Journal of Novel Applied Sciences
Available online at www.jnasci.org
©2014 JNAS Journal-2014-3-S2/1671-1674
ISSN 2322-5149 ©2014 JNAS

An Investigation of Sudden Contraction Effect of Different Cross-Sections on Velocity and Pressure in Pipes with 90° Bend

Behrooz Zad Shakouyan* and Rasool Daneshfaraz

Department of Civil Engineering Maragheh Branch, Islamic Azad University, Maragheh, Iran

***Corresponding author*:** Behrooz Zad Shakouyan

ABSTRACT: In the present study, the flow in a pipe line with 90 degree bend with different sudden contractions in bend inlet is simulated numerically by use of Fluent Software. The fluid flow regime in bend is turbulent and laminar with Reynolds numbers varying from 100 to 10000 (100, 500, 850, 1800, 4800, 10000). The main objective of this study is investigating the effect of sudden contraction. This amount is expressed as relative contraction parameter in percent which include 0 (n contraction), 20, 40 and 60 percent.
The results indicate that in spite of sudden contraction in bend inlet, the maximum velocity of incidence is in 67.5° and 90°. Flow divergence in velocity profiles is not perceived except in cross sectional angle of 45° and 67.5° in which a great inclination towards divergence is observed. Also, a negative pressure in bend with a sudden contraction of 20 % in Reynolds number of 10000, in bend with sudden contraction of 40% in Reynolds number of 4800 and 10000 and in bend with sudden of 60% in Reynolds number of 850, 1800, and 4800 and 10000 is observed. The presence of sudden contraction results in an increase in pressure drop in 90° bend in compare to neutral state. This effect is more significant in higher Reynolds numbers. Increasing relative contraction, the highest pressure drop occurs in 45° while in non-contraction condition, the highest amount of pressure drop occurs at 67.5 ° and 90°.

***Keywords*:** bend, numerical simulation, Fluent, Sudden Contraction, Velocity and pressure distribution.

INTRODUCTION

Bending and flow divergence is an inherent part of piping system, extrusion, dam structures such as diversionary tunnels, water passages and so on. For this reason, available patterns of flow in bends make analytical solutions almost impossible. On the other hand, the sudden contraction which occurs, for example due to construction failure of blockage in flow passages, disturbs velocity and pressure distribution which in turn results in hydraulic system efficiency decrease.

The literature review in this study is limited to under-pressure cross sections. Bovendeerd. (1978) studied flow velocity of profile and counter in 90° bend based on finite element method. They used parabolic section as inlet condition and presented a coherent definition of flow field throughout the bend, the intensity of secondary movements and axial profile of velocity in a diversity of sections. They also compared their findings with previous studies in which uniform profiles had been used and mentioned the difference of flow development in different inlet conditions (1).

Van de Vosse . (1989) modeled a 3D under pressure flow in 90° bend based on finite element (Galerkin). The comparison of velocity profiles with experimental results indicated a good agreement (2). The isolated turbulent flow in U-shaped sections examined by several researchers as Hsieh and Chen among others (3). Nakayama . (2003) conducted their experimental study on 180° duct and investigated the results of measurement in separated sections and Reynolds' stress distribution in flow passages (4).

Sparrow . (2009) studied separation in diverging conical duct with regard to the effect of Reynolds number and divergence angle and depicted a diagram to change dimensionless parameter without diameter separation in compare to Reynolds number based on divergence angles (5). Sadeghfam and Akhtari (2012) investigated numerically the separated flow in bends with different angles and developed some equations for height and thickness of separated domain (6). Daneshfaraz (2013) simulated a flow pattern in an under pressure bend with 90° angle by use of Fluent Software and investigated velocity profile and pressure distribution behavior in different cross sections throughout the bend (7).

According to aforementioned issues, this paper aims to have a closer look at flow pattern from numerical point of view and investigates the effects of sudden contraction on velocity and pressure distribution profiles.

2. Model geometry, meshing and boundary conditions

2.1 model geometry

In this study, flow in a pipe line with 90° bend was simulated by help of Fluent Software. The flow in bend was pressurized. The investigation of the effect of flow contraction is the main objective of this study. To this end, at first, flow simulation conducted in neutral state, i.e., contraction-free state, in Reynolds number set at 300 to confirm simulation validity. In general, the given bend had two horizontal and perpendicular arms of 300 and 150 mm respectively and the diameter of 8 mm. in order to get a fully developed flow in bend inlet, the length of input arm assigned longer than output arm. Moreover, to remove the dimension effect, all diagrams and parameters are represented dimensionless in this section. It’s worth to note that entering flow regiment was in domain of laminar and turbulent regime. Curvature radius of the bend was 24 mm. due to the small size of sample and for removing dimension effect, the finding of this study are investigated by dimensionless parameters. During flow analysis, the state of flow assumed steady and in domain of laminar and turbulent regime.

