Frequency of shedding vortices in the wake of a bluff body is used to measure the flow rate in vortex flowmeters. A smooth bluff body shape may not generate stable vortices and needs sharp edges for the consistency of pulsed output. However, such bluff bodies give higher-pressure loss and drag force and therefore a need exists for optimising the body shape. In the present work, the flow over various body shapes in a pipe have been analysed using computational fluid dynamics to study the effect of body shape on pressure loss and drag coefficient. It is observed that the body shape with sharp corners have higher-pressure loss and drag compared to the cylindrical one. However, a triangular body shape with splitter plate or diamond shape has low coefficients of permanent pressure loss and drags compared to the other sharp cornered bodies and hence are better suited for use in the vortex flowmeters.

Vortex flowmeters are a good choice for metering devices to work under hazardous atmosphere in the case of the clean liquids and gases. These devices have high precision, low maintenance requirement and their output is independent of the properties of construction material. These devices work on the principle of measurement of vortex shedding frequency in the wake of a bluff body assuming that Strouhal number remains constant over the operating range of Reynolds numbers. The Strouhal number is expressed as:

\[ St = \frac{f d}{u} \]  

(1)

where \( f \) is frequency of the shedding (Hz), \( d \) is width of bluff body (m) and \( u \) is average velocity of flow (ms\(^{-1}\)).

Thus for a given bluff body, the vortex shedding frequency is directly proportional to the average velocity and the flow rate through the pipe can be determined by:

\[ Q = \frac{Au}{St} = \frac{f}{K} \]  

(2)

where \( A \) is cross-sectional area of the meter (m\(^2\)) = \( \frac{\pi}{4} D^2 \), \( D \) is Pipe diameter (m) and \( K \) is meter constant (pulses/flow volume).

The phenomenon of vortex formation and shedding in the flow past a circular cylinder has been studied by many investigators. It has been observed that such flows generally have three types of flow instabilities namely, boundary layer instability, separated shear layer instability and Karman vortex instability. However, the existence of sharp edges on a bluff body improves both the accuracy and regularity of the vortex-shedding phenomenon. Singh et al. have experimentally observed that the size and shape of a bluff body play a significant role on the performance of the flow meter and there exists a need to optimise the bluff body configuration. According to them, a body shape with sharp corners helps in clear shedding of vortices resulting in high turndown ratio. Different shapes of bluff bodies have been evolved to optimise the performance. Experimental investigations by Thomson et al. have shown that the Strouhal number is dependent on Reynolds number and also increases with increase in the body width (\( d \)). At high Reynolds numbers, aerodynamic shapes can be preferred due to minimum blockage and less variations in Strouhal number. Brennan and Lomas reported that the \( K \)-factor remains same for all liquids, gases and cryogenic fluids in a vortex flowmeter. Bearman found that the splitter plate helps in stabilizing the wake and reduces the drag on two-dimensional bluff bodies. May experimentally measured the drag force on the bluff bodies having circular, semi-ellipsoid and square cross-sections with and without splitter blades/end.

*For correspondence: (e-mail: sidnathsingh@hotmail.com)
discs. He found that the drag coefficient of a bluff body can be reduced by around 20% by using a splitter plate, 33% by using end disc and 38.5% by using the combination of the two. Numerical analysis of a vortex flowmeter has been carried out by Motosmaga et al. who reported reasonable agreement of the numerical predictions with the experimental observations of the flow field. They predicted the dependence of Strouhal number on Reynolds number over a wide range of flow varying from laminar to turbulent regimes. According to them, this method can also be used to determine the optimum shape of the bluff body.

In the present work, the flow around various bluff body shapes has been studied using FLUENT, a commercial computational fluid dynamics code (CFD). Analysis has been carried out to investigate the variation of permanent pressure loss coefficient and drag coefficient with Reynolds number for various body shapes and sizes. The effect of addition of splitter plate in rectangular and triangular shapes has also been studied and the results are compared to determine the optimum body configuration. Steady state analysis has been carried out to optimise the shape. Calculation of Strouhal number requires determination of shedding frequencies, which is only feasible if unsteady flow analysis is carried out. Hence, Strouhal number has not been evaluated in the present work.

