Effect of Pipe Roughness on Pressure Losses of Newtonian Fluids in Concentric Annulus

Mehmet Sorgun[1*], Deniz Gülşen Haşıloğlu[2]

[1]Department of Civil Engineering, Izmir Katip Celebi University, Izmir, Turkey,
mehmetsorgun@gmail.com
[2]Department of Civil Engineering, Ataturk University, Erzurum, Turkey,
civil_engineer_deniz@hotmail.com
* Corresponding author / İletişimden sorumlu yazar

Abstract

Pressure loss is one of the important parameters in hydraulic engineering. Especially for the large Reynolds Number, relative roughness has a significant effect on pressure losses in annulus. In this study, water flowing through rough and smooth concentric annulus for turbulent flow is modeled using Computational Fluid Dynamics (CFD). CFD software based on the finite element method predictions of pressure losses are compared with experimental data obtained from the literature. The model predicts measured frictional pressure gradient with an error less than ±15% in all cases. Furthermore, pipe roughness influences on frictional pressure losses of water are also examined for various pipe roughness values, inner and outer diameters and Reynolds Number. The analysis results revealed that as the diameter ratio increases, the influence of roughness on pressure gradient increases.

Keywords- Concentric annulus, CFD, roughness, pressure loss, water

1 Introduction

The estimation of frictional pressure losses inside the annulus is one of the main issues for numerous engineering branches such as civil, petroleum and chemical engineering. Many researchers proposed Darcy friction factor correlations for smooth and rough pipes [1-5]. Blasius [1] developed the following user friendly friction factor equation for smooth pipe and Reynolds Number < 10^5:

$$f = 0.316 \ \text{Re}^{-0.25} \tag{1}$$

Nikuradse [2] conducted one of the pioneering studies of turbulent flow in rough pipes. Prandtl [3] proposed an accurate friction factor equation for smooth pipes:

$$\frac{1}{\sqrt{f}} = -2.0 \ \log\left(\text{Re}\sqrt{f}\right) - 0.8 \tag{2}$$

The most widely used friction factor correlation in order to calculate friction factor for turbulent flow in rough pipes are given by Colebrook [4]. Colebrook equation is

$$\frac{1}{\sqrt{f}} = -2.0 \ \log\left(\frac{k_s}{3.7D} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \tag{3}$$

where f is the friction factor, k_s is the average roughness height, D is the pipe diameter and Re is the Reynolds Number. However, since it is an implicit equation, a number of iterative solutions of this equation are available in the literature [6-10]. Moody [6] introduced the oldest approximation of Colebrook's equation.

A simple and accurate equation is proposed by Haaland [7]):

$$\frac{1}{\sqrt{f}} = -1.8 \ \log\left[\left(\frac{k_s}{3.7 \ D}\right)^{1.11} + \frac{6.9}{\text{Re}}\right] \tag{4}$$

Manadilli [8] introduced an approximate equation as follows:

$$\frac{1}{\sqrt{f}} \approx -2 \ \log_{10}\left[\frac{k_s}{3.7 \ D} + \frac{95}{\text{Re}^{0.983}} - \frac{96.82}{\text{Re}}\right] \tag{5}$$

A simple pressure gradient equation as a function of flow rate for the concentric water flow is developed by Kelessidis et al. [11] as

$$\frac{\Delta P}{\Delta L} = 1.508 * 10^7 \, Q^{1.773} \tag{6}$$

where $\Delta P/\Delta L$ is the pressure gradient (pa/m) and Q is the flow rate (m^3/s).

In this study, Newtonian fluid flowing through rough and smooth concentric annulus is simulated using Computational Fluid Dynamics (CFD) based on the finite element method.

