COMPUTATIONAL FLUID DYNAMICS SIMULATION WITH AERODYNAMIC TREATMENTS ON HIGH RISE BUILDINGS

Mrudul P. Patel[1], Bansal R. Patel [2]

[1]P. G. Student, Department of Applied Mechanics, L. D. College of Engineering, Ahmedabad

[2] Assistant Professor, Department of Applied Mechanics, L. D. College of Engineering, Ahmedabad

---***---

Abstract - *Due to development of new techniques and advanced computational tools in construction industry now a days tall structures created are very flexible & slender. Such types of slender and flexible structures are very sensitive to dynamic wind loads and it can adversely affect the serviceability of building and comfort of occupants. To ensure the functional performance of such type of structures and to control the wind induced motion on the tall buildings various types of aerodynamic modifications to the shape/geometry of the buildings are possible such as horizontal treatments like corner modification and vertical treatments like height and shape modification along height of building. The purpose of this study is to investigate the effect of corner modification on high rise building by average wind pressure coefficient distribution on external surface of building models. Four shapes with horizontal treatments such as rounded, chamfered, recessed, double recessed compared with simple square model with 20 % corner modification at angle of incidence of 0 to 90 degree at 15 degree interval are taken to examine wind pressure coefficient variation along periphery of model. Flow patterns around the building are examined to understand the phenomena occurring around the tall building. ANSYS CFX is used to compute flow pattern, wind pressure and forces on external surface of the building. Furthermore, the polynomial equations for calculating mean pressure coefficients on each face for different angles of attack is calculated using MATLAB.*

Key Words: **Corner modification, Computational Fluid Dynamics, k-ε Method, Average pressure co-efficient, ANSYS CFX, MATLAB.**

1. INTRODUCTION

When fluid flows past the bodies it creates certain effects on surface of buildings. This effect includes large amount of forces e.g. pressure forces, suction forces, drag forces and lift forces or vertex shedding and flow separation around any structure e.g. Buildings, space crafts, submarines, cars etc.

In order to make these structures safe against such type of forces it is necessary to study phenomenon of flow of wind in order to make structure safe against wind.

Generally speaking in field of structural engineering there are lateral forces such as earthquake and wind which governs the design of structure for design combinations. When low rise buildings are considered earthquake forces governs design, however as the height increases building behaves as cantilever and wind force comes into picture. Since after certain height fluctuations of wind force increases. It is often not possible for designer to calculate this behavior of wind under static methods. For this type of tall buildings gust factor approach or wind tunnel procedure is adopted to study phenomenon of aerodynamics means fluctuations of wind around building in order to design safe building that gives human comfort as well as safety of users.

As per current scenarios construction of tall buildings are increased due to less space and more users, so it is necessary for engineer to predict nearly accurate behavior of wind around buildings. Therefore aerodynamics plays important role in design of tall structures.

Generally, the way of mitigation of this wind induced motion can be divided into two ways; 1) structural modification 2) aerodynamic modification. Over two prerequisites can be satisfied with the utilization of mass damper like Taipei 101 or with an efficiently adjusted structure like Burj Dubai. As the improvement, building shape has been made over streamlined shape advancement which can diminish the arrangement of swirls and lessen the impact of vortex destroying.

Further, aerodynamic modifications can be classified into two categories called horizontal modification and vertical modification. Horizontal modification intended to provide corner modification of building plane such as chamfer, recession, roundness, and so on, While Vertical modification refers to changing shape along height of building. It includes providing taper along height, twisting, opening and setback. Albeit Vertical aerodynamic alterations have seen as viable in lessening across-wind loads and reaction of buildings, minor changes on flat state of tall structures are some of the time more helpful or attainable than vertical aerodynamic treatments for designer and extraordinarily more effortlessly acknowledged by proprietors of the structures.

The various aerodynamic adjustments are applied to the high rise buildings to forestall or lessen the breeze excitations that can be partitioned into major and minor changes. Major modifications are that which creates considerable effect on the architectural as well as structural ideas, for example setbacks along the height, tapering, openings at top, shifting the state of structures, setbacks, twisting of building. Minor modifications that have limited effects on the architectural and engineering ideas, for examples corner alterations like slotted corners, chamfered corners, corner recession, roundness of corners and orientation of building in relation to the most frequent strong wind direction. The latter is the focal point of the present numerical investigation

1.1 Aim & Objective of study

The work is mainly based on comparative study on various horizontal treatment on building using computational fluid dynamics tool in ANSYS 2019 R2.

The aim of this study is to evaluate the flow pattern of wind under aerodynamic effect around the building and to evaluate pressure and forces at various faces and angle of incidence.

1.2 Literature Review

Literature was collected, studied, analysed and sorted So many publications. After sorting the papers and publication which were relevant to the topic where zeroed down. Furthermore, these papers & publications which were sorted to the relevancy of the current topic.

Many analytical research is going on in this field by some researchers using various finite element softwares which also have tools to simulate the fluid behaviour around any object or surfaces. Such type of software currently available are STAR CCM+, ANSYS, ABACUS, MIDAS etc. However, ANSYS is giving facility of advanced fluid dynamics simulation under various categories as well as conditions. ANSYS provides package called ANSYS Fluent and ANSYS CFX to determine behaviour of fluid flow. Out of these two ANSYS Fluent is used to solve wide variety of problems with forming equations, While ANSYS CFX can be used particularly for aerodynamic applications and turbo machinery problems.

Many researches were conducted in wind tunnel as well as computational studies. Some of these are described here. J. A. Amin and A. K. Ahuja (2008) have conducted experimental study on irregular plan shape buildings such as L and T shapes. Kim and Kanda (2010) studied behaviour of aerodynamics on square plan building with height variation using experimental study. Eswara kumar bandi, Yuiko Tamura, and others (2013) have carried out experimental investigation to study behaviour of wind on triangular shape buildings using wind tunnel studies. Norbert Jendzelovsky, Roland Antal and Lenka Konecna (2017) have studied behaviour of flow on external surface of triangular high rise building façade and they compared results of experimental study with CFD tool. They found that overall difference between results were around 6 %. Yi Li, Xiang Tian and co-authors (2018) have studied behaviour of high rise building in wind tunnel by 10 % corner modifications. Prasenjit Sayal and Sujit Kumar Dalnui (2018) have conducted study of effect of courtyard and opening on rectangular plan shaped tall building under wind load. K. Shruti, P. N. Rao and G.R. Sabareesh (2019) have studied across wind flow using CFD tool ANSYS FLUENT. Canter of attraction of their research was to study flow conditions of square and cooling tower in standalone and two structure cases.

Main focus of literature review was to search the work done to determine the behaviour of high rise building under dynamic wind loads. Both experimental as well as analytical study are done in this area. However main focus of researchers was experimental study using wind tunnels, and determination of response of building and surroundings due to wind load. Based on one literature it can be seen that computational tools also provides nearly same results to experimental work.

2. METHODOLOGY

2.1 Numerical analysis

Five models were selected for purpose of study the behaviour of corner modification of building. These models are square, chamfered, rounded, recessed and double recessed having 20 % of corner modification.

