International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:8, No:2, 2014

Heat Transfer and Turbulent Fluid Flow over Vertical Double Forward-Facing Step

Tuqa Abdulrazzaq, Hussein Togun, M. K. A. Ariffin, S. N. Kazi, A. Badarudin, N. M. Adam, S. Masuri

***Abstract*—**Numerical study of heat transfer and fluid flow over vertical double forward facing step were presented. The k-w model with finite volume method was employed to solve continuity, momentum, and energy equations. Different step heights were adopted for range of Reynolds number varied from 10000 to 40000, and range of temperature varied from 310K to 340 K. The straight side of duct is insulated while the side of double forward facing step is heated. The result shows augmentation of heat transfer due to the recirculation region created after and before steps. Effect of step length and Reynolds number observed on increase of local Nusselt number particularly at recirculation regions. Contour of streamline velocity is plotted to show recirculation regions after and before steps. Numerical simulation in this paper done by used ANSYS FLUENT 14.

***Keywords*—**Turbulent flow, Double forward, Heat transfer, Separation flow.

I. Introduction

THE crises of energy in global has been encourage researchers to look for new techniques which improve of thermal performance. One of common method to increase efficiency of energy system equipment is by change the design geometry of channel. The fluid flow over forward or backward-facing step found in many practical applications such as heat exchangers, chemical process, turbine blades, and power plants. There are many experimental and numerical study of effect separation flow at forward or backward-facing step on increase of enhancement heat transfer rate. Heat transfer to fluid flow over single and double forward-facing step experimentally studied by Shakouchi and Kajino [1] with using laser Doppler anemometer where the results appeared that the reattachment region by a new heat flux probe, and used jet discharge by [2], [3]. Saldana and Anand [4] performed numerical study of laminar fluid flow over a horizontal forward-facing step in three dimensions. They showed that the increase of Reynolds number leads to increase of reattachment length and average Nusselt number. Moreover, reattachment region appeared after the separation region and these phenomena are addressed in numerous numerical and experimental studies [5]-[14]. In order to improve of enhancement in heat transfer Oztop et al. [15] used obstacles with horizontal double forward-facing step as the results found that the increase augmentation of heat transfer with increased of aspect ratio of obstacle, Reynolds number and step height.

The purpose of this paper is to study effect of the length step, Reynolds number, and wall temperature of vertical double forward–facing step on improvement of heat transfer. Also from the literature that study of heat transfer to fluid flow over a vertical double forward–facing step has not been performed yet.

Tuqa Abdulrazzaq is with the Department of Mechanical and Manufacturing Engineering, Universiti Putra Malaysia, 43400 Selangor, Malaysia (Corresponding author;Phone: 0060126331856; e-mail: tuka_eng@yahoo.com).
Hussein Togun is with the Department of Mechanical Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia. He is also with the Department of Mechanical Engineering, University of Thi-Qar, 64001 Nassiriya, Iraq,
M. K. A Ariffin, N. M Adam, and S. Masuri are with the Department of Mechanical and Manufacturing Engineering, Universiti Putra Malaysia, 43400 Selangor, Malaysia.
S. N. Kazi and A. Badarudin are with the Department of Mechanical Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia.

II. Geometry Description

The geometry which considered in numerical simulation presented in Fig. 1. Vertical double forward facing step with different length steps varied 10 to 20 mm for both the first and second steps are investigated with three case see Table I. The straight side of the vertical channel is unheated while forward facing step side is heated with wall temperature between 310K and 340K. The total length of vertical channel is 1600mm (L) with 50mm diameter (D) at the upstream is 800 mm but the length (a, b) after first and second step is 400 mm and 200mm respectively. The Reynolds number is calculated based on entrance diameter (D) of the channel and varied from 10000 to 40000.

TABLE I
Cases and Dimensions of Geometries

Cases	H1(mm)	H2(mm)
1	10	10
2	20	10
3	10	20

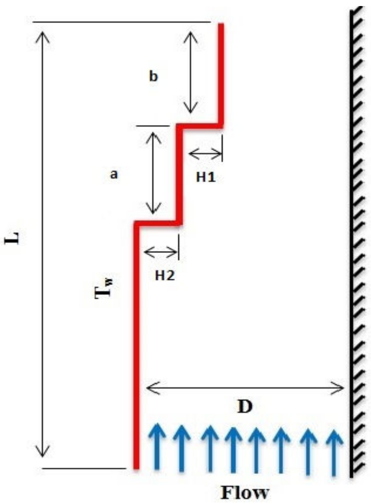


Fig. 1 Geometry model

III. Numerical Model and Data Validation

Finite Volume Method was used in computational fluid dynamics (CFD) in ANSYS FLUENT for solving continuity, momentum, and energy equations. Due to Ansys ICEM has capability tools therefore used to build the geometry and meshing process then export to FLUENT14. The assumptions as adopted were two dimensional, steady state, and turbulent flow, and the working fluid is air. Generally, the set equations which employed in simulation can be written by (1)-(4).