**Computational Procedure**

The commercial CFD code - FLUENT, which is the state of art software package, has been used for analysing fluid flow and heat transfer problems involving complex geometries. The numerical scheme employed belongs to a finite volume group and adopts integral form of the conservation equations. The solution domain is subdivided into a finite number of contiguous control volumes and conservation equations are applied to each control volume. Surface and volume integrals are approximated using suitable quadrature formulae.

The following conservation equations for mass and momentum have been used to solve the steady flow problem. The equation for conservation of mass can be written as,

\[
\frac{\partial}{\partial x_i} (\rho u_i) = S_m
\]

where \(\rho\) is mass density (kgm\(^{-3}\)), \(t\) is time (s), \(x_i\) is longitude co-ordinate in \(i^{th}\) direction (m), \(u_i\) is mean longitudinal velocity in \(i^{th}\) direction (ms\(^{-1}\)) and \(S_m\) is mass added to continuous phase (kg).

The above equation is valid for incompressible as well as compressible flows. The source \(S_m\) is the mass added to the continuous phase from the dispersed second phase (e.g. due to vapourization of liquid droplets) or by any other user-defined source. Conservation of momentum for the \(i^{th}\) direction in an inertial reference frame is written as,

\[
\frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} (\tau_{ij} + \rho g_i) + F_i \quad \ldots (4)
\]

where \(x_j\) is longitude co-ordinate in \(j^{th}\) direction (m), \(u_j\) is mean longitudinal velocity in \(j^{th}\) direction (ms\(^{-1}\)), \(p\) is static pressure (Nm\(^{-2}\)), \(\rho g_i\) is body force per unit volume in \(i^{th}\) direction (Nm\(^{-3}\)), \(\tau_{ij}\) is shear stress tensor (Nm\(^{-2}\)) and \(F_i\) is body force (N).

The shear stress tensor \(\tau_{ij}\) is given by,

\[
\tau_{ij} = \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_i}{\partial x_i} \right] \delta_{ij} + (\tau_{ij}) \quad \ldots (5)
\]

where \(\mu\) is effective dynamic viscosity (Nm\(^{-2}\)), \(\delta_{ij}\) is Kronecker delta function and \((\tau_{ij})\) is Reynolds stress tensor (Nm\(^{-2}\)).

The second term on the right hand side is the effect of the volume dilatation. For calculating the pressure, the method is derived from the SIMPLE algorithm of Caretto et al. The geometry taken for the CFD analysis configures a bluff body placed in the middle of 2 m long pipe of 50 mm diameter as shown in Fig. 1. The volume of the geometry was meshed by Hex-Cooper/Tet-primitive meshing schemes. Fine grids for region close to bluff body were chosen and coarse grids were adopted for the remaining volume to optimize the computer time. Different body shapes analyzed in the present work are shown in Fig. 2 and their dimensions are listed in Table 1. Air is taken as the fluid and the flow is assumed to be steady and three-dimensional. The total numbers of elements.
selected were always more than $10^5$ for different shapes. Velocity at the pipe inlet and pressure at the pipe outlet were specified as the boundary conditions. Flow is assumed to be turbulent and the velocity profile at the inlet is assumed to follow one-seventh power law distribution.

