Application of CFD for assessment of galloping stability of rectangular and H-sections

*M. Keerthana and P. Harikrishna
Wind Engineering Laboratory, CSIR-Structural Engineering Research Centre, Taramani, Chennai-600113, INDIA

Received 18 July 2012; revised 20 December 2012; accepted 15 March 2013

Wind induced aerodynamic forces are considered to be important for the design of buildings and structures. Additional aerodynamic forces, which can cause instability, are induced in flexible structures like suspension/cable stayed bridge decks owing to wind-structure interaction effects. Hence, it becomes necessary to assess the stability of cross-sections of such bridge decks against the effects like galloping and flutter. In the present study, towards the application of CFD as a numerical tool in assessing the stability of bridge cross-sections against transverse galloping, smooth flows around three 2-D rectangular sections and a 2-D H-section representing a realistic bridge deck cross-section have been numerically simulated for various angles of wind incidence at Reynolds number of 61000. Two of the popular RANS turbulence models, namely Realizable k-ε (RKE) and Shear Stress Transport k-ω model (SST) available in commercial CFD code FLUENT have been used. The numerically evaluated aerodynamic drag and lift force coefficients and their variations with angle of wind incidence have been used to assess the stability of the sections against transverse galloping using quasi-steady assumption. Further, the numerically evaluated aerodynamic force coefficients have been compared with the experimental results available in literature for the validation of CFD simulations.

Keywords: Aerodynamics, Computational Fluid Dynamics (CFD), galloping, smooth flow

Introduction

With the advancement in materials and design methodologies, modern day lightweight and flexible structures such as bridge decks vibrate/oscillate in wind flow, causing interaction between the structural motion and wind flow. Windstructure interaction in such structures results in additional aerodynamic forces, which cause a variety of instabilities like galloping (transverse or torsional modes) and flutter, resulting in vibration of the structure with very large amplitudes. Hence, it becomes essential to assess the stability of cross-sectional shapes of such structures against galloping/flutter. Most of the earlier experimental investigations on transverse galloping stability analysis reported in literature are based on quasi-steady assumption.

For rectangular sections with various side ratios (B/D; where B is the breadth of the rectangle along the flow direction and D is the depth of the rectangle normal to the flow), experimentally measured force coefficients have been used to evaluate the galloping response in quasi-steady mode. Further, experimental studies on different rectangular sections for various angles of wind incidence have been reported in literature. Recently, CFD has become a viable numerical tool in simulating the flow around buildings/structures. However, the numerical studies carried out on rectangular sections with B/D of 2 or 5 are either at 0° or 90° angle of wind incidence. Only a few numerical studies have been reported in literature for rectangular sections under different angles of wind incidence.

In the present study, CFD has been applied to assess the transverse galloping stability of four different cross-sections by considering quasi-steady assumption. Three rectangular sections with B/D of 2, 4 and 5, have been considered with D=0.06 m. Further, an H-shaped cross-section which is typical of original/first Tacoma Narrows bridge section, with B/D ratio of 5 has been considered (Figure 1). The numerically evaluated aerodynamic force coefficients have been used to assess the stability of the sections against transverse galloping.

Analysis for transverse galloping

A single DOF galloping model where the rectangular section is supported on spring in order to allow only transverse motion (y(t)) has been considered. The model is subjected to smooth flow with velocity U and density...
of $\dot{\varphi}$. As the rectangular section oscillates in transverse direction about its static equilibrium with velocity $\dot{y}$, the effective flow velocity/relative velocity has an angle of incidence of $\alpha$ with respect to the mean flow direction. This implies that the drag and lift forces ($F_D$ and $F_L$ respectively) have resolved components in the transverse direction. Taking mass per unit length as $m$, damping coefficient due to dissipation within the structure as $\zeta$, stiffness of the spring per unit length as $K$ and the natural frequency of the structure as $\omega_n$, the linear dynamic equilibrium equation considering the aerodynamic forces in the transverse direction is obtained as Equation 1 for small angles of wind incidence. The intermediate steps have not been provided for brevity.

$$m \ddot{y} + 2m \omega_n \left( \xi - \frac{\rho UD C_s}{4m \omega_n} \frac{\partial C_s}{\partial \alpha} \right) \ddot{\alpha} + K y = -\frac{1}{2} \rho U^2 D C_{D1} \left|_{\alpha=0} \right.$$  

