Al-Azhar University Civil Engineering Research Magazine (CERM)
Vol. (39) No. (1) January, 2017

Comparative study of Wind loads and drag coefficients of single story buildings using CFD and the international and local codes of loads.

G. H. Mahmoud1, Nasr E. Nasr2, Nourhan.S3
1Associate Professor, Structural Engineering Department, Ain-Shams University, Abbasseya, Cairo, Egypt;
2Asistant Professor, Structural Engineering Department, Ain-Shams University, Abbasseya, Cairo, Egypt;
3Demonstrator, Structural Engineering Department, Ain-Shams University, Abbasseya, Cairo, Egypt;

الملخص:

يحدد هذا البحث معاملات الرياح مثل توزيع الضغط، معامل السحب، شكل توزيع السرعة، والبيانات اللازمة لتصميم هيكلي لمنشأ نموذجي متكون من دور واحد اعتمادا على تقنيات CFD.وتشمل الحالات التي تمت دراستها على جمالون ذو منحدرات مزدوجة و بحر طويل و قصير. والمعاملات الأساسية التي يعتمد عليها تحليل القوى المؤثرة على الجمالون هى ميل السقف، واتجاه الرياح. للمباني، وتم تسليط الضوء على تغير معاملات الضغط على حسب المنطقة المدروسة لمقارنة تطبيق تقنية CFD مع المعايير الرياح الدولية و الاكواد العالمية.ان أهم الاساليب الحالية لحساب ضغط الرياح التي يسببها على المباني مستخرجة من اختبارات نفق الرياح. ولكن تجارب نفق الرياح مكلفة وغير متوفرة دائما للمهندسين التصميم. ولذلك فان هذا البحث هو محاولة لتعظيم الاستفادة من استخدام التقنيات الرقمية المتاحة لتوسيع قاعدة البيانات المتوفرة لحساب قوى الرياح على الجمالونات. يقدم هذا البحث دراسة مستفيضة عن أحمال الرياح ومعاملات الضغط الخارجي والسحب على بعض المباني المحددة باستخدام التقنيات الحسابية وديناميكا الموائع (CFD). والهدف من هذا البحث هو توفير قاعدة بيانات واسعة النطاق للمعاملات الضغط والشفط لمنشأ الجمالون مزدوج الميل. و النتائج المستخرجة سوف يتم مقارنتها مع كل من ASCE7، كود الأحمال والنتائج المعملية.

Abstract:

This research determines the wind parameters as pressure distribution, drag coefficient, velocity profile, and necessary data for the structural design of typical single story building adopting the Computational Fluid Dynamics CFD techniques. The studied cases include single span short and long gable structure with double slopes. The basic parameters considered in the analysis of gable building including roof slops, wind direction. For gable buildings, the different roof zones coefficients of pressure has been highlighted to compare the application of CFD technique with the international wind standards and codes of practice. The most current provisions for calculating the wind induced pressure on buildings extracted from boundary layer wind tunnel tests. The wind tunnel experiments is expensive and not always available for design engineers. This research is an attempt to maximize the benefit of using the available numerical techniques to extend the available database for wind codes provisions. This paper presents extensive study of wind loads and coefficients of external pressure and suction of some common single story buildings using the Computational Fluid Dynamics (CFD) techniques. The aim of the paper is to provide wide range database for coefficients of pressure and suctions for double slops gable roof. The obtained results compared with each of the ASCE7, code of loads.

Introduction

The existing wind loads codes are based on measured values of pressure coefficients from boundary layer wind tunnel tests. For structural design engineers, the calculations of the wind loads on buildings is one of the important items of the design process. The determination of the flow field and the wind pressure distribution of the buildings are one of the main objectives of several numerical and experimental researches.

The application of CFD simulation by T. Tamura et al (1) in the pioneer investigation of three-dimensional flow around a rectangular cylinder present the application of finite difference method to study the aero elastic instability phenomena. The adopted numerical techniques succeeded to describe the pressure distribution on the side of forced oscillation rectangular cylinder. J.D Gingera et al (2), present wind tunnel tests to study long low-rise steep roof buildings to obtain the effect of length/span ratio on external wind pressure distribution. The experimental coefficient of pressure and suction compared with the Australian wind load standard (AS 1170.2-1989) (3) and show underestimation of the codes values in some locations of studied buildings. The actual Australian/New Zealand wind load standard, (AS/NZS 1170.2-2002) (4) increase the suction values at the leeward side of the buildings. The investigation showed underestimation of pressure coefficient in other principal standards and cods.

