Wind and Structures, Vol. 5, No. 2~4 (2002) 337-346
DOI: http://dx.doi.org/10.12989/was.2002.5.2_4.337

A comparative investigation of the TTU pressure envelope. Numerical versus laboratory and full scale results

S. A. Bekele† and H. Hangan‡

The Boundary Layer Wind Tunnel Laboratory, University of Western Ontario, London, Ontario, Canada

Abstract. Wind tunnel pressure measurements and numerical simulations based on the Reynolds Stress Model (RSM) are compared with full and model scale data in the flow area of impingement, separation and wake for 60° and 90° wind azimuth angles. The phase averaged fluctuating pressures simulated by the RSM model are combined with modelling of the small scale, random pressure field to produce the total, instantaneous pressures. Time averaged, rsm and peak pressure coefficients are consequently calculated. This numerical approach predicts slightly better the pressure field on the roof of the TTU (Texas Tech University) building when compared to the wind tunnel experimental results. However, it shows a deviation from both experimental data sets in the impingement and wake regions. The limitations of the RSM model in resolving the intermittent flow field associated with the corner vortex formation are discussed. Also, correlations between the largest roof suctions and the corner vortex "switching phenomena" are observed. It is inferred that the intermittency and short duration of this vortex switching might be related to both the wind tunnel and numerical simulation under-prediction of the peak roof suctions for oblique wind directions.

Key words: RSM turbulent model; pressure peak values; corner vortex; TTU building.

1. Introduction

The high suctions on the roof corner of a low-rise building induced by oblique wind's have been associated with the formation of conical vortices. The mechanism and condition of occurrence of large peak suctions has been experimentally studied Kawai and Nishimura (1996), Wu, Sarkar and Mehta (1999), Taniike and Taniguchi (1999), through wind tunnel pressure measurements and flow visualizations. The TTU experiment generated valuable full-scale data, which constitutes a benchmark for further wind tunnel and numerical investigations (Surry 1991, Cochran and Cermak 1992, Mochida, Murkami, Shoji and Ishida 1993, Selvam 1992, Tieleman, Surry and Metha 1996, He and Song 1997, Banks *et al*. 2000). Also, numerical studies of the pressure and velocity fields add to the understanding of the formation of these vortices in the separation zone, (Lakehal and Rodi 1997, Thomas and Williams 1999).

The numerical study of bluff body generated complex flow fields, with high-pressure gradients and multiple circulation regions, requires a careful choice of the turbulence models, Murakami (1993). For low Reynolds numbers an accurate analysis of simple (generic) bluff body related flows

† Research Assistant
‡ Assistant Professor

can be obtained via Direct Numerical Simulations (DNS). However, for wind engineering related problems the high Reynolds number flows and rather complicated geometries eliminate DNS as a suitable option at present time. LES(large eddy simulations) have increasingly become the trend in resolving some of the fundamental problems in bluff body aerodynamics (Rodi 1993, Shah and Ferziger 1997, Yu and Kareem 1997).

The requirement for significantly higher computing resources implies that the use of LES should be justified by showing the inadequacy of RANS turbulent models to handle particular aspects of the flow fields involved. The Reynolds Stress Model (RSM) has been tested for the generic problem of a cube for mean pressure field in boundary layer and reasonable results have been obtained (Wright and Eason 1999). Also Franke and Rodi (1993) concluded that within the RANS based models, full Reynold stress models are best for the case of an isolated cylinder at *Re* = 22,000. Following these arguments as well as taking into consideration the marked anisotropy of the corner vortex problem the RSM model is chosen here for the investigation of the pressure field on the TTU building. Comparison with full scale, wind tunnel and previous numerical work are presented.

2. Numerical setup

2.1. Grid

The computational domain has the dimension of 5 H as a vertical height from the ground, 20 H a downstream length, 10 H an upstream length and a width of 17.5 H, where H is the full scale building height of 4 meters, Fig. 1. The size of the computational domain is chosen so that the blockage ratio (ratio of the frontal area of the cube to the vertical cross sectional area of the computational domain) less than 3%. This criteria helps to eliminate any significant influence of the computational domain envelope on the flow field characteristics which is close to the building. The 3% is adapted after Baetke and Werner (1990), which is based on numerical experiment on a flow around a generic cube. The grid is generated using GAMBIT, Fluent (1998), with the mesh size of $100 \times 80 \times 25$ cells, corresponding to the length, width and height respectively. The mesh is refined in the impingement, separation and wake areas. The minimum length used at the building corner is 0.17 H and the maximum size is 0.5 H at the outlet.

