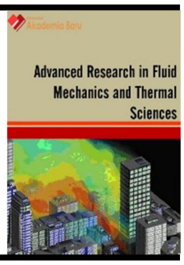
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html

ISSN: 2289-7879



Numerical Investigation of Forced Convection Flow over Backward Facing Step Affected By A Baffle Position

Open Access

Aadel Abdul Razzaq Alkumait[1], Maki Haj Zaidan[1], Thamir Khalil Ibrahim[2,*]

[1] Department. of Mech. Eng., College of Eng., Tikrit University, Tikrit, Iraq
[2] Department. of Mech. Eng., College of Applied Eng., Tikrit University, Tikrit, Iraq

ARTICLE INFO

Article history:
Received 3 July 2018
Received in revised form 19 October 2018
Accepted 21 October 2018
Available online 3 December 2018

Keywords:
Backward facing step, Baffle, Heat transfer, Laminar regime

ABSTRACT

Backward-Facing Step (BFS) flow is one representative model for separation flows, which can be widely seen in aerodynamic flows (airfoil, spoiler, high attack angle process), engine flows, condensers, vehicles (cars, boat), heat transfer systems, and even the flow around buildings, etc. Aiming to enhance the heat transfer characteristics of backward-facing step flow in a channel, three dimensional study of backward-facing step with different baffle's positions under Laminar forced convection flow has been numerically investigated. The range of Reynolds number (from 50 to 400) with water as working fluid have been considered in this study. The governing equations represented by momentum, continuity, and energy equations were solved by employing SIMPLE algorithm with Finite Volume Method to link the velocity and pressure fields. The results revealed that the highest Nusselt number grows near the wall, then transfers further downstream as the position of the baffle moves in the direction of the streamwise. The skin friction increases as the distance between the baffle and the inlet section decreases. Additionally, the maximum Nusselt number and skin friction assigned at (d=40 mm distance of the baffle) meanwhile the minimum Nusselt number and skin friction assigned at channel without baffle. Comparing the results of the cases with or without the installation of baffle, the maximum augmentation on average Nusselt number is about 213% at (d=40 mm distance of the baffle).

1. Introduction

The separation and reattachment considered one of the most effective methods to improve the thermo-physical phenomena, such as heat transfer using backward-facing step, are employed commonly in various engineering applications such as electrical machines [1-5]. Therefore, the flow over backward facing step technique for heat transfer convection considered interest topics in several investigations. The backward facing step technique is employed widely in much industrial equipment like engines of gas turbine, aircraft and combustors [6-11]. It is noticeable that the flow

* *Corresponding author.*
E-mail address: thamirmathcad@yahoo.com (Thamir Khalil Ibrahim)

separation and reattachment can change the structure of the flow and effect on the mechanism of the heat transfer meanwhile effect on the equipment thermal performance [12-17].

In the past decades, several investigations on internal separation and reattachment flow have been implemented experimentally and numerically [15, 18-22]. These studies support the benefit of using backward facing step technique in different directions and covering several boundary conditions with various baffles, ribs, and grooves [23-32].

The heat transfer by forced convective with constant wall heat flux, over a backward facing step installation of baffle on the upper wall leads to enhance the heat transfer. Comparing the slotted and solid baffles showed that solid baffle produces higher pressure drop and decrease in average Nusselt number as compared to slotted baffle [33] . In 3D numerical investigation, indicated that highest Nusselt number grows near the side walls [34]. Cheng and Tsay [33] studied numerically the effect of the inclined baffle by comparing the result with the vertically orientated baffle under forced convection heat transfer condition. Moreover, Heshmati *et al.*, [35] studied the effect of inclined baffle under mixed conviction. The results did not show any significant change in flow structure but the reduction in the recirculation zone size. Meanwhile, state that the highest Nusselt number assigned to the inclined baffle.

Selimefendigil and Öztop [36] studied the influence of baffle height, and compared the pulsating flow condition with the steady-state flow condition at constant wall temperature. The results revealed that with increase in fin height and Reynolds number the Nusselt number increases, but the addition of a baffle is not beneficial for heat transfer in unsteady flow.

Very less attention is given to the effect of baffle in backward facing step, with different distance between baffle and backward facing step when bottom wall is maintained at constant heat flux. Therefore, the primary goal in the current investigation is to study the enhancement of heat transfer and flow behaviour over backward facing step with installed baffle in the upper wall of the channel. The objective of present study is to provide the answer to these questions and special attention is given to control the fluid flow and heat transfer behavior with the help of baffle location so that it can be used in much more effective ways.

2. Numerical Implementation

Three dimensional simulation of water with laminar flow under forced convection heat transfer over backward-facing step in a rectangular channel has been investigated for four geometries with and without a fixed baffle on the channel wall in upper side, and the schematic diagram of the computational domain is illustrated in Figure 1. In the studied geometrical model, the channel has upstream height (h= 10 mm), downstream height (H= 20 mm) and the width (W= 80 mm), while, the backward-facing step height (S=10 mm) with an aspect ratio (AR = W/S = 8) and an expansion ratio (ER= H/(H − S)= 2). The values of these dimensional parameters have been selected and motivated based on the fact of the laser-Doppler measurement are obtainable for this model and can be utilized for the validation of the flow simulation [18]. The technique of flow over backward-facing step with installed baffle on the channel wall in the upper side with varied distance between the backward-facing step and the baffle of (d_1 = 40, d_2=70 and d_3=100 mm) respectively as shown in Figure 2. Moreover, the values of (D = d/H) which represent the normalized distance have been adopted to be 1, 2.5 and 4, respectively, meanwhile the channel without baffle represented by the normalized distance of (D = ∞). The selection of the computational domain length and the downstream of the sudden expansion as 20 mm and 500 mm respectively have been adopted to confirm that the sudden expansion at the step in the channel dose not effect on the inlet section flow, and the flow is fully developed at the outlet section. However, the increase of computational domain length was confirmed did not effect on the behavior flow domain in the backward-facing step region. Furthermore, Table 1 illustrates the characteristic dimensions of the studied geometry and the boundary conditions.