A.Shklyar et al (5) investigate numerically the three-dimensional isothermal flow patterns and mass fluxes in a full-scale greenhouse. The studied building was pitched-roof, single-span and the results compared with similar full-scale test results by Hoxey et al (6). The turbulent flow with higher values of Reynolds-number selected for the study accompanied by free stream conditions. The flow features around the leeward side, roof ridge and windward wall described to determine the ventilation rate and direction of greenhouses. The airflow through complex truss were examined byA.Nakayama et al, (7) using Large Eddy Simulation (LES). From the adopted simulation, it is possible to calculate the mean drag coefficient, which agrees with empirical values. In addition, the dynamic flocculation of both drag and suction forces of either the complete structure or its constituent obtained. The flow feature of the local velocity distribution obtained. Tominaga et al (8) formed a wind tunnel experiments and CFD simulations of three buildings with different roof pitches slopes to examine the pressure variation along each one, comparing the results with wind tunnel test output, and found that the best performance was for RNG k-Ɛ model. Guirguis NM (9) Formed a Study on mono slope model tested in a wind tunnel and evaluated the wind pressure distribution as a function of roof slope, height and wind direction and deduced the recommended zones of pressure coefficient to be adopted by deign codes.The employment of the numerical techniques such as the Computational Fluid Dynamics “CFD” used here to demonstrate the possibility of providing the structural design engineer with the necessary wind pressure data.

The study focus on the Gable roof structures, which is widely, used in industrial buildings considering the effect of roof slope angles on the distribution of wind pressure. The verification of the application of CFD with previously available experimental and numerical results followed with comparison between the solution techniques and the mesh sensitivity effect. The pressure distribution of the Gable roof building studied for different angles of wind attack. Finally, a comparison between the obtained results with ASCE (10) design code presented.

Theoretical Background

Wind loads used for design specified as

Fa = q . Ce . Cpa . Cg . A

where,

Fa is the wind load,

q is the dynamic wind pressure equals ½ . ρ . v2

Ce is the exposure factor,

Cf is the effective pressure coefficient,

and Cg is the gust factor.

CFD technique for the simulation of the wind effect on buildings; applies numerical methods (called discretization) to approximate the governing equations of fluid mechanics in the domain of the air flow, and it is made by converting this domain into a group of cells "grids" and applying equation of motion on each node, then solving it simultaneously to provide solution.

The governing equations of the flow are:

Mass conversation equation $\frac{\partial \mathrm{ui}}{\partial \mathrm{xi}} = 0$

Navier Stokes equation

$$\frac{\partial \mathrm{ui}}{\partial \mathrm{t}} + \mathrm{u_j}\frac{\partial \mathrm{ui}}{\partial \mathrm{xj}} = -\frac{\partial \mathrm{p}}{\rho.\partial \mathrm{xi}} + \frac{\partial^2 \mathrm{ui}}{\mathrm{Re}.(\partial \mathrm{xj})^2} + \mathrm{fi}$$

Where, $\mathrm{u}, \mathrm{p}, \mathrm{t}$ and Re denote velocity, pressure, time and Reynolds number respectively.

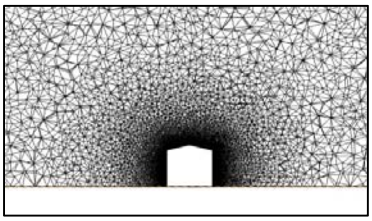
General conservation (transport) equations for mass, momentum, energy, etc., discretized into algebraic equations. The discretized conservation equations solved iteratively. A number of iterations are usually required to reach a converged solution when the Changes in solution variables from one iteration to the next are negligible, Residuals provide a mechanism to help monitor this trend, and Overall property conservation achieved.

