SIGNIFICANCE OF CORRELATION COEFFICIENT IN EVALUATING REYNOLDS STRESSES

S. Mandal\textsuperscript{1}, C.S.P. Ojha\textsuperscript{2}, and P. Bhargava\textsuperscript{2}

1) Senior Res. Fellow (CSIR), Dept. of Civil Engrg., Univ. of Roorkee, Roorkee, U. A. – 247 667, India.
2) Assoc. Prof., Dept. of Civil Engrg., Univ. of Roorkee, Roorkee, U. A. – 247 667, India.

1. INTRODUCTION

Turbulent flow is marked with the presence of fluctuating velocity components $u$, $v$, and $w$ in three directions $x$, $y$, and $z$ of a flow field resulting in Reynolds stresses. Various stress models, either linear or non-linear in nature, have been proposed in the literature. Boussinesq’s (1877) \cite{1} constitutive relations are examples of linear models while Speziale’s (1987) \cite{2} model belongs to a class of non-linear models. One of the components of these models is the expression of $-uw$ which reflects the joint variation of root mean square of fluctuating components $u$ and $w$. It is the representation of this component, which requires the use of correlation coefficient between $\sqrt{u^2}$ and $\sqrt{w^2}$. Sometimes recourse to the use of correlation coefficient is a must to estimate $-uw$. Truly speaking, the correlation can vary from one control volume to another and it may not be reasonable to use one correlation coefficient based on the record of all the measurements of $u$ and $w$ in a flow domain. Some of the text books \cite{3} mention that the correlation coefficient varies from 0.45 to 0.55 for turbulent boundary layers. Also, certain dependence between correlations in spatial and time domain may exist \cite{4,11}. However, the use of these values or concepts may lead to unrealistic results. Thus, the objectives of the present paper are centered around the use of correlation coefficient with particular emphasis to the use of correlation coefficient based on entire flow domain or use of correlation coefficient based on control volume or a smaller domain of flow field. For this purpose, comprehensive data of a turbulent flow field around a bluff body based on LDA measurements are considered from \cite{5} and issues such as the use of correlation coefficient within a range of 0.45 to 0.55 or that based on spatial correlation are analyzed.

2. GOVERNING EQUATIONS

A 2-D turbulent flow field can be described by Reynolds Averaged Navier Stokes (RANS) equations comprising of continuity, momentum and turbulence transport equations \cite{6}. The RANS momentum equation involves Reynolds stresses $\overline{-u_j u_l}$. Of these, the component $-uw$ is of interest here as data in \cite{5} does not provide this. Boussinesq’s (1877) linear \cite{1} and Speziale’s (1987) non-linear \cite{2} constitutive relations, respectively, express these turbulent stresses in terms of gradients of mean velocity components as

\begin{equation}
-uw = \nu_t \left( \frac{\partial U}{\partial z} + \frac{\partial W}{\partial x} \right)
\end{equation}

\begin{equation}
-uw = C_\mu k^2 \left( \frac{\partial U}{\partial z} + \frac{\partial W}{\partial x} \right)
+ 4C_D C_\mu \frac{k^3}{\varepsilon^2} \left[ \left( \frac{\partial U}{\partial z} \frac{\partial W}{\partial z} \right) - \left( \frac{\partial U}{\partial x} \frac{\partial W}{\partial x} \right) + t_3 \right]
\end{equation}

where, $C_D = 1.68$ (an empirical constant \cite{2}), $C_\mu = 0.09$, and the $t$ terms are as,

\begin{equation}
t_3 = \frac{\partial}{\partial x} \left[ \frac{U}{2} \left( \frac{\partial U}{\partial z} + \frac{\partial W}{\partial x} \right) + \frac{\partial}{\partial z} \left( W \left( \frac{\partial U}{\partial z} + \frac{\partial W}{\partial x} \right) \right) \right]
\end{equation}

Using the expression for turbulent eddy viscosity, the terms $C_\mu \frac{k^2}{\varepsilon}$ and $C_\mu \frac{k^3}{\varepsilon^2}$ appearing in (2) can be replaced by $\nu_t$ and $\frac{U^2}{\varepsilon}$, respectively.
3. COMPUTATION OF REYNOLDS STRESSES

Computation of Reynolds stresses also requires the computation of eddy viscosity \( \nu_t \). For airflow, the kinematic viscosity is negligible, and hence, k-equation takes the form,

\[
U \frac{\partial k}{\partial x} + W \frac{\partial k}{\partial z} = \frac{\partial}{\partial x} \left[ \nu_t \frac{\partial k}{\partial x} \right] + \frac{\partial}{\partial z} \left[ \nu_t \frac{\partial k}{\partial z} \right] + P_k - \varepsilon
\]

(4)

where, the term \( \nu_t \) has been omitted as \( \nu_t = 1.0 \). On a typical control volume (see 3 in Fig. 2), (4) can be integrated and one obtains

\[
(U_x k_x - U_{x'} k_{x'})_{\Delta x} + (W_x k_x - W_{x'} k_{x'})_{\Delta x} = \left[ \left( \nu_t \frac{\partial k}{\partial x} \right)_{x_{2}} - \left( \nu_t \frac{\partial k}{\partial x} \right)_{x_{1}} \right]_{\Delta x}
+ \left[ \left( \nu_t \frac{\partial k}{\partial z} \right)_{z_{2}} - \left( \nu_t \frac{\partial k}{\partial z} \right)_{z_{1}} \right]_{\Delta z} + (P_k)_r \Delta x \Delta z - \left( \frac{C_f k^2}{u_i} \right)_r \Delta x \Delta z
\]

(5)

Alternatively, the left-hand side of (5) can be approximated to yield

\[
U_x (k_x - k_{x'})_{\Delta x} + W_x (k_x - k_{x'})_{\Delta z} = \left[ \left( \nu_t \frac{\partial k}{\partial x} \right)_{x_{2}} - \left( \nu_t \frac{\partial k}{\partial x} \right)_{x_{1}} \right]_{\Delta x}
+ \left[ \left( \nu_t \frac{\partial k}{\partial z} \right)_{z_{2}} - \left( \nu_t \frac{\partial k}{\partial z} \right)_{z_{1}} \right]_{\Delta z} + (P_k)_r \Delta x \Delta z - \left( \frac{C_f k^2}{u_i} \right)_r \Delta x \Delta z
\]

(6)

where, \( \varepsilon \) has been replaced using eddy viscosity and the subscripts E, W, N, S denote the location for considering variable values in adjacent control volumes (see Fig. 2). \( P_k \) can be expressed in a simplified form as

\[
P_k = -u_{\tau}^2 \frac{\partial U}{\partial x} - w_{\tau}^2 \frac{\partial W}{\partial z} - \overline{u w} \left( \frac{\partial U}{\partial z} + \frac{\partial W}{\partial x} \right)
\]

(7)

After substituting for \( -u_{\tau}^2 \), \( -w_{\tau}^2 \), \( -\overline{u w} \) from linear and nonlinear models, two values of \( P_k \) can be obtained. Considering two types of \( P_k \) expressions, two types of discretization schemes [(5) and (6)] and two expressions of \( -\overline{u w} \) [(1) and (2)], it is possible to obtain six different estimates of \( -\overline{u w} \). Computational strategies 1 and 2 are based on the use of linear \( P_k \) in (5) and (6) respectively. Strategies 3 and 4 consider use of nonlinear \( P_k \) without t terms in (5) and (6) respectively, while strategies 5 and 6 consider use of \( P_k \) with t terms in (5) and (6) respectively.

