J. Basic. Appl. Sci. Res., 2(4)3268-3275, 2012
© 2012, TextRoad Publication

ISSN 2090-4304
Journal of Basic and Applied Scientific Research
www.textroad.com

Control of Turbulent Flow in a Stepped Duct with Using the Suction and Injection

Y.Bakhshan, S.Niazi, M.Mahmeli

Department of Mechanical Eng., University of Hormozgan, P.O.Box 3990,Iran

ABSTRACT

In this research, the turbulent flow sudden expansion in a channel with suction and blowing at the walls has been studied numerically. For this purpose, the channel withrectangular cross-section, which changes suddenly and creates a step and is suitable for flow separation study, has been considered. In the down part of wall of the backward-facing step, a tape shape area for suction or injection process flow is intended.Fluid flow is supposed in turbulent regime, incompressible, 2D and steady state. Different turbulent models have been applied in modeling of turbulence and the comparison of results with available experimental data shows good agreement between the numerical and experimental results. Also, in our studies thestandard k–ε turbulence model has better compatibility from other models for this kind of problems.
KEYWORDS: Channel flow, Backward-facing step, Reattachment control, Blowing; Suction, k–ε.

NOMENCLATURE

Latin Equation Symbols

a	**effective slot gap (m)**
d_h	hydraulic diameter (m)
h	step height (m)
p	wall pressure (Pa)
w	channel width (m)
C_p	pressure coefficient
C_Q	suction/blowing flow coefficient
F	cross-section area (m^2)
H	channel height (m)
Re	Reynolds number
U	mean velocity
ρ	fluid (air) density (kg/m3)

Subscript Equation Symbols

e	**at the inlet of the test section**
s	at the suction/blowing slot

1. INTRODUCTION

Shear flow with sudden expansion of cross-section has several applications in Mechanical Engineering and Civil Engineering courses. Flow separation will be resulted to negative effects such as drag increase, lift decrease and noise production.

One of the most practical methods in drag reduction and lift increase in the control of boundary layer is the using of suction and injection methods.

In the flow over airfoils, use of suction and blowing process not only lead to lift increase and considerable drag decrease, but also is very effective in considerable noise decrease and the sound which is resulted of airfoil back trail. This method has been considered widely in recent years. With this method, the size of the separation region can be easily controlled. So, the study of suction/blowing and geometric parameters (such as intended surface for blowing/suction, locating the place of blower/sucker in the channel or the geometry of channel) and also fluid parameters (such as velocity of fluid blowing/suction to the channel, pressure control in the effect of blowing/suction and or the effects of blowing/suction in flow turbulence) of flow in the channel is very important and can empower the engineering ability in production of various kinds of airplanes and submarines.

The most common aim of control of separated flows is the control of flow separation region. In the year 1904, Prantdl had presented a modern method for the control of boundary layer. He, in his famous article, had provided successful experiments in which he had been able to considerably control the boundary-layer separation. Prantdl in his experiments, had used flow suction method to delay the boundary-layer separation on the cylinder[1].

***Corresponding Author:** Y.Bakhshan, Department of Mechanical Eng., University of Hormozgan, P.O.Box 3990,Iran
Email: ybakhshan47@gmail.com

Flow on a step is very likely to its geometry (but the structure of the flow is more complex). Several Experimental data are collected by use of hot wire and Laser, and observing flow in simple 2D case of abackward-facing stepin Reynolds numbers which of such can address to the Biswasresearches in 2D case and Leein 3D case and in the above-mentioned Reynolds numbers[2,3].Sakurabawas of first researchers who studied the flow control in recirculation area[4].Li and his co-operators had researched in the field of optimized control of loop flows of the bottom (wake) of the cylinder. The flow control will be performed through suction/blowing on the cylinder shell. Here, governing equations on the fluid movement are the same 2D naviar stocks, incompressible which in this research is solved by the method of finite volume and depends on time boundary conditions[5].

Kanna et al.studied the Conjugate heat transfer specifications for the issue of flow on a backward-facing step. They studied the effects of conjugate in relation with four parameters of Re, Pr, K and b and the effect of four parameters on the local nusselt number, interface temperature and average nusselt number are expressed in detail. Here, the energy equations which are dominant on the fluid and solid areas are solved simultaneously[6].