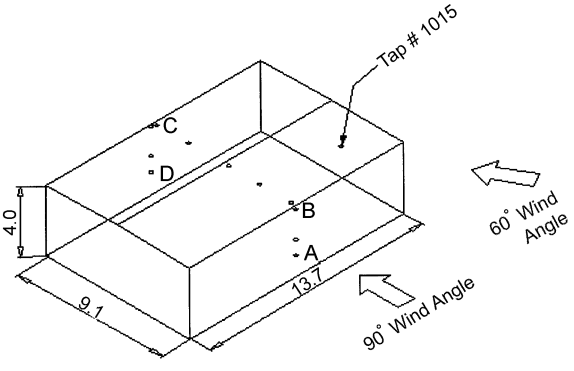

Fig. 1 TTU building dimensions, investigated taps locations and wind directions

2.2. Boundary conditions and solver

Flow analysis is performed using FLUENT 5, Fluent (1998). Boundary conditions are specified as follows: a velocity inlet with an approximate full scale velocity profile of power low $\alpha = 0.14$ with roof height velocity 8.9 m/s and turbulent intensity 20%, symmetry for the side and top boundaries; wall boundary conditions for the wall and roof of the building as well as for the floor of the computational domain; outflow boundary condition for the flow outlet. The general flow equations are discretized using a second order scheme. The near wall regions are solved using wall function for the first few time steps and then are changed to two layer zone treatment which is adequate for the wall area. The reason for using the wall function at initial stage of the computation is to produce a stable flow domain for further unsteady calculations. The wall reflection term in the turbulence model is not used, since the presence of this term creates a problem in the stability of the numerical scheme and its benefit compared to the difficulty in achieving stability is not well established (Wright and Easom 1999). In order to ensure stability a Courant number of 0.5 is used. A normalized convergence criteria of 10^{-4} is used.

2.3. Mathematical formulation

For the usual case of steady simulations the total, instantaneous, variables are expressed based on the double decomposition :

$$\Phi(x_i, t) = \overline{\Phi(x_i)} + \phi(x_i, t) \tag{1}$$

with $\overline{\Phi(x_i)}$ a time average and $\phi(x_i, t)$ the fluctuating component.

When a non-stationary flow is simulated by an unsteady RANS approach the tripple decomposition (Hussain and Reynolds 1970) is suitable :

$$\Phi(x_i, t) = \overline{\Phi(x_i)} + \phi_c(x_i, t) + \phi(x_i, t) = \langle \Phi(x_i, t)\rangle + \phi(x_i, t) \tag{2}$$

where $\phi_c(x_i, t)$ is the organized (coherent) component of the flow, and $\phi(x_i, t)$ represent the small scale, random turbulent motions. The time average is combined with the $\phi_c(x_i, t)$ component in a ensemble average (more specific phase average) term :

$$\langle \Phi(x_i, t)\rangle = \lim_{N \to \infty} \frac{1}{N} \sum_{n=0}^{N} \Phi(x_i, t + n\tau) \qquad \text{where}$$

τ is the quasi-period of the organized component, see Hussain and Reynolds (1970).

Using this decomposition the unsteady RANS (phase averaged) conservation equations become :

$$\frac{\partial}{\partial x_i}(\rho\langle U_i\rangle) = 0 \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\langle U_i\rangle) + \frac{\partial}{\partial x_k}(\rho\langle U_k\rangle\langle U_i\rangle) = -\frac{\partial\langle P\rangle}{\partial x_k}\delta_{ki} + \mu\frac{\partial^2}{\partial x_k^2}(\langle U_i\rangle) - \frac{\partial}{\partial x_k}(\rho\langle u_i u_k\rangle) \tag{4}$$

Where $<U_i>$ and $<P>$ are phase-averaged fluid velocity and phase-averaged pressure respectively.

