The University of Western Ontario

Department of Mechanical and Materials Engineering

APPLIED COMPUTATIONAL FLUID MECHANICS
AND HEAT TRANSFER
Course MME 9614

Assignment No.2

"A comparison of Numerical solution with Exact or Empirical Solution in case of Laminar and Turbulent flow through pipe on different Reynolds number"

Dated: 25 October 2016

Submitted to:
Prof. Chao Zhang
Department of
(Mechanical and Material Engineering)

Submitted by:
Name: Jagdeep Singh
Student No. 250911825
Meng. (MME)

`

1.Problem Description:

Problem 1:

This first problem of the assignment involves numerical and analytical study of 2-D Laminar and Turbulent flow through pipes. This problem has two components which is given below and explain in Fig. (1);

1) Study of Laminar flow when R_e=100 and $R_{e=}$1000
2) Study of Turbulent flow when $R_e=10^5$ and $R_{e=}10^7$

The focus of study is on comparison of entrance length and velocity profile of analytical or empirical solution with that computed with the help of numerical solution(CFD) on different Reynolds numbers.

Case 1 & 2: Laminar Flow Properties Table

Sr.No.	Properties	R_e=100	R_e=1000
1	Diameter	0.1 Meter	0.1 Meter
2	Density of Fluid	998.2 kg/m^3	998.2 kg/m^3
3	Viscosity @ 20 Degree Celsius	1.002e-3 Pa s	1.002e-3 Pa s
4	[1]Entrance Length(L_e), $L_e/D=0.06R_e$	0.6 Meter	6 Meter
5	Length of Pipe	1.2 Meter	12 Meter
6	Inlet Velocity of fluid	0.001 m/s	0.01 m/s

Case 3 & 4: Turbulent Flow Properties Table

Sr.No.	Properties	$R_e=10^5$	$R_e=10^7$
1	Diameter	0.1 Meter	0.1
2	Density of Fluid	998.2 kg/m^3	998.2 kg/m^3
3	Viscosity @ 20 Degree Celsius	1.003e-3 Pa s	1.003e-3 Pa s
4	[1]EntranceLength(L_e), $L_e/D=4.4(R_e)^{1/6}$	3	8.9
5	Length of Pipe	6 Meter	20
6	Inlet Velocity of Fluid	1.0048	100.48

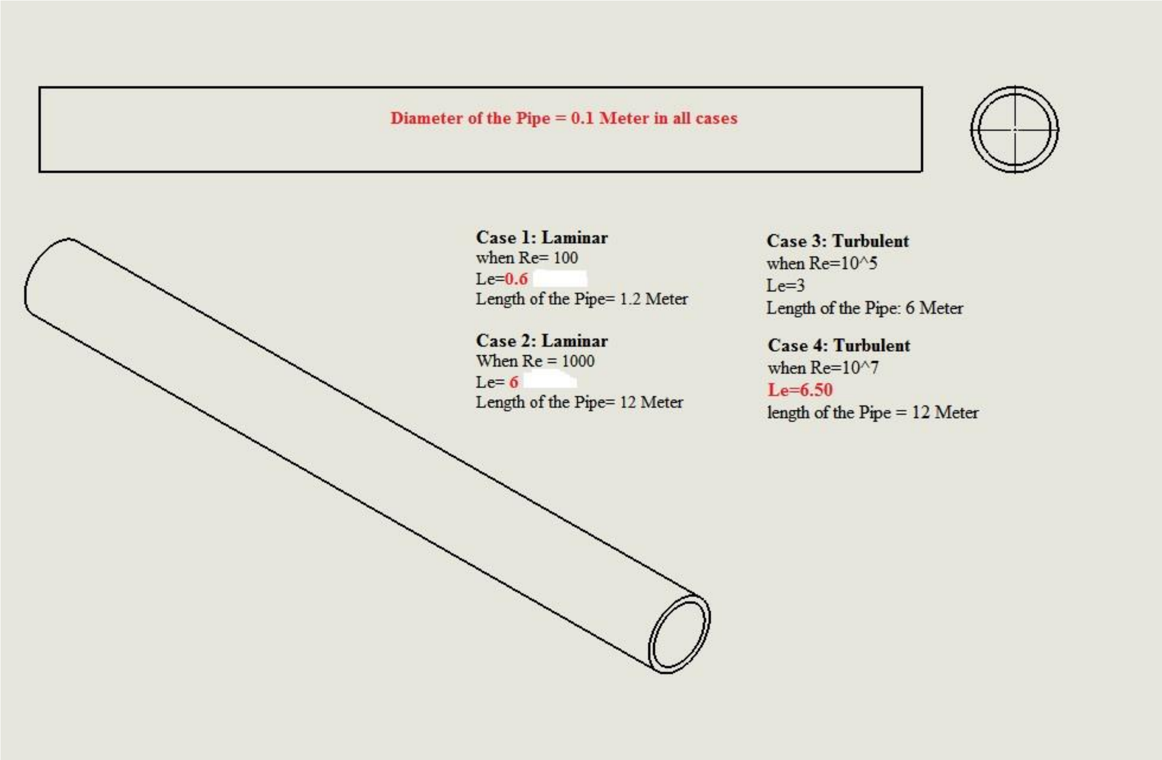


Fig: (1)

Problem 2

The second problem involves numerical calculation of the velocity and temperature of turbulent flow through 2-D pipe when Reynolds number is 10^6 with the following boundary conditions:

a) T_e=300 degree Celsius (Temperature of the fluid while entering the pipe)
b) T_0=25 degree Celsius (Uniform Temperature of the surface)
c)

Properties of the Pipe:

a) Diameter of the pipe= 0.1 Meter
b) Length of the pipe= 5 Meter

`

2. Mathematical Models of the Problem 1 (Case 1 to 4):

a) Mathematical Model for the Case 1 and 2(For Laminar Flow through Pipes)

$$U = U_{max}\left(1 - \frac{r^2}{R^2}\right) \qquad \text{Eq. 2.1}$$

$$where,\ \ U_{max} = \left(-\frac{dp}{dx}\right)\frac{R^2}{4\mu} \qquad \text{Eq. 2.2}$$

$U_{max} = 2\ V_{avg}$ (In fully developed flow) Eq. 2.3

$$\mathrm{L_e} = 0.06 * \mathrm{R_e} * \mathrm{D} \qquad \text{Eq. 2.4}$$

Where;

$U_{max} = Maximum\ Velocity$

$V_{avg} = average\ Velocity$

R = radius of the pipe

r= any distance along radial direction up to R

The Eq. (2.1) can be obtained from the **Navier's stoke equation** in cylindrical co-ordinates by applying the following assumptions;

The flow is only one direction that is in Z-direction and varying with radial direction therefore other two components of the velocity will be zero, $U_r=U_\Theta=0$ and constant pressure drives the flow. Therefore $\frac{dp}{dx}$ is consatnt. The flow is steady, incompressible and viscosity is constant.

b) Turbulent Model:

[1]The velocity distribution along the cross-section of the pipe within the fully developed flow part we concerned could be derived as follow:
Let k=0.41 and B=5.0, get:

$$u_r = u^*\left[\frac{1}{0.41}\ln\frac{(R-r)u^*}{\upsilon} + 5\right] \qquad \text{Eq. 2.5}$$

Where ν is the kinematic viscosity $\frac{\mu}{\rho}$,

$$u^* = \frac{u\mathrm{Re}}{2R}\left(\frac{f}{8}\right) \text{ and } f = \left(1.8\log\frac{\mathrm{Re}}{6.9}\right)^{-2} \qquad \text{Eq. 2.6}$$

`

The velocity profile for turbulent flows can be written as

$^{1}u_r = u^*[\left[\frac{1}{4}\ln\frac{(R-r)u^*}{v}\right] +5]$ Eq. 2.7

For the entrance length of turbulent flows, the empirical correlation for entrance length is:

$\frac{l_e}{D} = 4.4(Re)^{\frac{1}{6}}\, for\ turbulent\ flow$

Where D is the diameter of the pipe.

