www.ijprems.com
editor@ijprems.com
INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
Vol. 04, Issue 06, June 2024, pp: 1959-1973
e-ISSN : 2583-1062
Impact Factor: 5.725

ANALYSIS OF HORIZONTAL FORCES OVER THE STRUCTURE BY AERODYNAMIC SHAPE MODIFICATIONS

Gauravkumar Ashok Neware[1], Dr Swati Ambadkar[2], Dr Swati Ambadkar[3]

[1,2]Department Of Civil Engineering, G H Raisoni University, Amravati, India.

[3]Guide, Department Of Civil Engineering, G H Raisoni University, Amravati, India.

ABSTRACT

The Wind is air in motion relative to the surface of the earth, whose primary cause is traced to earth's rotation. The wind generally blow horizontal to the surface at high speed and can be assessed with the aid of Anemometers. Tall and slender structures are flexible and exhibit a dynamic response to the wind. Therefore it is necessary to determine behaviour of structure. The square shape high building with aerodynamic modification is taken. In this study different models such as Corner cut, Setback and Opening are taken to determine the efficient shape modification among the models such as square, setback, corner cut with opening, setback with corner cut and setback with void/opening.

The square model is created by using SolidWorks as well as in Design Modeller in ANSYS software. After modelling, computational domain is provided. The tetrahedron meshing and acceptable fine mesh is provided for convergence. The setup is required following boundary conditions, the inlet velocity is 10m/s at reference point and udf.h(user define function) file is provided to follow power law. The air properties are also applied as per literature. The outlet pressure is zero pascal. The wall are having zero shear and treated as enhanced wall treatment. Then by using the fluent solver force coefficient is calculated. The square models is tested in computational fluid dynamics (CFD) simulations using ANSYS Fluent as well as CFX and validated with the literature results.

For the further study four different models are taken into study to determine the wind effects such as Drag and Lift coefficients and concluded the efficient shape modification to reduce wind load. The setback with opening model shows better results in order to reduce wind force coefficient as compared to square, setback and other models. After setback with opening, setback with corner cut shows better results. The ascending order of the models for the force coefficient are SQ < CC+VOID < SB < SB+CC < SB+VOID. The drag and lift force coefficient of the models are compared with square model and concluded.

Keywords: Anemometer, Drag coefficient, Lift coefficient, SolidWorks, etc.

1. INTRODUCTION

WIND

The Wind is air in motion relative to the surface of the earth, whose primary cause is traced to earth's rotation. The wind generally blows horizontal to the surface of earth at high speeds. Several atmospheric phenomenon like Cyclonic storms, Thunderstorms, Dust storms or Vigorous monsoons, Tornados, have extremely high wind speeds. These causes severe damage to the structure. Therefore structure is need to be wind resistant. The high building should be designed for wind loads.

The wind speeds recorded at any locality are extremely variable and in addition to steady wind at any time, there are effects of gusts, which may last for a few seconds. These gusts cause increase in air pressure and their effect on stability of the building may be important. Gusts can also be extremely important for design of structures with high Slenderness ratios. High and Slender structures are flexible and exhibit a dynamic response to the wind. High structure vibrate in wind as well as that generated by the structure itself. Thus there is a mean and a fluctuating response to the wind. Besides, the dynamic forces act not only in the direction of wind flow but also in perpendicular direction to the flow (lift forces), so that high structures also exhibit an across-wind response. Along-wind response has a mean component (time-invariant load obtained from the mean wind speed) and a fluctuating component.

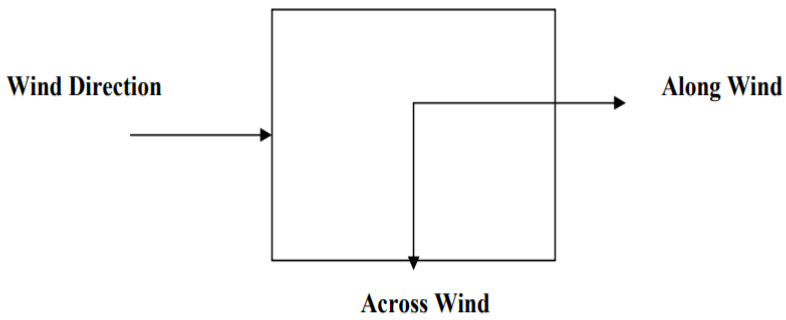


Figure: 1 Response of along-wind and across-wind

INDIAN STANDARD CODE:

Flexible slender structures and structural elements shall be investigated to a certain, the importance of wind induced oscillations or excitations along and across the direction of wind. In general, the following guidelines may be used for wind induced oscillations as per IS 875 Part3 2015.

- Buildings and closed structures with a height to minimum lateral dimension ratio of more than about 5.0.
- Buildings and closed structures whose natural frequency in the first mode is less than 1.0 Hz.

A TYPICAL HIGH-RISE STRUCTURES:

The high structures has no specific definition, as a structure height is greater than its adjacent structure. In many countries, it is define on the basis of number of floors or the height of the structure. It has advantages such as it give aesthetic view. To fulfil such a demand, A typical high-rise structure which are unique in geometry are came to existence. Based on geometry, there are different types of structure such as symmetric and asymmetric. High structures tend to have well-established occupier profiles. The other appeal factors for high rises are that they offer all the conveniences of modern life including swimming pool, gymnasium, grand entrance lobbies and high speed elevators.

Figure :2 Atypical High-Rise structures

Classification

Based on Geometry, there are many atypical high rise structures are as follow:

- Basic
- Square
- Rectangle
- Circle
- triangle
- Corner modification
- Tilted
- Tapered
- Helical
- Void
- Setback
- Combinations

Advantages

1. High Density developments, which concentrate multiple dwellings, and offices, etc. into a single development. This reduces the strain on urban infrastructure like roads, water supply, sewage, etc. as the connectivity needs to be provided to only one location as compared to low-rises structures where every dwelling unit needs a public infrastructure connectivity.
2. High-rise structures offer much better security to tenants than a ground level development which provides multiple points of access for intruders.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
www.ijprems.com
editor@ijprems.com
Vol. 04, Issue 06, June 2024, pp: 1959-1973
e-ISSN : 2583-1062
Impact Factor: 5.725

3. The high-rise structures can accommodate many more people on a smaller land than would be the case with low-rise structures on the same land. They are suitable for highly populated or overpopulated countries where there is a shortage of land.
4. It provides day lighting and greater flow of air.
5. It also gives pleasant as well as aesthetic view.

Disadvantages

1. Difficulties in preventing congestion and bad internet connection.
2. Privacy and social problems and neighbour interference.
3. Due to people imbalance load on municipal services like water supply and electricity.
4. Even if the association has no rule in banning pets in the apartment complex, taking your pet down for a walk becomes difficult if you lives in higher floor.
5. Difficulties in maintenance.

COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a branch of mechanics that uses numerical analysis and data structures to analyse and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high- speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels.

The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows. In the finite volume method, the governing partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes.

2. METHODOLOGY

GENERAL

In this chapter, model description and its configuration and their boundary condition as well as flow condition and other parameters are studied and perform the CFD simulation using ANSYS software in order to achieve the objectives which are already stated in previous chapter.

ANSYS OVERVIEW

ANSYS Computational Fluid Dynamics (CFD) simulation software allows you to predict the impact of fluid flows on your model — throughout design and manufacturing as well as during end use. The software's unparalleled fluid flow analysis capabilities can be used to design and optimize new equipment and to troubleshoot already existing installations. ANSYS renowned CFD analysis tools include the widely used and well- validated ANSYS Fluent and ANSYS CFX, available separately or together in the ANSYS CFD bundle. Because of solver robustness and speed, development team knowledge and experience, and advanced modelling capabilities, ANSYS fluid dynamics solutions provide results which are in close range with wind tunnel test. ANSYS CFD solutions are fully integrated into the ANSYS Workbench platform. This environment delivers high productivity and easy-to-use workflows.

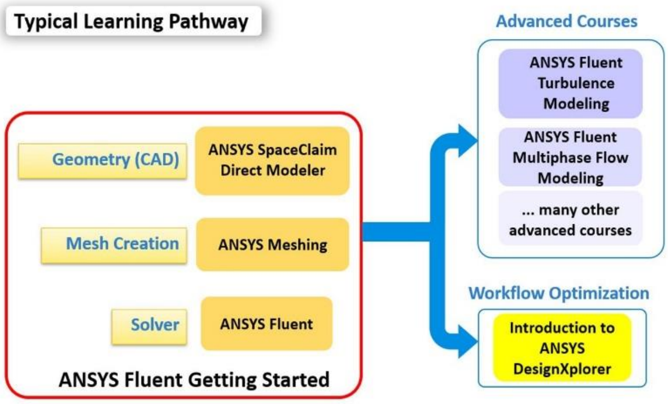


Figure:3 Ansys Workflow [17]

3. MODELING AND ANALYSIS

Model drafting

The model used in the study is square-shaped in plan. The structure at full scale has an outer diameter of 50m and the height is 400m, giving a B/H ratio is 1/8. The model has been scaled down to a 1:1000 scale to facilitate wind tunnel tests.