4. ANALYSIS OF DATA AND DISCUSSION

Experimental data are provided for non-dimensionalised mean velocities \((U, W)\) and respective fluctuating components \((u, w)\) measured using Laser Doppler Anemometer in [5]. As \( v \) was not provided, a parameter \( F_k \) is introduced to obtain \( k \) in the absence of \( v \) as

\[
k = F_k \left( \overline{u^2} + \overline{w^2} \right)
\]

(8)

In the analysis, \( F_k \) is varied from 0.5 to 1 with increment of 0.1.

For the present analysis, \( -\overline{u w} \) is also statistically calculated from the data of [5] using a correlation coefficient, \( R \), as

\[
-\overline{u w} = R \sqrt{u^2 \overline{w^2}}
\]

(9)

Certain typical variations of \( R \) reported in the literature are shown in Fig. 1. In the present study, \( R_1 \) (local) and \( R_g \) (global) are defined based on the available measurements of \( \overline{u^2} \) and \( \overline{w^2} \) within the control volume and within the entire flow domain respectively. As the objective is to examine the worth of \( R_1 \) and \( R_g \), an error term \( E \) is defined as

\[
E = 100 \times \left| \frac{-\overline{u w} \text{statistical}}{-\overline{u w} \text{computed}} \right|
\]

(10)

The two statistical estimates of \( -\overline{u w} \) are available depending on the use of \( R_1 \) or \( R_g \). Similarly, six estimates of \( -\overline{u w} \text{computed} \) are obtained depending on the use of six computational strategies.

For different values of \( F_k \) varying between 0.5 to 1.0 (see Fig. 1a for typical variations), the correlation coefficient between \( -\overline{u w}_l \) and \( -\overline{u w}_g \) with six computed Reynolds stresses are given in Table 1a and 1b respectively. Corresponding to correlation coefficients of Table 1, for two typical values of \( F_k \) as 0.6 and 0.9, a plot of \( E \) values for different computational strategies are shown in Fig. 3. With the exception in certain strategies for control volume 1 and 2, it can be seen from Fig. 3 that almost in all cases analyzed in the study. Despite the limitation of the data in terms of non-availability of \( v \), the significance of the correlation coefficient in evaluating Reynolds stresses has been highlighted. The study also indicates the limitations of using correlation.

5. CONCLUSIONS

The implications of using reported values of correlation coefficients between certain fluctuating components of velocities have been examined using linear as well as non-linear Reynolds stress models and less support for their use has been found. The computed values of correlation coefficients either in different parts of flow field or based on the consideration of entire flow field are no where close to the reported values of 0.45 to 0.55. Rather results are better at \( F_k \) adopted as 0.6. Use of correlation coefficient based on the measurements of velocity field within a control volume is found to perform relatively better (Fig. 3) in majority of cases analyzed in the study. Despite the limitation of the data in terms of non-availability of \( v \), the significance of the correlation coefficient in evaluating Reynolds stresses has been highlighted. The study also indicates the limitations of using correlation.
coefficient based on spatially measured velocities. Future efforts can concentrate on the analysis using complete flow field data and the sensitivity of errors to the turbulence parameters. In view of this, recourse must be taken to rely on the correlation within the time domain despite the existing views regarding certain correlation between spatial and temporal variation of the turbulence parameters.

ACKNOWLEDGEMENTS

This study was financially supported (Ref. File no. 9/143(356)/97-EMR-I-RKM) by the Council of Scientific and Industrial Research (CSIR), New Delhi, India.

Keywords: k-ε model, turbulent or eddy viscosity, spatial and temporal correlation coefficient, Reynolds stresses, control volume method.

APPENDIX I: References


APPENDIX II: Notations

The following symbols are used in this paper:

- $C_D$ = empirical constant in nonlinear constitutive relations;
- $F_k$ = factor to relate $u$ and $w$ with $k$;
- $H$ = height of the building model;
- $k$ = turbulence kinetic energy per unit mass;
- $P_k$ = production term in $k$-equation;
- $R$ = correlation coefficient in (9);
- $t_3$ = terms in nonlinear Reynolds stress models;
- $u$, $v$, $w$ = dimensionless fluctuating components of velocity in $x$, $y$, and $z$ directions;
- $U, W$ = dimensionless mean velocity components in $x$ and $z$ directions respectively;
- $-\overline{uw}$ = turbulence (Reynolds) stress per unit mass;
- $\Delta x, \Delta z$ = control volume size in $x$ and $z$ directions;
- $\varepsilon$ = rate of dissipation of $k$;
- $\nu_t$ = turbulent eddy viscosity;
- $\rho$ = density of fluid;
- $\sigma_k$ = empirical constant in $k$-equation;

subscripts:

P, E, W, N, S,
EE, WW, NN, SS = indicate the location where the variables are evaluated.

Table 1a: Correlation coefficients between $(-\overline{uw})$ and computed Reynolds stresses

<table>
<thead>
<tr>
<th>$F_k$ Stresses</th>
<th>0.5</th>
<th>0.6</th>
<th>0.7</th>
<th>0.8</th>
<th>0.9</th>
<th>1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$(-\overline{uw})$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$(-\overline{uw})_1$</td>
<td>0.449</td>
<td>0.576</td>
<td>0.485</td>
<td>0.604</td>
<td>0.427</td>
<td>0.413</td>
</tr>
<tr>
<td>$(-\overline{uw})_2$</td>
<td>0.119</td>
<td>0.574</td>
<td>0.330</td>
<td>0.599</td>
<td>0.745</td>
<td>0.850</td>
</tr>
<tr>
<td>$(-\overline{uw})_3$</td>
<td>0.488</td>
<td>0.573</td>
<td>0.398</td>
<td>-0.18</td>
<td>0.754</td>
<td>-0.01</td>
</tr>
<tr>
<td>$(-\overline{uw})_4$</td>
<td>0.447</td>
<td>0.572</td>
<td>0.395</td>
<td>0.599</td>
<td>0.760</td>
<td>0.445</td>
</tr>
<tr>
<td>$(-\overline{uw})_5$</td>
<td>0.446</td>
<td>0.571</td>
<td>0.420</td>
<td>0.599</td>
<td>0.756</td>
<td>0.202</td>
</tr>
<tr>
<td>$(-\overline{uw})_6$</td>
<td>0.445</td>
<td>0.569</td>
<td>0.005</td>
<td>0.531</td>
<td>0.385</td>
<td>0.248</td>
</tr>
</tbody>
</table>