Equation (1)

The net damping factor has aerodynamic term (second term in the parentheses of Equation 1) in addition to the structural damping term (first term in the parentheses of Equation 1). Here, when the net damping is negative, the oscillations tend to increase with time and galloping takes place. From this, the necessary condition for galloping can be obtained as given in Equation 2.

$$\frac{\partial C_s}{\partial \alpha} \left|_{\alpha=0} \right. > 0 \quad \text{or} \quad \left[ \frac{\partial C_L}{\partial \alpha} + C_D \right] \left|_{\alpha=0} \right. < 0 \quad \text{for instability}$$  

Equation (2)

When the slope of transverse force coefficient is greater than zero, the section is potentially unstable for transverse galloping and is stable otherwise (when the slope of transverse force coefficient is negative). Detailed information about the formulation can be found elsewhere.

### Numerical Simulations

The turbulent wake structures comprise of a wide range of eddies with different length scales and frequency content. Directly simulating all the scales of eddies using Direct Numerical Simulation (DNS) on a very fine grid with very small time step is prohibitively expensive. Though Large Eddy Simulation (LES) reduced the grid requirement and computational demand compared to DNS, by modeling only dissipative eddies and simulating other eddies, it has still been found to be computationally intensive. RANS (Reynolds Averaged Navier-Stokes) based models have been observed to be computationally cheap and viable for engineering practices, wherein the entire range of eddies are modeled. RANS based two-equation turbulence models, viz, $k-\varepsilon$ based models and the $k-\omega$ based models have become industry standard models and are commonly used for most of the engineering problems. The equations governing the fluid flow are solved numerically to get the flow variables.

### Governing equations

The basic governing differential equations for an unsteady incompressible flow as per RANS based two-equation turbulence models are given by Equations 3 and 4 Reynolds-averaged continuity equation:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0$$  

Equation (3)

Reynolds-averaged Navier-Stokes equation:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \tau'_{ij} \right]$$  

Equation (4)

where $\bar{u}_i$ are the mean velocity components in $x$ ($i=1$), $y$($i=2$) and $z$($i=3$) directions in Cartesian coordinate system, $p$ is the pressure, $\dot{j}$ is the fluid density, $\nu$ is the kinematic viscosity of the fluid and $\tau'_{ij}$ is the Reynolds stress tensor, as a result of ensemble averaging, which has six unknown terms. To obtain closed set of equations, Reynolds stress tensor is modeled with the Boussinesq eddy viscosity assumption, which postulates that Reynolds stress tensor $\tau'_{ij}$ is proportional to the mean strain rate tensor in analogy to the viscous stresses in laminar flows, as given below:
In Equation 5, $V_t$ is the kinematic eddy viscosity (which is not a flow property, but a variable that depends on the flow), $\delta_{ij}$ is the Kronecker delta function ($=1$ for $i=j$ and 0 otherwise). The kinematic eddy viscosity is dimensionally expressed as

$$\frac{\tau'_\theta}{\rho} = V_t \left\{ \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right\} - \frac{2}{3} k \delta_{ij}$$

(5)

Where, $l_o$ and $\tau_o$ are turbulent length scale and turbulent time scale, respectively. Hence, Reynolds stresses are characterized using single length and time scales. In $k-\varepsilon$ based models, $l_o$ and $\tau_o$ are obtained using the turbulent kinetic energy ($k$) and the turbulent energy dissipation rate ($\varepsilon$). In $k-\omega$ based models, $l_o$ and $\tau_o$ are obtained using the turbulent kinetic energy ($k$) and the specific dissipation rate ($\omega$). The standard $k-\varepsilon$ model has been found to perform poorly in highly strained flows with highly curved streamlines (e.g. swirling / rotating flows) near bluff bodies and in anisotropic turbulent flows, whereas standard $k-\omega$ model, which is substantially more accurate than the standard $k-\varepsilon$ model in the near wall layers, has been successful for flows with moderate adverse pressure gradients, but fails for flows with pressure induced separation. To overcome these limitations, improved turbulence models like Realizable $k-\varepsilon$ model (RKE) and Shear Stress Transport $k-\omega$ model (SST) have been formulated. RKE model is an improvement over standard $k-\varepsilon$ model, in which the model satisfies certain mathematical constraints on Reynolds stress in consistent with the physics of turbulent flows, which provides superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. SST model combines the positive features of standard $k-\varepsilon$ model and standard $k-\omega$ model by adopting standard $k-\varepsilon$ model near the wall and standard $k-\omega$ model near the boundary layer edge, thereby showing good performance under adverse pressure gradients. In the present study, RKE and SST models have been considered. The transport equations for RKE and SST models are available elsewhere.