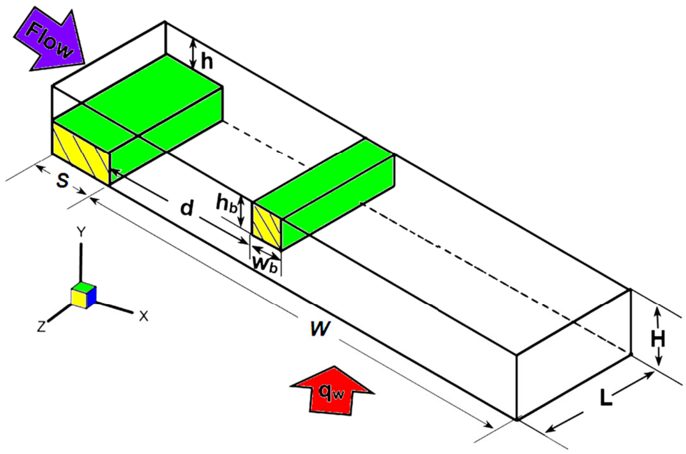


Fig. 1. Geometry description

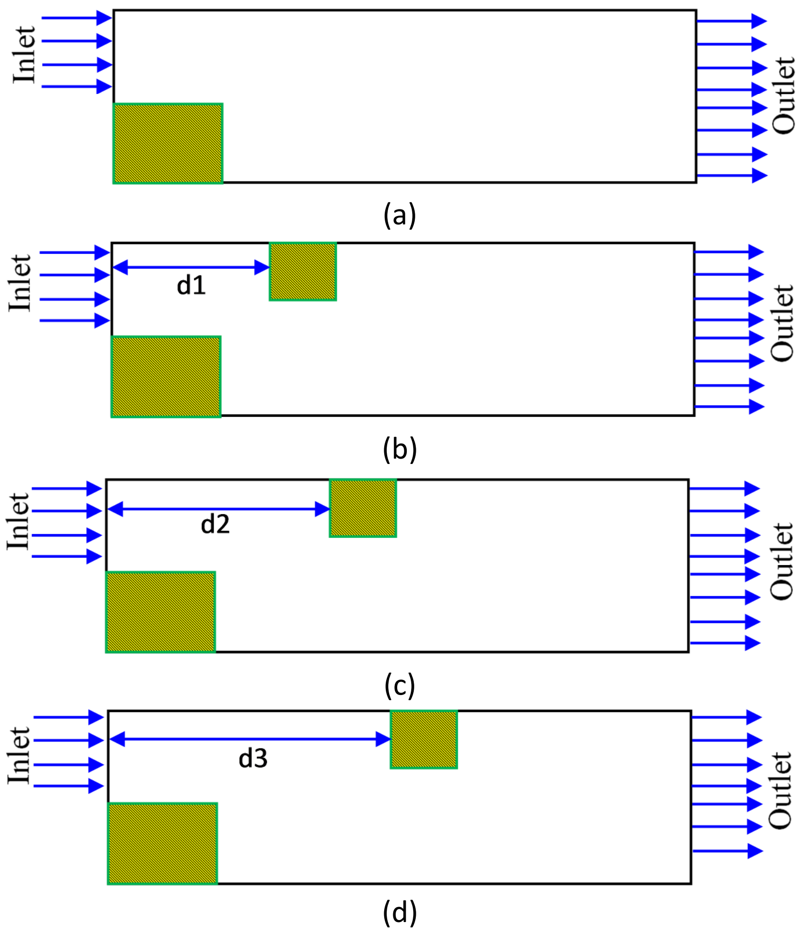


Fig. 2. Schematic diagram of the computational domain

In order to predict the numerical results, a 3D simulation of the flow and heat transfer of water in coper channel is carried out for the considered cases using ANSYS FLUENT 15 CFD code [37]. The numerical simulation of the thermal and the flow field has been implemented using finite volume method in order to solve the steady laminar three-dimensional Navier–Stokes and continuity equation as well as the energy equations [34].

Continuity equation

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0 \quad (1)$$

Momentum equation

$$\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho uv)+\frac{\partial}{\partial z}(\rho uw)=-\frac{\partial P}{\partial x}+\mu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}) \quad (2)$$

$$\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)+\frac{\partial}{\partial z}(\rho vw)=-\frac{\partial P}{\partial y}+\mu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}) \quad (3)$$

$$\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial y}(\rho vw)+\frac{\partial}{\partial z}(\rho ww)=-\frac{\partial P}{\partial z}+\mu(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}) \quad (4)$$

Energy equation

$$\frac{\partial}{\partial x}\left(\rho C_p uT\right)+\frac{\partial}{\partial y}\left(\rho C_p vT\right)+\frac{\partial}{\partial z}\left(\rho C_p wT\right)=k(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}) \quad (5)$$

where the pressure is represented by (p), temperature (T), and (u, v, w) are the velocity components of (x, y, and z) coordinate directions respectively. The thermophysical properties of water are employed in the simulation and estimated at 27 ^{o}C inlet temperatures.