2 CFD Modeling

To predict frictional pressure losses of Newtonian fluids in rough concentric annulus, Computational Fluid Dynamics are undertaken. In this study, ANSYS Workbench 12.0 [12], a commercial CFD code based on the finite element method is used. Four different concentric annular geometries (Table 1) are created and then it should be appropriately meshed to generate the computational grids. A well meshed geometry can improve considerably computation speed and accuracy of calculation. Tetrahedral mesh number in geometry was increased until it did not affect pressure loss results in annulus. As seen from Figure 1, up to a point, as the number of mesh increases, the frictional pressure losses decrease. However, after a certain mesh number, it did not change frictional pressure loss in annulus. For all of the cases, the concentric annulus is divided approximately to 2.9×10^6 tetrahedral meshes.

Table 1. Concentric annulus geometry used in the study

	Outer Diamater (m)	Inner Diamater (m)
Annulus# 1	0.07	0.04
Annulus# 2	0.074	0.046
Annulus# 3	0.1	0.053
Annulus# 4	0.089	0.068
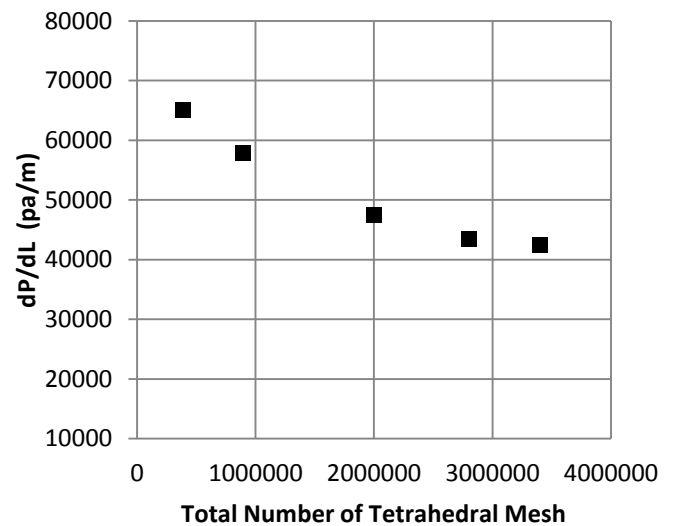
Figure 1. Pressure gradient predictions of CFD model for five different total number of tetrahedral mesh and Reynolds Number= 95600

After the meshed geometry is imported to CFD, the boundary conditions and initial values have to be described. The inlet was defined as an inlet velocity which depends on the average velocity at the inlet. The outlet was specified as atmospheric pressure and the flow was assumed to be steady, incompressible, isothermal and k-ε model used for turbulent flow. In k-ε model, the local turbulent viscosity is determined from the solution of the transport equations for the turbulent kinetic energy k and ε, the rate of dissipation of turbulent energy.

3 Experimental Work

Experiments were conducted on Middle East Technical University. Flow loop consists of a 3.66m long concentric horizontal annulus having 0.1- 0.053m and 0.074– 0.046m geometric configurations. Schematic drawing of concentric annulus is shown in Figure 2. Fluid is flowing through between two pipes. The data acquisition system records flow rates and frictional pressure losses of water during experiments. Water from the tank is pumped and passed through the concentric annulus using a centrifugal pump. The tests are performed with various average fluid velocities (0.98 m/s–2.3 m/s). During each experiment, differential pressure loss data obtained from digital analogue pressure transmitters at various water flow rates are recorded.
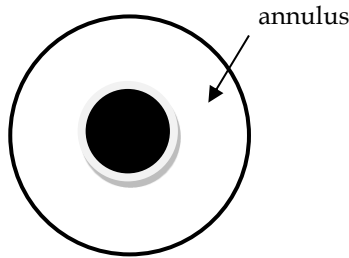

Figure 3. Predicted and measured pressure gradient of Kelessidis et al. data (Annulus # 1), k_s=0.024 mm