Table 1b: Correlation coefficients between $(-\overline{uw})_k$ and computed Reynolds stresses

<table>
<thead>
<tr>
<th>$F_k$ Stresses</th>
<th>0.5</th>
<th>0.6</th>
<th>0.7</th>
<th>0.8</th>
<th>0.9</th>
<th>1.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$(-\overline{uw})_1$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$(-\overline{uw})_2$</td>
<td>0.570</td>
<td>0.692</td>
<td>0.623</td>
<td>0.691</td>
<td>0.317</td>
<td>0.354</td>
</tr>
<tr>
<td>$(-\overline{uw})_3$</td>
<td>0.570</td>
<td>0.692</td>
<td>0.471</td>
<td>0.693</td>
<td>0.713</td>
<td>0.841</td>
</tr>
<tr>
<td>$(-\overline{uw})_4$</td>
<td>0.571</td>
<td>0.692</td>
<td>0.536</td>
<td>-0.19</td>
<td>0.728</td>
<td>-0.09</td>
</tr>
<tr>
<td>$(-\overline{uw})_5$</td>
<td>0.572</td>
<td>0.691</td>
<td>0.539</td>
<td>0.695</td>
<td>0.718</td>
<td>0.384</td>
</tr>
<tr>
<td>$(-\overline{uw})_6$</td>
<td>0.572</td>
<td>0.691</td>
<td>0.561</td>
<td>0.697</td>
<td>0.710</td>
<td>0.124</td>
</tr>
</tbody>
</table>
Fig. 1: Estimates of Parameters $F_k$ and $R$

Fig. 2: 2-D Computational Domain

Fig. 3: Relative Performance of Two Statistical Estimates
1. INTRODUCTION

Though many studies of computational fluid dynamics (CFD) about flows past bluff bodies are carried out, most of these studies are on flows past various bluff bodies in uniform smooth flow. Actual structures like buildings and bridges, on the other hand, are in nature wind that is turbulent and is not uniform. Characteristics of wind load on structures in smooth flow and in turbulent flow are quite different. Therefore, in investigating flows around structures, simulations need to be carried out with turbulent inflow.

In the present work, a large-eddy simulation (LES) is carried out for flow past a square and rectangular cylinders with and without free stream turbulence to examine if the LES can reproduce the free stream turbulence effects properly. A stochastic method is used to generate homogeneous turbulence with specified power and cross spectra in the approaching flow.

2. CALCULATION METHOD

Flow field to be simulated is that around a rectangular cylinder of length $B$ and height $D$ placed in a uniform flow of $U_w$ as shown in Fig.1. The coordinates and the boundary conditions are also shown. The cross section of the cylinder is either square ($B/D=1.0$) or rectangular ($B/D=2.5$). The inflow at the far left plane is generated by the method described in the next section. For the sub-grid stress, we use the standard Smagorinsky model with the values of the constant $C_s=0.13$.

LES method used here is summarized in Table.1, which is very close to that verified by Nakayama and Noda$^1$ for smooth flow past square cylinder. Calculations were done for the four cases shown in Table.2. These cases correspond to the experiments conducted by the present authors$^2$ and the Reynolds number Re based on the mean velocity of the inflow $u_w$ and the height $D$ of the cylinder is 50,000.

3. GENERATION OF INFLOW TURBULENCE

Stochastic method$^3$ was used to generate homogeneous inflow turbulence with specified power and cross spectral density in the oncoming flow. Similar method has been used in various engineering applications including seismic vibrations. Kondo et al.$^4$ and Maruyama et al.$^5$ have tried to apply to generation of inflow turbulence for CFD. Use of this method for simulations of inflow turbulence, however, is relatively

![Fig.1 Definition of flow field](image-url)
The inflow velocity must be generated at a huge number of positions and need to be defined at very closely spaced positions compared with other direct applications such as simulation of seismic waves. This method may cause a numerical instability. Therefore we need special care. Generated velocity does not necessarily satisfy the continuity equation and momentum equations in three dimensions. Especially, when continuity equation is not satisfied, subsequent flow calculation load becomes enormous. These problems are avoided using appropriate modification by Noda et al. In using this technique of generating fluctuating velocity components ($u, v, w$), we assumed that the power spectrum of Karman type as follows

$$S_s(n) = \frac{4\sigma^2 L_s/U_i}{[1 + 70.8(nL_s/U_i)^2]^{7/6}}$$  (1)

where $L_s$ is the integral length scale, $\sigma$ is the variance, $U_i$ is the mean velocity at point $i$. Since we like to generate isotropic turbulence, the cross spectra between $u$ at point $i$ and $v$ at point $j$, $v$ at $i$ and $w$ at $j$ and $w$ at $i$ and $u$ at $j$ are all set to zero but the cross spectra between $u$'s at points $i$ and $j$, $v$'s at points $i$ and $j$, or $w$'s at points $i$ and $j$ separated by $\Delta s$ is assumed to take the form specified by the Davenport type root coherence as

$$\sqrt{\text{Coh}(n)} = \frac{S_u}{\sqrt{S_u S_v}} = \exp \left\{ -\frac{k n \Delta s}{U} \right\}$$  (2)

where $\Delta s$ is the separation distance between points $i$ and $j$, and $k$ is a constant related to the decay rate. The root coherence between $u$ at points $i$ and $j$ is given by setting $k=8$, and $k=4$ for coherence between $v$'s and $w$'s.

Examples of the spectra of generated velocity components are shown in Fig.2. The Karman spectrum that is the target prescribed in the generation is also shown. It is seen that $u$ and $v$ spectra are reproduced very well except at high wave number region while the $w$ spectrum is generated to be slightly smaller than the target even in the low wave-number range. All spectra are seen to reduce their magnitudes with the streamwise distance from the plane of generation. This is mostly due to the fact that the generated velocity components do not satisfy the continuity equation. But they appear to settle within acceptable range from the prescribed one.

