Proceedings of ENCIT 2010

13th Brazilian Congress of Thermal Sciences and Engineering
December 05-10, 2010, Uberlandia, MG, Brazil

NUMERICAL SIMULATION OF FLOW OVER A BACKSTEP: Prediction of Velocity and Temperature Profiles

Tamires Lopes de Azevedo, tamires_loaz@yahoo.com.br
Yuu Itai , yuuitai@ufpa.br
Danielle Guerra, drsguerra@hotmail.com
Laboratory of Fluid Mechanics, Faculty of Mechanical Engineering, Federal University of Pará, Belém-PA-Brasil, CEP:66075-110.

Abstract. *Turbulent separated flows with reattachment were classified as complex flows, characterized with high turbulent intensity and reverse flow. The phenomena of separation of turbulent boundary layer have been investigated for many years, since the presence of adverse pressure gradient and hence the appearance of separation and reattachment, have a practical and considerable significance, because they are of great importance in the efficiency of many projects such as hydrodynamic: the jet engine, airfoils with separation bubbles, helicopter blades, turbine blades, supported bodies, combustion engines, pipelines with expansion project in logical systems that involves fluid. The separation is a result of a slowed movement of the fluid adjacent to the wall, and the friction's force is an important factor in this process. The presence of a separation zone along with reattachment flow, increases the instability, pressure fluctuations, vibrations in structures and noise. Therefore, problems with separated flow cover a wide velocity range with many types of boundary condition, where the shapes, properties, and the phenomenon manifestation are inexhaustible. Thus, the surveys were conducted in this matter were primarily motivated to understand the physics of the phenomena of separated flow and a commitment to find possible methods to mitigate the negative impacts of the process. In this context the present work is a numerical simulation study of fluid dynamic phenomena of separation, which was conducted with turbulent flow using air as fluid. The simulation was conducted over a step in a wind tunnel open circuit with high intensity turbulent. The aim of this work is to present the effects of the turbulent air flow in a wind tunnel open circuit with a step of 50 mm x 670 mm through the commercial code FLUENT 6.0. Three turbulence models were applied to obtain the mean velocity field, pressure gradients and temperatures. In this study the grid of the geometry was developed with the same dimensions of the wind tunnel that was used by Guerra et.al. (2000). The wind tunnel has a section of 670 mm X 670 mm and the of wall temperature may be increased until 383,15K through electric current which passes through two resistances and is controlled by two thermostats.*

Keywords*: separation, reattachment, boundary layer*

1. INTRODUCTION

Turbulent flows with separation, as known as a complex flow, were more frequently in nature and have been studied due the various practical applications and technologies. Despite the many studies on this type of flow, the structure of the separation region is not completely understood. In these flows the boundary layer separation is followed of a reattachment in solid surface, where is observed the presence of undesirable pressure gradient.

Studies on this subject have as the main the understanding the separation and reattachment phenomena, this way the knowledge can provide some control of the adverse impacts that can affect both positively and negatively the efficiency of various fluid dynamic applications.

The principal features of turbulent BFS flow are described as a turbulent boundary layer which develops on a flat plate, encounters a backward facing step of height, H. The sudden change in surface geometry causes the boundary layer separation at the sharp step edge. The behavior of the resulting flow at downstream position is like a free shear layer, with a higher velocity on the upper side and lower velocity on the bottom side. In some downstream distance, the shear layer impinges on the surface and then forms a closed recirculation region containing turbulent moving. A small counter-rotating, a "corner eddy", developing below the mean recirculating bubble flow may also exist in this region. The instantaneous location of reattachment varies slightly in time and can be found about a mean position, which means that is not a point but a region. Downstream of reattachment, the boundary layer begin to develop a standard turbulent boundary layer state.

Due the simplicity of the geometry, the problem of turbulent flow in a flat plate with a backstep and a bidimensional flow has been used in the investigations of flows with separation. This configuration is a fluid mechanics classic case, often is used to analyze turbulent models performance. The backward facing step (BFS) flow has been extensively studied, but many aspects of the fluid dynamics and the structure of the flow remain incompletely explained.

Komoda (1961) have determined the surface pressure distribution and velocity fluctuations of a flow over a step, and flow over a rectangular cavity. The experiments were carrying out in a wind tunnel.

