Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2015
(ICMERE2015) 26 – 29 November, 2015, Chittagong, Bangladesh

ICMERE2015-PI-143

NUMERICAL MODELING OF TURBULENT FLOW THROUGH BEND PIPES

Rana Roy Chowdhury[1,*], Md. Mahbubul Alam[2] and A.K.M Sadrul Islam[3]

[1-2]Department of Mechanical Engineering, CUET, Chittagong- 4349, Bangladesh
[3]Department of Mechanical and Chemical Engineering, IUT, Gazipur- 1704, Bangladesh
[1,*]engr.rrc@gmail.com, [2]mahbub87@yahoo.com, [3]sadrul05@gmail.com

Abstract- *A numerical investigation has been carried out for incompressible turbulent flow through 90^{0}bend pipes. The model is based on the numerical solution of conservation equations of mass and momentum. The results of the simulations of the flow in the form of contour and vector plots for two types of pipe bend both mitre and smooth bend with seven different Reynolds numbers (2.0+E04, 2.5+E04, 3.0+E04, 3.5+E04, 4.0+E04, 4.5+E04 and 5.0+ E04) are presented in this paper. From the obtained results, it is seen that the mitre bend pipes produces more turbulent kinetic energy, eddy viscosity and skin friction factor compared to that of the smooth bend pipes. To further investigation on smooth bend pipes, the static pressure distribution along the inner, outer wall and the pressure loss factor with different Re numbers are analyzed for different R/D ratios. However, it is noticed that as Re increases the pressure gradient changes rapidly at the inner and also the outer wall of the bend. Again, the total pressure loss factor (k) increases as the R/D ratio decreases and due to higher velocity heads, factor k decreases as Re increases. The numerical results are validated against existing experimental results.*

Keywords: Turbulent flow, Turbulent kinetic energy, Eddy viscosity, Skin friction factor, Pressure loss factor

1. INTRODUCTION

Various researches had been carried out to study the flow phenomena of bend pipes in experimentally and numerically. CFD technique is widely used to analyze the flow characteristics of a bend pipe. The researchers still work on this topic to know about secondary flow, the pressure, velocity variations along the inner and outer walls of the bend pipes. In addition to that, researchers want to conclude the effect of curvature ratio on flow phenomena and flow accelerated corrosion analysis for different engineering purposes.

Some of the major applications include oil and gas production field with their distribution networks, the energy conversion systems found in same design of nuclear reactor, heat exchangers, solar collectors, components of internal combustion engines (e.g. exhaust manifolds) and cooling of industrial machines and electronic components [2,7]. As Bangladesh is approaching to the era of nuclear power generation, in future this paper can be used for flow measurement and FAC (flow accelerated corrosion) rate analysis for the bend under high pressure and temperature where the difficulties arise on due to the restriction of reactor structure and wicked measuring environment [3,5].

When a fluid flows through a bend it causes the fluid particles to change their motion. The secondary flow generated from the curvature is superimposed on the primary flow. In addition to that, the strong action of centrifugal force produces higher wall pressure in the outer wall and reverse is occurred in the inner wall side of the bend [1]. Thus, the fluid experiences an adverse pressure gradient at bend and this disturbance of flow exists further downstream of the bend.

The aim of this paper is to compare the flow phenomena between mitre and smooth bend, then to analyse the effect of curvature ratio i.e. R/D ratio on the smooth bend with different Re numbers.

2. SOLUTION METHODOLOGY

The mass and momentum equation with the standard k-ε model has been adopted for the computation of turbulent flow through the bend pipes.

The SIMPLE algorithm which is based on a finite volume discretization of the governing equations as suggested by Patankar (1980) is used for numerical modeling [6].

2.1 Governing Equations

Governing equations for the present analysis respectively continuity, navier-stokes, turbulence kinetic energy and turbulence dissipation rate equation are given below [8].

$$\nabla.\vec{V} = 0 \tag{1}$$

$$\frac{\partial \vec{V}}{\partial t} + \vec{V}.\nabla\vec{V} = -\nabla p + \nabla.\left[(\upsilon + \upsilon_t)\nabla\vec{V}\right] \tag{2}$$

$$\frac{\partial k}{\partial t}+\vec{V}.\nabla k=\mathrm{P_k}-\varepsilon+\nabla.\left[(\upsilon+\upsilon_t/\sigma_k)\nabla k\right] \quad (3)$$

$$\frac{\partial \varepsilon}{\partial t}+\vec{V}.\nabla \varepsilon=C_{\varepsilon 1}\frac{\varepsilon}{k}P_k-C_{\varepsilon 2}\frac{\varepsilon^2}{k} + \nabla.\left[(\upsilon+\upsilon_t/\sigma_\varepsilon)\nabla \varepsilon\right] \quad (4)$$