**Computational domain**

A typical computational domain for rectangular section with side ratio of 2 at 10° angle of wind incidence is shown in Figure 2. The depth of the section is 0.06m (D) and breadth of the section which is along the flow (B) has been obtained based on side ratio (B/D) of the rectangular section considered (or that of enclosing rectangle in case of H-section). The computational domain has been chosen with respect to the cross-section in such a way that the solution is independent of the locations of the boundaries. The inlet boundary has been chosen at a distance of 15B from the front wall of the core region $R_1$. The outlet boundary has been chosen at a distance of 25B from the rear wall of $R_2$. The symmetry
(free-slip) boundaries were chosen at a distance of 15B from the side walls of R₁. The computational domain consists of core region (R₁) around the cross-section and Region (R₂), which extends beyond the core region (R₁) up to the boundaries of the computational domain. The dimensions of the core region vary based on angle of wind incidence and side ratio, while R₂ remains the same.

Meshing

The region near the wall of the cross-section is very sensitive as the gradients of flow variables are very high. To simulate flow in such regions, mesh has to be made very fine in the near wall region. Instead of going for very fine mesh, suitable near-wall treatments are generally adopted, which does not solve flow very near the wall, instead employ semi-empirical formulas in the near wall region. Standard wall functions have been used for near wall treatment in the present study. The distance of first grid point should be in such a way that the wall unity (y* in FLUENT) is around 30. This ensures that the centroid of first cell adjacent to the wall is in the log layer of the wall.

\[ y^* = \frac{y u^*}{v} \]  

Where y is the distance of first grid point from the wall, \( u^* \) is the shear friction velocity, which is function of Reynolds number. Based on the y* criteria with Reynolds number of 61000 considered in the present study, meshing of the R₁ region has been carried out using suitable distance of first grid point from the wall. Unstructured mesh of triangular elements has been used in this regionof R₁. This is acceptable in the present case of bluff body aerodynamics, in which the aerodynamic forces are mainly due to pressure/form forces rather than due to skin friction, which requires boundary layer mesh. Beyond the region of R₁, the distance between grid lines from the edgeof R₁ to the domain boundaries has been varied using a stretching ratio of 1.1 (ratio of sizes of adjacent cells is maintained as 1.1) following a geometric series. The enlarged view of the mesh showing core region (R₁) and a part of surrounding region (R₂) for 10° angle of wind incidence for the side ratio of 2 is also shown in Figure 2.

Boundary conditions and numerical schemes

The mean velocity \( \bar{u} \) (same as U) at the inlet boundary has been given as 15m/s, which corresponds to Reynolds number of 61000 (using \( \bar{\partial} \bar{\partial} \) as characteristic dimension). Turbulent intensity (I) of 0.2% and turbulent viscosity ratio \( \nu = \bar{v} / v \) of 10 were chosen for smooth flow condition. The values of turbulent kinetic energy (k), dissipation rate (\( \bar{\Omega} \)) and specific dissipation rate of turbulent kinetic energy (\( \gamma \)) at the inlet boundary were obtained based assuming isotropic turbulence. At the outlet, gauge pressure has been set equal to zero and the backflow turbulent parameters were set equal those at inlet boundary in order to reduce the difficulties in convergence. At the symmetry boundary, the velocity in the direction normal to the boundary has been set equal to zero and also the derivatives of all the velocity variables in the direction normal to the boundary were set equal to zero. Wall (no-slip) boundaries were chosen for all the sides of the rectangular section at which the velocities in the flow and normal to flow directions were set equal to zero. The pressure field was linked to velocity through SIMPLE (Semi-Implicit Method for Pressure Linked Equations) pressure-velocity coupling algorithm. Second order upwind scheme was employed for solving the transport equations of momentum, turbulent kinetic energy and turbulent dissipation rate specific turbulent diffusion. The unsteadiness in the present simulations is mainly due to periodic vortex shedding in the wake behind the rectangular section. Based on the literature, the vortex shedding frequency for the chosen values of and D, have been observed to be in a range of 20 Hz to 30 Hz. Hence, the time step \( \Delta t \) must be less than 0.0167 s based on Nyquist frequency. In the present study, a time step size of 0.0005 s has been chosen with 75 iterations per time step for the purpose of convergence. For convergence, the residual criteria for x, y momentum, k, \( \bar{\Omega} \) and continuity equations have been set to be \( 10^{-4} \). Temporal integration has been done using second order implicit method. The aerodynamic force coefficients have been monitored until the results converged, that is, periodic variation with time. Simulations have been carried out for a range of critical angles of wind incidence of -10° to +10°.