By using a second moment turbulent closure (such as RSM) the phase averaged Reynold-stress term $<u_i u_j>$ is not approximated by isotropic (turbulent viscosity type) closures. Instead the Reynolds stress transport equation is used :

$$\frac{\partial\langle u_i u_j\rangle}{\partial t} + \langle U\rangle_k \frac{\partial\langle u_i u_j\rangle}{\partial x_k} = P_{ij} + D_{ij} + \Pi_{ij} - \varepsilon_{ij} \tag{5}$$

where the left side of the above equation represents the substantial derivatives of $<u_i u_j>$ and the terms on the right side are the rates of production P_{ij}, diffusion D_{ij}, pressure-strain Π_{ij} and dissipation ε_{ij} .

The exact form of both the D_{ij} and the Π_{ij} term include fluctuating pressure term correlations, which consequently influence (through the Reynolds-stress term) the calculation of the phase-averaged pressures for the N-S equations.

In order to close the Reynolds-stress transport Eq. (5) at a second order level the diffusion, pressure strain and dissipation terms need to be modelled.

An isotropic simplification of the Daly & Harlow model has been used for the diffusion term:

$$D_{ij} = \frac{\partial}{\partial x_k}\left(\frac{\nu_t}{\sigma_k}\frac{\partial\langle u_i u_j\rangle}{\partial x_k}\right) \tag{6}$$

It is believed that this simplification eliminates the actual contribution of the fluctuating pressure correlation in the diffusion term.

Nevertheless, modeling of the pressure strain term (Π_{ij}) retains the fluctuating pressure contribution on the Reynolds stress transport equations and hence implicitly on the RANS simulated phase averaged pressure fluctuations.

Also the dissipation rate of $<u_i u_j>$ is modeled as isotropic :

$$\varepsilon_{ij} = \frac{2}{3}\rho\langle\varepsilon\rangle\delta_{ij} \tag{7}$$

where the phase averaged dissipation rate $<\varepsilon>$ is obtained through a modelled $<\varepsilon>$ transport equation.

It is clear that the RSM model used here in conjunction with unsteady RANS equations provides phased averaged pressure fluctuations and not the total (instantaneous) pressure fluctuations. The missing term, as per the triple decomposition (2), is the random pressure fluctuation.

This missing random pressure term can be calculated using a r.m.s pressure model, see Paterson and Holmes, 1989 and Selvam, 1992. Here Selvam's model has been adapted to calculate the r.m.s of the random pressure field :

$$C'_p = 2|C_p|(1.414V\sqrt{k} + k)/V^2 \tag{8}$$

Where V and K are values of velocity and K at the building height in the approaching flow and the phase averaged value $<Cp>$ has been used instead of the time average value employed in Selvam's steady- state simulations. Time series of random pressure fluctuations with the r.m.s value modelled by Eq. (8) have been generated. Finally, the total, instantaneous pressure field has been reconstructed from the phase averaged RSM simulated pressures and the modelled random pressure fluctuations following the tripple decomposition (2).

3. Comparison with full and wind tunnel results

The model and full-scale TTU main experimental data employed for comparison with the numerical data presented here is from Surry (1991). Also, the results are compared with the wind tunnel experiment from Ham and Bienkiewicz (1998). In addition comparison is provided with other numerical simulations available, Mochida, Murakami, Shoji and Ishida (1993), Selvam (1992), Selvam (1997).

The full scale dimensions of the TTU building are 9.1× 13.7× 4.0 m with a very slight roof slope of 1:60, see Fig. 1. The wind tunnel data was collected from a 1:100 scale model at a sample rate of about 500 samples per second for 60 seconds. The full-scale data has been collected for 15 minutes duration at a sample rate of 10 Hz Levitan and Mehta (1992), Levitan, Mehta and Vann (1991). A time step of 0.1 second is used for the numerical simulation and approx. 3000 time steps are considered in order to obtain statistically comparable result with full scale and wind tunnel data. The comparison is based on eleven pressure tap locations of the full-scale building near the centerline of the building as shown in Fig. 1.

In general the mean values obtained from both wind directions show good agreement with the model and full-scale data in most areas. However, the RSM predicted peak pressure values deviate in the impingement and wake areas from the full-scale and wind tunnel data.

