International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:3, No:5, 2009

Computational Simulation of Turbulence Heat Transfer in Multiple Rectangular Ducts

Azli Abd. Razak, Yusli Yaakob, and Mohd Nazir Ramli

***Abstract*—**This study comprehensively simulate the use of k-ε model for predicting flow and heat transfer with measured flow field data in a stationary duct with elucidates on the detailed physics encountered in the fully developed flow region, and the sharp 180° bend region. Among the major flow features predicted with accuracy are flow transition at the entrance of the duct, the distribution of mean and turbulent quantities in the developing, fully developed, and sharp 180° bend, the development of secondary flows in the duct cross-section and the sharp 180° bend, and heat transfer augmentation. Turbulence intensities in the sharp 180° bend are found to reach high values and local heat transfer comparisons show that the heat transfer augmentation shifts towards the wall and along the duct. Therefore, understanding of the unsteady heat transfer in sharp 180° bends is important. The design and simulation are related to concept of fluid mechanics, heat transfer and thermodynamics. Simulation study has been conducted on the response of turbulent flow in a rectangular duct in order to evaluate the heat transfer rate along the small scale multiple rectangular duct

***Keywords*—**Heat transfer, turbulence, rectangular duct, simulation.

I. Introduction

THE performance of turbulence and heat transfer models in predicting the velocity and temperature fields of relevant industrial flows has become importance during the last few years.

This requirement for improved predictive performance is also true for turbulent duct flows which occur frequently in many industrial applications such as compact heat exchangers, gas turbine cooling systems, recuperates, cooling channels in combustion chambers, inter-coolers, nuclear reactors and others[1]. The cross-section of these ducts might be orthogonal (square or rectangular) or non-orthogonal (such as trapezoidal), in which the generated flow is extremely complex[2]. Sometimes, the ducts are can be wavy or corrugated in the streamwise direction and might be manufactured with ribs in order to achieve faster transition to turbulence.

Turbulent flows can be created by introducing geometrical elements periodically in the channel to enhance heat transfer between the fluid and the surface[3, 4]. Turbulent flow can control the temperature along the duct.

The local-averaged heat transfer coefficients along the length of the flow passage except for the regions of the bends of the passage can be nearly constant and closely coincide with the fully developed heat transfer coefficient for flow in a flat rectangular duct[5, 6].

The calculation of developing flow in 180° bend regions can be carried out with initial conditions similar to a fully developed channel and then integrating in time until the flow reached a statistically stationary state. The time evolution of bulk quantities such as Nusselt number, wall heat flux, friction drag losses, and form drag losses were observed until all values were established, sampling to obtain mean and turbulent quantities was carried out for approximately 10 time units in each calculation[7]. The corners and recesses geometrical features can cause abrupt changes in flow direction, separation and consequently large-scale vortices. The subsequent convection of vortices gives rise to significant flow unsteadiness and consequently temporal heat variations. They also claimed that due to the massive separation, the flow unsteadiness and heat transfer in a 180° sharp bend duct are much stronger than for U-bends[8].

In this study the ducts are analyzed without any ribs and have sharp 180° bend region to be considered and analyzed as shown in Fig. 1. Thus, the prediction of the turbulent flow and heat transfer characteristics in engineering duct flows still requires Reynolds number approach using suitable turbulent closure model for both the velocity and temperature correlations. It is known that secondary motions develop in the sharp corners of non-straight duct where those motions are important since they redistribute the kinetic energy, influence the streamwise velocity, and thereby affect the wall shear stress and heat transfer[9].

Azli Abd Razak is with Universiti Teknologi MARA, 40450 Shah Alam Selangor Malaysia (phone: 603-55435154; fax: 603-55435160; e-mail: azlirazak@ salam.uitm.edu.my). He is the Senior Lecturer and also Head of Programme at Faculty of Mechanical Engineering.

Yusli Yaakob, was with Universiti Teknologi MARA, Permatang Pauh Pulau Pinang, Malaysia (e-mail: yusli662@ppinang.uitm.edu.my). He is now Lecture at Faculty of Mechanical Engineering and also Head of Research Management Institute, UiTM Penang.

Mohd Nazir Ramli is with the Schlumberger Well Services, P. O. Box 2847 Al-Khobar 31952, Kingdom of Saudi Arabia (e-mail: mohdnazir@al-khobar.oilfield.slb.com). He is now with the Oil Field Department as Field Engineer.