Simpson et al. (1974) realized an experimental boundary layer analysis with separation using laser anemometry and observed that the law of the wall is only valid until the region next to separation point. In this region the terms of motion equations associated with the normal stress have a great importance.

Dimitriev (1990) analyzed the mechanism of turbulent boundary layer with separation under the influence of a positive pressure gradient and showed that the separation is determined by the nature of the flow in a certain part of the internal boundary layer, where friction is an important factor. This region is highly exposed to the action of positive pressure gradient and where the stagnation zone first appears.

In a study realized by Chen et al. (2006), simulations of turbulent convection flow adjacent to a two-dimensional backward-facing step are presented to explore the effects of step height on turbulent separated flow and heat transfer. Two-equation low-Reynolds-number model is employed to achieve the turbulent Prandtl number. In this work, they noted that the primary and secondary recirculation regions increase in size as the step height increases. It's noted that near the step and below the step height, the turbulent kinetic energy becomes smaller as the step height increases.

A numerical study based on the large eddy simulation methodology was made of heat transfer in locally disturbed turbulent separated and reattached flow over a backward facing step. In this study the local disturbance was given to the flow by a sinusoidal blowing/suction of the fluid into a separated shear layer. Mehrez et al. (2009) observed a great amount of disturbance, St = 0.25, where heat transfer is significantly higher in the recirculation zone. The influence of frequency and amplitude of the disturbance, the positions of maximum heat transfer and maximum local Nusselt number were studied in this work.

Among several authors who studied the heat transfer phenomena in flow over step, Abu-Nada (2007) in their work made the study using nanoparticles as Ag, Cu and TiO2 in the fluid. In this work it was obtained the distribution of the Nusselt number on the walls of the tunnel.

The aim of this work is to compare three turbulence models inserted on the computational code FLUENT 6.3 analyzing their performances to provide the behavior of the flow in a presence of a backstep. This study provides the mean velocity data and pressure distribution in a two-dimensional turbulent flow in a channel. The numerical results are compared with experimental results of velocity profiles obtained by Guerra (2000).

2. METHODOLOGY

2.1. Experimental description

The measurements were obtained in an open channel wind tunnel located at the Laboratory of Turbulence Mechanical - COPPE/UFRJ. The wind tunnel has 670 *m*m X 670 *m*m X 3000 *m*m, a high turbulent intensity and the flow velocity can varies between 0 to 3,0 m/s. The test section is divided into three equal sections and many configurations can be adapted. The flow is generated through a centrifugal blower equipped with a filter; the velocity is controlled by a frequency converter coupled to the engine. The section of stabilization has one meter and the test section has two meters which has smooth surfaces and a heating system. The heating system of the inner surface are made by stainless steel, so the heating is produced by electrical currents that pass through two resistors on the plates which allow temperature elevation on the wall up to 383,15 K.

All the functions in the wind tunnel were controlled by a microcomputer. The bench of instrumentation has computers for acquisition, oscilloscope, hot wire anemometer, multimeter, and an automatic position system for sensors, there is also a thermometer attached to the inner wall at the top for temperature control. A Pitot tube with about 8.0 *m*m external diameter and 457.2 *m*m in length was used as a reference. More precise details about the experimental apparatus can be obtained at Guerra (2000).

2.2. Numerical description

The tunnel geometry and numerical simulation were built using the computational codes Gambit and Fluent 6.3. The geometry was created using the same dimensions of the original wind tunnel that was used to carried out the experiments by Guerra (2000).The backstep has 1.0m in length, 0.67 in width and 0.05m in height. The tunnel with the backstep geometry is shown in Fig.1.

It is important to mention that the grid was developed by dividing the channel in 11 parts, this division facilitate the grid creation in the step and provided a mesh growth factor, the grid geometry can be seen in Fig. 2. The grid contains around 67000 elements with fixed zones of velocity inlet, pressure outlet and walls. Elements were used in spacing QUAD MAP mode ranging between 0.05 and 0.08 depending on the area of geometry.

In the computational code was fixed the same initial conditions used by Guerra (2000), as show in the Table 1. It was selected three turbulence models that will be compared with experimental data. The models used for development this work were the standard K-ε, the RNG K-ε model and Reynolds stress model.