<table>
<thead>
<tr>
<th>Table 1 Summary of Computational Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spatial difference scheme</td>
</tr>
<tr>
<td>Time advancing</td>
</tr>
<tr>
<td>Pressure coupling</td>
</tr>
<tr>
<td>Sub-grid scale model</td>
</tr>
<tr>
<td>Calculation domain</td>
</tr>
<tr>
<td>Grid size</td>
</tr>
<tr>
<td>Minimum grid spacing</td>
</tr>
<tr>
<td>Reynolds number</td>
</tr>
<tr>
<td>Time step</td>
</tr>
<tr>
<td>Averaging time</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2 Computational case</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case</td>
</tr>
<tr>
<td>Case1</td>
</tr>
<tr>
<td>Case2</td>
</tr>
<tr>
<td>Case3</td>
</tr>
<tr>
<td>Case4</td>
</tr>
</tbody>
</table>

Fig.2 Power spectra of generated turbulence
4. CALCULATION RESULTS AND DISCUSSION

4.1 INSTANTANEOUS VORTICITY DISTRIBUTION

Typical instantaneous flow field of the calculation results are shown in Fig.3 in terms of the contours of spanwise component of the vorticity. Fig.3(a) and (c) are the “smooth flow” results without the free stream turbulence and Fig.3(b) and (d) are the “turbulent flow” results with free stream turbulence. In the cases of B/D=1.0(Case 1 and Case 2) shown in Fig.3(a) and (b), the separated shear layers off the upstream corners curve around the downstream corners of the cylinder resulting in the beginning of the vortex shedding. This formation of vortex shedding is blurred a little and the location of formation of vortex is moved closer to the body in the turbulent flow case. Otherwise the overall pattern of the flow is almost the same as that without turbulence.

In the cases of B/D=2.5, however, the case without the free-stream turbulence shows formation of vortex shedding which is larger than the case of B/D=1.0, while in Case 4 the separated shear layers reattach on the side surfaces and no clear vortex shedding is seen in the wake. The structure is broken into smaller scale motion. This is in good agreement with the experimental observation.

4.2 AERODYNAMIC PARAMETERS

The mean drag coefficient and the RMS fluctuations of the instantaneous drag and lift coefficients that are defined by

\[
C_D = \frac{\bar{f}_D}{\frac{1}{2} \rho U_m^2 D}, \quad C'_D = \frac{f'_D}{\frac{1}{2} \rho U_m^2 D}, \quad C'_L = \frac{f'_L}{\frac{1}{2} \rho U_m^2 D}
\]

where, \( \bar{f}_D \) is the mean drag force per unit length of the cylinder and \( f'_D \) and \( f'_L \) are r.m.s. of fluctuating drag and lift forces, respectively. The values of these parameters from the present simulation calculation and the

<table>
<thead>
<tr>
<th>B/D</th>
<th>Free stream turb.</th>
<th>Intensity</th>
<th>Scale</th>
<th>Re</th>
<th>St</th>
<th>( C_{pb} )</th>
<th>( C_D )</th>
<th>( C'_D )</th>
<th>( C'_L )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculation</td>
<td>Smooth</td>
<td>5.00 x 10^4</td>
<td>0.133</td>
<td>-1.435</td>
<td>2.358</td>
<td>0.215</td>
<td>1.238</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5.0%</td>
<td>1.0D</td>
<td>0.138</td>
<td>-1.418</td>
<td>2.323</td>
<td>0.287</td>
<td>1.140</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Smooth</td>
<td>6.89 x 10^4</td>
<td>0.131</td>
<td>-1.483</td>
<td>2.164</td>
<td>0.207</td>
<td>1.180</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5.3%</td>
<td>1.125D</td>
<td>0.133</td>
<td>-1.318</td>
<td>1.989</td>
<td>0.203</td>
<td>1.105</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Experiment(1)</td>
<td>Smooth</td>
<td>5.00 x 10^4</td>
<td>0.066,0.301</td>
<td>-0.879</td>
<td>1.639</td>
<td>0.062</td>
<td>0.127</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5.0%</td>
<td>1.0D</td>
<td>0.057,0.157</td>
<td>-0.705</td>
<td>1.382</td>
<td>0.179</td>
<td>0.304</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Smooth</td>
<td>5.16 x 10^4</td>
<td>0.070</td>
<td>-0.746</td>
<td>1.562</td>
<td>0.054</td>
<td>0.256</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>5.3%</td>
<td>1.5D</td>
<td>0.049,0.169</td>
<td>-0.558</td>
<td>1.346</td>
<td>0.119</td>
<td>0.217</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Fig.3 Calculated instantaneous vorticity distributions
experiments are summarized in Table 3, together with the base-pressure coefficient $C_{pb}$ and the Strouhal number $St$.

The experimental results obtained in the “smooth” flow with the tunnel turbulence intensity of 0.2% are also shown. For the case of $B/D=1.0$, calculation results of the Strouhal number $St$, the mean drag and lift coefficient $C_D$, $C_L'$ and the base pressure coefficient $C_{pb}$ are all seen to agree with the experiment fairly well indicating the quality of the basic LES simulation. In the case of $B/D=2.5$, the fluctuation of the drag coefficient $C_D'$ agrees well with experiment, but the base pressure is predicted smaller and as a result, the mean drag coefficient is larger than the experiment. It is known from the experiments by Nakaguchi et al. that the flow changes sharply near $B/D=2.8$. Also Okajima showed that for high Reynolds number flow of $Re>10^4$ the vortex shedding has two different frequencies when $B/D$ is in the range of 2.0 to 3.0. Therefore the flows of this case sensitively depend on $B/D$ and the Reynolds number. In the present LES, the reattachment of the separated flow happens to be predicted smaller.

As to the effects of turbulence, the experimental results for the case of $B/D=1.0$ indicate that the free-stream turbulence increases $St$ slightly, increases $C_{pb}$ decreases $C_D$ and decreases $C_L'$. The calculation reproduces all these trends correctly at least qualitatively. Quantitatively, the pressure recovery is under predicted. The above effects are about the same for the larger $B/D$ of 2.5, except that the fluctuating drag coefficient increases. These are all calculated correctly except that the fluctuating lift coefficient $C_L'$ is predicted larger in the case with free-stream turbulence. This is thought to be due to the widened effective width of the wake caused by the shorter reattachment length.

5. CONCLUSIONS

The present paper presents a method of generating homogeneous turbulence by a stochastic method and LES calculation results using the generated turbulence as the inflow passing square and rectangular cylinders. Calculations are also conducted with no free stream turbulence for comparison. These results are compared with experimental results and the following may be concluded.

The LES with standard Smagorinsky subgrid stress model appears to be sufficient for representing the free-stream turbulence effects. Particularly, the enhancement of boundary layer reattachment is well reproduced and the conditions for generation of vortex shedding are well predicted. The effects of the free stream turbulence on the forces on the cylinders are also well predicted but the magnitude of the pressure on the rear surface is slightly over predicted and correspondingly the drag coefficient is slightly under-predicted. This is the trend of most LES calculation with more refined subgrid models. The present result is an indication that with moderately fine grid, LES with a low-level subgrid model can reproduce properties of turbulent flow around a bluff body with reasonable accuracy for engineering purposes.