Antonia et al. [7] investigated experimentally the effects of suction on a spatially evolving turbulent boundary layer. When high suction rates were implemented, relaminarization was produced downstream of the slot and a considerable reduction in the total skin friction was achieved. The influence of constant suction on velocity and thermal fluctuations over a surface was also experimentally investigated by Antonia et al. [8] and Antonia and Fulachier [9]. Near the wall region, suction induced a stabilization effect. Park et al. [10,11] carried out experimental studies in a wind tunnel to examine the flow structures behind time-periodic blowing/suction in a flat.

Urba et al. studied experimentallythe channel flow on a backward-facing stepwith the method of blowing/suction. The gap of the bottom of the step has been in the different forms of rectangular or jagged, and the current strength has been controlled by the suction/blowing flow coefficient. Performed experiments include various areas of Reynolds numbers. The initial results showed that both suction and blowing methods are able to decrease the length of the separation region to the quantity of 1/3 of its quantity in un-controlled case and in the result, the recommended control method based on suction or blowing in step is effective in the field of shortening the length of the recirculation region. In order to experimental study of the length of the separation region for blowing, suction, the method of indirect measurement by use of pressure distribution of the shell has been used[1].

The aim of this research is solving the equations which are dominant on passing fluid flow of the step surface in order to study the effect of blowing, suction and sudden change of cross section on the velocity field and pressure coefficient.

The main hypothesizes in this simulation are:

1. The flow is turbulent and steady.
2. The gravity power is ignored.
3. The fluid flow is supposed as incompressible.
4. The flow is supposed 2D.
5. Boundary conditions and the geometry of issue are determined in a way that the result of this numerical simulation can be compared with the experimental data of Urba et al[1].

2. Computational Domain

As mentioned before, in order to be able to finally perform a comparing of numerical simulation with experimental data, the computational domain is supposed in a way that be fit with experimental setup in the research of Urba et al [1]. So, considering the assumption of 2D flow, we will define the computational domain as below:

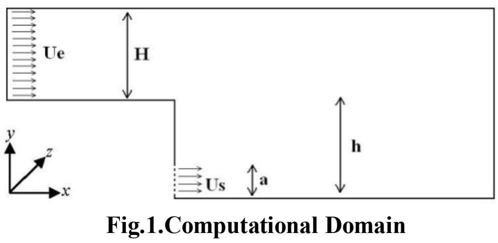

Fig.1.Computational Domain

The channel's specifications,are shown in Tables 1 and 2.

Table 1.

W	0.1 m
H	0.25 m
H	0.025 m
$Re(d_h, U_e)$	5×10^4

Table 2.

Type	a (m)	F_s (m^2)	F_e (m^2)
Rect.	2.25×10^{-3}	225×10^{-6}	25×10^{-3}

3. MODEL FORMULATION

Governing equations on the fluid movement in this research are the same of general basic equations which, considering the assumptions of turbulent, steady and incompressible flow and ignoring the gravity in the 2D case will be in the following form:

$$\frac{\partial U_i}{\partial X_i}=0 \qquad (1)$$

$$\rho U_j \frac{\partial U_i}{\partial X_j}=-\frac{\partial P}{\partial X_i}+\mu\left(\frac{\partial^2 U_i}{\partial X_j \partial X_j}\right) \qquad (2)$$

Where U is velocity, P is pressure, $\rho$is density, and $\mu$is the coefficient of air viscosity. A standard method for solving the turbulent flow is the Reynolds averaging method. In this method, the quantities of the properties and solving variables will be separated to two parts of fluctuation and average. So, for the present variables will have:

$$U=\bar{U}+U' \qquad (3)$$

$$P=\bar{P}+P' \qquad (4)$$

So, governing equations on turbulent flow for governing equations of continuity and Navier- Stocks for turbulent flow will be as below:

$$\frac{\partial \bar{U}_i}{\partial X_i}=0 \qquad (5)$$

$$\rho \bar{U}_j \frac{\partial \bar{U}_i}{\partial X_j}=-\frac{\partial \bar{P}}{\partial X_i}-\rho\frac{\partial \overline{U_i' U_j'}}{\partial X_j}+\mu\left(\frac{\partial^2 \bar{U}_i}{\partial X_j \partial X_j}\right) \qquad (6)$$

Which in the above equations, $\bar{U}$&$\bar{P}$respectively express velocity and pressure average of the fluid and also U'&P' are respectively express velocity and pressure fluctuations.