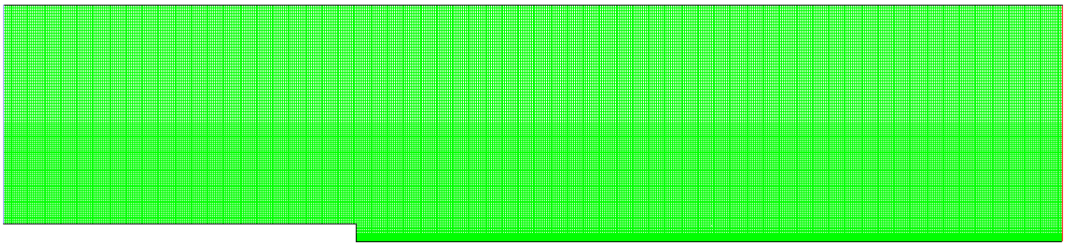

Fig. 1 – Geometry of wind tunnel generated in Gambit.

Fig. 2 – Illustrative scheme of the grid geometry.

Table 1 – Initial conditions of flow

Initial conditions	
Velocity inlet	3.0 m/s
Step height (H)	0.05m
Initial temperature	21°C
Pressure outlet	101325 Pa

3. RESULTS

Three turbulent models was used to provide results of the flow over a backstep, however the following figures show only the results obtained with the standard K-ε model. In general it was observed that the pressure distribution along the channel walls is strongly determined by the backstep. The pressure gradients induced by the step are clearly observed.

The pressure contours are displayed in Fig.3. There is a slight negative pressure gradient just prior to the step as indicated by the blue color in the colormap. Downstream the reattachment point, the pressure gradient remains positive down and then it reverses to an adverse pressure gradient. In the Fig. 4 we can see a red region that indicates a static pressure peak that must be the reattachment region. An amplified view of the velocity vectors of the recirculation region is shown in Fig. 5.

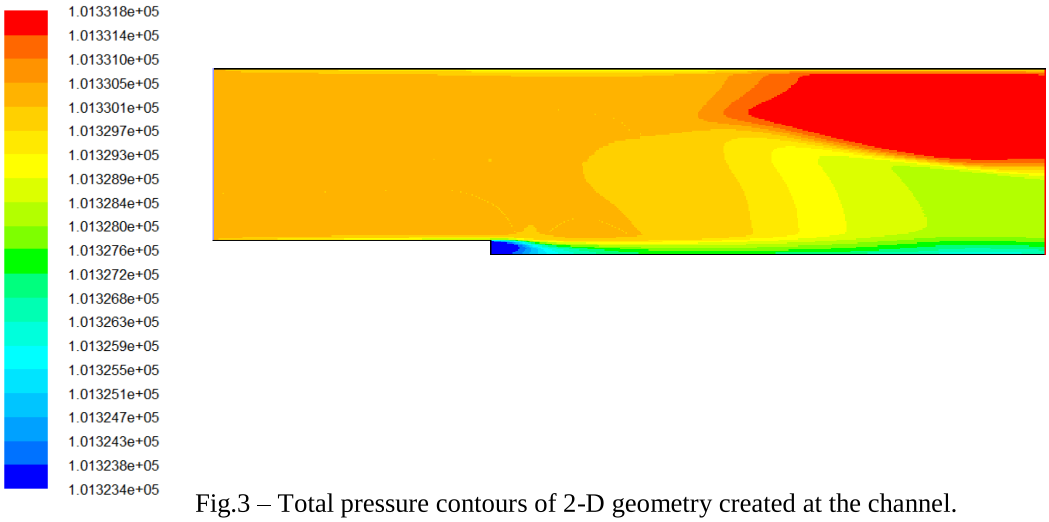

Fig.3 – Total pressure contours of 2-D geometry created at the channel.

