#7463

Journal of Wind Engineering and Industrial Aerodynamics, 46 & 47 (1993) 129–134
Elsevier

Numerical simulation to determine the effects of incident wind shear and turbulence level on the flow around a building

Y.Q. Zhang*, A.H. Huber**, S.P.S. Arya* and W.H. Snyder**

*Department of Marine, Earth and Atmospheric Sciences, North Carolina State University, Raleigh, North Carolina, 27695-8208, USA

**Atmospheric Sciences Modeling Division, Air Resources Laboratory, National Oceanic and Atmospheric Administration, Research Triangle Park, North Carolina, 27711, USA On assignment to the Atmospheric Research and Exposure Assessment Laboratory, U.S. Environmental Protection Agency, Research Triangle Park, North Carolina, 27711, USA

Abstract

The effects of incident shear and turbulence on flow around a cubical building are being investigated by a turbulent kinetic energy/dissipation (κ-ε) model (TEMPEST). The numerical simulations demonstrate significant effects due to the differences in the incident flow. The addition of upstream turbulence and shear results in a reduced size of the cavity directly behind the building.

The accuracy of numerical simulations is verified by comparing the predicted mean flow fields with the available wind-tunnel measurements of Castro and Robins (1977). Comparing our results with experimental data, we show that the TEMPEST model can reasonably simulate the mean flow.

1. INTRODUCTION

The development of the flow field around a building strongly depends on the nature of the upstream flow. The upstream flow may be characterized by the profiles of the wind velocity and turbulence intensity. Other important factors determining the flow around a building are the shape of the building and its orientation relative to the flow direction. Our present study has focused only on a cube with the inflow normal to the building face. The presence of upstream wind shear and turbulence in the flow approaching a block-shaped building is known to have important effects (Castro and Robins, 1977). However, not much information is available on the effects of shear or turbulence alone on the resulting flow field. Numerical modeling provides an ideal tool to investigate this. We have used a turbulent kinetic energy/dissipation (κ-ε) model for this purpose. Wind-tunnel measurements of Castro and Robins (1977) are used to validate the numerical model and to investigate possible limitations of the model in simulating complex flow patterns around buildings.

2. NUMERICAL SIMULATION

2.1 The TEMPEST model

The TEMPEST (Transient Energy Momentum and Pressure Equations Solutions in Three dimensions) model is a three-dimensional, transient, nonhydrostatic numerical model which was developed at Battelle Pacific Northwest Laboratory and has been applied to a broad range of engineering and geophysical problems. It includes the ability to account for small density variations through the Boussinesq approximation.

Cylindrical, Cartesian, or polar coordinates may be used. It has the ability to use variable grid spacing along any coordinate and the inflow/outflow boundaries can be either specified or computed. Turbulence is treated using a turbulent kinetic energy/dissipation (κ-ε) model. The solution technique in TEMPEST is similar to the marker-and-cell (MAC) technique (Amsden and Harlow, 1970) whereby at each time step, the momentum equations are solved explicitly and pressure equations implicitly; temperature, turbulent kinetic energy (TKE), dissipation of kinetic energy (DKE), and other scalar transport equations are solved using an implicit continuation procedure.

2.2 Numerical methods

The standard formulation for the κ-ε model (Trent and Eyler, 1989) was used in our simulation. A staggered grid system was adopted for all the simulations.

At solid walls, no-slip boundary conditions are used for the mean velocity; a modified law-of-the-wall formulation is used to relate the level of turbulence to the mean shear. The rationale behind this modified law-of-the-wall formulation is explained by Launder and Spalding (1973).

Our numerical simulation used a variable-spaced grid of 52 cells long × 25 high × 43 wide with the domain dimension about $14H_b$(length) × $4.8H_b$(height) × $9H_b$(width). The smallest cell in the domain is 1/10 of the building height H_b which is 60m for our numerical simulations.

