A Study of Simulation of Computational Fluid Dynamics and Turbulence Modelling On Hip Roof Buildings

Rajendra Pd Verma[1], Dr. Eswara Veera Raghav Rao[2]

[1]Research Scholar of Sri Satya Sai University of Technology & Medical Sciences, Sehore, M.P., India.
[2]Research Supervisor of Sri Satya Sai University of Technology & Medical Sciences, Sehore, M.P., India.

Abstract

This study presents detailed portrayal of simulation work performed on low rise hip roof building utilizing different procedures of Computational Fluid Dynamics (CFD). The simulated information in the wind-tunnel has been utilized for the inflow boundary conditions, the boundary conditions, close to wall treatment, and so on for the CFD investigation. This wind tunnel was an open circuit, constant flow, suction type tunnel with a solitary blower fan (125 HP) and it has test segment of 2.1m x 2.0m. The length of test segment was 15m. A circular effuser profile with constriction ratio 9.5:1 alongside a square holed honeycomb at the passage (6mx6m) assists with building up a uniform smooth flow in the test segment. A physically controlled turntable was located at 12m downstream of the effuser on which unbending model under examination can be set.The results show that the roof pitch fundamentally influences the roof pressure on the hip-roofs and its variation has influenced both magnitudes just as example of distribution. Simulation of wind flow around two hip-roof buildings, set sideways and one after the other, utilizing RNG k-ε CFD simulation strategy have been finished. Impact of variation of distance of buildings on pressure coefficient is additionally concentrated by taking four instances of distance between the buildings, viz. 0.25B, 0.5B, 0.75B and 1.0B.

Keywords: *Computational Fluid Dynamics, Turbulence Modelling, HIP Roof Buildings.*

1. INTRODUCTION

Ongoing headways in computational offices made scientists to shift center from conventional methodologies of wind examines like wind-tunnel examinations or full-scale field perceptions to computer based logical or numerical methodologies like Computational Fluid Dynamics (CFD).

Use of CFD in wind engineering began around 30 years prior, in any case, the methodology was being utilized before in much wide scope of industrial and non-industrial application zones. As clarified in past sections, CFD essentially makes a virtual atmosphere to examine the impact of fluid-structure communication.

The improvements in the CFD planned to give an ability equivalent to other CAE (Computer-Aided Engineering) instruments, for example, stress examination codes. The CFD has primarily lagged behind when contrasted with different apparatuses because of the huge intricacy of the fundamental conduct of fluid and fluid–structure connection, which blocks exact fluid flow conduct, just as time economy. In ongoing past elite processing hardware and user-friendly interfaces with moderate accessibility have prompted upsurge of premium in CFD for the more extensive industrial application. The underlying venture expenses of a CFD capacity are equivalently a lot higher than other CAE instruments, while it is simply part of expenses for a great experimental office. Additionally, there are a few one-of-a-kind benefits of CFD over experiment-based ways to deal with fluid frameworks design, they are:

1. CFD decreases considerable times and cost.
2. Ability to contemplate frameworks where controlled experiments are troublesome or difficult to perform (e.g., large frameworks)
3. The examination can be under unsafe conditions at and past their ordinary performance limits (e.g., security studies and accident situations)

4. Practically limitless degree of detail of results.

The experimental expense, as far as office and labor, fluctuates with the quantity of information points and configurations tried in full scale and wind tunnel tests. While CFD simulations can deliver amazingly large volumes of results at virtually no additional expense and it is modest to perform parametric examinations.

2. COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD is the interaction of numerically demonstrating an actual wonder including fluid stream and tackling it mathematically utilizing the computational ability.

At the point when a designer is entrusted with designing another item, for example a triumphant race vehicle for the following season, aerodynamics assumes a significant part in the designing interaction. Nonetheless, streamlined cycles are not effectively quantifiable during the idea stage. Generally, the lone path for the architect to streamline his designs is to lead actual tests on item models. With the ascent of PCs and consistently developing computational power (on account of Moore's law!), the field of Computational Fluid Dynamics turned into a generally applied device for creating answers for fluid streams with or without strong association. In a CFD programming investigation, the assessment of fluid stream as per its actual properties like speed, pressure, temperature, thickness and consistency is directed. To basically create a precise answer for an actual wonder related with fluid stream; those properties must be considered all the while.

