Journal of Civil Engineering and Urbanism

Volume 2, Issue 1: 35-39 (2012) (Received: August 18, 2011; Accepted: December 16, 2011; Published: January 25, 2012) ISSN-2252-0430

ORIGINAL ARTICLE

Numerical Investigation of Length and Thickness of Separation Zone after Sudden Change of Direction in Closed Sections

Sina Sadeghfam[1*], Ali Akbar Akhtari[2]

[1] *MSCE Student, Department of Civil Engineering, University of Razi, Kermanshah, Iran*
[2] *Assistant Professor, Department of Civil Engineering, University of Razi, Kermanshah, Iran*

*Corresponding author's Email: s.sadeghfam@gmail.com, akhtari@razi.ac.ir

ABSTRACT:
Separation phenomenon which is created in result of the inverse pressure gradient reduces the efficiency of the hydraulic systems. Due to eddy flows, analytical solution is not possible. In this paper, the relationship between the length and the thickness of the separation zone versus velocity variations has been investigated by using Fluent, in closed sudden change of direction in square sections with different divergence angles. First, to verify the accuracy of the numerical model, and some other factors such as turbulent model, the results of numerical model have been compared with experimental results in a sudden expansion at several sections. Based on the results of sudden expansion analysis, the turbulence model k-ε (RNG) and steady flow mode are used in analysis of two dimensional bends. The results show that for the sharp curvature with central angle of 45 degrees regardless of flow velocity, separation in internal wall doesn't happen. For the sudden change of direction with a central angle of 45 to 90 degrees, flow separation is the function of divergence angles and flow velocity. In this study, the equations in terms of length and thickness of the separation are collected via abundant numerical analysis.

Keywords: Separation, Inverse Pressure Gradient, Sudden Change of Direction, Fluent, Turbulent Model k-ε (RNG)

INTRODUCTION

The fluid might cause increase of the pressure in the flow direction while facing an obstacle. Such changes in pressure are called inverse pressure gradients. The fluid, flowing in the area of boundary level, is also affected by pressure increase, as a result velocity of fluid decreases. But because the kinetic energy of the fluid inside the boundary layer is less, it is possible that the flow stops and its direction is reversed. Thereby the boundary layer separates from the boundary and is distracted. The main flow separating from boundary layer is called "Separation" that is created in result of the inverse pressure gradient. The inverse pressure gradient is a necessary condition but not sufficient for flow separation. In other words inverse pressure gradient can be existed without separation whereas inverse pressure gradient without separation isn't possible [1]. In Figure 1, the separation mechanism on a convex boundary is shown. The point of the boundary layer, where the pressure begins to reverse, is called D. At this point, velocity and its variations in y direction are zero [2]. In the following equations u and v are respectively velocity in x and y direction and p and ρ are stand for pressure and density.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{dp}{dx}+v\frac{\partial^2 u}{\partial y^2} \quad (1)$$

$$\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}=0 \quad (2)$$

$$\left(\frac{\partial u}{\partial y}\right)_{y=0}=0 \quad (3)$$

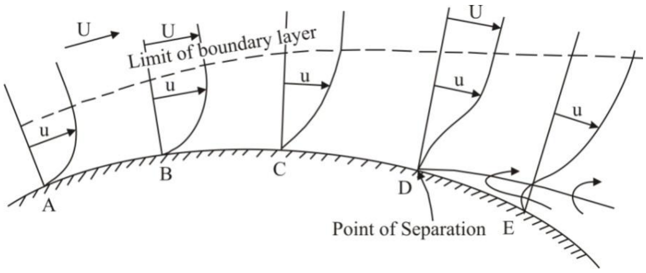

Figure 1. The separation in boundary layer through increasing pressure gradient

Generally, with no experimental data, predicting the location on the boundary, on which this condition can be established, is very difficult. Ideally, first the unsteady irrotational flow around the object should be analyzed and then by using pressures that is obtained by this analysis (Equations 1 and 2), the boundary layer for reaching above equation's conditions should be evaluated. But it is worth noting that, the abovementioned resolving method for irrotational flow, is not related to the actual flow and the calculations are generally difficult, and numerical calculations are needed. On the other hand, by reducing pressure to reach to the limit of vapor pressure of the fluid, the liquid will be boiled and will release insoluble air bubbles. With fallowing this process and increasing the size of the bubbles, closure in pipe is possible that it greatly reduces efficiency of the system. If these tiny bubbles in separated zone flow to area with more pressure, it will suddenly burst and severe hammer-like collision will occur in the boundary and an undesirable cavitation

To cite this paper: Sadeghfam S., Akhtari, A.A. 2012. Numerical Investigation of Length and Thickness of Separation Zone after Sudden Change of Direction in Closed Sections. *Journal. Civil Eng. Urban.* 2(1): 35-39.
Journal homepage: http://www.ojceu.ir/main/

will be created in the system that should be avoided in design of the hydraulic systems [3]. In Figure 2, the mechanism of flow separation and cavitation in a tube, that has been reduced its diameter, is shown.

