Available online at www.CivileJournal.org

Civil Engineering Journal

Vol. 3, No. 4, April, 2017

Numerical Analysis of Energy Loss Coefficient in Pipe Contraction Using ANSYS CFX Software

Kourosh Nosrati [a], Ahmad Tahershamsi [b*], Seyed Heja Seyed Taheri [a]

[a] *Postgraduate Student of Civil and Environmental Engineering, Amirkabir University of Technology, Tehran, Iran.*

[b] *Professor of Civil and Environmental engineering, Amirkabir University of Technology, Tehran, Iran.*

Received 4 March 2017; Accepted 29 April 2017

Abstract

The purpose of this study is the numerical analysis of energy loss coefficient in pipe contraction using ANSYS CFX software. To this end, the effect of the dimensionless parameters of Euler number, Reynolds number, and relative roughness on energy loss coefficient has been investigated and eventually an overall formula to determine the energy loss coefficient in these transitions has been provided. In order to solve the fluid turbulence equations in the pipe, standard K-Epsilon model has been used. For this purpose, first the geometry of pipe transitions was designed in 3-D, using Solid Works software, and then the transitions were meshed by ANSYS MESHING. The initial simulation of transitions including boundary conditions of outlet, inlet and wall, was carried out by a pre-processor called CFX-PRE. Furthermore, to solve the equations governing the fluid flow in the pipes (Navier-Stokes equations) the CFX-SOLVER was used. And finally, the results were extracted using a post-processor called CFD-POST.

The results indicated that the energy loss coefficient, contrary to the findings of previous researchers, is not only related to transition geometry, but also is dependent on the Reynolds number, relative roughness of the wall and Euler number. By increasing the Reynolds Number and turbulence of fluid flow in transitions, the energy loss coefficient is reduced. Moreover, by increasing the relative roughness in the transition's wall the energy loss coefficient is reduced. The increase in pressure fluctuation causes the increase of Euler number which leads to the linear increase of energy loss coefficient.

Keywords: Energy Loss Coefficient; Pipe Contraction; Standard K-Epsilon; ANSYS CFX; ANSYS Meshing.

---

1. Introduction

Pipe transitions are considered as one of the most common types of fittings in water transmission lines. These fittings are usually used in the installation of hydro mechanical equipment's in which the diameter of the pipe needs to be changed. Furthermore, in many cases, while designing the water pipeline and outlets in dams and on-site installation of butterfly valves and splits, due to the speed limit, the diameter of the pipe needs to be changed. The energy loss is mainly caused due to the friction between fluid particles with each other and also friction between fluid particles and the pipe wall. When a real fluid (viscose) flows in the pipe, it consumes part of its energy for the cohesion among particles. Through internal friction and turbulence flow, this energy is converted to thermal energy. Such an energy conversion, which is considered an energy loss, is divided into two types. The first type, which is basically in the entire length of the pipe, is called friction loss (major loss – linear loss). The second type is called minor loss (singular loss- local loss) which is caused by fittings such as valve and bends which are placed in the flow path and or caused by cross section pipe (contraction or expansion). Friction loss is mainly due to fluid viscosity and flow regime (turbulent or laminar flow). This loss is significant in the pipe length and cannot be ignored. In general, major loss (h_f) is calculated by Darcy-Weisbach (1845) equation [1]:

---

* Corresponding author: tshamsi@aut.ac.ir

$$h_f = f\frac{L}{D}\frac{V^2}{2g} \tag{1}$$

In the above equation the friction coefficient f is a function of Reynolds Number, relative roughness, and fluid regime, L pipe length, D pipe diameter, V average velocity and g is 9.81 m/s^2. To estimate the amount of energy loss (h_m), the following formula is often used [1]:

$$h_m = K\frac{V^2}{2g} \tag{2}$$

K is the energy loss coefficient which, in different resources, there are tables and charts for its calculation with regard to the type of fittings such as the bends, valves and pipe contraction or expansion, and V is the flow velocity in the pipe. Laboratory studies to determine the energy loss coefficient in sudden pipe contraction for turbulent flow were carried out by Benedict (1996) and Gerami Tajabadi (1965) [2]. Subsequent experiments to determine the loss coefficient for transition zone of flow until 7000 Reynolds Number and area ratio of 0.82 were reported by kays (1950) [3]. Energy loss coefficient measurements in a wide range of Reynolds Number from 20 to 2000 and area ratio of 0.61 were carried out by Astarita and et al (1968) [4]. Gerami Tajabadi reported his results for five different area ratios and the Reynolds numbers ranging from 50000 to 230000. Contrary to Benedict, Gerami Tajabadi considered the effect of Reynolds Number for the calculation of energy loss coefficient [5].

Prions et al (1993) studied the turbulent flow in 21-degree gradual pipe contraction. The flow rate was measured by laser velocimetry. K-epsilon turbulence model was applied. The investigations showed the poor effect of the flow characteristics of 100000 and 150000 Reynolds Number [6]. E. S. D. U. (1977) reported the energy loss coefficient for pipe contraction for compressible and incompressible fluids [7]. Durst et al (1985) studied laminar flow regime, Reynolds Number ranging from 23 to 1213 (based on the diameter of upstream pipe) and the area ratio of 0.582 using a two-beam laser velocimetry. In this study, the maximum speed location in the pipe and velocity profile in downstream contraction was done for the Reynolds Number greater than 196. They estimated that the secondary current of the created bubbles, caused by the turbulent flow in the pipe, begins at Reynolds Number of about 300. By increasing the Reynolds number, the length and depth of these bubbles increased [8].

Hooper and William (1988) presented empirical equations for calculating the energy loss coefficient in pipe and square contraction (contraction and expansion) [9]. G. Satish et al. (2013) investigated the water flow in gradual and sudden contractions (contraction and expansion) for unsteady flow and by using K-epsilon turbulence model using fluent software in order to study the velocity and pressure in these contractions. These researchers also observed that with increasing inlet mass flow in the pipe and also with increasing inlet flow speed the rate of energy loss increases [18]. Zhao Haiyan et al. (2009), by using numerical methods, showed that the rate of backflow in sudden contractions increases with increasing diameter rate [19]. Zhou Zai Dong et al. (2012), by numerical methods, showed that using higher range of Reynolds numbers in modelling for sudden contractions leads to more violent eddy currents and larger impact zone [20]. Yingpeng Li et al. (2015), by comparing experimental and theoretical methods in sudden contractions, observed that there is 1.33 per cent difference in energy loss coefficient and it is because of ignoring the roughness of the pipe walls [21].