3. Numerical Schemes:

For Laminar flow pressure velocity coupling scheme is used and for discretization of governing equation of fluid flow a second order upwind scheme is used for robustness and accuracy. While on the other hand for turbulent flows k-Epsilon model is used to capture turbulence velocity profile with second order upwind scheme is used for discretization.

The general Transport Equation is given below:

$cv \int \frac{d\emptyset\rho}{\partial t}.dV + \oint \rho\emptyset U.dA = \oint \Gamma_{\emptyset}\,\nabla_{\emptyset}.dA + V\int S_{\emptyset}.dV$ Eq. 3.1

where

ρ = Density of fluid

U = velocity vector

A = surface area

$\Gamma_{\emptyset}$= diffusion Coefficient for Scalar

$\nabla_{\emptyset}$= gradient of scalar

$S_{\emptyset}$= source Term

The discretized form of the equation is given below:

$\frac{d\emptyset\rho}{\partial t}V + \sum_f^{Nfaces}\rho.\nabla_{\emptyset}.A_f\ +\ \sum_f^{Nfaces}\Gamma_{\emptyset}.\nabla_{\emptyset}.A_f\ \ + S_{\emptyset}.V$ Eq(3.1.2)

where

N_{faces} = number of faces enclosing cell

$\emptyset_f$ = value of $\emptyset$ convected through face f

$\rho.\nabla_{\emptyset}.A_f$ = Mass flux through the face

`

$\Gamma_{\emptyset}.\nabla_{\emptyset}.A_f$ = gradient of $\emptyset$ at face f

V = cell volume

3.2 K-epsilon model

[2]For turbulent flow, k-epsilon model is used to capture the turbulence velocity profile. In this model two equations are used. The first one is transport equation of turbulent kinetic energy (k) and the second one is transport equation of dissipation rate of turbulent kinetic energy (ε). The general form of these equations can be written as the follows. k-transport equation:

$$\frac{\delta}{\delta t}(\rho k) + \frac{\delta}{\delta x_i}(\rho k u_i) = \frac{\delta}{\delta x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \qquad \text{Eq. 3.2}$$

ε-transport equation

$$\frac{\delta}{\delta t}(\rho\varepsilon) + \frac{\delta}{\delta x_i}(\rho\varepsilon u_i) = \frac{\delta}{\delta x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

Eq. 3.2

where

$\mu_l = \rho c_\mu \frac{k^2}{\varepsilon}$, $G_k = -u_i u_j \frac{\partial u_j}{\partial x_i}$, $G_b = \beta g_i \frac{\mu_t}{Pr_t}\frac{\delta T}{\delta x_i}$, $Y_M = 2\rho\varepsilon M_t^2$, $C_{3\varepsilon} = tanh\frac{|v|}{|u|}$, σ_k=1.0 , $\sigma_\varepsilon = 1.3$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$ and $C_\mu = 0.09$

Energy equation:

The energy equation combined with turbulence model is used for part (b) problem of the assignment.

The Energy equation is solved by fluid in the form given as following:

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\big(v(\rho E + p)\big) = \nabla.\left(k_{eff}\nabla T - \sum_j h_j \vec{J}_j + (\overline{\overline{T_{eff}}}.\ \vec{v})\right) + S_h \qquad \text{Eq. 3.3}$$

Grid Independence Test:

Grid Independence Test is performed on three different meshes for each case and difference in error is approximately less that 1 percent therefore any grid can be implemented to obtained the desired results in each case. This grid independence test can be clearly seen in the Table (1), (2), (3), (4) for example for Re=100 the below results are obtained:

<table>
<tr><th colspan="4">Re =100</th></tr>
<tr><td></td><td>Mesh1</td><td>Mesh2</td><td>Mesh3</td></tr>
<tr><td>Entrance length</td><td>1.19192</td><td>1.19598</td><td>1.20301</td></tr>
<tr><td>Velocity</td><td>0.000979138</td><td>0.000988666</td><td>0.000990717</td></tr>
<tr><td>Difference % error entrance length</td><td colspan="2">0.33947056</td><td></td></tr>
<tr><td></td><td></td><td colspan="2">0.584367545</td></tr>
<tr><td>Difference % error velocity</td><td colspan="2">0.963722835</td><td></td></tr>
<tr><td></td><td></td><td colspan="2">0.207021783</td></tr>
</table>

Mesh 20 x 120

Mesh Quality:

Minimum Orthogonal Quality = 1.00000e+00
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)

Maximum Ortho Skew = 3.74044e-11
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)

Maximum Aspect Ratio = 2.07237e+00

Mesh Size

Level Cells Faces Nodes Partitions
0 1071 2270 1200 1

Mesh Size 20x240

Level Cells Faces Nodes Partitions
0 4541 9340 4800 1

1 cell zone, 5 face zones.

Mesh Quality: 20 x 240

Minimum Orthogonal Quality = 1.00000e+00
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)

Maximum Ortho Skew = 0.00000e+00
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)

Maximum Aspect Ratio = 2.15413e+00

Mesh Size: 40 x 480

Level Cells Faces Nodes Partitions
0 18681 37880 19200 1

1 cell zone, 5 face zones.

Mesh Quality:

Minimum Orthogonal Quality = 1.00000e+00
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)

Maximum Ortho Skew = 0.00000e+00
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)

Maximum Aspect Ratio = 2.19508e+00

```
R=0.05;
Vavg=0.001;
Umax=2*Vavg;
for r=0:0.005:0.05;
    U=Umax*(1-r^2/R^2)
end
```

Turbulent:

```
Re = 100000;
v= 1.0048;


for r = 0 : 0.005: 0.05
     u = (0.0469)*((1/0.41)*log(((R-r)*0.0469)/0.000001004)+ 5)
end
```

`

Results of Velocity Profile for Re=100.

The center line velocity in case of analytical solution is 0.002 m/s and CFD Velocity is 0.00198127 and on the other hand, Entrance length in case of analytical is 0.6m and with CFD, it is calculated around 0.61 meter which is approximately the same value.

Discussion of Results, Velocity profile for Re=100:

The Velocity profile should be parabolic in nature as per analytical solution for laminar flow through pipes. Here, CFD solution looks promising as we get parabolic profile in velocity distribution in case of Re=100 as shown in Fig. (1) and Fig. (2) and Table 1. Velocity solution are very similar in nature when compared CFD and exact Solution. The center line velocity in case of analytical solution is around 0.00198127 getting on Mesh Size 40 x 480 while in case of analytical solution it is 0.002 m/s.

On the other hand, the entrance length for analytical solution in case of Re=100 is 0.6 while on CFD it is 0.65 therefore entrance length approximation by CFD solution is a good deal as shown in Fig. (4) after 0.65 the velocity value becomes constant therefore obtained the fully developed flow after in 0.65 in case of CFD solution.

The analytical solution of Velocity Profile is obtained with the help of MATLAB is given below for Re=100;

```
R=0.05;
Vavg=0.001;
Umax=2*Vavg;
for r=0:0.005:0.05;
    U=Umax*(1-r^2/R^2)
end
```

The same code can be implemented for evaluating the velocity profile for Re=1000 by changing the average velocity.

Results of Velocity Profile for Re=1000.

The center line velocity in case of analytical solution is 0.02 m/s and CFD Velocity is 0.0198334 and on the other hand, Entrance length in case of analytical is 6m and with CFD it is calculated around 6.01 meter.

Discussion of Results Velocity profile for Re=1000:

As the Reynolds number increases the average velocity of fluid flow is also increases because it is directly related in dimensionless number. In case of analytical solution, the maximum velocity is 0.02 m/s while with CFD solution the velocity value is very near to analytical solution that is 0.0198334 m/s in case of Mesh Size 40 x 2400 as shown in Table.