In the numerical results, the average Nusselt number is defined as follow.

$$Nu=\frac{hD}{k} \quad (6)$$

The friction factor for tube with or without twisted tape can be calculated using pressure loss, Δp, across the test length, *L*, using the following equations.

$$f=\frac{1}{2}\frac{\Delta p}{L}\frac{D}{\rho\bar{V}^2} \quad (7)$$

Table 1
Characteristic dimensions and boundary condition

Parameters	Details	Parameters	Details
H	20mm	d1	40mm
H	10mm	d2	70mm
h_b	10mm	d3	100mm
w_b	10mm	Reynolds Number	50-400
W	180mm	T_{in}	27^{o}C
L	40mm	Constant Heat flux	10,000 w/m^2
S	20mm	Working fluid	Water
Hydraulic diameter	16mm		

In the computational domain, the inlet boundary conditions are assumed with constant temperature T_{in} and constant flow velocity V_{in}. Furthermore, the solver with double precision is adopted for accurate fining during the simulation. Moreover, in order to discretize the pressure term the standard scheme is used. Meanwhile, to discretize the energy and momentum terms the second-order upwind scheme is utilized and the SIMPLE algorithm is employed in order to switch the link between the pressure and velocity field. In addition, some assumptions are considered to simulate this problem such as; the effect of gravity is negligible, incompressible flow and Newtonian fluid with constant physical properties.

3. Grid Independence Study

In order to guarantee the most suitable computational mesh grid and evaluate the effect of using different mesh size on the results a grid independence study is carried out. In this study, four different computational mesh grids are adopted, which are 50x60x150, 50x50x200, 50x60x200 and 50x60x300 at Re= 200. Furthermore, the results of the local Nusselt number were similar for all grid densities with maximum deviation of 0.75% as it is clearly shown in Table2 and Figure 3. To obtain the accurate result with consider the consuming time a domain grid of 50x60x200 was adopted. Furthermore, the minimum convergence criterion for the velocity and continuity equation is 10^{-6} and for energy equation is 10^{-8}.

Table 2
Grid independence test results

Nods	Average Nusselt number, Nu	Percentage Error (%)
50x60x150	4.59	3.64
50x50x200	4.63	2.16
50x60x200	4.72	0.81
50x60x300	4.03	12.4

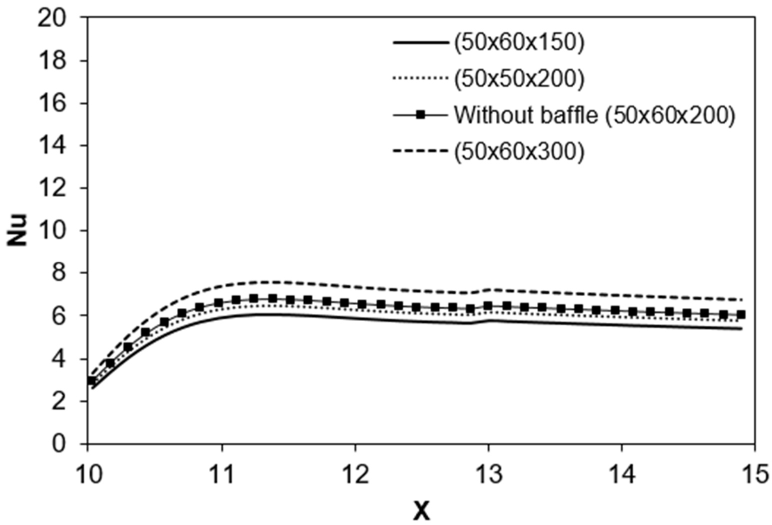
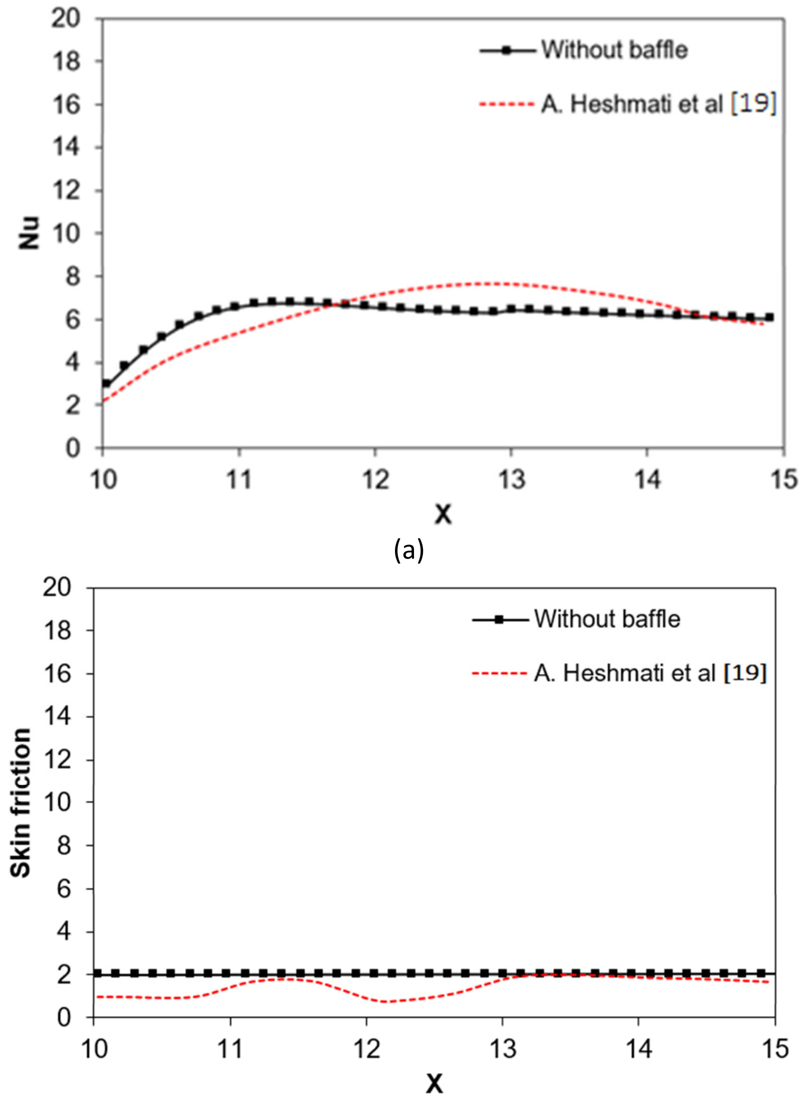


