

Effect of Wall Tapper and Attack Angle on Mean Flow Structure around a Pyramid

Mr. Subhrajit Beura^a, Dr. Dipti Prasad Mishra^{b,*}

^aInstitute of Technical Education and Research, S'O'A University, Bhubaneswar – 751 030, Odisha, India

^bBirla Institute of Technology, Mesra, Ranchi – 835215, Jharkhand, India

Abstract: Numerical investigations of a surface mounted pyramid have been carried out by solving the conservation equations of mass and momentum. The resulting equations have been solved numerically using finite volume technique in an unstructured grid employing eddy viscosity based two equation $k-\varepsilon$ turbulence model. It has been found from the computation that there exists optimum apex angle and attack angle for maximum turbulent intensity. From numerical investigations it is also found that there exists optimum apex angle and attack angle for maximum reattachment distance. For same height and volume, the reattachment distance is more for the case square based prism compared to any other shape bluff bodies. The computed velocity profiles at the rear side of the pyramid shows that the intensity of back flow is more towards the bottom of the domain.

Key words: Pyramid, attack angle, apex angle, back flow, reattachment distance, vertex

Nomenclature

B Breadth of the domain (m)

$$D_k \quad \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$

$$D_\varepsilon \quad \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$

g Acceleration due to gravity (m/s^2)

H Height of the domain (m)

h Height of the pyramid (bluff body) (m)

k turbulent kinetic energy (m^2/s^2)

L Length of the domain (m)

P Pressure (Pascal)

Re Reynolds number

U_∞ Velocity of air at inlet to domain (m/s)

V Velocity (m/s)

x Distance in x-direction from inlet (m)

X Reattachment distance (m)

Greek Symbol

ρ Density (kg/m^3)

ρ_∞ Density of ambient (kg/m^3)

μ Dynamic viscosity (Pa.s)

μ_t Turbulent viscosity (Pa.s)

ε Dissipation rate of turbulent kinetic energy (m^2/s^3)

ζ Apex angle (Degree)

α Attack angle (Degree)

σ Prandlt Number

ϕ Scalar variable either k or ε

I. Introduction

The flow around the surface mounted bluff bodies is very complex which gives rise to large vortical structures, flow separation and reattachments. Flow separation results in strong shear layers and velocity and pressure fluctuations in those layers are very influential. The wake in resulting flow is highly turbulent and forms turbulent mixing zones around and downstream of the bluff bodies creating vortexes which strongly affect the flow pattern and mixing characteristics. There are also great impacts of sizes, shapes and positions of bluff bodies on flow structure, pressure field as well as mixing of the fluid at the rear side of the bluff bodies.

Few researchers have performed two dimensional numerical analysis to study the fluid flow characteristics of symmetrical bluff bodies [1, 2, 3, 4] which are placed in-line to the flow of fluid. But, considerable research works have been carried out experimentally to analyze the flow field and heat transfer characteristics around the bluff bodies.

Castro and Robins [5] experimentally measured surface pressure and mean fluctuation velocities within the wake region of surface mounted cube. Igarashi [6] investigated the average heat transfer coefficient a square prism and concluded that the average heat transfer coefficient was minimum at an attack angle 12° - 13° and maximum value of heat transfer coefficient at an attack angle of 20° - 25° . In case of circular cylinder Agui and Andreopoulos [7] found that there is always a primary vortex present in the flow which induces an eruption of wall fluid which often results in the formation of counter rotating vortices. Tieleman et al. [8] discussed about

the magnitude and the distribution of mean and fluctuating pressure coefficients associated with corners and edges on the top surface of surface-mounted rectangular prisms immersed in a variety of turbulent shear layers and observed that pressure variation with the turbulence intensity and the azimuth angle of the incident flow. Flow visualization study of a linearly tapered, oscillating cylinder at low Reynolds numbers flow revealed the formation of two different primary vertical patterns along the span of the cylinder Techet *et al.* [9]. Martinuzzi and Havel [10] experimentally investigated the flow around two in-line surface-mounted cubes in a thin laminar boundary layer as a function of obstacle spacing and they found that for small spacings, the shear layer separating from the first cube reattaches on the sides of the second obstacle and wake periodicity can only be detected in the wake of the downstream cube. Castro and Rogers [11] experimentally studied the vortex shedding behind flat tapered plates placed normal to an air stream. Martinuzzi and Abuorma [12] had conducted an experimental investigation on turbulent flow around square-based wall-mounted pyramids in thin and thick boundary layers as a function of the pyramid apex angle and angle of attack. For thin boundary layers, wake periodicity and for slender pyramids ($15^\circ < \zeta < 75^\circ$), the periodic formation and shedding of vortices is observed. Abu Omar and Martinuzzi [13] experimentally investigated the turbulent flow around square-based, surface-mounted pyramids, of height h , in thin and thick boundary layers. Using a modified pressure coefficient, it is found that the mean wall pressure distribution in the wake collapses on to a single curve for several different apex angles of pyramids and angles of attack. Abu Omar and Martinuzzi [14] experimentally studied vortical structures around a surface-mounted sharp-edged pyramid in a thin boundary layer and found for slender pyramids ($\zeta < 75^\circ$), periodic flow in the wake of the slender pyramid is a result of regular vortex shedding. The flow pattern is similar to an owl-face of the second kind. Chyu and Natarajan [15] comparatively examined five basic geometries (cylinder, cube, diamond, pyramid & hemisphere) at certain Reynolds number to determine the effect of single roughness element on heat and mass transfer. The results shows that the upstream horseshoe vortex system and the inverted arch shaped vortex immediately behind the element are dominating effect in element end wall interaction.