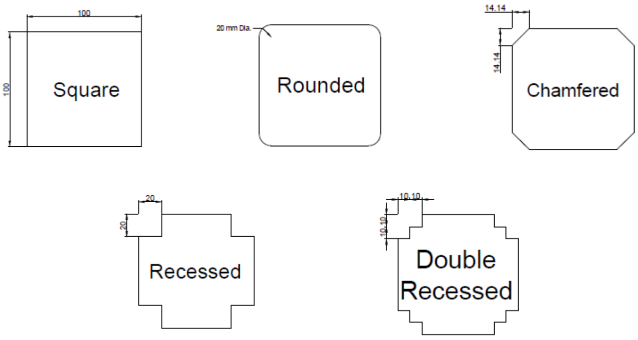


Fig - 1: Shapes considered for purpose of study

These shapes having plan dimensions of 100 mm x 100 mm and having height of 800 mm. Geometric scale of model is taken 1:300. The scaled down velocity of Bombay zone is taken as 11 m/s. Rotation of building is applied from 0 to 90 degree at 15 degree interval.

A domain to apply flow of wind around building is selected as per recommendation of Franke et al (2004). Having spacing as shown in figure 1.

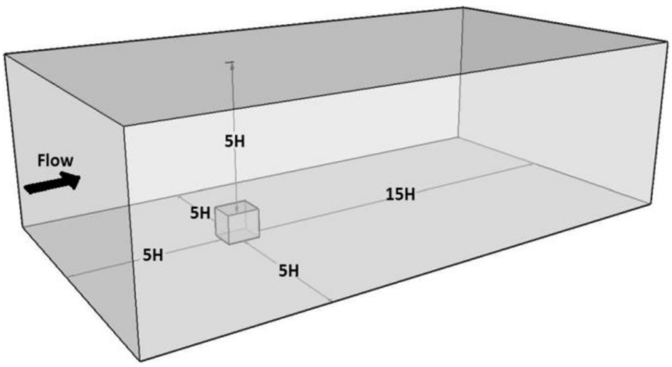


Fig - 2: Domain of CFD simulation

Such large domain is good enough for vertex generation on leeward side of building to avoid backflow of wind.

ANSYS CFX is used for purpose of analysis since it mainly focuses to study aerodynamic applications.

Meshing of the domain is done using tetrahedral elements. Meshing is done finer near surfaces of building in order to get good results on surface of building.

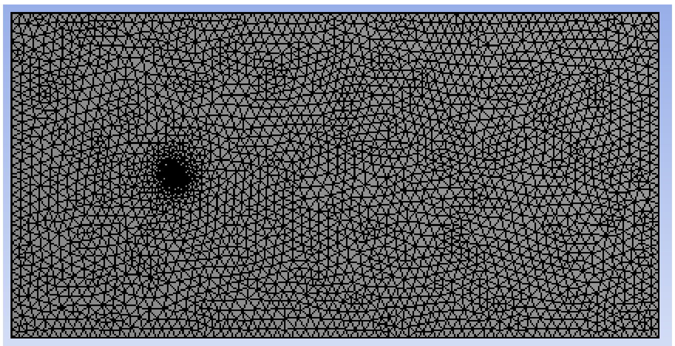


Fig - 3. Meshing of Domain

Velocity at inlet is taken 11 m/s as mentioned before, and outlet relative pressure is taken 0 Pa. The atmospheric pressure is taken as operative pressure.

2.2 k – ε Method

Continuity and momentum in ANSYS CFX is calculated using various equations of fluid dynamics. Various models such as standard k-epsilon model, RNG k-epsilon model, Resizable k-epsilon model, SST model & k-Omega model is used. However for current thesis based on literature review done standard k-epsilon model is used for all analytical models.

The k- ε models utilize the angle dispersion theory to relate Reynolds stresses to mean velocity gradients and turbulent viscosity. Turbulent viscosity is demonstrated as the result of turbulent length scale and turbulent velocity. k is the turbulent kinetic energy and is characterized as the variance of fluctuations in velocity. It has dimensions of L^2T^{-2}. ε is the turbulence eddy dissipation which is actually the rate at which the velocity fluctuation dissipates and has measurements of per unit time.

Following equation of continuity and momentum in standard k- epsilon model is used to compute velocity of flux in solver by iteration.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U_j)}{\partial x_j} = 0$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{\partial P'}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{\text{eff}}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_M$$

Where S_M is the addition of body forces, μ_{eff} is the effective viscosity accounting for turbulence and P' is the modified pressure.

Density and velocity are denoted by and U. i and j are two mutually perpendicular directions. The k- ε model is based on the concept of eddy viscosity, so that,

$$\mu_{\text{eff}} = \mu + \mu_t,$$

Where μ_t is turbulent viscosity.

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon}$$

The values of k and epsilon come from the differential transport equations of turbulence kinetic energy and turbulence dissipation rate.

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k U_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

$$\frac{\partial (\rho \epsilon)}{\partial t} + \frac{\partial (\rho \epsilon U_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu\epsilon}} + C_{1\epsilon}\frac{\epsilon}{k} C_{3\epsilon} P_b + S_\epsilon,$$

Where P_k represents the generation of turbulence kinetic energy because of the mean velocity gradients, P_b represents the generation because of buoyancy, Y_m represents the contribution of fluctuating dilatation in compressible turbulence to in general dissipation rate and C1 and C2 are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k (turbulence kinetic energy) and ε (dissipation rate). The values considered for C1, σ_k and σ_ε are taken as 1.44, 1 and 1.3, respectively, as per the recommendation of Jones and Launder (1972).

2.3 External Pressure Co-efficient

External pressure coefficient in this thesis is found out by general equation given below,

$$Cp = \frac{P}{0.5\ \rho\ U^2}$$

In above equation P is pressure computed from ANSYS CFX, while ρ is density of fluid for wind tunnel model fluid is air having density 1.225 kg/m³, and U is velocity of flow.

2.4 Force Co-efficient

Force coefficient is found out using following equation,

$$Cf = \frac{F}{P\ X\ A}$$

Where 'F' is the estimation of total force exported from ANSYS CFX in the desired direction, 'P' is the wind pressure and 'A' is the anticipated surface area to the wind.

2.5 MATLAB for Current Work

For current thesis work MATLAB is used to develop equation of curve fitting for external pressure coefficient and force coefficient.

External pressure coefficient over surface can be changed only at angle of rotation. Curve fitting tool is used in MATLAB to find out the equation of curve fitting for similar types of models.

These information are then fitted as a 6th degree polynomial,

$C_p = \alpha_0 + \alpha_1\theta + \alpha_2\theta^2 + \alpha_3\theta^3 + \alpha_4\theta^4 + \alpha_5\theta^5 + \alpha_6\theta^6$ by least - squares regression method, here θ is angle of incidence differs from 0 to 90.

The polynomial coefficients along with the regression coefficients (R^2) for various faces. It is discovered that greater part of the polynomials are fitted well with sixth degree least-squares polynomial. All R^2 values are 1 which is very much adequate to construct a model with least-squares regression polynomial.

2.6 Validation

The current problem in ANSYS CFX is validated by taking one rectangular shape building which have dimensions 100 mm × 150 mm and having height 700 mm. & angle of incidence is 0 degree. The free stream velocity is assumed as 10 m/s at inlet.

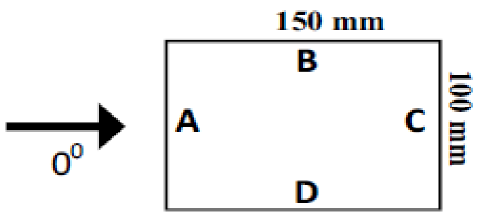


Fig - 4: Various Faces of Model

The face normal estimations of coefficient of pressure are controlled by ANSYS CFX and contrasted with different global codes.

