Ali A. Mahamied. Int. Journal of Engineering Research and Application *www.ijera.com*
ISSN : 2248-9622, Vol. 6, Issue 11, (Part -1) November 2016, pp.84-88

RESEARCH ARTIC OPEN ACCESS

Calculation of Fluid Dynamic for Wind Flow around Reinforced Concrete Walls

Ali A. Mahamied , Amjad A.Yasin ,

ABSTRACT:
A study on the flow phenomena around free-standing walls is important in practical building construction. In the present paper a numerical study is conducted for two- dimensional incompressible steady flow around free-standing walls using low-Re k-co turbulence model. The separation regions downstream the wall and on the roof of the leeward were predicted. Finally, results of numerical simulation are presented in the form of velocity vectors, velocity contour, pressure contours and streamlines.

Keywords: *Fluid dynamic, wind flow, two- dimensional incompressible steady flow, walls*

I. INTRODUCTION

Wind loads on free-standing walls are important because of their poor performance under wind loading. For example, falling walls in wind stomis have been responsible for approximately one death per year on average in the United Kingdom (Letchford and Holmes 1994, Letchford 1986, Holmes 1985).

In addition, free-standing walls (F.S.W) are widely used for boundary demarcation, land escaping, screening, security, and noise barriers on urban motorways and railways to improve the quality of the built environment. When properly constructed, free-standing walls have proved to be extremely durable and pleasing in appearance. In most cases, a durable and stable wall will cost little more than a sub-standard one. All that is required is the selection of suitable materials combined with an efficient arrangement of brickwork. More often than not, free-standing walls are exposed to the full effects of the weather. The action of wind, as a lateral force, is catered for by the strength design. But, the combined effect of rain and driving wind requires the use of suitable materials and correct constructional details. Most F.S.W are constructed of brick and are designed following the guidance contained in the Brick Development Association document “Design of free-standing walls” (Korff 1984).

Lack of a quantitative theory to explain airflow near wind barriers in a turbulent boundary layer has hindered theoretical and experimental programs in barrier research and made optimum barrier design code for practical applications difficult. The main objective of the present study was to develop a quantitative, theoretical simulation of airflow around freestanding walls, to detect the flow behavior of the turbulent shear layer and its reattachment length, to be used as a reference to the present and future pressure design guide.

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 full scale 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 1995. Case was run for h = 5.0 m and a boundary layer wind profile with roughness thickness of 0.015m for turbulent flow simulation using the low-Re k-ω turbulence model, FIDAP allows the computation of flows over smooth and rough surfaces. This property of CFD package used to apply the required surface roughness characteristics to a portion of the computational boundary (FIDAP 1995).

II. 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 2a. One of the main features of the present mesh is that the individual elements are graded in size in various directions from certain key points. The reasoning behind this grading is in order to make the elements smaller in regions of interest as shown in Figure 2b.

Boundary Conditions

For modeling free-standing walls, where only a two dimensional section throughout the wall is investigated, a uniform U_0 velocity profile is used at inflow following the log law as in figure 3. In this turbulence model the inflow values of V 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:

$$L(z) = (25 Z^{0.35})/(Zo^{0\,063}) \qquad (1)$$

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

$\varepsilon=(0.09 * \underline{k}^{3/2})/(0.1 * L(z))$ (2)

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

$I(z) = 1/\log_e(10Z)$ (3)

In the above, k is the turbulence kinetic energy and s its rate of dissipation.
Both normal and tangential velocity values are set to zero at solid boundaries. All the free stream boundaries are placed far enough from the obstructions, so that these boundaries have no effects upon the flow inside the flow domain.

III. TURBULENCE MODEL

The numerical simulation presented in this paper was obtained by solving the Navier- Stokes equations in two dimensions using a finite element method. The new two-equation k- co turbulence model of Wilcox has been chosen for the present study. The high efficiency and accuracy of this model has been clearly confirmed when applied to such simple flow field (FIDAP 1995).