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \quad (2)$$

$$\frac{\partial}{\partial x_i}[u_i(\rho E + P)] = \frac{\partial}{\partial x_j}\left[\left(\lambda + \frac{c p \mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_j} + u_i(\tau_{ij})_{eff}\right] \quad (3)$$

where

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}k\delta_{ij} \quad (4)$$

The k-ω Model for shear stress transport (SST) is applied where the transport equations in this model are performed by (5), (6):

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k \quad (5)$$

$$\frac{\partial}{\partial x_i}(\rho \omega k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega \quad (6)$$

where, Gk is denoted the generation of turbulent kinetic energy by mean velocity gradients, Gω is represented the generation of ω, Yk, and Yω as denotes the dissipation of k and ω by turbulence, Dω refers the diffusion term.

The Reynolds number is calculated based on entrance diameter (D) of the channel.

$$Re = \frac{u_{av} D}{\nu} \quad (7)$$

To obtain validation that compared present results with Abu-Mulaweh [16] for velocity distribution and showed satisfy agreement as shown Fig. 2. Also three size of grid varied from 59594, 89394, 119194 elements used for more validation and then 89394 elements selected for grid independent due to that the difference results less than 1% compared to size grid 119194 elements.

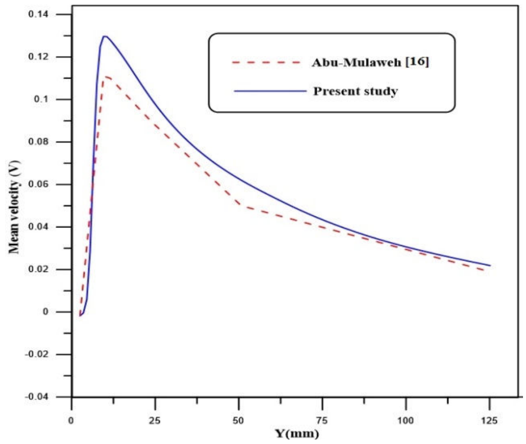


Fig. 2 Comparison of velocity profile with [16]

IV. Results and Discussion

Two dimensional of fluid flow and turbulent heat transfer over vertical double forward facing step was investigated numerically. Distribution of local Nusselt number over vertical double forward-facing step at Reynolds number 10000 with different temperature is presented in Fig. 3 Increase of local Nusselt number with increased wall temperature observed on whole side of double forward-facing step case1. Also the results found that the effect of Reynolds number on increase of local Nusselt number as shown Fig. 4 where the maximum of local Nusselt number has been seen at Reynolds number of 40000 and particularly at the first and second steps of flow passage. Comparison of local Nusselt number for three cases with different length of step and different Reynolds number plotted in Fig.5. The results found that the increase of local Nusselt number obtained after the first and second steps of vertical channel. The highest improvement of heat transfer was at case 2 compared to other cases. The contour of streamline velocity with different Reynolds number for case 1 is illustrated in Figs. 6 (A)-(D) It can be seen that the increase size of recirculation zone after and before the first and the second steps as the biggest of recirculation zone observed with Reynolds number of 40000 result to increase of static pressure. Effect of the length step on the recirculation zone after and before the first and second steps of vertical channel for three cases is clarified in Fig. 7 where the biggest size of recirculation zone observed after and before the steps for case 2 compared to case 1 and 3.

