e-ISSN: 2455-5703

CFD Simulation on a Group of Tall Buildings

[1]Yadav Varsha [2]Jignesh Arvindbhai Amin
[1]P.G Student [2]Professor
[1,2]Department of Civil Engineering
[1,2]SVIT, Vasad 388306, Gujarat, India

Abstract

Calculation of wind induced loads is essential for design of tall structures. With the advancements in computer technologies, CFD simulations are now a good alternative for wind tunnel tests for knowing the wind induced responses of a structure. In current scenario, presence of nearby structures exhibit Interference effects. In this paper, CFD simulation is carried out for a particular arrangement of group of tall buildings. Pressure coefficients on the same have been studied using Computational Fluid Dynamics (CFD) technique. Buildings of different heights are taken into consideration. The effect of different wind incidence angles on the pressure coefficients on all the faces of Principal Building has been investigated. Also, the pressure coefficient contours on all the faces of all buildings for wind incidence angles of 0°, 30°, 60° and 90° are plotted. ANSYS CFX is used for carrying out the CFD simulation.
Keyword- CFD, Interfering Building, Pressure Coefficients, Principal Building, Wind Incidence Angle

I. INTRODUCTION

With the present advanced materials and construction techniques, it is possible to construct tall structures with irregular geometries. These modern tall buildings are susceptible to wind excitations. Now, the existence of various tall structures in the proximity creates significant interference. Evaluation of wind loads in such cases considering the effects of nearby structures with greater confidence is of utmost importance. Wind induced loads are usually calculated by existing codes and standards. The codes are based on wind tunnel experiments conducted on isolated buildings. The presence of up-stream buildings in the neighbouring environment creates some effects on the flow of wind which depends upon building geometry, arrangement and the wind incidence angle. Pressure coefficients on a building are calculated by three methods. The first method is using full scale measurements but it is highly costly and the amount of control on the experiment is limited. Second way is the wind tunnel tests. Wind tunnel test have been extensively validated and show good controllability but it suffers from limitations such as model dimensions and scaling. Third way is use of CFD technique which is cheaper than the former two methods and does not have their limitations. CFD is used to determine wind loads on building. It is an acronym for Computational Fluid Dynamics, a method to describe and analyze the fluid flow. Simulations in this method are easy to repeat and control. In all the methods, efforts are made to make the models as realistic as possible but they can never be exactly same as every method has its uses and flaws. This fact causes discrepancies in results of tests, calculations and simulations. Amin et. al (2011) investigated the mean interference effects between two rectangular buildings located in close proximity in a geometrical configuration of 'L' and 'T' plan shape using wind tunnel tests. Chakraborty et. al(2014) studied the results of wind tunnel studies and numerical studies using CFD technique on a '+' plan shaped tall building and found that although there are some differences on certain wall faces, the numerical results are having a good agreement with the experimental results. The objectives of the present study is to investigate the pressure distribution on various faces on the group of tall buildings for different wind angles i.e. 0°, 30°, 60° and 90°. Buildings of different heights are located in a row at a spacing of 50m. Contours of pressure coefficients on all the faces for each case are then plotted.

II. NUMERICAL STUDY

A. Details of Model

Numerical simulations have been carried out using Computational Fluid Dynamics package of ANSYS, namely ANSYS CFX. To compute the pressure coefficients, three high rise buildings with a scale of 1:100 were modelled in ANSYS CFX. All the buildings are square in plan having width of 35m. Buildings are placed in a row at a spacing of 50m and the heights of the buildings are in an ascending pattern i.e. 70m, 90m and 110m. The arrangement of buildings is shown in Fig. 1 and 2. Buildings of 70m and 90m height are present on the upstream side of 110m tall building. Building of height 70m is termed as Interfering Building 1 and of 90 m height is termed as Interfering Building 2. 110m high building is known as Principal building.