The external pressure coefficient 'CP' is counted by below mentioned formula mentioned below,

$$Cp = \frac{P}{0.6\ V_Z^2}$$

Here, P is the wind pressure & V_Z is the design wind speed. The external surface pressure coefficients acquired by taking average pressure value of face, C_P (Face average value), for various faces of model are recorded and contrasted with different international standards.

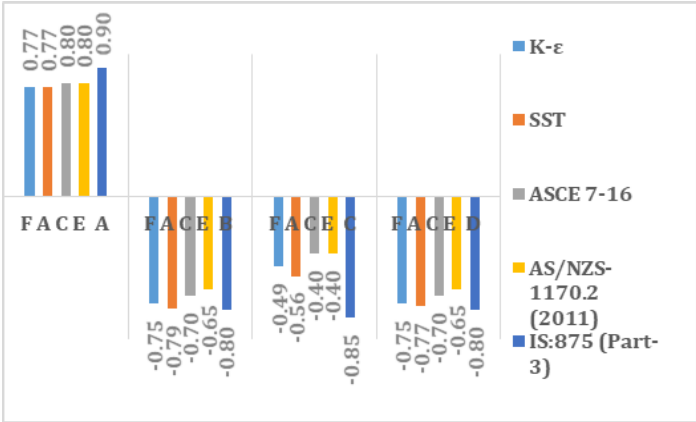


Chart - 1: Comparison of software values with codes

It can be seen from chart 1 that values are matching with minor variation from some code values. So we can say that software gives reliable results.

To look at the outcomes found from the model with those from experimental studies, a correlation is made among present and sharath et al. (2015) results. Inevitably dimensions and all other parameters identified to the wind flow are coordinated with the numerical studies. For better understanding between two turbulence models and experimental outcomes the pressure coefficients along the horizontal centerlines around the building periphery for 0 degree wind incidence are compared.

Chart - 2: Comparison of software results with experimental studies

As shown in chart 2 it can be said that analytical results are matching with that of experimental studies.

It is concluded that value obtained from the horizontal centerlines pressure coefficient k-€ model have a good similarity with experimental results than those from SST turbulence model, so further work is done based on k-€ model.

3. RESULTS AND DISCUSSION

Analysis of various models were conducted using ANSYS CFX 2019 R2. These models are prepared using Design modeler in ANSYS. Various models were rotated with respect to one point from 0 degree to 90 degree at 15 degree interval.

Meshing is done as described in numerical study and to simulate flow inlet velocity is considered 11 m/s. while outlet is considered having relative pressure of 0 Pa.

Different limit conditions, for example, free slip wall for side walls & no slip wall for building and ground are considered.

In solver control iteration size is considered 1000 so it can go up to 1000 iteration for solution and residual target is considered for accuracy of 0.0001.

By trial and error procedure in solver various values related to fluid such as k (Turbulent Kinetic Energy) and ε (Energy dissipation of kinetic energy) is obtained. While as secondary unknowns velocity in various directions such as X, Y and Z is obtained.

Based on above data in ANSYS produces results in terms of relative pressure. Global pressure, eddy viscosity temperature etc.

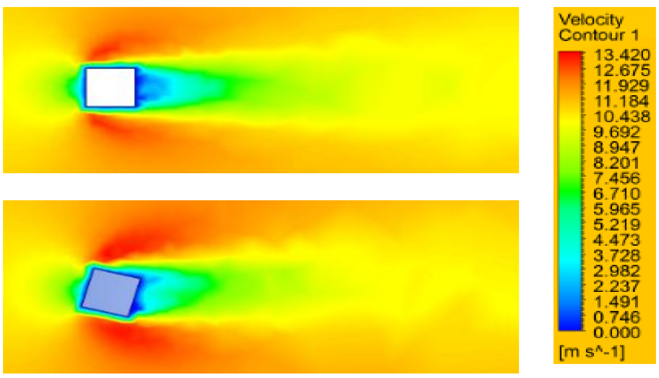
After obtaining results from ANSYS average pressure is found on surface of building, and as mentioned above equation is used to find out average external pressure co-efficient.

Similarly average force co-efficient is found in X and Z direction by taking average force on building, and putting value in equation described previously. Face area is considered on single face in order to find average force coefficient.

Based on results obtained of pressure co-efficient, curve fitting is done in MATLAB and polynomial equation is developed by least square regression method.

3.1 flow around building

Analysis is carried out in ANSYS CFX and results in post processing is obtained. Flow around building is observed at height 0.782 m and results of various shapes are shown below.



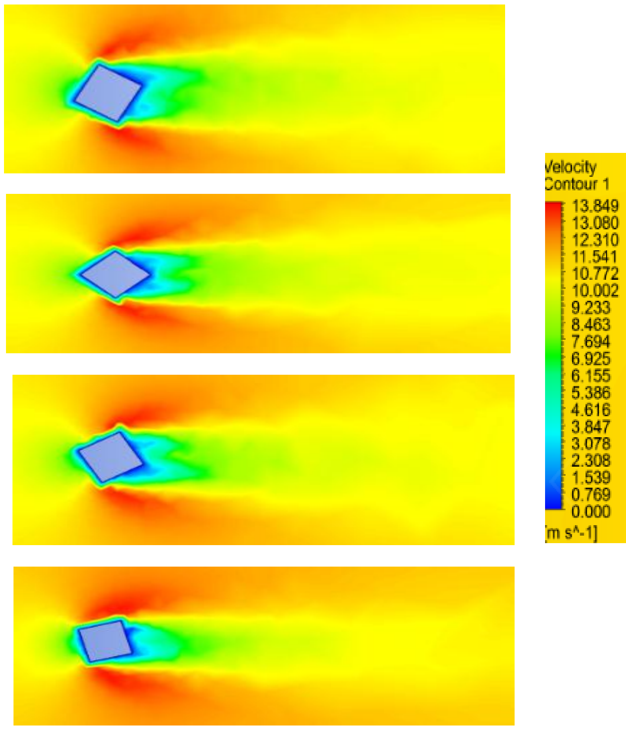
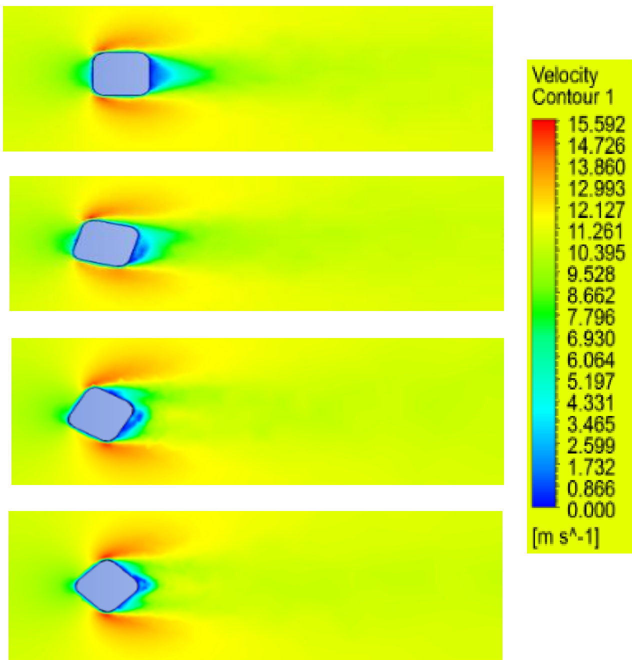


Fig - 5: Flow around square model

As shown in figure for various angle of incidence flow is observed around building and it is observed that largest wake is observed in 0 degree while smallest wake is observed at 15 and 75 degree.