As it was already mentioned, the main objective of this study is investigating the possible effect of sudden contraction in bend section with 90° angle. For this purpose, three sudden contraction with different intensities embedded in bend inlet. Thus, the effect of three contraction with a gradient of 20, 40 and 60 % is investigated. The represented figures in percent imply that for example 20% of bend section inlet involved in sudden contraction. Flow regime in 90° bend in both conditions of contraction and non-contraction was laminar and turbulent and Reynolds numbers, as the most important indicator of flow was set at 100, 500, 850, 1800, 4800 and 10000. As a result of numerical simulations of above mentioned Reynolds numbers, velocity and pressure parameters in different sections extracted and investigated. Ithas to be remarked that in this study, section angels in bend are named according to the angle of the section relative to linear axis. Therefore, the angle of these sections include 0° (bend inlet), 22.5, 45, 67.5 and 90° (bend outlet) (see figure 1).

2. 2. Boundary conditions

Boundary conditions are essential in differential equations solution. Fluent Software considers all boundary conditions so that users can select the most compatible one according to their problem.

The bend in non-contraction condition in Gambit environment is shown in the figure as follows. In this study, 4 boundary conditions are considered including inlet boundary in which inlet velocity is given and the outlet boundary. Pipe casing is considered as wall boundary condition. No-slip condition is presupposed in this type of boundary condition. The forth boundary assigned as symmetry boundary condition because of pipe symmetry and calculation simplification and increasing convergence time. The following figure depicted symmetry boundary condition of the bend.

2.3 meshing

Gambit Software was used for meshing. The applied Meshing was hexahedral structured grid. Because of model’s unstructured irregular geometry in bends with sudden contraction and difficulty of applying the meshing on them, structured meshes with shorter analysis time and more favorable results applied for bend simulation. The schematic diagram of the meshing of the bend with sudden contraction of 60% is shown in Fig. … .As it can be seen, meshes in the region of sudden contraction are applied with smaller intervals to show the model changes more preciously.

3. Numerical solution and results

As already discussed in preceding sections, in order to implement Navier- Stokes equations, they must be discretesized at first. There are various methods in Fluent Software ford is cartelization of equations. In the present study, standard design is used for discretization, for discontinuation of multiline equations’ expression and turbulent equation, upwind design was used and simple algorithm was used for concurrent coupling of velocity and pressure.

3.1 model verification

In this section, the results of numerical simulation in neutral state (non- contraction) are compared with the findings of experimental study of Olson *etal.* in Reynolds number of 300 in different cross- sections. According to depicted figures, there is a good agreement between the numerical findings of the two study. This agreement validates the numerical simulation of the present study. In these figures, the horizontal axis represents the normalized pipe diameter (0 to 1) and perpendicular axis represents the rate of axial velocity to mean inlet velocity.

RESULTS AND DISCUSSION

Following numerical analysis, the results of velocity profile in Reynolds number of 100 to 10000 were investigated. For turbulent flow, RNGk-ε model was implemented. In this section, the results of velocity profile of 90° bend with relative contraction of 0, 20, 40 and 60 % are provided. Indepicteddiagrams, horizontal axis represents normalized diameter of pipe section and perpendicular axis represents dimensionless velocity which is the rate of axial velocity to mean inlet velocity. Moreover, in the presented diagrams, the amount of velocity profile of different cross- sections (0 to 90°) are illustrated in figures legend.Increasing Reynolds number results in velocity profile inclination towardsexternal wall of bend. Flow separation was not observed in velocity profiles except in cross sectional angles of 45 and 67.5° in which there was a great inclination towards separation. Also, inspecting incidence spot of maximum velocity showed that it occurred in 67.7° and 90º angles.