Reference

Key Words: large eddy simulation, rectangular cylinder, inflow turbulence
DNS for aerodynamic characteristics of a 3D square cylinder in the turbulent boundary layer

T. Nozu¹ and T. Tamura²

1) Institute of Technology, Shimizu Corporation, 4-17, Etchujima 3-chome, koto-ku, Tokyo 135-8530, Japan
2) Tokyo Institute of Technology, 4259, Nagatsuda, Midori-ku, Yokohama 226-8505, Japan

1. INTRODUCTION
In the case of the wind resistant problems and environmental problems of buildings, we have to consider the effect of turbulence characteristics. Therefore, it is important for numerical simulations as well as wind tunnel experiments to give a velocity profile and turbulence intensity to the approaching flow conditions similar to natural wind. So, we proposed a method of generating turbulent boundary layer with another paper¹¹ before.

In this study, we simulate the flows around a 3D square cylinder in the three approaching flow (uniform flow and two turbulent boundary layer) , and we compare the flows and aerodynamic characteristics in the numerical simulation with those in the wind tunnel test. Especially, we analyze the turbulent effects concerning the fluctuating pressure spectra of a 3D square cylinder.

2. OUTLINE OF ANALYSIS
2.1. Governing Equation
Governing equations of the 3D-incompressible flow are given by the continuity equation and the Navier-Stokes equation (x-component) as (1)(2). In this simulations, we approximate convection terms using the higher order interpolation method².³

\[
\begin{align*}
\frac{\partial \bar{u}}{\partial t} + \nabla \cdot (\bar{u} \bar{u}) &= 0 \\
\frac{\partial \bar{p}}{\partial t} + \frac{1}{Re} \nabla \cdot (\bar{u} \bar{u}) &= 0
\end{align*}
\]

(1)

(2)

where \((u, v, w)\), \(p\), and \(Re\) denote the velocity, pressure, and Reynolds number. Especially, \(\delta\) represents fourth-order central difference, and \(\delta_i\) represents fourth-order interpolation.

2.2. Generation of Turbulent Boundary Layer
Before we calculate the flow around a 3D square
cylinder, we generate turbulent boundary layer. The calculation domain is shown in Figure 1. We give a slight constant pressure gradient in the driver region, and we generate turbulent boundary layer. In this simulation, we place some roughness blocks on the floor, and we change the turbulent characteristics of the inflow. Besides, by using a periodic condition for a main flow direction, we get a fully advanced turbulent boundary layer.

In another calculation domain (computational region), we simulate the flow of a 3D square cylinder in the turbulent boundary layer that generated in the driver region. At this time, we calculate these two regions at the same time. Here, Re number is \( \frac{U_H D}{\nu} \approx 2,000 \). Figure 2 shows generated profile in wind tunnel test and generated profiles in the numerical simulation. The power-law index \( a \) of the measurement\(^3\) (\( Re=12,000 \)) is about \( 1/4 \), and it almost coincide with the \( a \) of the inflow profile of the case 2 in the numerical simulation. Also, the turbulent intensity shows a good agreement with those of the measurement. On the other hand, the power-law index \( a \) of the inflow profile in case 1 equal to \( 1/7 \).

### 3. COMPUTED RESULTS

Figure 3 shows instantaneous pressure contours. We can show small pressure fluctuation on the calculation domain in case 1 and case 2. By the vertical section, we can observe the variation of the pressure distribution on the cylinder surface, and the stagnation point moves above.

In the wake of the cylinder, the downwash in uniform flow is stronger compared with that in the turbulent boundary layer (case 1 and case 2).

Figure 4 shows power spectra of the wind force coefficients. First, the drag coefficient is discussed. The power spectrum in case 2 has high values at low frequency region. This is caused that the approaching flow in case 2 contains the intensive velocity fluctuation. This region represents the buffetting component. Second, we discuss about the lift coefficient. The lift power spectrum by the numerical simulation in uniform flow is higher than that by the wind tunnel test at all region. This reason is that the fluctuating lift coefficient in numerical simulation shows high value in table 1. On the other hand, in the turbulent boundary layer (case 2), the shape of the lift power spectrum looks like that in wind tunnel test, but the position of the peak is different from that in the measurement. Also, we can confirm that the lift coefficients has a broad band spectrum if the inflow contains intensive fluctuation.

We present the other aerodynamic characteristics in table 1. It is almost good coincidence except for the fluctuating lift coefficient in uniform flow.

Figure 5 shows the power spectra of fluctuating...
pressure on the surface of a 3D square cylinder. On the front surface, evident differences could not be observed between the shape of case 1 and case 2. However, case 2 has a slight high power in all frequency region. On the side surface, case 2 has broad peak in comparison with constant flow and case 1. Besides, this peak get to be sharp as the location get to be low. On the back surface, there are no evident differences.

4. CONCLUSIONS

We simulate around a 3D square cylinder in various approaching flow, and we compare the flows and aerodynamic characteristics in simulation with those in the wind tunnel test. Especially, we showed that the power spectra of the fluctuating wind forces and fluctuating pressure changed obviously, if the turbulent component in approaching flow changed.
Figure 6. Power spectra of fluctuating pressure on the surfaces of a 3D square cylinder.

REFERENCES
key words: 3D flow simulation, Turbulent boundary layer, Local wind force
Numerical Simulation of Moderate Reynolds Number Wake past a Square Plate Normal to Uniform Flow

Inwon Lee, Hitoshi Ishikawa and Masaru Kiya

Graduate School of Engineering, Hokkaido University, Sapporo 060-8628, Japan

1. INTRODUCTION

The wake structure around three-dimensional bluff bodies is of prime importance in the understanding of flow-induced fluctuating forces acting on the body. The flat plate normal to the incoming freestream is the simplest bluff body configuration where the wake can be characterized by the fixed separation points at the edge of the plate. Until recently, many experimental numerical studies have been made to delineate the vortical structure in the flow past a two-dimensional flat plate placed in a uniform stream [1,2].

The flow past a three-dimensional rectangular plate, however, has been scarcely dealt with even at low Reynolds numbers, in spite of its importance from the engineering viewpoint. Experimental report on time-mean wake characteristics was made by Fail et al. [3], who did not investigate the wake dynamics. The wake structure is considered to reveal even more energetic, three-dimensional behavior than that of flow past a two-dimensional plate.

In this study, the results from numerical simulations for the flow past a square plate at low Reynolds number will be presented. Emphasis is made on the dynamic behavior of the vortical structure of the wake.

2. NUMERICAL METHOD

In this study, a direct numerical simulation of the wake was made by NAGARE3D.DH software produced by the Institute of Computational Fluid Dynamics Co. Ltd, Tokyo. This software employs the third-order upwind-difference scheme of Kawamura and Kuwahara [4].