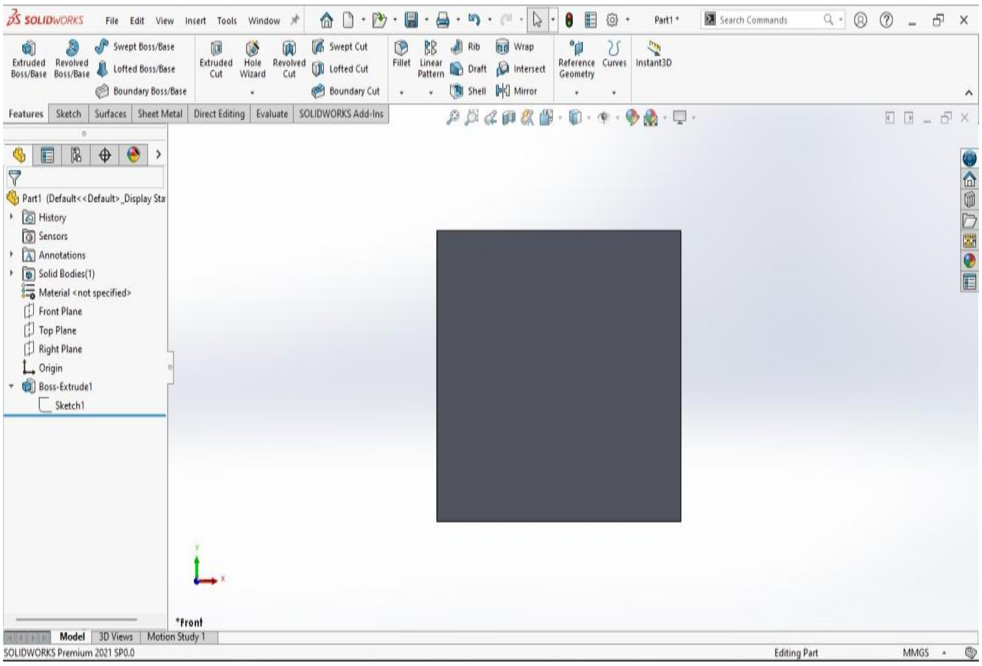


Figure : 4 Plan View/Top View

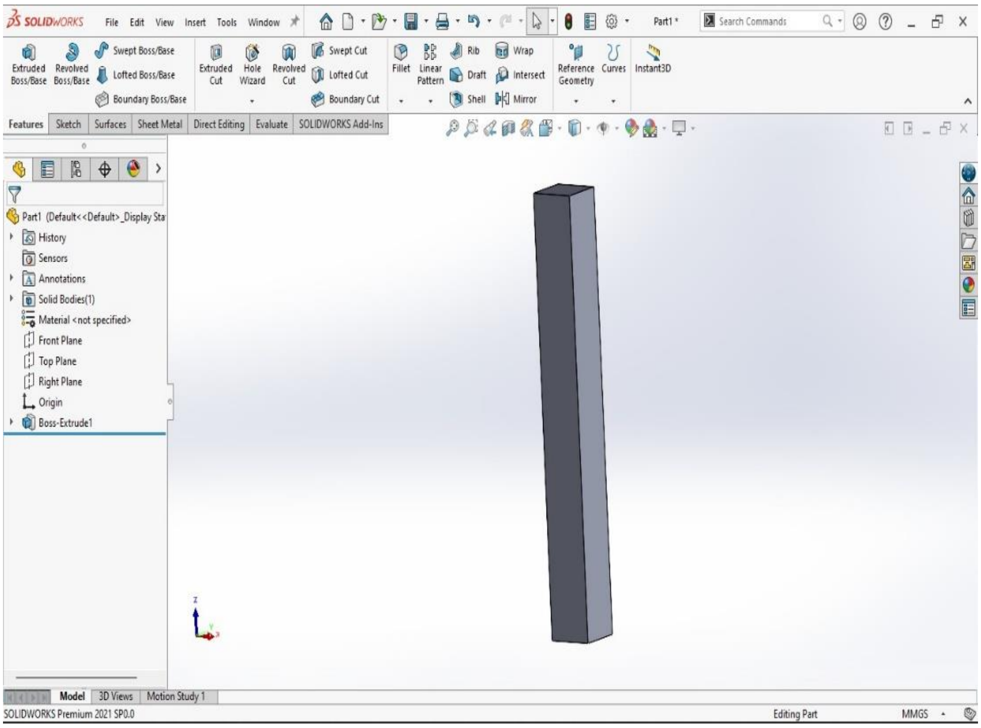


Figure:5 Side View

Computational domain

The wind tunnel is a rectangular box. It is designed to have the upstream fetch 5B, side clearance and height to be 5B and the downstream fetch to be 15B, where B is the width of the building.

Meshing

Meshing is done using ANSYS. Mesh refinements are made so that the solution variables converge down to acceptable levels. Tetrahedron meshing is used and is inflated near all boundaries. The meshing is done and around 15-20lakh nodes and 10lakh elements are shown in statistics.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
www.ijprems.com
editor@ijprems.com
Vol. 04, Issue 06, June 2024, pp: 1959-1973
e-ISSN : 2583-1062
Impact Factor: 5.725

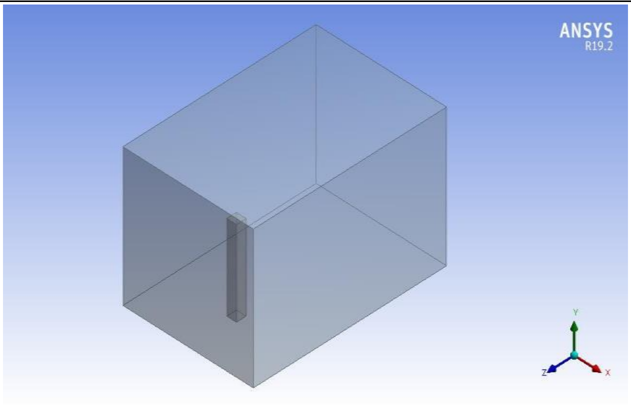


Figure :6 Computational domain from design Modeller

ANSYS fluent solver

Grid convergence studies were conducted for all the models. The mesh conditions are changed, increasing the number of elements, to achieve convergent values of coefficients of pressure and force. The free stream wind velocity applied is 10 m/s and the outlet gauge pressure is zero with no slip wall condition.

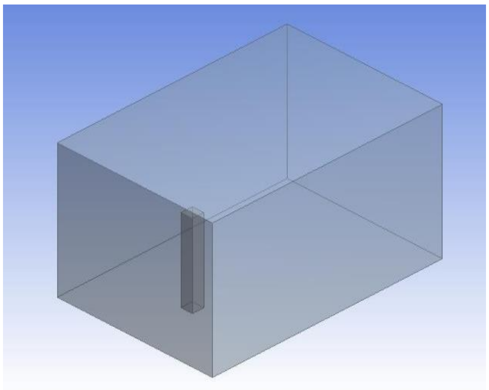
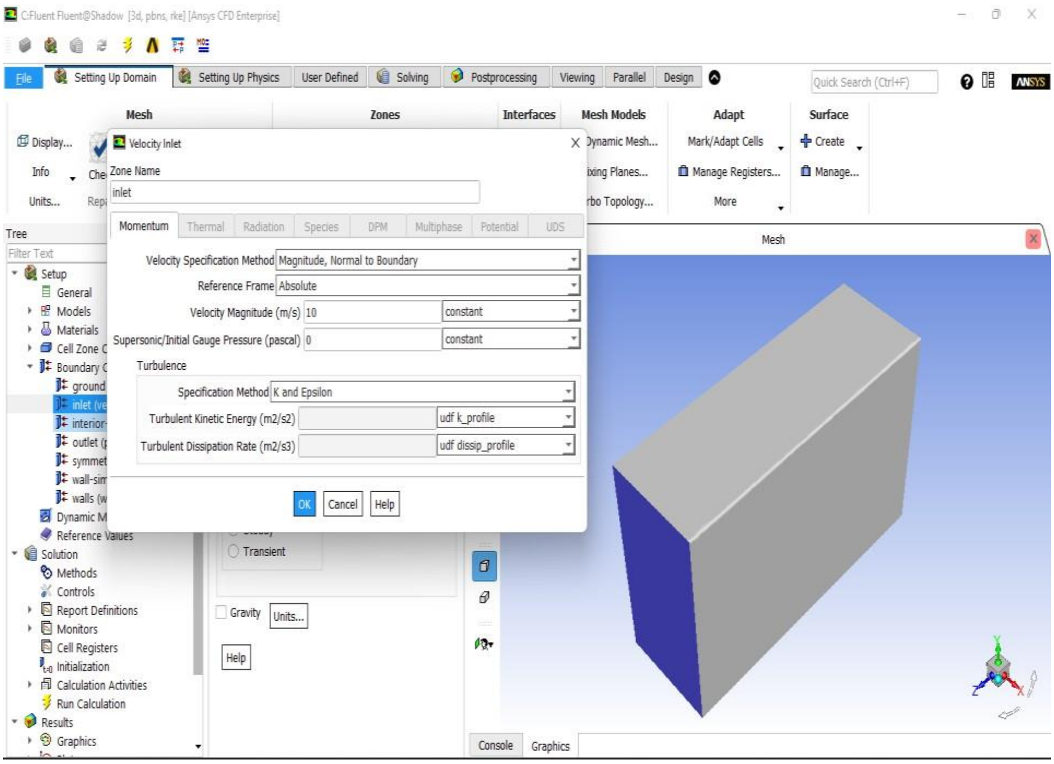