`

(2). The velocity profile is parabolic in nature as seen in Fig. (5) and Fig (6). From the analytical solution, one can say the velocity is maximum at the centerline as we can see in Table. (2) and decreases radially and become zero at the wall due to no slip boundary condition.

On comparing entrance length for Exact and Numerical solution it is approximately the same value as seen in Fig. (7), Fig. (8), Fig (9). Where the CFD solution matches, the values calculated by analytical.

<u>Case 3 and Case 4:</u>

Turbulent Flow: for Re=10^5 and 10^7

Case 3: When Re=10^5

Results of Turbulent Flows When Re=10^5:

The center line velocity in case of turbulent flow is calculated with the help of empirical relation as per given in the Frank M. White Book. The value of velocity at the center line is 1.12170 in case of empirical relation and that calculated with the help of CFD is 1.18213 and on the other hand, the value of entrance length is 3 meter while in CFD it is 3.35 meter is very near to the empirical value.

Discussion of Results:

[1]The velocity profile in case of Turbulent flow is not parabolic and it can be clearly noticed in Fig. (10) because there is no exact solution for turbulent flow till date and data is computed empirical relations. There are two components of velocities in case of turbulent flows when is mean velocity and other is fluctuating velocity. So, the velocity profile calculated from the log law as shown in the Eq. (2.7). The analytical solution is obtained with the help of MATLAB code as tabulated in the Table. (3). The velocity profile, that calculated from CFD solution is very similar as shown in Fig. (10), Fig. (11) and Fig. (12) to empirical relation calculated values therefor the CFD solution can be adapted for turbulence modeling.

Results of Turbulent Flows When Re=10^7:

The center line velocity for turbulent flow in case of Re=10^7 is 110.415 calculated with the help Empirical relations and that calculated with the help of CFD is 108.8345. on the other hand, empirical entrance length is 8.9m and in case of CFD it is 8.95 meter as shown in Table (4).

Discussion of Results of Turbulent Flows When Re=10^7:

As discussed earlier in case of $Re=10^5$ that the velocity profile is not parabolic in case of turbulent flow. This is also the turbulent case having large Reynolds number there the effect of inertial forces is more as compared to viscous forces. The velocity profile that is obtained with the help of log Law empirical relation is very close to that we have computed numerically with the help of CFD as shown in Fig. (17) a similar thing also happened in case of Entrance Length which can be clearly seen in the Fig. (18).

Conclusion:

To conclude, from all the above simulation that are done with the help of CFD using ICEM for geometry and meshing one can say the results are obtained with help of CFD are reliable when compared to analytical or empirical relations. In case of turbulent fluid flow problem there is no single analytical solution is obtained till yet. Therefore, for turbulent flow CFD plays a vital role for simulation as it can be clearly seen from the above problem solved on turbulent fluid flow. In laminar flow, the velocity profiles must be parabolic as stated by analytical solution and obtained parabolic by CFD simulation similarly in case of entrance length similar results are obtained. The same results can be seen for other problem also.

References:

[1] White, Frank.M "Introduction to Fluid Mechanics" pp 357 to 367.
[2] Incropera, Frank P. Incropera " Introduction to Heat transfer" pp 484-485.
[3] Ansys Theory Guide.

`

Comparison of Numerically calculated value with exact solution of Velocity Distribution on Reynold number= 100:

R-Position	Exact Values	Velocity Magnitude(m/s) Mesh Size 10 x 120	Velocity Magnitude(m/s) Mesh Size 20 x 240	Velocity Magnitude(m/s) Mesh Size 40 x 480
0.05	0.000000	0	0.000000	0
0.045	0.000380	0.0003865	0.000384	0.000382669
0.04	0.000720	0.00072305	0.000724	0.000723665
0.035	0.001000	0.00101803	0.001022	0.00102342
0.03	0.001300	0.00127252	0.001280	0.00128177
0.025	0.001500	0.00148657	0.001497	0.00149872
0.02	0.001700	0.00166051	0.001672	0.00167459
0.015	0.001800	0.00179492	0.001807	0.00180999
0.01	0.001900	0.00189059	0.001903	0.00190579
0	0.002000	0.0019576	0.001977	0.00198127
-0.01	0.001900	0.00189059	0.001903	0.00190579
-0.015	0.001800	0.00179492	0.001807	0.00180999
-0.02	0.001700	0.00166051	0.001672	0.00167459
-0.025	0.001500	0.00148657	0.001497	0.00149872
-0.03	0.001300	0.00127252	0.001280	0.00128177
-0.035	0.001000	0.00101803	0.001022	0.00102342
-0.04	0.000720	0.00072305	0.000724	0.000723665
-0.045	0.000380	0.0003865	0.000384	0.000382669
-0.05	0.000000	0	0.000000	0

Table. (1)

`

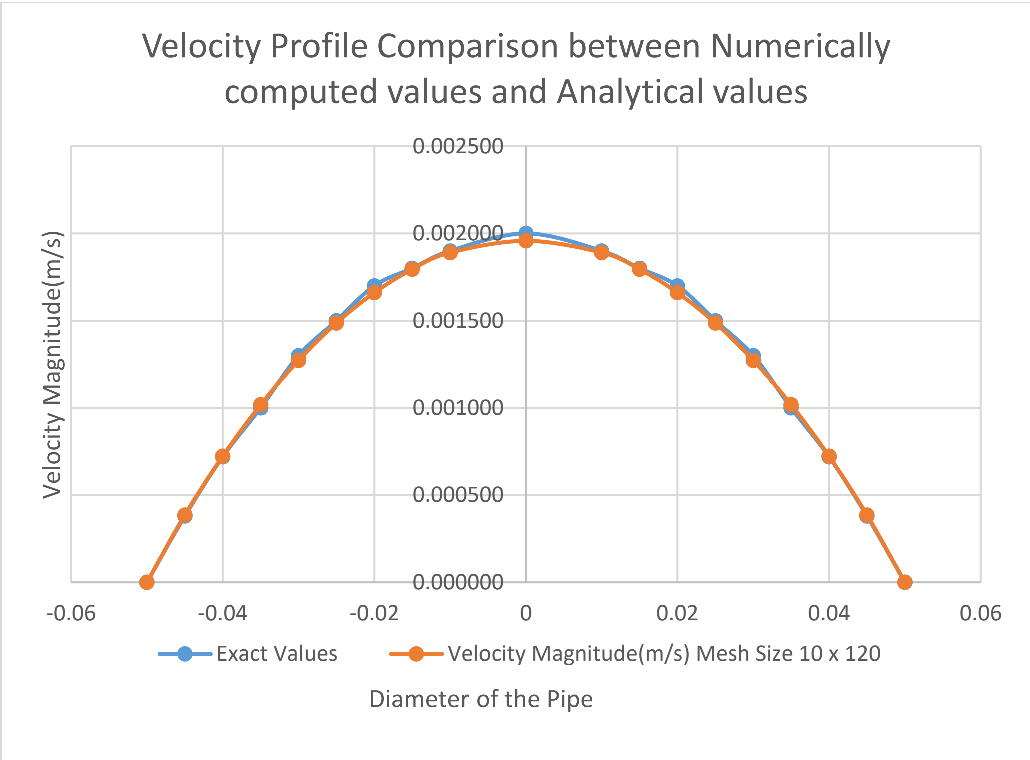


Fig. (1) Velocity Profile on Mesh Size 10 x 120(Re=100)