In the entrance boundary condition, the developed profiles flow with the free flow entrance velocity of 5.6 m/s is applied. The walls have the condition of non-slip.

Blowing and suction at the bottom of stepare also entered to consider boundary conditions and we used the following non-dimensional coefficient, for flow rate:

$$C_Q=\frac{\rho_s U_s F_s}{\rho_e U_e F_e} \qquad (7)$$

Which in this equation, the positivevaluesof$C_Q$mean blowing process and the negative values of $C_Q$show the suctionprocess. The case of C_Q=0 shows the non-blowing and non-suction case. Also, as mentioned before, to be able to finally compare the result of numerical simulation with experimental data, the computational domain is in intended as the experimental domain of Urba et al.In terms of geometric dimensions and flow specifications[1].

To solve physics equations, the finite volume method has been used. At the conditions used here, the mach numberis very low and so the compressibility of the flow is ignored. Considering the incompressibility of the flow, density changes resulted from changes of pressure and temperature, were not affected the flow equations, and the simpler algorithm has been used for coupling the pressure and velocity field. Thus with considering the network complexity and issue convergence, the implicit formulation has been applied. The flow solved in the form of steady state and three models of turbulence (standard K-ε, K-ε Realizable and K-ωhave been used.

4. RESULTS AND DISCUSSIONS

4.1Grid generation

As mentioned in the previous section, the generated mesh is in theCartesian vertical system and is compressed in the near ofboundaries and in the place of flow sudden expansion. For mesh independency study, the results of numerical simulation have been extracted for different values of control volumes generated in the computational domain and compared with each others, so in this study the optimum value of control volumes have been obtained about108200 and the results have not been changed with increasingcontrol volumes from this value so, the calculation in continuinghas performed with this value. Figures 2 and 3show the computational domain with optimum generated grids:

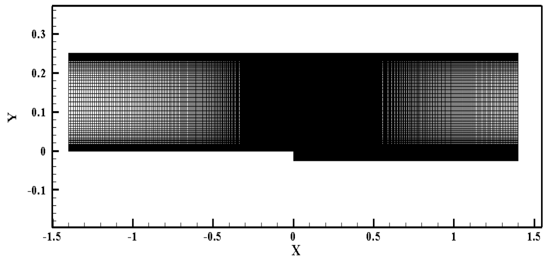

Fig.2. Grids generated in the computational domain

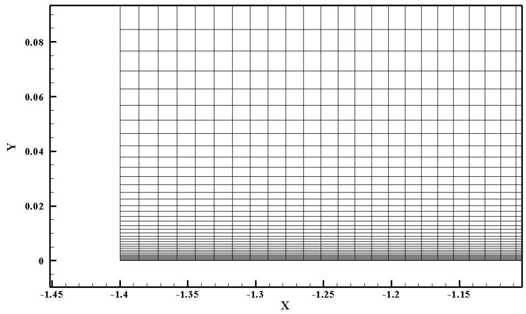

Fig.3. Generated gridsis compressed at the channel wall

4.2The effect of flow break down on the velocity and pressure profiles:

In order to study the impact of step on the velocity and pressure profiles, thegoverning equations are solved without considering the conditions of blowing and suction. Figure4, shows the distribution of velocity contoursin the whole of computational domain and the figure 5,showsstreamlinesin the whole of computational domain.