Figure :7 Tetrahedron Meshing **Figure : 8** Inlet Condition

Air properties

The air has properties such as density, viscosity etc. The air density has taken 1225kg/m^3. It is already available in the database of ANSYS FLUENT.



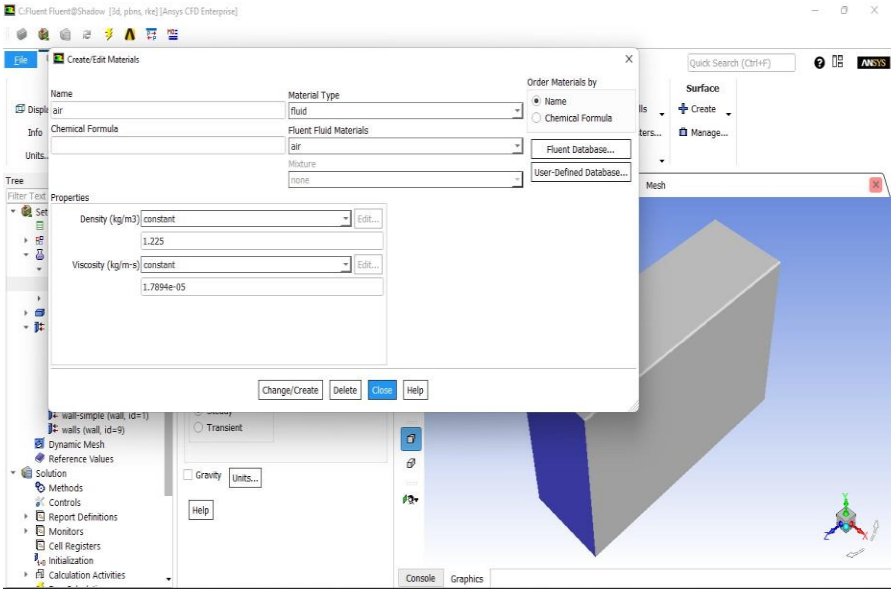


Figure :9 Air Properties

4. RESULTS

The pressure and velocity contour of the square tall building is shown as below:

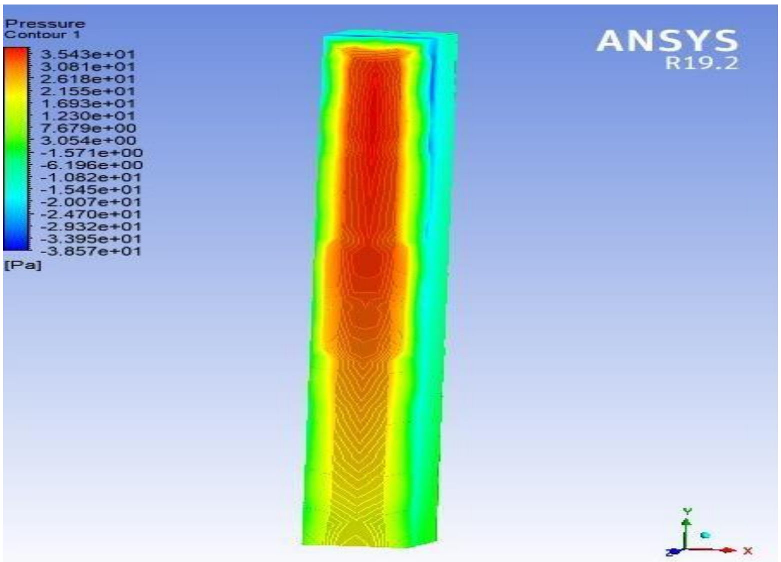


Figure :10 The pressure contours of the XY plane

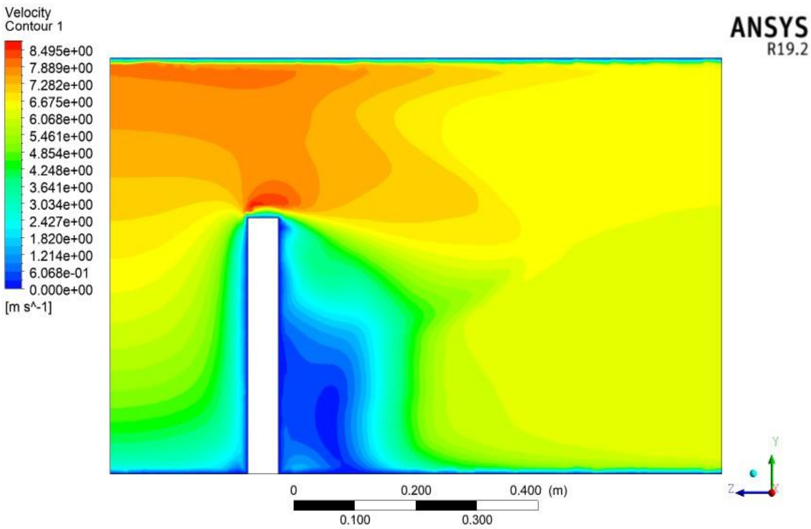


Figure : 11 Velocity contour

To confirm the accuracy of the work, the velocity profile that was developed must be evaluated. The wind tunnel data utilized and the computed profile are compared. Since the wind environments employed in both situations are similar, demonstrates that the simulated profile reflects the profiles from wind tunnel studies and the power law equation with sufficient accuracy. As a result, it may be said that the study's flow characteristics are acceptable.

