The Eurasia Proceedings of Science,
Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 2018

Volume 2, Pages 323-333

ICRES 2018: International Conference on Research in Education and Science

Numerical Flow Analysis of The Variation of Central Axial Velocity Along The Pipe Inlet

Hasan DUZ
Batman University

Abstract: Due to no slip flow condition at the wall, the fluid enter the pipe with a smooth velocity start to develop along the flow to comply the zero velocity at the wall and maximum at the pipe center. After a certain distance where the development completed, the velocity profile becomes fully developed and no longer changes observed along the pipe flow. The region flow where the velocity profile developes is called developing flow or inlet flow and the region flow where the fully developed profile govern are called fully devloped flow. Computation of the flow properties in the fully developed region can be enabled with various empirical theories, but the complex flow styructure in pipe inlet region still has not been solved exactly. However It is quite important to know the flow behavior at the pipe inlet to compute the right pumping power especially in the fluid heating and cooling short pipe flow processes. the study performed, the steady pipe flows with Newtonian fluid were simulated numerically at low Reynolds numbers (ranged 1000 and 25000) covering the three flow regimes (laminer, transition and turbulence). High turbulence level and smooth velocity profile were assigned to the flow at pipe inlet. Turbulence flows were solved according to the time mean flow assumption. On the numerical results obtained, the variation of axial central velocity along the flow was examined for different relative roughnesses. Consequently, a numerical correlation which define the axial velocity and fit the numerical values well is proposed.

Keywords: Entrance length, Pipe flow, Developing flow

Introduction

Osborne Reynolds (1842-1912) has discovered laminar and turbulent flow behavior by injecting ink into glass-tubular flow in his experiments. At low flow velocities the ink followed a uniform flow path and not mixed to the flow along, while at high flow velocities the ink mixed with the flow over the entire cross section as in the move downstream. In the laminar flows, due to low flow velocity, fluid particles follows a smooth flow path, for this reason the laminer flows is smooth. Whereas in the turbulent flow, the instabilities in the flow cause the flow to get mixed in so that the fluid particles do not follow a uniform flow path. In general, all flows must be laminar, however some factors that degrade the flow stability, such as surface roughness, upstream turbulence, and heat transfer in the flow, force the flow to be turbulent. For this reason, if precise flow conditions are provided, all flows will remain laminar (Özışık, 1985). Turbulent flows include fluid clusters that are formed continuously near the wall and move during the flow while spin its around. These moving fluid clusters are called turbulent structures or swirl motions. These vortex structures form continuously near the wall, move, divide and disperse in the flow and eventually turn into sensible heat in the flow then disappears. They are responsible of the conversion of some of the mechanical energy into sensible heat energy. This describe why the energy loss is greater in turbulent flows. For example, especially for flows over a solid surface (flow through aircraft, turbine, and compressor blades) the flow being turbulent not only increases energy loss but also create vibration and noise in the flow. As a result, in addition to the viscous forces in the turbulent flow, there are also additional drag forces, so that the energy losses in turbulent flows are much higher than laminar flows of which viscous forces are dominated. For the flows over a flat surface, flow first begins as laminer in the leading edge and a certain distance away from the inlet the flow stability is deteriorated and it become a turbulent flow structure. Figure 1 illustrate three stages of the fluid flowing over the inside surface of a pipe as being laminer,

- Selection and peer-review under responsibility of the Organizing Committee of the Conference

transient and turbulent. The flow distance from the pipe inlet to the location where the flow first disturb to turbulent is called the transition length *(L_t)*. After transition length, a transitional flow region ocur for a while then the flow becomes fully developed. The measured distance from inlet to where the flow to become full turbulent is called entrance length. It is seen in many experimental works that the flow distances is depend on the flow velocity, surface roughness, free stream turbulence, surface vibrations, and heating and cooling processes (Minkowycz et al. (2009), Zanoun et al. (2009)). Though some empirical correlations are proposed for both flow distances through the experiments, a general solution to the problem is not still be clarified well due to many parameters effects on the flow.