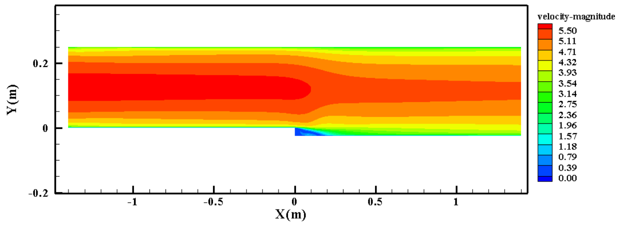

Fig.4. Velocity contours in the computational domain

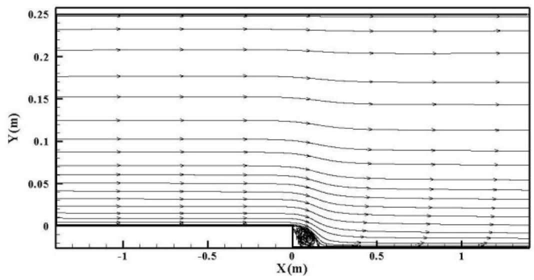

Fig.5. Streamlines in the computational domain

With attention to the figures 4 and 5, it will be clear that in the bottom of step, we see the sharp drop in the pressure due to existence of fracture, and the quantity of velocity in the bottom of step shows 20 times more pressure drop than the upper section. In figure 6, pressure distribution contour in the whole of computational domain has been shown. It shows that in the bottom of the step, we will observe the increase in pressure drop in turbulent flow.

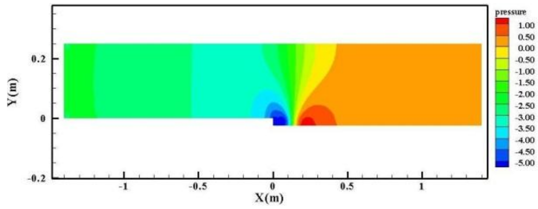

Fig.6. pressure contours in the computational domain

4.3Validation of results

In order to study the accuracy of performed simulation, the results of this numerical simulation are compared with the experimental data of Urba et al [1]. At first, governing equations are solved with the boundary conditions in the case of without blowing and suction by applying thestandard K-ε, K-εRealizable and K-ωturbulence models, so, the pressure coefficient is calculated and compared with experimental data (figure 7).

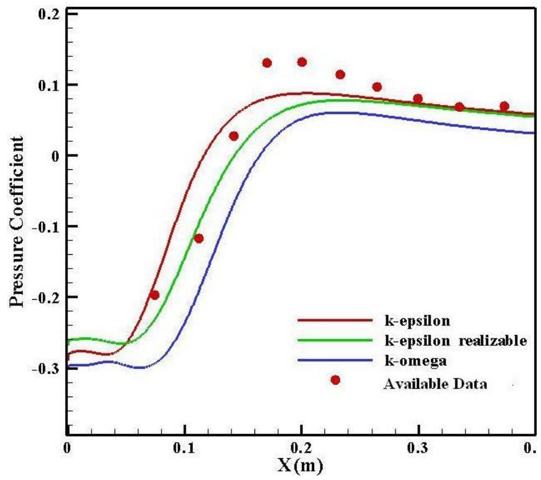

Fig.7. pressure coefficient of the bottom of the step and along the flow

As shown in figure 7, we can find that, in the case of without blowing and suction, there is a good agreement between numerical simulation and the experimental data.Also the standard K-εturbulence model shows a better compatibility with experimental data.

4.4Studying the effect of blowing and suction on the flow

As mentioned before, because of high pressure drop, noise and velocity drop in the place of turbulent flow break down on the step, we will used the blowing and suction mechanism to control the flow. Blowing and suction at the bottom of the step are entered as boundary condition for the flow rate and to show the effect of them, we useda non-dimensionalcoefficient[C_Q]. Figures8, 9,10, 11, 12and 13show the results of simulation in cases of without blowing and suction and with suction/blowing. Indeed, these figures show the effect of blowing and suction on optimizing streamlines and decreasing the drops, which are resulted from flow sudden breakdown.Fig.9 shows the streamlines of the flow with no suction or injection and the circulation of flow after the step is created due to pressure drop at the step and this increases the risk of flow separation from the wall. The streamlines are shown in fig.11 with injection from the wall, and it affects the size of generated wake at the bottom of step and decreases the risk of flow separation from wall.Fig.13 shows the streamlines of flow with suction and its effect on the circulation of flow at the bottom of step, it is clear that the risk of flow separation from the wall is decreases mostly. The increasing of pressure coefficient is shown in the figures 14 and 15 with applying the injection and suction at the wall. These figures show the increasing of the pressure coefficient with applying the injection and suction and this means the control of flow without separation in the duct.

