin, B. Tee, and C. Tan, "Indoor airflow simulation inside lecture room: A CFD approach," in *IOP Conference Series: Materials Science and Engineering*, 2015, vol. 88, no. 1: IOP Publishing, p. 012008.

[8] M. Mokhtarzadeh-Dehghan, M. El Telbany, and A. Reynolds, "Transfer rates in single-sided ventilation," *Building and Environment*, vol. 25, no. 2, pp. 155-161, 1990.

[9] S. S. Ayad, "Computational study of natural ventilation," *Journal of wind engineering and industrial aerodynamics,* vol. 82, no. 1-3, pp. 49-68, 1999.

[10] G. Gan, "Effective depth of fresh air distribution in rooms with single-sided natural ventilation," *Energy and buildings,* vol. 31, no. 1, pp. 65-73, 2000.