NUMERICAL AND EXPERIMENTAL INVESTIGATION OF THE NEAR WAKE OVER A SLITTY BLUFF BODY

Yiqing DU[1] and Wenhua TANG[1]

[1] School of Energy and Power Engineering, Huazhong University of Science and Technology , Wuhan 430080, P.R. CHINA

ABSTRACT

The evolution of the near wake over a slitty bluff body was numerically investigated using the renormalization group (RNG) k- ε model at Reynolds number of 470,000. The slitty bluff body is to be used as a flame holder with a gap ratio from 0 to 0.48. The coherent structure of near wake was identified by the vortex shedding simulation. To explain vortex shedding, a mechanism that single vortex of large size suddenly immerses two shear layers was proposed. To quantitatively compare the near wakes at different gap ratio, a vortex shedding character dimension was first proposed. Particle-image velocimetry (PIV) measurements in a close wind tunnel were also carried out to confirm the observation from the numerical study. The evidence shows that the numerical results are of good agreement with the experiments.

NOMENCLATURE

k   turbulent kinetic energy
ε turbulent dissipation rate
Re   Reynolds number
U   velocity
Ls   vortex shedding character dimension
G/D gap ratio
τ normalized time

INTRODUCTION

Bluff body flame holders are widely used to make flames stable in industry, especially for combustors with high velocity flows. Many research conducted during the past four decades indicates that the wake flow structure behind the bluff body is directly related to the flame-holding performance. The wake contains the complexities of separation and recirculation, mass and momentum transport across the shear layers, and vortex shedding dynamics. However, the study of the flow structure has not been thoroughly explored. Yang and Tsai (1993) have experimentally investigated the effects of the width of gap on the recirculation zone and the drag in the near wake of a slitty bluff body (v-gutter) in a confined tunnel at Re=120,000. According to their experimental evidence, an asymmetric wake structure is developed behind the symmetric slit (G/D=0.2 and 0.4). Compared with traditional bluff body the slitty bluff body has better flame holding ability and less pressure loss because the gap flow induces great reverse flow and greater back pressure in the near wake. Their investigations also indicates that the gap flow provokes more extensive transport across the shear layers and reduces both the turbulent intensity and the Reynolds shear stress of the wake, but no more data with

different gap ratios and detailed analysis on the wake structure has been reported.

The understanding of flame holding mechanism of slitty bluff body is incomplete without understanding of the coherent structure in its near wake. Flow characteristics and vortical motion within the near wake are major interest of present work. The simulation on flows of engineering and industrial interest, characterized by high Reynolds numbers and complex geometry in confined tunnel, generally requires proper turbulence modeling. The analyses on different models, e.g. the standard $k - \varepsilon$, the RNG $k - \varepsilon$ and the large eddy simulation (LES), were provided. The flow structure behind the slitty bluff body was investigated numerically by using RNG k - ε . Experimental results using PIV measurements in a close wind tunnels were also provided to confirm the observation from numerical study.

NUMERICAL SIMULATION

Calculation model and condition

Figure 1 shows the geometrical parameters and the co-ordinate system used for the 2D case considered. The slitty bluff body of triangle cross-section with side dimension D and gap dimension G, is immersed in a uniform velocity stream U. The resulting characteristic Reynolds number (Re=ULi/ Υ) is fixed to 470000. The dimensions of the computational domain are (Li, Lx, Ly) = (250mm, 450mm, 200mm). The slitty bluff body is in the middle of the confined tunnel with its axis normal to the inlet side.
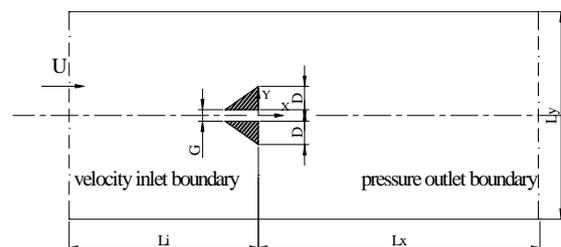
Figure 1: Definition of the co-ordinate and the geometrical parameters for the flow past a slitty bluff body