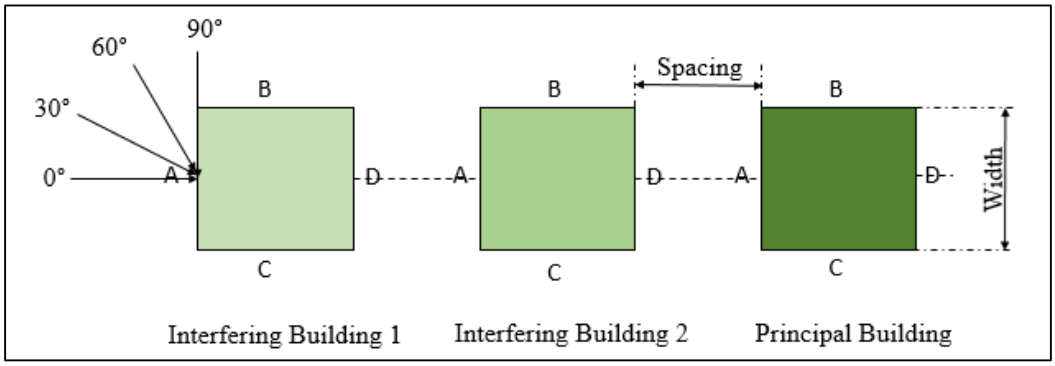


Fig. 1: Top view of the modelling arrangement

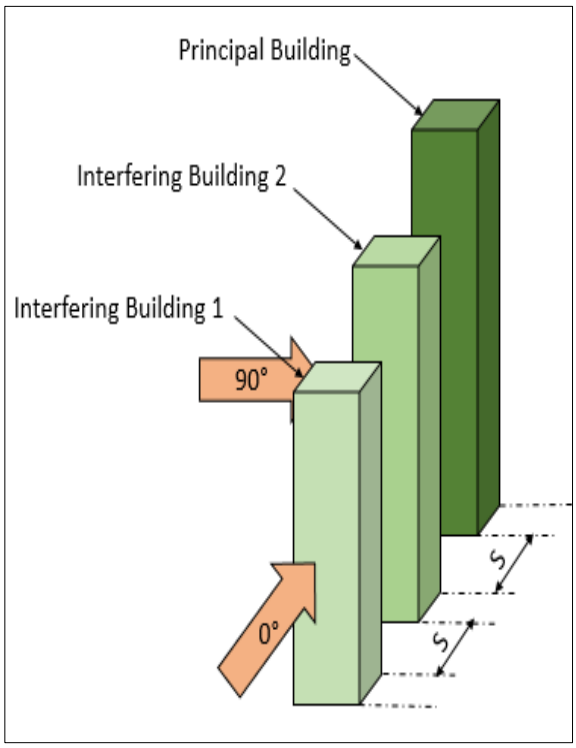


Fig. 2: Isometric view of the modelling arrangement

B. Domain Size and Meshing

The domain used in this study is as recommended by Franke et. al(2004). Inlet, side walls of domain and top of the domain will be at 5H distance whereas the outflow boundary is placed at a distance of 15H from the last building in the arrangement. H stands for the maximum height of the building. Triangle surface meshing is used to mesh the domain and mapped face meshing is used for meshing the buildings. Finer mesh is adopted near the buildings for better accuracy of results. k-ε turbulence model has been used to simulate the wind flow. The k-ε model solves for two variables: k, the turbulent kinetic energy and ε, the rate of dissipation of kinetic energy. The governing equations for k-ε model are given by Eqns. 1 and 2:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - Y_M + S_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon \quad (2)$$

$$\text{Where, } C_1 = \max\left[0.43, \frac{n}{n+5}\right], = S\frac{k}{\varepsilon}, S = \sqrt{2S_{ij}S_{ij}} \quad (3)$$

G_k is the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy and Y_m represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_1$and C are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k (turbulence kinetic energy) and ε (dissipation rate). The other notations are having their usual meaning. The values considered for $C_{2\varepsilon}$, σ_k and σ_ε are 1.9, 1 and 1.2 respectively. ρ is the density of air in ANSYS CFX taken as 1.224 kg/m^3.Turbulence intensity was taken as 10%.

