Proceedings of Fifth International Conference on **Enhanced, Compact and Ultra-Compact Heat Exchangers: Science, Engineering and Technology**, Eds. R.K. Shah, M. Ishizuka, T.M. Rudy, and V.V. Wadekar, Engineering Conferences International, Hoboken, NJ, USA, September 2005.

CHE2005 – 58

SIMULATION OF FORMATION MECHANISMS AND TRANSPORT OF LONGITUDINAL VORTICES IN 3D BOUNDARY LAYER

K. Khallaki[*], S. Russeil and B. Baudoin

École Des Mines De Douai, Département Énergétique Industrielle, 941, Rue Charles Bourseul
B. P. 838, 59508 DOUAI Cedex - France
[*] E-mail: khallaki@ensm-douai.fr

ABSTRACT

This paper presents a numerical study of the mechanisms of development and transport of longitudinal vortices. Computations are carried out in three-dimensional turbulent boundary layer using three turbulence models: Realizable $k-\varepsilon$, SST $k-\omega$ and RSM. Results are compared to experimental data of Pauley and show that characteristics and turbulent structures are accurately predicted.

INTRODUCTION

Heat transfer enhancement in turbulent channel flow can be achieved by introducing artificial surfaces on the wall such as vortex generators (*VG*). These obstacles produce longitudinal vortices which have been used in many industrial applications. This vortices cause the swirling of the flow around the main flow direction and increase the mixing between the fluid and the hot wall.

Several studies have been performed to analyze the effect of *VG* on the turbulent boundary layer and the mechanism of formation and transport of embedded vortices. Pauley (1988) has conducted experiments on longitudinal vortices imbedded in turbulent boundary layer produced by a pair of *VG* with changing different geometric parameters. He showed that the major effects of the pairs are the thickening of the boundary layer in regions called upwash where the secondary flow is away from the wall and thinning in downwash region where this flow is toward the wall. He also concluded that the heat transfer enhancement in the common flow down pattern is much larger than common flow up pattern because of the stronger interaction with the wall. Tiggelbeck et al (1992) performed local and mean heat transfer measurements for one-row and aligned double-row configurations of delta winglet pairs in a wind tunnel. They found that the local heat transfer is highest near the rear edge of second-row but decreases a little faster in the streamwise direction than in the rear of the first row. Fiebig et al (1995) studied many geometries of *VG* and they concluded in their works that wing-type generators are especially suited for heat transfer enhancement of compact heat transfer surfaces. They showed from numerical investigations that the maximum average heat transfer enhancement of up to a factor five could be achieved at small angles of attack for the in-line symmetric winglet-*VG* configuration but the additional pressure loss is also high. Torii et al (2002) proposed an innovative strategy that augment heat transfer and reduce pressure-loss in finned tube heat exchanger in relative low Reynolds number flow, by deploying delta vortex winglet-vortex generators. In case of staggered tube banks, the heat was increased by 10-30 % and pressure loss was reduced by 34-55%. In the case of in-lined tube banks, the heat transfer was augmented by 10-20% with pressure loss reduction of 8-15 %.

Many computational studies have been performed by Lee et al (1999) for three-dimensional turbulent boundary layer with longitudinal vortices. Their results, computed by the Reynolds stress Model (RSM) and Standard $k-\varepsilon$ model and compared with experimental data, showed that the RSM model gives the best predictions. Yang et al (2001) studied the effect of the flow field and the heat transfer created by interactions between a pair of vortices in rectangular channel flow. They used the two-layer turbulence model and their results for vortex characteristics are reasonably close to experimental data.

Nevertheless, It is worth to note that in these studies, the *VGs are* not inserted in computational domain and longitudinal vortices are supposed to be already generated at the entrance of this one. Indeed, they are modeled as Rankine vortices or the experimental data are set. As consequence, these works only concerned the transport of longitudinal vortices behind *VG*.

We propose, in the present paper, to introduce the *VG* in the computational domain and study numerically the generation of longitudinal vortices in turbulent boundary layer and their transport downstream. Our investigations also aim to the better understanding of turbulence in this flow topology and check the ability of CFD codes to produce accurate predictions of the flow and heat transfer characteristics using several RANS turbulence models.