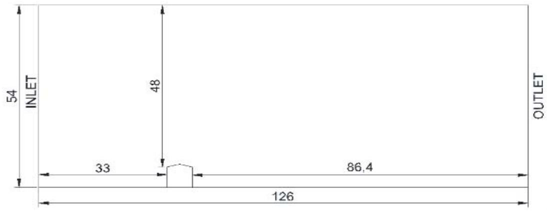
The accuracy of a converged solution is dependent upon appropriateness and accuracy of the physical models, grid resolution and independence, and problem setup.

Numerical simulation of Gable Roof

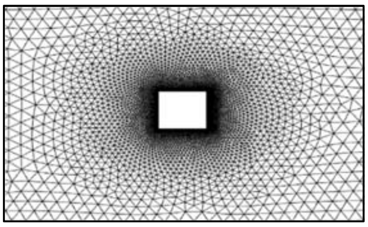
A gable roof building with dimensions 6.6m length X 6.6m width X 6m height "Hg" is subjected to wind flow with velocity 100 MPH. The suggested domain of 21Hg length X 9Hg width X 9Hg height used so that it can enclose the whole flow and represent the full path of air around the building upstream and downstream. The wind loads will be applied in three angles of incidence "0°" perpendicular to ridge, parallel to ridge "90°" and oblique direction "45°".

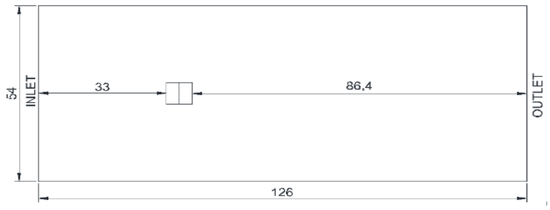
ANSYS FLUENT 15.0 software was used to perform steady flow computations based on a control volume approach for solving the flow equations. And gable roof models are tested "Laminar & LES".





(a) Elevation of Domain Mesh (b) Elevation of Domain Dimensions





(c) Plan of Domain Mesh (d) Plan of Domain Dimensions

Fig. (1) Gable roof domain dimensions and mesh.

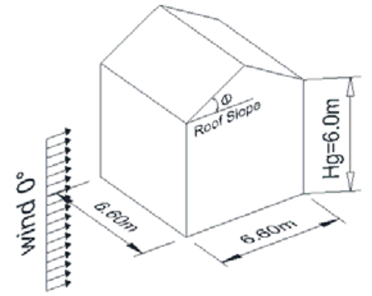


Fig. (2) Schematic view of gable roof model.

The standard case of design codes is adopted here; so the air flow around building is simulated as laminar model, solved using 2nd degree pressure, momentum and the default air density 1.225 kg/m3, Number of iteration is assigned so that the results converge and the output pressure coefficient stabilize, and in this case it is taken 500 iteration.

The dimensions of this model assigned to be same as the previously studied model at wind tunnel of Niigata Institute of Technology by Tominaga et al (8). At first, the performance of this model with roof slope equal to 5:10 (26.6°) investigated. The surrounded domain divided into 2355829 tetrahedral cells instead of hexahedral one to increase the accuracy of results. This model is analyzed with three different types of flow; Laminar, Turbulent RNG k-Ɛ and LES, then a comparison was made to verify the results of the adopted method in this paper, and to study difference between the effect of each type of flow on the structure. The contours lines of the coefficients of pressure demonstrated in Figure (2) while the average pressure coefficient for each model surface calculated by fluent program and summarized in table (1).

Table (1)- comparison between the three flow types for the studied Gable Roof

Model face	Average Pressure Coefficient		
	Laminar flow	RNG k-Ɛ	LES
Front Face	0.814	0.777	0.769
Rear Face	-0.381	-0.409	-0.412
Front Roof	-0.617	-0.578	-0.566
Rear Roof	-0.640	-0.395	-0.483

The results of the three studied flow models showed that there is a difference between the velocity diagrams of each one due to the difference between the particles movement through domain and the formation of vortex at turbulent model. However, there is a slight difference between each flow effect on building surface. The values of pressure distribution around the structure is same with slight change in distribution, also the average pressure coefficients of model surfaces are almost the same for both RNG and LES flow type while Laminar flow model gives more conservative values.