Fig. 1 Schematic Diagram of computational domain

Figure 1 illustrates the computational domain for the square plate case. The square plate has the thickness of H/3, H being the length of one side of square. Rectangular uniform grid with the resolution of 129×65×65 was used and the time step was automatically determined after the CFL condition. As shown in Fig. 1, the coordinate axis is located on the
centroid of the front face of the square plate. The Reynolds number based on \( H \) was 200. Uniform flow with velocity \( U_0 \) at the inlet plane was impulsively started from the rest. After initial transients ended at \( tU_0/H = 80 \), time-mean statistics was evaluated. The whole numerical simulation proceeded then to \( tU_0/H = 375 \).

3. RESULTS AND DISCUSSION

The time history of drag coefficient is given in Fig. 2, along with the time history of base pressure coefficient. It is worth mentioning that the skin friction drag on the side faces was considered in the evaluation of the drag coefficient, since the plate in the present simulation has finite thickness. It is seen that the initial transient disappears after \( tU_0/H = 80 \). It is notable that both the drag coefficient and the base pressure coefficient exhibit periodic component with period of \( TU_0/H = 50 \).

![Figure 2. Time history of drag coefficient](image)

This unsteadiness might seem to result from the periodic vortex shedding. However, the time history of streamwise velocity component taken at \((y,z) = (0.5H,0)\) and \((0,0.5H)\) at \(x/H = 4.0, 6.0\) and \(8.0\) in Fig. 3 reveals another faster periodic component with period \( TU_0/H = 6 \). A Closer examination of Fig. 2 indicates that there is small, faster component in the drag coefficient. Moreover, the velocity time history shows notable low-frequency modulation whose period is comparable to that of the aforementioned slow component in drag coefficient.

![Figure 3. Time history of streamwise velocity](image)

![Figure 4. Time history of average vorticity](image)

In order to examine more closely the relation between the two unsteadiness, the time history of \( yz\)-plane averaged vorticity vector with \( x/H = 2.0, 4.0 \) and \(8.0\) is plotted along with the base pressure coefficient in Fig. 4. Here, the vorticity corresponds to the magnitude of the vorticity vector. From this vorticity time history at \( x/H = 8.0\), it becomes clear that the faster component corresponds to the 'shedding' of the vortical structure. In addition, the
Strouhal number based on this period gives $St = 0.16$, which is comparable to $St = 0.115$ reported by Fail et al.[4], although the direct comparison is not so meaningful because the latter is a high Reynolds number result with $Re = 3 \times 10^5$.

From Figs. 2 and 4, it is found that there exists a certain relationship between the drag coefficient and the averaged vorticity. When the drag becomes high and the base pressure becomes low, the vorticity at downstream locations, i.e., at $x/H = 4$ and 8 shows energetic shedding behavior. On the contrary, low drag phase is associated with relatively quiet shedding, i.e., small magnitude of shedding component. This tendency is clearly demonstrated in Figs. 5 and 6, where the isosurfaces of vorticity is illustrated. In Fig. 5, the formation is hairpin vortices is clearly discernible. Thus, the energetic vorticity variation results from the passage of coherent hairpin vortices. In Fig. 6, however, the vortical structure is not clear, which manifests small magnitude of unsteadiness in vorticity.

It is quite notable that the above-mentioned low-frequency modulation of drag is associated with the variation of vortical structure. Najjar and Balachandar [2] reported very similar behavior in case of two-dimensional plate, where the drag coefficient undergoes a low-frequency modulation. According to their investigation, the phase of high drag (termed ‘Regime H’) is characterized by shorter separation bubble and coherent spanwise/streamwise vortical structure, while ‘Regime L’ is denoted by longer bubble length and unorganized vortical structure. In spite of the large difference in the geometry, this virtually identical behavior between the two cases is remarkable. Najjar and Balachandar further attributed this low-frequency unsteadiness to the resultant phase imbalance between the formation cycles of the spanwise and the streamwise vortices.

In order to further investigate the dynamics of vortical structure, the overall inclination of vortical structure, i.e., the direction of hairpin vortices is examined. From Fig. 5, it is seen that the direction of each hairpin vortex is not uniform. Namely, one in the downstream points almost to vertical ($+z$) direction, while the upstream one to oblique direction. Thus, a certain vibration or rotation of hairpin vortices seems to exist. The orientation angle of vortices $\theta$ in a certain $yz$-plane is evaluated from the first moment of the vorticity as follows:

$$M_y = \iint \omega y \, dy \, dz, \quad M_z = \iint \omega z \, dy \, dz$$

$$\theta = \arctan \left( \frac{M_z}{M_y} \right)$$

Figure 7 and 8 show the Rissajous curve defined from $M_y$ and $M_z$, the time history of $\theta$ and the
histogram of time fraction of the quadrant occupation of vortices at $x/H=2.0$ and 8.0, respectively. As shown in these figures, the orientation angle $\theta$ is seen to undergo a low-frequency modulation. In addition, in the near wake at $x/H=2.0$, vorticity is mainly concentrated in the third quadrant of $(M_y, M_z)$, i.e. the third quadrant of $yz$-plane. When flow goes downstream at $x/H=8.0$, this tendency has been somewhat relaxed and the histogram indicates near-uniform distribution.

Direct numerical simulation of a flow past a square plate normal to uniform flow has been performed with Reynolds number of 200. The time dependence of drag coefficient and average vorticity exhibited a low-frequency modulated behavior of vortex shedding characteristics. The observed vortical structure demonstrates remarkable resemblance with two-dimensional normal plate case. Along with many previous studies, the low-frequency unsteadiness is again considered as an universal feature of separated shear flow regime.

Reference
3) Fail, R., Lawford, J. A. and Eyre, R. C. W., 1959, Low-Speed Experiments on the Wake Characteristics of Flat Plates normal to an Air Stream, NACA Reports No.3120
Generation of inflow turbulence and its effects on a wind flow around a tall building

Kojiro NOZAWA and Tetsuro TAMURA

1) Izumi Research Institute, Shimizu Corp., Tokyo 100-0011, Japan
2) Tokyo Institute of Technology, Yokohama 226-8502, Japan

1 INTRODUCTION

In predicting the unsteady pressure force on a bluff body immersed in the atmospheric boundary layer using Large eddy simulation (LES), it is required to generate time series of unsteady inflow data. Recently, several methods of generating inflow data were proposed, but none of them satisfied the characteristic of the real atmospheric turbulence. It follows from this that there has been only a few cases to simulate flows around a bluff body with oncoming turbulence[1]. The objective of this study is to propose the method of generating a turbulent boundary layer under zero pressure gradient which evolves over a rough wall, and to study the effects of oncoming turbulence on wind flow around a tall building. Turbulence of a generated boundary layer were investigated comparing with the atmospheric and laboratory data. To estimate the effects of oncoming turbulence, a square prism (height:width = 4B:B) was settled on a floor and features of flow around a prism was studied. The Reynolds number of this simulation was high enough compared with most Reynolds number studied in wind tunnels.