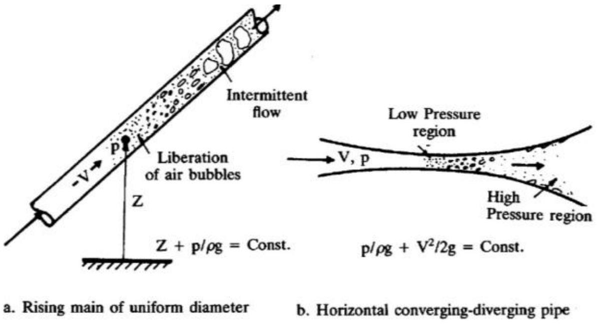

Figure 2. Separation and Cavitation in Reduced-Diameter Pipe

Mayle [4] divided the crated bubbles into small and large types and concluded that large bubbles create more loss. Mayle and Schultz [5] found that with small variation at Reynolds number, larger bubbles are created and for this reason in these type of researches Reynolds number is considered. The separated flow in U-shaped ducts with a square section is investigated by some researchers. As an example Hsieh et al. [6] investigated separated flow in these ducts likewise Nakayama et al. [7] presented profile and contour of the velocity at different sections in a turbulent flow. Also there are some researches for controlling and reducing undesirable behavior when the separation happens. For example Nagib et al. [8] introduced oscillating edges that are applied by Chung and Sung [9] in air tunnel to control the separated flow. Sparrow et al. [10] investigated the separation in ducts under the effect of Reynolds number and divergence angles, thereby variation of dimension-less parameters of separation versus Reynolds number are presented in some diagrams.

Numerical modeling of sudden expansion for verifying the model

Eaton and Johnston [11] presented the dimensionless velocity profiles at various sections by experimental investigation of turbulent flow in a sudden expansion. In this study the abovementioned expansion modeled via Fluent and the results are used to verify the bend models. Some part of these result presented in Figure 5.

Eaton and Johnson's study domain, which is illustrated in figure 3, is the turbulent flow of an incompressible fluid (water) that encounter the sudden expansion. The Reynolds number based on inlet average velocity is 132000. The ratio of step height to outlet section height is 3 $\left(\frac{H}{h}=3\right)$. In numerical modeling the conditions of inlet and outlet boundaries are, respectively, velocity inlet and pressure outlet. It is worth mentioning that the inlet velocity profile is distributed. Based on studies, between the turbulent models, k- ε (RNG) has the most performance in predicting separated flow, thereby in this study k- ε (RNG) is used for modeling of turbulent flow.

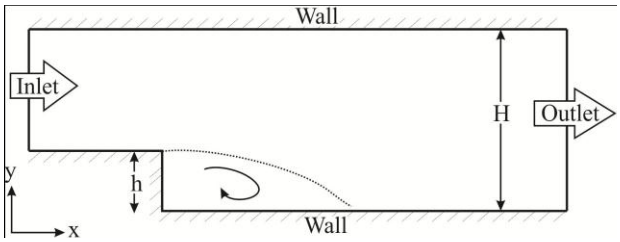

Figure 3. Schematic geometry of sudden expansion and boundary conditions

The type of meshes that are used in the modeling of sudden expansion is quadrant and map, that its size is different in some region. In this case the number of meshes is 4380 that the details are illustrated in figure 4.

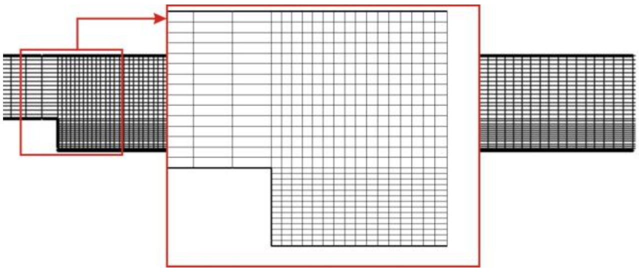

Figure 4. The details of meshes in sudden expansion

The sudden expansion that is solved for a flow with Reynolds number of 132000 and the velocity profiles compared with the experimental results of Eaton and Johnson at 4 sections (Figure 5). It is worth mentioning that the presented diagrams are dimensionless and *u* is the longitude velocity and U_0 is the average velocity of inlet. Also *y* is vertical coordination and *h* is the step height. The diagrams are presented for 4 different sections that the amount of *x/h* is 1.33, 2.67, 5.33 and 6.22.