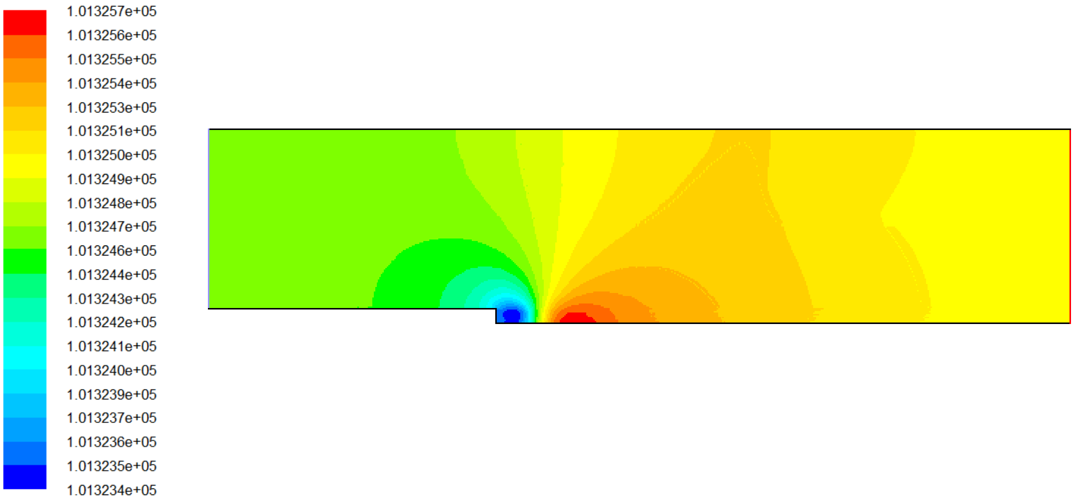

Fig. 4 – Static pressure contours in 2-D geometry created in wind tunnel

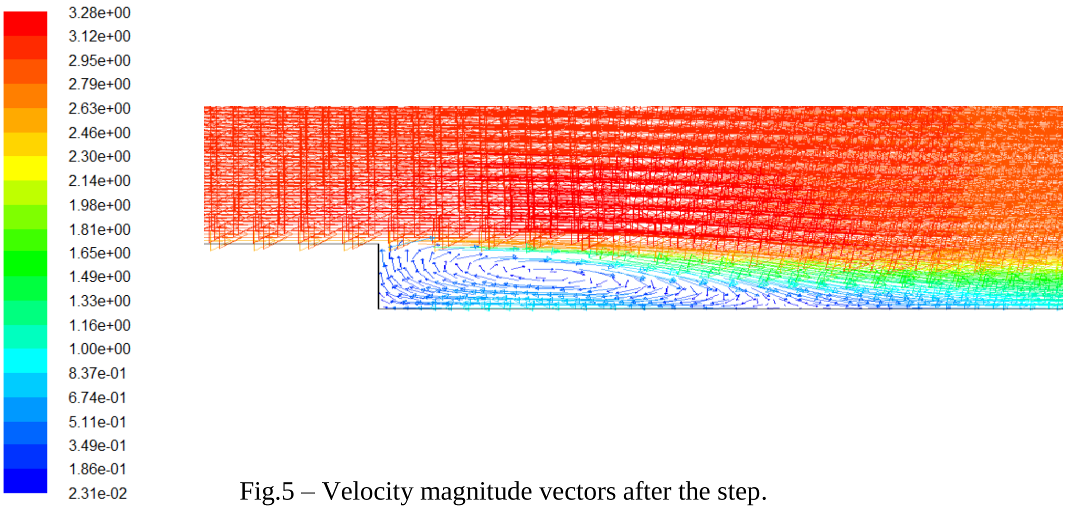

Fig.5 – Velocity magnitude vectors after the step.

The velocity profiles obtained through the three turbulent models were separated in some positions to compare with experimental data from Guerra (2000). The positions were chosen after the flow pass over the backstep as 0.3m, 0.35m, 0.40m, 0.45m, 0.50m, 0.60m, 0.80m and 0.90m. As the step has 1.0m in length, the first velocity profile was obtained at x= 1.30m. These positions are located in the redevelopment boundary layer region. Due the limitation of the instrument to measure the velocity profiles, the experimental data was obtained only after the reattachment region and the velocity numerical profiles were plotted in such positions. In Fig.6 we present the evolution of the mean velocity profiles

The position x=1.30 means one meter equivalent to the step length plus 0.30m in front of the step. This position is located at the reattachment region, which can be observed in Fig. 6a,b. The numerical results provided the behavior of the flow. In that it is possible to identify a deflection region which indicates velocities in a opposite direction. The numerical results are not in agreement with the experimental data. This is due to the fact that the experimental velocity profiles were obtained with a single hot-wire anemometry which does not have sensibility to the velocity vector in an opposite direction. Due the instability of the stagnation region where occurs the reattachment flow, the measurements were done at some distance from the reattachment region as well x=1.40m.