Fig. 3. Grid independence test

4. Results and Discussion

It is important to approve the results of the friction factor and Nusselt number for working fluids with trusted references and experimental results. The validation for the simulation code was achieved based on geometry and boundary conditions reported by Heshmati *et al.*, [19]. The study concentrate on numerical determination of mixed convection flow over backward facing step in channel with rectangular cross section area to study influence of the slotted baffle on heat transfer and fluid flow. The comparison for the Nusselt number and skin friction demonstrate a good agreement as shown in Figure 4.



(b)

Fig. 4. Code validation (a) Nusselt number, (b) Skin friction at Re=100

Figure 5 clarify the distribution of local Nusselt number along the heated channel wall. Furthermore, there is an increase of Nusselt number at the baffle position for all the geometries. It can be observe that the plot trend effected by the baffle position and gets highest Nusselt number at the location of the baffle due to the initiated secondary flow near the baffle region. After that, there was a dramatic decline for all cases with baffle, The Nusselt number is then increased gradually to reach maximum rate close to the baffle wall. This is due to the presence of the upturned flow coming from the swirling region and contacted the heated downstream wall which formulae a swirling flow. It's observed that the channel without baffle provides the lowest Nusselt number in comparison with other geometry with baffle.

According to Figure 6 the average Nusslet number increase with increasing Reynolds number, and the values of Nusslet number effected significantly by the baffle distance, where the highest Nusselt number assigned to the baffle with (d=40). Moreover, the increase in Reynolds number lead to increase the flow interruption and expand the fluctuation size and circulation region around the baffle.

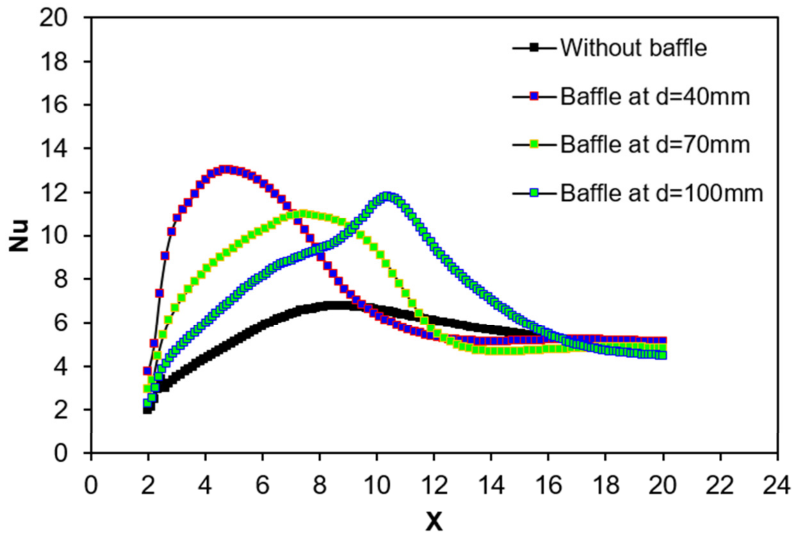


Fig. 5. Local Nusselt number of the channel with and without baffle at Re=200

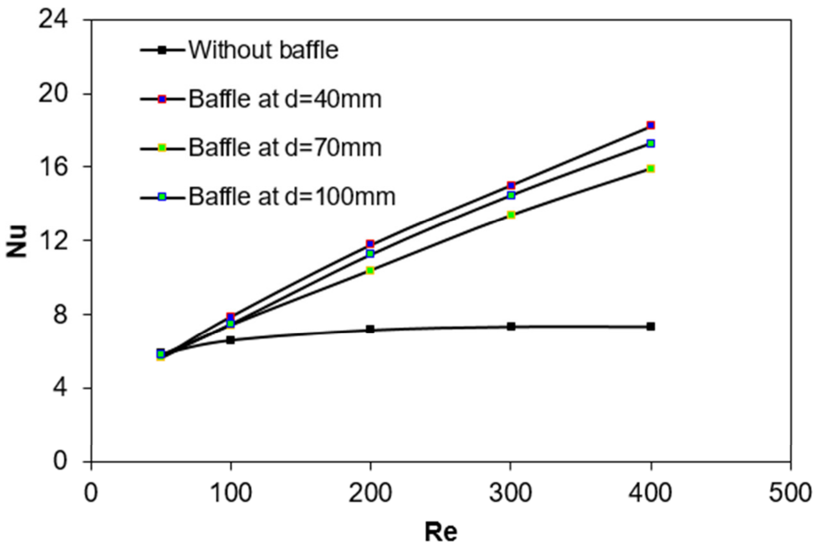


Fig. 6. Average Nusselt number of the channel with and without baffle

Figure 7 shows the skin friction along the channel. It is clearly observed that, the use of the baffle caused a significant skin friction around the baffle region. Nevertheless, the channel without baffle provides a minimum skin friction compared with the geometry with baffle. It can be attributed to the effect of the initiated main and secondary flow around the baffle wall. However, it can be concluded that lowest value is assigned to geometry without baffle meanwhile, the highest skin friction coefficient is assigned to the baffle with (d=40 mm).