Those slitty bluff body problems involve very complicated flow behavior that cannot be accurately simulated using improper computational methods. In addition, the physical geometries are generally quite arbitrary and require special attention when attempting to predict the associated

flow. In most problems, the flow is unsteady and very turbulent. Traditionally, numerical simulations of these flows are performed using the Reynolds-averaged Navier-Stokes (RANS) equations and using phenomenological model to fully represent the turbulence. The available turbulence models vary extensively in complexity, in particular, from simple algebraic eddy viscosity relationships to complex formulations involving several additional differential equations. In each model, values for the empirical constants are obtained from turbulent flows that are usually fundamentally simpler than the complexity of the practical problem concerned.

It is theoretically possible to directly resolve the whole spectrum of turbulent scales using an approach known as direct numerical simulation (DNS) to capture the fluctuations. DNS is not, however, feasible for practical engineering problem, as DNS approaches are too computationally intense. An alternative for numerical simulation of complex turbulent flows is the large eddy simulation (LES).

The LES approach seems to be gaining popularity over the traditional RANS simulation, which relies primarily on the rapid advancements in supercomputer technology as well as the recent developments in the methodology itself. It should, however, be stressed that the application of LES to industrial fluid simulations is in its infancy. As highlighted in a recent review publication, typical applications to date have been for simple geometries. This is mainly because of the large computer resources required to resolve the energy-containing turbulent eddies. Most successful LES has been done using high-order spatial discretization, with great care being taken to resolve all scales larger than the inertial subrange. The degradation of accuracy in the mean flow quantities with poorly resolved LES is not well documented. In addition, the use of wall functions with LES is an approximation that requires further validation. Evidence shows that LES is improper to calculate the 2D slitty bluff body flow.

The RANS equations represent transport equations for the mean flow quantities only, with all the scales of the turbulence being modeled. The approach of permitting a solution for the mean flow variables greatly reduces the computational effort. If the mean flow is steady, the governing equations will not contain time derivatives and a steady-state solution can be obtained economically. A computational advantage is seen even in transient situations, since the time step will be determined by the global unsteadiness in the mean flow rather than by the turbulence. The Reynolds-averaged approach is generally adopted for practical engineering calculations, and uses models such as Spalart-Allmaras, k - ε and its variants, k - ω and its variants, and the RSM.

The RNG k - ε model is one of the k - ε variants of RANS derived using a rigorous statistical technique called renormalization group theory. It is similar in form to the standard k - ε model, but includes the following refinements:

- The RNG model has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard k - ε model uses user-specified, constant values.

- While the standard k - ε model is a high-Reynolds-number model, the RNG theory provides an analytically-derived differential formula for effective viscosity that accounts for low-Reynolds-number effects.

These features make the RNG more accurate and reliable for a wider class of flows than the standard k – ε model. Thus RNG k – ε model in FLUENT 6.0 is used in the present investigation using. Detail RNG model analysis is available in FLUENT6.0 user's manual.

The computational meshes employed are non-uniform grids. The number of grid cells is about 140,000 (slightly varied with different gap ratio), these quadrilateral cells are obtained with interval size 1mm using the Quad/Tri-Pave meshing Scheme in GAMBIT, which creates a paved mesh that consists primarily of quadrilateral elements but employs triangular mesh elements in any corners, the edges of which form a very small angle with respect to each other.

Velocity Inlet boundary conditions include the flow velocity, and all relevant scalar properties of the flow. The total properties of the flow are not fixed, so they will rise to whatever value necessary to provide the prescribed velocity distribution. Figure 1 defined the inflow velocity Re=470000 (normal to the boundary).

