Abstract—A computational study supported by experimental analysis is conducted on a square cylinder to curtail the drag force acting on the square cylinder. Upstream rods of different shapes and sizes (keeping the cross-sectional area same) are used to accomplish the task by varying distance between the upstream rod and the square cylinder. Two flow modes are observed while varying the distances: Cavity flow and wake impingement mode. A maximum of 78.8% of drag coefficient is reduced using circular rod, while 84.1% reduction is reported using square rods. The triangular geometry showed an abnormal behaviour where drag coefficient was unexpectedly negative at certain L/D signifying the opposite nature of the drag force.

Keywords—Drag reduction, cavity flow mode (CFM), wake impingement mode (WIM), turbulence model.

I. INTRODUCTION

LUFF body aerodynamics has been of keen interest to the researchers pertaining to the importance of drag reduction on such objects. Automobile, aircrafts, naval and architectural structures, such as bridge decks, ship decks and monuments etc. are practical examples where reduction of drag is important either to improve their performance or even to an extent of avoiding damage due to wind loading. Most of these objects roughly have either circular or rectangular cross-section. Hence flow past cylinders remains an area of research in past as well as at present. Roshko [1] conducted experiments on a circular cylinder at high Reynolds number for reporting drag. Experiments were performed on a pressurized wind tunnel at Re=1.0×10^6 to 1.0×10^7, it was reported that the drag coefficient increases from its low supercritical value to a value 0.7 at Re = 3.5×10^6 and then becomes constant. Bearman [2] carried out experiments to diminish the steady drag by monitoring flow on a rectangular cylinder by fitting a splitter plate onto the rear of the rectangular cylinder.

Igarashi [3] observed two flow modes characterized by the behavior of the upstream rod wake. First one is defined as the cavity flow mode (CFM) and second is wake impinging mode (WIM). In the study, an upstream rod is placed ahead of a square cylinder. As the spacing between the rod and the square cylinder (L) is varied, two major flow modes, characterized by the behavior of the rod wake, are observed. In CFM, the gap between the rod and cylinder is small and there is no vortex shedding from the rod.

In WIM, both the rod and the square cylinder shed vortices for a large spacing ratio and the wake behind the rod impinges on the front side of the square cylinder. In the junction of the two modes, the value of C_D changes abruptly, and is known as 'jump Phenomenon'. The spacing, where the jump phenomenon appears, is defined as critical length L_c.

Sakamoto and Haniu [4] studied the suppression of the fluid forces acting on a cylinder adding a control rod to the system. They remarked that the time-averaged mean drag force could be reduced until approximately to 50%, and that the drag forces could be reduced up to 85% by using a control rod. Prasad and Williamson [5] experimentally investigated the drag of the system consisting a circular cylinder and a flat plate upstream of the cylinder in tandem arrangement. An optimal geometrical configuration in order of one-third of the side of square cylinder (D) is placed at 1.5D upstream of the cylinder, which produced 62% reduction in a total system drag in comparison of a bare cylinder (i.e. cylinder without any upstream rod). Zhang and Wang [6] demonstrated aerodynamic characteristics of square cylinder with an upstream rod in a wind tunnel at Re = 82000. It is observed that use of upstream rods has shown a considerable reduction in C_D.

It is revealed from literature that the studies carried out in the past usually used lower sub-critical range of Re. Moreover, the studies with upstream rods a have been with two similar shape cylinders in tandem, the present study tries to investigate the effect of shape change in the upstream rods as well, at high Reynolds number in view to encompass all left outs in the literature regarding flow control through upstream rods.

II. EXPERIMENTAL METHODOLOGY

Experiments are carried out in an open-mouth flow generation set-up at a free-stream velocity (U∞) of 15 m/s, which corresponds to Re=1×10^5 for the upstream circular rod to understand the first-hand behavior of upstream rod on C_p and C_D over a square cylinder. Upstream rods of different shapes are used to reduce the overall drag over a square cylinder shown in Fig 1.
Fig 1 Arrangement of upstream rod and cylinder

The cylinder with a square cross-section of 10×10cm had 5 pressure tapings attached on each face for measuring wall static pressure with the help of a digital micro manometer and a pressure scanner (Make: Furness Control, UK). Experiments are done for various L/D (0.9 to 3.6) and d/D (0.1 to 0.5) ratios and their effects on drag have been studied. Pressure coefficient ($C_p$) is calculated at each location, whereas, drag coefficient ($C_D$) is calculated for each set of experiment. Detailed parametric study is conducted for the upstream rod in tandem with the square cylinder.