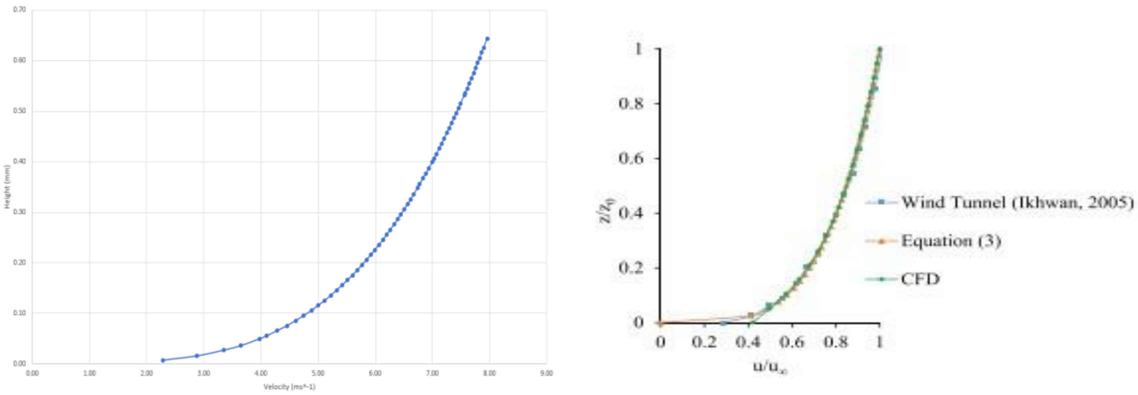


Fig :12 Wind profile Diagram

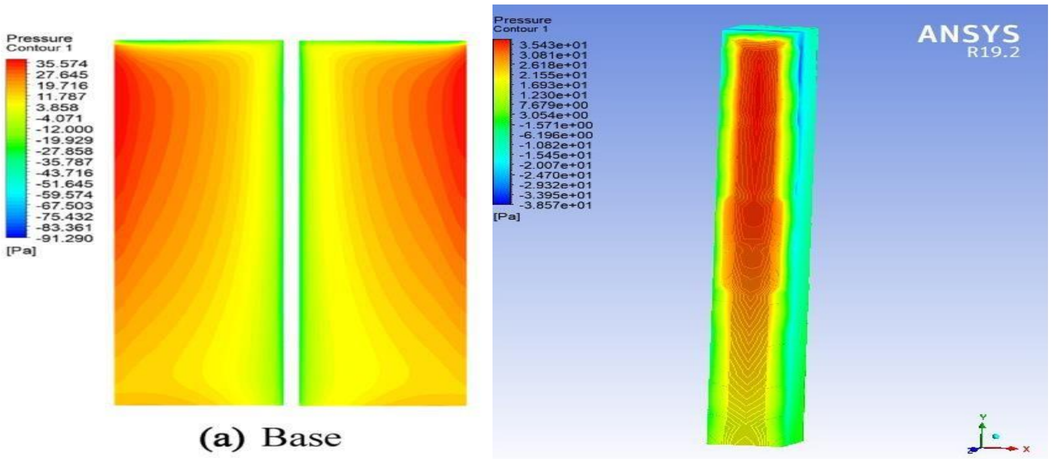


Fig.:13 Pressure contour

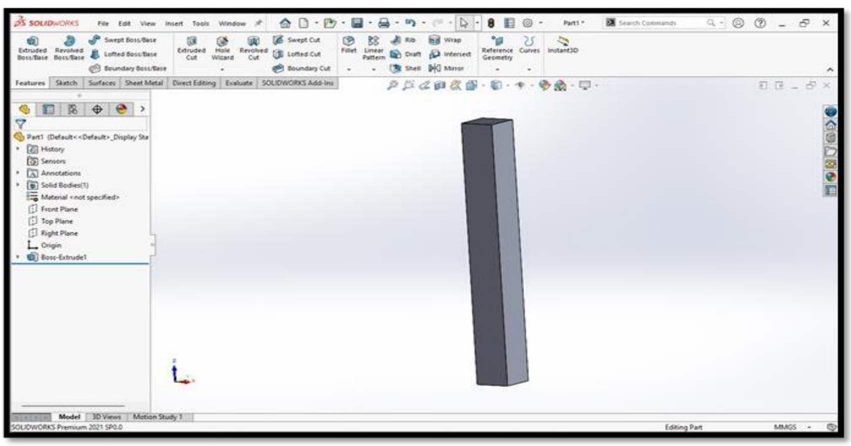
Table.1 Result comparison

Parameters	Literature Values	Validation Results
Drag Force Coefficient	0.5817	0.5

5. RESULTS AND COMPARISON

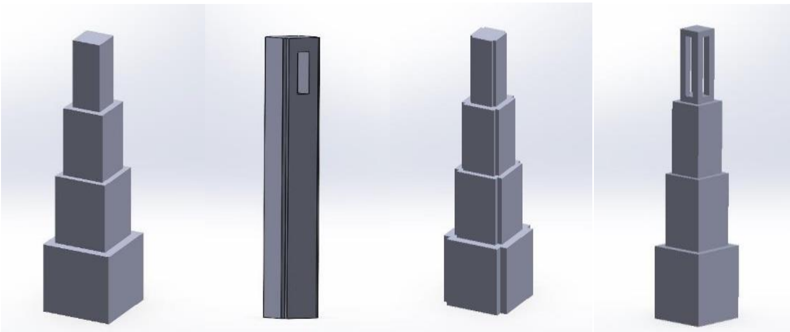
1. Results from the review paper and validation results are matching in the range of 85-100%.
2. Force coefficient values for validation results are lesser as compared to research paper values.
3. Pressure contours values are slightly lesser as compared to research paper values.
4. The velocity contour is used to predict the behaviour of the wind around the model.

6. MODEL DRAFTING



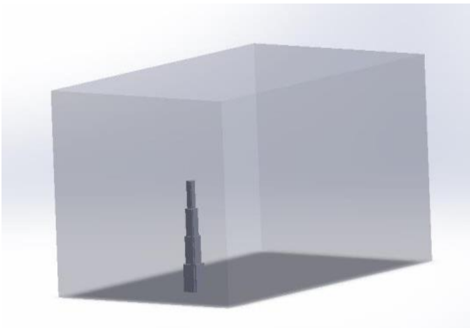
The models which are take in this study are with the help of reference paper. The models corner, setback and void modification is used in this study. The model used in the study ae modified square-shaped in plan. The building at full scale has a breadth of 50m and the height is 400m, giving a B/H ratio is 1/8 and taking Reynolds number as $2*10^5$. The model has been scaled down to a 1:1000 scale to facilitate wind tunnel tests. The models of the building were made with three aerodynamic modifications including setback, corner cut, void\opening and their combinations such as corner cut with opening, setback with corner cut, setback with void\opening. Model 1 is square, Model 2 is setback which are used for comparative study, Model 3 is corner cut with opening, Model 4 is setback with corner cut and Model 5 is setback with opening. The SolidWork Software and ANSYS Software is used for modelling purpose as shown in figure.

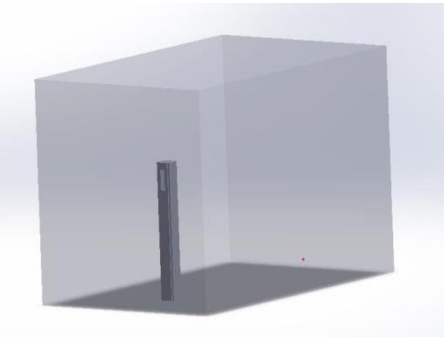
Model 1



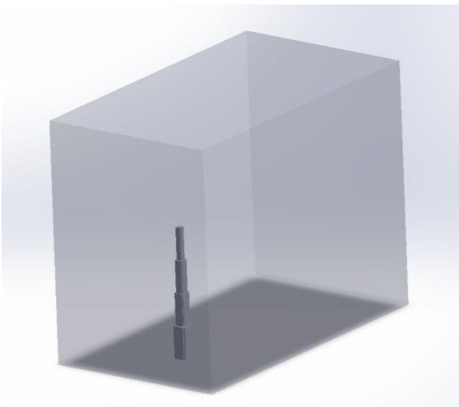
Model 1 Model 2 Model 3 Model 4

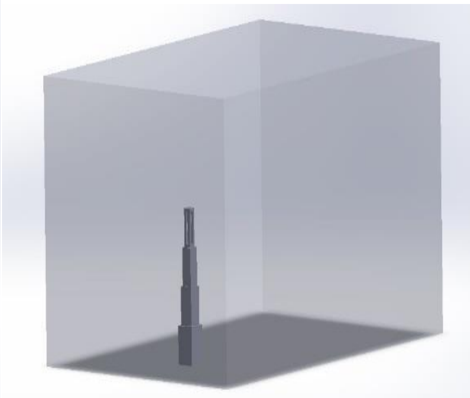
COMPUTATIONAL DOMAIN:





Model 5 Model 6





Model 7 Model 8

Figure : 14 Computational Domain

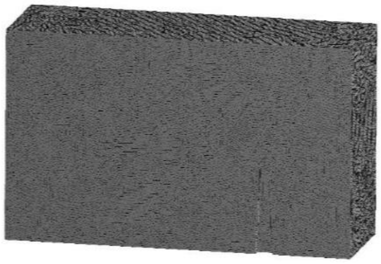
MESHING:

Meshing is done using ANSYS. Mesh refinements are made so that the solution variables converge down to acceptable levels. Tetrahedron meshing is used and is inflated near all boundaries.

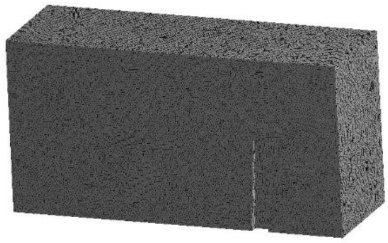
ANSYS FLUENT SOLVER

Ansys Fluent is the industry-leading fluid simulation software known for its advanced physics modelling capabilities and unmatched accuracy. Ansys Fluent is the industry-leading fluid simulation software known for its advanced physics modelling capabilities and unmatched accuracy.