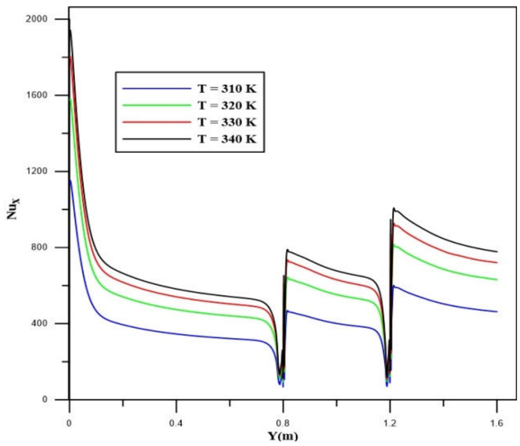


Fig. 3 Distribution local Nusselt number at Re =10000 case 1 with different temperature

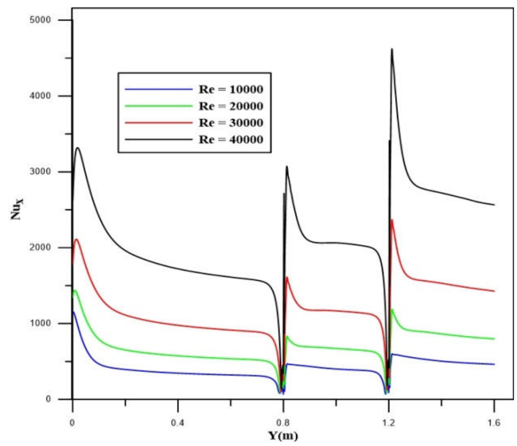


Fig. 4 Distribution local Nusselt number at T=310K case 1 with different Reynolds number

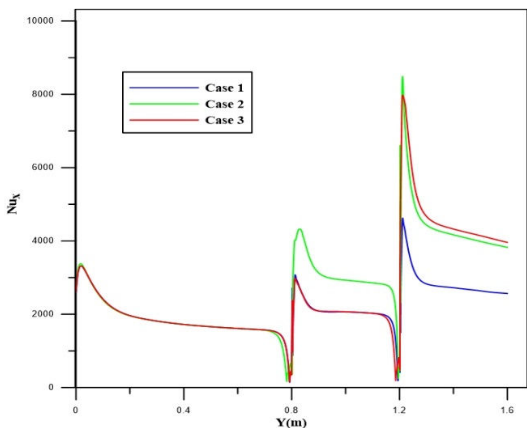


Fig. 5 Comparison of local Nusselt number for case 1, 2, and 3 with different Reynolds number