Some investigators have performed three-dimensional numerical analysis of bluff bodies. Yaghoubi and Velayati [16] studied numerically the conjugate heat transfer for three-dimensional developing turbulent flows over an array of cubes in cross-stream direction. They established new correlations to predict average Nusselt number and fin efficiency for an array of inline cubes. Farhadi and Rahnama [17] studied flow over a wall-mounted cube in a channel at a Reynolds number of 40000 and found that implementation of a wall function does not improve the results considerably.

After a thorough review of works of different investigators it has been seen that the three dimensional numerical analysis of flow around the bluff bodies having complicated geometry has received much less attention. In the present work we have carried out three dimensional numerical analysis of flow field around the bluff bodies with complicated structure. Initially investigation is started from pyramidal structure because numerical analysis of this geometry has not yet been done. Subsequently the analysis will be extended to other bluff bodies such as prisms, cubes, cylinders and cones for comparisons. It is attempted to match our present numerical results with the existing experimental results (Martinuzzi and Abuorma [12]) for velocity field.

II. Mathematical Formulation

The numerical investigation has been carried out for a bluff body of pyramidal shape placed at a distance of x , from the entrance of a wind tunnel as shown in Fig. 1. The bluff body is placed in a wind tunnel of length L ,

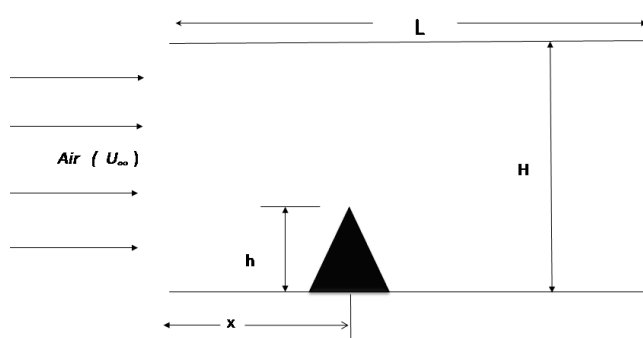


Fig. 1 Schematic diagram of Computational Domain with Bluff-Body

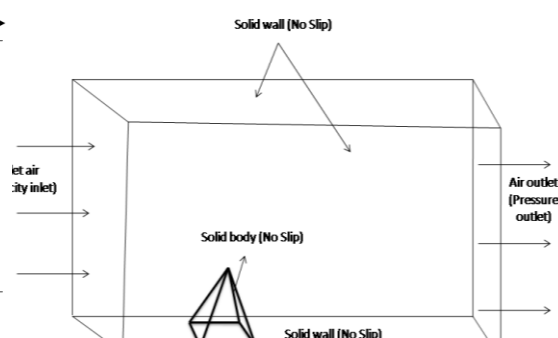


Fig. 2 Boundary conditions applied to Computational domain

Breadth B , (cannot be visible in the Fig. 1) and height H . The height of the bluff body in the present case considered to be ' h '. The air entering the wind tunnel from one side with a certain free stream velocity ' U_∞ ' and flows over the bluff body. The effects of apex angle, attack angle and shape subsequently have been varied to study the flow characteristics. The flow field in the domain will be computed by using three-dimensional

incompressible Navier-Stokes equations with a two equation based $k-\varepsilon$ (standard) turbulence model. The fluid used in the simulation is air and is treated to be incompressible while flowing around the bluff body.