2. Materials and Methods

In this study, three diameter ratios of 1.2, 1.5, and 2.1 have been investigated according to the standards of the Engineering society of US for five pipe contractions (including gradual and sudden) with apex angle transition of 15, 30, 45, 60 and 90 degrees [10, 17]. In order to simulate the fluid flow in pipe contraction, first, 3-D geometry transition was designed using Solid Works. To ensure the development of fluid flow in the pipe, as much as 10 times the diameter of the inlet pipe, assuming turbulent flow, a length of the pipe was considered in the beginning of the transitions [11, 17]. As an instance, for a ratio of 1.5, the geometry of conversions for two states of gradual and sudden is as Figures 1. show. For meshing the contractions, ANSYS MESHING software is used and tetrahedral mesh is used as the elements of meshing Figure 2. The initial simulation of transitions including boundary conditions of outlet, inlet, wall, fluid temperature, fluid type, selection of turbulence model, and heat transfer settings were carried out by a pre-processor called CFX-PRE. To this end, fourteen primary mass flows were considered as an inlet for speeds of 0.1 to 10 meters per second. For the outlet, the pressure is assumed equal to atmospheric pressure (relative pressure of zero). The temperature of the fluid is considered 15°C and the type of fluid is considered water and turbulence model is selected K-epsilon.

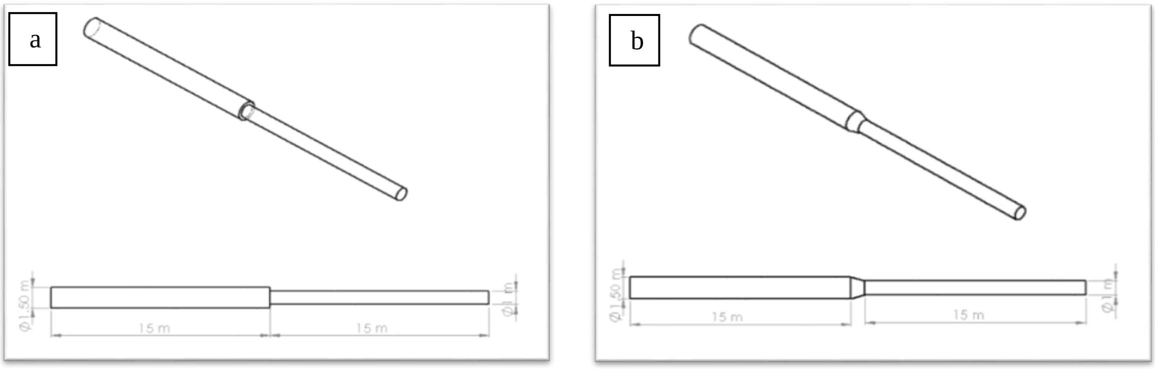

Figure 1. Geometry of sudden pipe contraction (a) and gradual pipe contraction (b) for diameter ratio of 1.5 drawn in Solid Works software

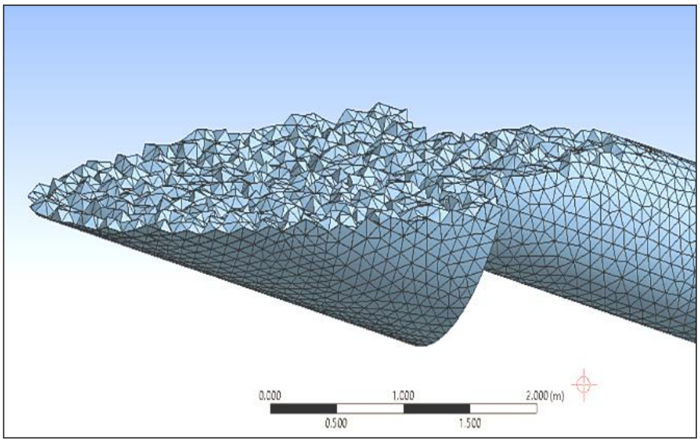

Figure 2. Tetrahedral mesh for a sudden pipe contraction in ANSYS MESHING software

The foundation of all numerical methods in hydraulic engineering is the solving of flow equations including the continuity equation, momentum equation, and sometimes energy equation. The continuity equation or the principle of mass conservation in a fluid flow is generally expressed as follows [12]:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \tag{3}$$

In the above equation, ρ is the density of water in kilograms per cubic meter, u_i is velocity rate in line with x_i in meter per second, and t is time. Navier-Stokes equations are the momentum equations governing the viscos Newton fluid flow. In general, this equation is expressed as follows:

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=\frac{\partial p}{\partial x_i}+B_i+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)\right) \tag{4}$$

In this equation u_i is then i-velocity component, B_i is body force in line with i and μ is the dynamic viscosity. Usually the viscosity of the fluid can be derived out, which causes a slight and negligible error.

2.1. Turbulence Model

K-epsilon is the most famous binary equation model, because it is more straightforward to understand and apply it in programming. In this model, by solving the conservation equation for turbulence kinetic energy (K) and the energy dissipation rate (ε), the amount of turbulence viscosity is calculated. This model can be used to simulate the turbulent flow such as shear layer and circular jet flows. In this model the amount of y^+ must be always less than 300 [13]. The main weakness of this model is the prediction and modeling of the flows with circular areas. Based on the solution of passing flow through the flat plate, y^+ can be obtained from equation (5) [13].

$$y^+=\frac{\Delta y}{\sqrt{74}LRn_L^{-13/14}} \tag{5}$$

In which $\Delta \boldsymbol{y}$ is the actual distance of first node from wall, L is the characteristic length of flow, and Rn_L is the Reynolds number in terms of characteristic length. In case of too high of y^+, the density of networks by the walls can be increased [13]. In the selection process of turbulence model, first K- epsilon turbulence model was chosen by default, and after modeling the average value of y^+ was determined. For instance, for pipe contraction with 1.2

diameter ratios and 21×10^6 Reynolds number, the amount of 26.38 was obtained. Furthermore, ANSYS recommendations for internal flows (water flow in pipe) of K-epsilon turbulence model have been suggested [13]. In addition, if any other model is used, due to the low turbulence rate in pipe contraction, significant changes will not be achieved in results.

2.2. Boundary Conditions

Inlet boundary condition is included 14 numbers of initial velocities for different diameter ratio. In fact, proportional to inlet velocities and inlet diameter, mass flow is calculated and is considered as boundary conditions .The selection of these velocities is such that it encompasses velocity real changes range in common hydraulic structures (Table 1.) [17].