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:3, No:5, 2009

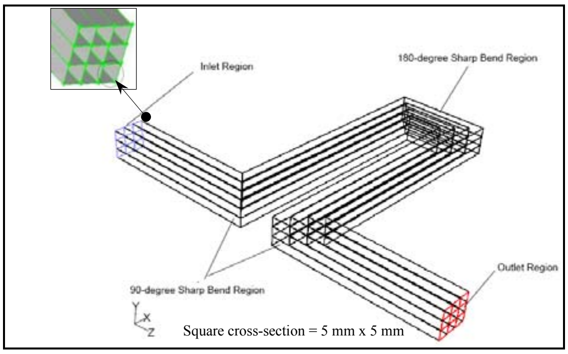

Fig. 1 Half-cut model for analysis

II. Schematic Model, Turbulent Model

A. Model Specifications

The full model of analysis was shown in Fig. 2 with the arrows of flow direction. It contains about 160 small rectangular ducts with 5 mm x 5 mm area per duct where the purpose is to allow certain amount of air per cubic area and direct it along the model. The arrangement are seems like the honeycomb typical arrangement along the full model of the 30 cm x 5 cm x 20 cm rectangular for the outer casing. It also contains baffles to elongate the duct in order to elongate the airflow distribution. For this analysis, the full model was cut into half size as shown in Fig. 1 and only nine small rectangular ducts will be analysed.

The purpose of downsize is the simulation and analysis would be much easier and practical to conduct, as it will represent the full-size model based on the similarity concept. The materials that been chosen for the ducts is high in thermal conductivity as it will absorb more heat of the airflow and those type will give different results from each other due to the properties itself.

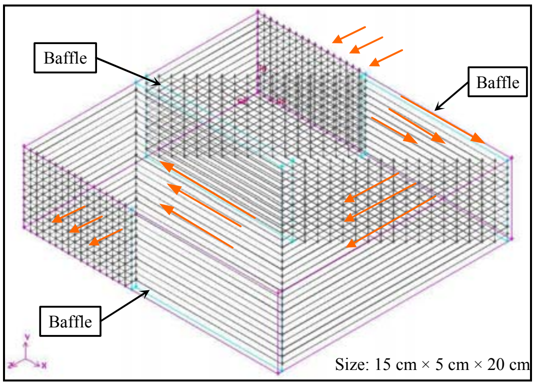

Fig. 2 Full model of multiple rectangular ducts

B. Turbulence Model

In this modeling one of the classical models are in this two-equation model *Standard k-ε* model and *Realizable k-ε* model. The realizable k-ε model has been choosing as it is consistent with the physics of turbulent flows. The transport equation of *Realizable k-ε model* is given by [10]:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial_k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_m \quad (1)$$

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial_\varepsilon}{\partial x_i}\right]+\rho C_1 S_\varepsilon$$
$$-\rho C_2\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b \quad (2)$$

where G_k represents the generation of turbulent kinetic energy due to the mean velocity gradient, G_b is the generation of turbulent kinetic energy due to buoyancy, Y_m represents the contribution of fluctuating dilatation in compressible turbulence to the overall dissipation rate.

III. Result and Discussion

A. Simulation Result

Three types of analysis of simulation results in the terms of temperature, pressure and velocity distribution were presented in the form contours and vectors plot to illustrate the flow. The results are based on the initial boundary conditions of 80 m/s of inlet velocity, 323 K of inlet temperature.

The first part will discuss about all the analysis from inlet to outlet. Another discussion part will focus on the 180° sharp bend region with same base of boundary conditions where this part could be considered as critical part because from the previous theoretical studies, this region gives the drastic impact to the results of any types of distribution analysis in the CFD simulation. The final part is the presentation of simulation in Iso-surface type where the distributions are more clearly and accurate in view.

Figs. 3 – 4 show the contour plot of static and total temperature distribution along the model where the range of the temperature value is from 323 K to 298 K along the channel. The changes of the temperature value were more effective when the flow pass through every corner or bend region of the duct. From this result, it shows that the temperature will decrease drastically when the turbulence take place. This result was supported by the results presented in Fig. 5. This figure presented that the value of the turbulent kinetic energy (m^2/s^2) increased after the flow reached first corner of the model in the range of 99.3 m^2/s^2 to 422 m^2/s^2.

