Proceedings of ASME FEDSM'00
ASME 2000 Fluids Engineering Division Summer Meeting
June 11-15, 2000 Boston, Massachusetts

FEDSM2000-11172

TURBULENT VORTEX SHEDDING FROM TRIANGLE CYLINDER USING THE TURBULENT BODY FORCE POTENTIAL MODEL

Xing Zhang Blair Perot

Department of Mechanical and Industrial Engineering, University of Massachusetts, Amherst
Amherst, MA 01003, U.S.A.

ABSTRACT

Numerical simulation of the turbulent flow around a triangular cylinder at a Reynolds number of 45,000 is presented in this paper. Both steady and unsteady vortex-shedding results are presented. A body force potential model is used to model the turbulent motion. This approach is able to model non-equilibrium turbulence accurately at a cost and complexity comparable to k-ε models. The numerical method used in this calculation is an unstructured staggered mesh scheme. The property that this method conserves kinetic energy both locally within cells and globally makes it a good choice for performing turbulence modeling.

For the unsteady solution, the Strouhal number and time-averaged velocity profile agree well with experiments. However, the steady solution that was obtained by using a symmetric boundary condition at the centerline leads to poor predictions of the time-averaged mean velocity profile.

INTRODUCTION

The flow around a triangle provides an example of bluff body flow with fixed separation points. If the Reynolds number is not too small the flow is inherently unsteady and a Von Karman vortex street appears with a well-defined frequency. If the Reynolds number is sufficiently high the flow will be turbulent and a turbulence model must be included to model the turbulent fluctuations. In the case of turbulent vortex shedding, we have the option of including the large scale vortex shedding in the turbulence model and calculating a steady mean flow, or of solving for the large scale vortex shedding by numerical scheme while only including the small scale turbulence in the model. The former approach is less expensive, but we show here less likely to give accurate predictions. This is hypothesized to be due to the fact that the large-scale vortex structures do not behave like equilibrium turbulence.

Sjunnesson (1991) measured the flow of a triangular cylinder in a duct. Their experimental study was motivated by the application to flame holders. Johansson et al. (1993) carried out numerical simulation of this flow using a k-ε model. Durbin (1994) also carried out the same simulation using a k-ε-v^2 model. Franke et al. (1991) compared the ability of different models to predict turbulent vortex shedding from a rectangular cylinder. Franke's conclusion is that some k-ε models do not predict the right shedding frequency and Reynolds stress transport models can produce results in good agreement with the experiments. The turbulent potential model is a simplified Reynolds stress transport model, which has the ability of modeling non-equilibrium turbulence with the computing cost and complexity comparable to k-ε model.

TURBULENCE MODEL AND NUMERICAL SCHEME

The primary difficulty of modeling unsteady turbulent vortex shedding is thought to be that the turbulence is not in equilibrium with the mean flow, because the large vortices move and decay at the same time-scale as the turbulence. The most common constitutive relation, the eddy viscosity hypothesis (or linear Boussinesq hypothesis) is probably incorrect in this case.

In the past, avoiding an algebraic constitutive relation for the Reynolds stresses required solving coupled transport equations for the Reynolds stress themselves. Recently, a new modeling approach, the turbulence potential model has been developed. It is capable of modeling the complex turbulent physics associated with separation and unsteady flow. This turbulent potential model is well suited to vortex shedding problem because it does not require a constitutive relation relating the Reynolds stress tensor to the mean flow. The model hypothesizes evolution equations for the scalar and vector potentials of the turbulent body force (the divergence of the Reynolds stress tensor). It has the accuracy of a Reynolds stress model, at a cost comparable to modern two equation models. The governing equations of the turbulence potential model will not be presented here. Their initial development is described in Perot (1999).

Our numerical method uses an unstructured staggered mesh scheme which can conserve mass, momentum, and kinetic energy to machine precision. The turbulence quantities are advected using an unwinding scheme to guarantee positivity constraints. The model integrates up to the wall, so wall functions are not used, but the first grid point should be in the laminar sub-layer to obtain accurate predictions. The details for this numerical method, including accuracy analysis and conservation property are discussed in Perot & Zhang (1999).