Figure 2. Schematic drawing of concentric annulus

4 Results and Discussion

First of all, the accuracy of the estimations of the CFD model is determined by comparing with experimental data that are available in the literature and conducted flow loop. Reynolds Number is calculated hydraulic diameter. For the rough concentric annulus, experimental data presented by Kelessidis et al. [11] are used. The frictional pressure gradients are calculated by using CFD model as well as Colebrook [4] and Fang et al. [13]. In Colebrook [4] and Fang et al. [13] equations, hydraulic diameter, D_h is used instead of D. Figure 2 demonstrates that comparison CFD model, Colebrook and Fang et al. equations with experimental results of Kelessidis et al. [11]. As seen from this figure, there are excellent matches to the estimations of Colebrook [4] equation with experimental data. In high fluid velocity, CFD model slightly overestimates the frictional pressure gradients, whereas Fang et al. [13] equation underestimates the pressure gradient. However, CFD model generally has satisfactory performance for rough concentric annulus. Model predictions and experimental measurements for smooth concentric annulus are shown in Figures 3. As seen from figure, CFD model estimates very well the frictional pressure losses of Newtonian fluids.
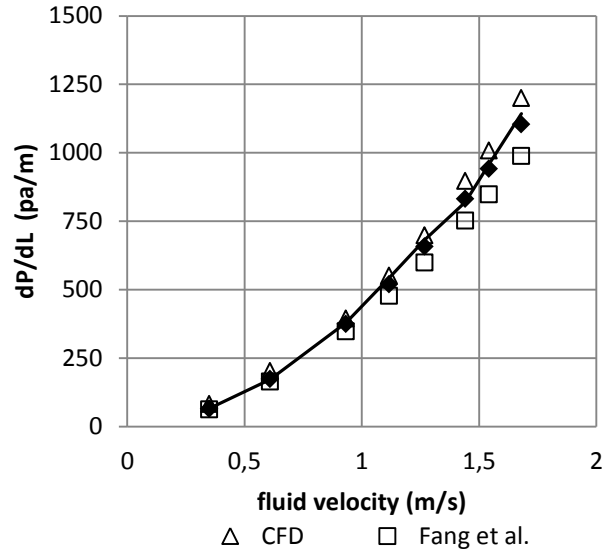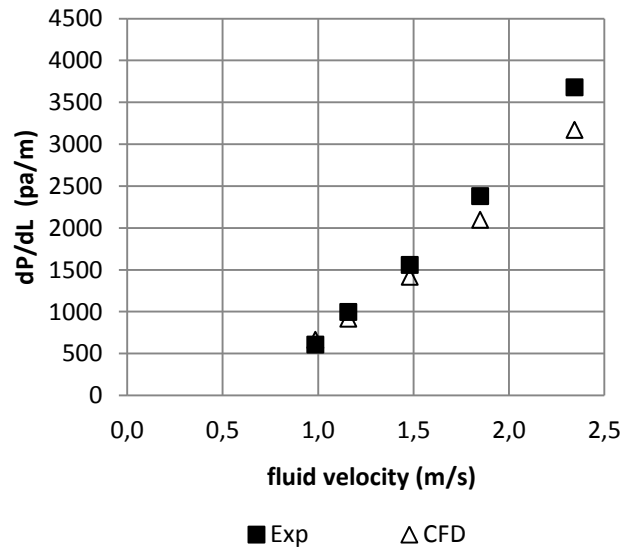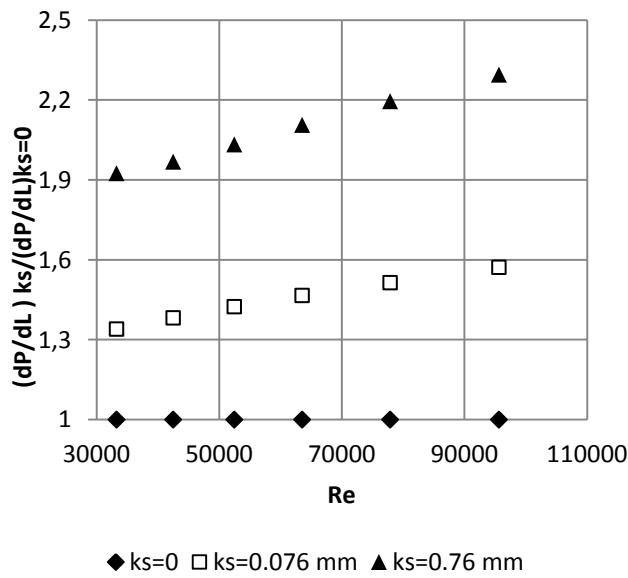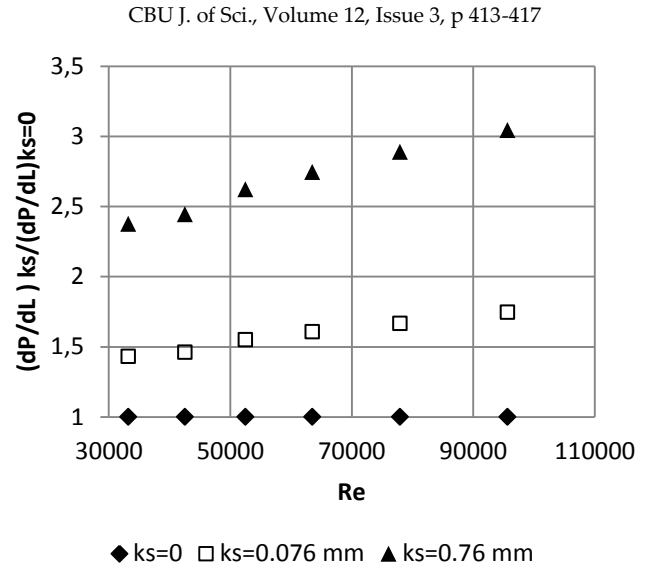