III. COMPUTATIONAL METHODOLOGY

Finite-volume technique based commercially available computational fluid dynamics (CFD) code ‘Fluent 6.3’ is used to carry out the flow simulation. The computational scheme mainly comprises of computational domain, grid generation, mathematical formulation and turbulence models, boundary conditions and solution scheme. The computational results are validated against the experimental data as shown in Fig.2. Further simulations are carried out with the upstream rod having various shapes like elliptical, square, and triangular shape, as shown in Fig.3.

A. Computational domain

The computational domain shaping as parallelepiped of dimensions 244 cm×110 cm×129 cm is shown in Fig 4. Other conditions remain same as maintained during experimentation. A Cylinder of size as shown in Fig. 5 is placed 39 cm downstream of from the inlet. To capture rear wake zone effectively, a larger domain is provided behind the cylinder.

Fig. 2. Variation of $C_p$ along the circumferential length for cylinder without an upstream rod

B. Grid generation

Commercial software Gambit 2.3.62 is used for grid generation in the computational domain. In order to obtain a major part of the computational domain as a structured mesh, hexahedron meshing is done.

Fig 6. Meshing in the computational domain

Grid independency checking

Any CFD solution heavily depends on the size and fitness of meshing. Therefore, care must be taken in selecting the grid types (coarse, medium or fine) such that the grid does not dictates the solution leading to erroneous results. In the present case, the computational domain with bare cylinder is meshed with different grid sizes and computed results are shown in Table 1. Mesh size 3 is found to be optimal in regard to economy and result independency and is therefore adopted as default mesh size for the further analysis. The meshed domain is shown in fig.6.
and transport equations are solved for two turbulence quantities to the class of two-equation models, in which modelled equations are required, namely the transport equations for \(k\) and \(\varepsilon\). In order to obtain closure solution, the two other modelled equations are required, namely the transport equations for \(k\) and \(\varepsilon\).

### Table 1. Checking of Mesh Independence for Bare Cylinder

<table>
<thead>
<tr>
<th>Mesh type</th>
<th>No. of volume cell</th>
<th>Total CPU time</th>
<th>(C_D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>930560</td>
<td>1 hr 20 min</td>
<td>1.037</td>
</tr>
<tr>
<td>2</td>
<td>1054815</td>
<td>2 hr 10 min</td>
<td>1.035</td>
</tr>
<tr>
<td>3</td>
<td>1128300</td>
<td>2 hr 35 min</td>
<td>1.040</td>
</tr>
<tr>
<td>4</td>
<td>1140918</td>
<td>3 hr 15 min</td>
<td>1.040</td>
</tr>
</tbody>
</table>

### C. Mathematical formulation

The general transport equations in tensor notation are written below.

**Continuity equation:** Flow is assumed to be steady and incompressible. Therefore,

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_j} = 0 \tag{1}
\]

**Momentum equations:**

\[
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial}{\partial x_j}\left[\mu \frac{\partial u_i}{\partial x_j}\right] - \frac{\partial p}{\partial x_j} \tag{2}
\]

### D. Turbulence Model

The proper selection of turbulence model for any flow computation using CFD demands rapt attention. The renormalized group (RNG) \(k-\varepsilon\) turbulence model [7] belongs to the class of two-equation models, in which modelled transport equations are solved for two turbulence quantities \(k\) and \(\varepsilon\). In order to obtain solution closure, the two other modelled equations are required, namely the transport equations for \(k\) and \(\varepsilon\).

**\(k\)-equation:**

\[
\frac{\partial}{\partial x_i}\left[\rho \frac{\partial k}{\partial x_i}\right] = \frac{1}{\sigma_k} \left[ \frac{\partial}{\partial x_i}\left( \mu \frac{\partial k}{\partial x_i} \right) \right] - \frac{\partial}{\partial x_j}\left( \frac{\mu}{\sigma_k} \frac{\partial u_i}{\partial x_j} \right) + \frac{\partial}{\partial x_j}\left( \frac{\rho}{\sigma_k} \frac{\partial u_i}{\partial x_j} \right) + \frac{\partial}{\partial x_i} \left( \frac{\partial \rho u_j}{\partial x_i} \right) \tag{3}
\]

**\(\varepsilon\)-equation:**

\[
\frac{\partial}{\partial x_i}\left( \rho \frac{\partial \varepsilon}{\partial x_i} \right) = \frac{1}{\sigma_\varepsilon} \left[ \frac{\partial}{\partial x_i}\left( \mu \frac{\partial \varepsilon}{\partial x_i} \right) \right] + C_{\varepsilon 1} \frac{\varepsilon}{k} \left( \frac{\rho}{\sigma_k} \frac{\partial u_i}{\partial x_i} \right) + C_{\varepsilon 2} \frac{\varepsilon^2}{k} \tag{4}
\]