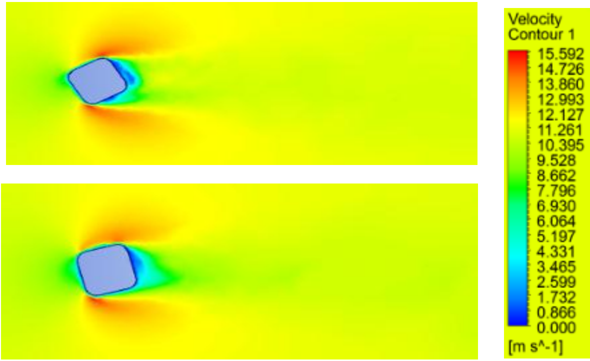


Fig - 6: Flow around rounded model

As shown in figure there is very small wake observed in rounded model compared with square model. However velocity around building goes up to 15 m/s. as observed in velocity contour while in square model it goes up to 13 m/s. However as rotation increases velocity increases around building.

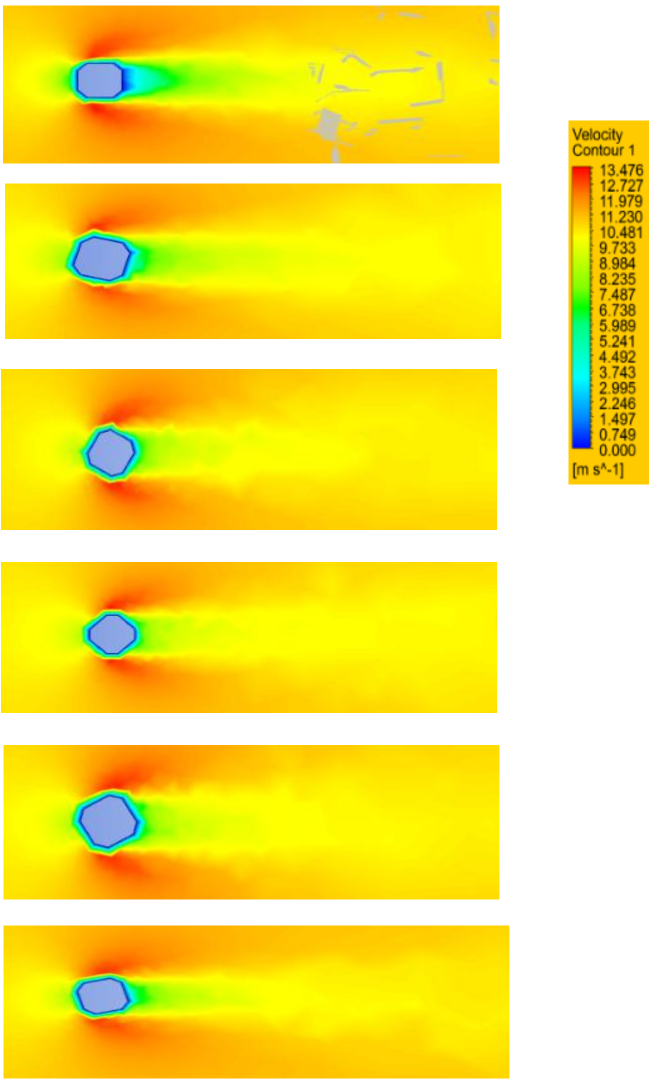


Fig - 7: Flow around chamfered model

For model chamfered similar behavior is observed like square model with velocity increase 13 m/s. However there is small wake observed compared to square model. Small wake is observed for angle 30 degree and 75 degree.

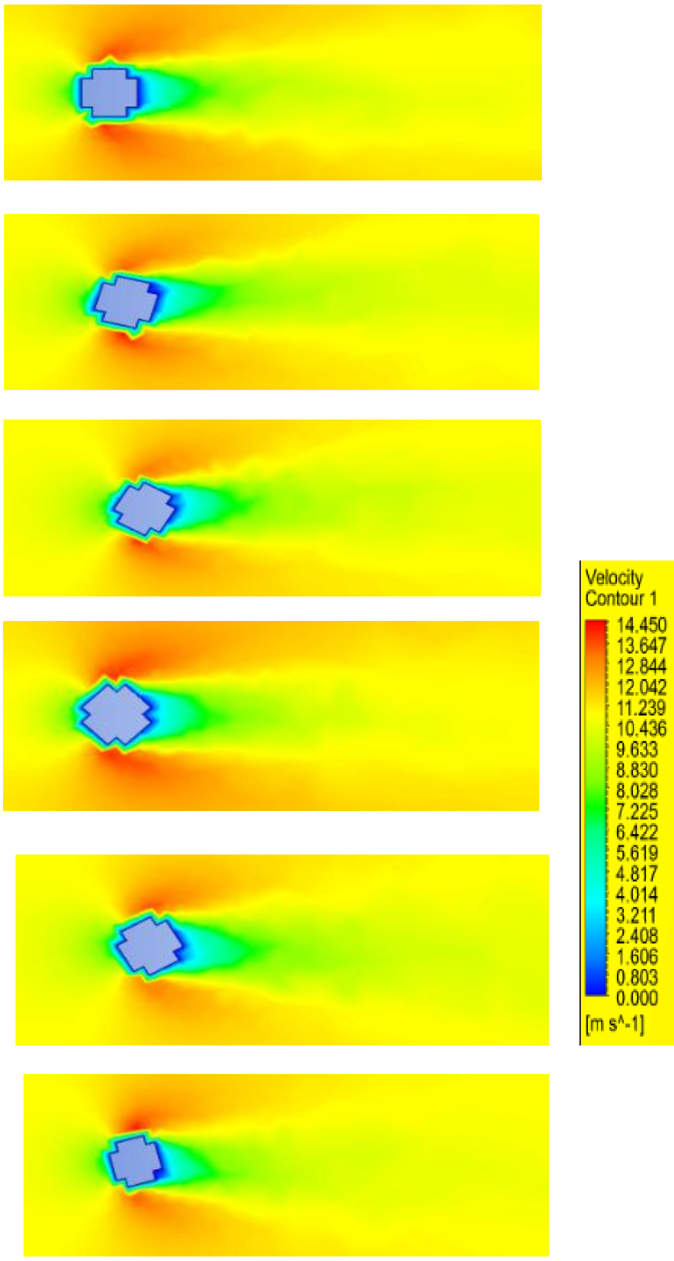


Fig - 8: Flow around Recessed Model

As shown in figure compared to square model recessed model has wider wake with velocity being greater up to 14 m/s. However comparing rotation of model shows 45 degree has narrow wake compared to other rotation angle.