A numerical model of the actual case and a mathematical technique are utilized in a CFD programming instrument to dissect the fluid stream. For example, the Navier-Stokes (N-S) conditions are indicated as the numerical model of the actual case. This portrays changes in every one of those actual properties for both fluid stream and warmth move. A numerical model shifts as per the substance of the issue, for example, heat move, mass exchange, stage change, synthetic response, and so on Also, the unwavering quality of a CFD examination exceptionally relies upon the entire design of the cycle. The check of the numerical model is critical to make a precise case for tackling the issue. Plus, the assurance of legitimate mathematical strategies is the way to produce a dependable arrangement. The CFD investigation is a vital component in creating a reasonable item improvement measure, as the quantity of actual models can be decreased definitely.

CFD Analysis

To lessen power losses and to improve the lifetime of the cutting edges of wind turbines in wind ranches, it is important to acquire a decent comprehension of the conduct of the wakes. A particularly understanding can be acquired by mathematical or exploratory examination. Further developed CFD models are currently accessible for the wake modeling, and addressing the three-dimensional (3-D) Navier–Stokes conditions. Nonetheless, two significant progressing difficulties are the precise portrayal of the rotor sharp edges and disturbance modeling. There are various purposes behind zeroing in on mathematical reenactment rather than on tests: Full-scale, great quality examinations are exorbitant, and have been restricted basically to give worldwide data on the stream field. CFD can give nitty gritty data both upstream and downstream of the turbine. Because of the fluctuation in air conditions, it is hard to track down the shared impact of turbines on one another. Advancement of a wind ranch format in a test setting is practically unthinkable. Notwithstanding, CFD calculations of wind ranches face different challenges: Accurately modeling both the stream over the turbine edges and the stream in the all over wake requires enormous PC assets. The stream field displays scales that range from the size of little swirls in the limit layer on the sharp edge, to the stature of the climatic limit layer. Reproducing the disturbance in the stream precisely and forestalling fake dispersion is a continuous test.

Applications of CFD

Where there is fluid, there is CFD. Having referenced previously, the underlying stage to direct a CFD reproduction is indicating a proper numerical model of the real world. Rapprochements and presumptions provide guidance through arrangement cycles to inspect the case in the computational area. For example, fluid stream over a circle/chamber is a dreary issue that has been educated by the instructor as an illustration in fluids courses. A similar marvel is basically accessible in the development of mists in the air which is undoubtedly huge

→ **Incompressible and Compressible stream**

In the event that compressibility turns into a non-immaterial factor, this sort of examination assists you with discovering arrangements in a hearty and exact manner. One model would be a Large Eddy Simulation of stream around a chamber.

→ **Laminar and Turbulent stream**

Diverse choppiness models assume a part in this sort of examination. A ton of registering power is needed to address disturbance recreations and its complex mathematical models. The trouble of choppiness is the reenactment of changes after some time. The whole space where the recreation happens should be recalculated after each time step. The valve is one potential use of a tempestuous stream examination.

→ **Mass and Thermal vehicle**

Mass vehicle reenactments incorporate smoke spread, aloof scalar vehicle or gas conveyances. To settle these sorts of reproductions, OpenFOAM solvers are utilized. Warmth exchanger recreations are one potential application.

CFD using SimScale

Computational Fluid Dynamics (CFD) is the part of CAE that permits you to reenact fluid movement utilizing mathematical methodologies. The cloud-based CFD programming office of SimScale permits the investigation of a wide scope of issues identified with laminar and violent streams, incompressible and compressible fluids, multiphase streams and then some. Those designing issues are tackled utilizing various incorporated mathematical solvers and innovations.