Results and Discussions

RANS based models have been used more commonly in literature for high Reynolds number flow simulation owing to reasonable accuracy at reduced computational cost. Since they are based on single length and time scales (as given in Equation 6), they have limitations in terms of capturing detailed time variation of single representative eddy size. In flow situations where
the unsteadiness in flow is mainly due to the periodic vortex shedding, these models have provided good quantitative comparison with the experimental results. However, when there are asymmetric recirculation regions and vortices, accurately resolving the flow is still a challenge for RANS based models. A typical example of such flow situation drawn from the present study is shown in Figure 3, which shows the contours of mean pressure coefficient for flow around 2-D H-section at 0º angle of wind incidence.

From the time histories of force coefficients of the unsteady simulation (after convergence), the statistical mean of the force coefficients have been obtained. The mean lift and drag coefficients were computed based on $\bar{D}$ as characteristic dimension. Figure 4 shows variation of mean lift coefficient (obtained by both RKE and SST models) with angle of wind incidence for rectangular sections with B/D of 2 as a typical case. Similar plots have been obtained for rectangular sections with B/D of 4, 5 and H-section. The slopes of transverse force coefficient have been evaluated and the values have been given in Table 1. It is to be noted that for evaluation of slope, the angle of wind incidence has to be considered in radians. The slope of transverse force coefficient has been observed to be positive for the rectangular section with B/D of 2, which indicates that the section is unstable against transverse galloping (Equation 2). The values of slope of transverse force coefficient for various cross-sectional shapes, viz, rectangular sections of various aspect ratios, angle section, etc is available in literature based on wind tunnel experiments. The actual values of the slope of transverse force coefficient in literature were obtained by considering the dimension of rectangular region parallel to the flow direction (B) as characteristic dimension. However, in the present study, the dimension of rectangular section perpendicular to the
flow direction (D) has been considered as characteristic dimension. Hence, the reported values have been normalized with \( \Delta \Delta \Delta \) as in the present study, to compare with the present results. The slope of transverse force coefficient for rectangular section with B/D of 2 has been given as 5.6. For rectangular section with B/D of 2, RKE model predicts the value of slope which has been observed to compare well with the value as given by literature, while SST model under predicts the slope by about 20%.

For rectangular section with B/D of 4 and 5, and H-section, the slope of transverse force coefficient has been observed to be negative which indicates that, the sections are stable against transverse galloping. This is consistent with the observations in literature\(^1\), which states that the sections with B/D greater than 2.8 are stable against transverse galloping. For rectangular section with B/D of 4, the slope predicted by RKE model shows maximum deviation of 55%, whereas the slope evaluated using SST model shows a deviation of 31%, even though the overall trends have been similar in comparison with literature. For rectangular section with B/D of 5 and H-section, the slope of transverse force coefficient predicted by RKE and SST models are close. Further to assessing the transverse galloping stability of cross-sections, the aerodynamic force coefficients that have been used to assess the galloping stability have been compared with the experimental values available in the literature, in order to validate the results of numerical simulation. Figure 4 also includes the comparison of mean lift coefficients evaluated from numerical simulations with literature for rectangular section with B/D of 2 for all angles of wind incidence. The comparisons have also been made for rectangular section with B/D of 5 as well as H-section. For rectangular section with B/D of 4, the mean lift coefficient values for all angles of wind incidence considered in the present study are not available in literature. Hence, comparison of mean lift coefficient with literature has not been included for this rectangular section.