NUMERICAL MODEL

A computational fluid dynamics code (Fluent) was used to simulate flow field. This code uses a control volume technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists in integrating the governing equations on each control volume, yielding to discrete equations that conserve each quantity on a control volume basis. Fluent stores discrete values of the conserved quantity at the cell centers and uses an upwind scheme for determining face values of the conserved quantity for the convective terms. A second–order scheme is used for higher accuracy. In order to provide the pressure field, the SIMPLEC scheme is adopted in the present solution. An unstructured grid is generated for the volume of *VG* and quadrilateral meshes are generated otherwise. The computational domain contains two millions cells essentially refined in the region where the longitudinal vortices are developed.

COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

Simulations are carried out in an identical configuration of Pauley: two meters long rectangular channel and a cross section of 13×30.5 cm^2 (figure1).

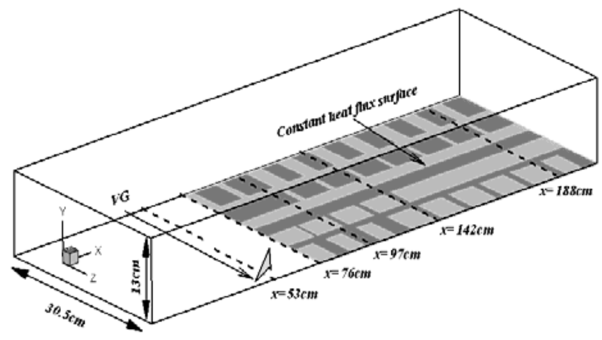


Figure 1. Computational domain

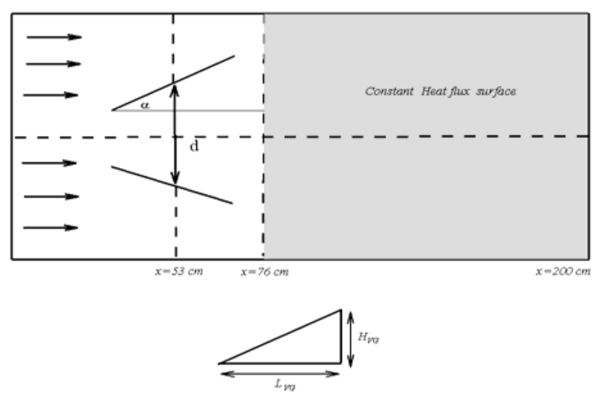


Figure 2. Common flow down configuration

Pairs of streamwise vortices were generated in the test section using half-delta-wing vortex generators, (2 cm high and 5 cm long), mounted at an angle of attack of 18° on the test wall, 53 cm downstream the inlet. The distance “d” between the *VG* is equal to 4 cm. A common flow down configuration is chosen (figure2). The flow enters with a nominal velocity of 16 m.s^{-1} and free-stream turbulence intensity of 0.3%. The Reynolds number based on the half height of the channel, $Re_{H/2}=U_0H/2\nu$, is equal to 67000. The heated surface start 76 cm downstream the inlet test section with a constant heat flux q = 817 W.m^{-2}.

RESULTS AND DISCCUSIONS

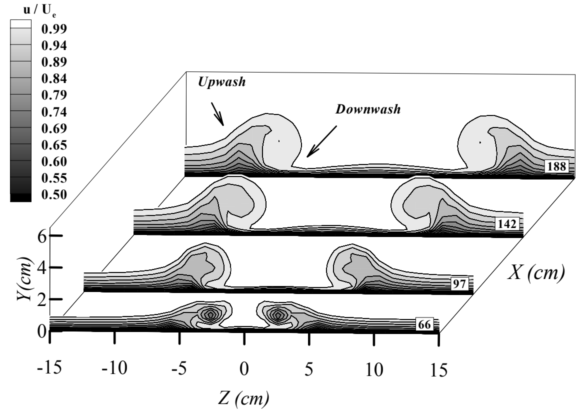
Dynamic and thermal results of computations are presented and discussed in this section. Streamwise velocity contours and secondary velocity vectors, given by SST $k-\omega$ model, show the flow pattern in this configuration. Profiles of velocity components and spanwise Stanton number are compared with experimental data of Pauley at different plans and locations.