The vertical profiles of velocity and turbulent kinetic energy measured in the wind tunnel (Castro and Robins, 1977) were used for the upstream boundary conditions in our numerical simulation. The dissipation of turbulent kinetic energy at the inflow is specified by $\varepsilon = C_\nu \kappa^2/\nu_t$, where ν_t is turbulent diffusivity, κ is the turbulent kinetic energy and $C_\nu = 0.09$.

3. RESULTS AND DISCUSSION

3.1 Grid independence

In order to confirm that our results do not depend on the grid resolution, two different grids were used in the simulations with the uniform approach flow. First, we used the grid specified above. In this case, the smallest cell was only one tenth of the cube height. We then increased the resolution in each direction so that the smallest cell was one twentieth of the cube height (a grid of 52 cells long × 31 high × 25 wide with the domain dimension about $11.9H_b$(length) × $4H_b$(height) × $3.5H_b$(width) and only half of the flow field was calculated). The results showed that the mean velocity fields in the two cases were virtually identical. The difference between the lengths of the recirculation zone on the ground behind the building in the two cases was less than $0.05H_b$. Therefore, the coarser resolution was considered to be adequate and was used in more detailed investigations of the effects of incident wind shear and turbulence on the flow around the cubical building.

3.2 Flow field

Four different simulations were conducted in order to investigate the effects of turbulence or shear alone on the flow field around a cubical building. Numerical simulation of the flow around a surface-mounted cube in shear flow with high turbulence is designated as CASE A and that with no turbulence as CASE B. Numerical simulation in uniform approach flow with turbulence is designated as CASE C and that with no turbulence as CASE D. Vertical profiles of longitudinal velocity above and downstream from the cube for CASE A are compared with wind-tunnel measurements (Castro and Robins, 1977) in Figure 1. Lateral profiles downwind of the cube are compared in Figure 2. Good agreement is found between the computed and observed mean velocity fields. The differences between the two are within 10 percent of the upwind free-stream velocity, the largest difference being in the recirculating cavity region. Comparisons are also made of the CASE D simulation with the available wind-tunnel data (Castro and Robins, 1977). In this case, the differences between the simulated and the wind-tunnel-measured profiles are much larger, as shown in Figures 3 and 4. In our simulations, we used the so called "wall functions" to relate the mean flow shear and turbulence near the solid surfaces. These formulations were derived assuming no separation of the flow from the wall and were based on empirical observations of pipe flow with smooth walls. The same wall functions are used in our application to the regions of flow separation and recirculating cavity which are more extensive for CASE D. The model underestimates the strength of the reverse flow in the cavity region.

The velocity fields in the vertical centerplane through the building for the four cases A-D are shown in Figures 5a to 5d, respectively.

A comparison of simulated velocity fields around the building with an approach shear flow with strong turbulence (A) and without any turbulence (B) can be made from Figures 5a and 5b. In both cases, there is a stagnation point on the upstream surface of the building whose position is not very sensitive to the presence of the turbulence in the approaching flow. The incident turbulence only slightly reduces the size of the upstream vortex near the building. However, the incident wind shear has a dramatic influence on the position of the upstream stagnation point and the resulting vortex. Both A and B exhibit stagnation points high up on the windward face of the building. By contrast, the stagnation points in CASES C and D are very near the ground. In a sheared boundary layer, incident wind speed increases with increasing height, so that the stagnation pressure is higher near the top of the front face; this pressure gradient drives a downward-directed flow on the front face. Shear has a stronger influence on the position of the stagnation point at the upstream face of the cube than that of the turbulence in the approach flow.

In CASE D (no shear and no turbulence in the upstream incident flow), flow separation occurs along the upwind edge of the cube and reattachment occurs on the ground. The addition of turbulence and/or shear almost entirely eliminates the recirculating flow on the roof-top and side faces in our simulations. In all cases, the flow separates from the upwind edge, but the reverse flow region on the roof top is too shallow to be resolved by our model, except in CASE D.

