POTENTIAL TURBULENCE MODEL PREDICTIONS OF FLOW PAST A TRIANGULAR CYLINDER USING AN UNSTRUCTURED STAGGERED MESH METHOD

Xing Zhang Blair Perot

Department of Mechanical and Industrial Engineering, University of Massachusetts, Amherst Amherst, MA 01003, U.S.A. E-mail: <xzhang@ecs.umass.edu>

ABSTRACT

Numerical simulation of the turbulent flow around a triangular cylinder at Reynolds number of 45,000 is presented in this paper. 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 Strouhal number and timeaveraged velocity profiles obtained from this simulation agree with experiments.

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. The large-scale motions of the vortices are not turbulence, so they should be resolved by the numerical scheme and only the small-scale fluctuations are modeled.

LDA measurement by Sjunnesson *et al.* [1] of vortex shedding flow past a triangular cylinder in a duct at Re_{D} =45,000 is a useful test case for unsteady turbulent flow of this kind. Their experimental study was motivated by the application to flame holders. Johansson et al [2] carried out numerical simulation of this flow using a k- ε model. Durbin [3] (1994) carried out a simulation using a k- ε - v^2 model. In some similar simulations by Franke et al.

[4], they compared the ability of different models to predict turbulent vortex shedding from a rectangular cylinder. Franke's conclusion is that some k-E models do not predict the right shedding frequency and Reynolds stress transport models can produce good agreement results in with the experiments. It is hypothesized that this is because the turbulence is not in equilibrium with the mean flow. The proposed 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.

EQUATIONS

Mean Flow Equations

$$\frac{\partial \boldsymbol{u}}{\partial t} + \nabla \cdot (\boldsymbol{u}\boldsymbol{u}) = -\nabla p + \nabla \cdot \boldsymbol{n} (\nabla \boldsymbol{u}^{\mathbf{T}} + \nabla \boldsymbol{u}^{\mathbf{T}}) - \nabla \cdot \boldsymbol{R}$$
$$\nabla \cdot \boldsymbol{u} = 0$$

where \mathbf{u} is the mean velocity vector, p is the mean pressure/density, \mathbf{n} is the kinematic viscocity, and \mathbf{R} is the Reynolds stress tensor.

Turbulence Modeling

In the turbulence vortex shedding problem, the turbulence motion is not usually in equilibrium with the mean flow. Prior evidence indicates that the commonly used Boussinesq hypothesis (linear eddy viscosity) is not a good approximation for this problem due to the assumption of the turbulent equilibrium. In the past, for this kind of problem to be effectively predicted, the modeled transport equations for the Reynolds stress tensor itself had to be solved. However, the Reynolds stress transport equations are significantly more difficult to solve than twoequation models. Recently, a body force potential model has been developed. This model is an alternative approach to modeling the Reynolds stress tensor. The primary quantities are the scalar and vector potentials of the divergence of the Reynolds stress tensor. The proposed model does not suffer from these difficulties. It has fewer variables than Reynolds stress transport models, the equations are not strongly coupled and are therefore not as numerically stiff, and no realizability conditions are needed. The governing equations of the turbulent potential model are discussed in detail in Perot [5] (1999). The Reynolds stress is related to the scalar and vector potentials of the turbulent body force by,

$$\nabla \cdot \vec{R} = \nabla f + \nabla \times \vec{y}$$

The governing equations for the turbulence quantities are:

$$\begin{split} \frac{Dk}{Dt} &= \nabla \cdot (\mathbf{u} + \frac{\mathbf{u}_{t}}{\mathbf{s}_{k}}) \nabla k + P - \mathbf{e} \\ \frac{De}{Dt} &= \nabla \cdot (\mathbf{u} + \frac{\mathbf{u}_{t}}{\mathbf{s}_{e}}) \nabla \mathbf{e} + \frac{\hat{\mathbf{e}}}{k} (C_{e1}P - C_{e2}\mathbf{e}) \\ \frac{Df}{Dt} &= \nabla \cdot (\mathbf{u} + \frac{\mathbf{u}_{t}}{\mathbf{s}_{k}}) \nabla \mathbf{f} - \left[2\mathbf{a} \frac{\hat{\mathbf{e}}}{k} + \frac{2\mathbf{u}(\nabla \mathbf{f}^{3/2} \cdot \nabla \mathbf{f}^{3/2})}{\mathbf{f}} + \frac{\mathbf{e}}{k} \frac{2}{1 + C_{p4}} \frac{\mathbf{u}}{\mathbf{u}} \right] \mathbf{f} \\ &+ C_{p1} \frac{\mathbf{e}}{k} \frac{(1 - \mathbf{a})}{k + C_{pw}/\text{Re}} (2k/3 - \mathbf{f}) + C_{p2} \frac{\mathbf{f}}{k} P + \\ (C_{p2} + C_{p3}) 2\mathbf{a} \left(\frac{\mathbf{y} \cdot \mathbf{y}}{\mathbf{u}_{t}} \frac{1}{1 + C_{pw}/\text{Re}} - P \right) \frac{\mathbf{f}}{k} \\ \frac{D\mathbf{y}}{Dt} &= \nabla \cdot (\mathbf{u} + \frac{\mathbf{u}_{t}}{\mathbf{s}_{k}}) \nabla \mathbf{y}^{\mathbf{v}} \left(\mathbf{a} \frac{\hat{\mathbf{e}}}{k} + \frac{2\mathbf{u}(\nabla k^{1/2} \cdot \nabla \mathbf{f}^{3/2})}{(k\mathbf{f})^{1/2}} + \frac{\mathbf{e}}{k} \frac{1}{1 + C_{p4}} \frac{\mathbf{u}_{t}}{\mathbf{u}} \right) \mathbf{r} \\ &+ \mathbf{f} \mathbf{w}^{\mathbf{v}} - C_{p1} \frac{\mathbf{e}}{k} \frac{(1 - \mathbf{a})}{1 + C_{pw}/\text{Re}} \mathbf{y}^{\mathbf{v}} + C_{p2} \left(\frac{\mathbf{y}}{k} P - \mathbf{f} \mathbf{w} \right) + \\ (C_{p2} + C_{p3}) 2\mathbf{a} \left(\frac{\mathbf{y} \cdot \mathbf{y}}{\mathbf{u}_{t}} \frac{1}{1 + C_{pw}/\text{Re}} - P \right) \frac{\mathbf{y}}{k} \end{split}$$