A. Governing equations

The governing equations for the above analysis can be written as:

Continuity

$$\frac{\partial}{\partial x_i}(\rho U_i) = 0 \quad (1)$$

Momentum

$$\frac{D(\rho U_i)}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \overline{\rho u_i u_j} \right] \quad (2)$$

Turbulence kinetic energy - k

$$\frac{D}{Dt}(\rho k) = D_k + \rho P - \rho \varepsilon \quad (3)$$

Rate of dissipation of k

$$\frac{D}{Dt}(\rho \varepsilon) = D_\varepsilon + C_{1\varepsilon} \rho P \frac{\varepsilon}{k} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (4)$$

$$\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} - \nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right); \nu_t = 0.09 \frac{k^2}{\varepsilon} \quad (5)$$

$$D_\phi = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\phi} \right) \frac{\partial \phi}{\partial x_j} \right] \quad (6)$$

$$P = -\overline{u_i u_j} \frac{\partial U_i}{\partial x_j} \quad (7)$$

σ_k and σ_ε are the Prandtl numbers for k & ε

$C_{1\varepsilon} = 1.44; C_{2\varepsilon} = 1.92; C_\mu = 0.09; \sigma_k = 1.0; \sigma_\varepsilon = 1.3$

B. Boundary Conditions

The boundary conditions can be seen from Fig. 2. All the sides of the domain (except entrance and outlet) and the surface of the bluff body are solid and have been given a no-slip boundary condition. Velocity inlet boundary condition has been imposed at the entrance (left side of the domain) through which the air enters into the domain where as pressure outlet boundary conditions have been employed at the outlet face of the domain.

$$\mathbf{V} = \mathbf{V}_{\text{inlet}} \quad (8)$$

At the outlet boundary of the computational domain $p = p_a$ (atmospheric pressure), (pressure outlet boundary) (9)

At the pressure outlet boundary, the velocity will be computed from the local pressure field so as to satisfy the continuity but all other scalar variables such as k and ε are computed from the zero gradient condition, Dash [18]. The turbulent quantities k and ε , on the first near wall cell have been set from the equilibrium log law wall function as has been described by Jha and Dash [19, 20] and Jha et al. [21]. The turbulent intensity at the inlet of the nozzle has been set to 2% with the inlet velocity being known and the back flow turbulent intensity at all the pressure outlet boundaries have been set to 5%. If there is no back flow at a pressure outlet boundary then the values of k and ε are computed from the zero gradient condition at that location.

III. Numerical Solution Procedure

Three-dimensional equations of mass, momentum and energy have been solved by the algebraic multi grid solver of Fluent 14 in an iterative manner by imposing the above boundary conditions. First order upwind scheme (for convective variables) was considered for momentum as well as for the turbulent discretized equations. After a first-hand converged solution could be obtained the scheme was changed over to second order upwind so as to get little better accuracy (the velocity profile is closed a little bit towards the existing experimental results). SIMPLE algorithm for the pressure velocity coupling was used for the pressure correction equation. Under relaxation factors of 0.3 for pressure, 0.7 for momentum and 0.8 for k and ε were used for better convergence of all the variables. Tetrahedral cells were used for the entire computational domain because it is one of the best choices for such a complicated geometry. Convergence of the discretized equations were said to have been achieved when the whole field residual for all the variables fell below 10^{-3} for V_x, V_y, V_z, p, k and ε .

IV. Results and Discussions

A. Grid independent test and validation with other results

A general arrangement of meshes in XY plane has been shown in Fig. 3. We have initiated the numerical investigation from a square based pyramid of height 0.05 m at an apex angle of 60° placed on a surface of wind