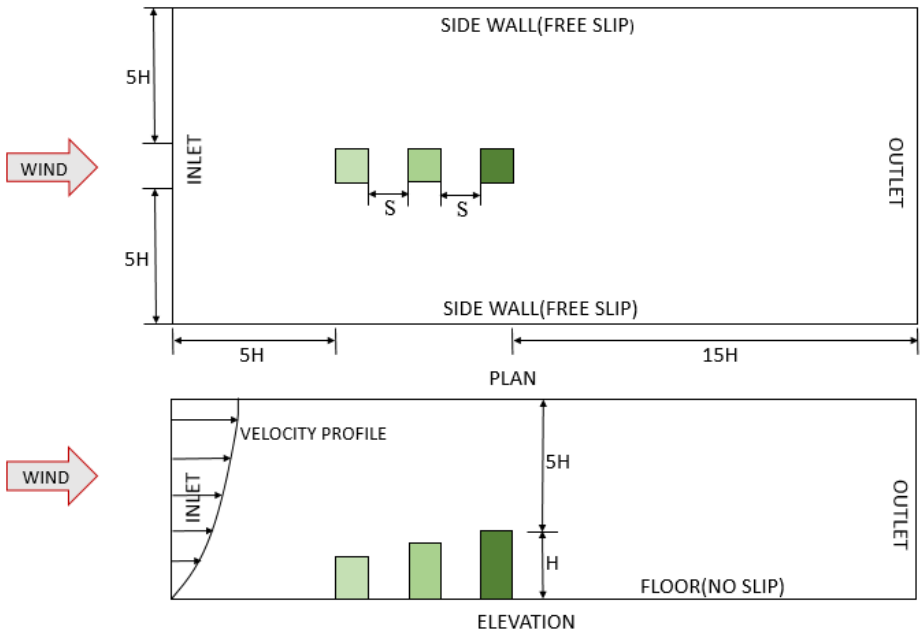


Fig. 3: Arrangement of building in computational domain

C. Velocity Profile

Boundary layer wind flow near the inlet boundary was generated using power law exponent formula by Eqn. 4.

$$\frac{U}{U0} = \left(\frac{z}{z_0}\right)\alpha \qquad (4)$$

Where U_0 is the basic wind speed which is taken as 12m/s and U is the velocity at some particular height z. α is the power law exponent which is taken as 0.14.