From the results of the studied flow types, the laminar flow model was chosen to use for the analysis of all gable roof models to estimate an accurate distribution of pressure coefficient and compare it with the design standard values

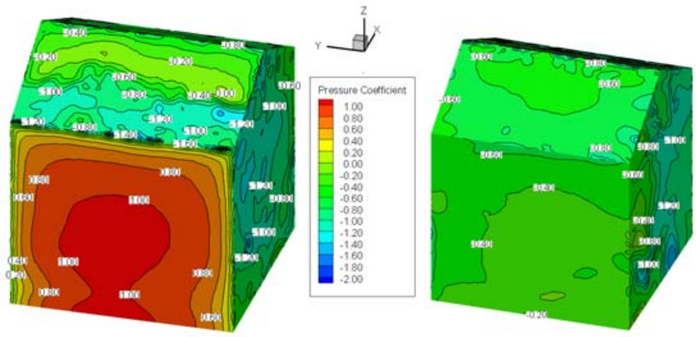


Fig. (3) Laminar flow pressure distribution Frontal & rear faces

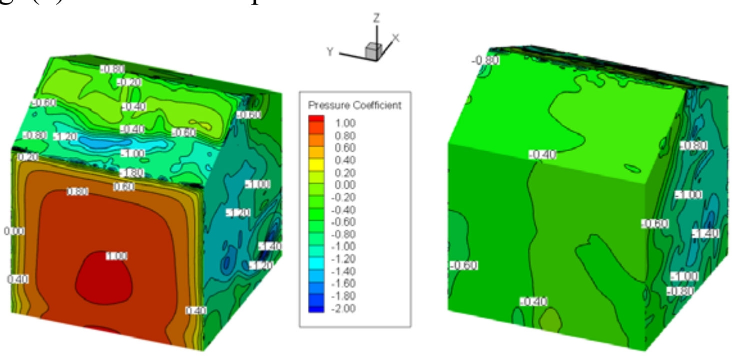


Fig. (4) RNG k-Ɛ flow pressure distribution Frontal & rear faces

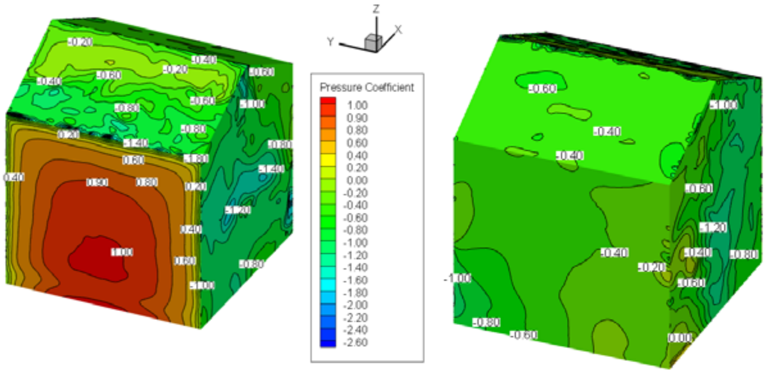
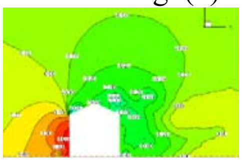
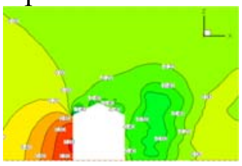


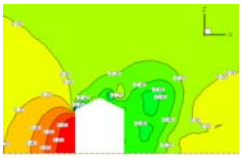
Fig. (5) LES flow pressure distribution Frontal & rear faces



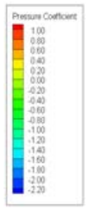
(a) Pressure distribution for roof slope 5:10 Laminar flow model