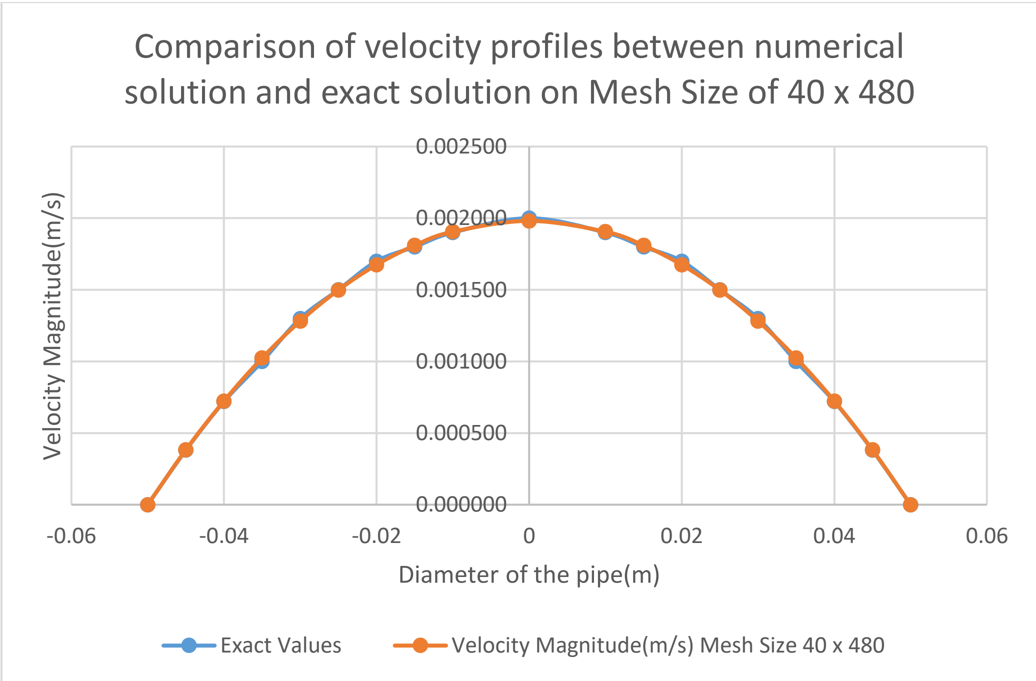


Fig. (2) Velocity Profile on Mesh Size 40 x 480 (Re=100)

`

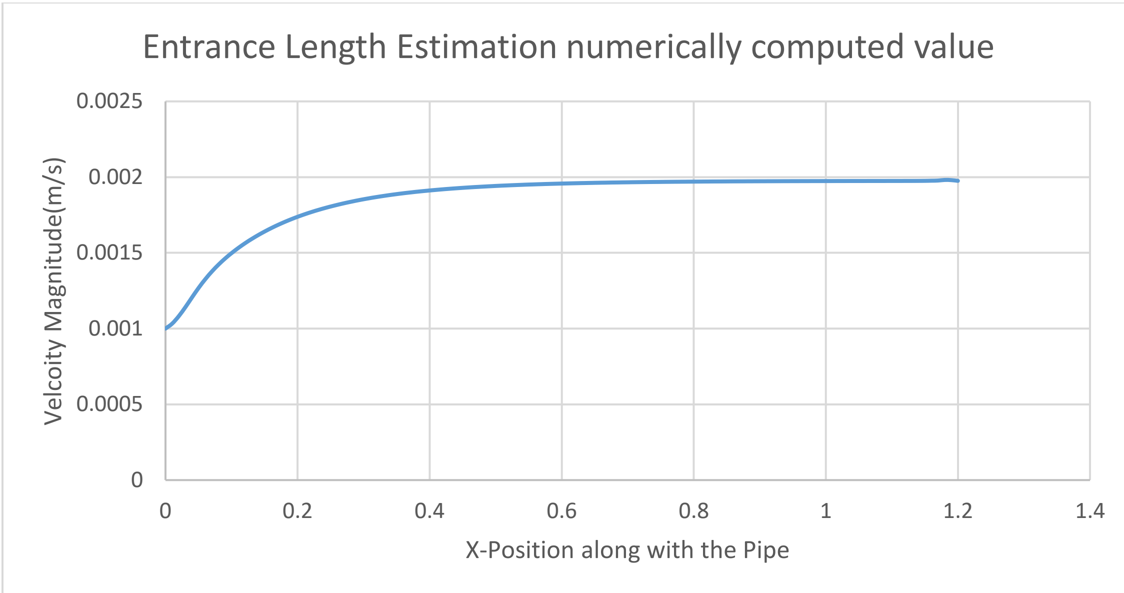


Fig. (3) Entrance Length on Mesh Size 10 x 120(Re=100)

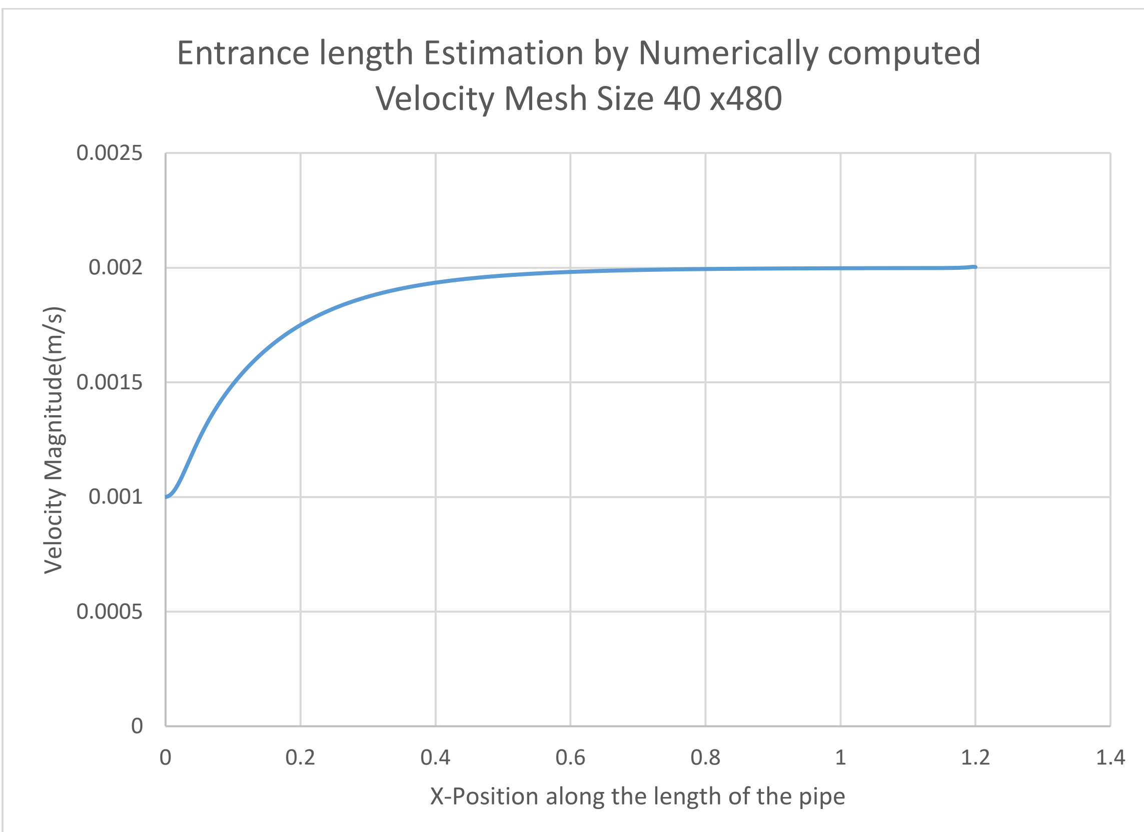


Fig. (4) Velocity Profile on Mesh Size 40 x 480(Re=100)

`

Comparison of Numerically calculated value with exact solution of Velocity Distribution on Reynold number= 1000:

R-Position	Exact Values	Velocity Magnitude(m/s) Mesh Size 10 x 600	Velocity Magnitude(m/s) Mesh Size 40 x 2400
0.05	0	0	0
0.045	0.0038	0.00386237	0.00382388
0.04	0.0072	0.00722652	0.00723226
0.035	0.0102	0.0101764	0.0102298
0.03	0.0128	0.0127231	0.012815
0.025	0.015	0.0148671	0.0149879
0.02	0.0168	0.016611	0.0167514
0.015	0.0182	0.0179604	0.0181109
0.01	0.0192	0.0189222	0.0190739
0	0.02	0.019597	0.0198334
-0.01	0.0192	0.0189222	0.0190739
-0.015	0.0182	0.0179604	0.0181109
-0.02	0.0168	0.016611	0.0167514
-0.025	0.015	0.0148671	0.0149879
-0.03	0.0128	0.0127231	0.012815
-0.035	0.0102	0.0101764	0.0102298
-0.04	0.0072	0.00722652	0.00723226
-0.045	0.0038	0.00386237	0.00382388
-0.05	0	0	0

Table (2)

`

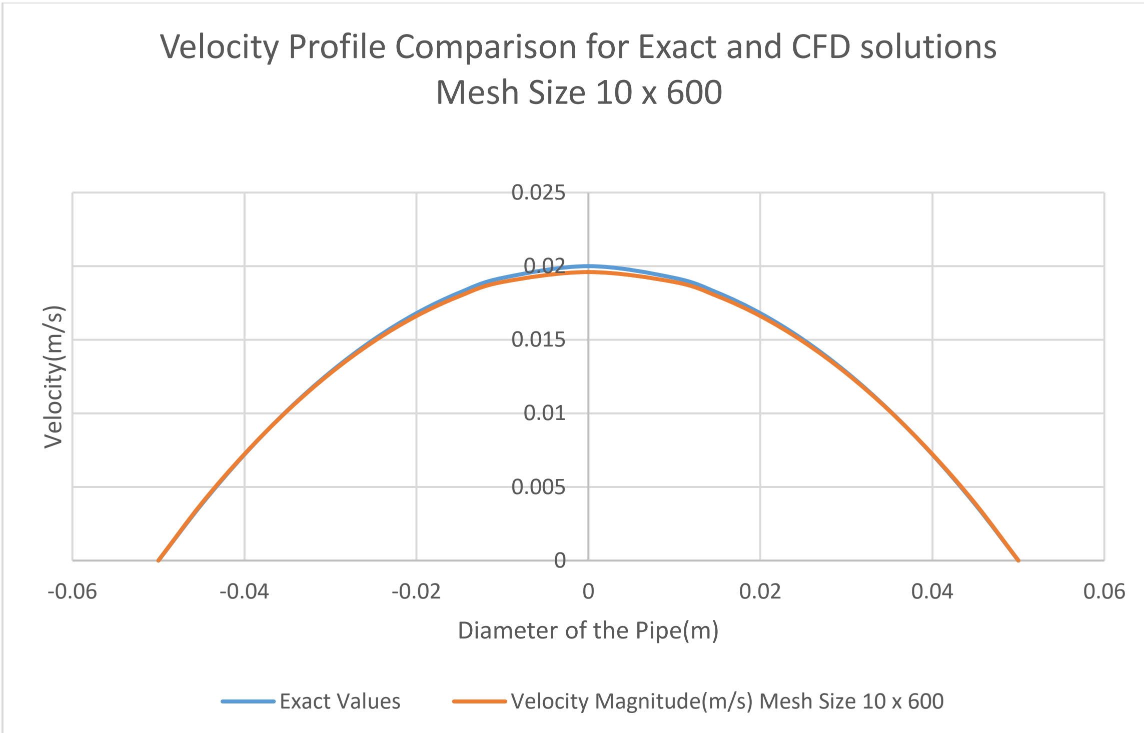


Fig. (5) Velocity Profile on Mesh Size 10 x 600 (Re=1000)

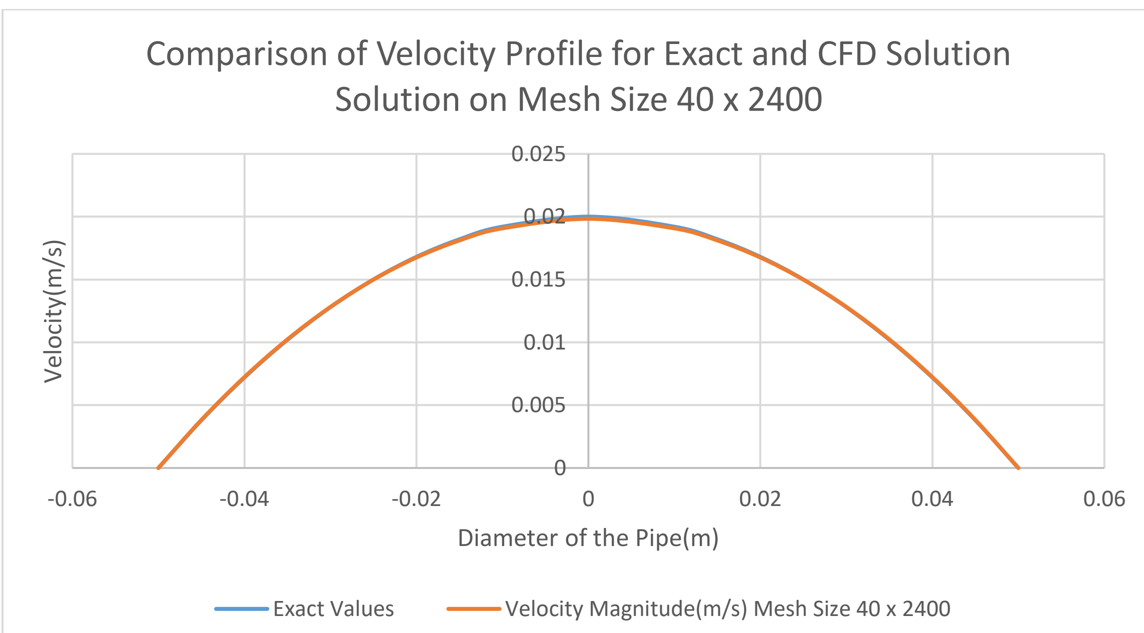


Fig. (6) Velocity Profile on Mesh Size 40 x 2400 (Re=1000)

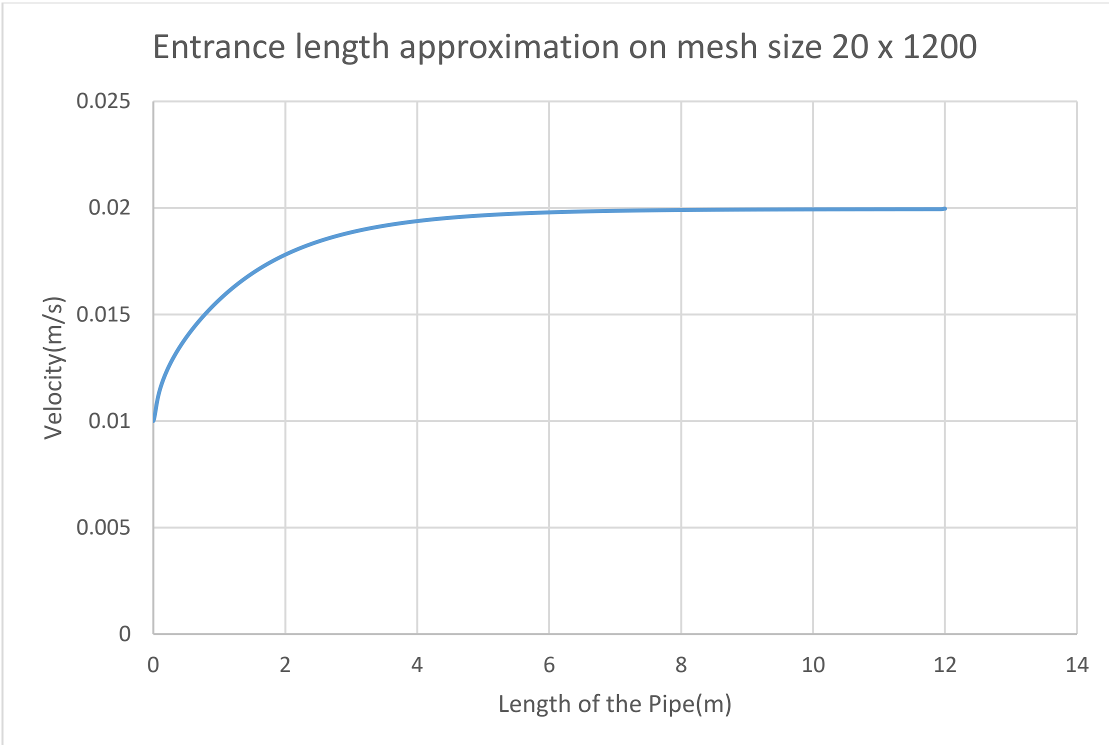


Fig. (7) Entrance Length on Mesh Size 20 x 1200 (Re=1000)