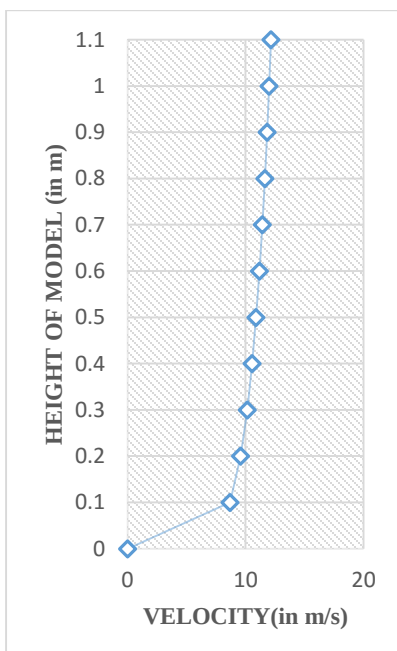
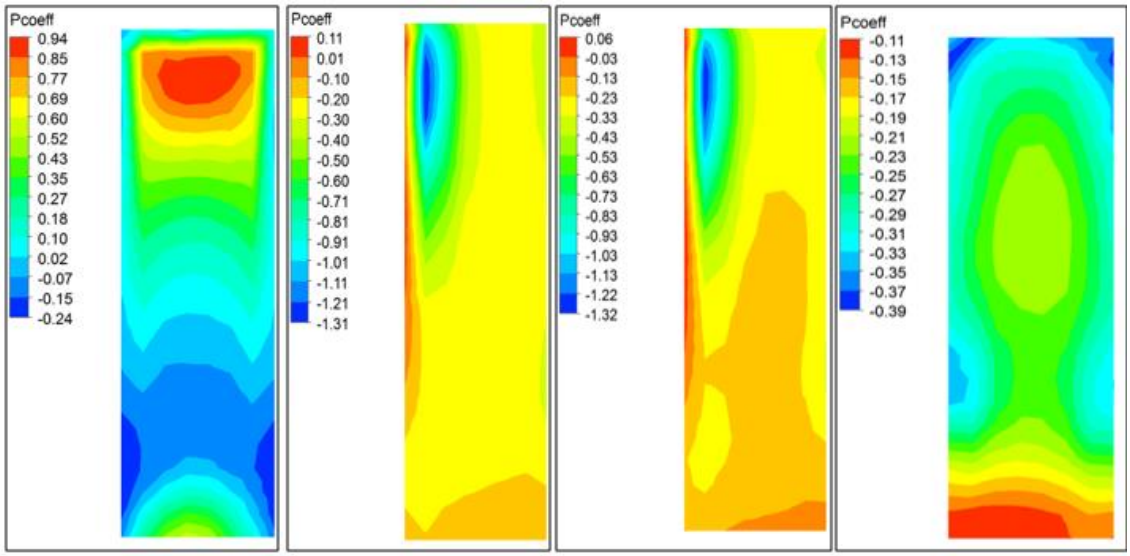


Fig. 4: Velocity Profile at inlet

III. RESULTS

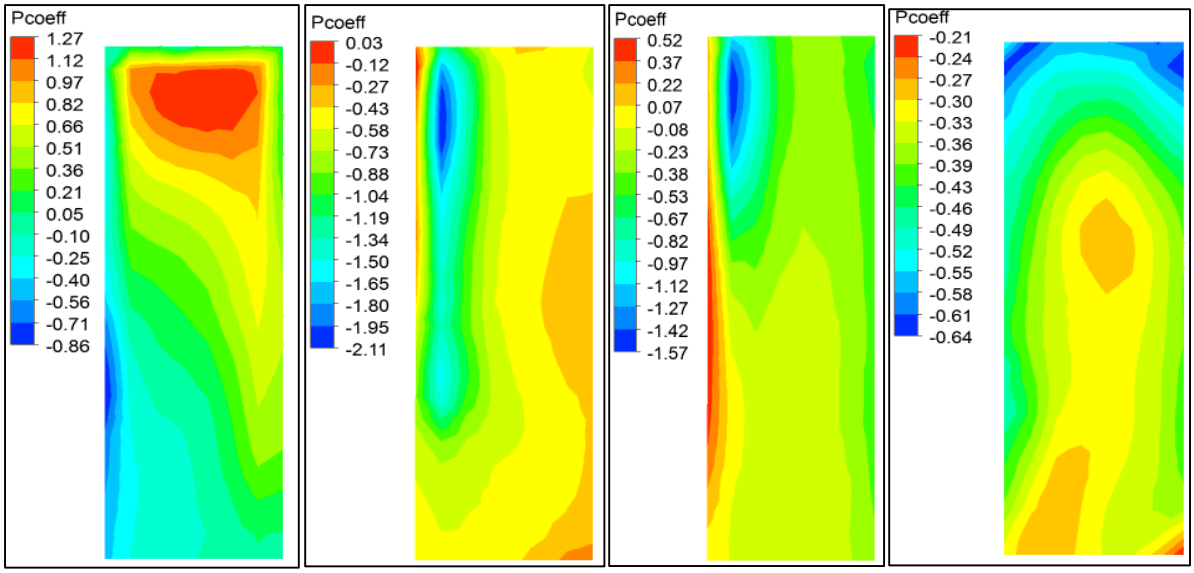
A. Pressure Coefficient Contours

Contours of pressure coefficients for all the four faces i.e. Face A, B, C and D of Principal building are plotted for the wind angles of 0°, 30°, 60° and 90° as shown in Fig. 5, 6, 7 and 8.