Pressure Outlet boundary conditions specify the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a higher rate of convergence when backflow occurs during iteration. The outlet boundary is far enough from the bluff body (x=Lx) with constant static pressure which can be measured experimentally.

In order to model the natural perturbations in any real flow, many numerical simulations usually use an explicit perturbation at the onset of the transient calculation. This numerical disturbance exists in the form of a deranged initial flow field often formed by applying slip velocity. This explicit perturbation is said to be necessary in order to disturb the Navier-Stokes equations and provoke or 'kick-start' the vortex shedding process by Anderson (1993). D.G.E. Grigoriadis et al. (2003) investigated incompressible turbulent flow past a long square cylinder using LES. They used a uniform stream U superimposed with Gaussian random divergence-free perturbations of intensity 2-5% w.r.t. the local value. After a transient time the flow rejects the initiate unrealistic condition and the shear layers at the cylinders' faces initiated vortex shedding.

In our investigation all flow fields were firstly calculated with a "stable" solver based on the assumption that the flow field is time-independent. This assumption is of signality, although it is undoubted that the real flow past a bluff body is a time-dependent problem. To investigate the intrinsic mechanism in the wake flow of a slitty bluff body, the inflow perturbation is unwanted. The perturbation can be reduced to a certain level, but it can never be removed completely or be diminished to very small level in a real flow. However the simulation can initiate the inflow perturbation to zero. All further time-dependent simulations are on basis of stable calculations.

Numerical result

The present investigation concentrates on the near wake, that is, the region at x < 200 mm. Development of the

wakes further downstream will not be considered in present study.

The stable simulation reveals that the canonical biased gap flow patterns can be identified at gap ratios 0.22-0.48. Corresponding example vector field at G/D=0.22 is given in Figure 2.
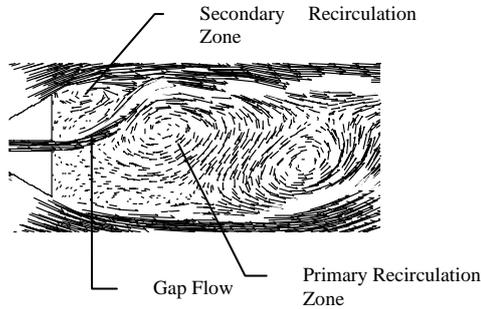
Figure 2. Coherent structure in flow over a slitty bluff body

Figure 2 shows that the gap flow divides the near wake into two recirculation zones, the big one is called primary recirculation zone (PRZ), the small one is called secondary recirculation zone (SRZ). Generally the recirculation zone in the near wake consists of a vortex pair closed in a zero streamline.

In most situations, the near wake flow at high Reynolds number is time-dependent and very turbulent. This unstable character depends on the perturbation and the flow structure itself. Roshko (1993) experimentally investigated laminar and turbulent wakes behind cylinders of different cross sections and observed geometric similarity among all vortex streets. From his data, he derived empirical features of vortex streets that are Reynolds-number independent, and he postulated a "universal" Strouhal number that is related to the wake width. The laminar and turbulent vortex-shedding process is slightly affected by other phenomena as well, e.g. oblique shedding modes, transition modes, Kelvin-Helmholtz instability, boundary-layer effects at the obstacle. However, the main characteristics of the vortex street are not strongly affected by these phenomena. Coherent structures in the near wake are related to fluctuation amplitudes of drag and lift and also to the mean flow quantities. All these evidence indicates that the stabilities of the wake and the perturbation itself intimately depend on the coherent structure of near wake.

Despite of the complexity of disturbance and the turbulent transition regime, we focused our investigation on the flow structure and how the structure influences the instability. All time-dependent computations begin with the stable converged results by setting the solver to unstable. The unstable numerical results reveal that gap flow shows obvious asymmetric in the wake flow field. The gap flow penetrating the leading open slit of the bluff body turns toward either the upper or the lower wing and then stays relatively 'stable' owing to the bi-stable status of the flow field at G/D=0.22-0.48. The gap flow does not diffuse symmetrically in the near wake region.