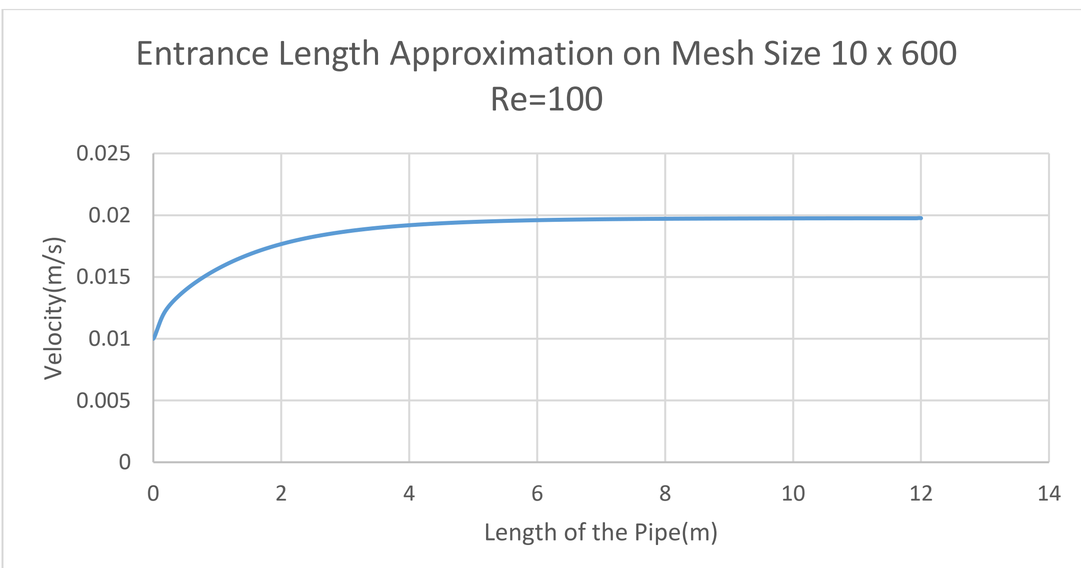


Fig. (8) Entrance Length on Mesh Size 10 x 600 (Re=1000)

`

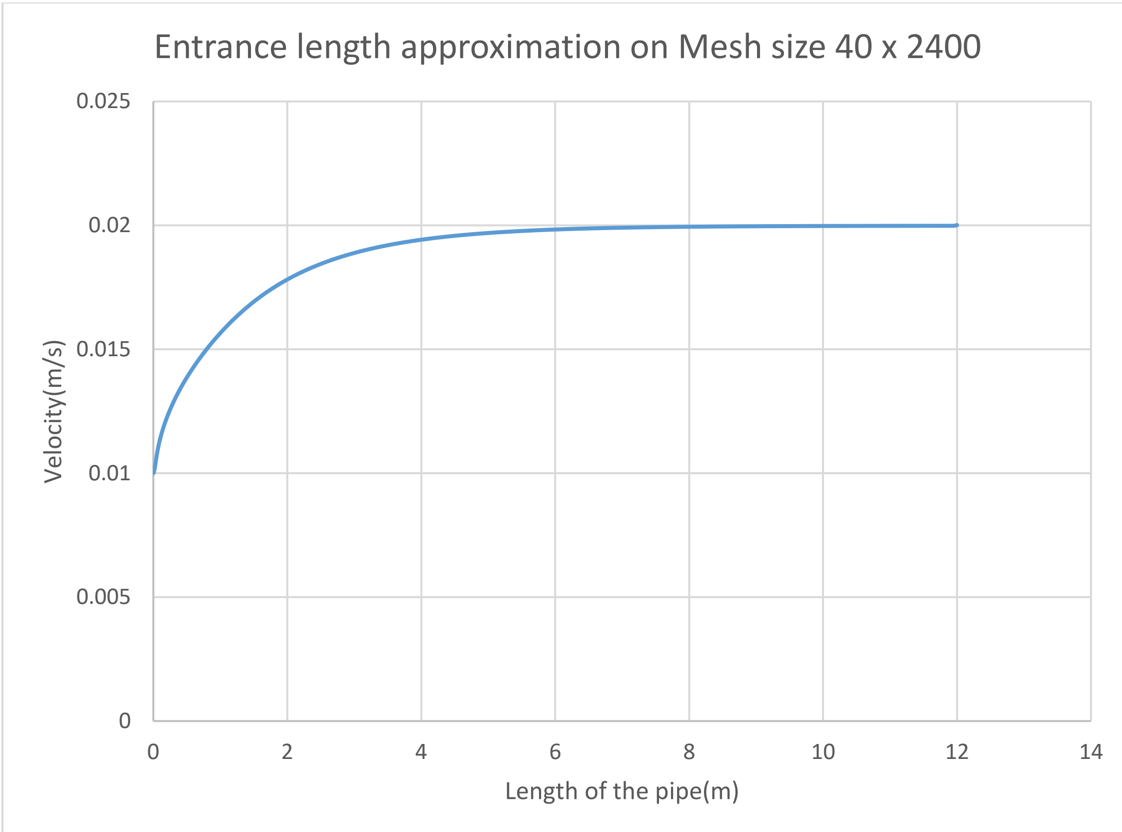


Fig. (9) Entrance Length on Mesh Size 40 x 2400 (Re=1000)

`

Comparison of Numerically calculated value with Empirical solution of Velocity Distribution on Reynold number= 100000:

X-Position	Empirical Value	CFD Solution Velocity 10x600	CFD Solution Velocity 20 x 1200	CFD Solution Velocity 40 x 2400
0.05	0	0	0	0
0.045	0.85830	0.872425	0.886505	0.889947
0.04	0.93760	0.970948	0.981066	0.982742
0.035	0.98400	1.03332	1.03861	1.03936
0.03	1.01690	1.08027	1.0808	1.08092
0.025	1.04240	1.11474	1.11399	1.1135
0.02	1.06330	1.14278	1.13996	1.13939
0.015	1.08090	1.16576	1.16035	1.15939
0.01	1.09620	1.17978	1.17476	1.17365
0.005	1.10970	1.1893	1.18331	1.18213
0	1.12170	1.19113	1.18588	1.18485
-0.005	1.10970	1.1893	1.18331	1.18213
-0.01	1.09620	1.17978	1.17476	1.17365
-0.015	1.08090	1.16576	1.16035	1.15939
-0.02	1.06330	1.14278	1.13996	1.13939
-0.025	1.04240	1.11474	1.11399	1.1135
-0.03	1.01690	1.08027	1.0808	1.08092
-0.035	0.98400	1.03332	1.03861	1.03936
-0.04	0.93760	0.970948	0.981066	0.982742
-0.045	0.85830	0.872425	0.886505	0.889947
-0.05	0.00000	0	0	0