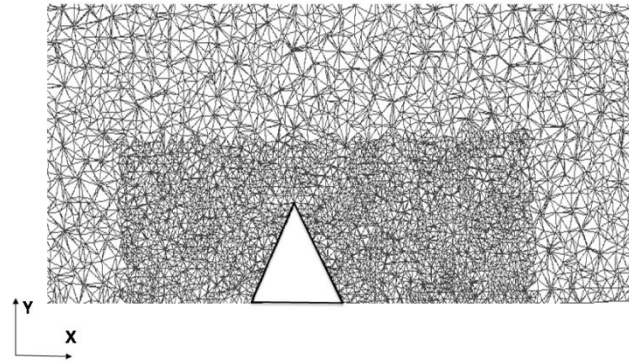


Fig. 3 A general arrangement of mesh in XY plane

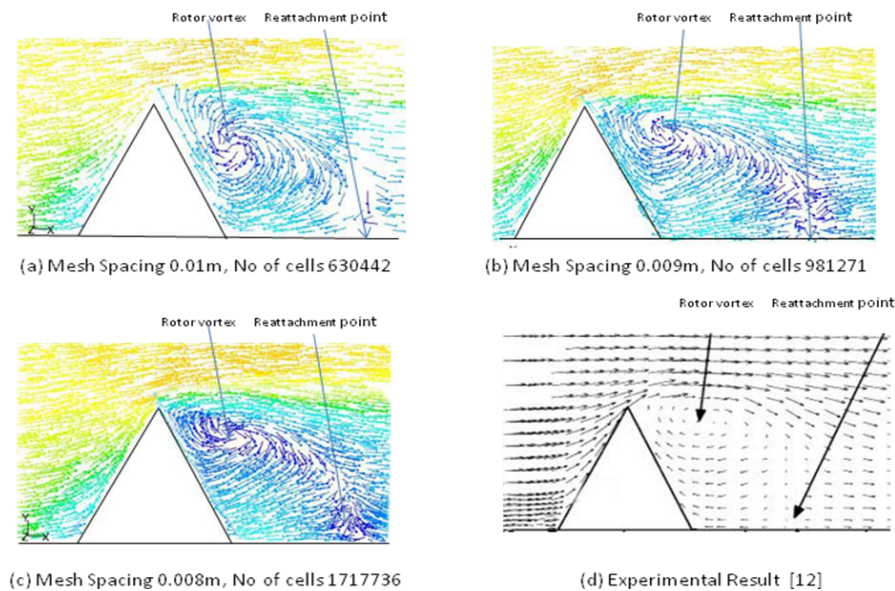


Fig. 4. Effect of mesh size on vortex and comparison with the existing experimental results

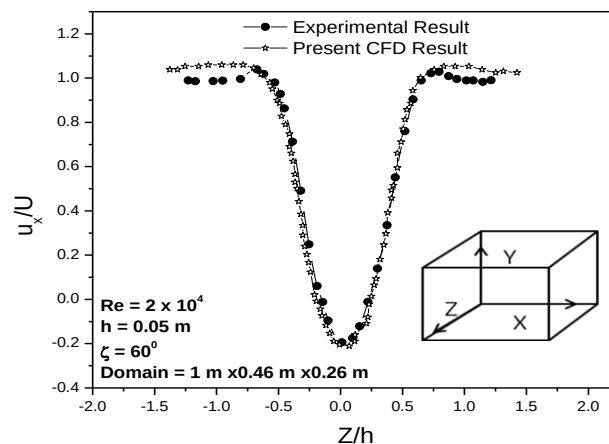


Fig.5. Mean stream wise velocity components downstream of the pyramid at $x/h=1$

tunnel of size a domain of length 1 m, height 0.26 m and width 0.46 m. The Reynolds Number at the entrance is considered for the numerical investigation is 2.3×10^4 . The investigation is started from a course mesh of tetrahedral cells. Detail of the grid independent test is shown in Fig. 4 and the corresponding most appropriate grid is chosen by comparing the rotor vortex of this investigation with existing experimental result of Martinuzzi and Abuorma [12]. For improvements in result we refined the meshes by adopting a region around the pyramid. With this grid velocity profile for the stream wise component through the centre of the two counter rotating vortices along the Z- axis is drawn (Fig. 5) at $x/h = 1$ downstream of the pyramid and at a height of $y/h = 0.3$ is computed for a Reynolds number of 2.3×10^4 and again compared with published experimental result of Martinuzzi *et al.* [12]. It is observed that, this simulation result for velocity profile matched well with the result (Fig.5) of Martinuzzi and Abuorma [12]. So for all further computation we have used $k-\epsilon$ turbulence model with same mesh structure as we have discussed above.

B. Effect of apex angle (ζ) on Turbulent Intensity

In our computation the turbulent intensity is measured at center line of the domain which is 0.05 m from the bottom wall (i.e. the line passed on the pyramid apex) for a Reynolds Number of 2.3×10^5 ($Re = \frac{\rho V h}{\mu}$). It can be

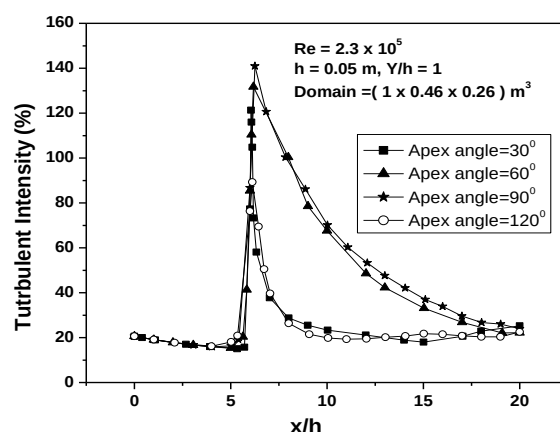


Fig. 6 Influence of apex angle on Turbulent Intensity, the optimum apex angle found to be at 90°

seen from the Fig. 6 the highest value of turbulence intensity in the fluid for the pyramid having an apex angle (ζ) 90° and lowest at $\zeta = 120^\circ$. Turbulent intensity increases from $\zeta = 30^\circ$ and reaches at the pick at $\zeta = 90^\circ$ then it falls at $\zeta = 120^\circ$. Turbulence intensity indicates the degree of disturbance in the flow field and it is evident from the result that for apex angle 90° the higher turbulent intensity indicates better mixing of the fluid and at the same time its adversely affects the stability of structure and its neighboring structure.