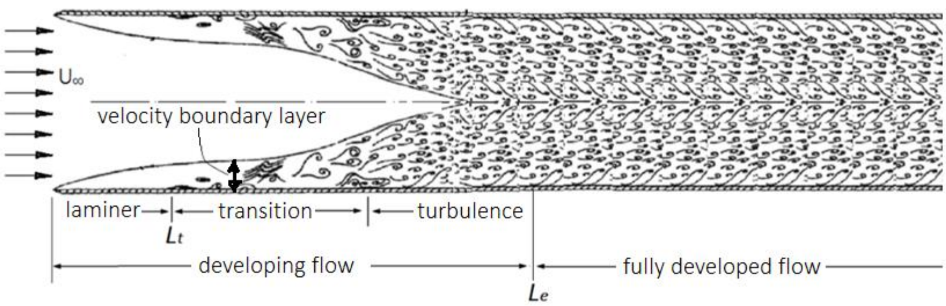


Figure 1. Developing and fully developed flow at pipe entrance

According to experimental studies, when a flow, contain high freestream turbulence level, pass over a full roughly surface, transition length lasted up at $Re_t = 10^5$ but for a flow not contained any tubulence in the freestream pass and over a smooth flat surface it lasted up to $Re_t = 10^6$ *as measured in the experiments* (Özışık (1985)). In case pipe inside flow study, due to pipe diameter limit the flow peripheral, transition distance from laminer to turbulence is being different than the flows over flat surfaces. Fig. 2 has shown the flow development after pipe inlet. The velocity boundary layer that forms as the result of the viscous effects from the pipe inlet, thickness of it increase along the inlet and since the thickness is limited by the pipe radius, the entire flow cross-section is filled with the boundary layer. From the pipe inlet, the viscous effects begin to change in the resulting velocity profile. This velocity profile changes along the flow until it become a constant velocity profile. The flow region where the velocity profile changes is called inlet flow or developing flow. The pipe flow, in which the velocity profile is along constant, is called the fully developed pipe flow. Different definitions are also available in the literature for fully developed pipe flow. For example, fully developed flow begins when such like two flow properties, wall shear stress or mean turbulent flow statistics reach the constant values (Anselmet et al. (2009), Patel&Head (1969)). Therefore, Zimmer et al. (2011) said that it should be required to define the fully developed flow as a flow that starts when the time-averaged turbulence flow statistics become constant. In the author's experimental study, it was reported that the developing flow distance is even longer when turbulence statistics measurements are based.

Along the fully developed pipe flow, the wall shear stress and the friction factor are constant since the pressure drop is linear. The fully developed laminar or turbulent pipe flows are largely solved with theoretical and empirical relations, while the developing flow portion has still not been fully solved. In engineering applications, pipe-tank connections generally become conical (bell mouth), square edged and reentrant. while a sharp edged inlet produce much turbulence in the flow, bell mouth inlet produces minimal turbulence. The amount of turbulence goes to pipe at the inlet is effect on the transition and entrance lengths (Tam et al. (2013), Augustine (1988)). It is evident that the transition and inlet lengths with high turbulent inlet are shorter than with the low turbulent inlet. Table 1 gives the entrance lengths reported in experimental studies of pipe flows.

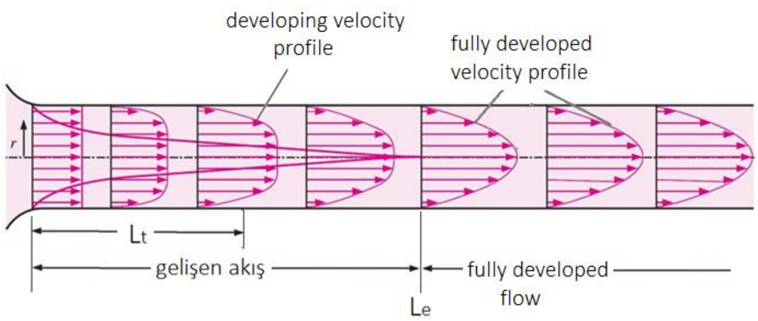


Figure 2. Variation of velocity profile along the developing and fully developed flow

Table 1. Dimensionless entrance lengths reported by the experimental studies

Dimensionless Entrance Length (L/D)			
Constant wall shear stress	**Mean turbulent statistics**	**Reynolds Number**	**Author**
80		-------------	Osborne Reynolds
$L_e/D = 2.09x10^{-8} * Re^{-1.66}$		5000-15000	Augustine (1988)
$L_e/D = 1.6\ Re^{1/4}$ $L_e/D = 4.4\ Re^{1/6}$		10^5 - 10^6	Anselmet et al. (2009)
A long Empirical formula		1,95 x 10^5	Salami (1986)
25 - 40		$3x10^3$ - $3x10^6$	Nikuradse (1966)
30		$5x10^4$ - $5x10^5$	Laufer (1954).
50 - 80		10^3 - 10^4	Patel & Head (1969)
70		$3x10^4$ - $1x10^5$	Zanoun et al. (2009)
--------	72	175000	Perry & Abell (1978)
50	80	$1x10^5$ - $2x10^5$	Doherty et al. (2007)
Not attain to 40		388000	Barbin&Jones (1963)
	70	$1.5x10^5$- $8.5x10^5$	Zimmer et al. (2011)

Numerical Study

Firstly, in order to gaining a validation to numerical solution, an experimental study has been carried out with four pipe types which is made of different materials. The selected pipe types, their relative roughness and pipe diameters are given in Table 2. Here it was aimed to see the effects of different relative roughness on flow conditions. The relative roughness of pipes, which is given in Table 2, was measured through the experimental work in which the pressure differences in the fully developed flow region is measured.

Static pressures were measured trough piezometres tubes fitted on pressure taps, which was welded to the holes drilled at seven different locations on the pipe. Pipe flows at each flow rate were recorded by a camera for three minutes. Time mean values of pressures are obained for each pressure taps from each flow record. The pressure values obtained from the numerical flows, which are parallel to experiment, are compared with the pressure values of the experiment as shown in Fig.3.