In this investigation four different geometries such as without baffle, with baffle of (d =40, 70 and 100), are numerically examined. The usage of the baffle on the upper wall side produces secondary flow over the baffle and this lead to enhance the convective heat transfer near the wall. The influence of the baffle distance from the backward facing step on the stream functions are obviously illustrated in Figure 8. Moreover, main and secondary flows are accruing in the channel as a result of baffle effect. Furthermore, It is can be seen that by using baffle with (d=40) provide much uniform heat transfer compared with other geometries.

The streamlines in the laminar flow regime for different distance at (Re=200) are presented in Figure 9. However, the size of the swirling region increases with using baffle and the flow at the step edge separates and a swirling region is detected around the baffle. It is seen from the plots of the streamline that the swirling region increases with the decrease of the distance (d). which can be enhance the heat transfer rate because of growth of the recirculation [38].

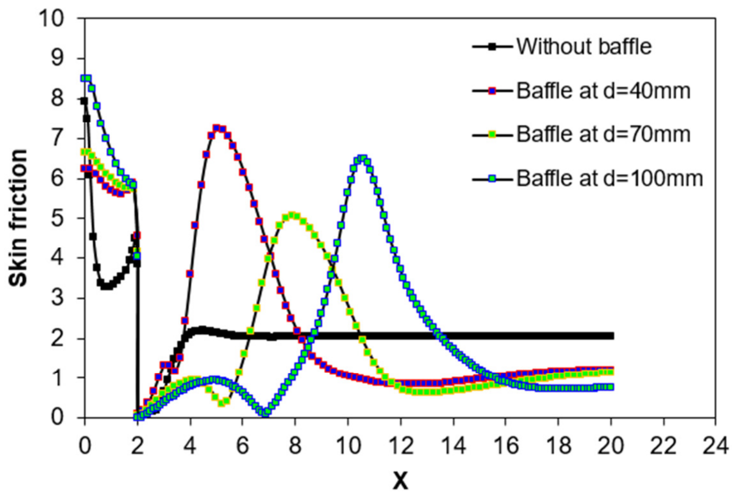
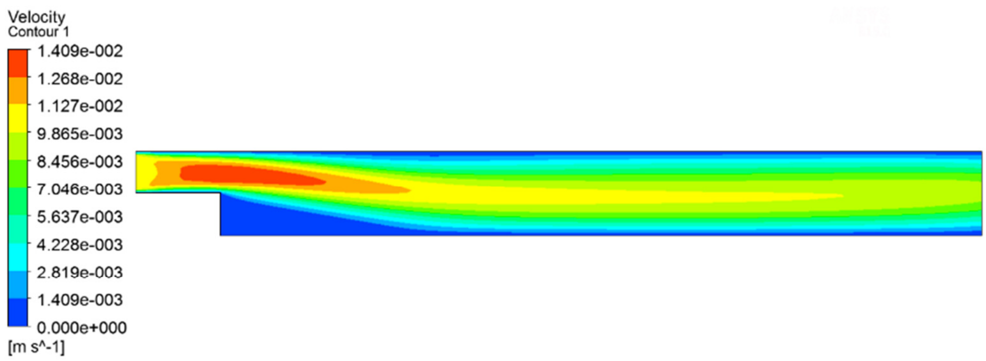
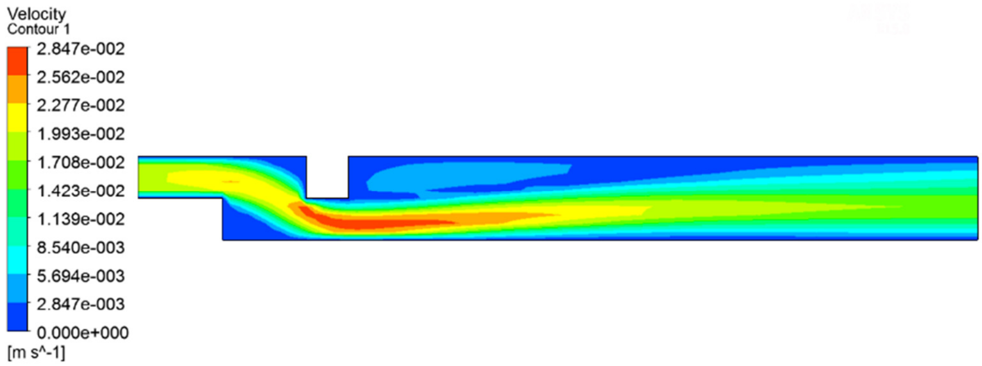