The velocity magnitude is shown in Fig. 6. From this contour plot it shows the velocity in z-direction will increases drastically when the flow passing through the corner but slightly decreased in the few centimetres ahead but the decreasing does not affect much to the temperature and pressure properties. This velocity shows that the force convection will take place throughout the channel.

From this simulation result of entire region it is clearly shown that the sharp bend gives significant effect of turbulent heat transfer in order to decrease the temperature.

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:3, No:5, 2009

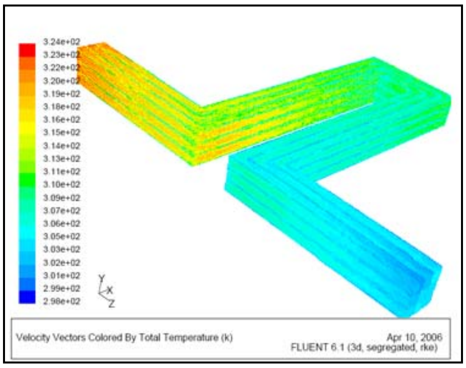

Fig. 3 Total temperature inside the dutcs

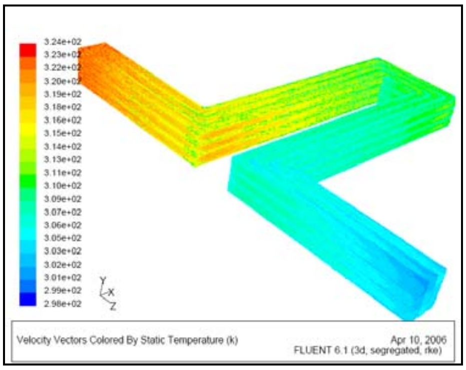

Fig. 4 Static temperature inside the ducts

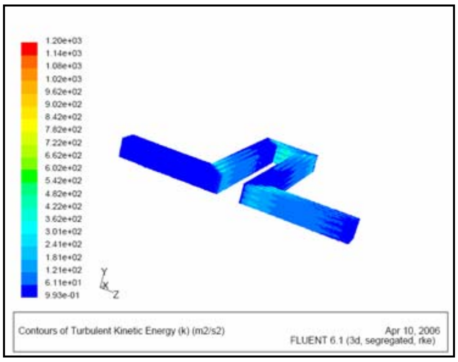

Fig. 5 Turbulent Kinetic Energy inside the dutcs

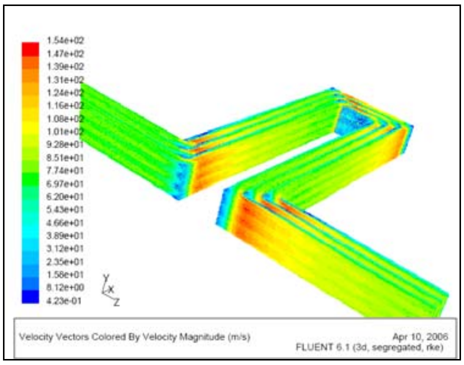

Fig. 6 Velocity Magnitude of the flow

B. The Effect of 180° Bend Region

The most critical region to analyse is 180° bend region since from the previous studies, especially for the sharp bend [8]. As we can see in the Figs. 7 – 9, and were discussed in the Fig. 3 - 6, all the value of the analysis are drastically change when the flows pass through this region in order to be directed to the outlet region.

The types of analysis were also in the same form as the first part that were discuss above which are in the form of temperature, pressure and velocity distributions. In this part of simulation, the distribution of Kinetic Energy, pressure and velocity viewed in plan. The simulation results shows that the effect of turbulent due to high velocity flow will affect the changes in temperature distribution. This shows the effectiveness of the bend region in order to create an obvious impact of turbulent flow.