Figure 4. Predicted and measured pressure gradient (Annulus # 2)

After the CFD model is validated, it is used for investigating the effects pipe roughness on frictional pressure gradient inside the concentric annulus. Three different annular geometries a nd two different average roughness heights are to be considered for the investigation. Figure 5-7 show roughness effects on pressure gradient for annulus#1, annulus#2 and annulus#3, respectively. In these figures, κ is diameter

ratio (D_i/D_o). If these figures are examined, the followings can be concluded: One of them, as the Reynolds Number and average roughness height increases, roughness effects on pressure gradient increase. The other one, as the diameter ratio increases, the influence of roughness on pressure gradient becomes more severe. An increase in the diameter ratio practically decreases the free flow area of fluid. This leads to an increase on the average flow velocity of the fluid in annulus, which causes a significant increase in the influence of roughness on pressure gradient. For example, for the same Reynolds Number and 0.76 mm roughness height, pressure gradient increases up to 130%, 200% and 270% at for annulus#1, annulus#2 and annulus#3, respectively.

◆ ks=0　□ ks=0.076 mm　▲ ks=0.76 mm

Figure 6. Roughness effects of the frictional pressure gradient for various Reynolds Number (Annulus# 2) (κ=0.63)

◆ ks=0　□ ks=0.076 mm　▲ ks=0.76 mm

Figure 5. Roughness effects of the frictional pressure gradient for various Reynolds Number (Annulus# 1), (κ=0.57)