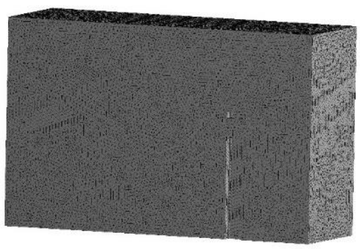
INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
Vol. 04, Issue 06, June 2024, pp: 1959-1973
e-ISSN : 2583-1062
Impact Factor: 5.725



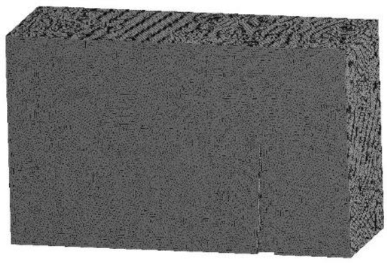
Model 9



Model 10



Model 11



Model 12

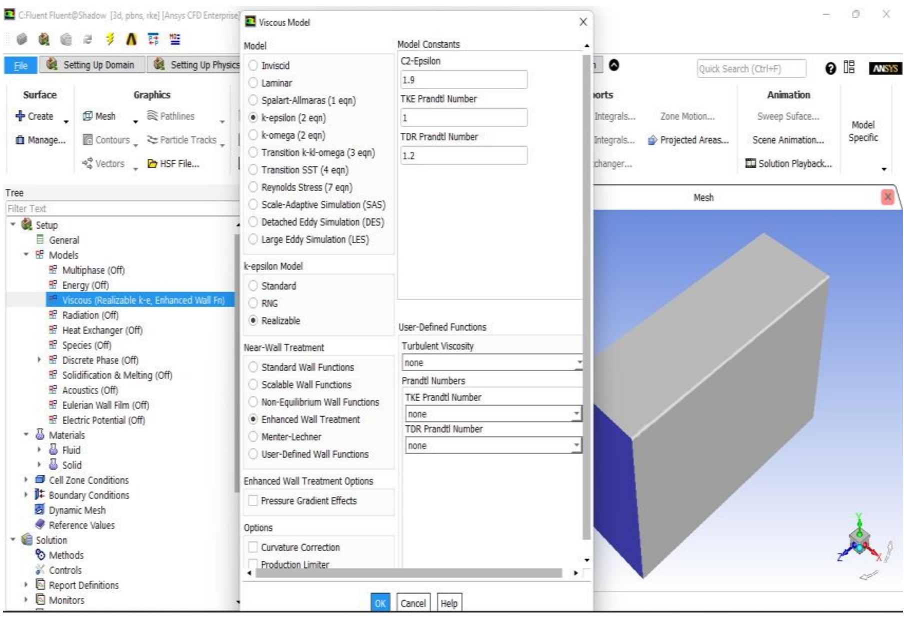


Figure :15 Fluent Solver

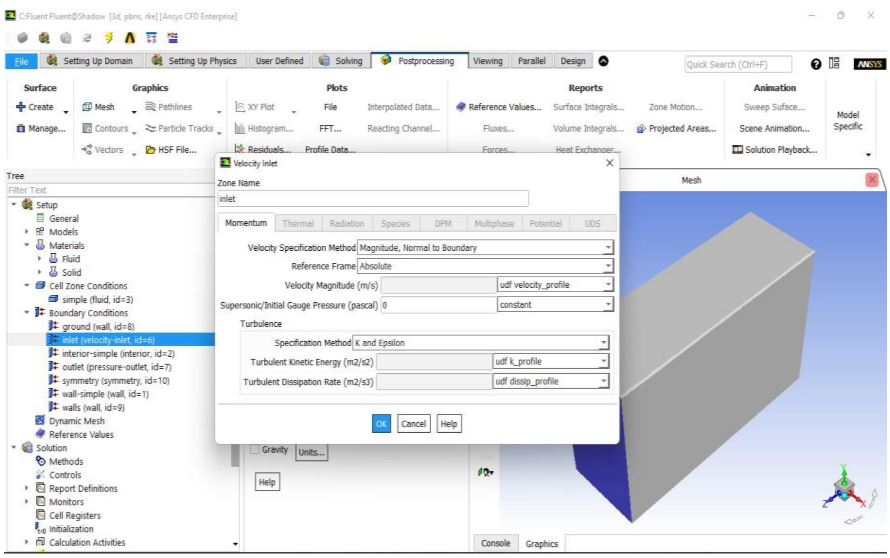


Figure :16 Inlet Velocity fluent

The velocity is provided at the top of model which is 10m/s

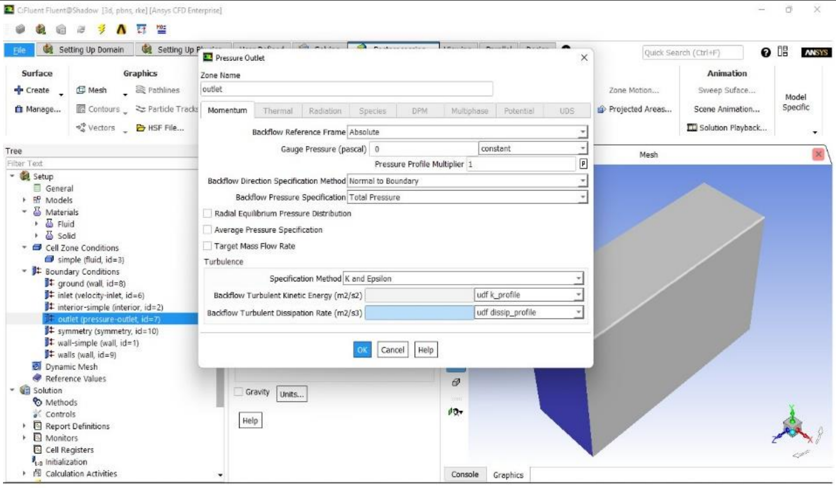


Figure :17 Boundary condition of wall

In the outlet boundary condition the gauge pressure is taken as zero pascal The boundary condition of wall is applied as enhanced wall treatment which is treated as no shear wall.

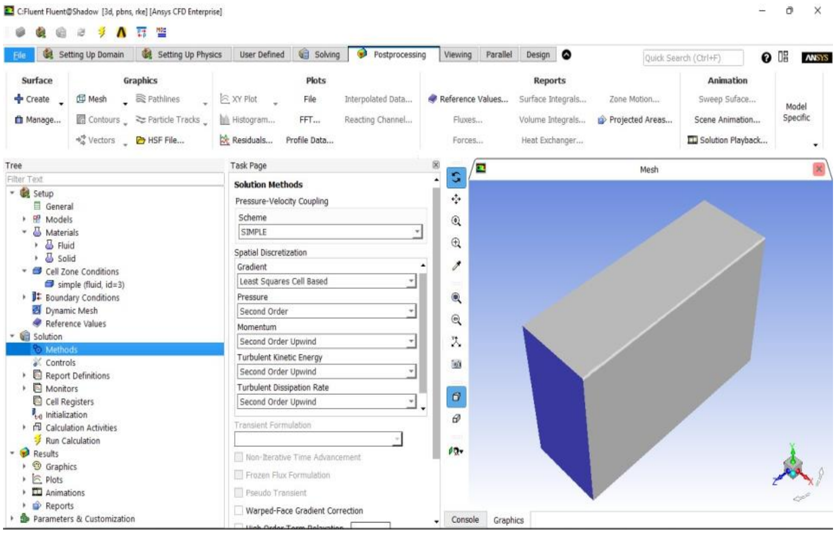


Figure :18 Solution methods

The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field as shown in below:

The run calculation is the process where all the iteration takes place and process takes time as per number of iteration.