For rectangular sections with side ratio of 2, 5 and H-section, it has been observed that the numerically evaluated lift coefficients have trends similar to those reported in literature. From Figure 4, for rectangular section with side ratio of 2, for angles of wind incidence between 0° and 5°, the mean lift coefficients are close to the experimental results, with 5-10% difference. The percentage of difference is more pronounced between 6° and 8° angles of wind incidence which is the range in which there is complete change in flow characteristics\(^1\) for such sections. The change in flow characteristics may be attributed to the shift in the location of flow separation from leading edge to trailing edge. Results from both RKE and SST models for B/D of 2 have shown lift coefficient peaking at angle of wind incidence of 7°, as reported in the literature, which indicates qualitative prediction of flow characteristics. However, the magnitude of mean lift coefficient has been observed to be under predicted by both the models.

For rectangular section with B/D of 5, SST-model has been observed to give better prediction of mean lift coefficient than RKE-model, though the mean lift coefficients evaluated using both the models have been

<table>
<thead>
<tr>
<th>Section</th>
<th>Numerical Model(s)</th>
<th>( \frac{\partial C_L}{\partial \alpha} ) (_{\alpha=0})</th>
<th>( C_D ) (_{\alpha=0})</th>
<th>( \frac{\partial C_L}{\partial \alpha} ) (_{\alpha=0})</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D rectangle</td>
<td>Realizable k-( \Omega )-model</td>
<td>-7.23</td>
<td>1.41</td>
<td>5.82</td>
</tr>
<tr>
<td>(B/D=2)</td>
<td>(RKE)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Shear Stress Transport k-( \chi ) model (SST)</td>
<td>-5.86</td>
<td>1.39</td>
<td>4.47</td>
</tr>
<tr>
<td>2-D rectangle</td>
<td>Realizable k-( \Omega )-model</td>
<td>61.04</td>
<td>1.20</td>
<td>-62.21</td>
</tr>
<tr>
<td>(B/D=4)</td>
<td>(RKE)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Shear Stress Transport k-( \chi ) model (SST)</td>
<td>25.34</td>
<td>1.10</td>
<td>-26.44</td>
</tr>
<tr>
<td>2-D rectangle</td>
<td>Realizable k-( \Omega )-model</td>
<td>37.54</td>
<td>0.98</td>
<td>-38.52</td>
</tr>
<tr>
<td>(B/D=5)</td>
<td>(RKE)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Shear Stress Transport k-( \chi ) model (SST)</td>
<td>30.82</td>
<td>1.08</td>
<td>-31.90</td>
</tr>
<tr>
<td>2-D H-section</td>
<td>Realizable k-( \Omega )-model</td>
<td>56.43</td>
<td>1.25</td>
<td>-57.68</td>
</tr>
<tr>
<td></td>
<td>(RKE)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Shear Stress Transport k-( \chi ) model (SST)</td>
<td>61.57</td>
<td>1.35</td>
<td>-62.92</td>
</tr>
</tbody>
</table>

Table 1: Slope of transverse force coefficients
found to be in close agreement. As the geometry becomes complex, in case of H-section, the simulations over predict the mean lift coefficient at higher angles of wind incidence. A maximum deviation of 65% has been observed with respect to the results of literature in mean lift coefficient for 8º angle of wind incidence. The mean drag coefficients obtained from the present simulation have been compared with literature values for 0º angle of wind incidence since has been used to evaluate the slope of transverse force coefficient, as can be seen in column 4 of Table 1. For rectangular section with B/D of 2, the mean drag coefficient for $\alpha = 0^\circ$ has been observed to be in the range of 1.4 to 1.69 as per literature. The evaluated mean drag coefficients using RKE and SST models have been observed to be under predicted, but, compare well with the lower bound value of 1.4 from literature. The reason for the under prediction of mean drag coefficient has been investigated by comparing the evaluated mean pressure distributions on the surface of the rectangular section from present numerical studies with numerical and experimental studies reported in literature (Figure 5). The distribution of mean pressure coefficient will aid in improved understanding of the performance of turbulence models. The mean drag force coefficient has been obtained by integrating the pressures on the front and rear surfaces of the rectangular sections. On the front surface (windward portion), the evaluated mean pressure distributions using RKE and SST models have been observed to compare well with those from literature. The negative pressures on the rear surface of the section have been observed to be under predicted, which has resulted in the under prediction of mean drag coefficient. It has also been observed that the negative pressures on the side walls of the rectangular section are under predicted.