Flow characteristics

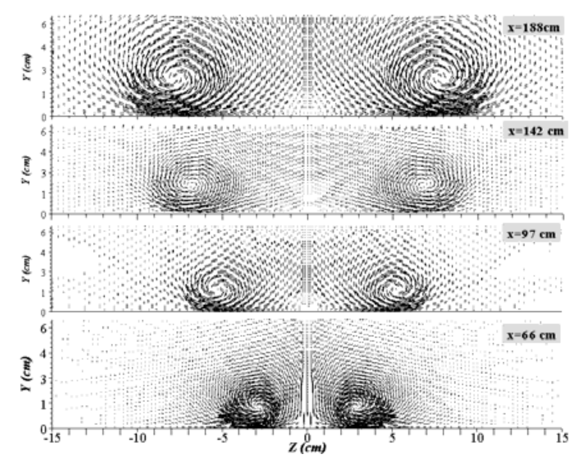
Figures 3a and 3b illustrate respectively the contours of streamwise velocity (*u*) and secondary velocity vectors (*v,w*) at four locations downstream the *VG* (*x = 66, 97, 142 and 188 cm*). It can be seen from figure 3a that vortices cause a strong modification of the boundary layer. It is thickened markedly in the upwash region where the vortex sweeps low momentum fluid away from the wall. In the downwash region, the boundary layer is thinned by the strong downflow and lateral outflow of boundary layer fluid. The streamwise velocity contours indicate a significant streamwise velocity deficit in the central region of the vortex. The velocity deficit decreases moving downstream. Velocity vectors of secondary flow are shown in figure 3b. We observe that the general features of the flow studied and described by Pauley (1988), are well reproduced by computations: the downwash flow toward the wall is observed between the vortices, and the upwash flow away from the wall is found in the outside region of the vortices.

Figures 4 and 5 show comparisons between numerical and experimental streamwise velocity profiles (*u*), normalized by the mean velocity *Ue*, at three spanwise positions *z = 3.5, 5 and 7 cm* for the plan *x = 97 cm* and *z = 6, 8 and 10 cm* for the cross section *x = 188 cm*. *Z* locations correspond respectively to the downwash region, the center of vortex and the upwash region. These locations present the most difficult regions in the flow field to be modeled due to strong secondary velocities and mean velocity gradients. These profiles indicate that the vortex pair causes significant distortion of the boundary layer including a region of negative velocity gradient in the center of the vortex. Numerical results indicate that RSM and SST $k-\omega$ models reproduced correctly experimental profiles of streamwise velocity in all regions of the cross-section at *x = 97* cm and *x = 188 cm*. In vortex core region, results obtained by the RSM model are in slightly a better agreement with experimental data and SST $k-\omega$ model matches successfully experimental results in the upwash region.

From figure 6 of profiles of normalized vertical velocity (*v*) at *x=97cm,* we observe some discrepancies between computations and experimental data. Vertical velocities are not so correctly predicted by simulation, but the difference between computations and experiments remains very low and presents 1 to 3% of the v/Ue value.