According to Figure 5, and with regard to the least square error that at the first (x/h=1.33), second (x/h=2.67), third (x/h=5.33) and fourth (x/h=6.22) sections are, respectively, 6.09, 7.56, 8.20 and 9.80 percent, reasonable agreement correlated with experimental results are observed.

Geometry and boundary condition in bends

In this study some bends with various deviation angels have been studied. As mentioned before k-ε (RNG) is used for modeling turbulent flow. According to figure 6, dimensions of model and boundary condition, is defined. The studied bends have 2 arms with length of 1 and 2.5m and each arm has diameter of 10cm. The inclined arm has various degrees from horizon and the values are: 45, 60, 75 and 90 degree. It worth mentioning that analysis mode is steady.

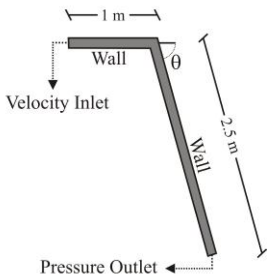

Figure 6. Geometry and boundary condition

To cite this paper: Sadeghfam S., Akhtari, A.A. 2012. Numerical Investigation of Length and Thickness of Separation Zone after Sudden Change of Direction in Closed Sections. *Journal. Civil Eng. Urban.* 2(1): 35-39.
Journal homepage: http://www.ojceu.ir/main/

Meshing is done by Gambit and uniform triangle meshes with regard to high velocity variation and the need for having fine meshes, is selected via sensitivity analysis. It worth mentioning that the distance between meshes is 5mm. Figure 7 shows an example of meshing.