### E. Boundary conditions

Boundary conditions used is similar to the experimental conditions. Same inlet velocity used for all cases which were found experimentally. Velocity inlet is taken as inlet boundary condition and zero gauge pressure is used as exit condition. To indicate the turbulence quantities at the inlet, like turbulent kinetic energy \((k)\) and turbulence dissipation rate \((\varepsilon)\), the following relation is used:

\[
k = 1.5 \left( \frac{U_{ave} I}{\lambda} \right)^2 \tag{5}
\]

\[
\varepsilon = \left( \frac{C_s}{k} \right)^{3/4} \left( \frac{k}{L} \right)^{3/2} \tag{6}
\]

Where, \(L\) = turbulence length scale = 0.07\(L_c\),

\[
I = 0.16 \left( \frac{Re}{18} \right)^{1/8} \tag{7}
\]

No-slip boundary condition is defined at the walls. Near-wall boundary condition is done using enhanced wall treatment method to account for the boundary layers formed during grid generation. Enhanced wall treatment combines two-layer model with enhanced wall functions to resolve the laminar sub-layer.

### F. Solution scheme

The segregated and implicit solver is selected for discretization of governing equations. The SIMPLE algorithm [8] is used for the pressure-velocity coupling. STANDARD scheme is selected for discretizing pressure, whereas, bounded central differencing numerical scheme is chosen for discretizing momentum equations. Convergence criterion for continuity and velocity is taken as \(10^{-4}\). The computational work has been performed on an IBM workstation with 8 GB RAM and Intel Xeon processor.

### IV. RESULTS AND DISCUSSION

#### A. Experimental Study

The experimental results with different upstream rod are analyzed for different \(L/D\) (1.8 to 3.6) and \(d/D\) (0.1, 0.15 and 0.25). Variation of \(C_D\) v/s \(L/D\) ratio for different \(d/D\) ratio is shown in Fig. 8 and variation of coefficient of pressure along the circumference of the square cylinder with \(d/D\) ratio 0.1 and varying \(L/D\)’s is plotted as shown in Fig.7.

#### B. Computational Study

Validating the experimental results with the computational results is as shown in Fig. 2, further computations were performed with varying \(d/D\) and \(L/D\) to have a better understanding of the effect of these variations on drag. Fig. 9-10 shows coefficient of pressure plotted against circumferential length for different \(L/D\) with constant \(d/D\) = 0.1. Value of coefficient of pressure \((C_p)\) in front side of the cylinder is reduced when rod is placed on the upstream of the cylinder. Although the value of the pressure coefficient on the front side is still positive, it is much less than that of a bare cylinder. This can be explained by the low incoming flow velocity induced by the rod wake. In case of bare cylinder, the flow strikes the front of the cylinder creating higher coefficient of pressure and hence higher drag. At low values of \(L/D\), the flow is characterized by cavity flow mode whereas for higher values of \(L/D\), wake impingement mode is observed as shown in Fig. 9-10 respectively. The figures clearly indicate that the pattern of \(C_p\) along the front face of the cylinder (till 100 mm on the X-axis of graph in fig. 9-10) is quite different as the flow switches from one mode to another (CFM in fig.9 and WIM in fig 10). Moreover there is a change in the nature of the curve along the rear, top and bottom faces (from 100 mm and more on the X-axis of graph in Fig.9-10). The cavity flow mode as shown in figure 9 extends from \(L/D\) 0.9 to \(L/D\) within 1.5 to 1.8 where
critical length occurs and a change in the nature of the curve is observed from L/D 1.8 and above. As shown in figure 10(Note that the results quoted regarding critical length and two modes for different L/D are with d/D 0.1 and will change with any changes in d/D ratio.

The effect of increasing the size of the rod is reduction in coefficient of static pressure on front side of the square cylinder. \( C_p \) for variation in size for fixed L/D = 1.5 shown in Fig. 9, L/D = 1.5 (cavity flow mode) is the case of minimum drag taking by the previous variation in L/d.

From Fig. 11 it is clear that value of coefficient of pressure decreased with increasing d/D at upward side of the cylinder. With increasing size of the rod static pressure at the front face of the cylinder gets reduced. For the case maximum reduction drag i.e. \( d/D=0.5 \) variation of L/D as clearly given in table 2.