where

$$P = \mathbf{y}^{\mathbf{V}} \cdot \mathbf{w}^{\mathbf{V}}, \mathbf{u}_{t} = C_{\mathbf{u}} \frac{fk}{e}, \mathbf{a} = \frac{1}{1+1.5 \frac{f}{k}}, \text{Re} = \frac{k^{2}}{ue}$$
$$\hat{\mathbf{e}} = \mathbf{e} - 2\mathbf{u}\nabla k^{0.5} \cdot \nabla k^{0.5} - \mathbf{u}\nabla f^{0.5} \cdot \nabla f^{0.5}$$
and
$$C_{\varepsilon_{1}}, C_{\varepsilon_{2}}, C_{\mu}, \sigma_{\varepsilon}, \sigma_{\varepsilon}, C_{p_{1}}, C_{p_{2}}, C_{p_{3}}, C_{p_{4}}, C_{p_{w}} \text{ are constants}$$

CODE

Numerical Scheme

The 2Densemble-averaged unsteady Navier-Stokes equations are solved numerically with an unstructured staggered mesh method. A two-dimensional triangular (Delaunay) mesh is generated for this computation. For this kind of mesh, there is a Veronoi dual mesh associated with it. The faces of the two meshes are always locally orthogonal. The orthogonality of these 'dual mesh' can be used to develop discretization operators that closely mimic their continuous counterparts. These discretization operators are ideally suited to representations of the Navier-Stokes equation based on the vorticity. A stream function formulation is used to eliminate the pressure term in the momentum equation.

For spatial discretization, a limited gradient method (a second order upwind scheme) is used for the convection term. As for the time advancement, the convection term is explicit



Figure 1. An example of 2D unstructured triangular mesh.

and a three-step second order Runge-Kutta scheme is used, the diffusion term is implicit. The symmetric semi-positive definite algebraic system resulting from the discetization is solved using a conjugate gradient iterative method. The detailed description of this Navier-Stokes solver is presented in Perot & Zhang [6]. The property that this method conserves kinetic energy both locally within cells and globally makes it a good choice for performing turbulence modeling.

Domain and Boundary Conditions

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 [7] – an automatic 2D Delaunay mesh-maker. There are approximately 20,000 triangles in our calculation (see Figure 2.)

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 Johnasson's paper.

$$U_{in} = 15.5 \text{ m/s}$$

 $k_{in} = (0.05U_{in})^2$
 $e_{in} = \frac{0.16k_{in}^{3/2}}{0.2\mathbf{l}}$

This inlet velocity is evaluated based on the total mass flow of their experiment. These values are also used as the initial value for the whole domain. \mathbf{l} is the height of the duct. A zero gradient boundary condition is used for all the variables at the outlet. A no-slip boundary condition is used for the duct wall.

RESULTS

Calculation of 2D unsteady turbulent flow around a triangle cylinder with Reynolds number $U_{in} H / n = 45,000$ is presented, where H is the height of the triangle. A Von Karmann vortex street is formed behind the triangle. No special triggering measure is taken to start the vortex shedding, the unsteadiness in the computational result evolved naturally.

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 3. It can be seen that an almost perfect periodicity exists. The shedding frequency is 99.98 (s⁻¹). The corresponding Strouhal number defined by,

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



Figure 2. Computational domain and mesh.



Figure 3. The stream-function of one point about one cylinder height behind the triangle near the centerline.

is 0.258, which should be compared with experimental data of 0.25 and the computed value of 0.27 in Johnnasson (1991).

The stream-wise velocity contour is shown in figure 4. The length of recirculation zone is about 25mm.

Figure 5 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. Although the instantaneous flow is asymmetric, the time-averaged fields are always symmetric or anti-symmetric.



Figure 5. Instantaneous velocity vector plot.

Figure 6 shows the stream-wise velocity at different locations behind the triangle. The calculated velocity profiles are in reasonable agreement with the experiment. However, far down stream in the wake zone, the velocity profiles do not recover as quickly as the experiments. The explanation for this is that we can not resolve the boundary layer in this computation due to the limit of our computing capacity. In the far field, the real wall turbulent boundary layer is much thicker than what was obtained in the calculation. Due to the conservation of mass, the experimental velocities near the centerline are thus larger than the computed ones.



Figure 4. Streamwise velocity Contours.



Figure 6. Mean stream-wise velocity behind the triangle: y, calculations; *, experiments. (a) 15mm, (b) 38mm, (c) 61mm, (d) 150mm, (e) 376mm

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 triangular cylinder are in reasonable agreement with experiment data. It is hypothesized that this agreement is a result of the non-equilibrium nature of the underling turbulence model.

ACKNOWLEDGMENTS

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

REFERENCES

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

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

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

[4] 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

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

[6] 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

[7] Jonathan Richard Shewchuk, *Triangle: Engineering a 2D Quality Mesh Generator and Delaunay Triangulator*, First Workshop on Applied Computational Geometry (Philadelphia, Pennsylvania), pages 124-133, ACM, May 1996.