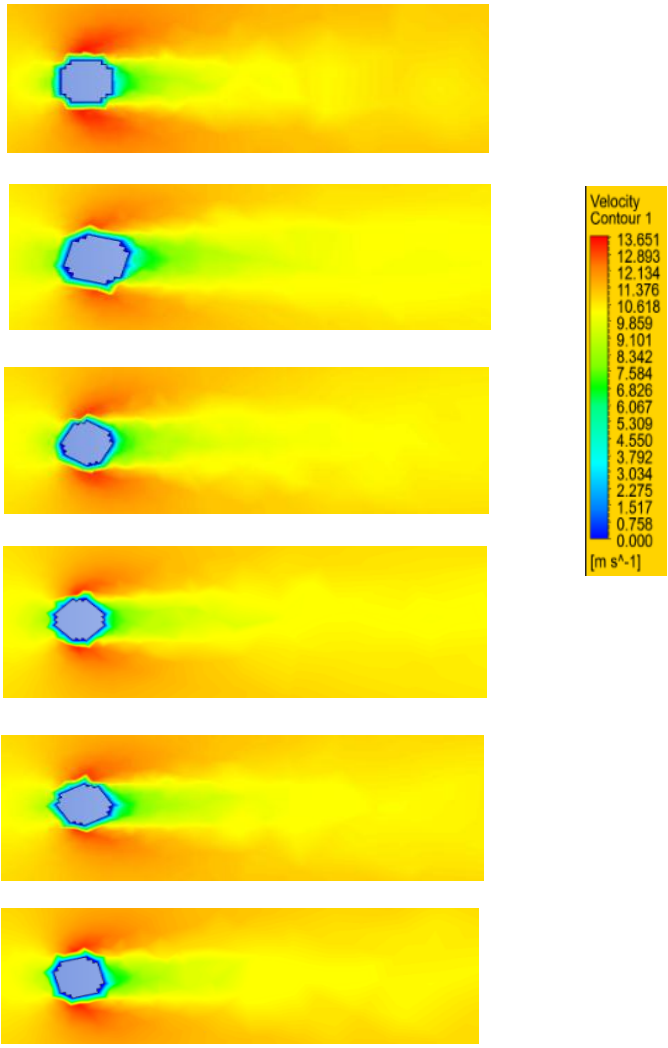


Fig - 9: Flow around Double Recessed Model

From figure it can be said that behavior of double recessed model is quite similar to model chamfered having similar velocity around 13 m/s. However in all rotation angle 15 degree observes largest wake.

3.2 Pressure contours on various faces of building

From post processing results in ANSYS pressure contours were taken and shown below.

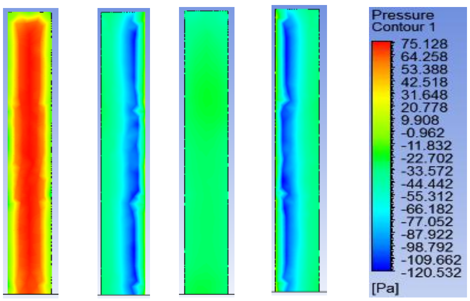


Fig - 10: Pressure contours for square 0 Degree

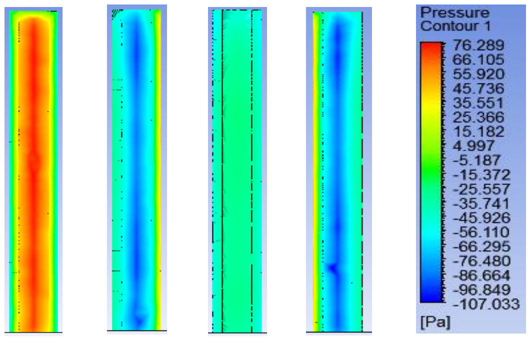


Fig - 11: Pressure contours for chamfered 0 degree

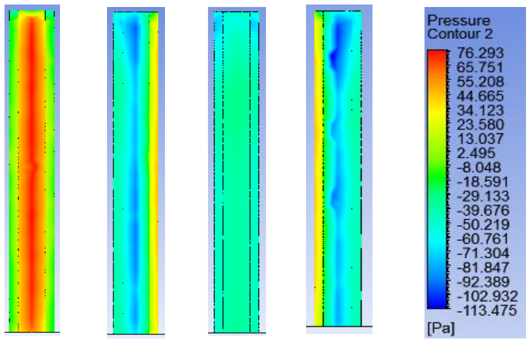


Fig - 12: Pressure contours for recessed 0 degree

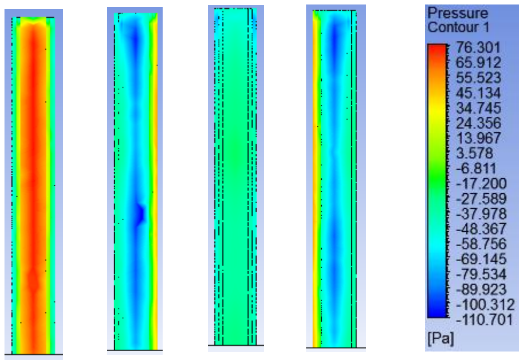


Fig - 13: Pressure contours for double recessed 0 degree

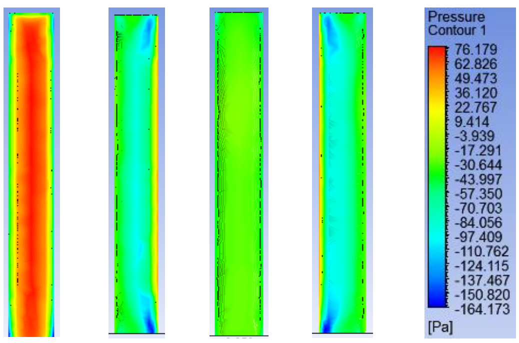


Fig - 14: Pressure contours for rounded 0 degree

Similarly, Pressure contours were captured for other angle of incidence, and following observations were made.

- Maximum negative pressure is observed in model rounded for all angle of incidence comparing other models.
- Maximum positive pressure is more or less similar in all models for all angle of incidence.

3.3 Average pressure co-efficient for various models

Comparison of various model with square model at various angle of incidence is done and average pressure coefficients were found out from above mentioned equation.

Table - 1: Average pressure co-efficient

Θ	Model	Face A	Face B	Face C	Face D
0	SQ	0.661	-0.858	-1.136	-0.858
	RO	0.314	-0.698	-0.422	-0.698
	CH	0.560	-0.983	-0.492	-0.983
	RE	0.397	-0.329	-0.619	-0.329
	DR	0.335	-0.523	-0.523	-0.523
15	SQ	0.613	-0.971	-0.764	-0.557
	RO	0.292	-0.645	-0.551	-0.454
	CH	0.527	-0.728	-0.518	-0.588
	RE	0.287	-0.820	-0.894	-0.377
	DR	0.137	-0.894	-0.577	-0.180
30	SQ	0.573	-0.942	-0.942	0.118
	RO	0.266	-0.670	-0.670	0.162
	CH	0.234	-0.662	-0.869	-0.180
	RE	0.046	-0.848	-1.092	-0.442
	DR	0.078	-0.927	-0.927	-0.089
45	SQ	0.333	-0.985	-0.985	0.333
	RO	0.253	-0.762	-0.762	0.253
	CH	0.111	-0.926	-0.926	0.111
	RE	-0.307	-1.199	-1.199	-0.307
	DR	-0.072	-0.945	-0.945	-0.072
60	SQ	0.247	-0.923	-0.923	0.325
	RO	0.059	-0.865	-1.070	0.367
	CH	0.203	-0.881	-0.881	0.347
	RE	-0.291	-1.029	-0.700	0.202
	DR	-0.245	-1.158	-0.743	-0.079
75	SQ	-0.143	-0.538	-0.775	0.568
	RO	-0.991	-0.591	-1.146	0.410
	CH	-0.059	-0.706	-0.706	0.517
	RE	-0.453	-1.191	-0.674	0.507
	DR	-0.217	-1.145	-0.824	0.087
90	SQ	-0.858	0.661	-0.858	-1.136
	RO	-0.698	0.314	-0.698	-0.422
	CH	-0.983	0.560	-0.983	-0.492
	RE	-0.329	0.397	-0.329	-0.619
	DR	-0.523	0.335	-0.523	-0.523