3.1. 90° Wind direction results & comparisons

For the 90° wind direction, the mean pressure coefficients for model and full scale experiments show good agreement for the separation zone with very small differences for the impingement and wake areas, Fig. 2. The numerical results with k - ε models (e.g., Selvam 1992) show a rather poor performance at the frontal corner of the roof. This is mainly due to the over-prediction of turbulence energy at the frontal corner, which is a main drawback of k - ε model as already discussed by Murakami (1993). The large eddy simulation results, Mochida *et al.* (1993), Selvam (1997) show good agreement with experiments in all areas. The RSM turbulence model tested in the present

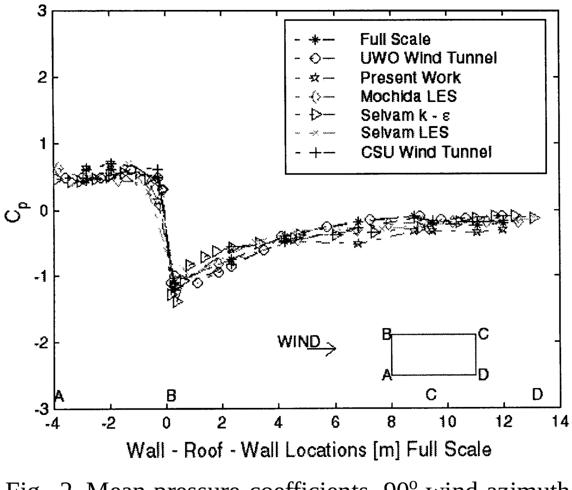

Fig. 2 Mean pressure coefficients, 90° wind azimuth

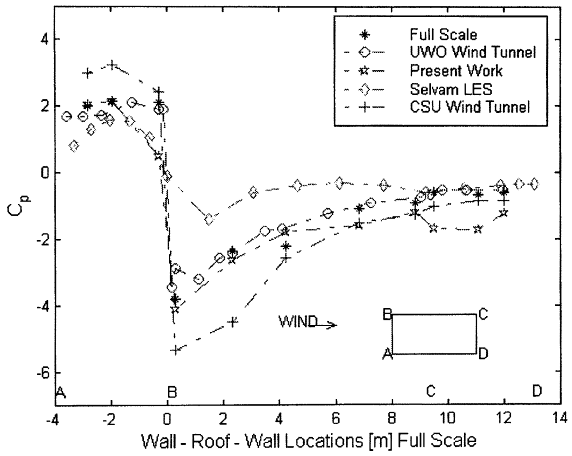

Fig. 3 Peak pressure coefficients, 90° Wind azimuth

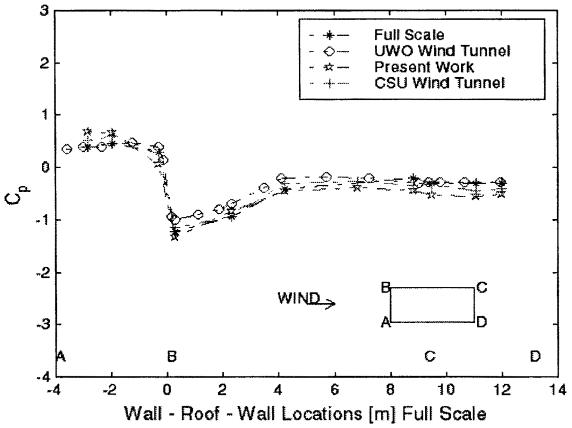

Fig. 4 Mean pressure coefficients, 60° wind azimuth

work also shows a good agreement in most of the areas.

The peak pressure coefficients of the full-scale data compared with UWO wind tunnel data show good agreement in most of the areas, Fig. 3. The CSU wind tunnel data shows a considerable over-prediction in the impingement and separation regions. The present numerical results agree with the full-scale data in the flow separation zone, but show some deviation in the impingement and wake regions, see Fig. 4. The only available numerical work to compare peak values at the selected tap locations is Selvam (1997) and these LES results show considerable deviations in most areas. This is mainly attributed to differences in inlet boundary condition, as explained by Selvam.

For this 90° wind direction a quasi-stationary separation bubble is generated at the leading edge of the roof Sarkar, Zhao and Mehta (1997), unlike the case of an oblique wind which generates unstable conical vortices.