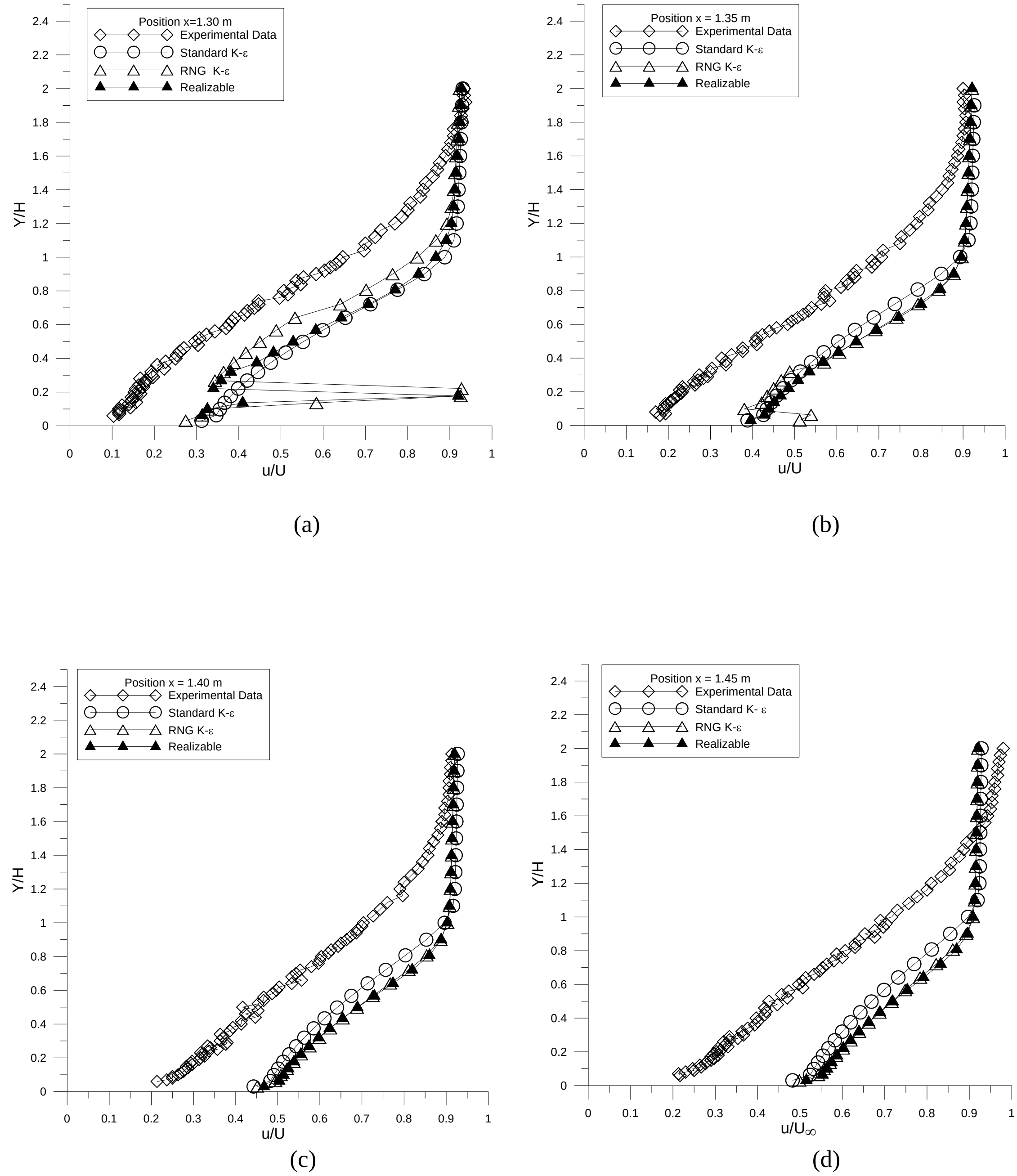

Fig. 6 – Mean velocity profiles along the wind tunnel located at position x:
(a) 1.30m, (b) 1.35m, (c) 1.40m and (d) 1.45m.