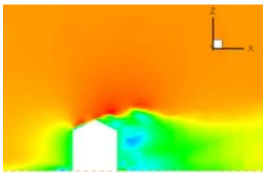


(b) Pressure distribution for roof slope 5:10 RNG model

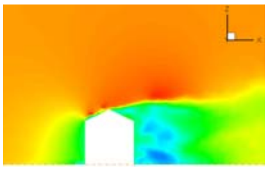


(c) Pressure distribution for roof slope 5:10 LES model

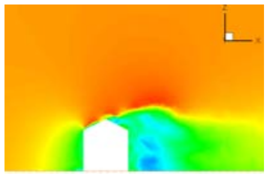




(d) Velocity distribution for roof slope 5:10 Laminar flow model



(e) Velocity distribution for roof slope 5:10 RNG model



(f) Velocity distribution for roof slope 5:10 LES model

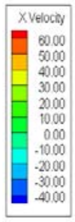
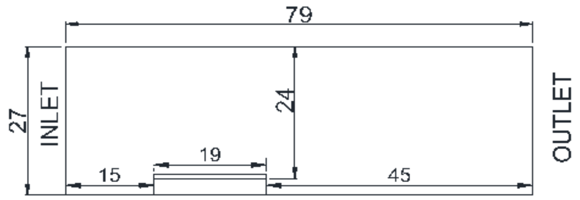


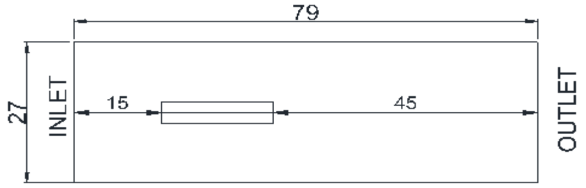
Fig. (6) Pressure coefficient and velocity distribution at domain section around Gable roof building 5:10

Comparative study between wind tunnel tests, ASCE code and CFD results of Gable Roof

Another gable roof building previously experimentally studied at both of the CSU wind tunnel and the INEEL (11) wind tunnel studded applying the CFD technique. The objective of the study is to compare the experimental results with each other and with each of ASCE code of practice and CFD technique. The full-scale dimensions of the model will be simulated using the fluent program and the results will be compared with the output data of the wind tunnel tests. CSU Wind tunnel test was performed on a scaled model 1:50, while INEEL wind tunnel scaled model is 1:25, the real house dimensions is 14ft* 10ft * 62ft long (4.3m * 3m * 19m) and with roof slope equal to 18°. The domain dimensions is chosen so that the clearance around building will be less than 5 H; dimension (27m * 27m * 79m).



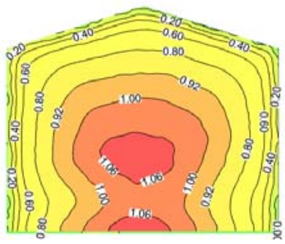
Elevation of Domain Dimensions



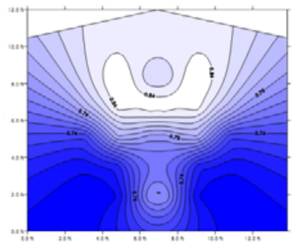
Plan of Domain Dimensions

Figure (7) Long Gable roof domain dimensions.

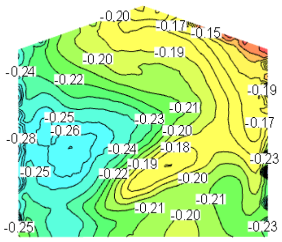
The surrounded domain is divided into 2987340 tetrahedral cells. The airflow through it is assigned as laminar flow with velocity 80mph (36m/sec) with 500 iterations. To investigate the effect of the length of the building on the pressure distribution, another model was built with the dimensions of building 14 ft* 10 ft * 14 ft long (4.3m * 3m * 4.3m) and with roof slope equal to 18° with the same domain of dimensions (27m * 27m * 79m). The comparison between the pressure distributions of experimentally obtained in the CSU wind tunnel test and the numerically obtained values using CFD technique demonstrated in Fig. (8). the comparison between the CFD simulation results, the experimental results for both CSU and INEEL and ASCE Code calculated values are presented in Fig. (9).



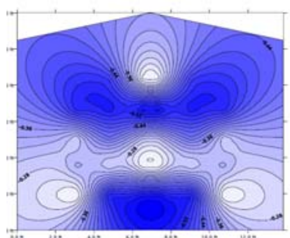
(a)Front wall pressure distribution of CFD model



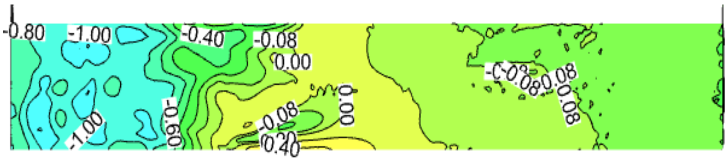
(b)Front wall pressure distribution of CSU model