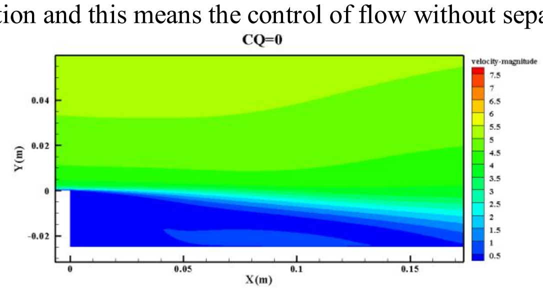

Fig.8. Velocity distribution contours at the bottom of step for $U_S = 0$ m/s

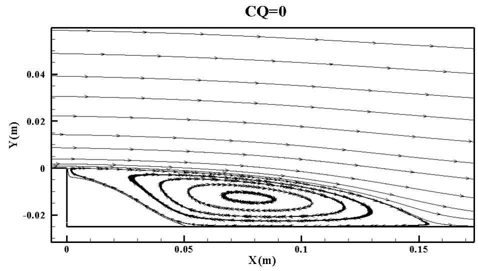

Fig.9.Streamlines at the bottom of step for $U_s = 0$ m/s

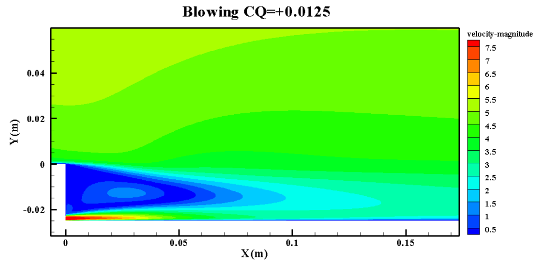

Fig.10. Velocity distribution contours at the bottom of step for $U_s = +7.72430556$ m/s

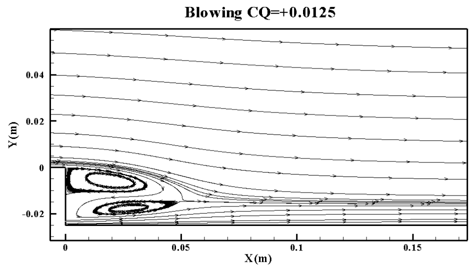

Fig.11. Streamlines at the bottom of step for$U_s = +7.72430556$ m/s

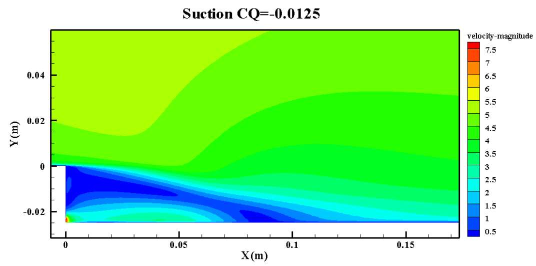

Fig.12. Velocity distribution contours at the bottom of step for $U_s = -7.72430556$ m/s

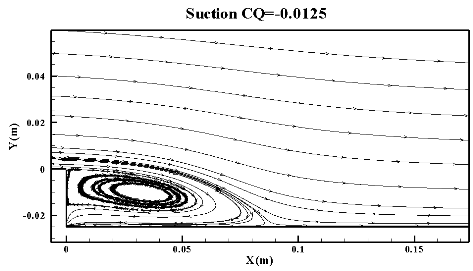

Fig.10. Streamlines at the bottom of step for $U_s = -7.72430556$ m/s

As mentioned in previous section, in the solving governing equations in this research, three turbulence models are used for modeling of turbulence and the result of standard K-ε shows a better compatibility with experimental data. So, in the case that blowing and suction are also entered to the issue as boundary conditions, justgoverning equations are solved by this standard method. In figure 11, a comparison has been performed between numerical simulation and experimental data for suction case. In figure 12, this comparison has been performed for the case of injection. They show a very good compatibility of the results of numerical simulation with the experimental.