The stationary nature of the separation bubble might be associated with reasonable prediction of peak pressures in the separation region by the RSM model.

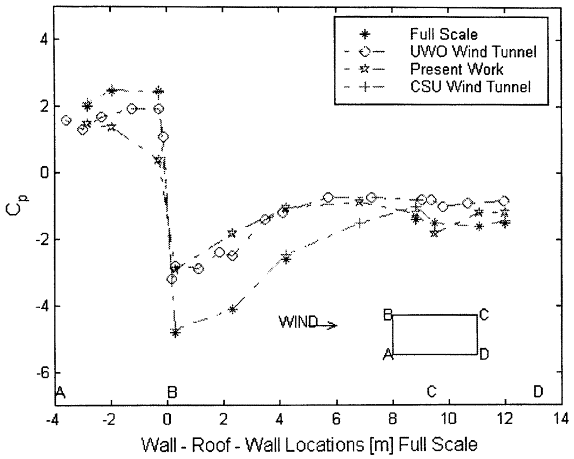

Fig. 5 Peak pressure coefficients, 60° wind azimuth

3.2. 60° Wind direction results & comparisons

The numerically predicted mean pressure coefficients present the same trend as for the 90° case, showing a good agreement with experiments in most of the areas see Fig. 4. It can be observed that RSM has the capability of predicting the mean and stationary values to a reasonable degree.

The peak pressure coefficient for 60° wind angle of the full-scale data and the CSU wind tunnel results show a good agreement in all areas. The model scale from UWO wind tunnel shows moderate discrepancies for the impingement region and appreciable under predictions for the separation and wake region.

The general trend of the full-scale results is not drastically changed between the two wind azimuths. However, for the 60 deg. wind direction, both CSU and UWO peak pressures are lower compared to the 90 deg. azimuth. The CSU data, which over predicted the 90 deg. case, fits now better the full scale results compared to the UWO data which now under predicts.

The numerically computed peak values for the roof and wake areas fall close to UWO wind tunnel results, but there are considerable differences for the impingement area, Fig. 5

The conical vortices responsible for large suction at oblique wind angles are non-symmetrical and are governed by a bi-stable flow mechanism. This switching occurs non-regularly (Taniike and Taniguchi 1999). We can therefore infer that the random nature of the switching phenomena makes it difficult for RSM turbulent model to capture the peak suctions for this wind angle.

4. Discussion

Discrepancies between full scale, wind tunnel as well as numerical results may be generally attributed to two main causes: (i) differences in inlet boundary conditions and/or (ii) the existence of intermittent flow (vortex) events at oblique azimuth angles, which events are not fully captured.

Detailed pressure measurements have been recently conducted at both UWO and CSU for a 1/100 scale model of the TTU building. Fig. 6 shows a large and narrow negative pressure peak corresponding to one of the corner vortex “switching” events at tap 1015 located on the roof near the windward