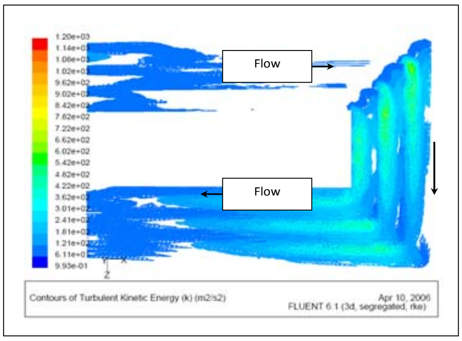

Fig. 7 Turbulent Kinetic Energy at 180° Bend

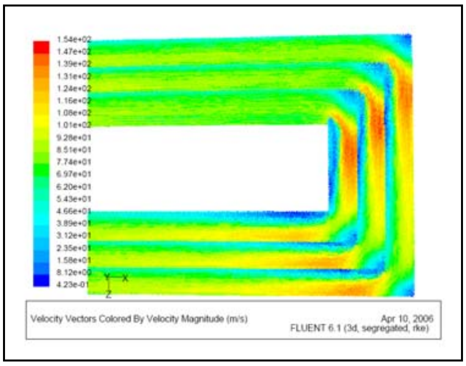

Fig. 8 Velocity Magnitude at 180° Bend

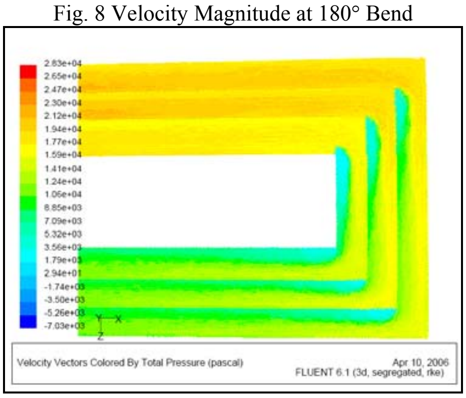

Fig. 9 Total Pressure at 180° Bend

C. Velocity and Turbulent Profile

In all analysis the flow channelling takes place near the wall and both sharp 90° and 180° bend region, due to the presence of constrained flow areas. Strong radial flow from the middle to the wall is also noticeable. It was also noticed that velocity increased even by 2 times the inlet velocity in some constrained areas of the rectangular duct as we can see in Fig. 6 and Fig. 8 in the 180° bend plan view.

Important objective of the optimum velocity inlet is to create a turbulence region with proper combination of boundary conditions variables. In the theory, velocity condition and profiles are corresponding to the turbulent flow created as can be seen in Fig. 5 and Fig. 7 where turbulent flow approach the maximum value in both 90° and 180° bend region as the value of velocity increased. In comparing the velocity and turbulent profile, then it could be summarize that the correlation between turbulent flow and velocity field exist.

In turbulent flow, the forces driving the secondary motion are concentrated in the region close to each corner. These motions were generate by gradients of the normal turbulent stresses. The turbulent shear stress is the essential element in the production of the turbulent kinetic energy and such determines the overall turbulent energy level of the flow. The secondary motions concentrated near the duct corners are strong and their effect on the streamwise flow is large.

D. Temperature Contours

The temperature contour plots for isosurface simulations using realizable k-ε model as shown in Fig. 10 and Fig. 11 was to illustrate on how temperature field changed with the velocity flow rate and the existence of high turbulent flows. This concept becomes clear when comparing the simulation results for temperature, velocity, turbulent and pressure contour and while analyzing the kinetic energy profiles (Fig. 12) that shows an increasing of kinetic energy in both 90° and 180° bend region as *Re* increases, then the correlation between heat transfer and flow field improves. The large-scale of turbulent flow in both 90° and 180° bend region the temperature field also shows oscillating behaviour and the heat transfer is decreased in those bend regions. Analysis also shows the active heat transfer region, corresponding to the maximum turbulent and velocity flow and minimum pressure is convey along the duct at a constant inlet velocity. In Fig. 11, the total temperature from the starting point of the 180° bend drops from 315 Kelvin to 307 Kelvin (about 2.5 % of temperature drops) at the end of the 180° bend respectively in condition of the increasing of velocity (about 27 % increased) and turbulence flow along that bend region. In the straight region, the correlation between heat transfer and flow field is not so good along the duct compare to the bend region (about 0.63 % of temperature drops). The power required for a given heat transfer rate can be reduced by decreasing the velocity of flow. However, a reduction in fluid velocity means that the required surface area must be increased, and hence a compromise must be made. Therefore, it can be concluded that there is strong relation of heat transfer rate between velocity and turbulent flow characteristics as the conforming to the theory and previous studies related to this topic. These results also can be use as a validation in the comparison of the theory and previous study.