Turbulence model used

Small high frequency fluctuations are present even in a steady flow and to account for these, time averaging procedure is employed which results in additional terms. These additional terms need to be expressed as calculable quantities for closure solution. The standard $k$-$\varepsilon$ model has been used in the present case. It is a semi-empirical model based on model transport equations for the turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$.

\[
\frac{\partial \rho u_i}{\partial x_i} \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_k} \right] \frac{\partial k}{\partial x_i} + G_k + G_b - \rho \varepsilon - Y_m \quad \cdots (6)
\]

\[
\frac{\partial \rho u_i}{\partial x_i} \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_\varepsilon} \right] \frac{\partial \varepsilon}{\partial x_i} + C_{1\varepsilon} \varepsilon \left( \frac{\partial u_i}{\partial x_i} \right)^2 - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad \cdots (7)
\]

where $k$ is turbulent kinetic energy (Nm), $\mu_t$ is eddy/turbulent viscosity (Nsm$^{-2}$), $\sigma_k$ is turbulent Prandtl number for $k$, $\sigma_\varepsilon$ is turbulent Prandtl number for $\varepsilon$, $G_k$ is generation term (buoyancy), $G_b$ is generation term (kinetic), $\varepsilon$ is turbulence dissipation rate (Nm$^2$kg$^{-1}$), $Y_m$ is fluctuating dilation and $C_{1\varepsilon}$, $C_2\varepsilon$ and $C_3\varepsilon$ are turbulence model constants.

Here, $G_k$ represents the generation of turbulent kinetic energy due to the mean velocity gradients calculated as,

\[
G_k = \mu_t S^2 \quad \cdots (8)
\]

where $S$ represents the modulus of mean rate of stress and is calculated as,

\[
S = \sqrt{2S_xS_y} \quad \cdots (9)
\]

The mean stress rate, $S_y$ is given by,

\[
S_y = \frac{1}{2} \left[ \frac{\partial u_y}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right] \quad \cdots (10)
\]

and $G_b$ represents generation of turbulent kinetic energy due to buoyancy which has been taken as zero as there are no temperature gradients. The eddy or turbulent viscosity, $\mu_t$ is computed using the equation given as,

Table 1—Dimensions of different bluff bodies

<table>
<thead>
<tr>
<th>S.No.</th>
<th>Title</th>
<th>Type of Shape</th>
<th>Dimensions in mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>B1</td>
<td>Rectangular</td>
<td>$d = 8$ and $b = 4$</td>
</tr>
<tr>
<td>2</td>
<td>B2</td>
<td>Square</td>
<td>$d = 8$ and $b = 8$</td>
</tr>
<tr>
<td>3</td>
<td>B3</td>
<td>Rectangular</td>
<td>$d = 8$ and $b = 12$</td>
</tr>
<tr>
<td>4</td>
<td>B4</td>
<td>Triangular</td>
<td>$d = 8$ and $\beta = 90^\circ$</td>
</tr>
<tr>
<td>5</td>
<td>B5</td>
<td>Triangular</td>
<td>$d = 8$ and $\beta = 60^\circ$</td>
</tr>
<tr>
<td>6</td>
<td>B6</td>
<td>Triangular with splitter plate</td>
<td>$d = 8$ and $\beta = 60^\circ$; splitter plate length = 3 and width = 3</td>
</tr>
<tr>
<td>7</td>
<td>B7</td>
<td>Triangular</td>
<td>$d = 8$ and $\beta = 45^\circ$</td>
</tr>
<tr>
<td>8</td>
<td>B8</td>
<td>Prismatic</td>
<td>$d = 8$ and $\beta = 60^\circ$</td>
</tr>
<tr>
<td>9</td>
<td>B9</td>
<td>Circular</td>
<td>$d = 8$ and $r = 4$</td>
</tr>
<tr>
<td>10</td>
<td>B10</td>
<td>Circular with splitter plate</td>
<td>$d = 8$ and $r = 10$, splitter plate length = 11 and width = 2.1</td>
</tr>
<tr>
<td>11</td>
<td>B11</td>
<td>Rectangular</td>
<td>$d = 20$ and $b = 10$</td>
</tr>
<tr>
<td>12</td>
<td>B12</td>
<td>Rectangular with splitter plate</td>
<td>$d = 20$ and $b = 4$, splitter plate length = 10 and width = 4</td>
</tr>
</tbody>
</table>
\[ \mu_i = \rho C_u \frac{k^3}{\varepsilon} \] … (11)

The values of model constants used are taken as \( C_1 = 1.44, C_2 = 1.92, C_3 = 0.09, \sigma_k = 1.0 \) and \( \sigma_\varepsilon = 1.3 \). As \( G_b \) has been taken zero, the value of \( C_3 \) is not needed.