The concepts of absolute and convective instabilities have provided new physical insights into wake control since 1985. The difference between absolute instability and convective instability is that the impulse response of a fluid system can propagate in both upstream and downstream direction in an absolutely unstable flow,

whereas it can only propagate in the downstream direction in a convectively unstable flow region. Therefore, a flow system that contains a sufficiently large region of absolute instability will respond to external forcing by developing time-amplifying global oscillations, a response that is fundamentally different from that of a system that is convectively unstable everywhere. In wake flow, both types of instability exist. The near wake is governed by absolutely instability, the wake then changes to convective instability at short distance from the solid boundary due to the rapid filling of the velocity deficit. In the absolutely unstable region, wake flow acts as a resonator where all unstable disturbances are self-excited or time-amplified. In the convectively unstable region, wake flow is similar to a spatial amplifier in that all unstable frequencies (for small-amplitude perturbations) will grow exponentially along the downstream direction. Therefore, the disturbances will first resonate in the absolutely unstable region and then serve as an initial perturbation in the convectively unstable region. PRZ sheds a large vortex street, whereas SRZ has a small vortex pair, the vortex shedding cannot be seen behind SRZ. As PRZ and SRZ are in the near wake and governed by absolutely stability, the unstable disturbance of PRZ or SRZ has an impact on each other through the gap flow. This results in fluctuation in the SRZ and the swing of the gap flow.

There are three main factors that will play important role in the interference between PRZ and SRZ. One is the distance G between two recirculation zones. The numerical study reveals the distance nearly equals to the gap width, a smaller distance G can make it easier to transfer the interference. The second is the proportion between PRZ and SRZ, the bigger the deflection angle of the gap flow is, the bigger the proportion is. The third factor is the gap flow strength.

Vortex shedding mechanism

The investigation over the vortex shedding mechanism over a slitty bluff body is very important not only in engineering applications but also in understanding the basic nature of turbulent flows.

Our knowledge on vortex shedding largely bases on previous phenomenological resolutions on the near wake over the circular cylinder. Gerrard (1966) proposed the shear layers entrainment in the motion of boundary layers during the vortex formation process in bluff body. Other investigators also gave phenomenological describes on vortex shedding process behind two- dimensional bluff bodies on the basis of experiments. But no detailed vortex shedding mechanism has been reported.

The formation and the downstream transport of the Strouhal vortices in the near wake of a circular cylinder is an excellent example of a difficult bluff body flow. The vortices themselves derive most of their large-scale vorticity from the separated shear layers and they organize downstream to form the well-known Karman vortex street. The upper and lower separated shear layers constitute the transverse outer regions of the formation regime. Both layers lie between the point of separation and the initial formation of the shed Strouhal vortices. Besides these shear layers, the fluctuating base pressure near the downstream stagnation location contributes to the Strouhal vortex formation as well.

Our understanding of the cylinder shear layer physics improved significantly due to the experimental investigation by Bloor(1964), and Wei and Smith(1986).

Bloor hypothesized that transition from laminar separation to turbulent vortex formation occurs within the separated boundary layer. Through flow visualization testing, Wei and Smith proposed different phenomena for the separated shear layer physics. They observed separate shear layer physics. They observed creation of small-scale secondary vortices between separation and the initial large-scale Strouhal vortex formation.

Generally, all the above evidences indicate that the circular cylinder vortex shedding physics is very complex because of its separated shear layers and the interaction between the second eddies and the Strouhal vortices. Other factors, like the disturbance from free stream or even three-dimensional contributions make experimental investigation almost impossible to understand the real physics of vortex shedding. Our slitty bluff body model is simpler than the circular cylinder, for the reason that there is no separate shear layer on the surface. Further more the gap flow can be easily measured by experiment, which can be used as index to evaluate the numerical results and theory analysis.