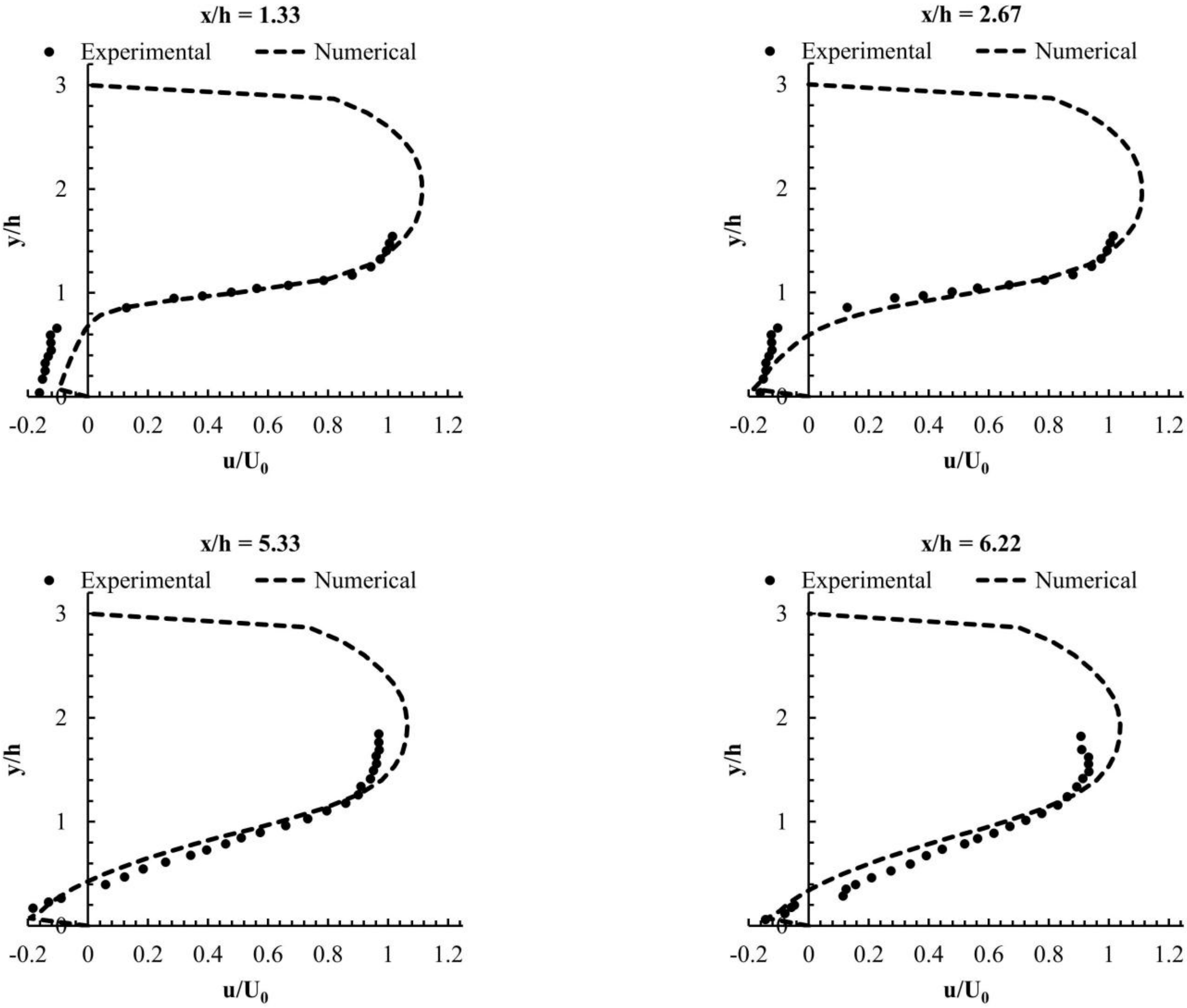

Figure 5. Velocity profiles of various sections in sudden expansion

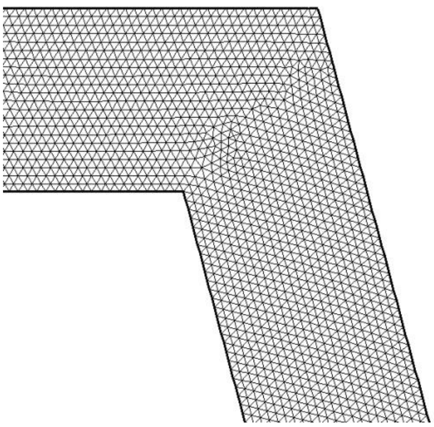

Figure 7. Meshing of bend

Separated zone

The mail goal of this study is investigating the length and thickness of separation zone. By using various velocity, two dimensionless parameter L/D and T/D (L and T are, respectively, length and thickness of separated area) vary. The type of this variation is evaluated by Tecplot software. As an example, velocity contour for the bend with division angle 75 degree and inlet velocity of 8m/s is illustrated in figure 8. The area that has negative velocity is marked with yellow and red colors and this area is separated zone. For identifying the accurate boundary, velocity contour is plotted for negative and positive velocity, thereby the length and thickness of separated zone is measurable.

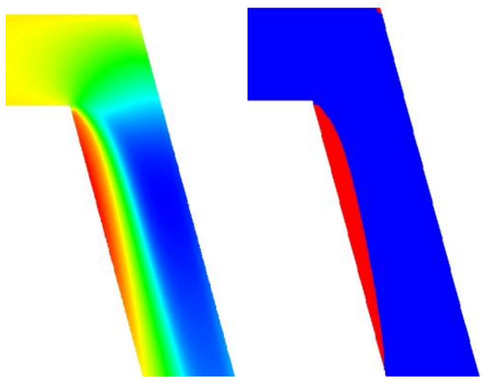

Figure 8. Velocity contour and separated area

To cite this paper: Sadeghfam S., Akhtari, A.A. 2012. Numerical Investigation of Length and Thickness of Separation Zone after Sudden Change of Direction in Closed Sections. *Journal. Civil Eng. Urban.* 2(1): 35-39.
Journal homepage: http://www.ojceu.ir/main/

Numerical analysis

As mentioned before numerical solution is done by Fluent. Creating flow domain and meshing is done by Gambit. In addition standard scheme is used for discretizing pressure and first order upwind scheme is used for discretizing momentum and turbulent kinetic energy and turbulent dissipation rate. Simple algorithm is used for pressure and velocity coupling. By applying velocities of 0.5, 1, 2, 4, 6, 8, 10, 13, 16 and 20m/s, as inlet velocities, variation of L/D for abovementioned bends versus Reynolds number is illustrated by figures 9 and 10. As it is observable in figures 9 and 10, by increasing the Reynolds number, the length and thickness of separated zone increased and by decreasing the angle of bends, separation parameters decreased. As a result there isn't any separation in bends with 45 degree divergence angles.