C. Effect of attack angle (α) on Turbulent Intensity

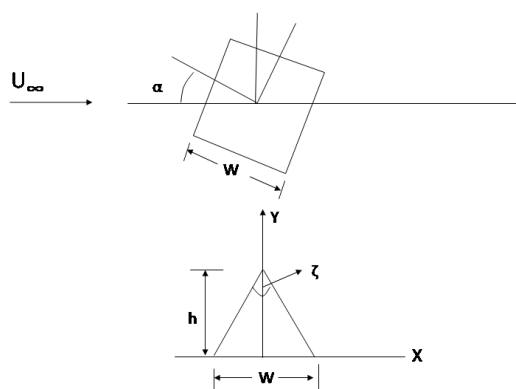


Fig. 7. Geometry of the base of the pyramid at an attack angle ' α '

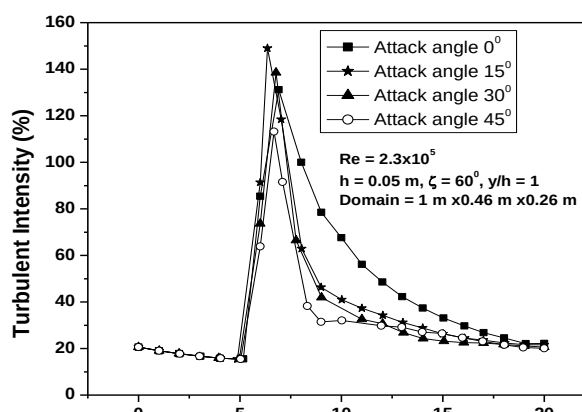


Fig. 8 Effect of attack angle on Turbulent Intensity, the optimum attack angle is found to be at 15°

The pyramid ($\zeta = 60^\circ$) is rotated through an attack angle α from 0° to 45° in anti-clock wise (as shown in Fig. 7) direction to investigate the effect turbulent intensity. As in the previous case the turbulent intensity is measured at the center line of the domain and at a height of 0.05 m from the bottom wall. It can be seen from the Fig. 8

that highest turbulent intensity is found at an apex angle of 15° . So a pyramidal object should be placed at an attack angle of 15° for better mixing of fluid.

D. Effect of apex angle (ζ) on Reattachment Distance

Fig. 9 shows the reattachment distance for different apex angle. Keeping the other parameters fixed we increase apex angle from 30° to 120° . Reattachment distance is the distance from the apex of the pyramid to a point

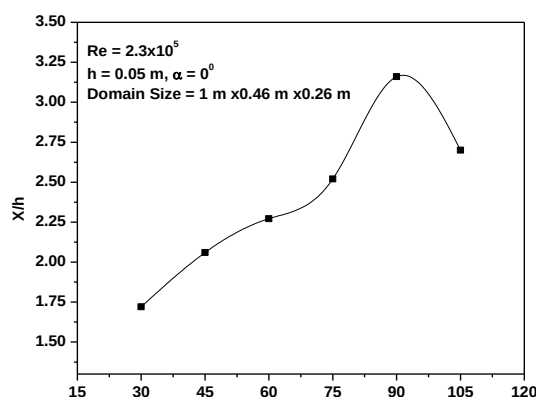


Fig. 9 Reattachment distance as a function of apex angle of the pyramid: the optimum apex angle can be seen to be at 90°

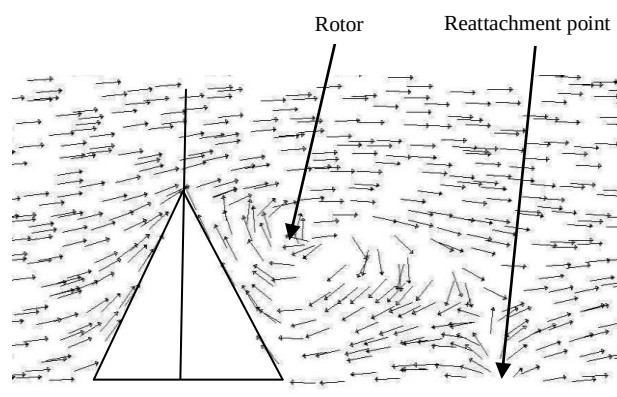


Fig. 10 Rotor vortex and Reattachment distance of pyramid in XY Plane

where the flow reattached to the bottom wall along the downstream of the flow. The reattachment distance is the indication of vortex size. It can be seen from the figure as the apex angle is increased (from 30° to 90°) the reattachment distance also increases and reached a maximum at an angle of 90° and then falls after that suggesting the existence of optimum value of apex angle for maximum reattachment distance. So in the present case it can be concluded that the size of the rotor vortex (as it can be seen in Fig.10) is highest in case of 90° apex angle. In bigger pyramids i.e. apex angle more than 90° due to the decrease in wall taper low separation does not occur and relatively weaker low pressure area is formed in downstream of the pyramid. So there is no reattachment point and rotor vortex formed in downstream.