Table 1. Initial condition

	Mass flow (kg/sec)		
diameter ratio	**1.2**	**1.5**	**2.1**
V(m/sec)			
0.1	113.04	176.63	346.19
0.25	282.60	441.56	865.46
0.5	565.20	883.13	1730.93
0.75	847.80	1324.69	2596.39
1	1130.40	1766.25	3461.85
2	2260.80	3532.50	6923.70
3	3391.20	5298.75	10385.55
4	4521.60	7065.00	13847.40
5	5652.00	8831.25	17309.25
6	6782.40	10597.50	20771.10
7	7912.80	12363.75	24232.95
8	9043.20	14130.00	27694.80
9	10173.60	15896.25	31156.65
10	11304.00	17662.50	34618.50

At the pipe's outlet, the flow regime is subsonic and the medium static pressure (zero relative pressure) is considered as outlet boundary condition. Pipe wall is No Slip Wall and four modes of relative roughness of pipe wall are considered according to the Table 2. [17].

Table 2. Wall boundary conditions

Type	relative roughness (mm)
smooth	0.00
Typical steel	0.06
concrete	1.65
Rivet steel	4.95

3. ANSYS CFX Software

In general, the analysis of a problem in ANSYS CFX consists of three main steps including: pre-processing (CFX-Pre), adjustment of the solver (CFX-Solver), and post-processing of the results (CFX-Post). In CFX-Pre the geometry and meshing is created. In CFX-Solver the settings related to the solver are done in the software and the problem is prepared for simulation and solving. Finally, the problem is simulated and in the last step in CFX-Post the results of the simulation is evaluated and the useful information is extracted. The ANSYS CFX solver does the discretization of Navier-Stokes equations with high resolution method and solves the equations simultaneously with the finite volume method. The project convergence criteria are to reach the root mean square (RMS) of 1×10^{-6}. Since the answer to the problem is achieved through trial and error, an amount of initial guess should be considered. In this problem the solver selects its own initial guess through inlet and outlet, and begins solving the problem. The number of repetitions considered to solve the problem is different regarding the diameter of the pipe and inlet mass flow and is obtained from 250 to 1000 receptions in each transition [17].

4. The Verification of Extracted Data from ANSYS CFX

In this paper, the experimental results of Levine (1970) have been used for sudden pipe contraction and King Brater (1963) for gradual pipe contraction without considering the effect of wall roughness. The experimental results of Levine are the basis of determining the energy loss coefficient in sudden pipe contraction of the US army corps of engineers. In this method, the wall roughness is ignored and includes the diameter ratio of 1.2 to 2.1. The range of Reynolds numbers for this researcher is 10^5 to 10^6 [13]. The experimental results of King Brater are used for determining of energy loss coefficient in gradual pipe contraction for transition apex angle of 5 to 150 degrees and the diameter ratio of 1 to 3 without considering the wall roughness. Unfortunately, the details of the experiments of this researcher are not available [14]. After the modeling, the results were extracted using CFX-PRE post-processor. To determine the energy loss coefficient in transitions, first, the average pressure in pre-transition and post-transition in pressure control sections were determined. Pressure control point, before and after the pipe contraction, the size of the output diameter from the end of the transition, is in the flow direction [17]. For an instance, for the diameter ratio of 1.5 the pressure counters and velocity vectors for 30-degree pipe contraction for the Reynolds Number of 15×10^6 have been extracted as can be seen in Figures 7 and 8.

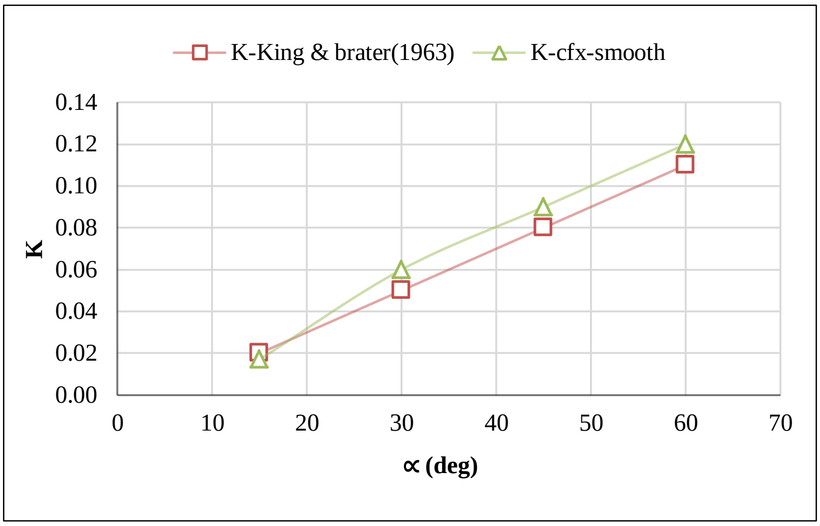

Figure 3. The verification of extracted data from ANSYS CFX with experimental results of Brater for diameter ratio of 1.2

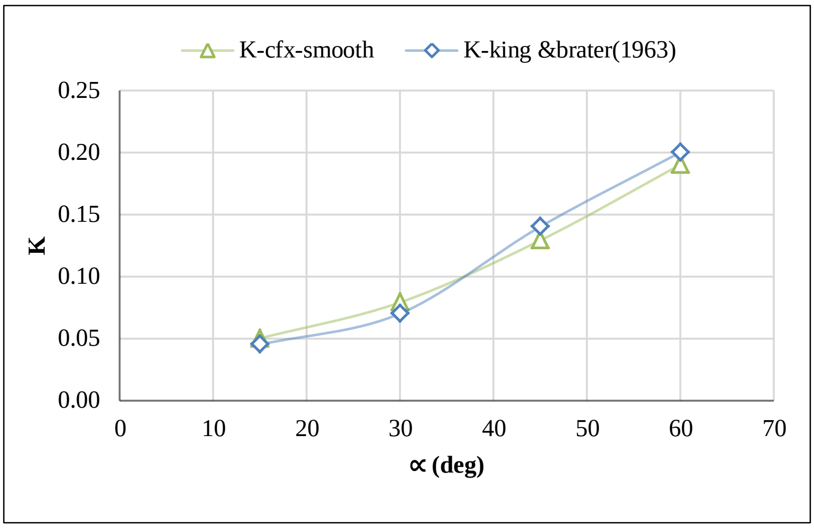

Figure 4. The verification of extracted data from ANSYS CFX with experimental results of Brater for diameter ratio of 1.5