3. COMPARISON OF INTERNATIONAL CODES WIND LOADS AND CFD RESULTS FOR LOW RISE BUILDINGS

The foundational layout of the building incorporates the estimation of the applied wind loads asone of the significant things in the design interaction. The accessible codes information for the recorded constructions in these codes separated from wind burrow tests however not cover every one of the designed designs types. There is a need to give the designer design wind boundaries thinking about that the experimental information is expensive and not generally accessible. This examination expects to work with the essential wind parameters for the foundational layout, for example, pressure appropriation, drag coefficient for some building adopting the Computational Fluid Dynamics (CFD) procedures. The approval of the application of CFD procedure with existing wind burrow results for certain designs is researched. At that point the application of the CFD methods performed on certain constructions, for example, single range short gable structure with mono and twofold inclines, brackets and arches. The essential boundaries considered in the examination of peak building including roof slops, wind course. Furthermore, for peak buildings, the diverse roof zones coefficients of pressing factor evaluated to contrast the use of CFD procedure and the global wind norms and codes of training.

The current wind loads codes depend on estimated estimations of pressing factor coefficients from limit layer wind burrow tests for the treated shapes and kinds of buildings. A few states of buildings and constructions are not recorded in these codes, and this missed design information should be covered through wind burrow tests. Nonetheless, wind burrow tests are broad and relentless and not available for the majority of architects, and furthermore it is hard to reproduce full scale Reynolds numbers experiment-count. The mathematical assurance of wind loads acquires advantage over the utilization of scaled models in limit layer tests. It is not difficult to mimic any Reynolds number, change the limit conditions and perform parametric investigation. Subsequently, mathematical reenactment of wind issues covers with incredible expected the hole of design wind information and broadens the codes of training for any mathematical state of constructions. The assurance of the stream field and the wind pressure appropriation of the buildings are one of the fundamental goals of a few test explores.

Low ascent building has broad wind burrow examinations to decide surface stream field and mean pressing factor coefficients. A solitary cubic building with various shapes and exhibit of cubic buildings tried to report wind pressure powers. Study inspected the power spectra of pressing factor fluctuation of pitched roof building for various wind points assault. The impact of model extents are inspected for four limited scope pitched roof buildings to determine pressure dispersion at mid building length. The variation of wind pressure with roof pitches for hip roof structure dictated by wind burrow test performed utilizing wind burrow inspected the long low-rise pitched roof building with steep roof to examine the impact of the length to traverse proportion on the outside wind pressure distribution. The qualities of wind stream including the perception, the temperature dispersion and the pressing factor appropriation were studied on mono slant limited scope model through wind tunnel test tried through wind tunnel tests, two limited scope obstructions with angled, and a pitched roof shapes to decide the violent motor energy profiles. Study formed a wind burrow experiments of three Gable buildings with various roof pitches slants to the pressing factor variety along every one.

The circular silos with cone shaped roofs tried in a barometrical boundary layer wind burrow. The outcomes describe the variety of wind pressure dispersion with the incline of pitch, stature to distance across proportion. To give aerodynamic information base to the wind load assessment framework, tried a limited scale model of spherical domes with various ascent/range and stature/length proportions for two kinds of violent limit layers. The investment tentatively and mathematically the wind stream field around a surface-mounted side of the equator of a fixed tallness for two different turbulent limit layers (meager and thick). The investigation consists of fierce stream field and power estimations for the two types of limit layers. Aerodynamic wind burrow tests on exaggerated paraboloid roofs with circular plane shapes have been explained with the target of ascertaining pressure coefficients. An examination is likewise made among circular and circular shapes to exhibit the effectiveness of the elliptical shape. It examines wind loads effects on roof sheathing of low-ascent, wood-outline houses. They experimentally examined 34 models to cover diverse parameters including roof shape, roof incline, building height, upstream territory and the presence of encompassing structures. It determines through wind burrow tests the drag coefficients of nine little scaled calculated steel transmission towers.

4. FLUID FLOW GOVERNING EQUATIONS

The numerical arrangement of any fluid flow issue requires the arrangement of the overall conditions of fluid motion, i.e., the Navier-Stokes and coherence conditions.

The Navier-Stokes' conditions, named after Claude Louis Navier and George Gabriel Stokes, depict the motion of fluid substances like liquids and gases. These conditions build up that adjustments in energy in infinitesimal volumes of fluid are basically the amount of dissipative viscous forces (like friction), changes in pressure, gravity, and different forces acting inside the fluid.