For rectangular section with B/D of 4, the mean drag coefficient for $\alpha = 0^\circ$ is observed to be in the range of 1.05 to 1.43 as per literature. The mean drag coefficients values of 1.30 and 1.43 from 2-D and 3-D LES computations of literature have been observed to be consistently more compared to other numerical results of literature. The results of the present numerical simulation agree closely with the other numerical results of literature with corresponding $C_D$ values being 1.05 and 1.17. For rectangular section with B/D of 5, the mean drag coefficient for $\alpha = 0^\circ$ evaluated using both the turbulence models has been found to compare well with numerical and experimental values. The evaluated mean pressure coefficients on the surface of the rectangular section with side ratio of 5 have been compared with the literature values. It has been observed that the negative pressure on the rear surface is well predicted by both RKE and SST models apart from on the front surface. Pressures on the side walls of rectangular section evaluated by using SST model have been observed to compare better than RKE model.

For H-section, the mean drag coefficients for $0^\circ$ angle of wind incidence accurately matches with the value of
1.40 from discrete vortex numerical simulations\textsuperscript{19}. But, the mean drag coefficient has been observed to be under predicted by numerical simulations (both present study and discrete vortex simulations) in comparison to the experimental results (value of 1.85). A maximum deviation of -27\% has been observed. For, $\dot{U} = 0^\circ$ at the windward portion, the flow encounters a thin rectangle (the ratio of dimension normal to flow and the dimension along the flow being 0.1), which is the front vertical part of H-section. It has been observed that for thin rectangles, the results from numerical simulations show deviations with respect to the experimental results\textsuperscript{20}. Also, it has been reported in literature that for rectangular section with B/D of 5, where the bluff shape has long after body, the numerical predictions corroborated well with the experimental results\textsuperscript{20}, which corroborated well in the present study also.

**Conclusions**

In the present study, quasi-steady assumption has been used for assessing the transverse galloping stability of rectangular and H sections. Based on the slope of transverse force coefficient obtained from numerical studies using two popular turbulence models namely RKE and SST models, it has been observed that the rectangular section with side ratio of 2 is unstable against transverse galloping while the section with side ratio of 4, 5 and H-section are highly stable for the same. These findings corroborate with the similar observations made in literature based on wind tunnel experiments. The magnitude of the slope of transverse force coefficient predicted by RKE model for rectangular section with side ratio of 2 compared well with the value reported in literature, whereas for rectangular section with B/D of 4, deviations have been observed in the prediction of both the turbulence models. The numerically evaluated mean lift coefficients for rectangular sections with B/D of 2, 5 and H-section have been compared with the experimental results of literature. As the angle of wind incidence increases, the mean lift coefficients have been observed to show deviations with respect to the experimental results of the literature, owing to increased complexity in the flow behavior around the cross-sections. The mean drag force coefficient at $\dot{U} = 0^\circ$ evaluated from RKE and SST models have also been compared with literature for all the cross-sections considered in the present study.

The present study proves the successful applicability of numerical studies using CFD to predict the stability of bridge deck cross-sections against transverse galloping. Even though the qualitative predictions have been observed to be useful to assess the stability, qualitative deviations in mean force coefficients have been observed. From the comparison of mean pressure coefficients, it can be observed that the windward pressure coefficients have been well predicted by both the turbulence models, whereas prediction of suction pressure coefficients in the wake of the cross-sections/after separation has been observed to show deviations with respect to experimental results, which could probably be due to limitations even in improved RANS based turbulence models. Presently, there is no universal turbulence model suitable for all kinds of flow problems. Improvements are needed in these turbulence models in order to make them applicable for complex flow problems involved with different angles of wind incidence.

**Acknowledgment**

This paper is being published with the kind permission of Director, CSIR-SERC, Chennai. The support rendered by the staff of Wind Engineering Laboratory of CSIR-SERC is greatly acknowledged.

**References**