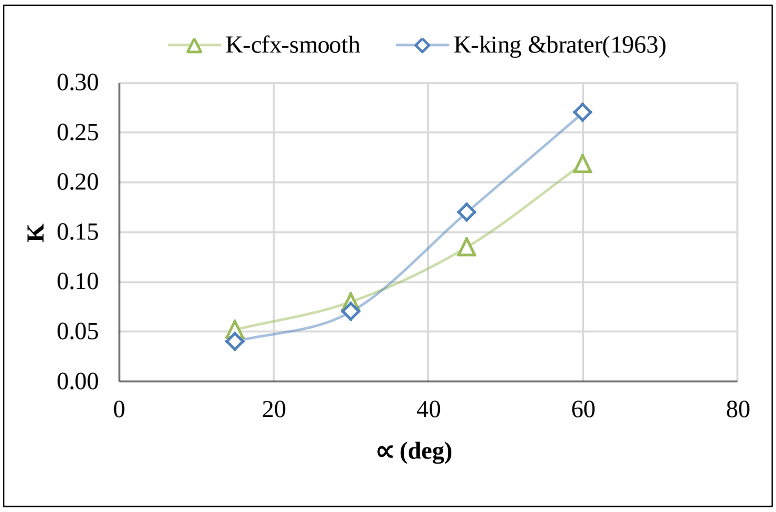

Figure 5. The verification of extracted data from ANSYS CFX with experimental results of Brater for diameter ratio of 2.1

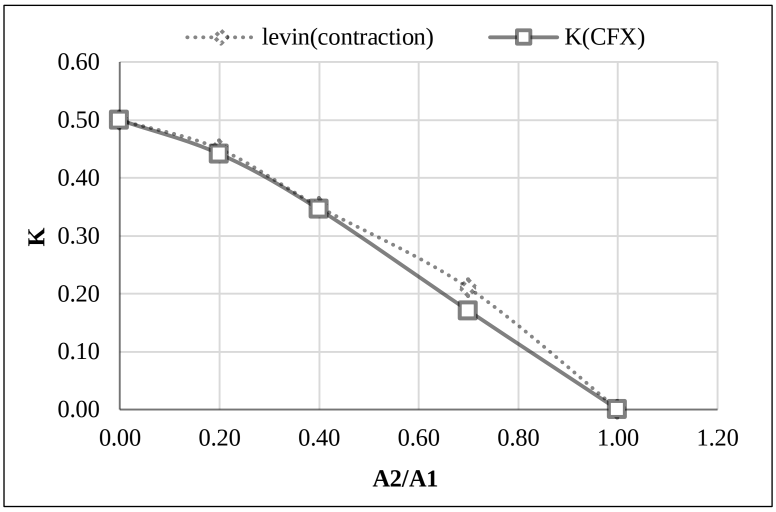

Figure 6. The verification of extracted data from ANSYS CFX with experimental results of Brater for sudden contraction pipe

In Figures 3 to 5. (3-diameters of 1.2, 1.5 and 2.1) the obtained energy loss coefficients via CFX numerical analysis and the experimental results for diameter ratios of 1.2, 1.5 and 2.1 for gradual pipe contraction have been extracted and compared with each other. In Figure 6. the obtained coefficients via CFX numerical method and experimental results of Levin for sudden contraction and in Figure 6. the experimental results of King Brater for gradual pipe contraction have been compared to each other. These diagrams for three diameter ratios of 1.2, 1.5 and 2.1 and five transition apex angles of 15, 30, 45, 60 and 90 degrees, without considering the effect of wall roughness, have been extracted. The average obtained error in calculation of energy loss coefficient in gradual pipe contraction has been obtained 7 percent for the diameter ratio of 1.2, 3 percent for the diameter ratio of 1.5, and 1 percent for the diameter ratio of 2.1. Furthermore, the average relative error for sudden contraction pipe has been obtained 5 percent.

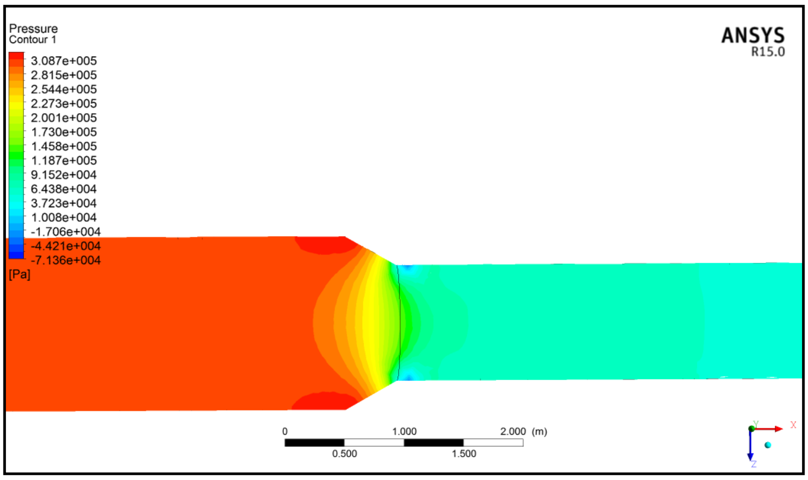

Figure 7. Pressure counter, gradual pipe contraction for diameter ratio of 1.5 and Reynolds Number of 15×10^6

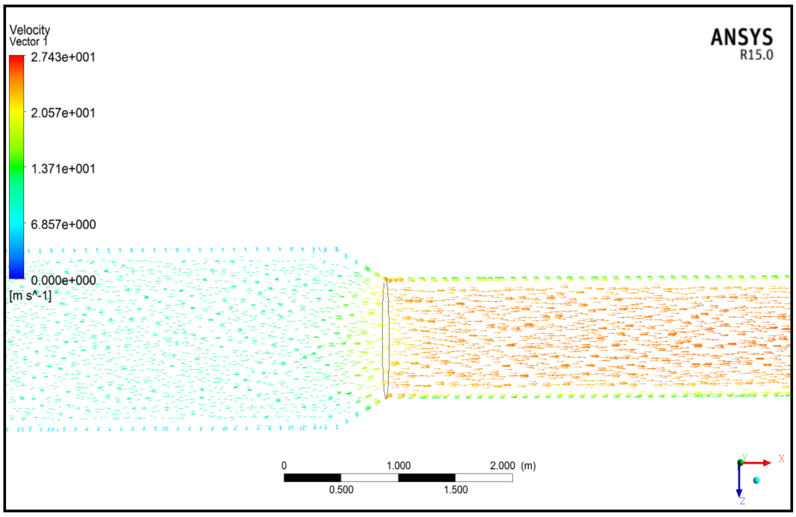

Figure 8. Vector velocity counter, gradual pipe contraction for diameter ratio of 1.5 and Reynolds Number of 15×10^6

4.1. Independence Analysis from Mesh

In order to investigate the independence of energy loss coefficient from mesh, a sudden contraction with 2.1 diameter ratio and smooth wall was investigated and in each step the obtained average relative error of energy loss coefficient from CFX numerical and experimental methods were determined and according to Figure 9, percent error diagram versus the number of elements was drawn.