In Figures 6c-g we can observe that the numerical velocity profiles are quantitatively far away from the velocity experimental profile. The last comparison in Fig. 6h, both experimental and numerical results have a tendency to collapse.

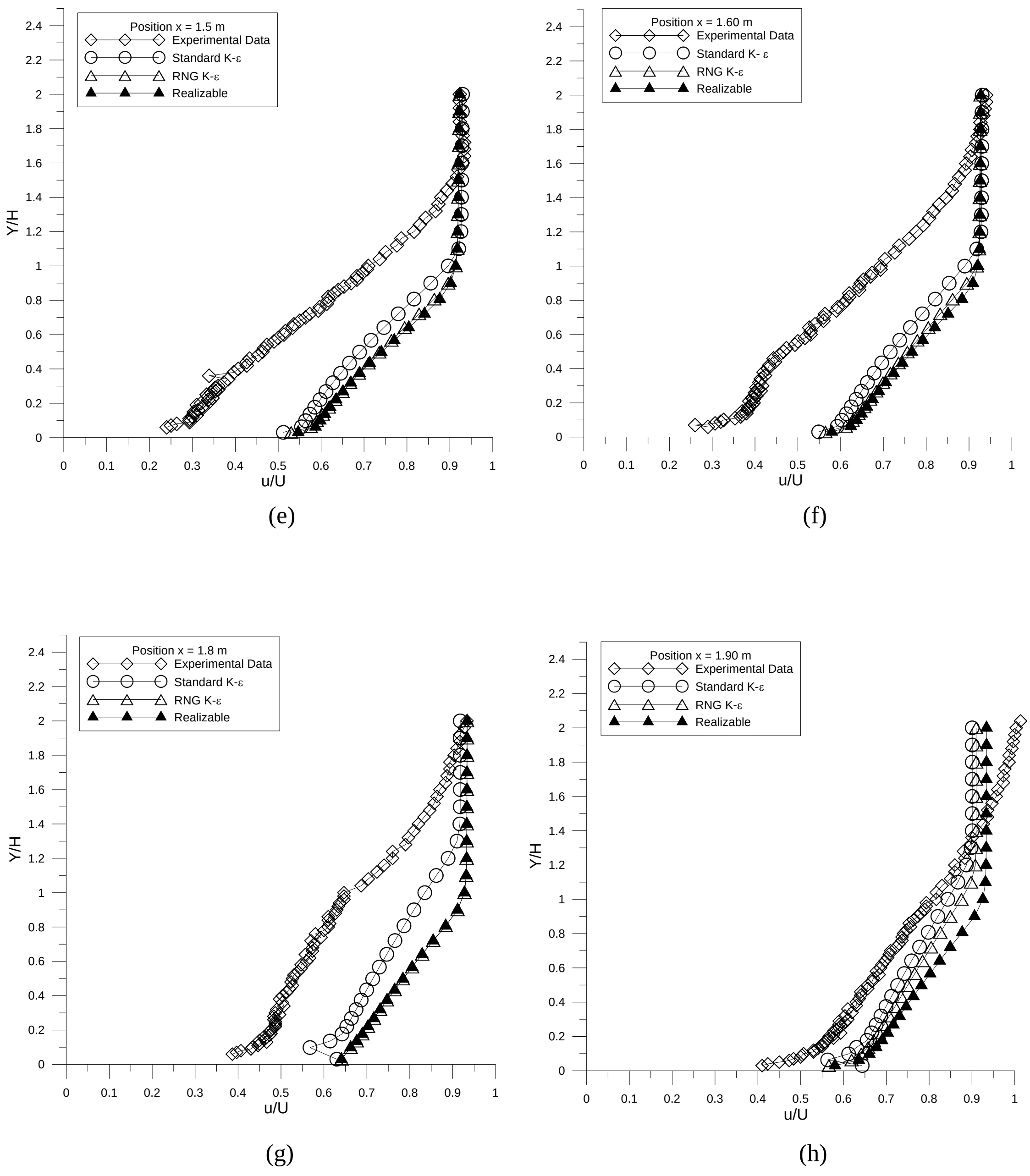

Fig. 6 – Mean velocity profiles along the wind tunnel located at position x:
(e) 1.50m, (f) 1.60m, (g) 1.80m and (h) 1.90m.