Table. (3) Velocity Comparison on Different Mesh Sizes

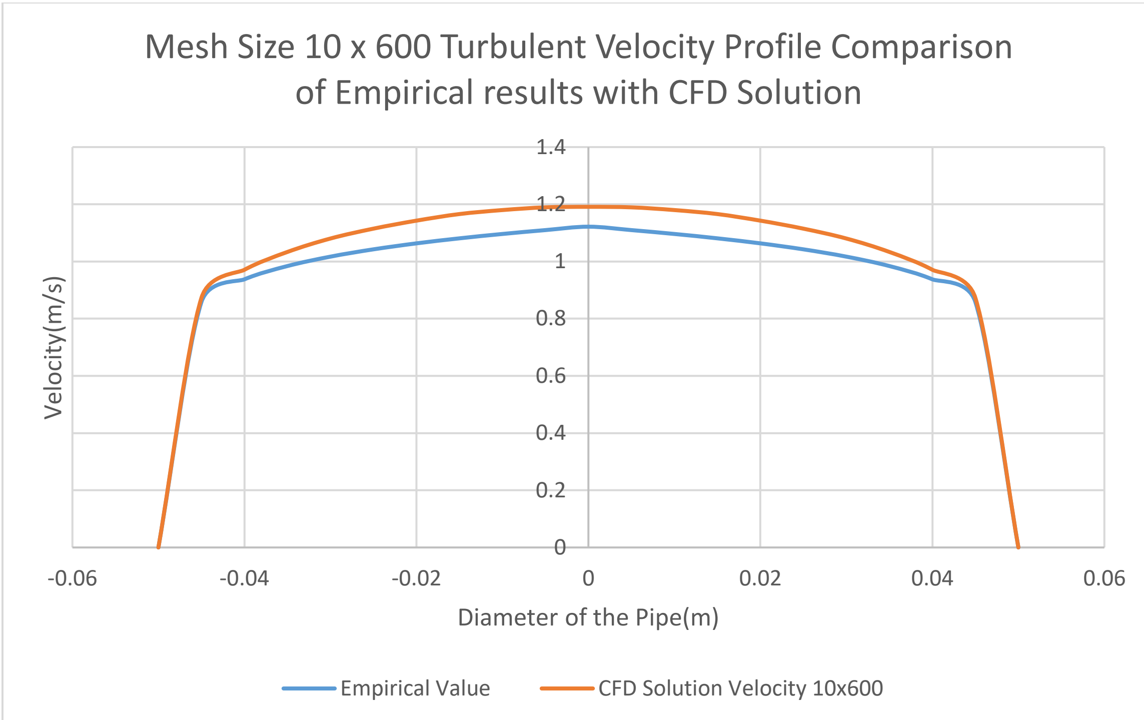


Fig. (10) Velocity Profile on Mesh Size 10 x 600 (Re=100000)

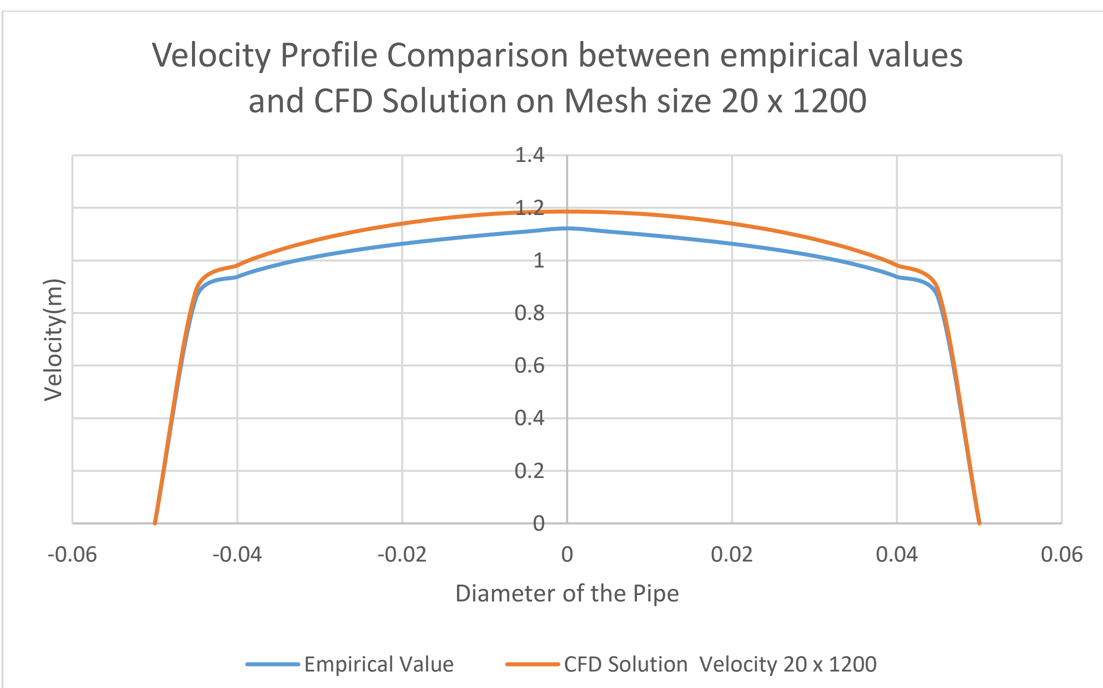


Fig. (11) Velocity Profile on Mesh Size 20 x 1200 (Re=100000)

`

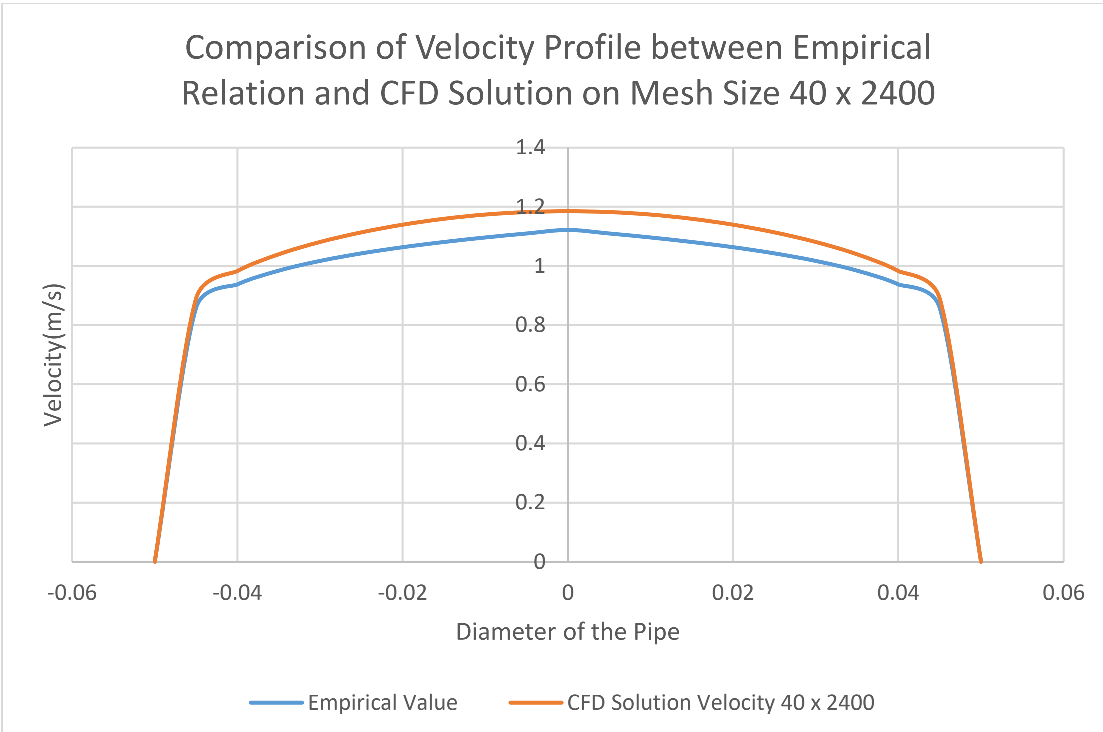


Fig. (12) Velocity Profile on Mesh Size 20 x 1200 (Re=100000)