Wilcox's Low-Re k-□ Turbulence Model

The k-ω model belongs to the so-called two-equation group of turbulence models and is therefore closely related to k-ε type turbulence models. The particular version of the ***k-ω*** model that is employed in FIDAP was developed by David Wilcox 1993, and for this reason we will refer to it as Wilcox's ***k-ω*** model. In this model the turbulent scales $\boldsymbol{u_t}$ and $\boldsymbol{\delta_t}$ are related to ***k*** and a turbulent frequency ω through the following expressions resulting from dimensional arguments:

$u_t\ \alpha\ k^{0.5}$; $\delta_t\ \alpha\ k^{0.5}/\omega$ (4)

The turbulent frequency ***ω*** is itself related to k and s via the simple expression ***ε= ω k***. The turbulent viscosity is obtained from equation 5,while the model transport equations(5 and 6) for ***k*** and ω in Wilcox's model are as follows,

$$\mu_t = c_\mu \rho \frac{\kappa}{\omega} \qquad (5)$$

$$\rho\frac{o\kappa}{\alpha} + \rho u_j \frac{ok}{\partial\chi_j} = \frac{\partial}{\partial\chi_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{ok}{\partial\chi_j}\right] + G - \rho\omega k \qquad (6)$$

$$\rho\frac{\partial\omega}{\alpha} + \rho\mu_j \frac{\partial\omega}{\partial\chi_j} = \frac{\partial}{\partial\chi_j}\left[\ \left[\mu + \frac{\mu_t}{\sigma_\omega}\right]\frac{\partial\omega}{\partial\chi_j}\right] + c_1\frac{\omega}{\kappa}G - c_2\rho\omega^2 \qquad (7)$$

The values of Wilcox's model constants are tabulated below:

Cl	C2	$^{C}\mu$	□k	□□
0.555	0.83333	0.09	2.0	2.0

IV. RESULTS AND DISCUSSION

This section will concentrate on the flow behavior around the F.S.W with particular emphasis on the reverse flow, shear flow and reattachment regions, to draw the best possible conclusions, and to pose some of the new problems encountered during the course of the investigation to which future work may be directed.

Velocity vectors and streamline

Numerical analysis with FIDAP 7.6 enabled the evaluation of the two velocity components $\underline{U}$ and $\underline{V}$, and therefore the two-dimensional mean velocity vector $\underline{V}$ at each node, presenting the mean velocity results as vector plots $\underline{V} = \underline{(U, V)}$ 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 are shown in Figure 4. From this figure several features can be identified. The separated flow region has been created behind the leeward side from **l h to 5 h** (see x-velocity contours). A secondary recirculation, in the opposite sense to the main reverse flow, occurs in the comer, at the bottom of the wall. It is observed that the horizontal distance of the comer recirculation is about 0.7h and the vertical height is about 0.5h. Also the present model showed that a small reverse flow appears on the roof of the wall at leeward roof slope. Beyond the reattachment point (5h) a new-sub-boundary layer started to develop which will develop into a fully turbulent boundary layer condition. Figure 5 shows the velocity vector plot for the windward face. The following feature was observed: near the windward comer, it is observed that the comer vortex has formed. Also from figure 5 it can be confirmed that there is no evidence of flow separation on the windward roof slope.

The data can also be plotted in the form of iso-contours for Longitudinal component at leeward and windward faces, as shown in figure 6, which highlight the spatial development of the x-velocity component magnitude in terms of contour level values, and to facilitate the discussion of the velocity vectors plots. For example, the negative contour levels with lowest negative values of -0.7 to -0.08 occurred in the reverse flow region in the suction side of the wall between lh to 5h. While the low magnitude of positive contour levels of 0.1-1 occurred in the pressure side of the wall (windward side). It can be seen from the x-component contour plot (figure 6) of the k-co model that the reattachment length has been occurred at 5h. It is observed that the deflection of the outer flow is increased significantly at axial distance 2h~4h, due to the effect of separation bubble underneath this region. The isometric contours of the x- velocity (Figures 6) clearly separate the comer recirculation region from the main flow and demonstrate how the mean velocity varies in both the longitudinal and traverse directions.

Another good feature of CFD is its ability to compute the streamlines. An example is shown in figure 7 of flow pattern around F.S.W, 45'roof pitch wall. The figure shows that the developments of the separated flow region traversely and horizontally behind the leeward sides, the separation flow region is seen to extend from lh to approximately 5h.