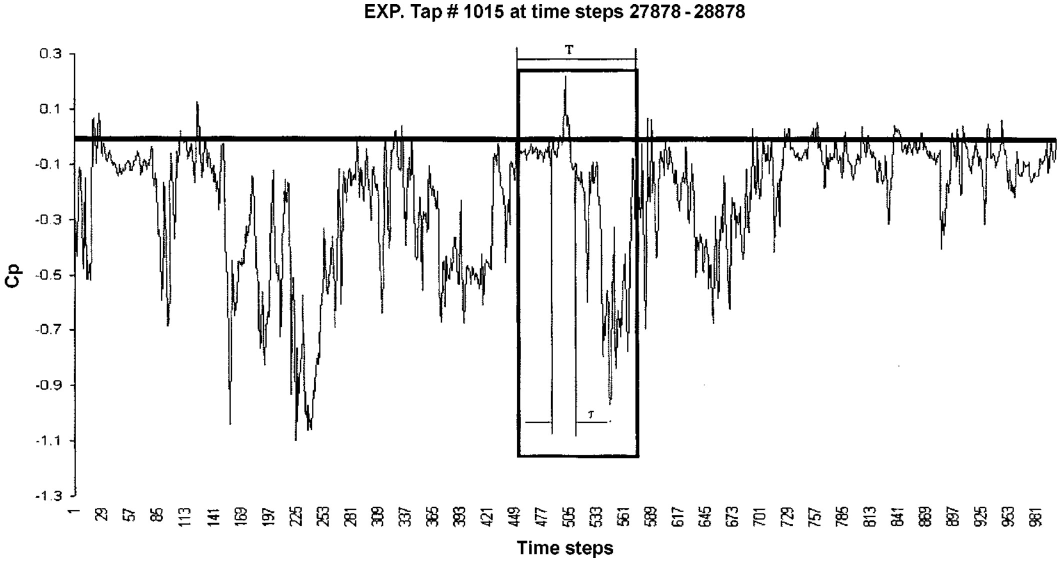

Fig. 6 Tap 1015 time series

corner, see Fig. 1. The total switching time is $T = 0.2$ (wind tunnel scale), see the box marker in Fig. 6. However, the duration of the switching peak is very short, of the order of $\tau = 0.01$ s for the 1/100 model scale. For typical wind tunnel tests the time scaling is of the order of 1/100 and the cut-off frequency is around 100 Hz. Therefore, at least some of these peak events might be lost in the sampling process. These observations are consistent with the increase in the magnitude of (average) negative peaks with the cut-off frequency, Ham and Bienkiewicz (1998).

The total time length of the RSM numerical simulation covered approximately 100 switching events (of T length) at a rate of approximately 32 time steps per event. This was considered acceptable given the second order numerical scheme employed.

The phased averaged pressure prediction of this model compare well for moderate changes in flow patterns (the case of the 90°) but will obviously filter out peaks related to intermittent flow structures, such as for the oblique wind case.

5. Conclusions

The intermittent flow events associated with bluff body corner vortex formation are difficult to measure accurately. Wind tunnel experiments for two wind directions couldn't agree with the full-scale data consistently i.e., the good agreement of peak values for one wind direction does not repeat for the other wind direction. These difficulties in capturing the instantaneous, intermittent phenomena accurately are also observed in the numerical results.

Comparisons of steady state numerical solutions with full and model scale mean values have been done by a number of researchers. These comparisons lead to improvements and new suggestions related to turbulence models in bluff body aerodynamics applications, Murakami (1993). The present work investigates the capability of a RSM turbulence model to simulate the instantaneous pressure field by phase averaging in combination with a random pressure field model. It has been found that the RSM simulations of the unsteady phase averaged pressure field combined with a random pressure fluctuation model can provide satisfactory predictions of peak pressures for quasi-stationary flow fields (90 deg. wind azimuth). However, for intermittent flow conditions (60 deg. wind azimuth) this approach proved to be rather poor. Further investigations into better modelling the random pressure component might improve the prediction of the instantaneous pressure field through RANS models.

References

Baetke, F. and Werner, H. (1990), "Numerical simulation of turbulent flow over surface-mounted obstacles with sharp edges and corners", *J. Wind Eng. Ind. Aerod.*, **35**, 129-147.

Banks, D., Meroney, R.N, Sarkar, P.P., Zhao, Z. and Wu, F. (2000), "Flow visualization of conical vortices on flat roofs with simultaneous surface pressure measurement", *J. Wind Eng. Ind. Aerod.*, **84**, 65-85.

Cochran, L.S. and Cermak, J.E. (1992), "Full- and model-scale cladding pressures on the Texas Tech University experimental building", *J. Wind Eng. Ind. Aerod.*, **41-44**, 1589-1600.

Fluent Inc. (1998), "GAMBIT modeling guide".

Fluent Inc. (1998), "Fluent user's guide".

Franke, R. and Rodi, W. (1993), "Calculation of vortex shedding past a square cylinder with various turbulence models", *In Turbulent Shere Flows* 8, F. Durst, *et al*. Editors, Springer, New York, 189-204.

Ham, H.J. and Bienkiewicz, B. (1998), "Wind tunnel simulation of TTU flow and building roof pressure", *J. Wind Eng. Ind. Aerod.*, **77 & 78**, 119-133.

He, J. and Song, C.C.S. (1997), "A numerical study of wind flow around the TTU building and the roof corner vortex", *J. Wind Eng. Ind. Aerod*., **67 & 68**, 547-558.