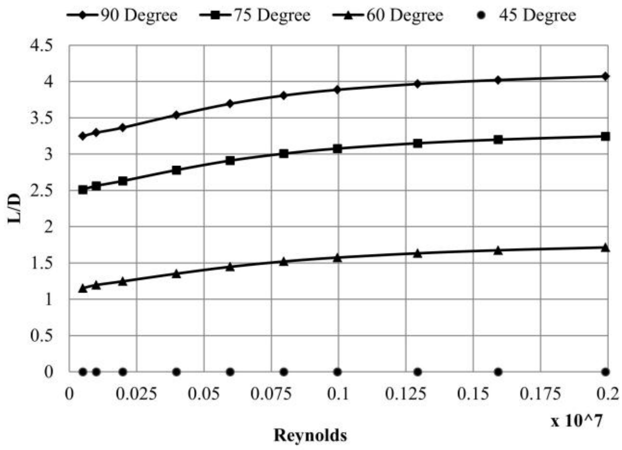

Figure 9. Variation of L/D versus Reynolds number

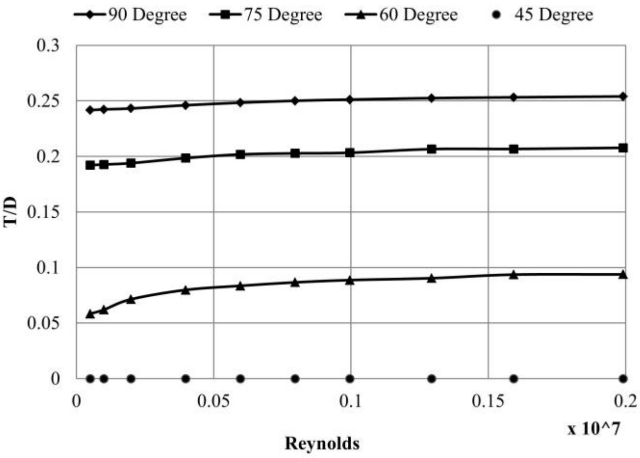

Figure 10. Variation of T/D versus Reynolds number

It appears that an equation exists between the results so polynomial equations are fitted with results; thereby some equation with high regression coefficient is observed (equation 4 to 9).

The obtained equations for 90 degree bend are:

$$\left(\frac{L}{D}\right)_{90\,\text{Degree}} = 2E-14\,(\text{Re})^2 + 9E-07\,(\text{Re}) + 3.199 \tag{4}$$

$$\left(\frac{T}{D}\right)_{90\,\text{Degree}} = 2E-14\,(\text{Re})^2 + 8E-09\,(\text{Re}) + 0.241 \tag{5}$$

The obtained equations for 75 degree bend are:

$$\left(\frac{L}{D}\right)_{75\,\text{Degree}} = -4E-13\,(\text{Re})^2 + 1E-06\,(\text{Re}) + 2.461 \tag{6}$$

$$\left(\frac{T}{D}\right)_{75\,\text{Degree}} = 3E-13(\text{Re})^2 - 3E-08\,(\text{Re}) + 0.193 \tag{7}$$

Finally the obtained equations for 60 degree bend are:

$$\left(\frac{L}{D}\right)_{60\,\text{Degree}} = -1E-13\,(\text{Re})^2 + 6E-07\,(\text{Re}) + 1.125 \tag{8}$$

$$\left(\frac{T}{D}\right)_{60\,\text{Degree}} = -2E-13\,(\text{Re})^2 + 1E-07\,(\text{Re}) + 0.052 \tag{9}$$

In above equations L and D are, respectively, length and thickness of separated zone and Re is Reynolds number. Equations 4 to 9 along with regression coefficient are displayed in figure 9.