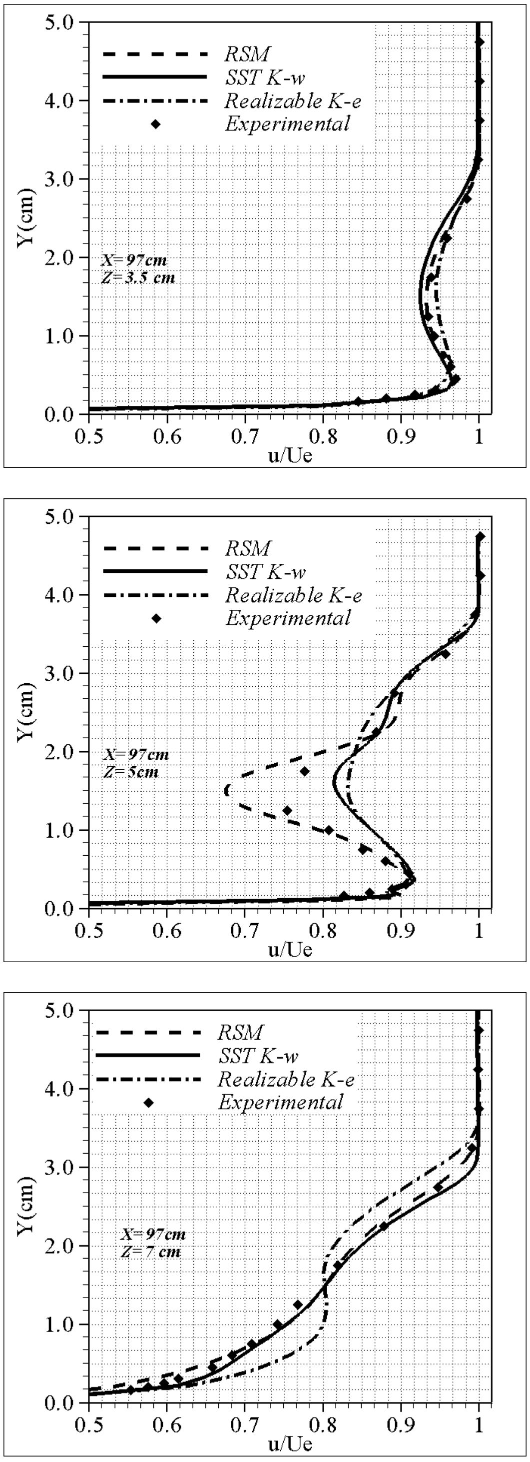


(a)



(b)

Figure 3. (a) Contours streamwise velocity
(b) Secondary velocity vectors – SST $k-\omega$ model



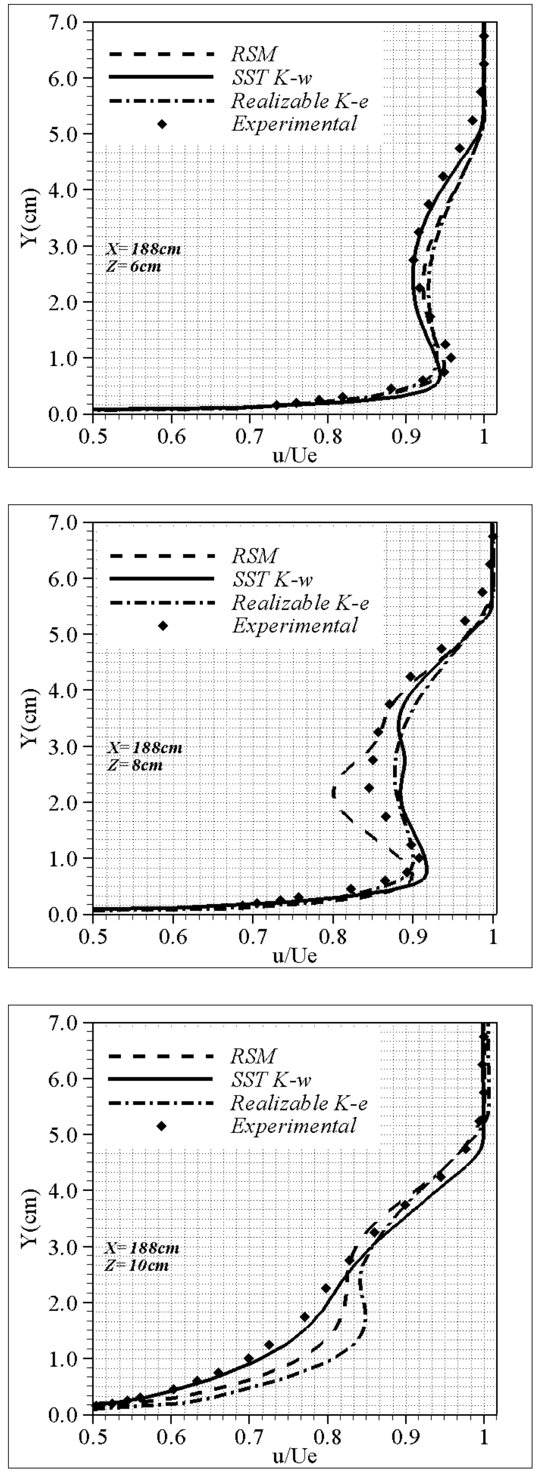


Figure 4. Normalised streamwise velocity profiles (u/U_e) at X = 97cm at different Z locations