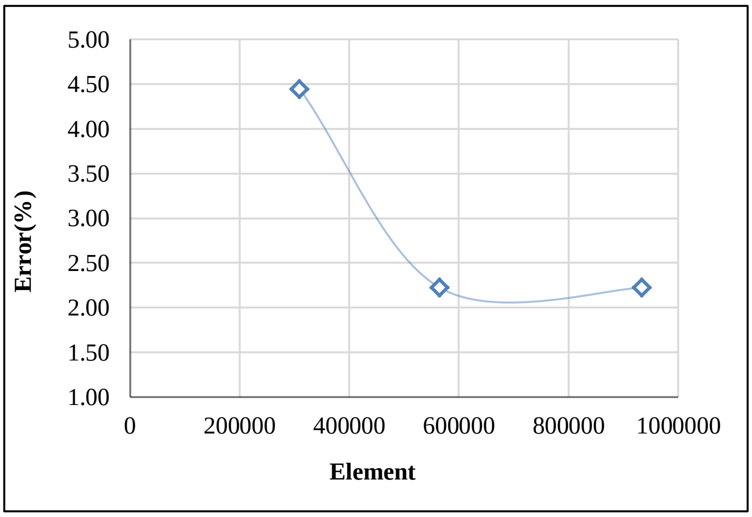

Figure 9. Independence analysis from mesh in sudden convergent contraction with 2.1 diameter ratio

According to Figure 9, in 310147 numbers of elements the obtained error is 4.44 per cent, in 565604 numbers of elements the obtained error is 2.22 and in 934213 numbers of elements the percent error is 2.22. In two final subsequent elements the relative per cent of errors is constant and is equal to 2.22 per cent. Therefore, the number of elements is considered 565664. It is seen that the smaller the mesh, the percentage of errors remains constant.

5. The Effect of Euler Number, Reynolds Number, and Relative Roughness on Energy Loss Coefficient

With regard to the conducted dimensional analysis in pipe contraction, the energy coefficient, in addition to the transition geometry, can be dependent on velocity, pressure, and the wall roughness. The effect of each of these variables in the form of the dimensionless parameters of upstream pipe Euler number (Eul_1), upstream Reynolds number (Rn_1), and upstream relative roughness (ε/d_1) are investigated in the following section. Euler number is expressed through the ratio of pressure force to inertia force [16]:

$$Eul_1 = \frac{\Delta p}{\rho V_1^2} \tag{6}$$

In which Equation 6. the Δp is the pressure changes in both sides of the transition, V_1 is the velocity in upper pipe, and ρ is the volume mass of the fluid.

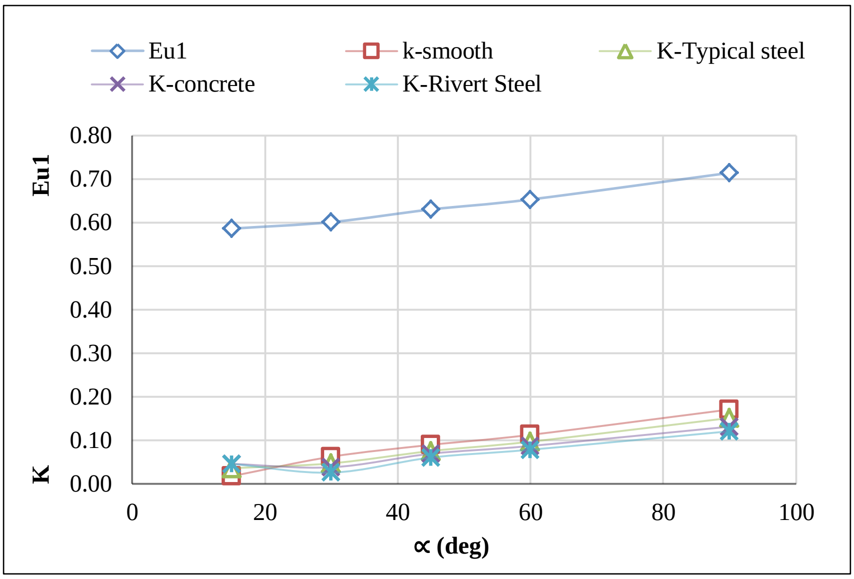

Figure 10. The effect of Euler number on energy loss coefficient for three diameter ratio of 1.2

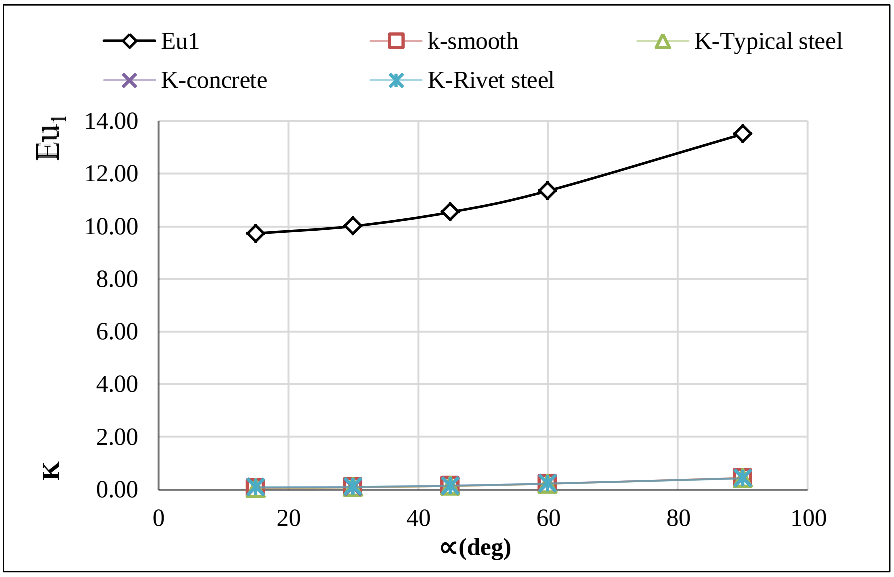

Figure 11. The effect of Euler number on energy loss coefficient for three diameter ratio of 2.1