SQ: Square, RO: Rounded, CH: Chamfered, RE: Recessed, DR: Double recessed

3.4 Curve fitting equation of average pressure co-efficient

Using MATLAB curve fitting equation is developed for average pressure coefficient for various models. The polynomial pressure coefficients for different faces are shown in Table below. It is shown that most of the polynomials are fitted well with sixth degree least-squares polynomial. All R^2 values are 1, this is very accurate to make a model with least square regression polynomial expression. Below equation shows sixth degree polynomial.

$$FACE = \alpha_0 + \alpha_1\theta + \alpha_2\theta^2 + \alpha_3\theta^3 + \alpha_4\theta^4 + \alpha_5\theta^5 + \alpha_6\theta^6$$

This curve fitting equation can be used to find out values of average pressure co-efficient at various faces of models for angle of incidence other than given here.

Table - 2: Curve fitting equation co-efficient

FACE		α_0	α_1	α_2
A	SQ	0.661	-0.06492	0.008606
	RO	0.314	-0.04132	0.006155
	CH	0.560	0.007119	-2.63E-05
	RE	0.397	-0.069	9.34E-03
	DR	0.335	-0.02122	3.83E-04
B	SQ	-0.858	-0.03467	0.003220
	RO	-0.698	0.01782	-0.001911
	CH	-0.983	-0.05365	0.01046
	RE	-0.329	-0.2256	0.02572
	DR	-0.523	-0.0345	0.0001834
C	SQ	-1.136	0.09145	-0.007296
	RO	-0.422	0.02844	-0.005347
	CH	-0.492	0.07080	-8.81E-03
	RE	-0.619	-0.1187	0.01444
	DR	-0.523	0.02376	-0.00216
D	SQ	-0.858	-0.01685	0.002901
	RO	-0.698	-0.09041	0.013040
	CH	-0.983	0.02477	-9.26E-05
	RE	-0.329	-0.0156	0.00208
	DR	-0.523	0.05844	-0.004378

		α_3	α_4	α_5	α_6
A	SQ	-0.0004135	8.84E-06	-8.70E-08	3.20E-10
	RO	-0.0003425	8.72E-06	-1.02E-07	4.42E-10
	CH	-7.23E-05	2.64E-06	-3.28E-08	1.35E-10
	RE	-0.0004888	1.11E-05	-1.13E-07	4.30E-10
	DR	2.59E-05	-1.32E-06	1.97E-08	-9.43E-11
B	SQ	-0.0001237	2.26E-06	-1.98E-08	7.10E-11
	RO	9.17E-05	-2.19E-06	2.38E-08	-9.18E-11
	CH	-0.0005321	1.16E-05	-1.15E-07	4.34E-10
	RE	-0.001184	2.54E-05	-2.55E-07	9.71E-10
	DR	5.09E-05	-1.53E-06	1.47E-08	-4.01E-11
C	SQ	0.0002454	-4.20E-06	3.64E-08	-1.26E-10
	RO	2.69E-04	-5.97E-06	5.98E-08	-2.19E-10
	CH	0.0003521	-6.70E-06	6.24E-08	-2.27E-10
	RE	-0.0007323	1.68E-05	-1.76E-07	6.78E-10
	DR	3.95E-06	1.64E-06	-2.94E-08	1.48E-10
D	SQ	-4.63E-07	-2.58E-06	4.78E-08	-2.53E-10
	RO	-5.19E-04	9.33E-06	-7.83E-08	2.46E-10
	CH	2.72E-05	-1.19E-06	1.81E-08	-9.32E-11
	RE	-0.0001226	3.03E-06	-3.01E-08	9.90E-11
	DR	1.87E-04	-4.22E-06	4.60E-08	-1.91E-10

3.5 Average Force Co-efficient for various models

Average force coefficients were found out for direction X and Z based on data of forces taken from ANSYS post processing forces divide by force computed from pressure at various faces.

Below table shows force coefficient for various models at various angle of incidence.

Table - 3: Average Force Co-efficient in X and Z direction

Direction	Model	C_{Fz}	C_{fX}
0	SQ	-0.01333	4.69E-05
	RO	-0.02298	-0.00111
	CH	-0.01374	-0.00018
	RE	-0.02176	-0.00014
	DR	-0.02345	0.000572
15	SQ	-0.01622	-0.0028
	RO	-0.02768	-0.00408
	CH	-0.01617	-0.00218
	RE	-0.02895	-0.00041
	DR	-0.05805	0.000373
30	SQ	-0.01938	-0.0009
	RO	-0.03564	-0.00214
	CH	-0.04002	-0.00164
	RE	-0.17603	-0.00184
	DR	-0.10344	-0.00143
45	SQ	-0.03533	-4.4E-05
	RO	-0.03753	0.000129
	CH	-0.08764	0.000236
	RE	0.02436	0.000174
	DR	0.109588	-0.00041
60	SQ	-0.04476	0.000983
	RO	-0.15869	0.001653
	CH	-0.04591	0.001135
	RE	0.027245	0.0015
	DR	0.032483	0.001099
75	SQ	0.069602	0.004906
	RO	0.008197	0.004098
	CH	0.146127	0.002167
	RE	0.018384	0.00083
	DR	0.037707	-0.00021
90	SQ	0.010265	-6.1E-05
	RO	0.010325	0.002474
	CH	0.00782	0.000323
	RE	0.02631	0.000115
	DR	0.015044	-0.00089

3.6 Percentage change in average pressure co-efficient

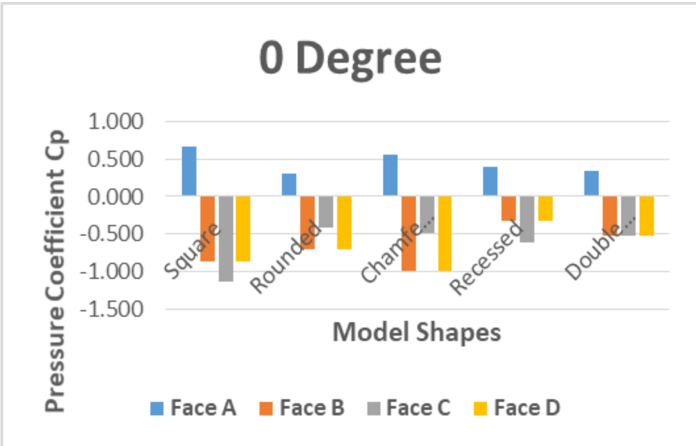


Chart -3: Comparison of models for 0 degree

Smallest value of face A and C is observed for rounded model. Percentage change in value of pressure coefficient is 52 % and 62 %.

For face B and D smallest values is observed for model recessed. Percentage change observed in values are 52 %.