The temperature profiles were obtained at the recirculation region to compare with the experimental data. As the temperature was measure by a thermocouple and this magnitude is a scalar, there was no problem to measure in the recirculation bubble. The numerical temperature profiles were obtained at the positions x of 1.05m, 1.10m, 1.15m, 1.25, 1.27m and 1.29m. These results are shown in Figures 7a-f.

In the Fig. 7a,b profiles are not in a good quantitative agreement, however they are in a good qualitative results. In Figures 7c-f the numerical and experimental temperature profiles collapse.

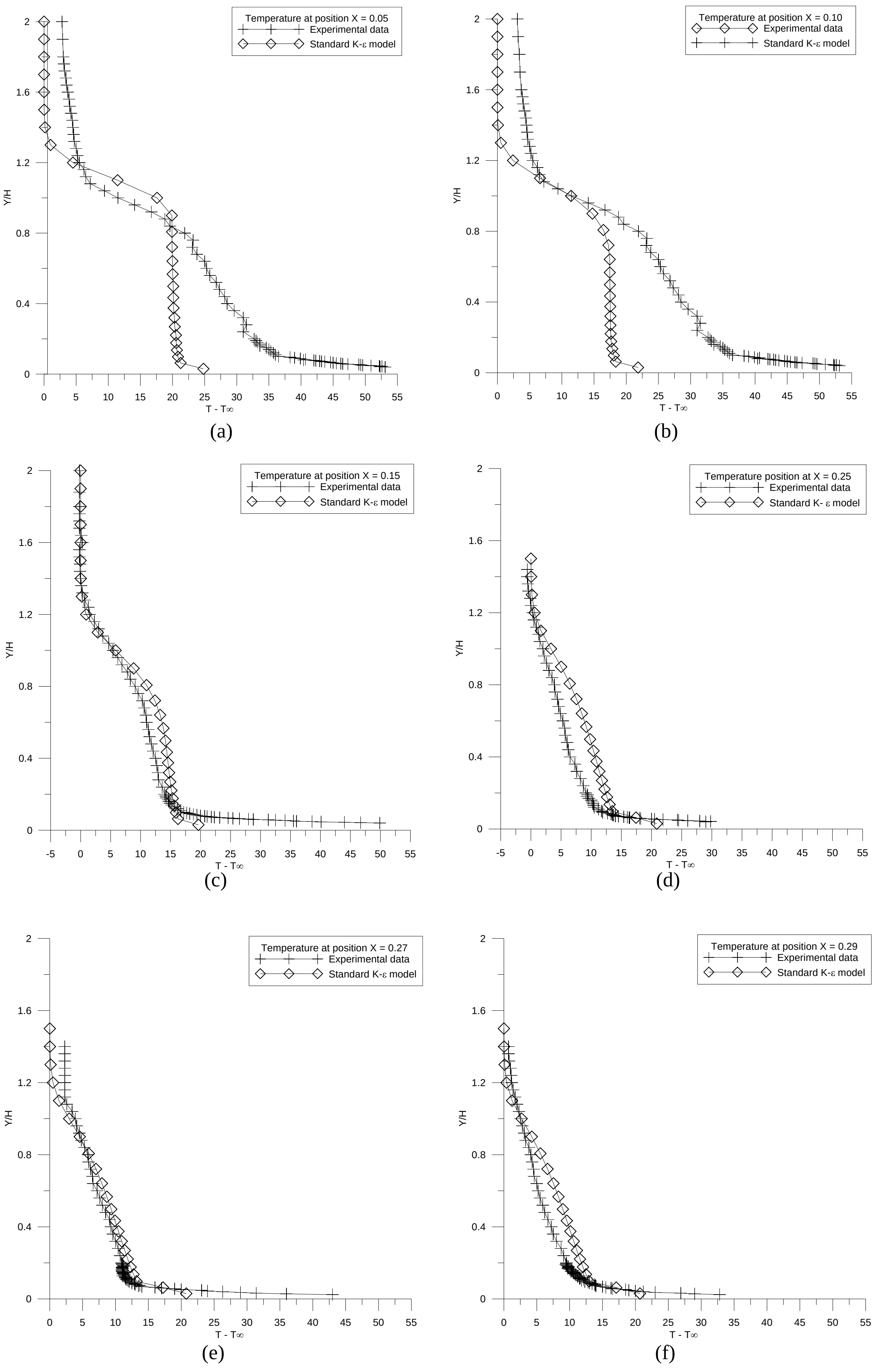

Fig. 7 – Comparative of temperature profiles with the Standard K-ε model located at position x: (a)1.05m, (b)1.10m, (c) 1.15m, (d) 1.25m, (e) 1.27m e (f) 1.29m.

5.CONCLUSIONS

A combined experimental and numerical study was undertaken to analyze a turbulent complex flow. The flow is produced by a turbulent flow over a backward facing step. Turbulent eddying zone, recirculation region, reattachment and redevelopment boundary layer have been observed. This work showed the correlation between experimental data and the data obtained in the simulation, providing a validation of simulation performed using the computer codes Gambit and Fluent 6.3.

It was possible to observe that some discordant between numerical and experimental results must be due to the wall friction effects. The experimental data was measured in a wind tunnel with wooden walls which has a roughness that was not considered on the numerical simulation.

Other observation is about the turbulence models, the three models did not show great advantages showing close results. In general we can consider that has a satisfactory agreement between the experimental results and the numerical predictions in this preliminary study.

6. REFERENCES

Abrunhosa, J.D.M., Nieckele, A.O., 2005, “Simulação do Problema do Degrau com Rans e Les”, Revista Iberoamericana de Ingeniería Mecánica. Vol. 9, N.º 3, pp. 57-66.