2 GENERATION OF TURBULENT INFLOW DATA

A method of inflow generation which accounts for spatial growth under zero pressure gradient with periodic boundary conditions was proposed by Spalart and Leonard[2], and then developed by Lund et al. [3] to optimize to a Cartesian coordinate system. In this paper, the method of Lund, which is orginally limited to a turbulent boundary layer over smooth surface, was advanced to a turbulent boundary layer over rough surface.

2.1 Advancing the Lund's method to a rough surface

Figure 1: Schematic of generating turbulent boundary layer.

In the Lund's original method, the velocity at the inlet plane is estimated based on the solution downstream and outflow boundary condition is applied at the exit boundary (see fig.1). The velocity at the downstream section is rescaled and re-introduced to the inlet. In the method, the mean flow is rescaled assuming the law of the wall in a wall region and the defect law in a outer region. Both momentum thickness θ and friction velocity \( u_r \) at the inlet and the downstream section determines scaling parameters in the rescaling procedure. At the downstream section both
quantities can be determined by the mean flow, and $\theta$ at the inlet is specified value.

In advancing the method to a rough surface, $u_r$ at the inlet must be determined by the quantities according to a suitable relation on a rough wall. In this study, $u_r$ at the inlet is determined using the experimental formulae of Prandtl et al. [4], which was approximated to a form as follows.

$$\frac{\theta_b}{\theta_a} - 1 = \frac{c'_f(x_a)}{2\theta_a} \left( \frac{u_r b}{u_r a} \right)^{2\gamma c} - 1$$

where, $r = 3.95/(2.87 + 1.58\log(x_a/k_s))$, $c'_f$ is the local skin friction, $x$ is the distance from a leading edge of a surface, $k_s$ is the equivalent sand roughness. The subscripts $a$ and $b$, indicate the variables at the inlet and the downstream section.

To realize the effects of roughness, rectangular blocks were staggered on a surface (see fig.5). From a standpoint that the loss of bulk momentum around a roughness block must be a key factor to realize a turbulent boundary over a rough wall, the cubic interpolated pseudo-particle (CIP) method [5][6] was applied to the convection terms in vicinity of blocks. By applying the CIP method, the momentum transfer could be evaluated exactly without concentrating grid points around a block. The top surface of a block was expressed only by $6 \times 8$ grid points with same spacing.

2.2 Turbulent characteristics of inflow data

The mean profiles of velocity and turbulence intensities are shown in fig.2. The thickness of boundary layer $\delta$ was about $2/3$ of the height of a computational domain. The Reynolds number based on $\delta$ and free stream velocity $U_\infty$ was about 110,000. The mean velocity profile fitted with $1/5$ power-law. The streamwise turbulence intensity was 11% at $z/B = 4$, and 19% at $z/B = 1$ (see fig.3). The power spectra of fluctuating velocities are shown in fig.4. They agreed well with the $-5/3$ power-law spectrum of Kolmogolov and they reduced the power at higher frequency due to the filtering of LES. Figure 5 shows instantaneous isosurface of second-invariants of deformation ten-
sor, complicated vortical structures could be seen formed in the wake of roughness blocks.

3 FLOW AROUND A TALL BUILDING

3.1 Numerical method

A flow around a rectangular prism was simulated using LES with turbulent inflow generated in the previous section. The inflow data was taken from the downstream section. The computational domain was decomposed to five sub-domains to concentrate grids around the prism and the floor without increasing number of grids (see fig. 6). The prism was submerged in a turbulent boundary layer and the ratio of the height \( H \), and \( \delta \) is about 0.6. The 3rd order upwind scheme was introduced to the convective term only to the area vicinity to the prism. The grid spacing upstream of the prism is smaller than the grid spacing used in generating the turbulent inflow. Any roughness block was not mounted around the prism. The statistical data were estimated during the period forty pairs of Karman type vortices shed.

3.2 Effects of oncoming turbulent on a flow around a prism

Figure 7 shows the pressure distribution on the surface of the prism. Mean pressure at the centerline on the front surface distributed due to the mean dynamic pressure of oncoming flow. We could see a quick pressure recovery at the leeward corner of the side near the ground. This might be caused by the reattachment of the flow separated from the windward corner and these characteristic can be seen in the experimental data[7]. On the back surface of the prism, we could see the area near the corner at the middle of the prism, where the fluctuation of pressure was strong. On the side fluctuation of pressure was large at the lee-ward corner of the middle part of the prism and it decreased at lower and higher part of the prism. At the lee-ward corner, there were two peaks of fluctuating pressure (see fig. 8), and peak frequency of higher one was 3–4 times higher than Strouhal number caused by Karman type vortices. This feature could not be seen in the flow with laminar inflow.

Figure 9 shows the time averaged streamlines on the surface of the prism. On the front surface a divided line could be seen at the height \( z/B = 3.3 \). On the back, there was a divided line at \( z/B = 2.1 \), and it indicated that a downwash flow from the top of the prism reattached to the back surface at the height. On the side, the flow separated from the windward corner reattached near the leeward corner, and this might cause the higher frequent pressure fluctuation on the leeward corner.

Reference


[2] P. R. Spalart and A. Leonard. Direct numer-
(a) mean pressure coefficient

(b) r.m.s of fluctuating pressure coefficient

Figure 7: Pressure distribution on the surface of the prism.

Figure 7: Pressure distribution on the surface of the prism.

Figure 8: Spectra of pressure fluctuation on the surface of the prism at $z/B=2.75$.

Figure 8: Spectra of pressure fluctuation on the surface of the prism at $z/B=2.75$.

Figure 9: Time averaged streamlines on surface of a prism.

Figure 9: Time averaged streamlines on surface of a prism.


key words: generation of turbulent inflow, rough surface, CIP method, square prism
Wind loads on free-standing walls with 45 degree top pitch

O.O. Badran1)*, A.E. Hold2, and J. Davies2)

1) Al-Balqa' Applied University, Department of Mech.Eng, P.O.Box 330116, Amman 11133 – Jordan
2) CFD Group, Faculty of Engineering, University of Hertfordshire, Hatfield, HERTS AL10 9A, U.K.

1. INTRODUCTION

Wind flow around free-standing walls is important for the guidance of civil and structural engineers, architects and builders, deals with the design and use of plain and reinforced free-standing walls not forming part of a building (1). There is very little information available on the sensitivity of results of wind flow on free-standing walls at fixed inlet boundary condition to the variability in turbulence models. Most of the studies performed on the free-standing walls used a wind tunnel measurements and the needs of CFD studies has risen recently over determination of the wind loads on free-standing walls. The geometry chosen was a wall model of a height of 5 m and 0.5m thickness with 45° top pitch, which represent the dimensions of a free-standing wall as shown in figure 1. The computational fluid dynamics CFD package FIDAP (version 7.6) has been used to solutions for two-dimensional walls of finite thickness (D) to height (h) ratio D/h =0.1 (figure 1). This ratio is approximately equivalent to the full-scale walls described by Robertson et al (2). Cases were run for h = 5.0 m and a boundary layer wind profile with roughness thickness of 0.015m for turbulent flow simulations using the two k-ε turbulence models.