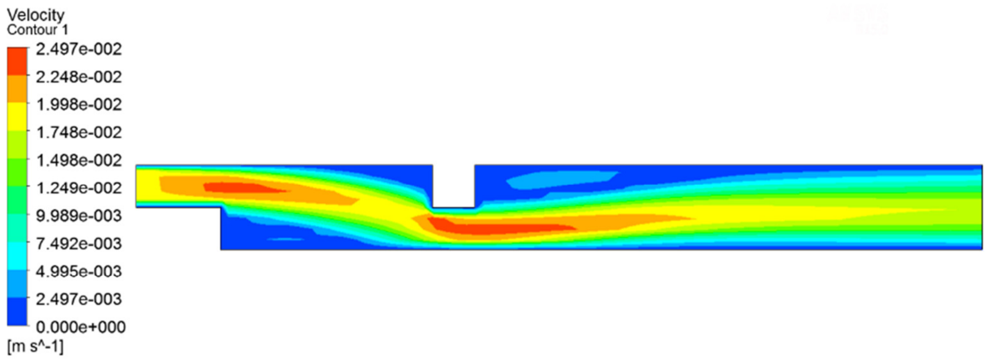
Fig. 7. Local skin friction of the channel with and without baffle at Re=200



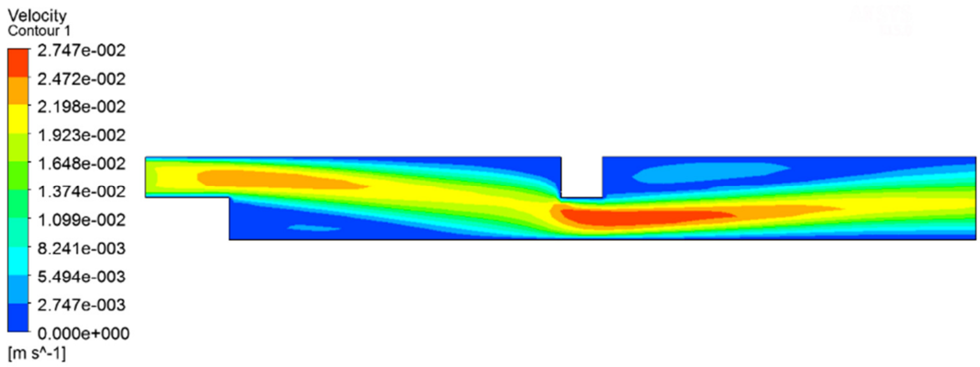
(a) d= 0 mm



(b) d=40 mm

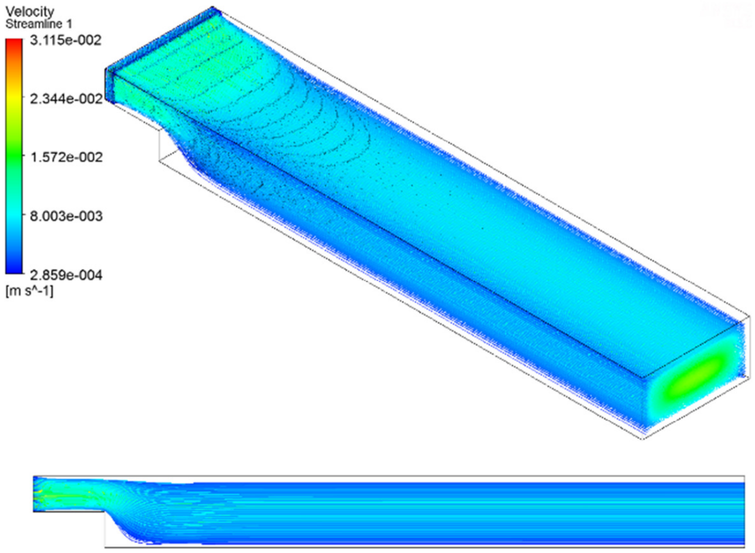


(c) d=70 mm

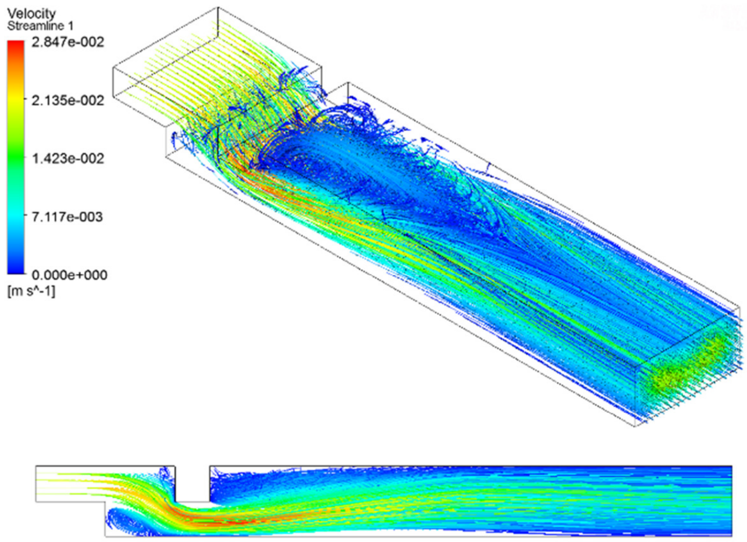


(d) d=100 mm

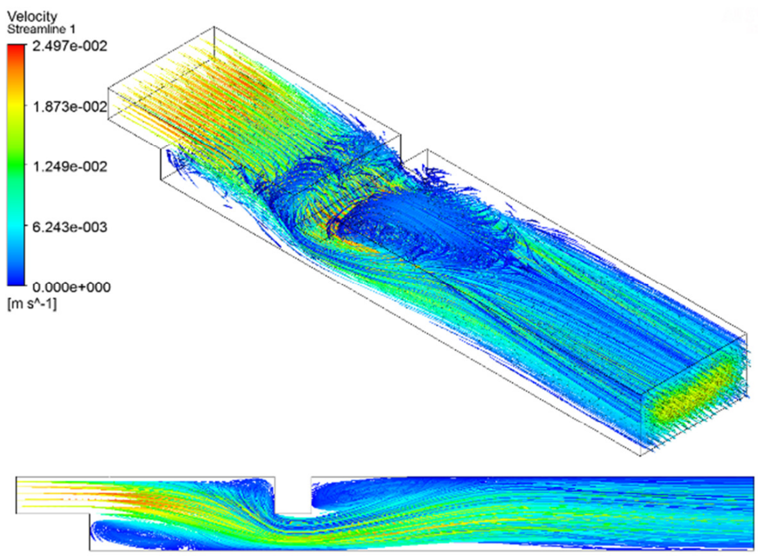
Fig. 8. Velocity contour for all studied geometry