Table 2. Pipe type, relative roughnesses and diameters

Pipe Type	**Diameter (mm)**	**Relative roughness ε / D**
Aluminium pipe	26	0,0016
Copper Pipe	26	0,00016
Steel Pipe	28	0,0024
Galvanized Pipe	28	0,0026
PPRC pipe	21	0,00033

Numerical Solution and Validation

Basically, fluid flows are defined by differential flow equations which is a results of mass, momentum and energy conservation. For this reason, the flow field in turbulent flows shows a continuous change temporarily and spatially. The time-dependent solution of a turbulent flow is difficult since it requires a solution of turbulence structures in time-dependent development that is available in the flow in a wide range. The numerical method used to solve the time-dependent fundamental flow equations of a turbulent flow is called direct numerical simuation (DNS). The solution is not possible with today computers except that of very simple flows. Since solution is required very large mesh numbers and time steps.

An another method suggested for the solution of turbulent flow is to get the instantaneus effects of the flow into time average effect. By this way, turbulent flows become time independent flows. The instantaneous drags existed by the turbulent structures against the flow form additional stresses in the time-averaged basic flow equations. These stresses are called Reynolds stresse or turbulent sresses. The existed time averaged conservation equations are called Reynolds averaged Navier-stokes equations (RANS). Only unknown in RANS equations is the Reynolds stresses. Therefore many turbulence models are developed to solve these Reynolds stresses. The solution of a turbulent flow with RANS equations is simple and the cost of numerical computation is very low in comparison to the DNS method.

In this study, turbulent pipe flows are solved via computer by applying finite difference numerical method to RANS equations to each flow field point. SST k-omea model are selected to solve the Reynolds stresses. To provide laminer to turbulent transition, Gamma-Theta model is selected. The pipe length has been selected long enough to cover the fully developed flow partly. Since the pipe flow is axis symmetrical, the flow area is limited to a small flow area sliced. The boundary conditions, fluid properties and flow type are defined in Table 3 below.

As shown in table 3, after setting up of boundary condition, flow and fluid properties, pipe flows are solved with CFX flow solver program. Numerical flows are kept parallel with experimental flows. As a result, the flow characteristics such as pressure, velocity, friction factor and wall shear stress were analyzed along the flow. Numerical and experimental values were compared each other in order to gain validity to numerical solution. The experimental and numerical values are compared in Fig 3 in terms of the pressure drop along the flow including all flows of each pipe type. As shown in Fig. 3, experimental values and numerical values are in well agree. The mean and maximum deviations of the numerical values from the experimental ones are given in Table 4 also. As can be seen in Table 4, numerical values of all pipe flows has deviated from the experimental values about 7-9% in average. The deviation amount is a tolerable one since it is natural to have such a deviation. Because physical conditions such as fluid temperature can not be precisely determined and faults that occur in flow measurements and in static pressure readings in experimental runs are thought to be caused by these deviations. For this reason, flow characteristics are analyzed by means of numerical data as given below.

Table 3. Boundary conditions and flow field properties

Numerical properties	
Flow state	Steady-state, incompressible and isothermal flow
Basic flow equations	RANS Equations
Turbulence model	SST k-omega model
Pipe inlet	Smooth velocity and high turbulent intencity $(T_U) = \%7$
Pipe wall	roughly
Pipe outlet	Open to atmosphere at gauge pressure
fluid	27 ^{o}C water