Consider now the characteristics of the flow behind the cube. The enhanced mixing and thickening of the shear layers shed from windward surfaces of the sharp-edged building in CASE A promote flow reattachment on the building surfaces. The velocity vector plots in vertical planes in Figure 5 demonstrate that the size of the downstream cavity region is reduced by the presence of upstream turbulence. Considering the wake behind the cube in CASE B, the shear layer has originated from the separation of the boundary layer at the rear of the body. The wake is, therefore, considerably lower than that for CASE D in which the shear layer has originated from the leading edge. Consequently, the length of the recirculation zone on the ground behind the building varies from about $1.7H_b$ in CASE B to $2.3H_b$ in CASE D and from $1.5H_b$ in CASE A to $1.8H_b$ in CASE C; this indicates the influence of shear and turbulence in the approach flow on the extent of the cavity region. Here, the length of the recirculating region is estimated directly from the raw velocity data on the centerplane through the building.

Both the shear and turbulence in the approach flow tend to reduce the extent of the cavity region in the lee of the building. The influence of each factor appears to be maximized in the absence of the other factor. For example, the reduction in the cavity length due to the effect of shear alone (no turbulence) is 26 percent (cf, CASES B and D) and that due to the effect of turbulence alone (no shear) is 22 percent (cf, C and D). The combined influence of shear and turbulence is to reduce the cavity length by only 35 percent (cf, A and D) which is less than the sum of their individual reductions.

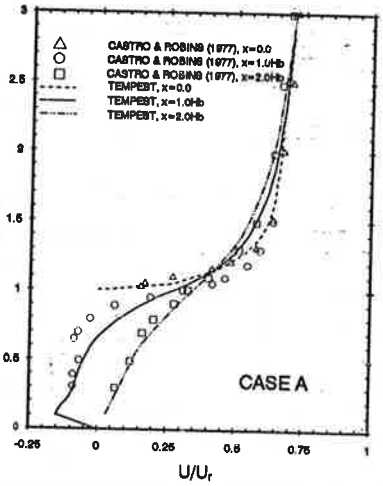

Figure 1. Vertical profiles of longitudinal mean velocity in centerplane; x=0 is at the center of the building.

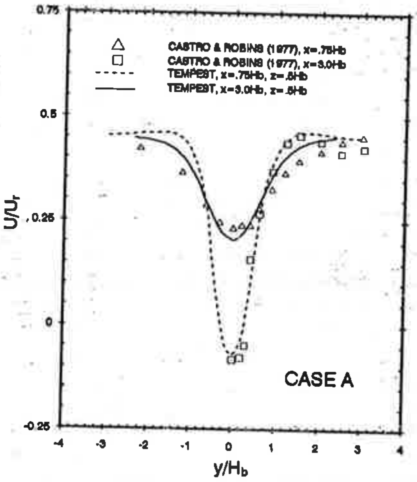

Figure 2. Lateral profiles of longitudinal mean velocity.

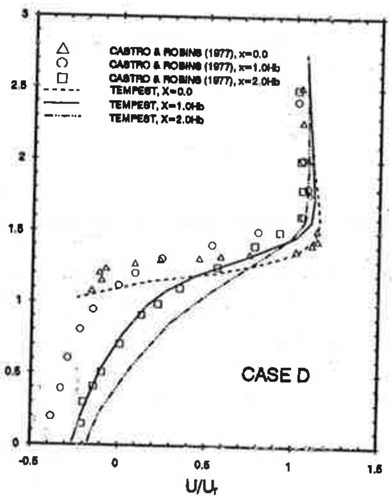

Figure 3. Vertical profiles of longitudinal mean velocity in centerplane.