The overall type of three dimensional incompressible momentary Navier-Stokes' conditions is as follows, in Cartesian tensor structure (subtleties of the induction of these conditions are alluded to Young:

$$\frac{\partial(\rho u_i)}{\partial t} = -\frac{\partial\left(\rho u_i u_j\right)}{\partial x_j} - \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + F$$

Acceleration term	Convection term	Pressure gradient	Effects of viscosity	Body force

and the continuity equation:

$$\frac{\partial\rho}{\partial t} + \frac{\partial\rho_i}{\partial x_i} = 0$$

They are quite possibly the most helpful arrangements of conditions since they depict the physical science of countless marvels of academic and economic interest. They might be utilized to model climate, sea flows, water flow in a pipe, flow around an air foil (wing), and the motion of stars inside a system. In that capacity, these conditions in both full and improved on structures are utilized in the design of airplane and vehicles, the investigation of blood flow, the design of force stations, the examination of the impacts of pollution, and so forth Combined with Maxwell's conditions, they can likewise be utilized to model and study magneto-hydrodynamics.

The Navier-Stokes' conditions are likewise of incredible interest in a simply numerical sense. Fairly shockingly, given their wide scope of practical uses, mathematicians still can't seem to demonstrate that in three measurements their answers consistently exist, or that in the event that they do exist, they don't contain any vast qualities, singularities or discontinuities. These issues are known as the Navier-Stokes' presence and smoothness issues.

In spite of what is regularly found in strong mechanics, the Navier-Stokes' conditions direct not position but instead velocity. An answer of the Navier-Stokes' conditions is known as a velocity field or flow field, which is a portrayal of the velocity of the fluid at a given point in reality. When the velocity field is addressed for, different amounts of revenue, (for example, flow rate, drag force, or the way a 'molecule' of fluid will take) might be found.

The straightforward instances of flow in laminar system can be tackled systematically by utilizing the coherence and Navier-Stokes' conditions. A large portion of the flows of the engineering importance are fierce. Fluid engineers need admittance to practical apparatuses equipped for addressing the impacts of turbulence.

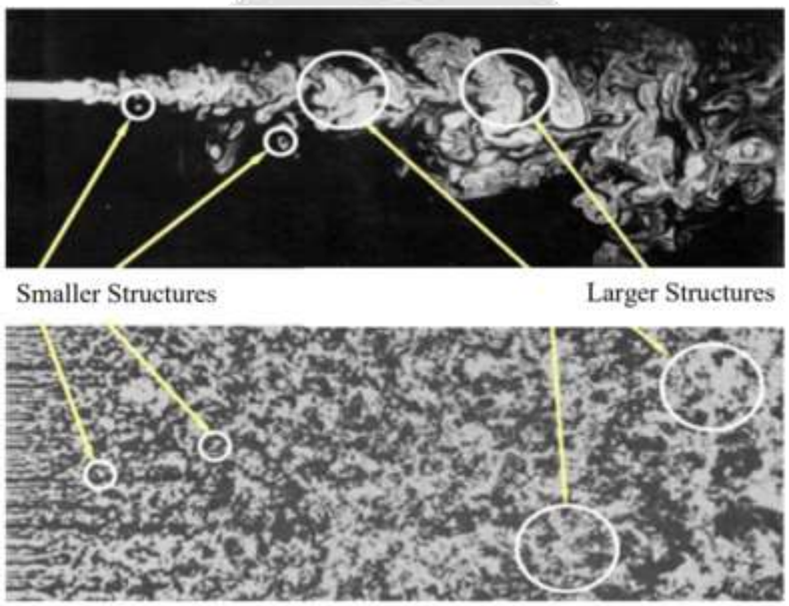

Figure 1: Small and large Turbulent eddies

5. COMPUTATIONAL FLUID DYNAMICS CODES WORKING

CFD codes are structured around the numerical calculations that can handle fluid flow issues. To give simple admittance to their addressing power all business CFD bundles incorporate refined user interfaces to enter issue parameters and to look at the outcomes. Thus, all codes contain three primary components: (i) a pre-processor, (ii) a solver and (iii) a post-processor.