Where, $\upsilon_t = C_\upsilon \frac{k^2}{\varepsilon}$, P_k represents the turbulence production and the empirical constants are:

$$C_\upsilon = 0.09,\ C_{\varepsilon 1}=1.44,\ C_{\varepsilon 2}=1.92,\ \sigma_k=1.0,\ \sigma_\varepsilon=1.3$$

2.2 Problem Specifications

Flow characteristics are analysed considering the incompressible fluid is water, density ρ=1000Kg/m^3, dynamic viscosity μ=0.008 Pa.s and the steady state turbulent flow through 90^0 bend pipe with 40 mm diameter along with upstream and downstream length of 120 mm.

Again, by varying the radius of curvature thus with different R/D ratios the flow characteristics are analysed with a view to see the R/D ratio effect on the smooth bend. Further, for validation purpose the geometry is adopted as specified in the paper [4].

3. RESULTS AND DISCUSSION

The velocity and pressure contour for both smooth and mitre bend with seven different Reynolds numbers (2.0+E04, 2.5+E04, 3.0+E04, 3.5+E04, 4.0+E04, 4.5+E04 and 5.0+ E04) are analyzed to see the flow phenomena for both smooth and mitre bend taking R/D=1.25.

Again, To put an emphasis on the smooth bend for six different R/D ratios (2.5, 3.0, 3.5, 4.0, 4.5, 5.0), the static pressure distribution along inner, outer wall and pressure loss factor with seven different Re numbers are analyzed.

3.1. Flow Characteristics for Smooth and Mitre Bend

Figure-1(a) and 1(b) shows the pressure contour for smooth and mitre bend at Re=2.0+E04 and Re=5.0+E04. Both figures show that higher pressure region are in the outer wall as the flow decelerates and lower pressure in the inner wall side as the flow accelerates throughout the bend. Similarly, the velocity vector for both bends as shown in Figure-2(a) and 2(b) indicates that the higher velocity appear in the inner wall of the bend then existed for the downstream stick to the outer pipe wall. Also, the lower velocity zone appear in the outer wall of the bend then at the downstream stick to the inner pipe wall from the separation point. Figure 3, 4 and 5 represent the graphical analysis for mitre and smooth bend flow characteristics such as Turbulent Kinetic energy, Eddy viscosity and Skin Friction Factor with seven different Reynolds numbers. From the graph, it is clear that the mitre bend produces more turbulent kinetic energy, eddy viscosity and skin friction factor and thus the mitre bend corroded more as compared to the smooth bend. It is worth mentioning that the above flow parameters increase as Re increases where skin friction factor changes rapidly than the others two parameters for the smooth bend.