E. Effect of attack angle (α) on Reattachment Distance

The numerical investigation is performed to determine the reattachment distance when the attack angle is changing from 0° to 45° in counter clockwise direction for a square based pyramid for an apex angle of 60° . It is seen from the Fig. 11 that the reattachment distance increases when the attack angle is increasing from 0° to 20° and it falls after that, indicating an optimum value of attack angle for maximum reattachment distance. So it is evident from the figure that the size of the rotor vortex is highest for the 20° attack angle. However, similar result is also obtained when the pyramid is rotated in clockwise direction. So better mixing can be made by twisting a pyramid through 20° angle either clockwise or anticlockwise direction.

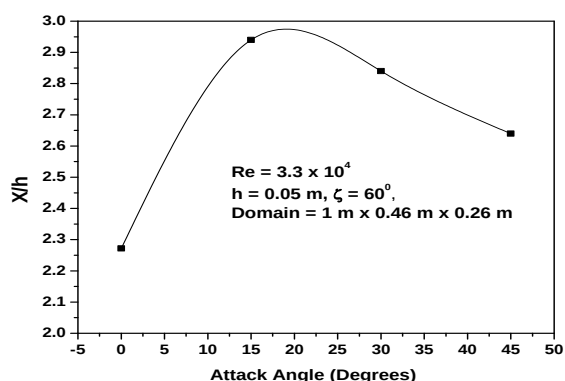


Fig. 11 Reattachment distance as a function attack angle, the optimum attack angle can be seen at around

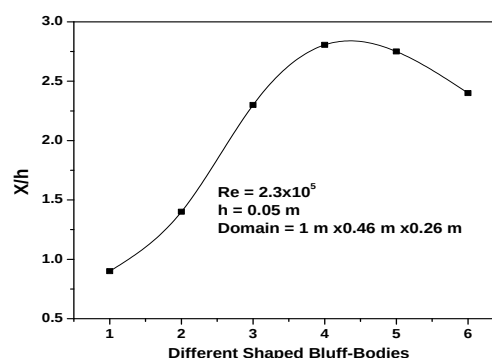


Fig. 12 Reattachment distance of different shaped Bluff Bodies having same volume and height (1 = Cone, 2 = Cylinder, 3 = Square Base Pyramid, 4 = Square base Prism, 5 = Triangular Base Pyramid, 6 = Pentagon Base Pyramid).

F. Reattachment distance for different shaped bodies

Numerical computations have been performed to see the effect of shape of the bluff body on the reattachment distance. In the present CFD analysis we considered six bluff bodies of different shapes viz. cone, cylinder, square based pyramid, square based prism, triangular based pyramid and pentagonal pyramid. The height and volume of the bluff bodies are same to that of square based pyramid having an apex angle of 60° . It can be seen from Fig. 12 the reattachment distance is maximum in case of square based prism and minimum for the case of cone. It clearly indicates that for the same height of different shapes bodies cone is more stable compared to any other shape. However, for better mixing point of view one should recommend square based prism as it has a largest rotor vertex.

G. Velocity profile along the downstream of the flow at the rear side of the pyramid at different height from the bottom wall

The velocity profile at the rear side of the pyramid in the downstream of the flow at different height from the bottom wall has been shown in the Fig. 13. In the present investigation we have considered a square pyramid with apex angle of 60° . The velocity profile is considered at different distance i.e. $y/h = 0.2, 0.3, 0.4$ and 0.5 from the base of the pyramid. It is seen from the figure negative velocity in near region of pyramid due to back flow. The velocity is zero on the wall of the pyramid which can be clearly visible at $x/h = 6$ and then it becoming more and more negative up to $x/h = 7$, where the maximum reversal flow occurs and then it becomes zero again when x/h is around 8. Then the velocity becoming positive which signifies no back flow occurs after $x/h = 8$. The plot also shows clearly the free stream velocity is achieved at a distance of $x/h = 12$ along the downstream of the flow. It is also marked from the plot at $y/h = 0.2$, i.e. towards the bottom wall the back flow is becoming more intense and when we move towards the top of the pyramid i.e. at $y/h = 0.2$ to 0.5 , the reversed flow decreases. This shows clearly that the back flow velocity is more towards the bottom of the pyramid compared towards the top of the pyramid.

H. Velocity profile along the downstream of the flow at the rear side of the pyramid at different apex angle

The velocity profile along the downstream of the flow at a constant height of $Y/h = 0.2$ from the bottom of the domain for different apex angle has been depicted in the Fig. 14. Keeping other parameter fixed we are changing the apex angle from 30° to 120° . It can be seen from the plot the back flow velocity is less for 30° apex angle compared to 60° and 90° case. It is also seen from the plot for smaller apex angle pyramid back flow occurs very close to the rear side of the pyramid where as it is farther for the case of larger (90°) apex angle pyramid. So it is prominent the vertex size is largest for 90° apex angle. It is also evident from the plot there is no back flow occurs in case of 120° apex angle. So from the figure it can be concluded the recommended value of apex angle in case of square base pyramid for maximum mixing is around from 60° to 90° .