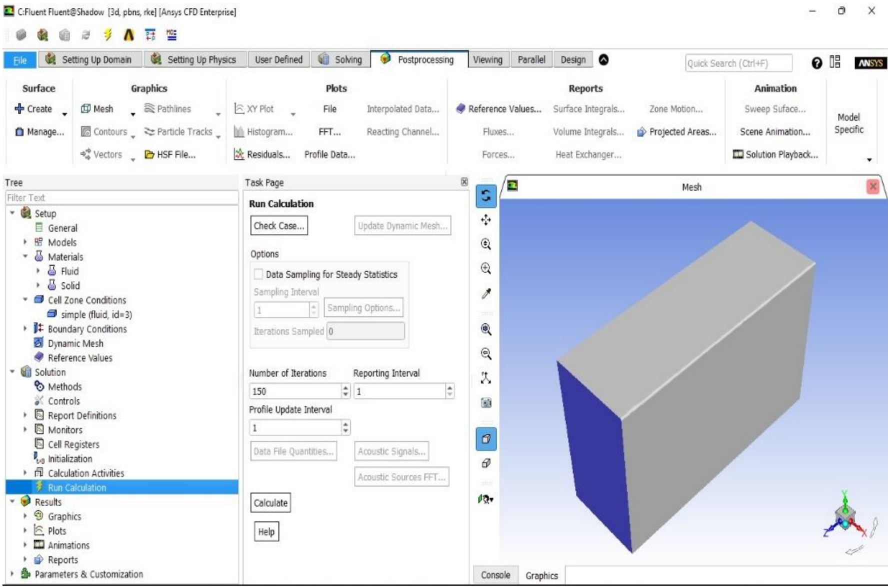


Figure. 19

Vol. 04, Issue 06, June 2024, pp: 1959-1973

e-ISSN : 2583-1062

Impact Factor: 5.725

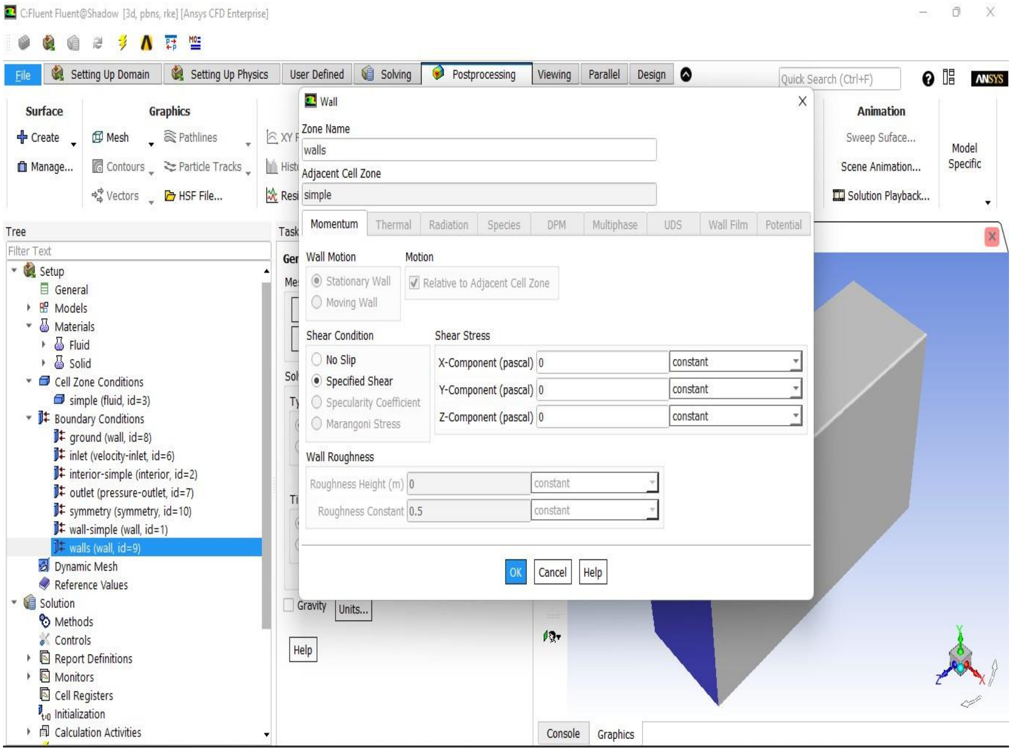
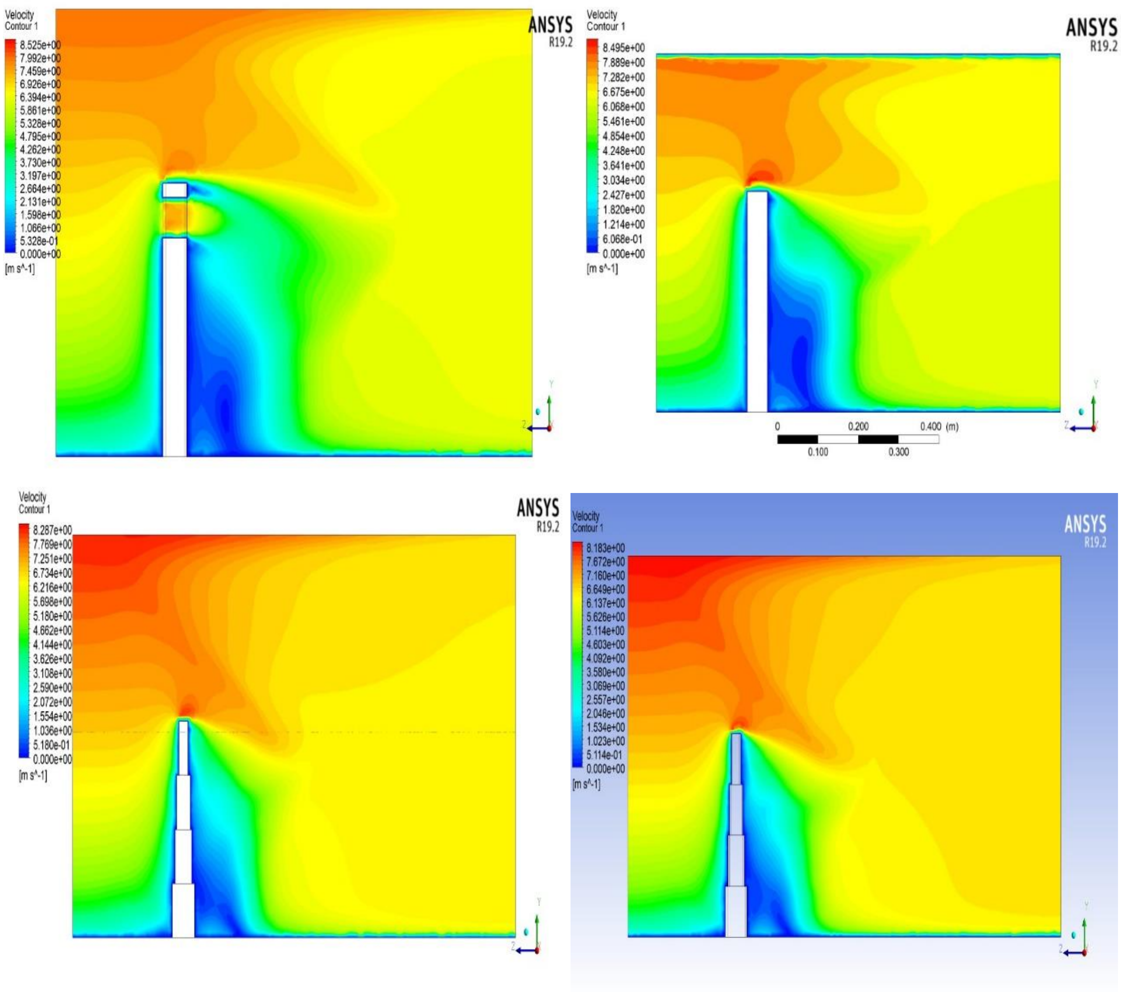


Figure :20 Run calculation

7. RESULTS

Velocity contour



The velocity contour from the elevation view are as follows:

e-ISSN : 2583-1062
Impact Factor: 5.725
Vol. 04, Issue 06, June 2024, pp: 1959-1973

The velocity contour from the top view as shown below:

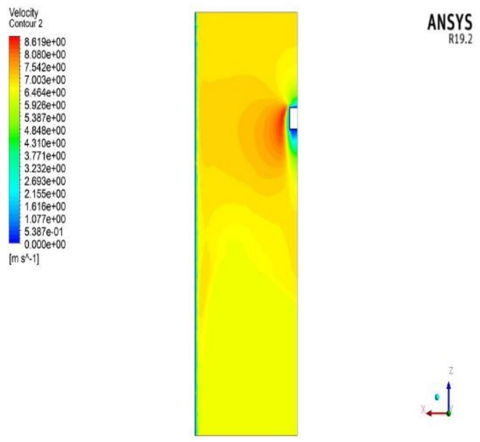


figure :21 Setbck +Cornercut

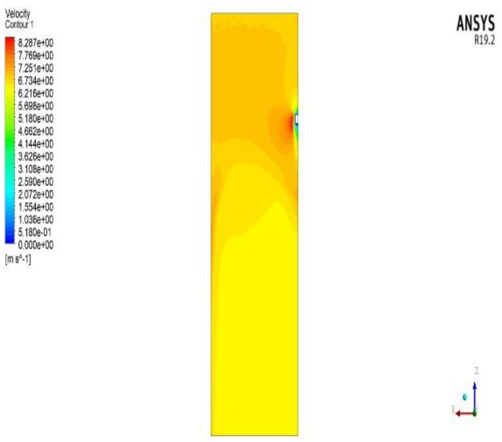


Figure :22 Setback model

PRESSURE CONTOUR

The pressure contour for all the model are as follows:

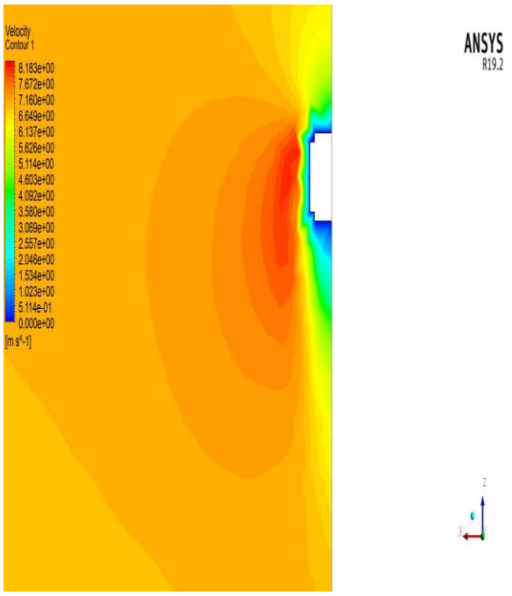


Figure :23 Setback model

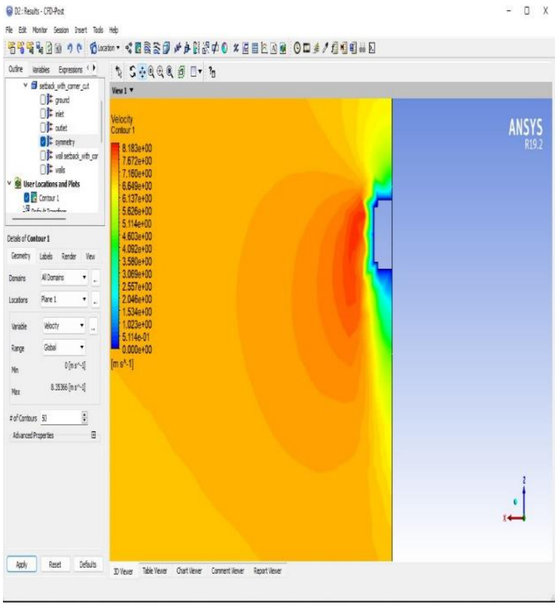


Figure : 24 Cornercut

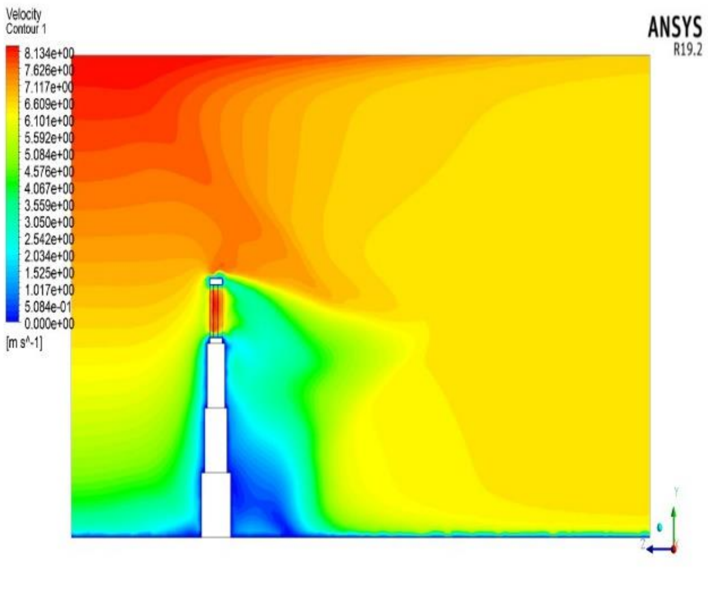


Figure.25 Setback + void

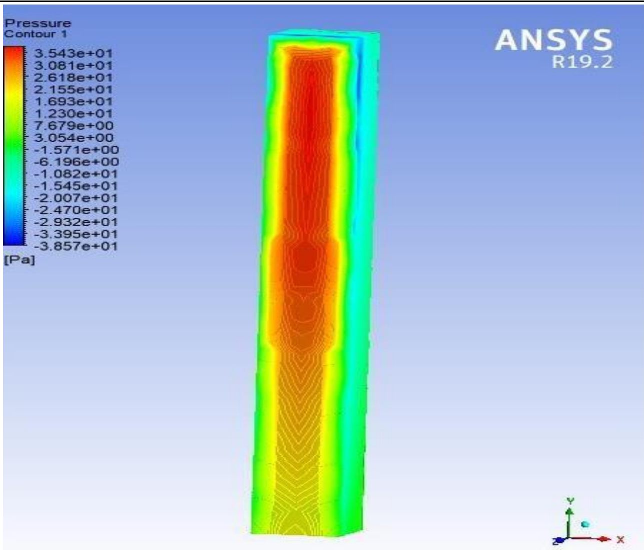
INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)

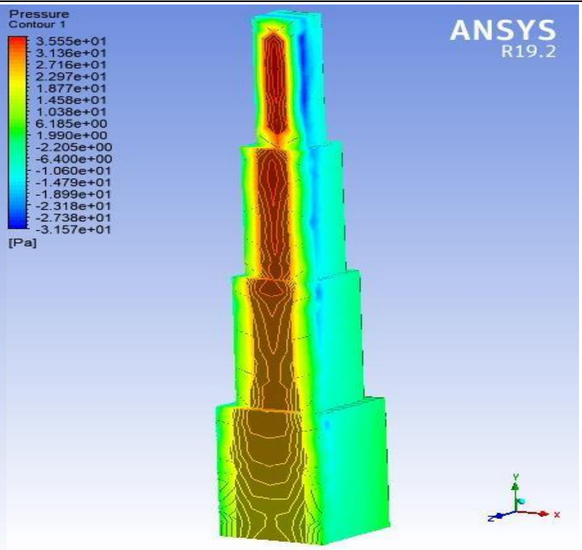
www.ijprems.com
editor@ijprems.com

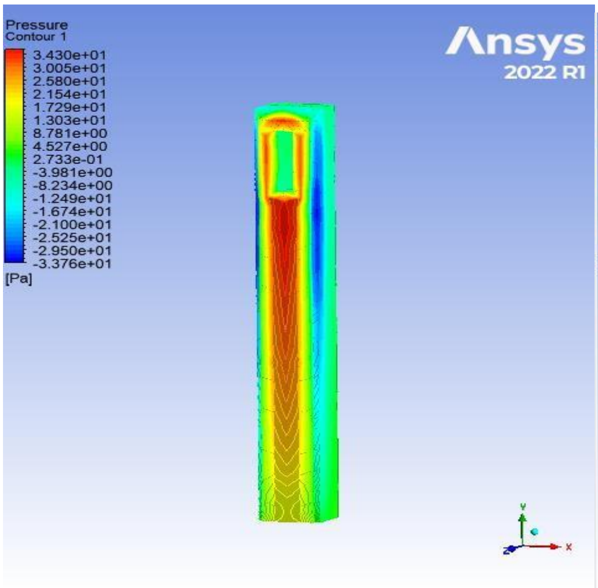
Vol. 04, Issue 06, June 2024, pp: 1959-1973

e-ISSN : 2583-1062

Impact Factor: 5.725







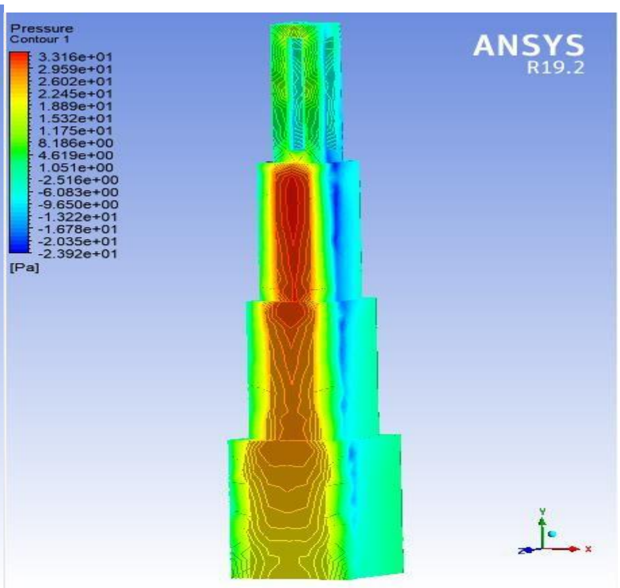


Figure :26 Square model

Figure :27 Setback model

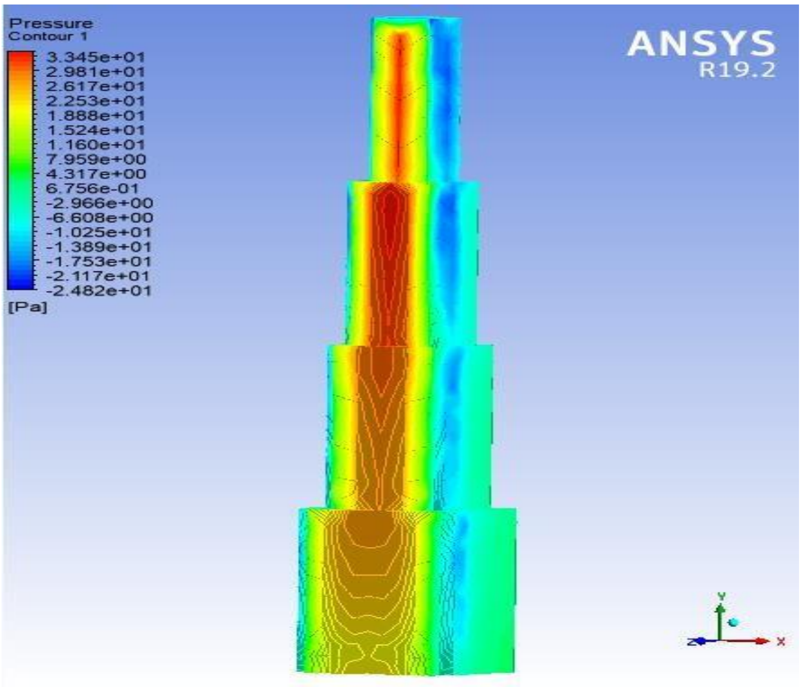


Figure.28 Setback + void model

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
www.ijprems.com
editor@ijprems.com
Vol. 04, Issue 06, June 2024, pp: 1959-1973
e-ISSN : 2583-1062
Impact Factor: 5.725

DRAG COEFFICIENT

The resistance force of fluid/air is known as drag force; this force always acts opposite to the motion of an object whih is moving submerged in a certain fluid/air.