<table>
<thead>
<tr>
<th>d/D</th>
<th>( C_D )</th>
<th>( C_D ) reduction (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>0.479</td>
<td>54.03</td>
</tr>
<tr>
<td>0.15</td>
<td>0.484</td>
<td>53.5</td>
</tr>
<tr>
<td>0.2</td>
<td>0.407</td>
<td>60</td>
</tr>
<tr>
<td>0.3</td>
<td>0.555</td>
<td>48.6</td>
</tr>
<tr>
<td>0.5</td>
<td>0.220</td>
<td>78.8 (max.)</td>
</tr>
</tbody>
</table>

To study the effect of using different cross-sectional shape upstream rods on the drag was the major objective in the present study and for that purpose circular, elliptical, square and triangular shape rods have been taken into consideration. Value of drag coefficients for L/D = 1.2 & 1.5 are shown in table 3 with various rod shapes. It is clear that minimum positive value of coefficient of drag is for square rod. Maximum drag reduction is 83.2% for square rod. Minimum drag reduction found in case of elliptical shaped rod. And an unexpectedly abrupt variation is recorded for the case of a triangular rod that a negative value of drag coefficient is found which indicates force is not in the direction of flow but opposite to it.

Ultimately the maximum drag reduction was observed as 84.1% in case of square rod upstream the cylinder, which is
the highest drag reduction for present study in terms of positive values of drag coefficient. By the velocity contours shown in Fig. 12 it is clear that when a square rod is placed on upstream then minimum positive drag is found, due to more frontal area and thus more shielding effect.

Drag coefficients \( C_D \) are calculated for various combinations of \( L/D \) and \( d/D \) ratios and also for various shapes of upstream rod. A complete flow diagnosis is performed to understand the flow physics. The results obtained from the experiments and the simulations are found to be in agreement (Fig.2). Some salient points of results are given here:

- **Variation in \( L/D \) for the smallest rod gives the maximum drag reduction of 54% at \( L/D = 1.5 \).** Drag coefficient \( C_D \) decreases as the rod moves from \( L/D = 0.9 \) to \(~ (1.5-1.8)\), and then increases from \( L/D = 1.8 \) to 3.6. It is evident from the flow simulation that the square cylinder falls within the rear wake region of the rod \( L/D = 0.9 \) to 1.5, and is called 'cavity flow mode'.

- **As \( L/D \) ratio increases, the cavity flow mode shifts to 'wake impingement mode', where vortices shedding from the control rod impinges on the pressure side of the cylinder.** This reduces the static pressure on the frontal side but experiences decrease in static pressure on the suction side.

- **Variation in \( d/D \) for the optimum \( L/D \) gives maximum drag reduction of 78.8% for \( d/D = 0.5 \).** Drag coefficient decreases with the increase of rod diameter.

- **Table 2** shows result of different cross-sectional shapes of upstream rod for a particular case at \( L/D =1.2 \) & 1.5 and \( d/D = 0.5 \). It gives maximum drag reduction of 84.1% for square rod.

- **In case of triangular shaped rod, \( C_D \) is found negative \((-0.014)\).** Negative value of drag means that direction of drag is opposite to the flow on cylinder. The separated shear layer from the front edges of the upstream cylinder reattached on the lateral surface of the downstream cylinder, forming two recirculation regions between the two cylinders, the flow reattachment of the separated shear layer on the downstream cylinder exposed the front face of that cylinder to a strong suction that created a negative drag coefficient.

**TABLE 3. VALUE OF \( C_D \) FOR DIFFERENT SHAPES OF ROD WITH \( L/D = 1.5 \) AND \( d/D = 0.5 \)**

<table>
<thead>
<tr>
<th>Shape of rod</th>
<th>( C_D(L/D=1.2) )</th>
<th>Reduction (%)</th>
<th>( C_D(L/D=1.5) )</th>
<th>Reduction (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular</td>
<td>0.199</td>
<td>80.8</td>
<td>0.22</td>
<td>78.8</td>
</tr>
<tr>
<td>Elliptical</td>
<td>0.518</td>
<td>50.1</td>
<td>0.517</td>
<td>50.2</td>
</tr>
<tr>
<td>Square</td>
<td>0.165</td>
<td>84.1 (max.)</td>
<td>0.174</td>
<td>83.2 (max.)</td>
</tr>
<tr>
<td>Triangular</td>
<td>-0.014</td>
<td>NA</td>
<td>-0.085</td>
<td>NA</td>
</tr>
</tbody>
</table>

In case of triangular cross section of rod coefficient of drag \( C_D = -0.014 \) found. Negative value of drag means that direction of drag is opposite to flow on cylinder. This is due to separated shear layer from the front edges of the upstream cylinder reattached on the lateral surface of the downstream cylinder, forming two recirculation regions between the two cylinders, the flow reattachment of the separated shear layer on the downstream cylinder exposed the front face of that cylinder to a strong suction as shown in Fig. 13 that created a negative drag coefficient.

**V. CONCLUSIONS**

Thanks are due to Department of Science & Technology (DST), Govt. of India for providing DST-FIST grant to create computational facility to accomplish this research.

**REFERENCES**