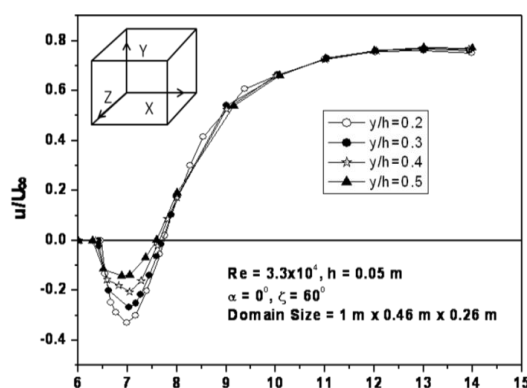


Fig. 13 Velocity profile along the downstream of the flow at different distance from the bottom of the

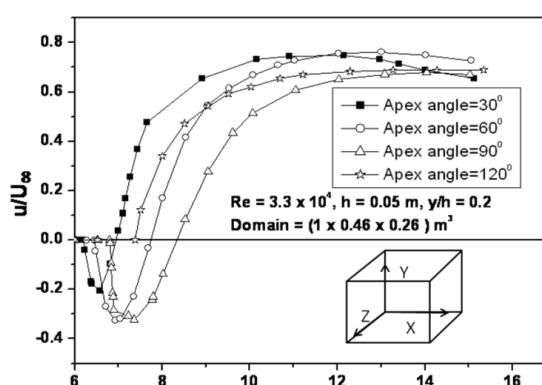


Fig 14 Velocity profile along the downstream of the flow at distance $y/h = 0.2$ from the bottom of the domain for different apex angle

I. Velocity profile vertical to the flow direction at different apex angle at the rear side of the pyramid

The velocity profile along the direction of flow has been computed for different apex angle ($30^\circ, 60^\circ, 90^\circ$ and 120°) at a distance of 0.05 m ($X/h = 1, Z = 0.23$ m) from the apex and at the rear side of the pyramid. It can be seen (Fig. 15) that highest back flow occurs for the pyramid having apex angle 60° and 90° . In case of smaller apex angle pyramid (30° for the present case) backflow velocity is almost negligible where as it is zero in case of larger apex angle pyramid (i.e. around 120°). In fact for smaller apex angle the size of the rotor vertex is small and it is also formed very close to the pyramid where as for larger apex angle pyramids the rotor vertex is not formed at all.

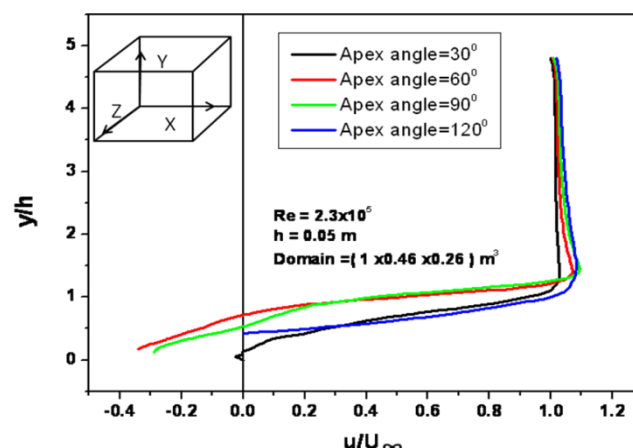


Fig. 15 Velocity profile vertical to flow direction at a distance $x/h = 1$ from the apex downstream of the pyramid for different apex angle

V. Conclusions

Flow around surface mounted bluff body has been numerically investigated. The conservation equations of mass and momentum has been solved along with two equation based $k-\epsilon$ model to determine turbulent intensity, reattachment distance and velocity profiles by changing different pertinent parameters. The computed velocity fields have been validated with the existing experimental results of Martinuzzi and Abuorma [12]. For a square based pyramid there exists optimum apex angle for maximum turbulent intensity and for the present case the optimum apex angle is found to be 90° . Keeping other parameter fixed it is also found that there also exists optimum attack angle for maximum turbulent intensity, from the present CFD analysis that value of attack angle is found to be 15° . Reattachment distance increases from smaller pyramid to larger pyramid. Pyramid having apex angle 90° is found to have highest reattachment distance and the maximum value of reattachment distance is obtained at attack angle 20° . It was also found that for same height and volume of bluff bodies the reattachment distance is highest for square based prism and lowest for cone. At the rear side of the pyramid the back flow intensity is highest towards the bottom wall of the channel and in the longitudinal middle line passing through the pyramid and it has been found that back flow velocity is highest for 60° and 90° apex angle pyramid.