This section also illustrates the results of pressure distribution in 90º bend with different ratios of contraction. Similar to velocity profiles, diagrams also depicted distinctly based on different Reynolds numbers in which pressure distribution presented based on cross section angle. Horizontal axis represents normalized pipe diameter and perpendicular axis representsaxial pressure ratio to mean inlet pressure. Note that here, pressure implies the total pressure that is the sum of static and dynamic pressures. according to pressure distribution diagrams, its concluded that a negative pressure in bend with sudden contraction of 20% in Reynolds number of 10000, in bend with sudden contraction of 405 in Reynolds number of 4800 and 10000 and in bend with sudden contraction of 60% in Reynolds numbers of 850, 1800, 4800 and 10000 is obvious. Afterward, the results of relative pressure drop in 90º bend with different cross sections illustrated. The correspondent diagrams of different contraction ratios are depicted separately. In these diagrams, the horizontal axis represents cross section angle and perpendicular axis represents pressure drop ratio to mean pressure in 0º section as dimensionless parameter. An increase in relative contraction leads to the highest relative pressure drop in 45º while in neutral state (non- contraction), the highest amount of pressure drop is observed in 67.5 and 90º. Moreover, the presence of sudden contraction causes a sudden increase in pressure drop in 90º bend in compare to neutral form. This amount is more obvious in higher Reynolds numbers.

4. Discussion and conclusion

In the present study, flow in pipe with 90º bend is numerically simulated by use of fluent software. Governing flow regime was laminar and turbulent flow with Reynolds numbers of 100 to 10000 (100, 500, 850, 1800, 4800 and 10000). Flow turbulence modelling conducted by k-εturbulence model. The main focus of the present study is investigating the effect of sudden contraction. To this end, flow in a bend with 90° angle simulated in four different conditions as non-contraction, relative contraction of 20%, relative contraction of 40% and relative contraction of 60%. Moreover, for verification of numerical modelling, the results of Olson velocity profile was used. in the study, two parameters of velocity and pressure(dimensionless) in different cross sections of 0° (bend inlet), 22.5°, 45°, 67.5° and 90° (bend outlet) are represented based on Reynolds numbers.

According to the study, increasing Reynolds number in spite of sudden contraction in bend inlet section results in velocity profile inclination towards exterior wall of bend. With regard to the findings of the study, in spite of a sudden contraction in bend inlet section, the maximum velocity is occurred in 67.5° and 90°. Also, separation of flow in velocity profiles were not observed except in cross sectional angles of 45 and 67.5° in which a great inclination towards separation was significant. Independent from Reynolds numbers and presence of sudden contraction in bend inlet, section curve angles increase results in maximum velocity drop.

Pressuredistributions diagrams depicted independently from relative contraction shows that increasing Reynolds numbers results in pressure distribution inclination towards exterior wall of pipe. negative pressure in bend with sudden contraction of 20% in Reynolds number set at 10000, bend with sudden contraction of 40% in Reynolds number set at 4800 and 10000 and in bend with sudden contraction of 60% in Reynolds number set at 850, 1800, 4800 and 10000 was observed.

Sudden contraction causes an increase in pressure drop in 90° bend in compare to neutral state. This effect is more obvious in higher Reynolds numbers. Increasingrelative contraction, the highest pressure drop occurs in 45° while in non-contraction conditions, the highest amount of pressure occurs in 67.5 and 90°.

REFERENCES

Acheson DJ. 1990. Elementary Fluid Dynamics, Oxford Applied Mathematics and Computing Science Series, Oxford University Press, ISBN 0-19-859679-0.

Batchelor GK. 1967. An Introduction to Fluid Dynamics, Cambridge University Press, ISBN 0-521-66396-2.

Bovendeerd PHM, Steenhoven AAV, Vosse FNVD and Vossers G. 1987. Steady entry flow in a curved pipe. J. Fluid Mech. 177, 233–246.

Hsieh SS, Chen PJ and Chin HJ. 1999. Turbulent Flow in a Rotating Two Pass Smooth Channel: J. Fluids Eng., (121) 725–734.

Nakayama H, Hirota M, Fujita H, Yamada T and Koide Y. 2003. Flow characteristics in rectangular ducts with a sharp 180-degree turn: Transactions of the JSME, 02(1028) 1171–1179.

Parker CB. 1994. McGraw Hill Encyclopaedia of Physics (2nd Edition), ISBN 0-07-051400-3.

Sparrow EM, Abraham JP and Minkowycz WJ. 2009. Flow Separation in a Diverging Conical Duct: Effect of Reynolds Number and Divergence Angle: J. Heat Mass Tran., (52) 3079–3083.

Sadeghfam S and Akhtari AA. 2012. Numerical Investigation of Length and Thickness of Separation Zoneafter Sudden Change of Direction in Closed Sections: J. Civil Eng. Urb., (2) 35-39.

Van De Vosse FN, Van Steenhoven AA, Segal A and Janseen JD. 1989. A Finite element analysis of steady laminar entrance flow in a 90 Curved tube. Int. J. Numer. Meth. F. 9,275-287.