Pressure distribution

The use of pressure contour plot can provide a clear spatial view of the distribution of the pressure values, and the development of the suction and pressure part of the pressure forces on the free-standing wall. The pressure distributions predicted by CFD modeling using k-w turbulence models for the F.S.W are shown in figure 8. From Figure 8 it can be noticed that the lowest pressure contour levels of about -1.7~ -1.1 occurred on the leeward roof slope, While the values of -1 to -0.2 occurred in the suction region from leeward side up to downstream distance 4h, which having steady reverse flow. At the windward side of the wall (pressure side) and on its roof slope the positive pressure contour values of nearly 0.09 to 0.4 occurred.

Pressure coefficient

Strong interest has risen recently in determining reliable pressure coefficient data for use in the design of free-standing walls, also recent developments in the wind loading codes for the UK, Australia and Europe have introduced new, more onerous, pressure coefficient data for the design of free-standing walls (Robertson et al 1995). The pressures on wall surfaces are presented in terms of pressure coefficients, i.e. Pressures normalized by upstream dynamic pressure at wall height (h=5m). The distributions of pressure coefficients for the wall are shown in Figure 9. It can be seen from figure 9 that the positive pressure coefficients (1- 1.07) occurred at the windward presenting a high wind load on this side, while the negative pressure values (-0.75- -0.67) occurred at the leeward side of the high suction region, which gives the evidence of a separation effect on pressure. Generally, it has been found that CFD can provide a useful aid to the interpretation of the relationships between any structure shape and pressure distributions around it (Badran et al 2001, Hoxey et al 1993, Ferziger 1993).

V. CONCLUSIONS

We have presented the results of some recently developed turbulence model (i-e k-ω) with an emphasis on applications connected with wind engineering by using free-standing wall. Recent developments in turbulence models for small scales give promise that simulation methods will be available for selected engineering applications in the not -too - distant future.

The present model predicted accurately the small separated flow region on leeward roof slope, large separated region behind the wall, and the comer recirculation's. The data obtained are useful for the understanding of the physics of separated flow behind free-standing wall and for testing the experimental studies.

REFERENCES

[1]. Badran, 0.0. Davis, J.B and HoldO, A.E. (2001), “Wind loads on rectangular free-standing walls”. Proceedings of the 2001 ASME pressure vessels and piping conference, PVP-Vol. 431, Fluid-Structure Interaction, Atlanta, Georgia, USA.
[2]. Ferziger, J. (1993), “Simulation of complex turbulent flows: recent advances and prospects in wind

engineering”. *J. Wind Eng. and Ind. Aerodyn,* Vol.46&47,195-212.
[3]. Fluid Dynamics International Inc. (1995), “FIDAP 7.06 Users Manual”.
[4]. Holmes, J.D. (1985), “Wind loading on free-standing walls at CISRO”. CISRO Division of Building Research, Melbourne, Internal Report 85/13, 1-30.
[5]. Hoxey, R.P. Robertson, A.P. Basara, B. and Younis, B.A. (1993) Geometric parameters that affect wind loads on low-rise buildings:Full-scale and CFD experiments, *J.Wind Eng. and Ind.Aerodyn.,* Vol.50, 243-252.
[6]. Korff, J. (1984), “ Design of free-standing walls”. Brick Development Association, Windsor.
[7]. Letchford, C.W and Holmes, J.D. (1994), “ Wind loads on free-standing walls in turbulent boundary layers”. *J. Wind Eng. and Ind Aerody.,* Vol.51, 1-27
[8]. Letchford, C.W. (1986), “Further studies of wind loads on walls”. Department of Engineering science, University of oxford, Report OUEL 1620/86,4-20.
[9]. Robertson, A.P., Hoxey, R.P. and Richards, P.J. (1995), “Design code, full-scale and
[10]. numerical data for wind loads on free-standing walls”. *J. Wind Eng and IndAerodyn.,* Vol.57, 203-214.
[11]. Wilcox, D.C. (1993), *Turbulence modeling for CFD,* DCW Industries, Inc.

Fig. 3. Inlet velocity profile.