1. Pre-Processor

Pre-processing comprises of the input of a flow issue to a CFD program through an administrator friendly interface and the ensuing transformation of this input into a structure appropriate for use by the solver. The user exercises at the pre-processing stage include:

- Definition of the calculation of the region of interest: the computational domain.
- Grid generation: the sub-division of the domain into various more modest, non-covering sub-domains: a grid (or cross section) of cells (or control volumes or components).
- Selection of the physical and chemical marvels that should be modelled.
- Definition of fluid properties.
- Specification of proper boundary conditions at cells which agree with or contact the domain boundary.

The answer for a flow issue (velocity, pressure, temperature and so forth) is characterized at hubs inside every cell. The accuracy of a CFD arrangement is represented by the quantity of cells in the grid. It very well may be said overall that the larger the quantity of cells the better the arrangement accuracy. Both the accuracy of an answer and its expense regarding important computer hardware and calculation time are reliant on the fineness of the grid. Optimal meshes are frequently non-uniform: finer in territories where large variations happen from one point to another and coarser in regions with generally little change. Eventually such projects will consequently refine the grid in spaces of fast variations. A generous measure of fundamental advancement work actually should be done before these procedures are sufficiently powerful to be consolidated into business CFD codes. At present it is still up to the abilities of the CFD user to design a grid that is an appropriate compromise between wanted accuracy and arrangement cost.

2. Solver

There are three particular surges of numerical arrangement procedures: finite distinction, finite element and spectral techniques. In diagram the numerical strategies that structure the premise of the solver play out the following advances:

- Approximation of the obscure flow factors through straightforward capacities.
- Discretisation by replacement of the approximations into the overseeing flow conditions and resulting mathematical controls.
- Solution of the algebraic conditions.

The fundamental contrasts between the three separate streams are related with the manner by which the flow factors are approximated and with the discretisation measures.

- **Finite Difference Method**

Finite contrast techniques portray the questions' Φ of the flow issue through point samples at the hub points of a grid of co-ordinate lines. Truncated Taylor arrangement extensions are frequently used to produce finite contrast approximations of subsidiaries regarding point samples of Φ at every grid point and its immediate neighbours. Those subordinates showing up in the governing equations are supplanted by finite contrasts yielding an algebraic equation for the values of Φ at every grid point. Smith (1985) gives a far-reaching record of all aspects of the finite distinction technique.

- **Finite Element Method**

Finite element techniques utilize basic piecewise capacities (e.g., Linear or quadratic), that are legitimate on elements to depict the nearby variations of obscure flow factors Φ. The governing equation is definitely fulfilled by the specific arrangement Φ. On the off chance that the piecewise approximating capacities for Φ are subbed into the equation, it won't hold precisely and a remaining is characterized to quantify the errors. Next the residuals (and consequently the errors) are limited in some sense by increasing them by a bunch of weighting capacities and coordinating. Subsequently, a bunch of algebraic equations for the obscure coefficients of the approximating capacities. The theory of finite elements has been grown at first for structural stress examination. A standard work for fluids applications is Zienkiewicz and Taylor (1989).

- **Spectral Method**

Spectral techniques inexact the questions through truncated Fourier arrangement or arrangement of Chebyshev polynomials. Dissimilar to the finite contrast or finite element approach the approximations are not nearby but rather legitimate all through the whole computational domain. Once more, we supplant the questions in the governing equation by the truncated arrangement. The imperative that prompts the algebraic equations for the coefficients of the Fourier or Chebyshev arrangement is given by a weighted residuals idea like the finite element technique or by causing the estimated capacity to harmonize with the specific arrangement at various grid points. Additional data on this specific technique can be found in Gottlieb and Orszag (1977).

- **Finite Volume Method**

The finite volume technique was initially evolved as an exceptional finite contrast definition. It is key to four of the five fundamentals financially accessible CFD codes: PHOENICS, FLUENT, FLOW3D and STAR-CD. The numerical calculation comprises of the following advances:

- Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain.
- Discretisation includes the replacement of an assortment of finite-distinction type approximations for the terms in the incorporated equation addressing flow cycles like convection, diffusion and sources. This believer the fundamental equations into an arrangement of algebraic equations.
- Solution of the algebraic equations by an iterative strategy.