**Validation of CFD code**

The flow past the bluff body has been analysed for different shapes. A typical variation of velocity vectors and static pressure contours for flow around a circular cylinder at a Reynolds number of \( 1 \times 10^5 \) are shown in Figs 3a and 3b respectively. These flow patterns agree reasonably well with those available in standard reference19. Based on the flow analysis, the coefficient of drag for cylinders of circular and square cross-section have been evaluated as 0.22 and 0.14, which are also in reasonable agreement with the reported values for these body shapes20. After validation of the code, the coefficients for permanent pressure loss and drag for different geometries of bluff body have been evaluated in the range of Reynolds number from \( 1 \times 10^4 \) to \( 1 \times 10^5 \) using following relationships:

Permanent pressure loss coefficient,

\[ C = \frac{P_b - P_{ub}}{\frac{1}{2} \rho u^2} \] … (12)

and drag coefficient,

\[ C_D = \frac{Drag \ force}{\frac{1}{2} \rho u^2 A_p} \] … (13)

where \( P_b \) is static pressure difference between inlet and outlet in the presence of bluff body (Nm\(^2\)), \( P_{ub} \) is static pressure difference between inlet and outlet without bluff body (Nm\(^2\)) and \( A_p \) is projected area of bluff body (m\(^2\)).

Analysis has also been carried out by varying the width of the body to study the effect of blockage on the above parameters.

**Results and Discussion**

Using CFD code, computations have been carried out to evaluate permanent pressure loss coefficient, drag force and drag coefficient for the different geometries (see Fig. 2). Initially, frontal area of each body is kept same \( (d/D = 0.16) \) to facilitate comparison at a constant blockage. The effect of blockage has also been studied by taking larger rectangular body \( (d/D = 0.4) \) with and without splitter plate.

**Permanent pressure loss coefficient**

The variation of permanent pressure loss coefficient and drag coefficient with Reynolds number for different body shapes is shown in Figs 4 and 5 respectively. It is seen that the trend of variation is same for all the bodies though the magnitudes of the two coefficients are different. There is a decreasing trend in the permanent pressure loss coefficient in each case up to a Reynolds number of \( 2.5 \times 10^4 \) after which it becomes almost constant at higher Reynolds numbers. The drag coefficient for all shapes remains almost constant in the range of Reynolds numbers investigated. The nature of the
variation of the two coefficients for different geometries can be summarized as: (i) Permanent pressure loss coefficient and drag coefficient decreases for rectangular bodies (B1, B2, and B3) as the body length in the direction of flow increases. The decrease in average value of permanent pressure loss coefficient is from 0.54 to 0.46 as b/d increases from 0.5 to 1.5 and the corresponding decrease in coefficient of drag is from 2.36 to 1.95. (ii) Both the coefficients decrease for triangular bodies (B4, B5, and B7) as angle on the front face of the obstruction decreases. Average value of coefficient of permanent pressure loss decreases from 0.52 to 0.32 as angle decreases from 90° to 45° and the corresponding decrease in drag coefficient is from 2.26 to 1.58. (iii) For triangular shape (B5 and B6), addition of tail decreases both the coefficients. Flow pattern around these bodies shows that the addition of tail in triangular body causes smoother vortex lines as compared to that observed without tail and this results in the decrease of pressure loss and drag force. The average value of permanent pressure loss coefficient and drag coefficients are 0.41 and 1.94 without tail and 0.38 and 1.69 with tail respectively. (iv) For diamond shape body, the coefficients of permanent pressure loss and drag are 0.36 and 1.6 respectively. These values are lower than the corresponding rectangular and triangular shape bodies. (v) In cylindrical shape (B9 and B10), addition of tail increases the permanent pressure loss and drag force. This may be attributed to the increase in flow separation due to downstream disturbances, which generates more negative pressure behind the bluff body. The permanent pressure loss coefficient and drag coefficient are 0.17 and 0.74 without tail and 0.26 and 1.04 with tail respectively.