FLOW OVER A TRIANGLE

In order to compare with the experimental data, we select a computational domain that is the same as the configuration of Sjunnesson's experiment. The mesh is generated by TRIANGLE – an automatic 2D-Delaunay mesh maker. There are approximately 25,000 triangles in our calculation (see Figure 1).

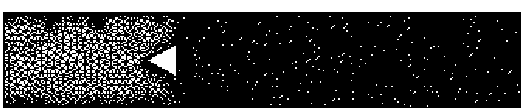

Figure 1. Computational domain and mesh.

In the present calculation, the inlet mean stream-wise velocity is a constant value, the vertical velocity is zero. For turbulent kinetic energy and dissipation rate, we use the same conditions described in Johansson's paper.

$$U_{in} = 17.0 \text{ m/s}$$
$$k_{in} = (0.05 U_{in})^2$$
$$\varepsilon_{in} = \frac{0.16 k_{in}^{3/2}}{0.2\ell}$$

The total mass flow was $\dot{m}_1 = 0.6$ kgs^{-1} in their experiment, and the inlet velocity is evaluated based on that value. These values are also used as the initial value for the whole domain. ℓ is the height of the duct. A zero gradient boundary condition is used for all the variables at the outlet. Slip-wall boundary conditions are used for the duct wall.

In the steady calculation, we use half of the domain mentioned above and imposed a symmetric boundary condition along the centerline.

RESULTS

Calculation of 2D unsteady turbulent flow around a triangle cylinder with Reynolds number $\mathrm{Re} = \frac{U_{in} H}{\nu} = 45{,}000$ is presented, where H is the height of the triangle. Unsteady behavior is due to vortices alternately shedding from the upper and lower edges of the cylinder, forming a Von Karmann vortex street behind the triangle.

No special triggering measure is taken to start the vortex shedding, the unsteadiness in the computational result evolved naturally. It was triggered by the machine error and asymmetry of the mesh.

To illustrate the periodicity of the flow, the stream function of a point about one triangle height behind the triangle near the centerline is shown in Figure 2. It can be seen that an almost perfect periodicity exists. The shedding frequency is 109.3 (s^{-1}). The corresponding Strouhal number defined by,

$$Sr = \frac{fH}{U_{in}}$$

is 0.257, which should be compared with experimental data of 0.25 and the computed value of 0.27 in Johnnasson (1991). Figure 3 shows an instantaneous velocity vector plot, we can see that the center of a vortex is rolled up at the lower edge and a new vortex is beginning to roll up at the upper edge. The vortex street can also be seen in the instantaneous vorticity contours plot shown in Figure 4.

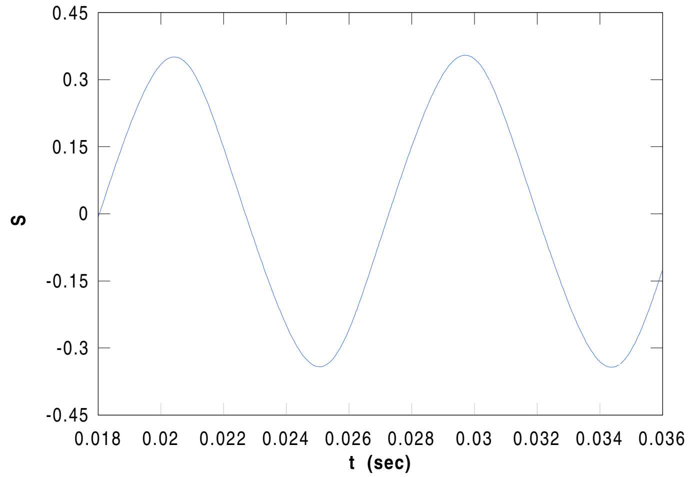

Figure 2. The stream-function of one point about one cylinder height behind the triangle near the centerline.