(a)

(b)
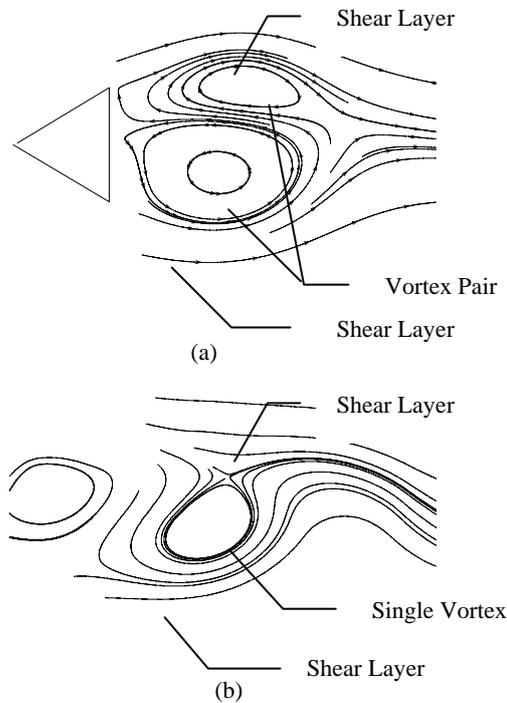
Figure 3. Vortex pair and single vortex between the two shear layers

A pair of vortices with different direction between the shear layers, see in PRZ, can be stable in the near wake at certain condition. For example, at low Reynolds number, there are a stable vortex pair in the near wake behind a bluff body, see in Figure 3a, the flow in the near wake is stable. As Reynolds number increases, the flow turns into unstable, one of the vortexes becomes bigger, finally it changes into a single vortex with a scale of 2D between two shear layers. A single vortex gains streamwise velocity from the shear layers so it will detach from the near wake, see in Figure 3b. Vortex shedding process was simulated by our unstable numerical computations. It should be noted that unlike some previous work the vortex shedding was not artificially initiated by numerically inducing slight twist to cylinder. For the present
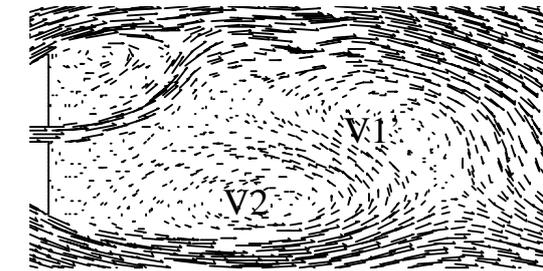
simulation, shedding of Strouhal vortices occurs naturally through time evolution of an unstable wake. The vortex shedding mechanism can be explained as following according to our numerical calculation.
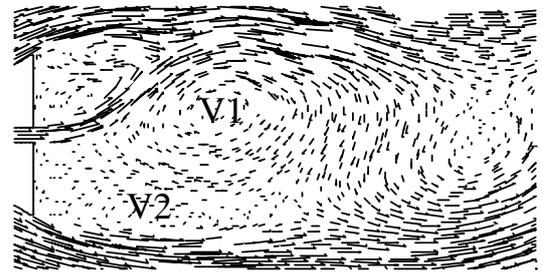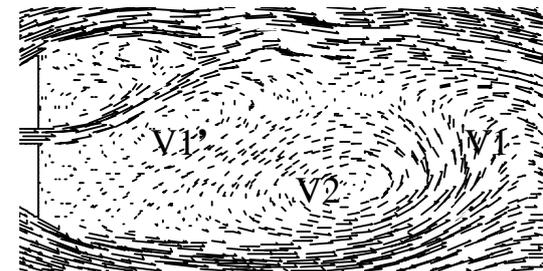