Figure 5. Normalised streamwise velocity profiles (u/U_e) at X = 188cm at different Z locations

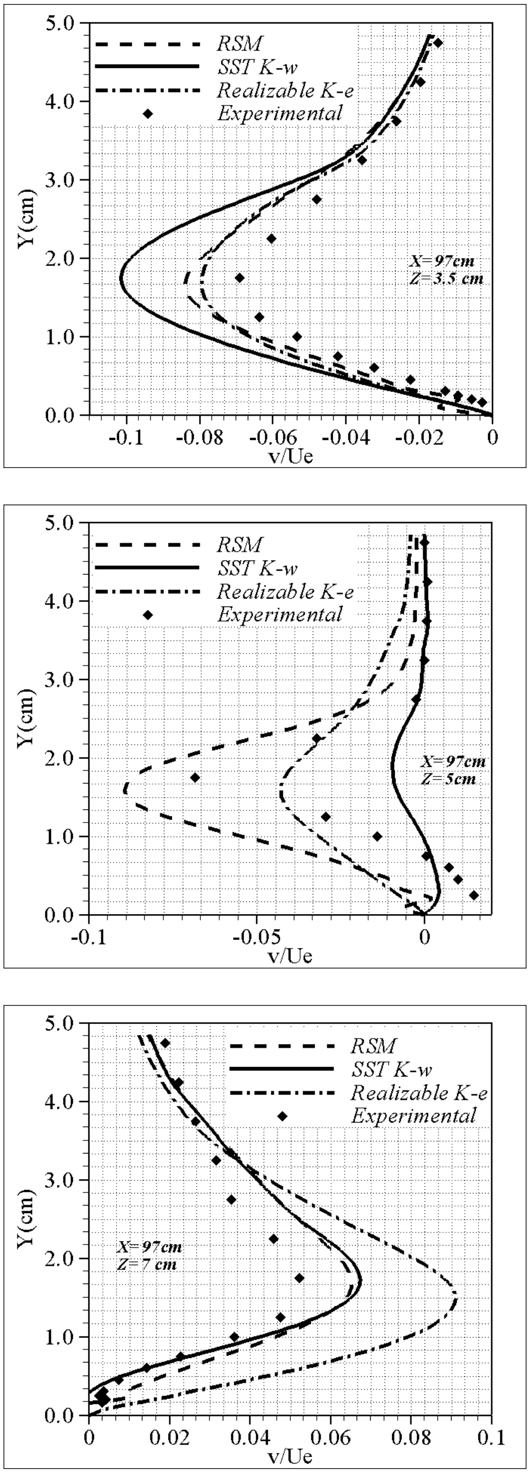


Figure 6. Normalised vertical velocity profiles (v/U_e) at X = 97cm at different Z locations

Heat transfer characteristics

It can be seen from spanwise profiles of Stanton number (figure 7) that the maximum heat transfer is located in the region where the secondary flow is directed toward the wall (downwash region) and the minimum values of heat transfer are found in the upwash region and when the secondary flow goes away from the wall. The peak level is typically 25 % greater than the undisturbed level. These peaks have a greater magnitude at the upstream location (x = 97 cm) than at the downstream location (x = 188 cm) but are distributed over a larger spanwise extent at the downstream location. All turbulence models reproduce qualitatively experimental data. Quantitatively, results of computations obtained by RSM and SST $k-\omega$ models are in better agreement to the experimental data than Realizable $k-\varepsilon$ model.

CONCLUSIONS

A numerical study has been carried out using different turbulence models to investigate the mechanisms of development of longitudinal vortices and examine their effects on dynamic and thermal turbulent boundary layer. Contrary to the other works, *VGs* are introduced in the computational domain in order to study the formation and transport of longitudinal vortices. This can be useful to get more information about operation conditions and design for heat exchangers. The major conclusions of this study are:

1. The main effects of vortex generators are the thickening the boundary layer in upwash region and thinning in downwash region.
2. The downwash region is a region of high heat transfer but the vortex core region is the region of lower mixing.
3. Computations are in good agreement with experimental data and the RSM and SST $k-\omega$ model produce the more accurate predictions for flow field and heat transfer.