Figures 4 and 5 show that the coefficients of permanent pressure loss and drag are very low for cylindrical shape compared to other shapes, because in cylindrical shape, the width of the wake is less and streamlines remain attached to the body for longer distance.

Effect of blockage (d/D ratio)

The rectangular body [B-11] having b/d ratio as 0.5 and d/D ratio as 0.4 is taken for this study. The variation of permanent pressure loss coefficient and drag coefficient with Reynolds number for this body shape is presented graphically in Figs 6a and 6b respectively. It is seen that the permanent pressure loss coefficient and drag coefficient decreases with increase in blockage ratio. This is due to the fact that with increase in blockage ratio, the flow separation is reduced and hence the pressure loss and drag force are also reduced.

![Fig. 4: Variation of permanent pressure loss coefficients with Reynolds number for different bluff body shapes (d/D = 0.16).](image1)

![Fig. 5: Variation of drag coefficient with Reynolds number for different bluff bodies (d/D=0.16).](image2)

![Fig. 6a: Effect of blockage (d/D ratio) on permanent pressure loss coefficient of rectangular body.](image3)

![Fig. 6b: Effect of blockage (d/D ratio) on drag coefficient of rectangular body.](image4)
loss coefficient becomes constant after a Reynolds number of $2.5 \times 10^4$ (Fig. 6a) and is around as 3.03. The drag coefficients show a similar trend as the permanent pressure loss coefficient and have a value around 5.60 (Fig. 6b). The increase in the values of permanent pressure loss coefficient and drag coefficient as compared to the rectangular body of same $b/d$ ratio but having different blockage ($d/D$ ratio=0.16) can be explained on the basis of the wake formed. The wake width is 24 mm for $d/D$ ratio of 0.4, which is much higher than wake width formed (=8.72 mm) by the rectangular body of same $b/d$ and $d/D$ ratio of 0.16.

Effect of splitter plate

The effect of addition of splitter plate has been studied for three body shapes namely, triangular, rectangular and circular. The dimensions of splitter plate added to different shapes are as: (a) 8 mm long and 3 mm thick for 60° triangular shape. (b) 10 mm

Fig. 7—Effect of splitter plate on the wake formed downstream of various body shapes
long and 4 mm thick for rectangular shape \((d/D = 0.5\) and \(b/d = 0.5\)) and (c) 11 mm long and 2.1 mm thick for circular shape.

The flow fields around these bodies have been observed at Reynolds number of \(2.5 \times 10^5\). The variation of velocity vectors around these shapes with and without splitter plate is shown in Figs 7 (a-f). It is seen that the wake formed behind the body reduces with the addition of splitter plate in triangular and rectangular body shapes and this is the reason for reduction of coefficients of permanent pressure loss and drag for these bodies (see Figs 4-6). However, for circular shape body addition of splitter plate shows increase in wake region. This results in increase in the magnitude of the two coefficients for the cylinder with splitter plate as compared to those observed for cylinder without splitter plate.

Conclusions
The flow around various bluff body shapes inside a pipe has been analyzed using a CFD code. The coefficients of drag and permanent pressure loss have been evaluated for different shapes. It is seen that the cylindrical shape shows minimum coefficients for pressure loss and drag. But the vortex flowmeter requires a body shape with sharp corners to generate stable vortex shedding frequency. A triangular shape of 60° angle with splitter plate or a diamond shape body appear to be the optimum choice. These shapes have low coefficients of permanent pressure loss and drag among all sharp corner shapes studied in the present analysis. The values of coefficients of permanent pressure loss and drag for these bodies are marginally higher compared to the cylindrical body.

References