(a) $\tau = 133$
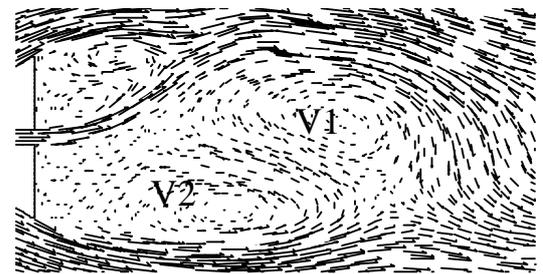
(b) $\tau = 1333$

(c) $\tau = 1600$

(d) $\tau = 2133$

Figure 4. The vortex shedding process in PRZ (G/D=0.22)

At a certain time $\tau = 133$, where the normalized time $\tau = tU/D$, there are a pair of vortices named V1 and V2 in PRZ. The smaller one V2 grows at $133 < \tau < 1333$, gradually pushing the bigger one V1 downstream. At $\tau = 1333$, V2 has the same size of V1. At $\tau = 1600$, V1 is put forward between the two shear layers just like the single

4

vortex mentioned above, getting a speed from the shear layers quickly. With a streamwise speed, V1 soon detaches from the near wake at 1600< τ <2133. At the sudden detachment of V1, the near wake initiates another small vortex V1' of the same direction of rotation. See Figure 4a~4d. This vortex shedding process can be divided into two stages, the 'growing' stage and the 'detaching' stage. At the "growing" stage, the small vortex becomes bigger and bigger slowly. At 'detaching' stage, the single vortex suddenly detaches downstream. The first stage spends more time ($\triangle \tau 1$) while the 'detaching' stage spends less time ($\triangle \tau 2$). The ratio $\triangle \tau 1/\triangle \tau 2$ is about 3 by our calculation.

(a)

(b)
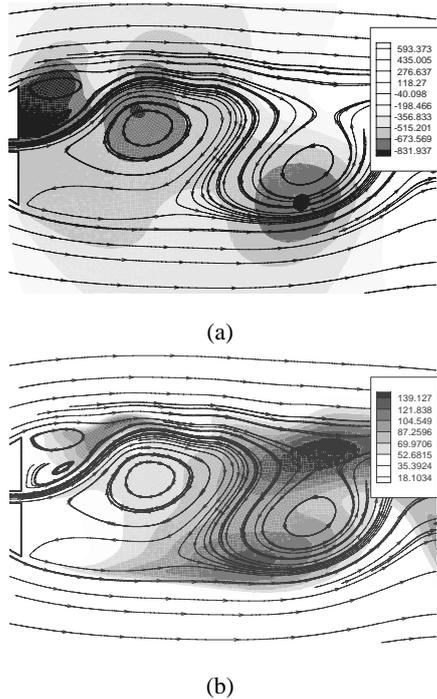
Figure 5. Instantaneous character of the near wake (a) static pressure (b) turbulent kinetic energy

The instantaneous character of vortex detachment in near wake is depicted in Figure 5. Yet the vortex shedding phenomena in PRZ over a slitty bluff body is largely the same as the near wake of a cylinder known as Von Karman vortex street. A single vortex confronts with the two shear layers, exchanging momentum and energy with the mean flow. At the same time, the vortex gets a speed of X component which makes the vortex shedding, and initiates sudden change of pressure and kinetic energy. Figure 5 depicts the disperse distribution of P and *k* at the detachment moment. The kinetic energy gathers near the saddle point and low pressure at the center of the vortex collapses, leads to the disturbance. The pressure disturbance can propagate in all direction at sound speed and the average velocity is low in the near wake with back flows, so the disturbance can propagate upstream. That is the reason why the near wake is absolute instable.

To compare the time-dependent flow over the slitty bluff body new character dimension is needed. According to the above model of the single vortex between two shear layers, we first define the vortex shedding character dimension Ls as the streamwise distance between the centre of the single vortex and the rear edge of bluff body. The single vortex gets momentum from the shear layers and detaches from the near wake rapidly, the detached vortex does not contribute to the flame instability, so the definition of Ls is of great signality in quantitively comparing the time-dependent wake flow over slitty bluff body.