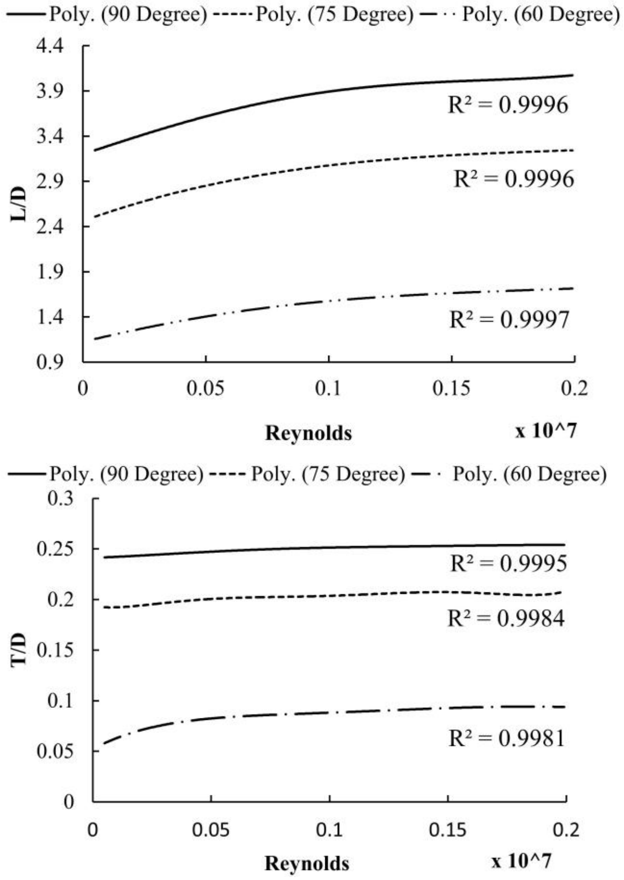

Figure 11. Fitted equations for L/D and T/D variation versus Reynolds number

CONCLUSION

In this study the relationship between the length and thickness of separated zone versus variation of Reynolds number has been investigated. Studies were done in sudden change of direction in closed sections (bends) via Fluent. First a sudden expansion has been modeled and the results have been compared with Eaton and Jonson's experimental results and acceptable agreement is observed. Numerical model of sudden expansion has good performance in predicting the initial point of separation. Thus some parameters of that (such as turbulent model and solvers) are used for analysis of the bends and flowing results are obtained. Length and thickness of separated zone has relationship with Reynolds number and this relationship is expressed through polynomial equations. The increases of the length and thickness of separated zone is increased by Reynolds number increases. For high Reynolds number this

To cite this paper: Sadeghfam S., Akhtari, A.A. 2012. Numerical Investigation of Length and Thickness of Separation Zone after Sudden Change of Direction in Closed Sections. *Journal. Civil Eng. Urban.* 2(1): 35-39.
Journal homepage: http://www.ojceu.ir/main/

variation is small. In other words equations have vertical asymptote. Finally for bends with divergence angles of 45 degree from horizon, any separation is observed.

REFERENCES

1. Shemes, I.H., 2003. Mechanics of Fluids. In: Boundary Layer Theory pp. 631-714. McGraw-Hill, New York.
2. Daugherty, R.L., Franzi, B.J., 1997. Fluid Mechanics with Engineering Applications. In: Steady Incompressible Flow in Pressure Conduits pp. 361-420. McGraw-Hill, New York.
3. Featherestone R.E., Nalluri C., 1995. Civil Engineering Hydraulics. In: Fluid Flow Concepts and Measurements pp. 47-90. Blackwell Science Ltd, Oxford.
4. Mayle, R.E. 1991. The Role of Laminar-Turbulent Transition in Gas Turbine Engines: J. Turboach., (113) 509-537.
5. Mayle, R.E., Schulz, A. 1997. The Path to Predicting Bypass Transition: J. Turboach., (119) 405–411.
6. Hsieh, S.S., Chen, P.J., Chin, H.J. 1999. Turbulent Flow in a Rotating Two Pass Smooth Channel: J. Fluids Eng., (121) 725–734.
7. Nakayama, H., Hirota, M., Fujita, H., Yamada, T., Koide, Y. 2003. Flow Characteristics in Rectangular Ducts with a Sharp 180-Degree Turn: Transactions of the JSME, 02(1028) 1171–1179.
8. Nagib, H.M., Reisenthel, P.H., Koga, D.J. 1985. On the Dynamical Scaling of Forced Unsteady Separated Flows. In the Proceedings of the 1985 AIAA Shear Flow Control Conference, pp. 12-14.
9. Chun, K.B., Sung, H.J. 1998. Visualisation of a Locally Forced Separated Flow over a Backward Facing Step: Exp. Fluids, (25) 133-142.
10. Sparrow, E.M., Abraham, J.P., Minkowycz, W.J. 2009. Flow Separation in a Diverging Conical Duct: Effect of Reynolds Number and Divergence Angle: J. Heat Mass Tran., (52) 3079–3083.
11. Eaton, J., Johnston, J.P. 1980. Technical Report MD-39, Stanford University, CA.