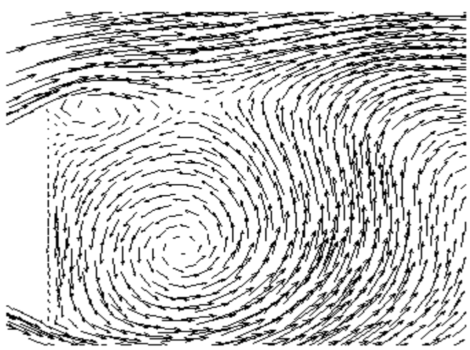

Figure 3 Instantaneous velocity vector plot.

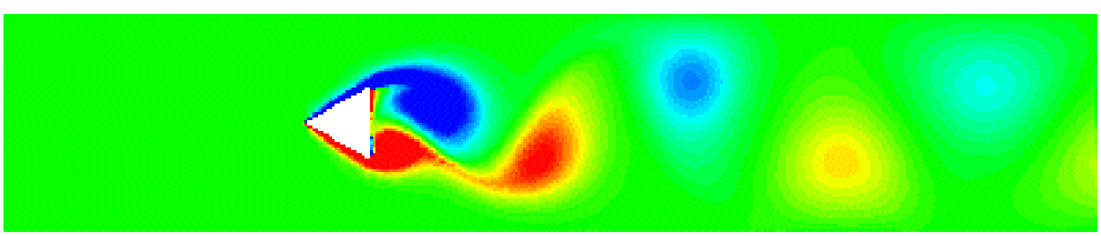

Figure 4. Instantaneous vorticity contours plot.

Although the instantaneous flow is asymmetric, the time-averaged fields are always symmetric or anti-symmetric. Figure 5 shows the stream-wise velocity at different locations behind the triangle. The calculated velocity profiles are in reasonable agreement with the experiment. However, it is hypothesized that the boundary layer on the triangle is not fully resolved due to mesh size restrictions. The computed boundary layer is much thicker than the real one, thus close to the back of the triangle, the fluid is slowed down and driven backwards more than it should be. This would explain the mean velocity profile close to the centerline at x=15mm where the velocity is under-predicted. Figure 6 shows the mean stream-wise velocity at the centerline. The length of recirculation zone is accurately predicted, while the location of the maximum negative velocity is slightly upstream compare with the experiments. The magnitude of the maximum negative velocity is also a little lower than the experiment data.

Figure 7 shows comparison of the mean stream-wise velocity contour plots between the time-averaged and the steady solution. The contour levels in each plot are the same. The predicted "steady-state" recirculation zone is much longer than the time-averaged unsteady solution. The reason for this is that the unsteady flow increases the momentum exchange between the wake and its surrounding, thus reducing the recirculation zone. The turbulence model does not adequately represent the momentum exchange due to these very large eddies