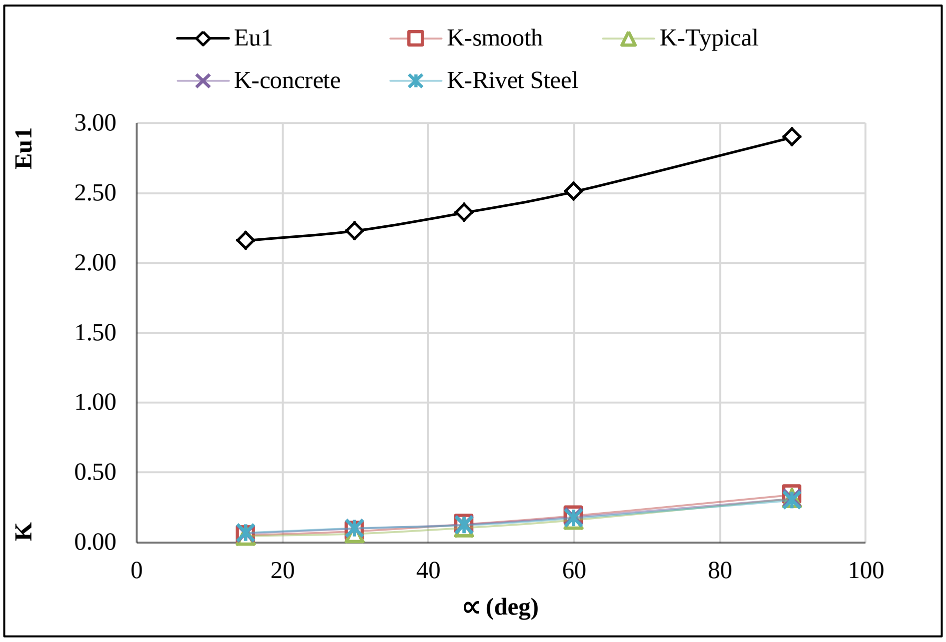

Figure 12. The effect of Euler number on energy loss coefficient for three diameter ratio of 1.5

Asbun Reynolds (1842-1912), to determine the laminar, transition, and turbulent flow regime, has suggested the following number [16]:

$$Rn_1 = \frac{\rho V_1 d_1}{\mu} \tag{7}$$

Equation 7. is known as Reynolds number. If the Reynolds Number is less than 2000, the flow is laminar, if this number is between 2000 and 4000 the flow is transition zone, and if the number is greater than 4000 the flow is turbulent. In this equation V_1 is the fluid velocity in inlet pipe, d_1 is the diameter of inlet pipe, and μ is dynamic viscosity of fluid. Furthermore, the relative roughness is defined as ε/d_1 in which ε is the wall roughness of the pipe. In Figures 13 to 15. in gradual pipe contraction, increasing of Reynolds Number causes the reduction of energy loss coefficient. This reduction is gradual and tends to some extent.

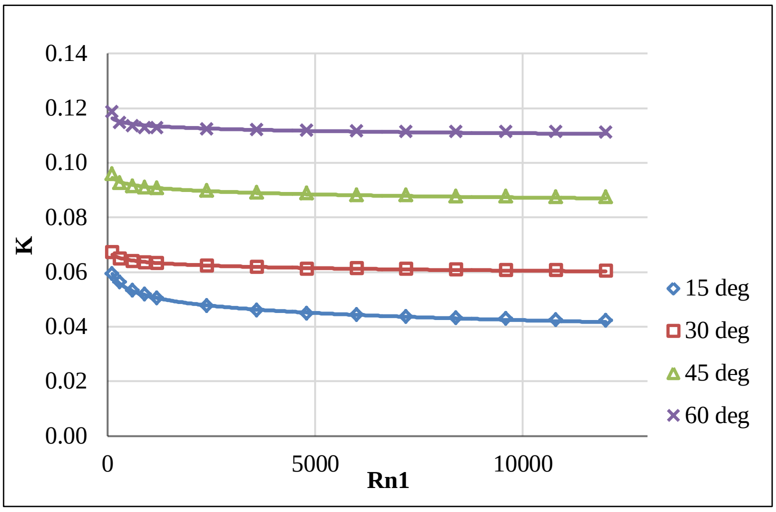

Figure 13. The effect of Reynolds Number on energy loss coefficient for diameter ratio of 1.2

In Figures 10 to 12. the effect of Euler number on energy loss coefficient on pipe contraction under pressure with three diameter ratio for four states of plastic pipe (smooth), typical steel (average roughness of 0.06 mm), concrete (average roughness of 1.56 mm), and rivet steel (average roughness of 4.59 mm) have been shown.

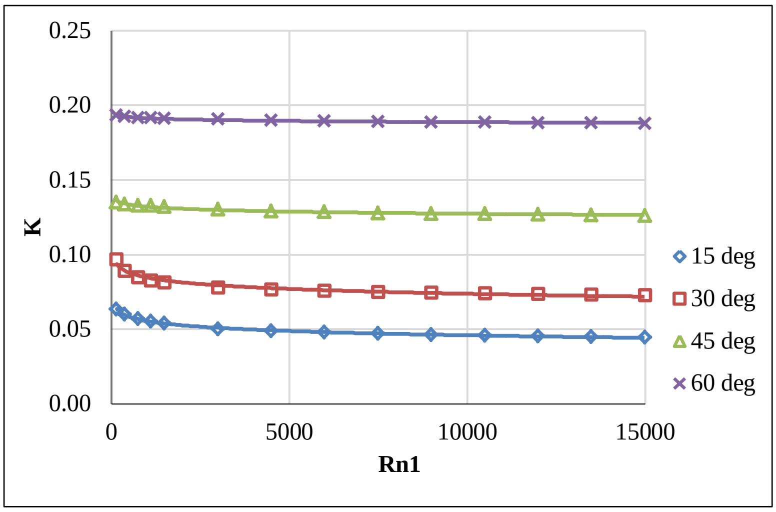

Figure 14. The effect of Reynolds Number on energy loss coefficient for diameter ratio of 1.5

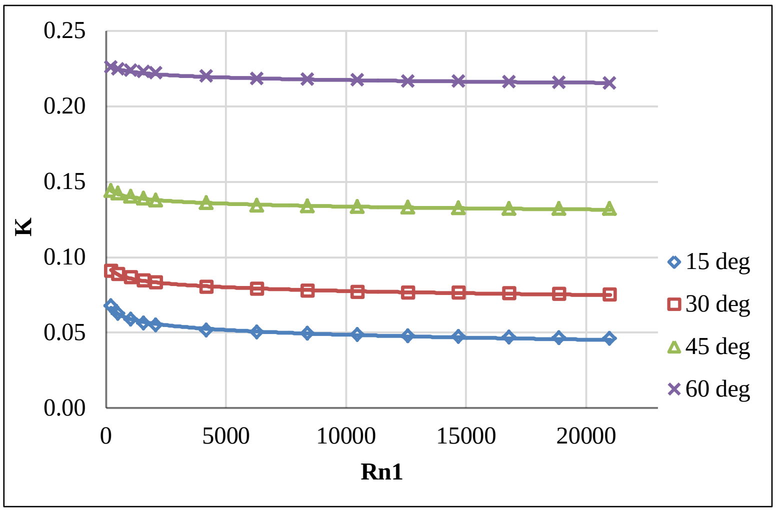

Figure 15. The effect of Reynolds Number on energy loss coefficient for diameter ratio of 2.1