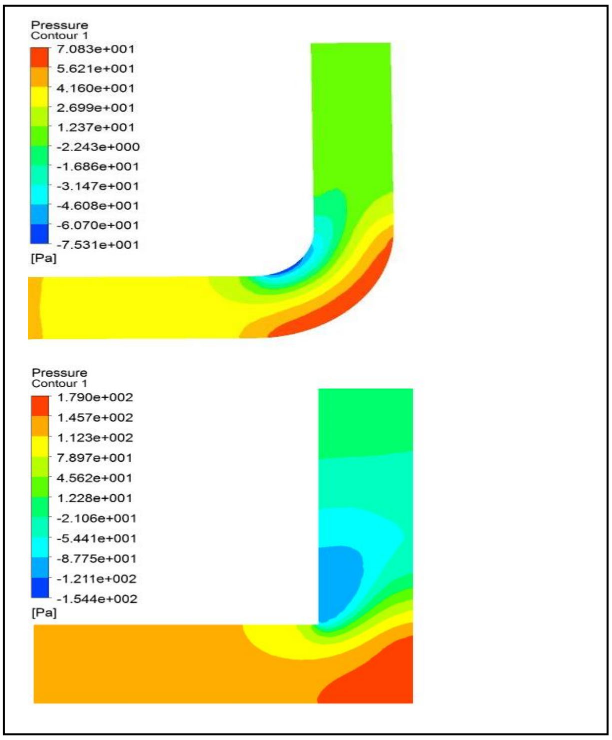

Fig.1(a): Pressure Contour for smooth and mitre bend at Re=2.0+E04

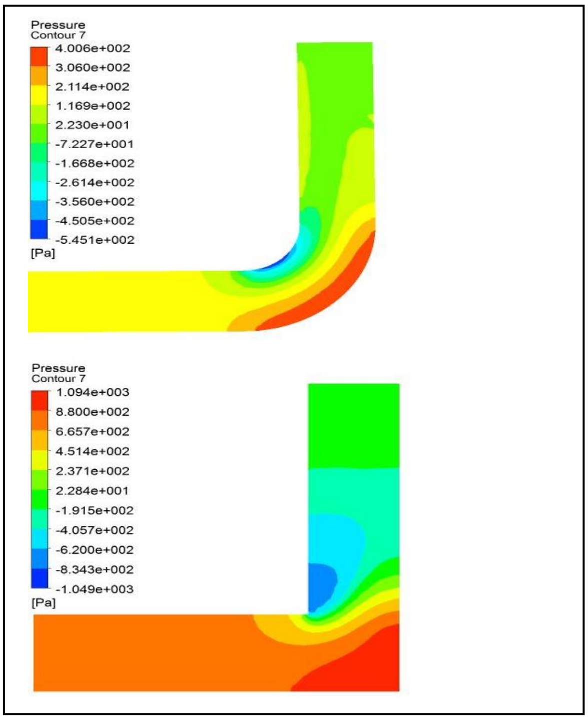

Fig. 1(b): Pressure Contour for smooth and mitre bend at Re=5.0+E04

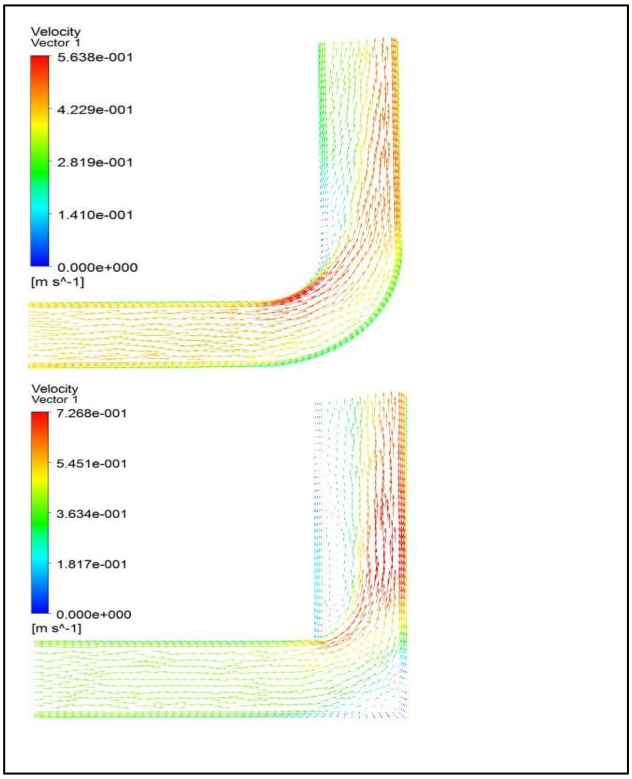

Fig. 2(a): Velocity Vector for smooth and mitre bend at Re=2.0+E04

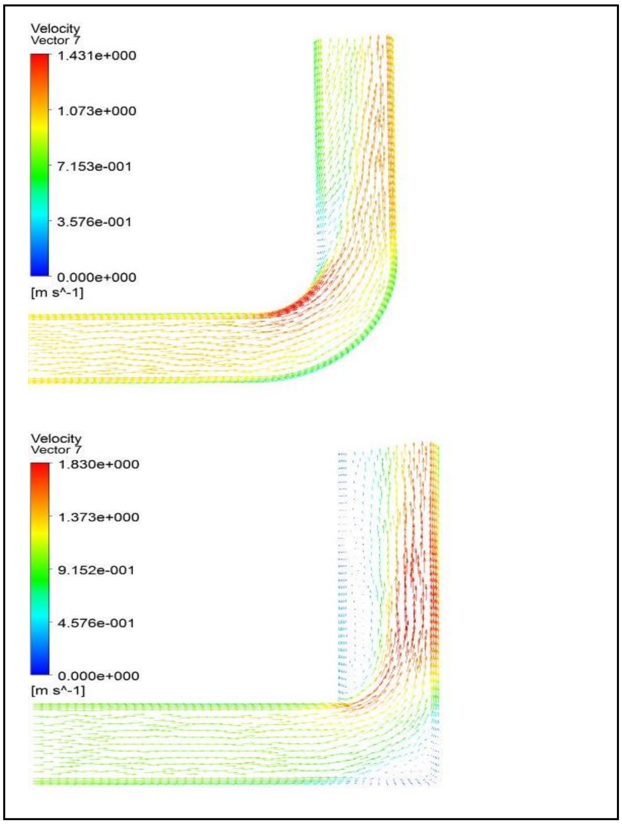

Fig. 2(b): Velocity Vector for smooth and mitre bend at Re=5.0+E04

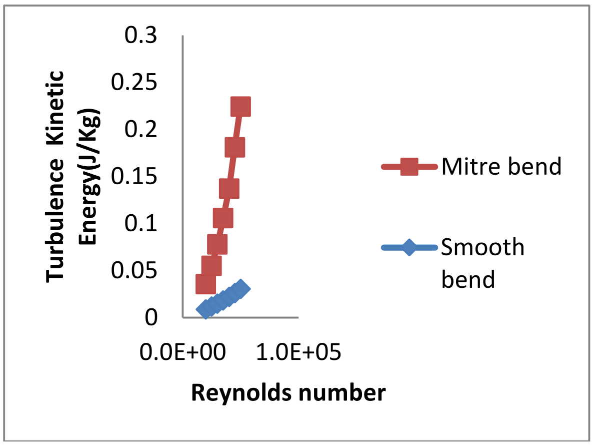

Fig.3: Turbulent K.E variation with different Reynolds number for smooth and mitre bend

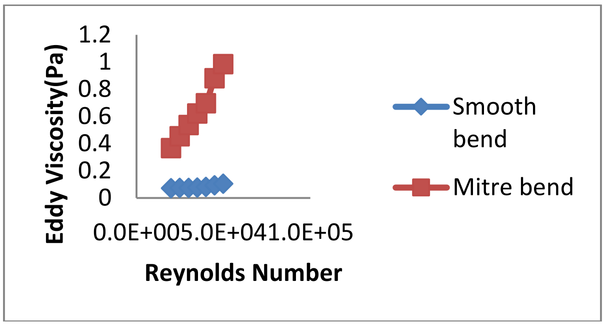

Fig.4: Eddy Viscosity variation with different Reynolds number for smooth and mitre bend

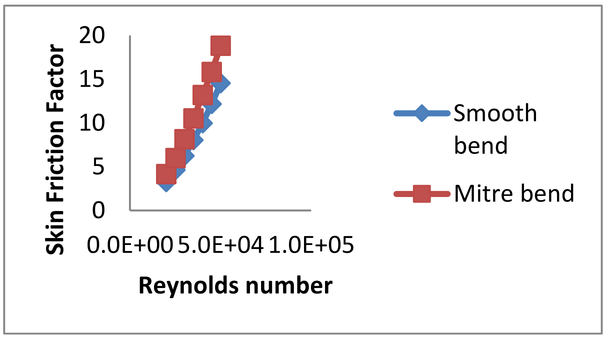

Fig.5: Skin friction factor variation with different Reynolds number for smooth and mitre bend

3.2 Focusing on R/D Ratios for Smooth Bend

The static pressure distributions along the inner and outer wall of the bend are represented in figure 6(a) to 6(d) respectively for R/D=2.5 and R/D=5.0. At the inner wall, the flow is accelerated into the bend where positive pressure gradient is seen at the vicinity of $\Theta=30^0$ location and then adverse pressure gradient appeared at further downstream. Again, reverse is occurred at the outer wall where the flow is decelerated. However, it is noticed that as Re increases the pressure gradient changes rapidly at inner and the outer wall of the bend. As R/D increases gradually, the two transition points of pressure gradient now appeared at $\Theta=15^0$ and 75^0 location at the inner wall.