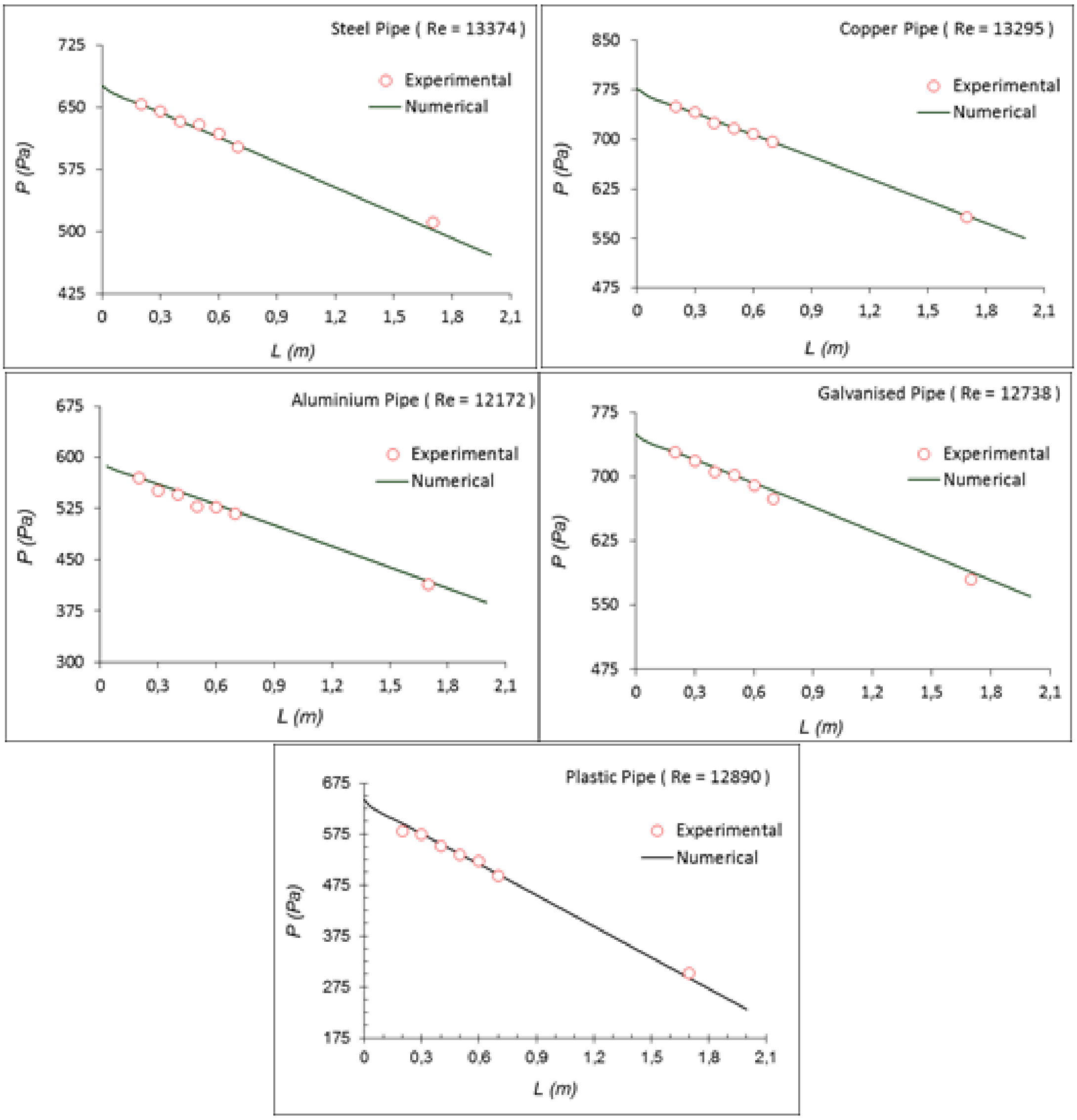


Figure 3. Comparison of numerical results with experimental data

Table 4. Percent deviation of numerical values from experimental data in terms of pressure variation along the flow

Deviation (%)	aluminium pipe	copper pipe	commercial steel pipe	galvanised pipe	Plastic pipe
maximum	±%20	±%24	±%36	±%35	±%18
average	%7.30	%7.70	%7.60	%9.40	%9

Numerical Analysis

In this section, the variation of central axial velocity in the pipe flows from the pipe inlet to fully developed flow is analyzed numerically. In this numerical study, pipe flows were performed with five pipe types at Reynolds numbers ranging from 2000 to 25000. The aim here is to examine the flow properties at low Reynolds numbered pipe flows where the transitional flow regime dominates. The uncertainty of the flow behavior, particularly at low Reynolds numbers, has led to prefer high Reynolds numbers in the design of heat exchangers pipe flows. In turbulent pipe flows with high Reynolds numbers, pressure losses are high and as well as energy consumption. Reducing fossil-based energy consumption for a cleaner environment has now become an obligation. In addition, reducing energy consumption also lowers the cost of energy consumption. The transition flow regime should be learned very well so that the designs of heat transfer also cover the low Reynolds numbered pipe flows. For this reason, more numerical and experimental studies are needed.

In this numerical study, a high turbulent flow presence at the conical pipe inlet is simulated along the dowstream of the pipe flow. Therefore, a conical inlet and high turbulent free stream is the limitataion of this numerical study . A conical pipe insert provide the inlet flow to be in a smooth velocity profile over the cross section. Therefore, a smooth velocity profile and a high turbulent level (I = 7%) is assigned as input in the inlet boundary condition. The smooth velocity profile at pipe inlet begins to change along the pipe flow due the fluid does not slip on the inner pipe wall. As a result of velocity variations, a velocity boundary layer develops along the flow with an increasing in thickness permenently. The velocity in the boundary layer increases from zero in the wall normal direction and ends in increasing at the boundary of the boundary layer. The flow velocity outside the boundary layer increases to compansate the same flow rate at each section of the flow conduit. The developing boundary layer ends when its thickness increases along the flow and is equal to the pipe radius at a certain distance. By combining of the boundary layer thickness at the pipe center, the changed velocity profile convert to a fully developed laminar or turbulent velocity profile.

The variation of velocity profiles in the developing flow and in fully developed flow region are visualized. The numerical results obtained were analyzed and compared with experimental studies. Here, the velocity field are visualized with color contours and vectors. For example, the velocity variations on the central plane in the pipe flow is shown in Fig.4.

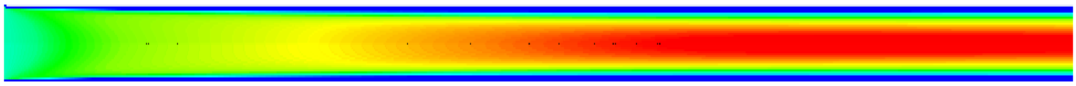


Figure 4. Display of the velocity distribution on the axial central plane by color contours

As shown in Fig. 4, the green color at the pipe inlet shows a low flow velocity, while the colours towards red show a high flow velocity. As can be seen the velocity colour varies from pipe inlet to a certain flow distance, but in the fully developed flow part, the colour of the velocity contour is unchanged. It also appears from the colour contour that the velocity decreases from the center line toward the pipe wall. The distribution of cross-sectional flow velocity at different downstream locations has been shown in the following figure. The state of velocity in the pipe flow can also be shown by vectors as shown in Fig. 5. In the Figure 5, the velocity at the pipe inlet (x = 0 m) is uniformly distributed. Downstream velocity vectors show that velocity profiles change along the flow. The velocity vectors at x = 0.8 m and x = 1.2 m appear to be the same, thus indicating the fully developed flow field. Comparisons of fully developed turbulent velocity profiles with experimental relationships and other experimental studies for all pipe flows are given in the following section.