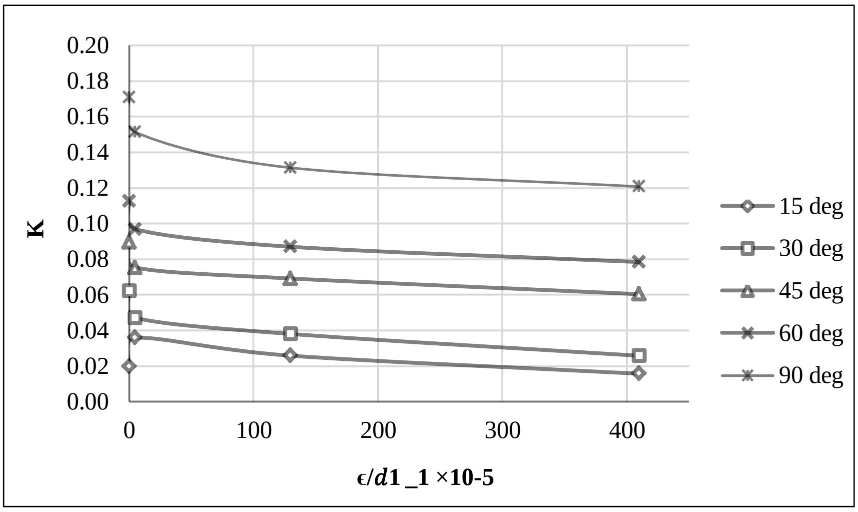

Figure 16. The effect of relative roughness on energy loss coefficient for diameter ratio of 1.2

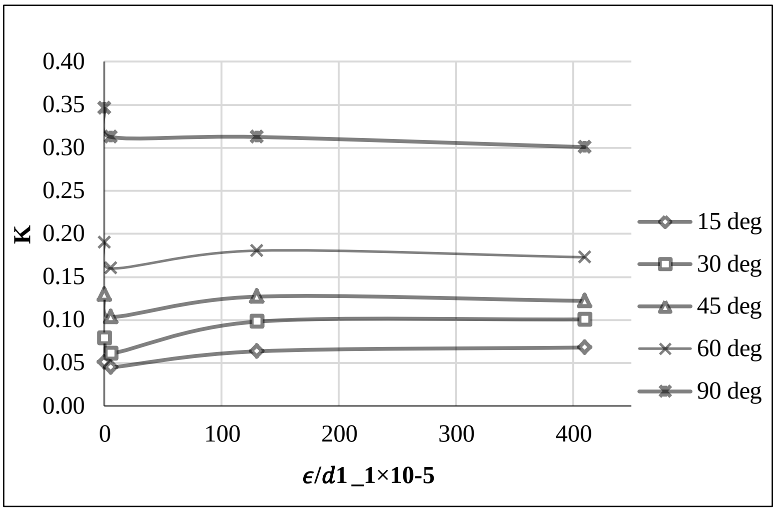

Figure 17. The effect of relative roughness on energy loss coefficient for diameter ratio of 1.5

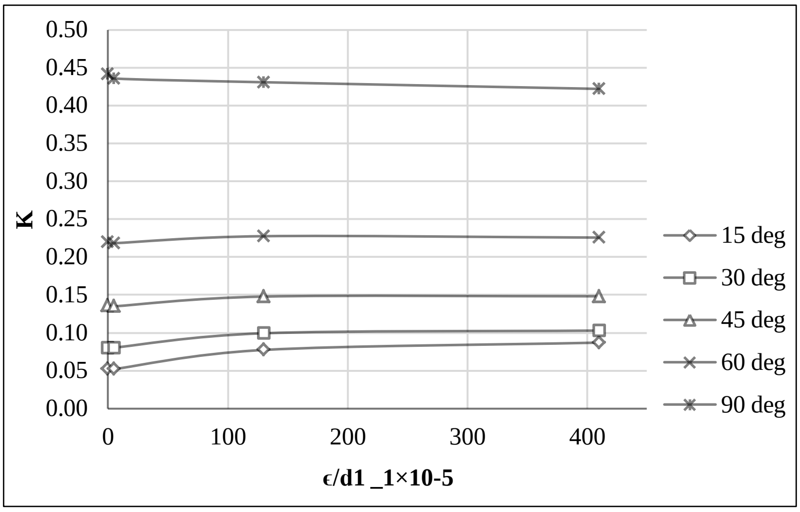

Figure 18. The effect of relative roughness on energy loss coefficient for diameter ratio of 2.1

For each apex angle and specified Euler number, with increasing of the wall roughness, the energy loss coefficient increases. According to the Figure 16. in diameter ratio of 1.2 for the less relative roughness and equal to 0.00005, the reduction of loss coefficient is sudden, and after this amount, the reduction is gradual, but this dependence has never been omitted. Increasing of relative roughness causes the reduction of energy loss coefficient, because by increasing roughness, the separation flow is delayed, and the amount of turbulence and subsequently the amount of loss coefficient is reduced.

6. Conclusion

In this study, gradual and sudden pipe contraction for three diameter ratios of 1.2, 1.5, and 2.1, and five apex angle transitions of 15, 30, 45, 60, and 90 degrees (sudden contraction) for four different pipe wall roughness using ANSYS CFX software have been modeled and the effect of the dimensionless parameters of Euler number, Reynolds Number and relative roughness on energy loss coefficient have been investigated and at the end the following results have been obtained:

- In gradual pipe contraction, the increasing of Reynolds Number causes reduction of energy loss coefficient. This reduction is gradual but never disappears. Moreover, with gradual apex angle transition, the loss coefficient reduces.
- In sudden pipe contraction, with diameter ratio of 1.2, for greater area ratio and equal to 0.17 and with diameter ratio of 1.5 and greater area ratio and equal to 0.53, the increasing of Reynolds Number has not any effect on energy loss coefficient. (The changes are gradual in a way that can be ignored).
- To determine the energy loss coefficient in pipe contraction in the form of a function of inlet Reynolds Number (Rn_1), diameter ratio (β), and transition apex angle (θ) for the range of $12 \times 10^3 \leq Rn_1 \leq 21000 \times 10^3$, $1.2 \leq \beta \leq 2.1$ and $0.26\, rad(15^o) \leq \theta \leq 1.57\, rad(90^o)$ the following equation is used:

  $$K = -0.176 + 0.094\beta - 4.4 \times 10^{-7} Rn_1 + 0.0036\theta$$

- In general, the increasing of Euler number causes the linear increase of energy loss coefficient in transitions. This result was observed for the transition apex angle greater than 30 in diameter ratio of 1.2.
- In 15-degree gradual pipe contraction for diameter ratio of 1.2 in 0.95 Euler number, diameter ratio of 1.5 in 2.61 Euler number, and diameter ratio of 2.1 in 9.7 Euler number, the increasing of roughness in the wall causes the increasing of energy loss coefficient.
- To determine the energy loss coefficient in pipe contraction in the form of a function of inlet Euler number (Eu_1), diameter ratio (β), and transition apex angle (θ) for the range of $0.6 \leq Eu_1 \leq 13.51$, $0.26\, rad(15^o) \leq \theta \leq 1.57\, rad(90^o)$ and $1.2 \leq \beta \leq 2.1$ the following equation is used:

  $$K = -0.0158 - 0.032\beta + 0.01 Eu_1 + 0.0034\theta$$

- To determine the energy loss coefficient in convergent pipe contraction in the form of a function of relative roughness ($\boldsymbol{\varepsilon}/d_1$), diameter ratio ($\beta$), and transition apex angle (θ) for the range of $0 \leq \boldsymbol{\varepsilon}/d_1 \leq 410$ and $0.26\, rad(15^o) \leq \theta \leq 1.57\, rad(90^o)$ the following equation can be used:

  $$K = -0.204 + 0.116\beta - 2.47(\varepsilon/d_1) \times 10^{-5} + 0.0033\theta$$

- The increasing of relative roughness causes the reduction of energy loss coefficient. Because, by roughness increasing, the flow separation is delayed and the amount of turbulence and, subsequently, the amount of loss coefficient decreases.
- In addition, instead of using the above equation, to determine the energy loss coefficient in the form of a function of diameter ratio (β), transition apex angle (θ), upstream Reynolds Number (Rn_1), upstream Euler number Eu_1), and upstream pipe relative roughness ($\boldsymbol{\varepsilon}/d_1$) in the mentioned ranges the overall following equation can be used. In this equation for $Rn_1 \geq 2000$ which the flow regime becomes turbulent in the pipe, the effect of Reynolds Number can be ignored in the calculation of energy loss coefficient:

  $$K = -0.083 + 0.028\beta + 0.003\theta - (2.3 \times 10^{-10}) Rn_1 + 0.007 Eu_1 - 3.5(\varepsilon/d_1)$$

7. References

[1] Weisbach J. “Die Experimental Hydraulic”, Freiberg, Germany: Engelhard. (1855).

[2] Liaqat Ali Hussain. ”The Experimental and Theatrical Analysis of Pipe Contraction Flow Fields", MPhil. Thesis, Kingston Polytechnic”. (1990)Udwadia, F.E. and Trifunac, M.D.

[3] Kays, W. M. “Loss Coefficients for Abrupt Change in Flow Cross Section with Low Reynolds Number Flow in Single and Multiple Tube System”. Technical Report No. 9. No. NP-3901; U-9356. Stanford Univ. (1950).

[4] Gianni Astarita and Guido Greco” Excess Pressure Drop in Laminar Flow through Sudden Contraction. Newtonian Liquids” Industrial & Engineering Chemistry Fundamentals 7 (1), 27-31. (1968).

[5] Gerami-Tajabadi, H. “The determination of pressure loss coefficients for an incompressible turbulent flow through pipe contractions”, MPhil. Thesis, Kingston Polytechnic. (1985).

[6] Prions, P. & Goulash, A. "Flow characteristics in a pipe with a gradual contraction. Journal of Hydraulic Research", 31(5), 587-600. (1993).

[7] E. S. D. U. "Pressure losses in f low through a sudden contraction of duct area", Data Sheet 78007. (1977).

[8] Durst, F. and T. Loy. "Investigations of laminar flow in a pipe with sudden contraction of cross sectional area. Computers & fluids" 13.1: 15-36. (1985).

[9] Hooper, W. B. "Calculate head loss caused by change in pipe size". Chemical Engineering, 95(16), 89. (1988).

[10] Benedict, R. P. and J. S. Wyler. "A generalized discharge coefficient for differential pressure type fluid meters". Journal of Engineering for Power 96.4 (1974): 440-448.

[11] Streeter, Victor. "Fluids Mechanics", Mac Graw-Hill Inc. 4th ed. (1966).

[12] John F. Newton "Pressure Loss in PVC Pipe Fittings Caused by Connection Gaps and Glue Beads" A thesis Master of Science in Civil and Environmental Engineering, Utah State University), (2005).

[13] ANSYS CFX-Solver Modeling Guide, Release 15.0, November. (2013).

[14] King, H. W. C. Wisler. And Woodburn, J.G. "Hydraulics 5th" New York, pp 206-209. (1948).

[15] Zarrati, Amirreza "Fluid mechanic and hydraulic laboratory" 2th Ed, (2012).

[16] Streeter, Victor. "Fluids Mechanics ,Mac Graw-Hill Inc. 4th Ed, (1966).

[17] Kourosh, Nosrati "Numerical Analysis of Energy Loss Coefficient in Pipe Contraction Using ANSYS CFX Software" Master thesis, Department of Civil Engineering and Environmental Amirkabir university of Tehran (Tehran Polytechnic), (2016).

[18]Satish, G. et al. "Comparison of flow analysis of a sudden and gradual change of pipe diameter using fluent software. Int. J. Res. Eng. Techno 2.12: 41-45. (2013)

[19] Zhao, H.Y. Jia, X.S. Yang, S.M. et al. Numerical Simulation of Separation Flow Filed of Sudden Expansion Tube. Scientific Technology and Engineering, 9, 5238-5240. (2009).

[20] Zhou, Z.D. Wei, C.Z. Sun, M.Y. et al. Numerical Simulation of Flow Form of Sudden Expansion Tube. Scientific Technology and Engineering, 12, 7983-7985. (2012).

[21] Li, Yinpeng, Changjin Wang, and Mingda Ha. "Experimental Determination of Local Resistance Coefficient of Sudden Expansion Tube. Energy and Power Engineering 7.04: 154. (2015).