2. MESH AND BOUNDARY CONDITIONS.

The solution domain extended 16h upstream, 16h downstream and was 16h height. It was decided to generate a rectangular section mesh for the model with the wall of the ratio (D/h) = 0.1 at its center figure 2. It should be noted also that, the conditions of numerical boundary condition are made as identical as possible for the simulations of the two turbulence models.

* Corresponding author, Telefax: 4613452, email: <o_badran@yahoo.com>
2.1 IN-FLOW BOUNDARIES.

In all turbulence models the inflow values of U velocity are set to zero while the length scale of longitudinal turbulence L for natural wind of the inflow boundary is approximated from the following empirical equation:

\[ L(z) = 25Z^{0.35}/Z_0^{0.063} \]

Where \( L(z) \) denotes the length scale of the velocity component \( U_0 \) in the x (or flow) direction at height \( z \). Also the \( \varepsilon \) value can be obtained from the following equation

\[ \varepsilon = \frac{0.09 \cdot k^{3/2}}{0.1 \cdot L(z)} \]

The turbulence intensity \( I(z) \) of natural wind at height \( z \) at inflow can also be approximated by an empirical equation, which is given by

\[ I(z) = 1 / \log_5(10z) \]

2.2 OUT FLOW BOUNDARIES.

Zero gradient out flow boundaries are used for all the variables.

2.3 SOLID BOUNDARIES.

Both normal and tangential velocity values are set to zero at solid boundaries.

3. TURBULENCE MODELS

The numerical simulations presented in this paper were obtained by solving the Navier-Stokes equations in two dimensions. The turbulence modeling enhancements in FIDAP 7.6 used in this paper includes: The two new two-equation turbulence models are the Renormalization Group (RNG) \( k-\varepsilon \) turbulence models, the \( k-\omega \) model of Wilcox.

The RNG \( k-\varepsilon \) turbulence model

The RNG \( k-\varepsilon \) turbulence model employed in FIDAP is that developed by Yakhot et al (3). The form of the \( \kappa \) and \( \varepsilon \) of the RNG \( k-\varepsilon \) model is as follows:

\[ \rho \frac{\partial k}{\partial t} + \rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + G - \rho \varepsilon \]

The model transport equations for \( k \) and \( \omega \) in Wilcox's model (4) are as follows,

\[ \rho \frac{\partial k}{\partial t} + \rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + G - \rho \omega \]

4. RESULTS AND DISCUSSION.

The objective of this section is to give an idea of the main structural features of the flow field around the free standing wall, with particular emphasis on the reverse flow, shear flow and reattachment regions.

4.1 VELOCITY VECTORS AND STREAMLINE.

The mean flow fields around a free-standing wall given by \( k-\omega \), and RNG models are compared in terms of mean velocity vectors, streamlines, and pressure contours. Numerical analysis with FIDAP 7.6 enable the evaluation of the two velocity components \( \bar{U} \) and \( \bar{V} \), and therefore the two-dimensional mean velocity vector \( \bar{V} \) at each node presenting the mean velocity results as vector plots \( \bar{V} = (\bar{U}, \bar{V}) \) will give an overall description of the direction and magnitude of the mean flow field. The velocity vectors fields for the leeward face region of the wall of the various models are shown in figures 3a and b. From these figures several features can be identified. The separated flow region has a different horizontal and vertical locations behind the wall, it can be noticed that for the RNG models the recirculating region is spreading far away downstream the leeward face (12h) and at higher vertical location, and two separated flow
regions formed divided by a region of a moderate positive values. Also the RNG model shows an existence of various sharp velocity gradients in the streamwise direction as well as in the vertical direction than the other model (Fig 3a).

Figure 3a. Velocity vectors field for RNG model at leeward face.

Figure 3b. Velocity vectors field for k-\(\omega\) model at leeward face.

While for the k-\(\omega\) model the separated flow region formed near the leeward face of the wall at lower vertical location as shown in figure 3b. A significant change is observed in the size and strength of the corner vortex occurred near the leeward corner which has the opposite sense to the main reverse flow between the two models, this probably caused by a change in size and strength of the main recirculating flow regions for each model. Both models showed that a small reverse flow appears on the roof of the wall at leeward side. Also they showed that beyond the reattachment point a new-sub-boundary layer started to develop which will develop into a fully turbulent boundary layer condition. To illustrate the differences between the various models at the windward face the velocity vector plot for those models shown in figures 4a and b. The following feature was observed: near the windward corner, it is observed that the corner vortex has different sizes, reflecting changes in the strength of the corner eddy.

4.2 PRESSURE DISTRIBUTION

Many researchers [5,6] admitted that the non-linear k-\(\varepsilon\) model, which used for two-dimensional solutions, could give a good approximation to the pressure distributions measurements. A study on the phenomena of the pressure distributions using various turbulence modeling is reported here to provide a clear understanding of flow behavior discussed in previous sections Figures 5 a and b. For the comparison purposes the contour levels for the various models were kept similar. From the comparison it can be noticed that the lowest contour levels of the pressure, -1.03 to -0.6, which occurred in the suction region downstream of leeward face between 1.5h to 2h for RNG model are nearly twice lower than the values of the k-\(\omega\) models. On the roof of leeward face at which steady reverse flow occurred the pressure value for k-\(\omega\) model is increased by about 40% compared with the RNG model, while on the windward side of the wall (pressure side) the pressure values were nearly similar for both models with a positive pressure contour values of nearly about (0.2 to 0.7). At the roof of the windward face the pressure contour of high positive values were observed to occur in k-\(\omega\) model which indicate significantly higher loads than have been detected in other model. These high values are may be due to the k-\(\omega\) enhancing the flow upstream the windward face.

5. CONCLUSIONS

The numerical simulations have shown significant differences between the recirculating regions associated with the two turbulence models studied. Also it can be observed from the results that the size of recirculating region is strongly dependent on the model used, with a quite evident growth of the recirculation bubble size from model to another, so that the present studies have shown that the real flow behavior can be very sensitive to different turbulence model. The pressure results pre-
sented, showed that the results of pressures on the windward face to be little affected by using various models but the suctions on leeward face varies significantly with different models due to different recirculation flow sizes. The data obtained are useful for the understanding of the physics of separated flow behind free-standing wall and for testing in the experimental studies.

References
1- J.O.A. Korff, Design of free standing walls. The Brick Development Association, U.K.

Key words: CFD, separation, turbulent flow, wind load, free-standing wall