◆ ks=0　　□ ks=0.076 mm　▲ ks=0.76 mm
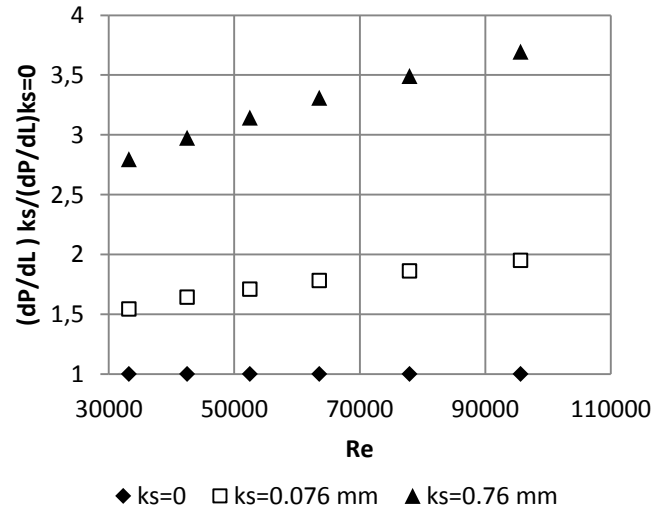
Figure 7. Roughness effects of the frictional pressure gradient for various Reynolds Number (Annulus#4) (κ=0.76)

5 Conclusions

In this study, Newtonian fluid flow inside rough and the smooth concentric annuli is simulated using CFD. Experimental data are presented for the pressure gradient of water in concentric annulus. The maximum error in predictions is the range of 15%

indicating that the CFD model successfully predicts frictional pressure gradients in rough and smooth concentric annulus. Results indicate that CFD code can be used to predict frictional pressure losses of rough concentric annulus in turbulent regime. For the constant Reynolds Number, simulation results show that as the diameter ratio increases, the influence of roughness on pressure gradient increases. Furthermore, turbulent flow regime drastically increases roughness effect on pressure losses of Newtonian fluid flowing through concentric annulus.

6 References

[1]	Blasius, H. Das Ähnlichkeitsgesetz bei Reibungsvorgängen in Flüssigkeiten. Forsch. Arb. Ing.,1913; 134.

[2]	Nikuradse, J. Laws of Turbulent Flow in Smooth Pipes. NASA, Report No. TTF-10, 1932.

[3]	Prandtl, L. Neuere ergebnisse der turbulenzforschung. VDI Z. (1857-1968), 1933; 77, 105-114.

[4]	Colebrook, C.F. Turbulent Flow in Pipes With Particular Reference to the Transition Region Between the Smooth and the Rough Pipe Laws, J. Inst. Civ. Eng. 1939; 11, 133–156.

[5]	McKeon, B. J., Zagarola, M. V., and Smits, A. J. A New Friction Factor Relationship for Fully Developed Pipe Flow. J. Fluid Mech. 2005; 538, 429–443.

[6]	Moody, L. F. An Approximate Formula for Pipe Friction Factors. Trans. ASME. 1947; 69, 1005–1006.

[7]	Haaland, S. E. Simple and Explicit Formulas for the Skin Friction in Turbulent Pipe Flow. ASME J. Fluids Eng. 1983; 105, 89–90.

[8]	Manadilli, G. Replace implicit equations with signomial functions. Chem. Eng. 1997; 104 (8), 129–130.

[9]	Chen, N. H. An Explicit Equation for Friction Factor in Pipe. Ind.Eng. Chem. Fundam. 1979; 18 (3), 296–297.

[10]	Avci, A., Karagoz, I. A novel explicit equation for friction factor in smooth and rough pipes. J. Fluid. Eng. ASME. 2009; 131 (6), 1–4 061203.

[11]	Kelessidis V.C., Dalamarinis P., Maglione R. Experimental Study and Predictions of Pressure Losses of Fluids Modeled As Herschel-Bulkley in Concentric and Eccentric Annuli in Laminar, Transitional and Turbulent Flows. Journal of Petrolem Science and Engineering. 2011; 77, 305-312.

[12]	ANSYS Version 12.0, 2009. ANSYS Workbench CFD. ANSYS Inc. Southpointe 275 Technology Drive Canonsburg, PA 15317, USA.

[13]	Fang, X., Xu Y., and Zhou Z. New Correlations of single-phase friction factor for turbulent pipe flow and evaulation of existing single-phase friction factor correlations. Nuclear Engineering and Design. 2011; 241, 897-902.