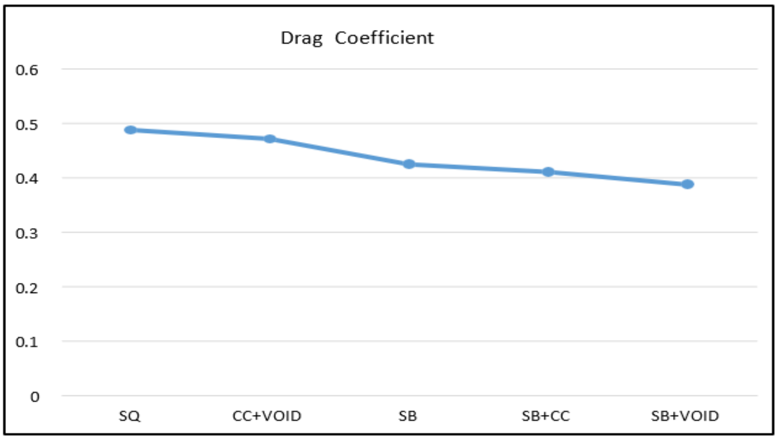


Figure : 29 Comparison of drag coefficient

LIFT COEFFICIENT

Lift coefficient is a dimensionless coefficient giving relationship between the lift generated by a lifting body to fluid density, fluid velocity and the associated reference area.

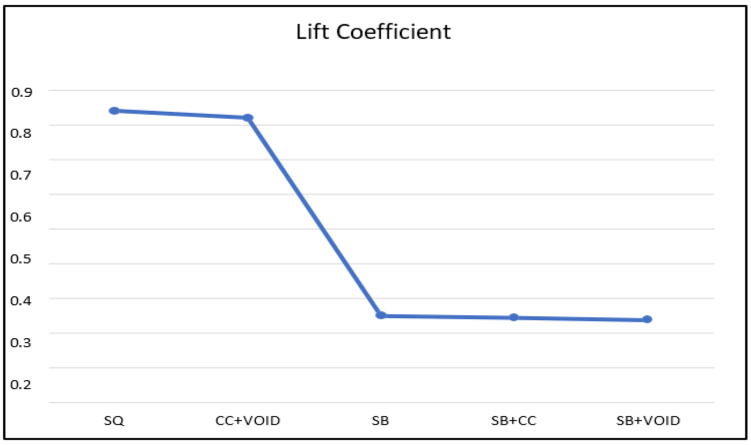


Figure.30 Comparison of lift coefficient

8. CONCLUSION

➢ **EFFECT OF DRAG COEFFICIENT**

- The drag of simple square model has highest value as compared with shaped modified models which is near to 1.
- The setback with opening model has lowest value as compared with other values of drag coefficient.
- The corner cut with opening model has value of drag coefficient which is lesser than square model by 3.33%
- The simple setback model has a value of drag coefficient which is lesser as compared to square model by 12.95%.
- The setback model is also show variation which is also less as compared to square model by 15.75%.
- The setback with corner cut model has greater value of drag coefficient when it is compared with setback with opening model by 5.71%.
- This variation helps to understand the shape/geometry effects on the drag of the building models.
- The setback model has major effect as compared to corner cut shape optimization method used for reducing the wind effect on the building.In setback modification, most efficient way to reduce drag is to combine it with opening/void as vented floors. It can reduce drag considerably by 8.78% when compared to simple setback model.
- . The corner cut model is combine with setback and void in which it shows that corner cut modification gives better efficient result with setback by 12.85% when compared to corner cut with opening model
- In opening modification with corner cut and setback, it shows reduction in drag coefficient in setback with opening by 19.14%.

- **EFFECT OF LIFT COEFFICIENT**

- The lift coefficient for the square model has maximum as compared with other models which comes near to 1.
- The setback with opening model has minimum value of lift coefficient which comes close to zero.
- In opening modification with corner cut and setback, it shows reduction in lift coefficient in setback with opening by 4.80%.
- In setback modification, most efficient way to reduce lift is to combine it with opening/void as vented floors. It can reduce lift considerably by 70.97% when compared to simple setback model.
- The corner cut model is combine with setback and void in which it shows that corner cut modification gives better efficient result with setback when compared to corner cut with opening model.

9. REFERENCES

[1] Yi Li lo, Xiang Tian, Kong Fah Tee, Qiu-Sheng Li and Yong-Gui Li (2018): Aerodynamic treatments for reduction of wind loads on high-rise buildings. Journal of Wind Engineering & Industrial Aerodynamics 172 (2018) 107–115. https://doi.org/10.1016/j.jweia.2017.11.006

[2] Yong-Gui Li, Ming-Yue Zhang, Yi Li, Qiu-Sheng Li and Si-Jia Liu (2020): Experimental study on wind load characteristics of high-rise buildings with the opening. Struct Design Tall Spec Build. 2020;e1734. https://doi.org/10.1002/tal.1734. © 2020 John Wiley & Sons, Ltd.

[3] Yong Chul Kim and Jun Kanda (2010): Characteristics of aerodynamic forces and pressures on square plan buildings with height variations. J. Wind Eng. Ind. Aerodyn. 98 (2010) 449–465. doi:10.1016/j.jweia.2010.02.004.

[4] Yong Chul Kim and Jun Kanda (2013): Wind pressures on tapered and set-back tall buildings. Journal of Fluids and Structures 39 (2013) 306–321. http://dx.doi.org/10.1016/j.jfluidstructs.2013.02.008.

[5] Yukio Tamura, Hideyuki Tanaka, Kazuo Ohtake, Masayoshi Nakai and Yongchul Kim (2010): Aerodynamic Characteristics of Tall Building Models with Various Unconventional Configurations. Structures Congress (2010)3104-3113. © 2010 ASCE.

[6] Yukio Tamura, Hideyuki Tanaka, Kazuo Ohtake, Masayoshi Nakai and Yongchul Kim (2012): Experimental investigation of aerodynamic forces and wind pressures acting on tall buildings with various unconventional configurations. J. Wind Eng. Ind. Aerodyn. 107–108(2012)179–191. https://linkinghub.elsevier.com/retrieve/pii/S016761051200116X.

[7] Y.C. Kim, Y. Tamura, H. Tanaka, K. Ohtake, E.K. Bandi and A. Yoshida (2014): Wind-induced responses of super-tall buildings with various atypical building shapes. J. Wind Eng. Ind. Aerodyn. 133 (2014) 191–199. http://dx.doi.org/10.1016/j.jweia.2014.06.004 & 2014 Elsevier Ltd. All rights reserved.

[8] Najah Assainar and Sujit Kumar Dalui(2020): Aerodynamic analysis of pentagon-shaped tall buildings. Asian Journal of Civil Engineering. https://link.springer.com/article/10.1007/s42107-020-00296-2

[9] Ahmed Elshaer and Girma Bitsuamlak (2018): Multiobjective Aerodynamic Optimization of Tall Building Openings for Wind-Induced Load Reduction. J. Struct. Eng., 2018, 144(10): 04018198. DOI: 10.1061/(ASCE) ST.1943-

[10] 541X.0002199. © 2018 American Society of Civil Engineers Chaorong Zheng, Yu Xie, Mahram Khan, Yue Wu, Jing Liu (2018): Aerodynamic treatments for reduction of wind loads on high-rise buildings. Journal of Wind Engineering & Industrial Aerodynamics 172 (2018) 107–115. https://www.sciencedirect.com/science/article/pii/S0167610517307109