However we can observe that overall smaller values are observed in double recessed model having consistency in values.

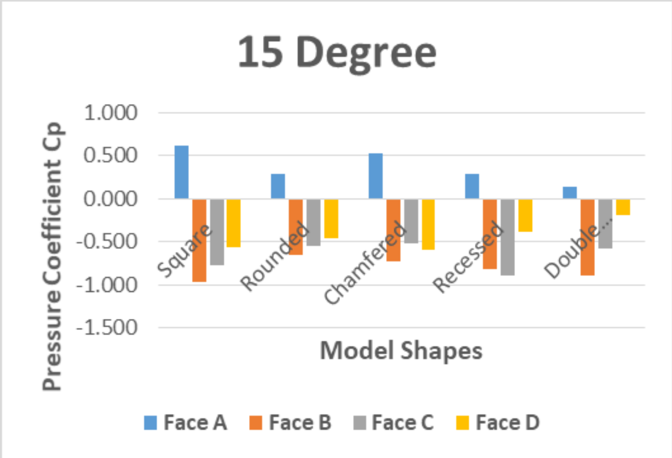


Chart - 4: Comparison of models for 15 degree

Smallest value of face A and D is observed for double recessed model. Percentage change in value of pressure coefficient is 53 % and 32 %.

For face B smallest values is observed for model rounded. Percentage change observed in values are 34 %.

For face C smallest values is observed for model chamfered. Percentage change observed in values are 32 %.

However we can observe that overall smaller values are observed in rounded model having consistency in values.

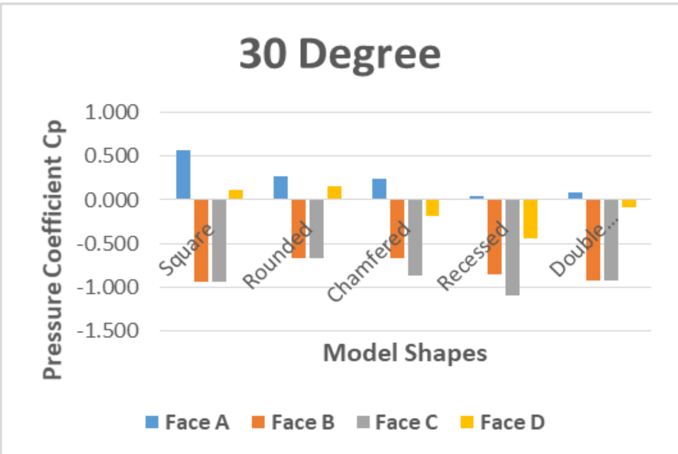


Chart - 5: Comparison of models for 30 degree

Smallest value of face A is observed for recessed model. Percentage change in value of pressure coefficient is 92 %.

For face B smallest values is observed for model chamfered. Percentage change observed in values are 29 %.

For face C smallest values is observed for model rounded. Percentage change observed in values are 28 %.

For face D smallest values is observed for model double recessed. Percentage change observed in values are 24 %. However behavior of section changes from pressure to suction.

However we can observe that overall smaller values are observed in rounded model having consistency in values.

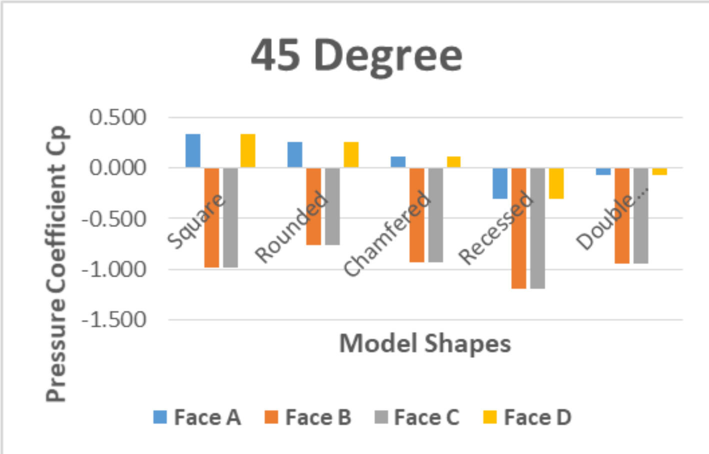


Chart - 6: Comparison of models for 45 degree

Smallest value of face A and D is observed for double recessed model. Percentage change in value of pressure coefficient is 34.96 %.

For face B and C smallest values is observed for model rounded. Percentage change observed in values are 24 %.

However we can observe that overall smaller values are observed in rounded model having consistency in values.

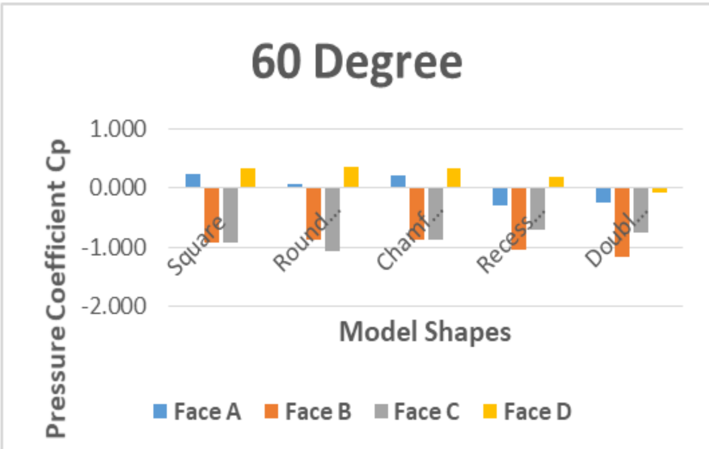


Chart - 7: Comparison of models for 60 degree

Smallest value of face A is observed for rounded model. Percentage change in value of pressure coefficient is 76 %.

For face B smallest values is observed for model rounded. Percentage change observed in values are 7 %.

For face C smallest values is observed for model recessed. Percentage change observed in values are 24 %.

For face D smallest values is observed for model double recessed. Percentage change observed in values are 75 %. However behavior of section changes from pressure to suction.

However we can observe that overall smaller values are observed in recessed model having consistency in values.

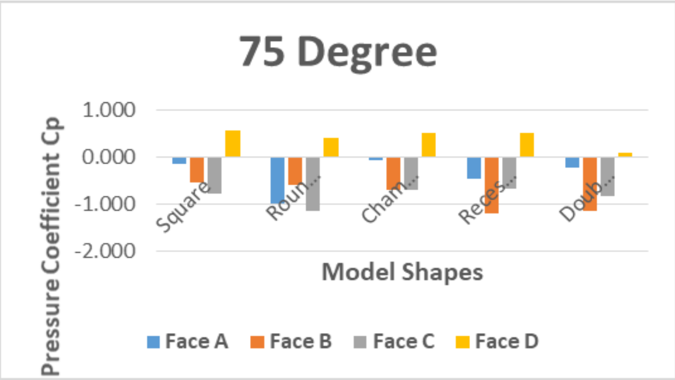


Chart -8: Comparison of models for 75 degree

Smallest value of face A is observed for chamfered model. Percentage change in value of pressure coefficient is 59 %.

For face B smallest values is observed for model square.

For face C smallest values is observed for model recessed. Percentage change observed in values are 13 %.

For face D smallest values is observed for model double recessed. Percentage change observed in values are 85 %.

However we can observe that overall smaller values are observed for chamfered model having consistency in values.