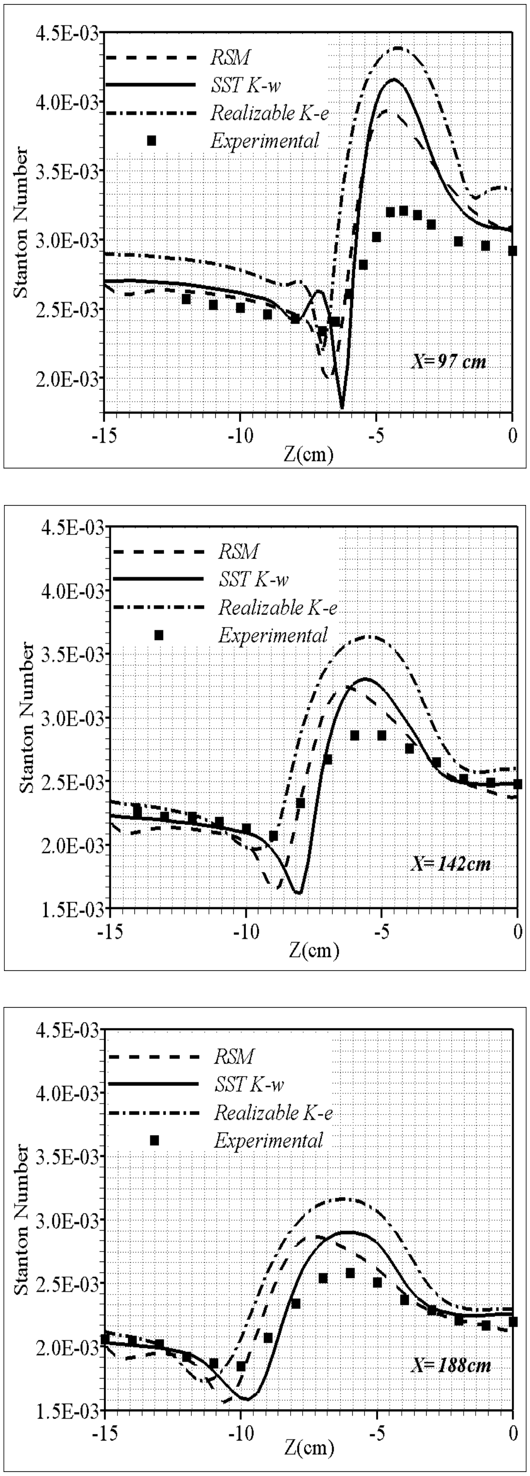


Figure 7. Spanwise distribution of Stanton Number at different X locations

NOMENCLATURE

H Height of the channel, m
k Turbulent kinetic energy, J/Kg
$Re_{H/2}$ Reynolds number
St local Stanton number
U_0 Velocity inlet, m/s

Greek symbol

ε Dissipation rate of turbulent kinetic energy
ρ Density of air, $kg.m^{-3}$
ν Kinematics viscosity, m^2/s
ω Specific dissipation rate of turbulent kinetic energy, s^{-1}

ACKNOWLEDGEMENTS

Financial support for this research work has been granted by European Community (FEDER)

REFERENCES

Fiebig, M. (1995) Vortex generators for compact heat exchangers, Journal of Enhanced Heat Transfer, 43-61.

Lee, S.H., Ryou, H., and Choi, Y. (1999) Heat transfer in three-dimensional turbulent boundary layer with longitudinal vortices, International Journal of Heat and Mass Transfer, 1521-1534.

Pauley, W.R. (1988) The fluid dynamics and heat transfer effects of streamwise vortices embedded in a turbulent boundary layer, PHD Thesis, Stanford University.

Tiggelbeck, S., Mitra, N., and Fiebig, M. (1992) Flow structure and heat transfer in a channel with multiple longitudinal vortex generators, Exp. Thermal fluid Sci, 425- 436.

Torii, K., Kwak, K.M., and Nishino, K. (2002) Heat transfer enhancement accompanying pressure-loss reduction with winglet-type vortex generators for fin tube heat exchangers, International Journal of Heat and Mass Transfer, 4327-4338.

Yang, J.S., Seo, J.K., and Lee, K.B. (2001), A numerical analysis on flow field and heat transfer by interaction between a pair of vortices in rectangular channel flow, Current Applied Physics, 393-405.