Hussain, A.K.M.F. and Reynolds, W.C. (1970), "The mechanics of an organized wave in turbulent shear flow", *J. Fluid Mech.*, **41**, 241-258.

Kawai, H. and Nishimura, G. (1996), "Characteristics of fluctuating suction and conical vortices on a flat roof in oblique flow", *J. Wind Eng. Ind. Aerod.*, **60**, 211-225.

Lakehal, D. and Rodi, W. (1997), "Calculation of the flow past a surface-mounted cube with two-layer turbulence models", *J. Wind Eng. Ind. Aerod.*, **67 & 68**, 65-78.

Levitan, M.L., Mehta, K.C. and Vann, W.P. (1991), "Field measurements of pressures on the Texas Tech Building", *J. Wind Eng. Ind. Aerod*., **38**, 227-234.

Levitan, M.L. and Mehta, K.C. (1992), "Texas Tech field experiments for wind loads part I: building and pressure measuring system", *J. Wind Eng. Ind. Aerod.*, **41-44**, 1565-1576.

Mochida, A., Murkami S., Shoji, M. and Ishida, Y. (1993), "Numerical simulation of flow field around Texas Tech Building by large eddy simulation", *J. Wind Eng. Ind. Aerod.*, **46 & 47**, 455-460.

Murakami, S. (1993), "Comparison of various turbulence models applied to a bluff body", *J. Wind Eng. Ind. Aerod.*, **46 & 47**, 21-36.

Paterson, D.A. and Holmes, J.D. (1989), "Computation of wind flow around the Texas Tech Building", *Proc., Workshop on Industrial Fluid Dynamics, Heat Transfer and Wind Engineering*, CSIRO-DBCE, Highett, Victoria, Australia, 33-34.

Rodi, W. (1993), "On the simulation of turbulent flow past bluff bodies", *J. Wind Eng. Ind. Aerod.*, **46 & 47**, 3-19.

Sarkar, P.P., Zhao, Z. and Mehta, K.C. (1997), "Flow visualization and measurement on the roof of the Texas Tech Building", *J. Wind Eng. Ind. Aerod.*, **69 & 71**, 597-606.

Selvam, P.R. (1992), "Computation of pressures on Texas Tech Building", *J. Wind Eng. Ind. Aerod.*, **41-44**, 1619-1627.

Selvam, P.R. (1997), "Computation of pressures on Texas Tech University building using large eddy simulation", *J. Wind Eng. Ind. Aerod.*, **67& 68**, 647-657.

Shah, K.B. and Ferziger, J.H. (1997), "A fluid mechanicians view of wind engineering large eddy simulation of flow past a cubic obstacle", *J. Wind Eng. Ind. Aerod*., **67 & 68**, 211-224.

Surry, D. (1991), "Pressure measurements on the Texas Tech Building: Wind tunnel measurements and comparisons with full scale", *J. Wind Eng. Ind. Aerod*., **38**, 235-247.

Taniike, Y. and Taniguchi, T. (1999), "Switching phenomena of conical vortices on varies flat roofs", *Con. on*

Wind Engineering Copenhagen / Denmark, 1203-1208.

Thomas, T. G. and Williams, J.J.R. (1999), "Simulation of skewed turbulent flow past a surface mounted cube", *J. Wind Eng. Ind. Aerod.*, **81**, 347-360.

Tieleman, H.W., Surry, D. and Metha, K.C. (1996), "Full/model-scale comparison of surface pressures on the Texas Tech experimental building", *J. Wind Eng. Ind. Aerod.*, **61**, 1-23.

Yu, D. and Kareem, A. (1997), "Numerical simulation of flow around rectangular prism", *J. Wind Eng. Ind. Aerod.*, **67 & 68**, 195-208.

Wright, N.G. and Easom, G.J. (1999), "Comparison of several computational turbulence models with full-scale measurements of flow around a building", *Wind and Structures, An Int. J.*, **2**(4), 305-323.

Wu, F., Sarkar, P.P. and Mehta, K.C. (1999), "Understanding the conical-vortex flow on roofs", *Con. on Wind Engineering Copenhagen / Denmark*, 1867-1874.