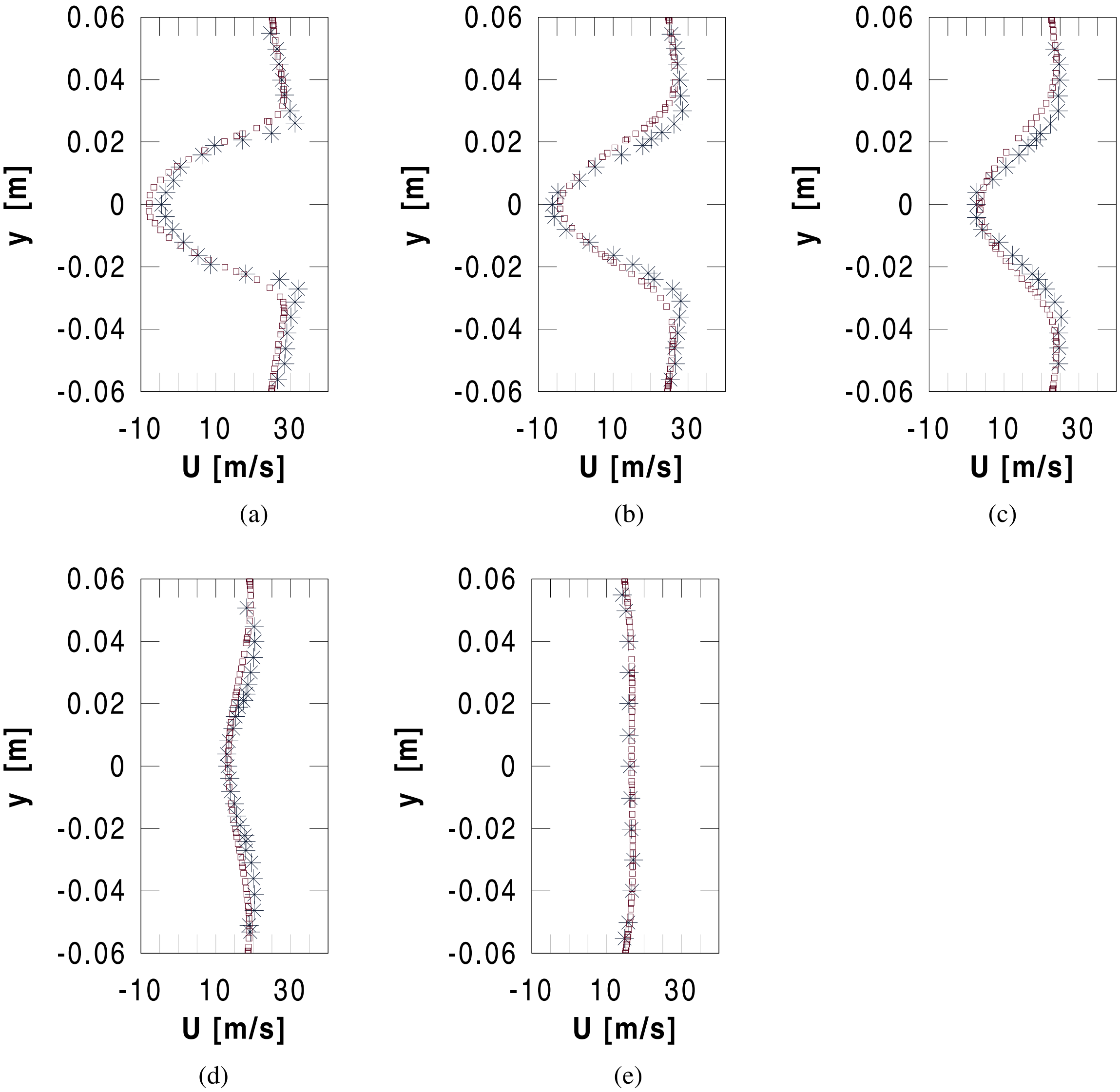

Figure 5. Mean stream-wise velocity behind the triangle: , calculations; *, experiments.
(a) 15mm, (b) 38mm, (c) 61mm, (d) 150mm, (e) 376mm

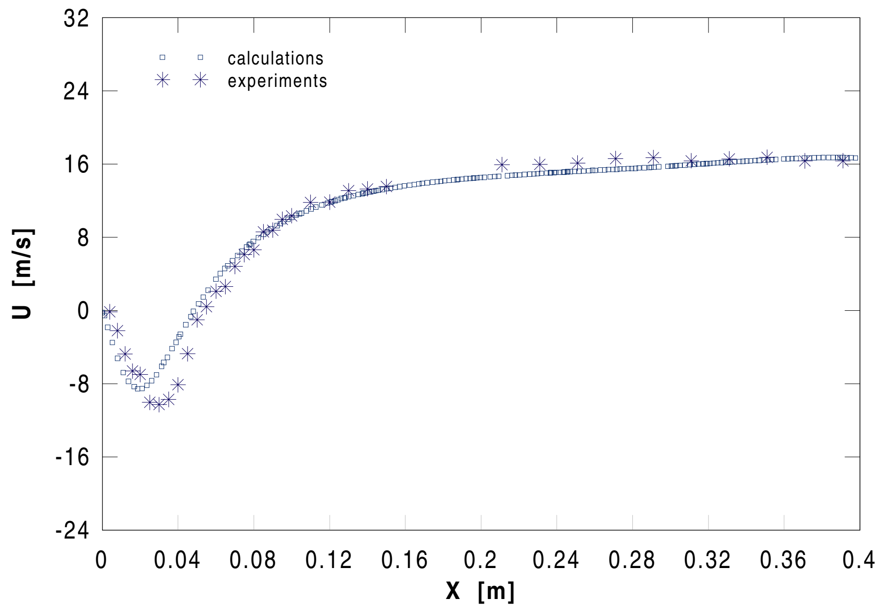

Figure 6. Mean stream-wise velocity at centerline.