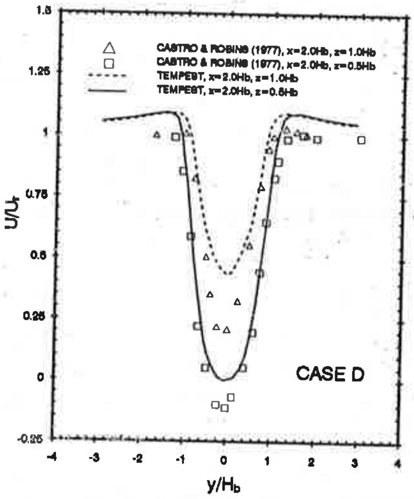

Figure 4. Lateral profiles of longitudinal mean velocity.

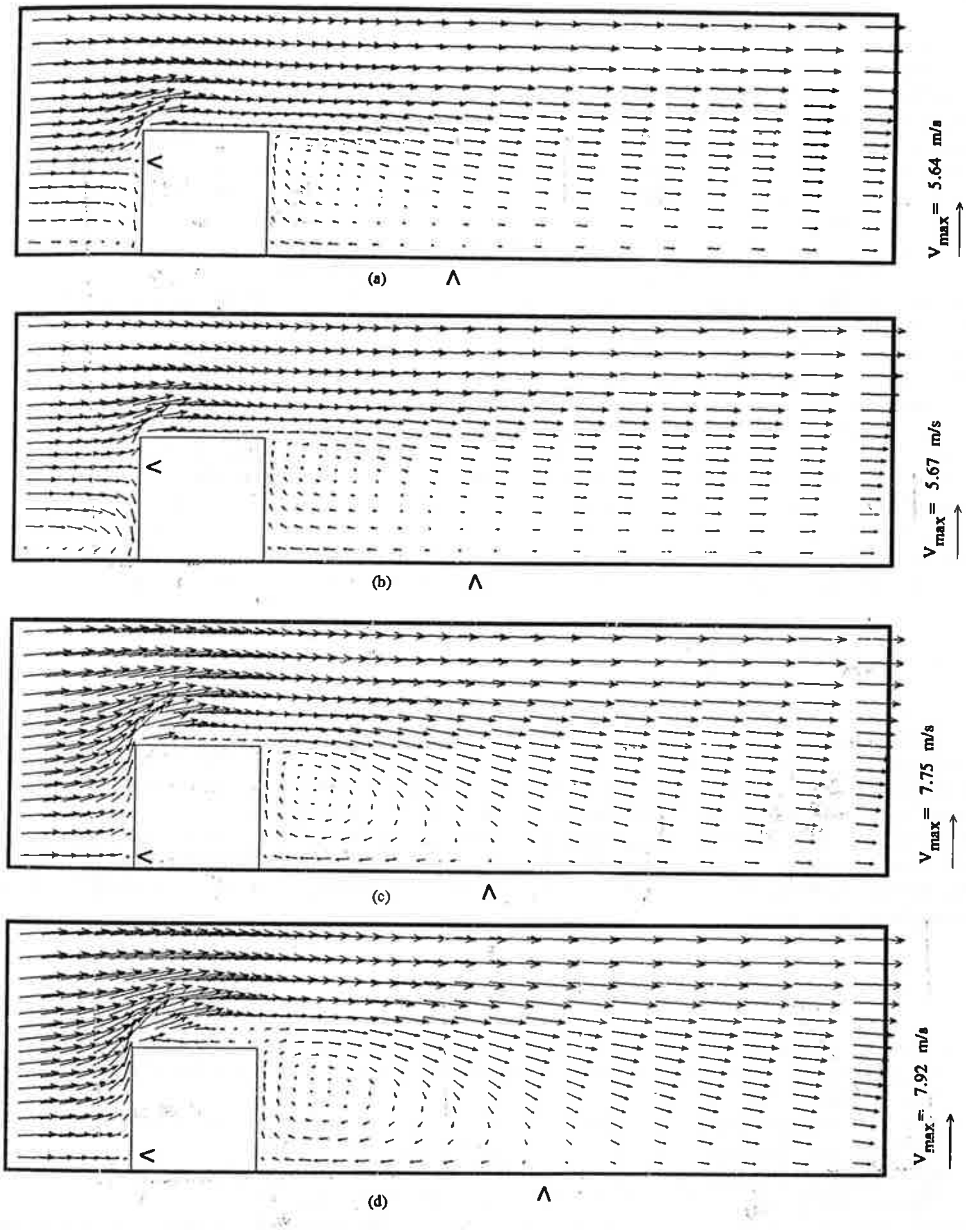

Figure 5. Velocity vector field in the vertical centerplane. Reference freestream velocity is 7m/s.
Carets mark the positions of stagnation and reattachment.
(a) CASE A; Shear with high turbulence inflow; (b) CASE B; Shear with no turbulence inflow.
(c) CASE C; Uniform with high turbulence inflow; (d) CASE D; Uniform with no turbulence inflow.

4 CONCLUSIONS

Numerical simulations of the atmospheric surface layer flow around a cubical building are compared with the wind-tunnel experiments of Castro and Robins (1977) for different approach flow conditions. The TEMPEST model simulates fairly well the important features of the complex flow field around the cubical building with a typical incoming shear flow (neutral surface layer) with strong turbulent mixing. An extreme case of uniform approach flow with no turbulence presents a more severe test of the model, which underestimates the strength of the reverse flow in the cavity region behind the building.

The numerical model is used to further examine the influence of shear and turbulence in the approaching stream on the flow around the cubical building. It is found that the upwind shear promotes the development of the upwind vortex at the front of the building, while it reduces the size and strength of the much larger cavity in the lee of the building. Turbulence in the approach flow further tends to reduce the lee-side cavity and also has some influence on the flow over the roof-top. The influence of shear or turbulence is maximum in the absence of the other factor.