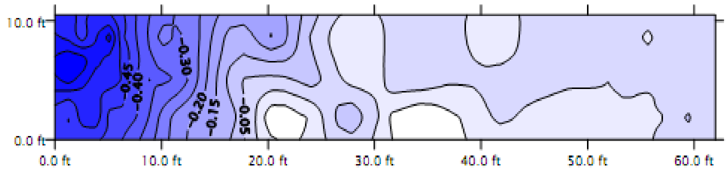
(c) Rear wall pressure distribution of CFD model



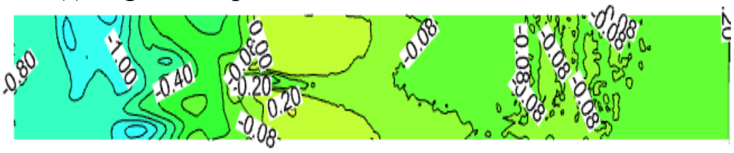
(d) Rear wall pressure distribution of CSU model



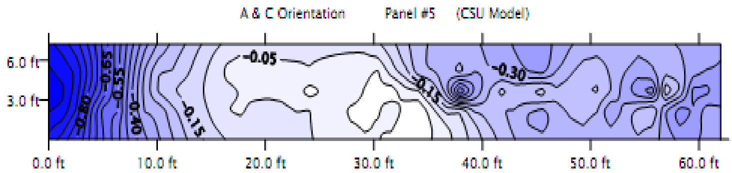
(e) Right wall pressure distribution of CFD model



(f) Right wall pressure distribution of CSU model

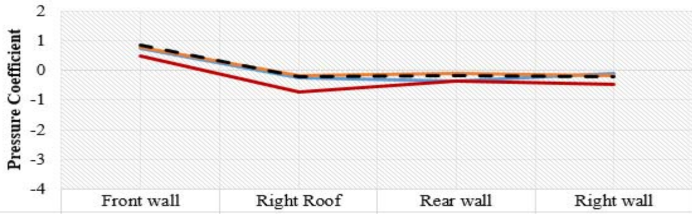


(g) Right Roof pressure distribution of CFD model



(h) Right Roof pressure distribution of CSU model

Fig. (8) Pressure coefficient comparison for Gable roof pitch angle 18°

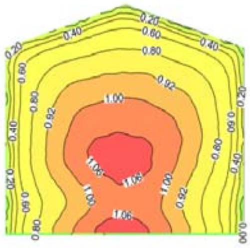


	Front wall	Right Roof	Rear wall	Right wall
CSU	0.76	-0.263	-0.376	-0.107
INEEL	0.773	-0.174	-0.092	-0.189
ASCE	0.49	-0.727	-0.35	-0.453
CFD 18	0.842	-0.220	-0.171	-0.230

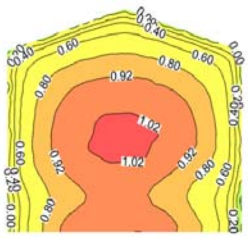
Fig. (9) Comparison between code, wind Tunnel and CFD Results

Table (2). Comparison between coefficients of pressure and suction of short and long gable roof house for different roof pitches slopes

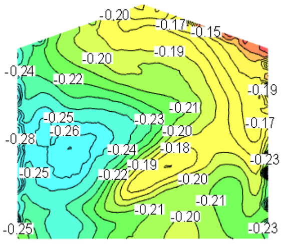
	Front Face		Right Roof		Back Face		Right Face	
Roof Angle	Long Model	Short Model	Long Model	Short Model	Long Model	Short Model	Long Model	Short Model
18°	0.842	0.813	-0.220	-0.967	-0.171	-0.228	-0.230	-0.813
30°	0.831	0.796	-0.209	-0.980	-0.190	-0.207	-0.260	-0.938
45°	0.826	0.820	-0.265	-0.987	-0.194	-0.337	-0.275	-1.095
60°	0.829	0.844	-0.294	-1.029	-0.175	-0.596	-0.329	-0.858