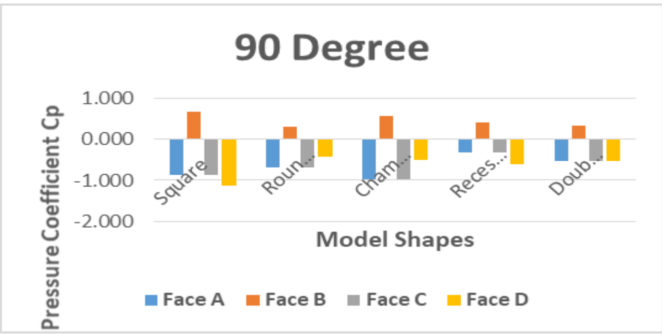


Chart - 9: Comparison of models for 90 degree

Smallest value of face A is observed for recessed model. Percentage change in value of pressure coefficient is 61 %.

For face B smallest values is observed for model rounded. Percentage change in value of pressure coefficient is 52 %.

For face C smallest values is observed for model recessed. Percentage change observed in values are 61 %.

For face D smallest values is observed for model rounded. Percentage change observed in values are 62 %.

However we can observe that overall smaller values are observed in model double recessed having consistency in values.

4. CONCLUSION

The present study is conducted to investigate the behavior of high rise building under wind load by rotating building at various angles and by applying corner modification. ANSYS CFX tool is used to conduct analysis by constructing wind tunnel like model. Wind flow is considered as fluid flowing in wind tunnel. Geometrical changes applied to building are changing corner 20 % modification. Parametric study of 35 building models have been performed using finite element analysis.

Based on results of study, the following conclusions were drawn,

- Corner modification and angle of incidence plays important role in reducing pressure on surface of high rise building.
- For angle of incidence 0 degree as well as 90 degree double recessed model shows minimum pressure values on surface of building.
- Another observation made is there is smaller wake for both angles, so double recessed can be effectively reduce pressure and suitable for these angles.
- For angle of incidence 15 degree, 30 degree and 45 degree rounded model shows overall minimum pressure on surface of building.
- Smallest wake is observed in rounded model for all angle of incidence.
- For angle of incidence 60 degree recessed model shows consistency in value of average pressure coefficient.
- For angle of incidence 75 degree chamfered model shows consistency in value of average pressure coefficient.

- Based on results polynomial analytical expression is developed for studying similar type of model having different faces of building model.
- Model accuracy is measured by R^2 value. These expression can be used to study similar type of model having angle of incidence between 0 and 90.
- Simply putting value of angle of incidence in this equation one can find out pressure coefficient at various faces of various models.
- Overall, it is observed that windward face experience positive pressure coefficient. However as angle of incidence changes for face-A it slowly changes to negative due to change in orientation.

5. FUTURE SCOPE

In current work, analysis of wind load is done using computational tool ANSYS CFX. Pressure on surface of building is found and average pressure coefficient is found out taking average of pressure on surface at various angle of incidence. Polynomial equation is developed using MATLAB. Conclusions were drawn based on various factors.

Furthermore, one can perform analysis of various triangular shape buildings as well as cluster of building can be studied taking real world problem and flow behavior can be studied on main building. Building with openings can be studied having irregular shape and size.

This extend the scope of research in the field of structural dynamics to make it even more effective, efficient and workable.

REFERENCES

1. Yi Li, Xiang Tian, Kong Fah Tee, Qiu-Sheng Li, Yong-Giu Li "Aerodynamic treatments for reduction of wind loads on high rise building", Journal of Wind Engineering and Industrial Aerodynamics, 2018, V.172, Pg. No. 107-115.
2. Youngchul Kim, Jun Kanda "Characteristics of aerodynamic forces and pressures on square plan building with height variation", Journal of wind engineering and Industrial Aerodynamics, 2010, V.98, Pg. No. 449-465.
3. Ahmed Elshaer, Girma Bitsuamlak, Ashraf El Damatty, "Wind load reduction due to building corner modification", Annual Conference of the CFD Society of Canada, Toronto, June-2014, Volume: 22.
4. K. Shruti, P. N. Rao, and G. R. Sabareesh, "Across Wind Load Analysis Using CFD for Sustainable Design of Tall Structures", Advances and Challenges in Structural Engineering. GeoMEast 2018. Sustainable Civil Infrastructures. Springer, Cham, 2019, Pg. No. 230-243.
5. Norbert Jendzelovsky, Roland Antal, Lenka Konecna "Investigation of the External Pressure Coefficients on the Facade of the Triangular High-Rise Structure with Curved Corners", Procedia Engineering, V.190, Pg. No. 398-405.
6. H. Sarath Kumar, S. Selvi Rajan, A. Joseph Andrew, G. Ramesh Babu, N. Shrinivasa Rao, J. Guru Jawahar "Aerodynamic coefficients for a rectangular tall building under sub-urban terrain using wind tunnel", Asian Journal of Civil Engineering(BHRC), 2016, V.17, Pg.No.325-333.
7. Prasenjit Sanyal, Sujit Kumar Dalui "Effects of courtyard and opening on a rectangular plan shaped tall building under wind load", International Journal of Advanced Strucutral Engineering, 2018, V.10, Pg.No.169-188.
8. J. Franke, C. Hirsch, A.G. Jensen, H.W. Krus, M. Schatzmann, P.S. Westbury, S.D. Miles, J.A. Wisse, N.G. Wright "Recommendation on the use of CFD in wind engineering", Von karman Institute for fluid dynamics, COST action C14 : Impact of wind and storm on city life and built environment.
9. Eswara Kumar Bandi, Yukio Tamuraa, ,Akihito Yoshida, Yong Chul Kim, Qingshan Yang "Experimental investigation on aerodynamic characteristics of various triangular-section high-rise buildings", Journal of wind engineering and Industrial Aerodynamics, 2013, V.122, Pg. No. 60-68.
10. J. A. Amin, A. K. Ahuja, "Experimental study of wind pressures on irregular plan shape buildings", BBAA VI International Colloquium on: Bluff Bodies Aerodynamics & Applications, 2008, Pg. No. 20-24.
11. Luigi Carassale, Andrea Freda, Michela Marrè-Brunenghi, "Experimental investigation on the aerodynamic behavior of square cylinders with rounded corners", Journal of fluids and structures, 2014, Pg. No. 195-204.
12. B.Patidar, A. R. Patil, K. Thiele, S.Mandal, "Across-Wind loading for Structures: An overview", 7th National Conference on Wind Engineering, 2014.
13. Abdollah Baghaei Daemei, Roya Aeinehvand, Mostafa Kazemi, Elham Mehrinejad Khotbehsara, "Numerical simulation of wind impacts on building blocks: Towards sustainable architecture strategies", European journal of sustainable development research, 2019.

14. IS:875 (2015) Indian Standard code of practice for design loads (other than earthquake) for buildings and structures, part-3 (Wind Loads). Bureau of Indian Standards, New Delhi.
15. AS, NZS: 1170.2 (2011) "Structural design actions, part 2: wind actions." Standards Australia/Standards New Zealand, Sydney.
16. ASCE: 7–16 (2016) "Minimum design loads for buildings and other structures." Structural Engineering Institute of the American Society of Civil Engineering, Reston
17. Bungale S. Taranath, "Reinforced Concrete Design of Tall buildings", CRC Press, Taylor and Francis Group.