Ls is the distance to the vortex at the moment of shedding. To quantify Ls, the global average mean can be used to get average velocity vectors and turbulent kinetic energy distribution in the near wake, Ls is determined from the critical point or from the point of maximum k.

PIV MEASUREMENT

The tested bluff body with a slit in a wind tunnel is shown in Figure 1. The experiments were conducted in a close-circuit wind tunnel blown by a blower, the speed of which was controlled by a frequency inverter. The free stream turbulence was controlled by a turbulence-generating screen place at the entrance of the test section, 800 mm upstream of the bluff body. The free stream turbulence intensity is lower than 0.5% and the non-uniformity of the mean flow distribution is less than 0.5%. Figure 6 gives a typical instantaneous velocity vector field.

(a)

(b)
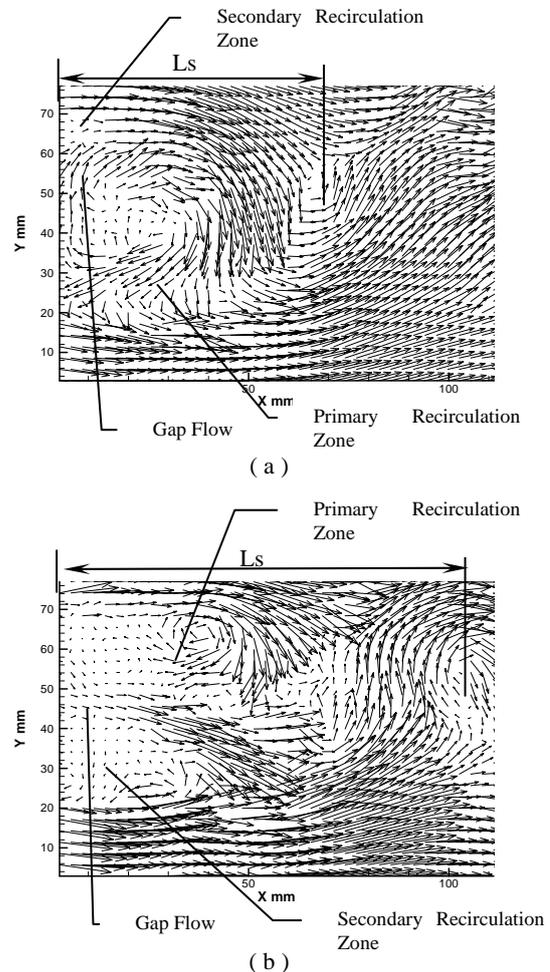
**Figure 6**. Instantaneous vectors of （a）（G/D=0）and （b）（G/D=0.22）by PIV

The velocity vectors by PIV also show the gap flow is biased. The results are in good agreement with the RNG simulation. By comparing the instantaneous velocity vectors, the coherent structures are similar except the

amplitude of disturbance of near wake by PIV is greater than the simulations, which is probably attributed to the greater disturbance in the free stream in experimental flow.
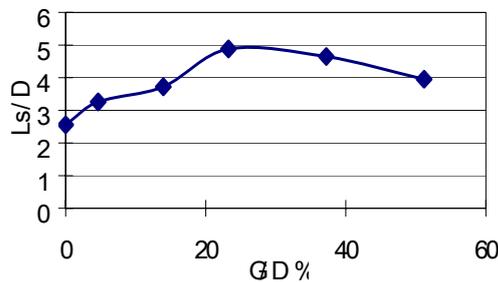
Figure 7. Effect of gap ratio on change of vortex shedding character dimension

Figure 7 gives the vortex shedding character dimension Ls at different gap ratio G/D. The Turbulent kinetic energy contour using the global average mean (Figure 8) shows that maximal k is exactly at the vortex shedding point (Ls from the bluff body rear edge). This evidence proves the above vortex shedding dynamics characterised by the Ls. The near wake gains a wider k distribution but less in the scale as the Ls increases. Our numerical study also reveals the same trend.