References

- [1] D. Jeff Eldredge, "Numerical Simulation of The Fluid Dynamics Of 2D Rigid Body Motion With The Vortex Particle Method" Journal of Computational Physics, Vol. 221, 2006, pp. 626–648.
- [2] A. L. F. Silva, A. R. Silva, and A. S. Neto, "Numerical Simulation of Two-Dimensional Complex Flows around Bluff Bodies Using the Immersed Boundary Method", J. of the Braz. Soc. of Mech. Sci. & Eng., Vol. 29 (4), 2007, pp. 379-387.
- [3] R. Mittal, and S. Balachander, "Direct Numerical Simulation of Flow Past Elliptic Cylinders", Journal of Computational Physics, Vol. 124, 1996, pp. 351–367.
- [4] S. Thete, K. Bhat and M. R. Nandgaonkar, "2D Numerical Simulation of Fluid Flow over a Rectangular Prism", CFD Letters, Vol. 1, 2009, pp. 1-17.
- [5] I. P. Castro and A. G. Robins, "The flow around a surface-mounted cube in uniform and turbulent streams", Journal of Fluid Mechanics, Vol. 79 (02), 1977, pp. 307 – 335.
- [6] T. Igarashi, "Heat transfer from a square prism to an air stream", International Journal of Heat and Mass Transfer, Vol. 28 (1), 1985, pp. 175-181.
- [7] J. H. Agui and J. Andreopoulos, "Experimental investigation of a three-dimensional boundary layer flow in the vicinity of an upright wall mounted cylinder", Journal of Fluids Engineering, Vol. 114 (4), 1985, pp. 566-576.
- [8] H. W. Tieleman, D. Surry and J. X. Lin, "Characteristics of mean and fluctuating pressure coefficients under corner (delta wing) vortices", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 52, 1994, pp. 263-275.
- [9] A. H. Techet, F. S. Hover and M. S. Triantafyllou, "Vortical patterns behind a tapered cylinder oscillating transversely to a uniform flow", J. Fluid Mech., Vol. 363, 1998, pp. 79-96.
- [10] R. J. Martinuzzi and B. Havel, "Turbulent Flow Around Two Interfering Surface-Mounted Cubic Obstacles in Tandem Arrangement", Journal of Fluids Engineering, Vol. 122 (1), 2000, pp 24-31.
- [11] I. Castro and P. Rogers, "Vortex shedding from tapered plates, Experiments in Fluids", Vol. 33 (1), 2002, pp. 66-74.
- [12] R. J. Martinuzzi and M. Abuorma, "Study of the flow around surface-mounted pyramids", Experiments in Fluids, Vol. 34 (3), 2003, pp. 379-389.
- [13] M. Abu Omar and R. J. Martinuzzi, "Experimental Study Of The Pressure Field And Flow Structures Around Surface -Mounted Pyramids", 16th ASCE Engineering Mechanics Conference, University of Washington, USA, 2003.
- [14] M. M. AbuOmar and R. J. Martinuzzi, "Vortical structures around a surface-mounted pyramid in a thin boundary layer", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96 (6-7), 2008, pp. 769-778.
- [15] M. K. Chyu and V. Natarajan, "Heat Transfer of the base surface on the three dimensional protruding element" Int. Journal of Heat Mass Transfer, Vol. 39 (14), 1996, pp. 2925-2935.
- [16] M. Yaghoubi and E. Velayati, "Undeveloped convective heat transfer from an array of cubes in cross-stream direction", International Journal of Thermal Sciences, 44 (8), 2005, pp. 756-765.
- [17] M. Farhadi and M. Rahnama, "Large Eddy Simulation of Separated Flow over a Wall-Mounted Cube", Scientia Iranica, Vol. 13 (2), 2006, pp. 124-133.
- [18] S. K. Dash, "Heatline visualization in turbulent flow", Int. J. Numerical Methods for Heat and Fluid Flow, 6 (4), 1996, pp. 37-46.

- [19] P. K. Jha and S. K. Dash, "Effect of outlet positions and various turbulence models on mixing in a single and multi strand tundish", *Int. J. Num. Method for Heat and Fluid Flow*, Vol. 12 (5), 2002, pp. 560-584.
- [20] P. K. Jha and S. K. Dash "Employment of different turbulence models to the design of optimum steel flows in a tundish", *Int. J. Numerical Methods for Heat and Fluid Flow*, Vol. 14 (8), 2004, pp. 953-979.
- [21] P. K. Jha, R. Ranjan, S. S. Mondal and S. K. Dash, "Mixing in a tundish and a choice of turbulence model for its prediction", *Int. J. Numerical Methods for Heat and Fluid Flow*, 13 (8), 2003, 964-996.