The most important parameter for bend pipe known as pressure loss factor k is defined as $K=\Delta p/0.5\rho u^2_{avg}$, where Δp is the total pressure loss across a bend. Figure 7 represent the total pressure loss factor with R/D =2.5, 3.0, 3.5, 4.0, 4.5 and 5.0 at different Re numbers for the bend. It is known that the separation zone is large for small R/D and for large R/D the influence of friction is dominant [9]. It has been observed that the total pressure loss factor k increases as the R/D ratio decreases and due to higher velocity heads factor k decreases as Re increases as shown in figure 7. The numerical results are found in a good agreement with experimental results published by NASA [4] as shown in figure 8.

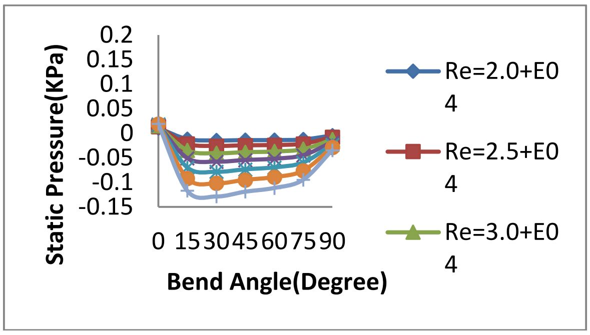

Fig. 6(a): Static Pressure variations along the inner walls of the smooth bend with different Re for R/D=2.5

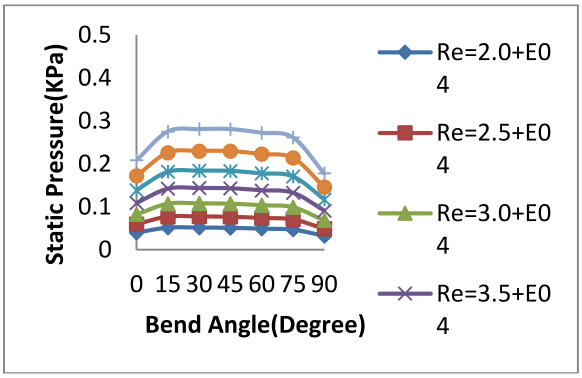

Fig. 6(b): Static Pressure variations along the outer walls of the smooth bend with different Re for R/D=2.5

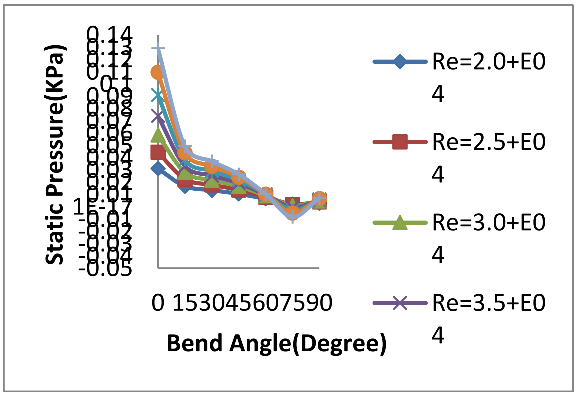

Fig. 6(c): Static Pressure variations along the inner walls of the smooth bend with different Re for R/D=5

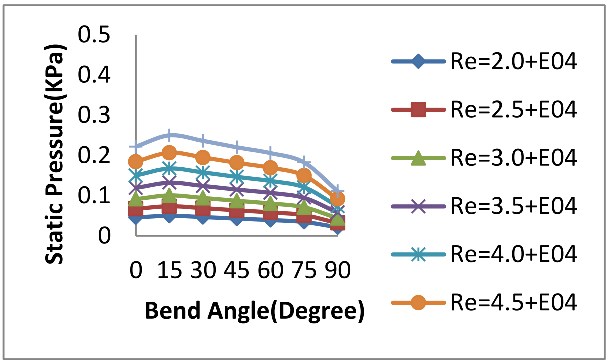

Fig. 6(d): Static Pressure variations along the outer walls of the smooth bend with different Re for R/D=5