The initial step for the control volume integration, recognizes the finite volume strategy from any remaining CFD procedures. The subsequent assertions express the (specific) conservation of significant properties for each finite size cell. This unmistakable relationship between the numerical calculation and the hidden physical conservation principal structures one of the fundamental attractions of the finite volume technique and makes its ideas a lot more straightforward to comprehend by engineers than finite element and spectral strategies. The conservation of an overall flow variable Φ, for instance a velocity part or enthalpy, inside a finite control volume can be communicated as a harmony between the different cycles having a tendency to increment or reduction it. The fundamental physical wonders are perplexing and non-linear so an iterative solution approach is required. The most mainstream solution systems are the TDMA line-by-line solver of the algebraic equations and the SIMPLE calculation to guarantee right linkage among pressure and velocity. Business codes may likewise give the user a determination of further, later, procedures, for example, Stone's calculation and conjugate gradient strategies.

3. Post-Processor

As in pre-processing a gigantic measure of improvement work has as of late occurred in the post-processing field. Attributable to the expanded fame of engineering workstations, a considerable lot of which have remarkable graphics abilities, the leading CFD bundles are presently outfitted with flexible information representation devices. These incorporate:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc.)
- Colour postscript output

All the more as of late these facilities may likewise incorporate liveliness for dynamic outcome show and notwithstanding graphics all codes produce trusty alphanumeric yield and have information export facilities for additional control outer to the code.

6. CONCLUSION

The values of the mean wind pressure coefficients are contrasted and the accessible wind tunnel information to check the accuracy of the CFD results. It tends to be finished up effectively from the discoveries that the numerical outcomes fall in the scope of the exploratory information as a rule, yet considerable errors exist close to the corners and sharp bends. These disparities are on the grounds that in the wind-tunnel tests it is hard to fix pressure taps at sharp bend and close to eaves, subsequently the pressure coefficients were not recorded in these regions. The pressure coefficients were either inserted or extrapolated close to the sharp bends.

- ✓ The computational outcomes got in the current investigation are in acceptable arrangement and they do show the right magnitude and patterns in the vast majority of the cases. It was likewise seen that the RANS turbulence models nearly anticipate the tentatively gotten leeward pressure distribution.

- ✓ All the turbulence models i.e., the Standard k-ε, the RNG k-ε, the Realizable k-ε the Standard k-ω and SST k-ω strategies gave comparable example of distribution of pressure coefficients; in any case, the tentatively got CFD values vary a bit in certain segment roof where the slope changes, leeward region and the zones close to sharp edges. The explanation might be because of the over creation of the violent kinetic energy close to the sharp edges. Precisely simulation of flow field is needed in the recirculation, wake and separation regions around the feign body. No improvement of the outcomes found with grid refinement past certain value.

- ✓ It has been tracked down that the Standard k-ε turbulence model has gained notoriety for its efficiency and simple execution. The Standard k-ε turbulence model can anticipate the overall wind conditions around the building sensibly well, aside from those in the isolated regions. Notwithstanding, the RNG k-ε turbulence model was discovered to be the most ideal decision among all the turbulence models for quick arrangements and great outcomes.

- ✓ The results show that the roof pitch fundamentally influences the roof pressure on the hip-roofs and its variation has influenced both magnitudes just as example of distribution.

- ✓ An expansion in the pitch of a hip roof caused an increment in the suction pressure for all attack angles aside from 0° in which pattern was opposite. 30° pitched hip roof, at 45° wind episode angle, encountered the greatest suction pressure at roof corner among the three tried hip roof models.

- ✓ The plan aspect ratio fundamentally influences the magnitude of the pressure coefficients; be that as it may, the example of pressure distribution remains practically a similar roof pressure on the hip-roofs. Variation of plan aspect ratio has altogether influenced magnitude while example of distribution stays undisturbed.