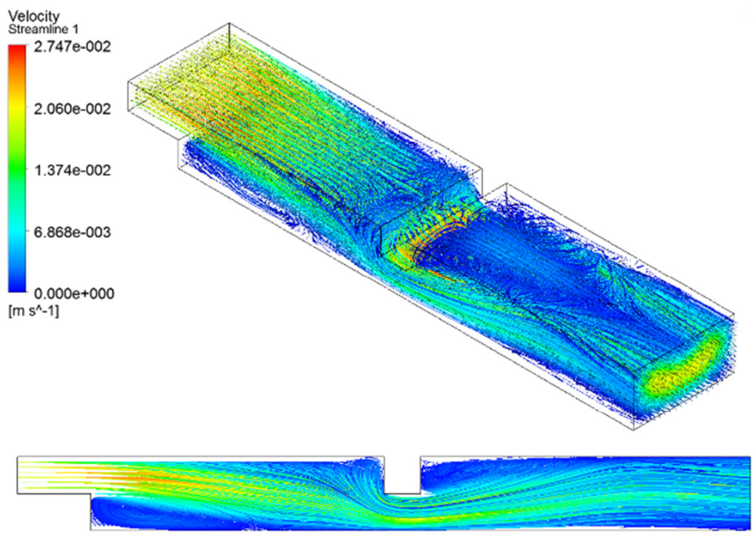
(a) d= 0 mm



(b) d= 40 mm



(c) d= 70 mm



(d) d= 100 mm

Fig. 9. Streamlines contour for all studied geometry

5. Conclusions

The numerical simulation outcome of 3D laminar forced convection flow over backward facing step in channel has been adopted to study influence of a baffle position n the heat transfer and fluid flow in this investigation. The range of Reynolds number (from 50 to 400) has been considered in this study. The reason behind instillation of a baffle onto the upper wall leads to improve heat transfer and provide high Nusselt number. The results revealed that the maximum Nusselt number assigned at channel with baffle of (d= 40 mm) followed by (d=100) and (d=70 mm) meanwhile, the minimum Nusselt number assigned at channel without baffle. The skin friction affected by the baffle instillation and position, where the highest skin friction value assigned at channel with baffle of (d= 40 mm) followed by (d=100) and (d=70 mm) meanwhile, the minimum skin friction assigned at channel without baffle.

Acknowledgement
The authors are grateful to Tikrit University for providing financial support to complete this project.

References
[1] Al-Sammarraie, Ahmed T., and Kambiz Vafai. "Heat transfer augmentation through convergence angles in a pipe." *Numerical Heat Transfer, Part A: Applications* 72, no. 3 (2017): 197-214.
[2] Al-Sammarraie, Ahmed Tawfeeq Ahmed, Raaid Rashad Jassem, and Thamir K. Ibrahim. "Mixed Convection Heat Transfer in Inclined Tubes with Constant Heat Flux." *European Journal of Scientific Research* 97, no. 1 (2013): 144-158.
[3] Al-Sammarraie, Ahmed Tawfeeq Ahmed, and Manar Salih Mahdi. "Theoretical Study of Heat Transfer through a Sun Space Filled with a Porous Medium." *Tikrit Journal of Engineering Sciences* 23, no. 2 (2016): 10-20.
[4] Ibrahim, Thamir K., Mohammed Marwah Noori, and Ibrahim Hassan. "Effect of perforation area on temperature distribution of the rectangular fins under natural convection." *ARPN Journal of Engineering and Applied Sciences* 11, no. 10 (2016): 6371-6375.
[5] Mhamuad, Aziz7 M., Thamir Kh Ibrahim, and Raid R. Jasim. "Determination of the temperature distribution the perforated fins under." *Tikrit Journal of Engineering Sciences* 15, no. 2 (2008): 63-78.
[6] Abu-Nada, Eiyad. "Application of nanofluids for heat transfer enhancement of separated flows encountered in a backward facing step." *International Journal of Heat and Fluid Flow* 29, no. 1 (2008): 242-249.