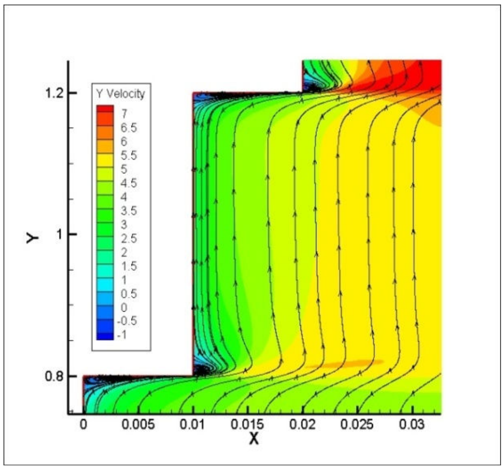


Fig. 6 (A) Contour of streamline velocity for case1 and Re=10000

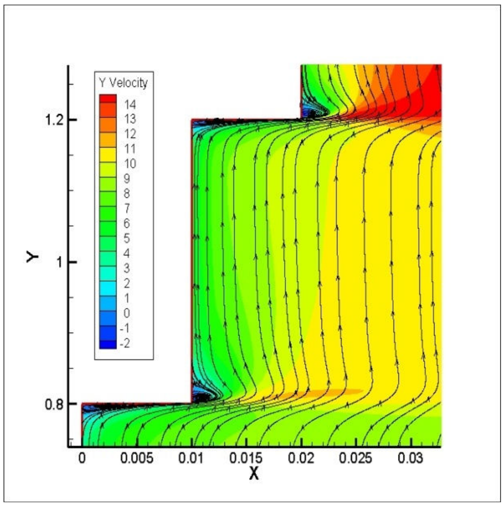


Fig. 6 (B) Contour of streamline velocity for case1 and Re=20000

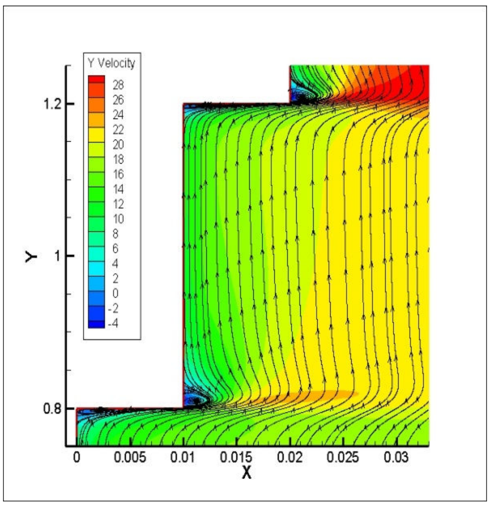


Fig. 6 (C) Contour of streamline velocity for case1 and Re=30000

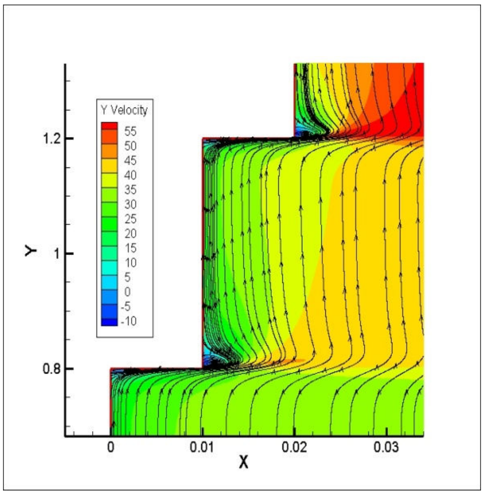
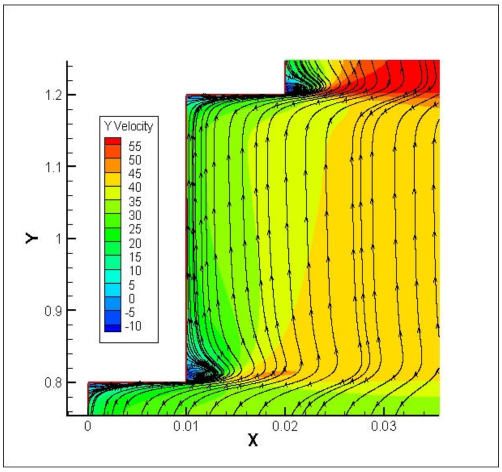
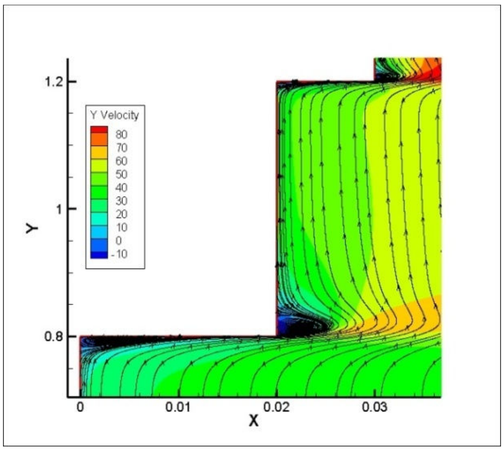


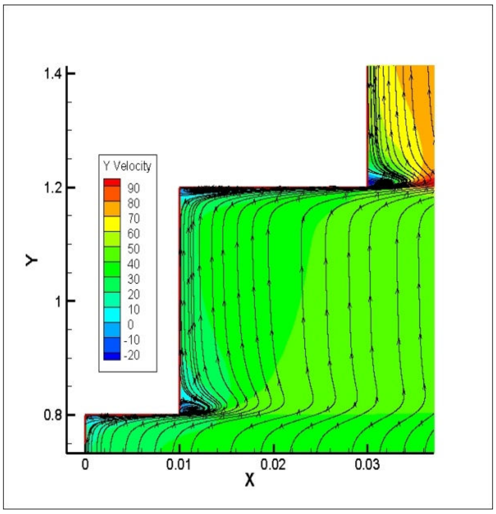
Fig. 6 (D) Contour of streamline velocity for case1 and Re=40000



Case 1



Case 2



Case 3

Fig. 7 Contour of streamline velocity at Re = 40000for case1, 2 and 3

V. Conclusion

Turbulent heat transfer to fluid flow over vertical double forward facing step was numerically studied by using k-w model with finite volume method. Three cases of different length of steps for vertical double forward facing step with Reynolds number varied from 10000 to 40000 and Temperature of wall varied from 310 to 340 K were used in this simulation. The presented results appeared that the increase of local Nusselt number with increase both Reynolds number and temperature of wall for all cases. Increase of length steps of vertical channel leads to increase size of recirculation zone which has more effect to improvement of heat transfer. Also the results revealed that the enhancement of heat transfer noticed with case 2 higher more than case 1 and 3.