- ✓ An expansion in the arrangement aspect ratio from 1.66 to 2.5 caused decline in the suction pressure for all angles of incidence. Model having plan aspect ratio of 1.66, with 45° wind attack angle, experienced greatest suction pressure on roof.

- ✓ Variation of height aspect ratio (0.4 to 0.6) gently influence the magnitude while example of pressure distribution stays undisturbed.

- ✓ The results showed that the height aspect ratio gently influences the roof pressure on the hip-roofs, besides now and again. The greatest suction of the roof was capable when the wind occurrence angle was 45^{o} at a height aspect ratio of 0.6.

- ✓ The overhang ratio additionally gently influences the roof pressure on the hip-roofs. The greatest suction pressure is competent at an overhang ratio of 0.38 when the wind occurrence angle is 45^{o}.

- ✓ Impact of wind pressure coefficients on the interference and shielding of hip-roof building utilizing RNG k-ε turbulence model at a dispersing of 0.25B, 0.5B, 0.75B and 1.0B have additionally been contemplated. It has been noticed the suction pressure diminishes when distance between the two buildings increments.

- ✓ Simulation of wind flow around two hip-roof buildings, set sideways and one after the other, utilizing RNG k-ε CFD simulation strategy have been finished. Impact of variation of distance of buildings on pressure coefficient is additionally concentrated by taking four instances of distance between the buildings, viz. 0.25B, 0.5B, 0.75B and 1.0B.

7. REFERENCES

1. A.K. Roy, N. Babu and P.K. Bhargava, ‘Atmospheric Boundary Layer Airflow Through CFD Simulation on Pyramidal Roof of Square Plan Shape Buildings’, VI National Conference on Wind Engineering, pp. 291–299, Dec. 14–15, 2012.
2. Abdelfatah, Nourhan & Mahmoud, Gamal & Eid, Nasr. (2017). Comparative study of international codes wind loads and CFD results for low rise buildings. Alexandria Engineering Journal. 57. 10.1016/j.aej.2017.11.023.

3. Abdelrahman, Mahmoud & Farag, Osama & Moustafa, Wael. (2016). The Role of CFD Simulation Software in Improving Residential Buildings' Efficiency: Case Study on Youth Housing in New Damietta . Journal of Al-Azhar University Engineering Sector. 2016
4. Ahmad, M. Muzzammil and I. Zaheer, 'Numerical Prediction of Wind Loads on Low Buildings', International Journal of Engineering, Science and Technology, Vol. 3(5), pp. 59–72, 2011.
5. Ahuja, A.K. & Roy, Amrit. (2013). Parametric Study of Wind Loads on Canopy Roofs.
6. Albaali, Ghani & Farid, Mohammed. (2006). Fundamentals Of Computational Fluid Dynamics. 10.1007/0-387-31129-7_4.
7. Alrawashdeh, Hatem & Elsharawy, Mohamed & Stathopoulos, Ted. (2014). Wind Pressures on Flat Roof Edges and Corners of Large Low Buildings.
8. Arch, Gie & Hajra, Bodhisatta & Suaris, W. & Zisis, Ioannis & Chowdhury, Arindam & Irwin, Peter & Moravej, Mohammadtaghi. (2016). Assessment of wind loads on gable and hip roof buildings through large scale measurements at the Wall of Wind.
9. Arya, Behnam & Jain, Anurag. (2012). Prefabricated Wood Truss Roof Failure under Wind Induced Loads. 1228-1237. 10.1061/9780784412640.130.
10. B. E. Launder, G. J. Reece and W. Rodi, ‘Progress in the development of a Reynolds-stress turbulence closure’, Journal of fluid mechanics, Vol. 68, pt. 3, pp. 537-566, 1975.
11. Bhattacharyya, Biswarup & Dalui, Sujit & Ahuja, Ashok. (2014). Wind Induced Pressure on 'E' Plan Shaped Tall Buildings. Jordan Journal of Civil Engineering. 8. 120-134.
12. Bre, Facundo & Gimenez, Juan & Fachinotti, Víctor. (2017). Prediction of wind pressure coefficients on building surfaces using Artificial Neural Networks. Energy and Buildings. 158. 10.1016/j.enbuild.2017.11.045.