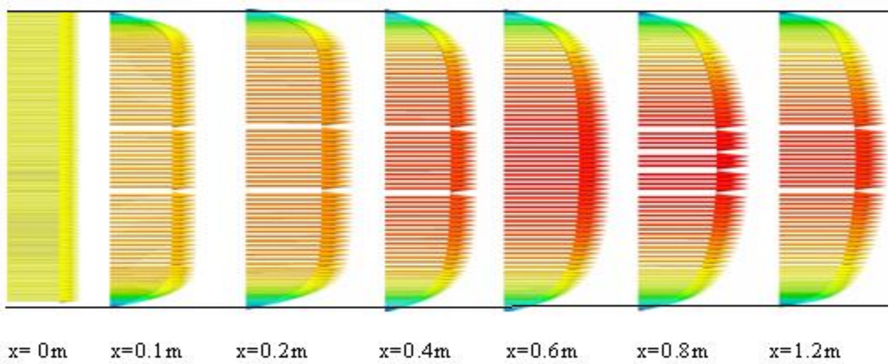
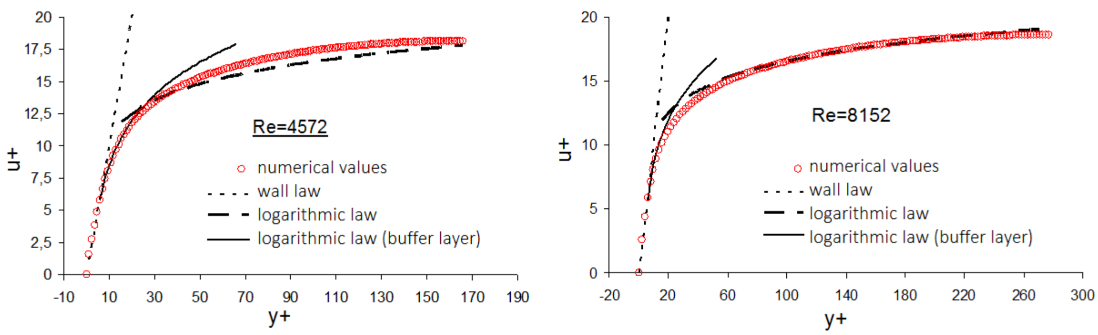


Figure 5. Cross-section velocity distribution vectors at different distances of the pipe



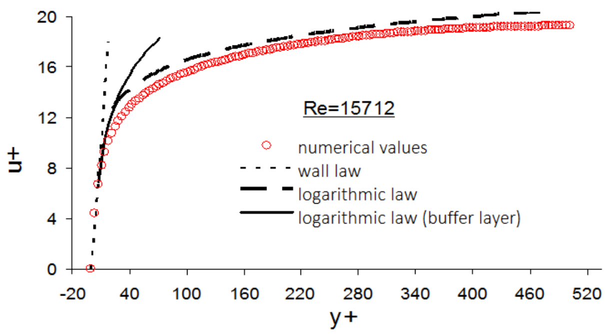


Figure 6. Comparison of dimensionless velocity profiles with empirical correlations at tree different Reynolds number flow of aluminium pipe aluminium pipe flows

Comparison with Empirical Correlations

In this section, velocity profiles obtained from the numerical study are compared with known empirical laws in order to provide the reliability of the numerical study. Due to fully developed flow conditions exists at 1.8m pipe location, The cross-section velocity profiles at that location have been used in comparisons. For example, velocity profiles of aluminium pipe flow at different Reynolds numbers are compared with the wall law and the logarithmic law in Fig.6. the wall law and logarithmic law are empirical relations that estimate the velocity variation in the sublaminar region and in the overlap region at high accuracy, respectively, in the fully developed turbulent pipe flow, In order to analyze the velocity variations in the numerical study, and to make sure that the resulting velocity profiles are correct, comparisons were made with those empirical relations

As shown in Fig. 6, the dimensionless velocity values in the fully developed flow are well agree with the wall law in the sublaminar flow region ($0 < y^+ < 5$). In this range, the wall law has shown an average deviation of 13% from the numerical values. In the case of $5 < y^+ < 30$, the numerical dimensionless velocity profiles deviate about $\pm\%10$ in average from the logarithmic law curve in the buffer layer. Numerical dimensionless velocity has been well agree with logarithmic law in the dimensionless distances of $y^+ > 30$. Here, the logarithmic law values deviate from the numerical values of about 5% in the range of $30 < y^+ < 60$, about 3% in the range of $60 < y^+ < 300$ and about 4% in the range of $300 < y^+ < 600$.

When the dimensionless distances between $30 < y^+ < 600$ is considered, it show a deviation from the numerical values about 3.5% in the general average. As a result of above comparisons, it is seen that logarithmic law has much better in agreement with numerical values than buffer layer equation and wall law. The three relationships that have to be seen here are the empirical correlation and the numerical velocity profiles are in very good agreement with those empirical correlations. This reinforces the reliability of numerical study.