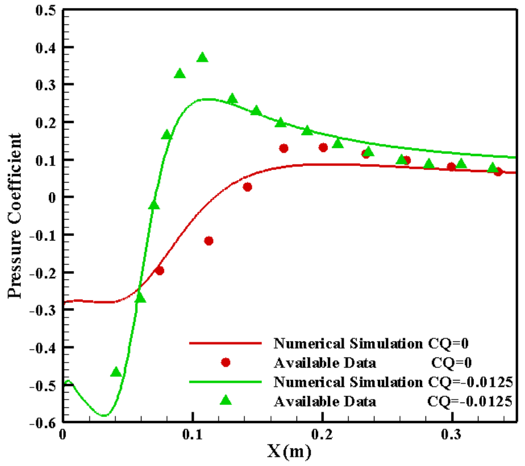

Fig.11.Comparison diagram of pressure coefficients of bottom of step and along the flow for the case of suction

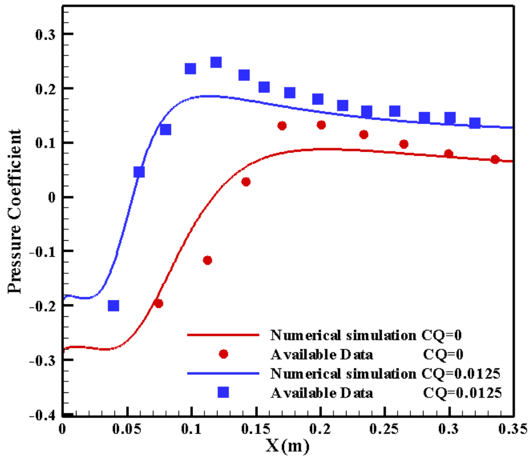

Fig.12. Comparison diagram of pressure coefficients of bottom of step and along the flow for the case of blowing

5. CONCLUSION

In this study, the numerical simulation of turbulent flow in a channel with sudden expansion has been donned. The results show, the injection or suction process can be suitable for control of turbulent flow and decreasing the risk of flow separation from the wall for this kind of problems. Comparison the numerical results with available experimental data show good agreement with them. Also this study results the standard K-ε turbulence model has better compatibility with numerical solution from other models.

REFRENCES

[1] Va´clavUruba, Pavel Jonas, Oton Mazur, 2007. Control of a channel-flow behind a backward-facing step by suction/blowing. International Journal of Heat and Fluid Flow 28 (2007) 665–672

[2] Va´clavUruba, Pavel Jonas, Oton Mazur, 2007. Control of a channel-flow behind a backward-facing step by suction/blowing. International Journal of Heat and Fluid Flow 28 (2007) 665–672

[3] Biswas, G., Breuer, M., Durst, F., 2004. Backward-facing Step flows for various expansion rations at low and moderate Reynolds numbers Transactions of the ASME. J. Fluids Eng. 126, 362–374.

[4] Sakuraba, K., Fukazawa, K., Sano, M., 2004. Control of turbulent channel flow over a backward-facing step by suction or injection. Heat Transfer – Asian Res. 33 (8), 490–504.

[5] Zhijin Li, I.M. Navon, M.Y. Hussaini, F.-X. Le Dimet, 2003. Optimal control of cylinder wakes via suction and blowing. Computers & Fluids 32 (2003) 149–171

[6] P. Rajesh Kanna, Manab Kumar Das, 2003. Conjugate heat transfer study of backward-facing step flow – A benchmark problem.

[7] R.A. Antonia, Y. Zhu, M. Sokolov, Effect of concentrated wall suction on a turbulent boundary-layer, Phys. Fluids 7 (10) (1995) 2465–2474.

[8] R.A. Antonia, L. Fulachier, L.V. Krishnamoorthy, T. Benabid, F. Anselmet, Influence of wall suction on the organized motion in a turbulent boundarylayer, J. Fluid Mech. 190 (1988) 217–240.

[9] R.A. Antonia, L. Fulachier, Topology of a turbulent boundary-layer with and without wall suction, J. Fluid Mech. 198 (1989) 429–451.

[10] S. Park, I. Lee, H. Sung, Effect of local forcing on a turbulent boundary layer, Exp. in Fluids 31 (2001) 384–393.

[11] Y. Park, S. Park, H. Sung, Measurement of local forcing on a turbulent boundary layer using PIV, Exp. in Fluids 34 (2003) 697–707.