Face A Face B Face C Face D

Fig. 5: Pressure Coefficient Contours for different faces of Principal Building for 0° wind angle



Face A Face B Face C Face D

Fig. 6: Pressure Coefficient Contours for different faces of Principal Building for 30° wind angle

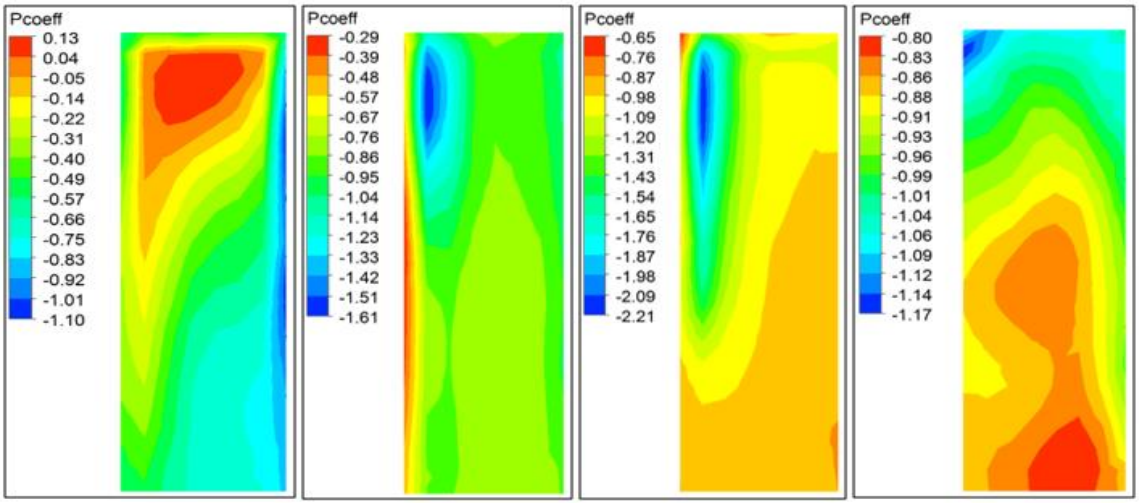
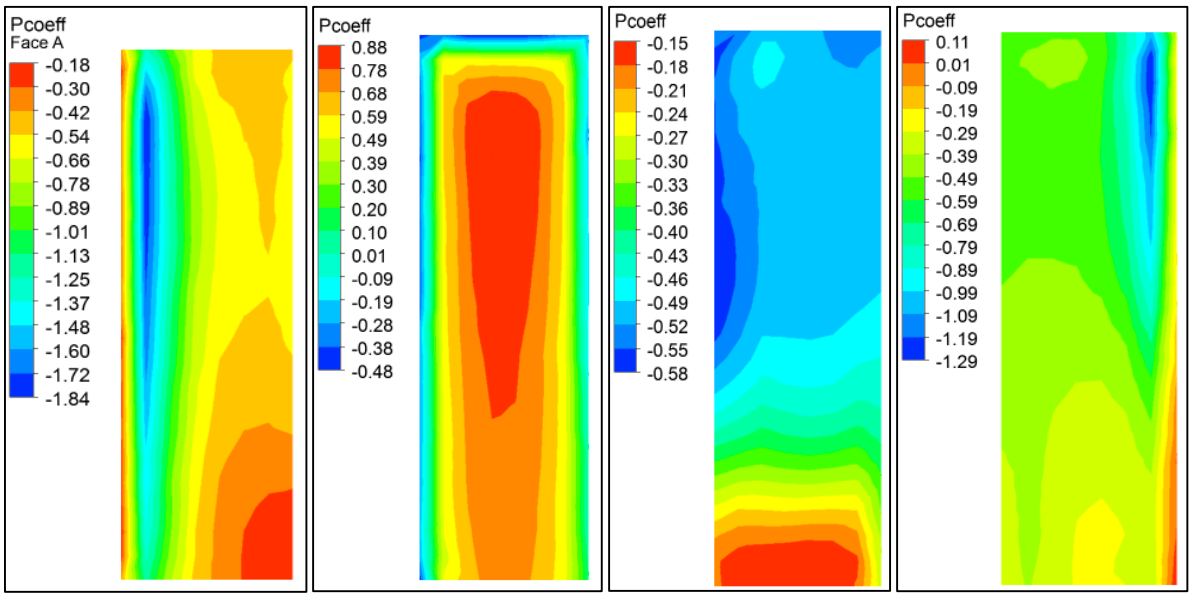