(a)

(b)
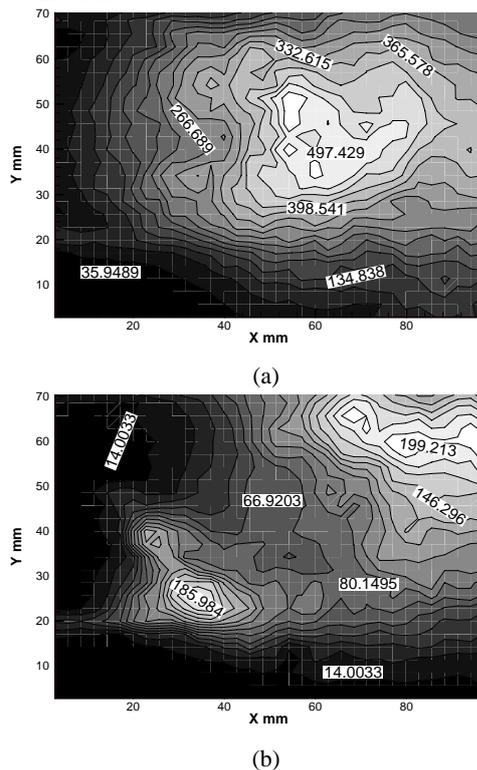
Figure 8. Turbulent kinetic energy contour by PIV (a) G/D=0 (b) G/D=0.22

CONCLUSION

Impetus of the present RNG k - ε simulation was to investigate the coherent structure of the near wake behind a triangle slitty bluff body. We computed the flow in the near wake behind a slitty bluff body at gap ratios from 0 to 0.48 at a Reynolds number of 470,000. Based on this study, the following concluding remarks are offered:

Detail analysis on the RNG k – ε model shows that this model is adequate in computing the near wake over bluff body, especially better than the standard k - ε model and LES.

The flow in the near wake behind the slitty bluff body is unstable structure. Without any perturbation initiation (kick-start), we got a time-dependent flow field. The reason is that there are numerical disturbance during the iterations.

At most gap ratios 0.22-0.48, the gap flow deflected to one side, and divided the near wake into two zones, the primary recirculation zone and the secondary recircularion zone. The near wake absolute instability lies on the interaction between PRZ and SRZ.

Bases on the computational results, the vortex shedding was analysed. A single vortex between shear layers mechanism was proposed. The production of perturbation and instability mainly depends on this sudden confrontation between a single vortex of large size and the two shear layers. Vortex shedding character dimension was first proposed to compare time-dependent wake according to vortex shedding dynamics.

The computations were further compared with PIV measurements. PIV vector field and Turbulent kinetic energy contour indicates that the simulation can get right information on the coherent structure of bluff body flow.

REFERENCES

YANG JT, Tsai GL,(1993), "Near wake flow of a V-Gutter with slit bleed", *J. Fluids Engineering,*115, 13-20.

ANDERSON CR,(1990), "On the accurate calculation of vortex shedding", *Phys Fluids,*A 2(6), 883-885.

GRIGOIADIS DGE, Bartzis JG,(2003), "LES of the flow past a rectangular cylinder using the immersed boundary concept",*Int J. Numer Meth Fluids,*41,615-632.

ROSHKO A, (1993), "Perspectives on bluff body aerodynamics", *J. Wind Eng. and Ind. Aero.,* 49,79-100.

GERRARD JH, (1966), "The mechanics of the formation region of vortices behind bluff bodies", *J. Fluid Mech.*, 25,401-413.

BLOOR MS., (1964), "The transition to turbulence in the wake of a circular cylinder", *J. of Fluid Mech.*,19,290

WEI, T., and SMITH CR., (1986), " Secondary vortices in the wake of circular cylinders", *J. of Fluid Mech.*,169,513-533.