The dimensionless velocity data obtained from all flows of five pipe types which is performed in the Reynolds number range of 2000-25000 are compared with the wall law and the logarithmic law. As a result, it has been seen that well agreement are found in all Reynolds numbers. Table 5 shows the general average percent deviations of the empirical relations from the fully developed dimensionless velocity profiles covering all study data.

Table 5. General mean deviation percentages from numerical values of experimental correlations

	aluminium pipe	**Copper pipe**	**Steel pipe**	**Galvanised pipe**	**Plastic pipe**
	Percent deviation- general average				
Wall law	± %14	± %15	± %13	±%18	±%18
Logarithmic law	± %3	± %3	±%5	±%7	±%3
Buffer layer Equation	±%9	±%6	±%10	±%13	±%6

According to the values given in Table 5, the logarithmic law is much better aligned with numerical values than the other two laws. It seems that the logarithmic law equation expressing the buffer layer is better aligned with the numerical values according to the wall law. The reason for the high deviation of the wall law is that the deviation to the dimensionless distances very close to the wall (y + <2) begins to become high. The reason is that the pipes are rough. Because the wall law is generally in good agreement with the empirical values of the smooth pipes. The above deviation percentages include comparisons up to 600 values of dimensionless distance. In the numerical study performed, the logarithmic law in the range of 30 $<y^+<600$, the wall law in the range of 0 $<y^+<5$ and the buffer layer equation in range of $5<y^+ <30$ is compared with the numerical dimensionless velocity values.

As a result, the fully developed velocity profiles in the numerical study are well adapted to the empirical relations. The comparison of numerical and empirical correlations here is intended to show the relaibility of the numerical solution. According to these comparisons, the accuracy level of numerical study is obtained high.

Findings and Results

The reliability of numerical work is provided through comparisons of numerical data with experimental study in terms of pressure drop and with empirical relations in terms of fully developed velocity profiles. The numerical data in this case can be used to analyze the developing flow part in the pipe flow. By analyzing the axial velocity profiles obtained from the numerical studies, some important correlations can be obtained for the developing flow part. The variation in central axial velocity along the flow, which is obtained from the numerical simulation of the aluminum tube is shown in Fig. 7. The central axial velocity is accelerated to increase from pipe inlet to reach a maximum velocity in the downstream in order to meet the reduced flowrate due to the boundary layer developing effect.

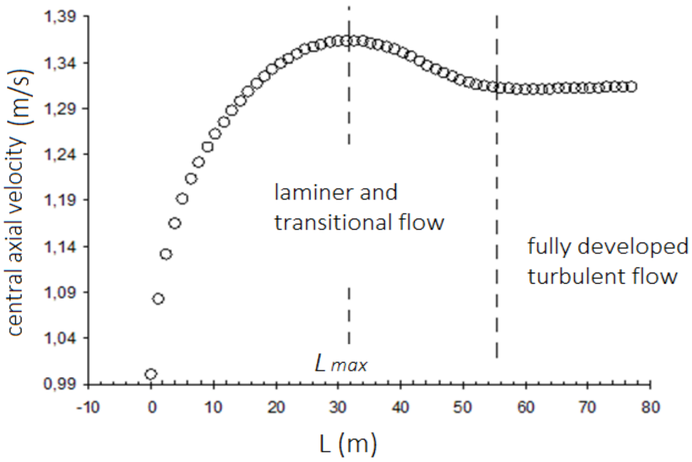


Figure 7. Variation of central axial velocity in Reynolds number of 4572 in the numerical aluminum pipe flow

As shown in Fig. 7, the axial velocity has shown a nonlinear increase from the pipe inlet towards peak point. At the peak point, the axial velocity reaches maximum and then decrease to a short distance to reach the unchanged values. The point at which the axial velocity has reached unchanged values is where the fully developed flow begins. In the fully developed flow section, two flow characteristics, such as axial velocity and pressure drop gradient, are no longer change throughout the flow. The flow point at which maximum velocity occurs is the flow point at which the boundary layer thickness combine at the pipe center. Here, fully developed flow occurs just beyond this point.

The transition distance to fully devloped turbulence from pipe inlet can be determined from the observation of the variation of axial flow velocity, but the critical transition distance where laminar transit to turbulence from pipe inlet can not be determined as well the transition distance. In most experimental studies, axial velocity values are generally found to be maximum at a range of 30-40D in pipe diameters (Anselmet et.al. (2009)). In numerical studies with five pipe types, the dimensionless diameters (L_{max} / D) at which the central axial velocities are maximum are shown in Table 6 against the Reynolds number. When the dimensionless axial velocity values in the table are examined, it is seen that there is a rapidly decrease up to Re=10000 Reynolds number. After $Re >10000$, it has passed to a low linear decrease of which the slope is low. The representation of L_{max}/D values given in Table 6 is shown on Fig. 8.

As shown in Fig. 8, L_{max} $/D$ values has shown a rapid decline up to 10000 Reynolds numbers. From this Reynolds number, it has gone to a low linear drop which is low in slope. The following numerical relation is obtained as a result of curve fitting works for linear drops beginning from 10000 Reynolds number.