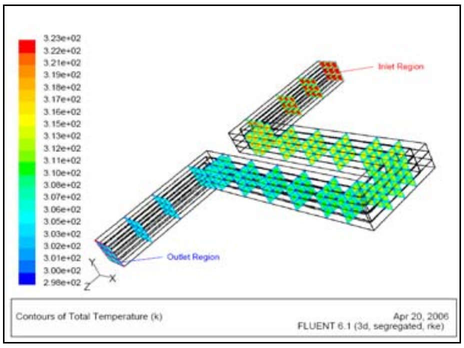

Fig. 10 Total Temperature in Whole Region

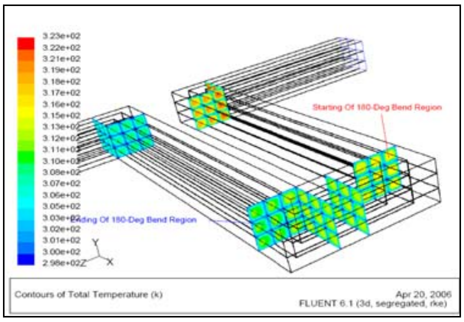

Fig. 11 Total Temperature in 90° and 180° Bend Region

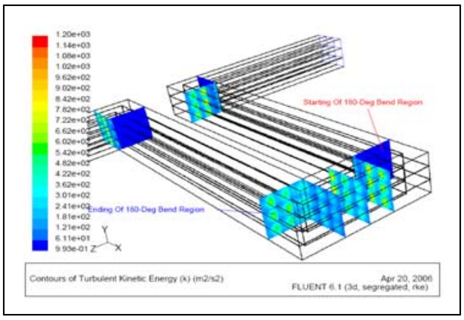

Fig. 12 Turbulent Kinetic Energy in 90° and 180° Bend Region

IV. Conclusion

From this analysis the result suggests that the flows will maintained turbulent along the ducts when the inlet velocity is at least in the range of 20 m/s – 80 m/s. A result also shows that the correlation between flow structure and heat transfer is found to be strong. It is found that the onset of flow oscillations is important as it dramatically enhances heat transfer. As mention before the temperature drop about 2.5% at the end of 180° bend while the velocity increases about 27%. Here the significant correlation between the sharp bend of the duct and velocity throughout the duct are shown.

ACKNOWLEDGMENT

Support of all technical staff at Faculty of Mechanical Engineering, Universiti Teknologi MARA is gratefully acknowledged.

REFERENCES

[1] Vazquez, M.S., W.V. Rodriguez, and R. Issa, Effect of ridged Walls on the heat transfer in a heated square duct International Journal of Heat and Mass Transfer, 2005. 48(10): p. 2050-2063.
[2] Rokni, M. and T.B. Gatski, Predicting turbulent convective heat transfer in fully developed duct flows. International Journal of Heat and Fluid Flow, 2001. 22(4): p. 381-392.
[3] Maeda, N., M. Hirato, and H. Fujita, Turbulent flow in a rectangular duct with a smooth-to-rough step change in surface roughness. International Journal of Energy, 2005. 30(2-4): p. 129-148.
[4] Yuan, J., M. Rokni, and B. Sunden, Simulation of fully developed laminar heat and mass transfer in fuel cell ducts with different cross-sections. International Journal of Heat and Mass Transfer, 2001. 44(21): p. 4047-4058.
[5] Abraham, J.P. and E.M. Sparrow, Fraction drag resulting from the simultaneous imposed motions of a freestream and its bounding surface. International Journal of Heat and Fluid Flow, 2005. 26(2): p. 289-295.
[6] Sparrow, E.M. and J.P. Abraham, Universal solutions for the streamwise variation of the temperature of a moving sheet in the presence of a moving fluid. International Journal of Heat and Mass Transfer, 2005. 48(15): p. 3047-3056.
[7] Sewall, E.A., et al., Experimental validation of large eddy simulations of flow and heat transfer in a stationary ribbed duct. International Journal of Heat and Fluid Flow, 2006. 27(2): p. 243-258.
[8] Chung, Y.M., P.G. Tucker, and D.G. Roychowdhury, Unsteady laminar flow and convective heat transfer in a sharp 180O bend. International Journal of Heat and Fluid Flow, 2003. 24(1): p. 67-76.
[9] Yuan, J., M. Rokni, and B. Sunden, Three-dimensional computational analysis of gas and heat transport phenomena in ducts relevant for anode-supported solid oxide fuel cells. International Journal of Heat and Mass Transfer, 2003. 46(5): p. 809-821.
[10] Bradshaw P., An introduction to turbulence and its measurement. Pergamon Oxford, 1971.