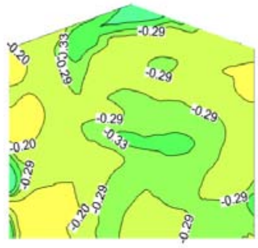
(a) Front wall pressure distribution of long model



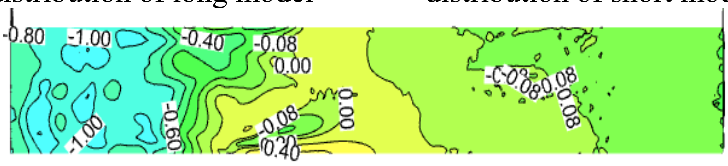
(b) Front wall pressure distribution of short model



(c) Rear wall pressure distribution of long model

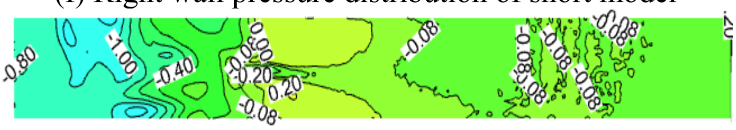


(d) Rear wall pressure distribution of short model

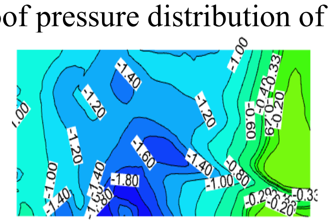


(e) Right wall pressure distribution of long model

(f) Right wall pressure distribution of short model



(g) Right Roof pressure distribution of long. Model



(h) Right Roof pressure distribution of short model

Figure (10) Contour Diagrams of Pressure coefficient on Long and Short pitched roof buildings.

Conclusions

This paper address the pressure flow field obtained from wind tunnel tests of double slopes gable building and examine the application of CFD numerical technique to predict this pressure field. The study results show good description of this flow field using the CFD technique. The values of the pressure coefficients predicted from the numerical technique agrees very well with the experimental corresponding values. The comparison between the CFD values of coefficients of pressure supported by the experimental values shows the ASCE code values over estimates the design values. The numerical application of CFD to compare the flow fields between long and short Gable Building show that there is significant discrepancies in the coefficients of pressure. The CFD technique capable to provide the structural engineer with useful data for wind pressure coefficient more accurate than that for wind loads codes as ASCE code.

References

Ginger, J. D. and J. D. Holmes (2003). "Effect of building length on wind loads on low-rise buildings with a steep roof pitch." Journal of Wind Engineering and Industrial Aerodynamics 91(11): 1377-1400.

Tamura T, Itoh Y. 1997. Three-dimensional vertical flows around a bluff cylinder in unstable oscillations. Journal of Wind Engineering and Industrial Aerodynamics 67–68:141-154.

Standards Association of Australia (1989) Minimum design load on structures wind loads, AS 1170.2-1989, North Sydney, Australia, 96 pp.

AS/NZS 1170.2:2002. Structural design actions Part 2: Wind actions.

Shklyar A, Arbel A. 2004. Numerical model of the three-dimensional isothermal flow patterns and mass fluxes in a pitched-roof greenhouse. Journal of Wind Engineering and Industrial Aerodynamics 92:1039-1059.

R.H. Hoxey , P. Moran, Full scale wind pressure and load experiments—single-span 7.0 22.6 m glasshouse, Division Note Dn. 1605, AFRC Engineering Research, Silsoe, February 1991.

Nakayama A, Okamoto D, Takeda H. 2010. Large-eddy simulation of flows past complex truss structures. Journal of Wind Engineering and Industrial Aerodynamics 98:133-144.

Tominaga Y. 2015. Flow around a high-rise building using steady and unsteady RANS CFD: Effect of large-scale fluctuations on the velocity statistics. Journal of Wind Engineering and Industrial Aerodynamics 142:93-103.

Guirguisa NM, Abd El-Aziz AA, Nassief MM. 2007. Study of wind effects on different buildings of pitched roofs. Desalination 209:190-198.

ASCE 7, “Minimum Design Loads for Building and Other Structures,” 1998.

Beaudoin DK. 2002. Modeling and Analysis of Wind Pressure Distribution on a Manufactured Home. In: Department of Civil and Architectural Engineering: The University of Wyoming. p 114.