Table 6. Dimensionless L_{max} / *D values where central axial velocity become maximum*

	aluminium pipe										
	4572	**5539**	**8152**	**9496**	**10539**	**11371**	**12172**	**13295**	**15712**	**18004**	**21043**
L_{max}/D	31.29	31.29	26.08	24.77	23.47	23.47	23.47	22.16	22.16	20.86	20.86
	copper pipe										
	3443	**4326**	**5738**	**7785**	**9002**	**9282**	**11079**	**13295**	**15856**	**18387**	**22084**
L_{max}/D	37.81	32.59	29.99	26.08	24.77	24.77	24.77	23.47	23.47	22.16	22.16
	steel pipe										
Re	**3609**	**5422**	**7101**	**8722**	**10643**	**11906**	**13374**	**13907**	**17257**	**19733**	**21688**
L_{max}/D	36.32	31.48	27.85	24.21	23.00	23.00	21.79	21.79	20.58	20.58	20.58
	galvanised pipe										
	2907	**4559**	**7295**	**9553**	**10643**	**11977**	**12738**	**14078**	**14861**	**15582**	**18439**
L_{max}/D	67.80	32.69	26.63	24.21	23.00	23.00	21.79	21.79	21.79	21.79	20.58
	Plastic pipe										
	3691	**4921**	**6890**	**8120**	**9842**	**12890**	**14458**	**16983**	**17540**	**20979**	**24317**
L_{max}/D	33.90	32.28	27.44	25.83	24.21	22.60	22.60	22.60	22.60	22.60	20.98

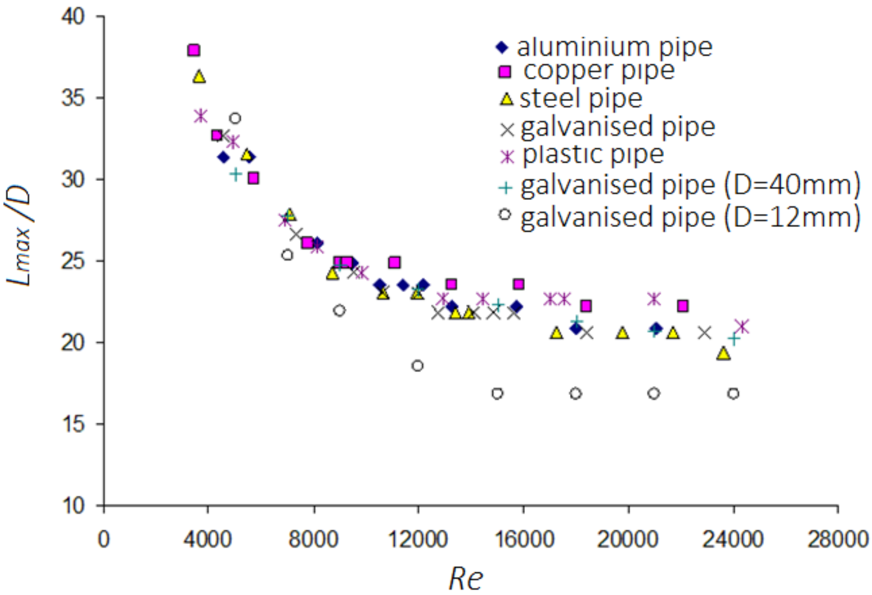


Figure 8. Variation of L_{mak} / *D values across Re for Maximum axial velocity*

$$\frac{L_{\max}}{D} = \frac{D^{0.2}}{\varepsilon^{0.06}}(28 - 0.00028\mathrm{Re}) \qquad \mathrm{Re} > 10000 \qquad (1)$$

In this numerical relation, the dimensionless diameter distance at which the axial velocity is maximum has been a function of the pipe diameter, roughness and Reynolds number. Diameter and roughness in this equation is just being a coefficient and it should be used their units in meter (m) in Equation. The correlation was found to deviate from the axial velocity values obtained from the entire numerical study by ± 8% in maximum and ± 3.5% in overall average. The estimated L_{max} / *D* values of the iron pipe and the galvanized pipe by Equ. (1) are shown in Fig. 9 below.

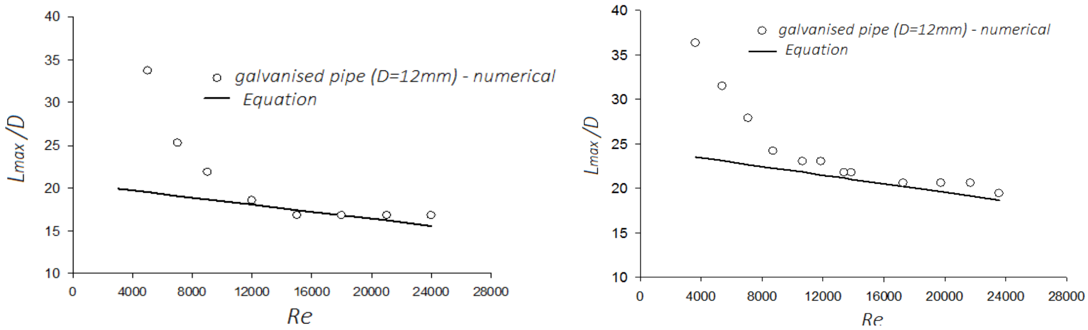


Figure 9. Estimation of L_{max}/D values where central axial velocity become maximumwith Equ. (1) for galvanised and steel pipe flows