[7] Kumar, Saurav, and S. Vengadesan. "Control of separated fluid flow and heat transfer characteristics over a backward facing step." *Numerical Heat Transfer, Part A: Applications* 73, no. 6 (2018): 366-384.
[8] Ibrahim, Thamir K., and M. M. Rahman. "Parametric simulation of triple-pressure reheat combined cycle: A case study." *Advanced Science Letters* 13, no. 1 (2012): 263-268.
[9] Ibrahim, Thamir K., and M. M. Rahman. "Effect of compression ratio on the performance of different strategies for the gas turbine." *International Journal of Automotive and Mechanical Engineering* 9 (2014): 1747.
[10] Mohammed, Mohammed Kamil, Omar I. Awad, M. M. Rahman, G. Najafi, Firdaus Basrawi, Ahmed N. Abd Alla, and Rizalman Mamat. "The optimum performance of the combined cycle power plant: A comprehensive review." *Renewable and Sustainable Energy Reviews* 79 (2017): 459-474.
[11] Ibrahim, Thamir K., and M. M. Rahman. "Optimum performance improvements of the combined cycle based on an intercooler–reheated gas turbine." *Journal of Energy Resources Technology* 137, no. 6 (2015): 061601.
[12] A.A. Razali, A. Sadikin, S. Ayop, "Structural analysis and morphological study of Al2O3 nanofluids in microchannel heat sink," 2018.
[13] S. Hajar, A. Mustaffa, F. Al, Z. Mohd Saat, E. Mattokit, "Turbulent Vortex Shedding across Internal Structure in Thermoacoustic Oscillatory Flow," 2018.
[14] Shaharuddin, N. H., and Mat Ali. "S., Muhamad, S., Shaikh Salim, SA and Usman, M."Flow simulations of generic vehicle model SAE type 4 and DrivAer Fastback using OpenFOAM."." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 37 (2017): 18-31.
[15] Sainju, Arpan Man, and Travis Atkison. "An Experimental Analysis of Windows Log Events Triggered by Malware." In *Proceedings of the SouthEast Conference*, pp. 195-198. ACM, 2017.
[16] M. A. Al-Dhaher, M. H. Al-Mosawi, A. Al-Sammarraie, "An Experimental Study on Forced Convection Heat Transfer from a Horizontal Cylinders Array in Cross Flow," 2003.
[17] Al-Sammarraie, Ahmed Tawfeeq Ahmed. "Natural Convection Heat Transfer from Two Parallel Horizontal Cylinders Embedded in a Porous Medium inside a Horizontal Cylindrical Enclosure."
[18] Ahmed, Mahmood H. Ali Ahmed Tawfiq, and Khalaf Ibraheem Hamada. "Numerical Study of Non-Darcian Natural Convection Heat Transfer in a Rectangular Enclosure Filled with Porous Medium Saturated with Viscous Fluid." *Tikrit Journal of Engineering Sciences* 15, no. 2 (2008): 90-111.
[19] Salimpour, Mohammad Reza, Ahmed T. Al-Sammarraie, Azadeh Forouzandeh, and Mahsa Farzaneh. "Constructal design of circular multilayer microchannel heat sinks." *Journal of Thermal Science and Engineering Applications* 11, no. 1 (2019): 011001.
[20] Fathinia1c, F., and Ahmed Kadhim Hussein. "Effect of Blockage Shape on Unsteady Mixed Convective Nanofluid Flow Over Backward Facing Step." (2009).
[21] Abubakar, S. B., and NA Che Sidik. "Numerical prediction of laminar nanofluid flow in rectangular microchannel heat sink." *J. Adv. Res. Fluid Mech. Therm. Sci.* 7, no. 1 (2015): 29-38.
[22] Singh, Yaswant, Rahul Nadda, Raj Kumar, Anil Kumar, Rajesh Maithani, and Sunil Kumar. "Heat Transfer Augmentation of Nanofluid flow in Ribbed Ducts: A Review." *Heat Transfer* 7, no. 5 (2017).
[23] Al-Aswadi, A. A., H. A. Mohammed, N. H. Shuaib, and Antonio Campo. "Laminar forced convection flow over a backward facing step using nanofluids." *International Communications in Heat and Mass Transfer* 37, no. 8 (2010): 950-957.
[24] Mohammed, H. A., A. N. Al-Shamani, and J. M. Sheriff. "Thermal and hydraulic characteristics of turbulent nanofluids flow in a rib–groove channel." *International Communications in Heat and Mass Transfer* 39, no. 10 (2012): 1584-1594.
[25] Wang, Lei, and Bengt Sundén. "Experimental investigation of local heat transfer in a square duct with continuous and truncated ribs." *Experimental Heat Transfer* 18, no. 3 (2005): 179-197.
[26] Naphon, Paisarn, Manachai Nuchjapo, and Jutarat Kurujareon. "Tube side heat transfer coefficient and friction factor characteristics of horizontal tubes with helical rib." *Energy Conversion and Management* 47, no. 18-19 (2006): 3031-3044.
[27] Wang, Lei, and Bengt Sundén. "Experimental investigation of local heat transfer in a square duct with various-shaped ribs." *Heat and Mass Transfer* 43, no. 8 (2007): 759.
[28] Colburn, Allan P. "Heat transfer and pressure drop in empty, baffled, and packed tubes1." *Industrial & Engineering Chemistry* 23, no. 8 (1931): 910-913.
[29] Molki, M., and A. Hashemi-Esfahanian. "Turbulent convective mass transfer downstream of a perforated baffle blockage." *International journal of heat and fluid flow* 13, no. 2 (1992): 116-123.
[30] Zaidan, Maki H., Aadel AR Alkumait, and Thamir K. Ibrahim. "Assessment of heat transfer and fluid flow characteristics within finned flat tube." *Case Studies in Thermal Engineering* 12 (2018): 557-562.
[31] Ibrahim, Thamir K., Marwah N. Mohammed, Mohammed Kamil Mohammed, G. Najafi, Nor Azwadi Che Sidik, Firdaus Basrawi, Ahmed N. Abdalla, and S. S. Hoseini. "Experimental study on the effect of perforations shapes on

vertical heated fins performance under forced convection heat transfer." *International Journal of Heat and Mass Transfer* 118 (2018): 832-846.