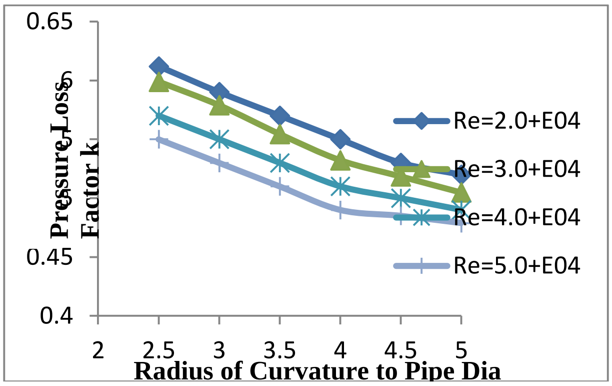

Fig.7: Variations of total pressure loss factor with R/D ratios for different Re

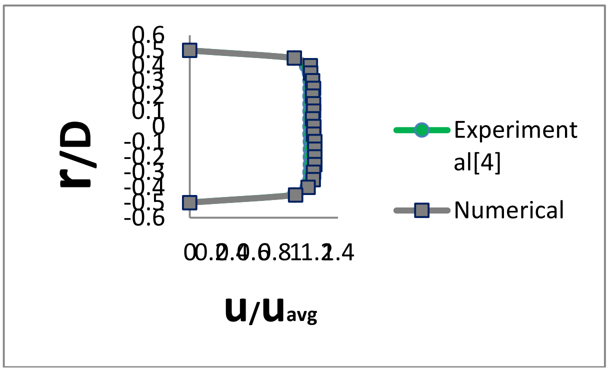

Fig.8: Comparison of Numerical and Experimental Results at x/D=-0.58

4. CONCLUSION

I. At the outer wall where the flow decelerates represent higher pressure and at the inner wall vice versa is occurred for both mitre and smooth bend.

II. The mitre bend produces more turbulent kinetic energy, eddy viscosity and skin friction factor and thus the mitre bend corroded more as compared to the smooth bend.

III. By increasing Re pressure gradient changes rapidly at the inner and outer wall with different R/D ratio for the smooth bend.

IV. The transition point/zone of pressure gradient at the smooth bend dependent on R/D ratio.

V. Total pressure loss factor k varies inversely with R/D ratios for the smooth bend and this conclusion specially refers for the present analysis. Further studies are necessary to give conclusion beyond this range of R/D.

5. REFERENCES

[1] Athanasia Kalpakli "Experimental study of turbulent flows through pipe bend" Technical Reports from Royal Institute of Technology, SE-100 44 Stockholm, Sweden.

[2] Philipp Schlatter, Azad Noorani, Athanasia Kalpakli1; & Ramis Örlü1; "Coherent Motions In Turbulent Flows Through Curved Pipes" 14th European Turbulence Conference, 1–4 September 2013, Lyon, France.

[3] C. Liqiang, H. Shouyin, and H. Chen, "Measuring methods for helium leakage rate of the primary coolant loop of HTR-10," Chinese Journal of Nuclear Science and Engineering, vol. 3, pp.234–239, 2007.

[4] M.M.Enayet, M.M.Gibson, "Laser Doppler measurement of laminar and turbulent flow in a pipe bend", NASA Contractor Report 3551, 1982.

[5] J. Barbe, F. Dijoux, C. Yardin, and T. Mac´e, "Measurement of helium micro flow for gas chromatography by the dilution method," Mapan: Journal of Metrology Society of India, vol. 27, no. 2, pp. 77–81, 2012.

[6] K Muralidhar, T Sundararajan, Computational fluid Flow and Heat Transfer, Narosa Publishing House.

[7] H. Benzenine, R. Saim, S. Abboudi and O. Imine "Numerical simulation of the dynamic turbulent flow field through a channel provided with baffles: comparative study between two models of baffles: transverse plane and trapezoidal" Revue des Energies Renouvelables Vol. 13 N°4 (2010) 639 – 651.

[8] J. M. McDonough "Introductory Lectures on Turbulence" Departments of Mechanical Engineering and Mathematics University of Kentucky.

[9] Losses in Flow, W.H. Hager, Wastewater Hydraulics, 2nd ed., Springer-Verlag Berlin Heidelberg, 2010.

6. NOMENCLATURE

Symbol	Meaning	Unit
$\vec{V}$	Velocity vector	(m/s)
p	Pressure	(Pa)
k	Turbulence kinetic energy	(J/Kg)
ε	Eddy viscosity	(Pa)
u_{avg}	Average velocity	(m/s)
ρ	Density	(Kg/m^3)
υ	Kinematic viscosity	(m^2/s)
R	Radius of curvature	(m)
D	Diameter	(m)
Re	Reynolds number	Dimensionless