We should point out that this κ-ε model is based on the eddy viscosity concept. Considerable differences in the distribution of turbulent kinetic energy around the windward corner between the numerical prediction from a κ-ε model and wind-tunnel measurements were found by Murakami and Mochida (1988). No experimental data are available at this time for comparisons. Further efforts including modifications of the so-called "wall functions" and the grid resolutions near the surface may be needed to improve the accuracy of the numerical simulation in the κ-ε model.

5 ACKNOWLEDGMENTS

The information in this document has been funded by the U.S. Environmental Protection Agency under Cooperative Agreement CR817931 with the North Carolina State University. The contents of this paper do not necessarily reflect the views and policies of the agency, nor does mention of trade names or commercial products constitute endorsement or recommendation for use. Computation time on the CRAY Y-MP8/432 at North Carolina Supercomputing Center was provided by NCSU and U.S. EPA. The authors would like to thank Dr. Don Trent and Dr. Loren Eyler of Battelle Pacific Northwest Laboratories for their consultation and support of the TEMPEST code throughout our study.

6 REFERENCES

Amsden A.A. and Harlow F.H. (1970), The SMAC method: A Numerical Technique for Calculating Incompressible Flows, Report LA-4370, Los Alamos Scientific Laboratory, Los Alamos, NM

Castro I.P. and Robins A.G. (1977), The Flow Around a Surface-Mounted Cube in Uniform and Turbulent Streams, *J. Fluid Mech.*, **79**, 307-335

Launder B.E. and Spalding D.B. (1973), The Numerical Computation of Turbulent Flows, *Comp. Methods in Appl. Mech. and Engr.*, **3**, 269-289

Murakami S. and Mochida A. (1988), 3-D Numerical Simulation of Airflow Around a Cubic Model by Means of the κ-ε Model, *J. Wind Engr. Ind. Aerodyn.*, **31**, 283-303

Trent S.D. and Eyler L.L. (1989), TEMPEST, A Three-Dimensional Time-Dependent Computer Program for Hydrothermal Analysis, Volume 1: Numerical Methods and Input Instructions, PNL-4348, Pacific Northwest Laboratory, Battelle, WA