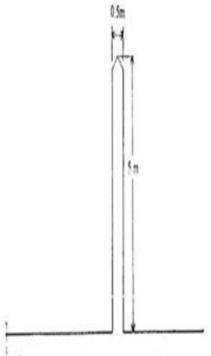

Fig. 1. Free-standing wall geometry

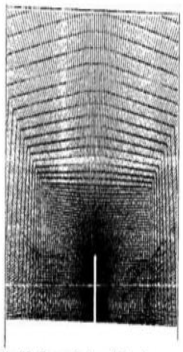

Fig. 2b. Zoomed view of the elements

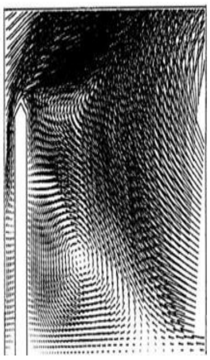

Fig.4. Velocity vectors field for k-ω model at leeward face

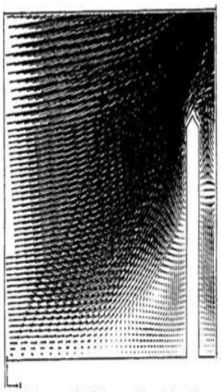

Fig.5. Velocity vectors field for k-ω model at windward face

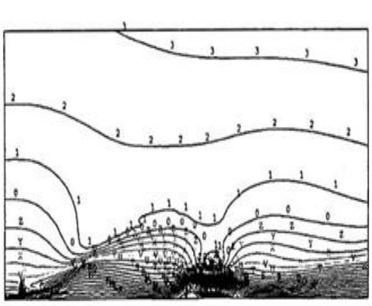

Figure 6. Isometric contours of x-velocity component for k-ω model

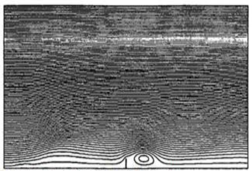

Fig. 7. Isometric contours of streamlines for k-ω model

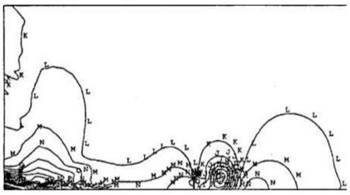

Fig. 8. Distribution of pressure contours for k-ω model

Fig. 8. Distribution of pressure contours for k-ω model

Error!

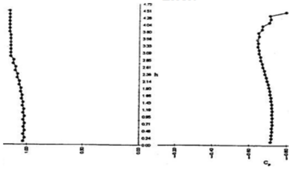

Fig. 9. pressure coefficients of k-ω model

AUTHOR's PROFILE

1. Ali Abdel-Fattah Mahamied was born in Amman 1/8/1960 Work in Facultyof Engineering Technology / Al-Balqa' Applied University Amman – Jordan

General Secondary Certificate (Scientific Stream) , Wadi–Seer Secondary School, Jordan . With Grade Total 737 out of inistry of Education, 1979 – Jorda Diploma (Associate degree) in Civil Engineering, February 1982,B.Sc. degree in “Applied Engineering” with major in “Highways, Bridges and Airport”/ Civil engineering department, June 2000,Al-Balqa Applied University, Amman College for engineering technology. M.Sc. degree in Civil engineering "Structures" February 2005, University of Jordan . Member of the Jordanian Engineers Association (JEA)
Tel. 962-6-4892345
962-6-5481595

2. Dr. Amjad A. Yasin, was born in Amman-Jordan at May 30, 1963, his education is as follows: Ph.D. Civil Engineering –Structures, University of Jordan-Jordan. M.Sc. Civil Engineering- Structures, University of Jordan-Jordan, B.Sc. in Civil Engineering,Yarmouk University -Jordan .His occupation is Assistant Professor in Structures, Civil Engineering Department, Faculty of Engineering Technology (FET),Al-Balqa’ Applied University (BAU) Amman-Jordan. He was head of civil engineering department in the academic year 2006/2007,He has published seven researches in different aspects related to his specialization mainly concentrated to concrete properties, Dr. Yasin is also chairman of many committees in the Jordanian Engineering Association (JEA) and also was member in preparation committees for many conferences held in Amman-Jordan under the supervision of JEA.

The mailing address is: Department of Civil Engineering Faculty of Engineering Technology.
AlBalqa’ Applied University .P.O Box 15008 Amman 11134, Jordan .Tel. 962-6-4790333.Mobile 00962-777 366126 and 00962790900194