Abu-Nada, E. 2007, “Application of nanofluids for heat transfer enhancement of separated flows encountered in a backward facing step”,Int. J. Heat and Fluid Flow 29, 242–249.

Bouda, N. N, Schiestel, R., Amielh, M., Rey, C., Benabid, T., 2008, “Experimental approach and numerical prediction of a turbulent wall jet over a backward facing step”, International Journal of Heat and Fluid flow.

Chen, Y.T., Nie, J.H., Armaly, B.F., Hsieh, H.T., 2006, “Turbulent Separated Convection Flow Adjacent to Backward-Facing Step - Effects of Step Height”, Int. J. of Heat and Mass Transfer 49,3670–3680.

Dmitriev, S.S., 1990, “Mechanism of Turbulent Boundary Layer Separation from a Smooth Wall”, Mekhanika Zhidkosti I Gaza, Nº 6, pp 69-77.

Guerra, D.R.S, 2000, Análise Teórico-Experimental de Escoamento Turbulento com Separação e Troca de Calor. Dissertação de Mestrado, Programa de Pós-Graduação em Engenharia Mecânica, UFPA.

Komoda, H.,Iuchi, M., Tani, I., 1961, “Experimental Investigation of Flow separation Associated with a Step or a Groove”, Aeronautical Research Institute, University of Tokyo Report Nº 364, April.

Mehrez, Z., Bouterra, M., Cafsi, El A., Belghith, A., Quere, Le P., 2009, “The Influence of the Periodic Disturbance on the Local Heat Transfer in Separated and Reattached Flow”, Int. J. Heat Mass Transfer 46:107–112.

Simpson, R. L., Strickland, J.H. and Barr, P.W., 1974, “The structure of a separating turbulent boundary layer”. J. Southern Methodist Univ. Thermal and Fluid Sciences Center, WT-3, Texas, USA

Simpson, R.L., Chew, Y.T. and Shivaprasad, B.G.,1981. “The structure of a separating turbulent boundary layer – Part 1: Mean Flow and Reynolds Stresses”. Journal of Fluid Mechanics, 113, 23.

Vogel,J.C., 1984, “Heat Transfer and Fluid Mechanics Measurements in the Turbulent Reattaching Flow Behind a Backward-Facing Step”, PhD Dissertation,Dept. of Mech. Engrg. Stanford Univ.

7. RESPONSIBILITY NOTICE