Conclusion

In this study, pipe flows in steady, incompressible and constant properties were simulated numerically in the Reynolds range of 2000-25000 for five pipe types with different relative roughness. In this study, pipe flows of steady, incompressible and constant properties were simulated numerically in the Reynolds numbers ranged between 2000 and 25000 by performing five pipe types in different relative roughness. In the solution of pipe flows, the RANS equations, which are time-averaged basic flow equations, were used. The SST k-omega model was used to include the turbulence effects into the stream. Flows were simulated with a lengthy pipe to cover the developing and fully developed flow region. Numerical results were compared with experimental work and other empirical velocity laws and it was seen that they fit very well. From the numerical results, the measured flow distances from the pipe inlet to flow point at which the central axial velocities were maximum were obtained. When the graphs of the variation of the axial velocity along the flow were analyzed, it was observed that the fully developed flow was slightly ahead of the flow point at which the axial velocity is maximum. The variation of the flow distances at which axial velocity is maximum was studied with Reynolds number and with different pipe types. Until Re<10000, the dimensionless flow lengths exhibit a rapid and non-linear decrease. However, in the Reynolds range of 10000 < Re < 25000, they show a drop in linear and at low slope. Variation of dimensionless flow lengths were investigated with pipe diameter, relative roughness and Reynolds number. As a result of curve fitting works, a numerical corrrelation including pipe diameter, relative roughness and Reynolds number effects and expressing dimensionless lengths in the range of 10000 <Re <25000 has been derived. The correlation was found to deviate by a maximum of ± 8% and an overall average of ± 3.5% from the axial velocity values obtained from the entire numerical study. The developing flow lengths can also be estimated by this relationship. However, the correlation can be suggested in the literature after testing with new experimental data.

References

Anselmet, F., Ternat, F., Amielh, M., Boiron, O., Boyer, P., & Pietri, L. (2009). Axial development of the mean flow in the entrance region of turbulent pipe and duct flows. *Comptes Rendus Mécanique, 337*(8), 573-584.

Augustine, J. R. (1988). *Pressure Drop Measurements in the Transition Region for a Circular Tube with a Square-Edged Entrance* (Master's Thesis). Bachelor of Science in Mechanical Engineering. The University of Southwestern Louisiana Lafayette, Louisiana.

Barbin, A. R., & Jones, J. B. (1963). Turbulent flow in the inlet region of a smooth pipe. *Journal of Basic Engineering, 85*(1), 29-33.

Doherty, J., Ngan, P., Monty, J., & Chong, M. (2007, January). The development of turbulent pipe flow. In *16th Australasian Fluid Mechanics Conference (AFMC)* (pp. 266-270). School of Engineering, The University of Queensland.

Minkowycz, W. J., Abraham, J. P., & Sparrow, E. M. (2009). Numerical simulation of laminar breakdown and subsequent intermittent and turbulent flow in parallel-plate channels: Effects of inlet velocity profile and turbulence intensity. *International Journal of Heat and Mass Transfer, 52*(17-18), 4040-4046.

Nikuradse J (1966). Gestzmassigkeiten der turbuleten stromung in glatten rohren. Forschung auf dem Gebiet des. Ingenieurwesens. Translated in NASA TT F-10, 359(3), 1932, 1-36.

Laufer, J. (1954). The structure of turbulence in fully developed pipe flow. NACA Report, Washington, National Bureau of Standards.

Ozisik, M. N. (1985). *Heat transfer: a basic approach*. New York: McGraw-Hill

Patel, V. C., & Head, M. R. (1969). Some observations on skin friction and velocity profiles in fully developed pipe and channel flows. *Journal of Fluid Mechanics, 38*(1), 181-201.

Perry, A. E., & Abell, C. J. (1975). Scaling laws for pipe-flow turbulence. *Journal of Fluid Mechanics, 67*(2), 257-271.

Salami, L. A. (1986). An investigation of turbulent developing flow at the entrance to a smooth pipe. *International journal of heat and fluid flow, 7*(4), 247-257.

Tam, H. K., Tam, L. M., & Ghajar, A. J. (2013). Effect of inlet geometries and heating on the entrance and fully-developed friction factors in the laminar and transition regions of a horizontal tube. *Experimental thermal and fluid science, 44*, 680-696.

White, F. M. (2003). *Fluid Mechanics*. 5th edition, McGraw–Hill Book Co, New York.

Zanoun, E. S., Kito, M., & Egbers, C. (2009). A study on flow transition and development in circular and rectangular ducts. *Journal of Fluids Engineering, 131*(6), 061204.

Zimmer, F., Zanoun, E. S., & Egbers, C. (2011). A study on the influence of triggering pipe flow regarding mean and higher order statistics. In *Journal of Physics: Conference Series* (Vol. 318, No. 3, p. 032039). IOP Publishing.

Author Information

Hasan Duz
Batman University, Tecnology Faculty, Automotive Engineering, Turkey
Batman Üniversitesi, Batı Raman Kampüsü, Lojmanlar, Loj 3 No:12, Batman, Turkey
E-mail: hasan.duz@batman.edu.tr