Acknowledgment

The authors gratefully acknowledge high-impact research Grant UM.C/HIR/ MOHE/ENG/45, UMRG grant RP012D-13AET, IPPP/PV113/2011A and the University of Malaya, Malaysia for support in conducting this research.

References

[1] T. Shakouchi, I. Kajino, "Flow and forced-convection heat transfer over forward-facing double steps (effects of step ratio)," Journal Name: Heat Transfer - Japanese Research; (United States) vol. 22, no. 7, pp. 716-730, 1994.

[2] K. Oyakawa, T. Taira, E. Yamazato, "Studies of heat transfer control by jet discharge at reattachment region downstream of a backward facing step," Trans JSME. Vol. B60, pp. 248-254, 1994.

[3] K. Oyakawa, T. Saitoh, I. Teruya, I. Mabuchi, "Heat transfer enhancement using slat at reattachment region downstream of backward facing step. Trans JSME. Vol. B61, pp.4426-4431, 1995.

[4] J. G. Barbosa-Saldana, N. K.Anand, "Flow over a three-dimensional horizontal forward-facing step," Numerical Heat Transfer, Part A vol. 53, pp.1-17, 2008.

[5] H. Togun, Hakim, S. Sultan, IrfanAnjum, S.N. Kazi"An experimental study of turbulent heat transfer separation external in an annular passage" International Conference on Applications and Design in

Mechanical Engineering (ICADME 2009), Universiti Malaysia Perlis (UniMAP).
[6] H. Togun, S.N. Kazi, A. Badarudin, "A Review of Experimental Study of Turbulent Heat Transfer in Separated Flow", Australian Journal of Basic and Applied Sciences, Vol.5,pp. 489-505, 2011.
[7] H. Togun, Y.K. Salman, H.S. Sultan Aljibori and S.N. Kazi"An experimental study of heat transfer to turbulent separation fluid flow in an annular passage"International Journal of Heat and Mass Transfer, vol. 54(4),pp. 766-77, 2011.
[8] C.S. Oon, H. Togun, S.N. Kazi, A. Badarudin, M.N.M. Zubir and E. Sadeghinezhad " Numerical simulation of heat transfer to separation air flow in an annular pipe" International Communications in Heat and Mass Transfer, vol. 39(8), pp. 1176-1180, 2012.
[9] C. S. Oon Hussein Togun, S.N. Kazi,A.Badarudin, M.N.M. Zubir,"A computational simulation of heat transfer to separation air flow in a concentric pipe"International Conference on Applications and Design in Mechanical Engineering 2012 (ICADME 2012),Universiti Malaysia Perlis (UniMAP).
[10] C.S. Oon, H. Togun, S.N. Kazi, A. Badarudin and E. Sadeghinezhad, "Computational simulation of heat transfer to separation fluid flow in an annular passage" International Communications in Heat and Mass Transfer, vol. 46, pp. 92-96, 2013.
[11] H. Togun, A.J. Shkarah, S.N. Kazi, A. Badarudin"CFD Simulation of Heat Transfer and Turbulent Fluid Flow over a Double Forward-Facing Step" Mathematical Problems in Engineering, vol. 2013, pp. 1-10, 2013.
[12] H. Togun, S.N. Kazi, Abdul Amir H. Kadhum, A.Badarudin, M.K.A. Ariffin, EmadSadeghinezhad"Numerical Study of Turbulent Heat Transfer in Separated Flow: Review" International Review of Mechanical Engineering (IREME), vol.7, pp. 337-349, 2013.
[13] H. Togun, T. Abdulrazzaq, S.N. Kazi, A. Badarudin and M.K.A. Ariffin"Heat Transfer to Laminar Flow over a Double Backward-Facing Step" World Academy of Science, Engineering and Technology, International Journal of Mechanical Science and Engineering vol. 7 (8), pp. 961-966, 2013.
[14] H. Togun, T. Abdulrazzaq, S. N. Kazi, A. Badarudin, M.K.A Ariffin"Numerical study of turbulent heat transfer in annular pipe with sudden contraction" Applied mechanics and materials, vol. 465-466, pp. 461-466, 2014.
[15] H. F.Oztop, K. S. Mushatet, I. Yılmaz, "Analysis of turbulent flow and heat transfer over a double forward facing step with obstacles" International Communications in Heat and Mass Transfer vol. 39, pp.1395-1403, 2012.
[16] H.I. Abu-Mulaweh"Turbulent mixed convection flow over a forward facing step - the effect of step heights" International Journal of Thermal Sciences vol. 44, pp. 155-162, 2005.