Fig. 7: Pressure Coefficient Contours for different faces of Principal Building for 60° wind angle

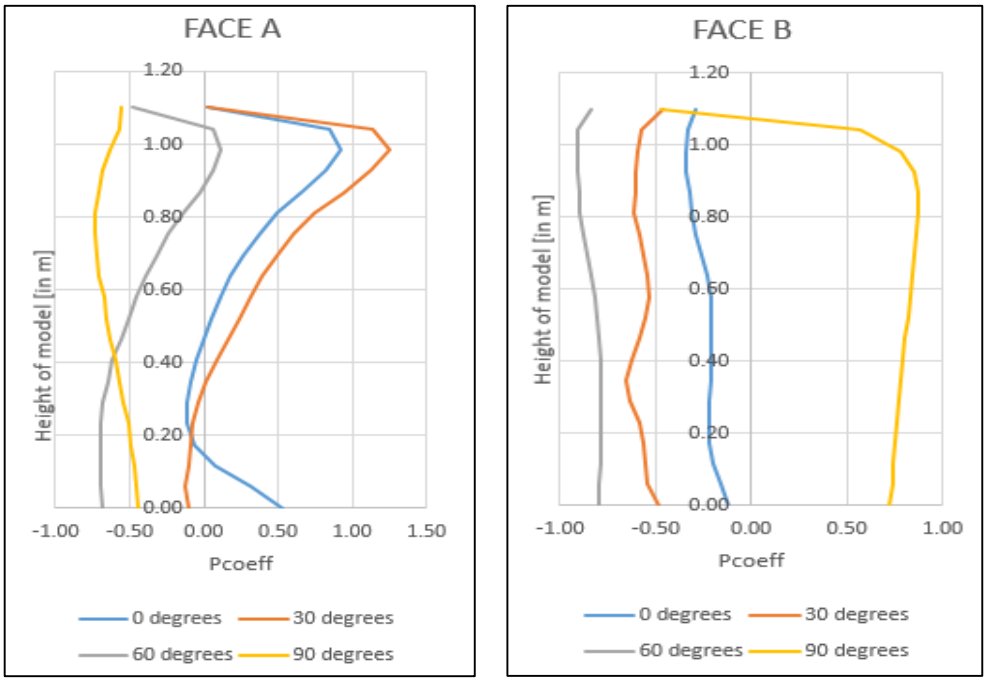


Face A Face B Face C Face D

Fig. 8: Pressure Coefficient Contours for different faces of Principal Building for 90° wind angle

B. Effect of Wind Incidence Angles

The effect of change of wind incidence angles on the faces of Principal building is represented by the graphical plots as shown in Fig 9.