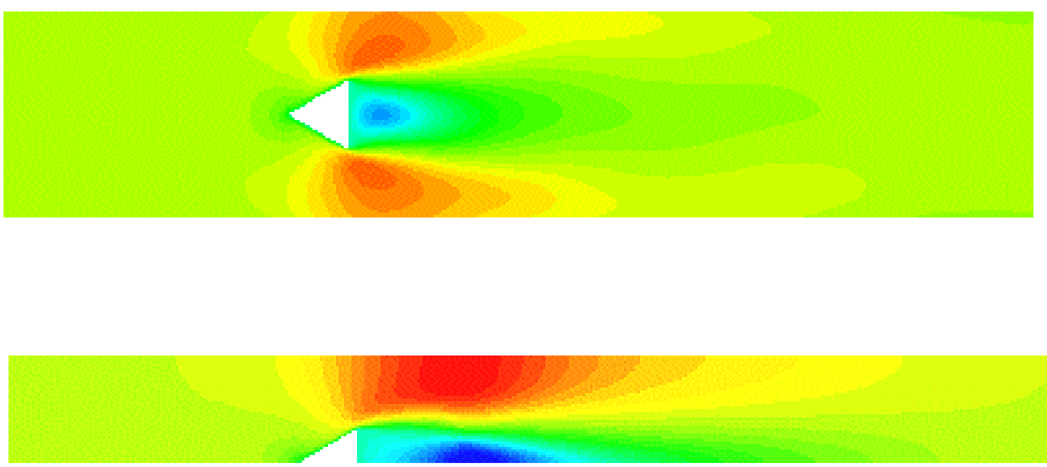

Figure 7. Mean stream-wise velocity contours of the time-averaged and steady solution

CONCLUSIONS

In this paper, numerical simulation of flow past triangular cylinder at high Reynolds number (45,000) is presented. The instantaneous flow situation is very complex due to the presence of vortex shedding and turbulence.

The calculation was performed using an unstructured staggered mesh scheme. A turbulent potential model is used to model the small-scale fluctuation motion.

The capability of the turbulent potential model to predict turbulent vortex shedding has been demonstrated in this calculation. Computed Strouhal number and mean velocity profiles down stream of the triangle cylinder are in agreement with experiment data. In addition, it has been shown that statistical unsteadiness produced by vortex shedding must be resolved in order to simulate the flow correctly. The steady state computation of this flow will lead to poor predictions.

ACKNOWLEDGMENTS

The financial support of the Office of Naval Research (grant number: N00014-99-1-0194) is gratefully acknowledged.

REFERENCES

J. B. Perot, Turbulence Modeling Using Body Force Potentials. Physics of Fluids, 11 (9), 1999

J. B. Perot and X. Zhang, Reformulation of the unstructured staggered mesh method as a classic finite volume method. Second International Symposium on Finite Volumes for Complex Applications: Problems and Perspectives, July 19-22,1999, Duisberg, Germany

Stefan H. Johansson, Lars Davidson and Erik Olsson, Numerical Simulation of Vortex Shedding Past Triangular Cylinders at High Reynolds Number Using a k-ε Turbulence Model. International Journal For Numerical Methods In Fluids, VOL. 16, 859-878 (1993)

P. A. Durbin, Turbulence modeling for separated flow. Annual Research Briefs-1994, Center for Turbulence Research, Stanford University

A. Sjunnesson, C. Nelsons and E. Max, LDA measurement of velocities and turbulence in a bluff body stabilized flame. Volvo Flygmotor AB, Trollhattan, 1991

R. Franke and W. Rodi, Calculation of vortex shedding past a square cylinder with various turbulence models. Proc. 8th Symp. On Turbulent Shear Flows, Munich, 1991, pp. 20.1.1- 20.1.6