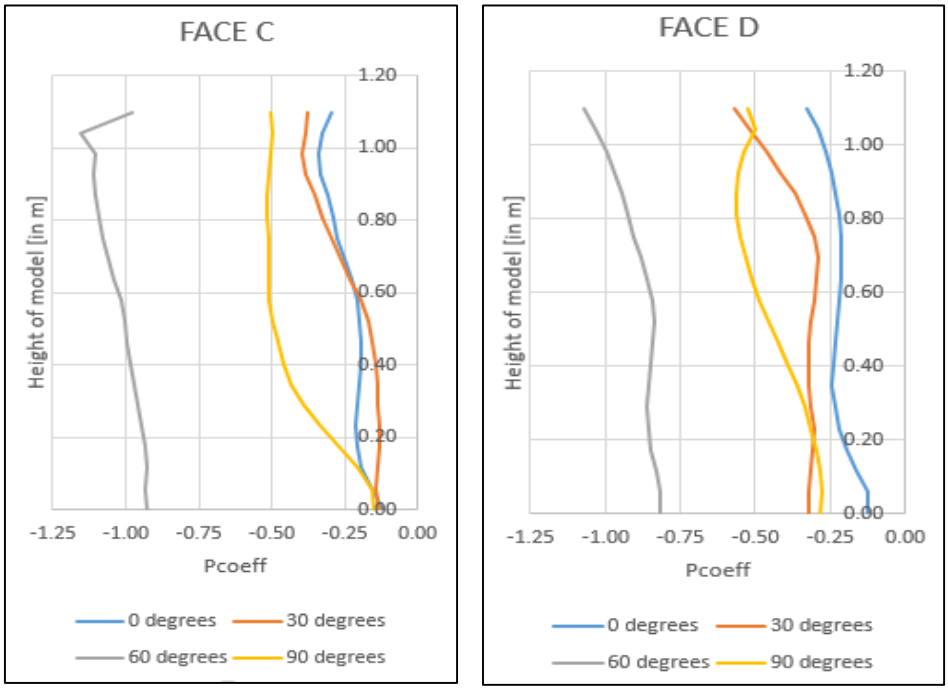


Fig. 9: Comparison of pressure coefficient along the vertical centerline on different faces of the Principal Building

IV. CONCLUSIONS

For face A, maximum pressures are observed for 0° and 30° wind angles whereas for wind angles of 60° and 90°, face A experiences negative pressures. Faces C and D experience negative pressures for all wind incidence angles. These faces experience maximum suction for 60° wind angle and minimum for 0° and 30° wind incidence angles. Face B is seen to observe positive pressures in case of 90° wind angle and for the angles of 0°, 30° and 60° negative pressures develop on face B.

Wind pressures are almost same for Face B and Face C in case of 0° wind angle due to symmetry of flow. No such symmetry is seen in case of 30° and 60° wind angles. Most of the parts of Face B and C are subjected to negative pressures. Leeward Face D experiences higher suction as compared to Face B and Face C. Because of presence of two interfering buildings, maximum pressure coefficient (0.86) is seen on top one third height on Face A for 0° wind angle, whereas the lower portion observes negative pressures. Magnitude of positive pressure coefficient on Face an increases significantly when the wind incidence angle is 30° as compared to 0° wind angle. As the wind angle changes from 30° to 60°, the values of negative pressure coefficients increases drastically on almost all the faces of the Principal building. For 90° wind angle, higher suction is seen on upper half portion of the Face C as compared to the lower portion and Face D is subjected to almost same pressure with least variation.

REFERENCES

[1] J. A. Amin and A. K. Ahuja, "Wind-induced mean interference effects between two closed spaced buildings," KSCE journal of civil engineering, India: Springer, 2012, Vol. 16(1), p. 119-131.
[2] K. Bairagi and S .K. Dalui, "Optimization of interference effects on high rise buildings for different wind angles using CFD simulation" EJSE, India, 2014.
[3] S. Mukherjee, S. Chokraborty, S .K. Dalui and A. K. Ahuja, "Wind induced pressure on Y plan shape tall building" Journal of wind and structures, 2014, Vol. 19, P. 523.
[4] S. Chokraborty, S .K. Dalui and A. K. Ahuja, "Wind load on irregular plan shaped tall building – a case study" Journal of wind and structures, 2014, Vol. 19, P. 59IS.
[5] IS: 875 (1987), "Indian standard code of practice for design loads for buildings and structures, part 3 (Wind loads)" Bureau of Indian standards, New Delhi.