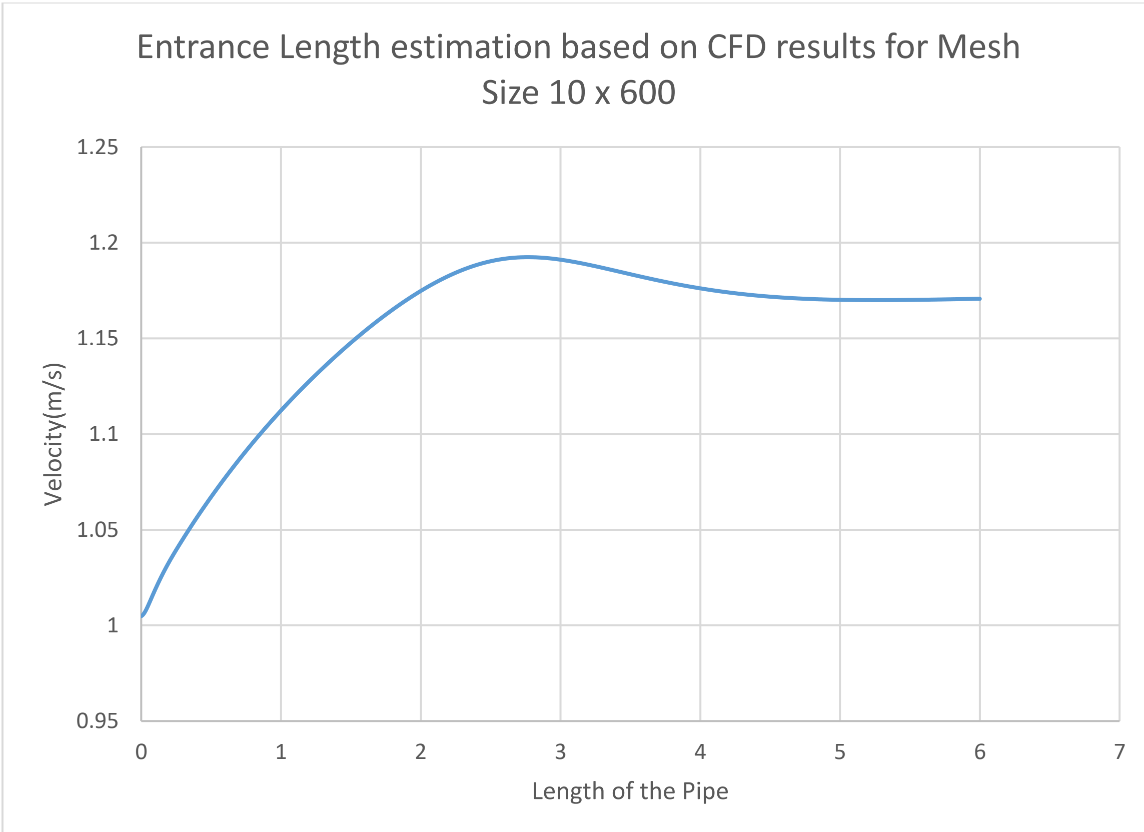


Fig. (13) Entrance Length on Mesh Size 10 x 600 (Re=100000)

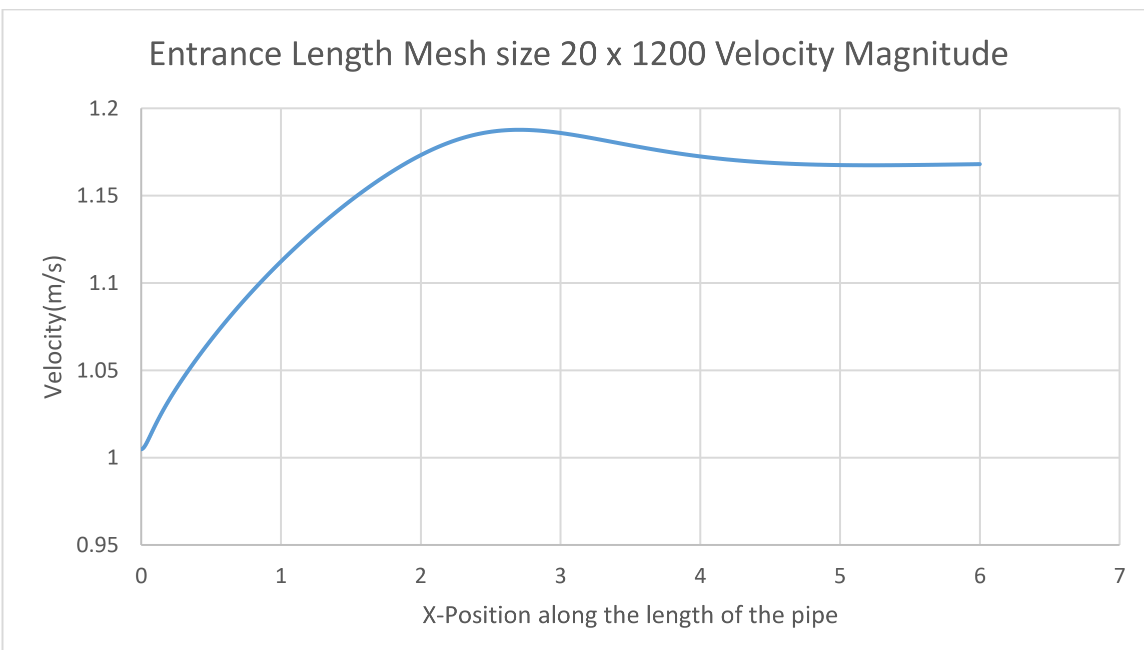


Fig. (14) Entrance Length on Mesh Size 20 x 1200 (Re=100000)

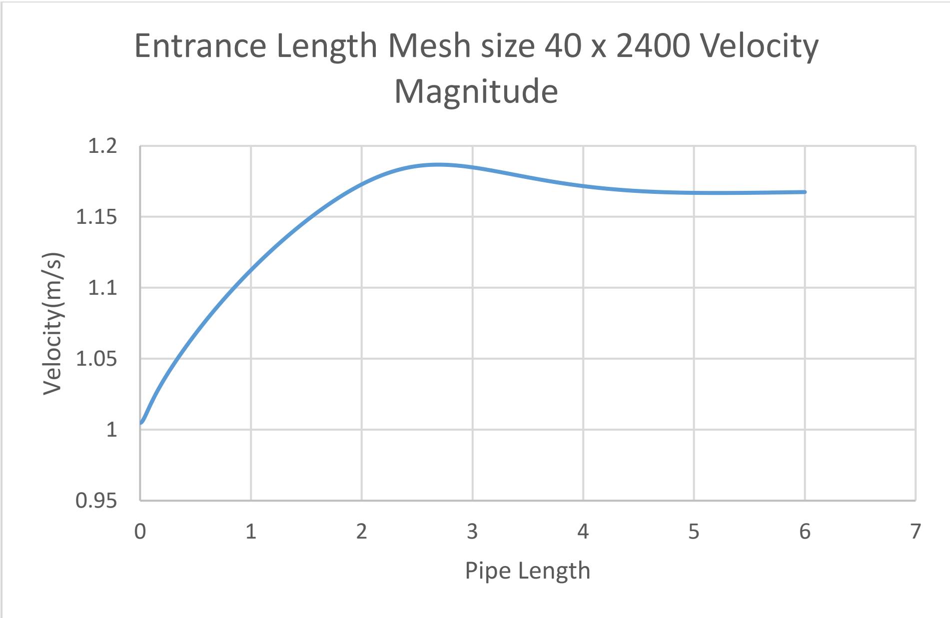


Fig. (15) Entrance Length on Mesh Size 40 x 2400 (Re=100000)

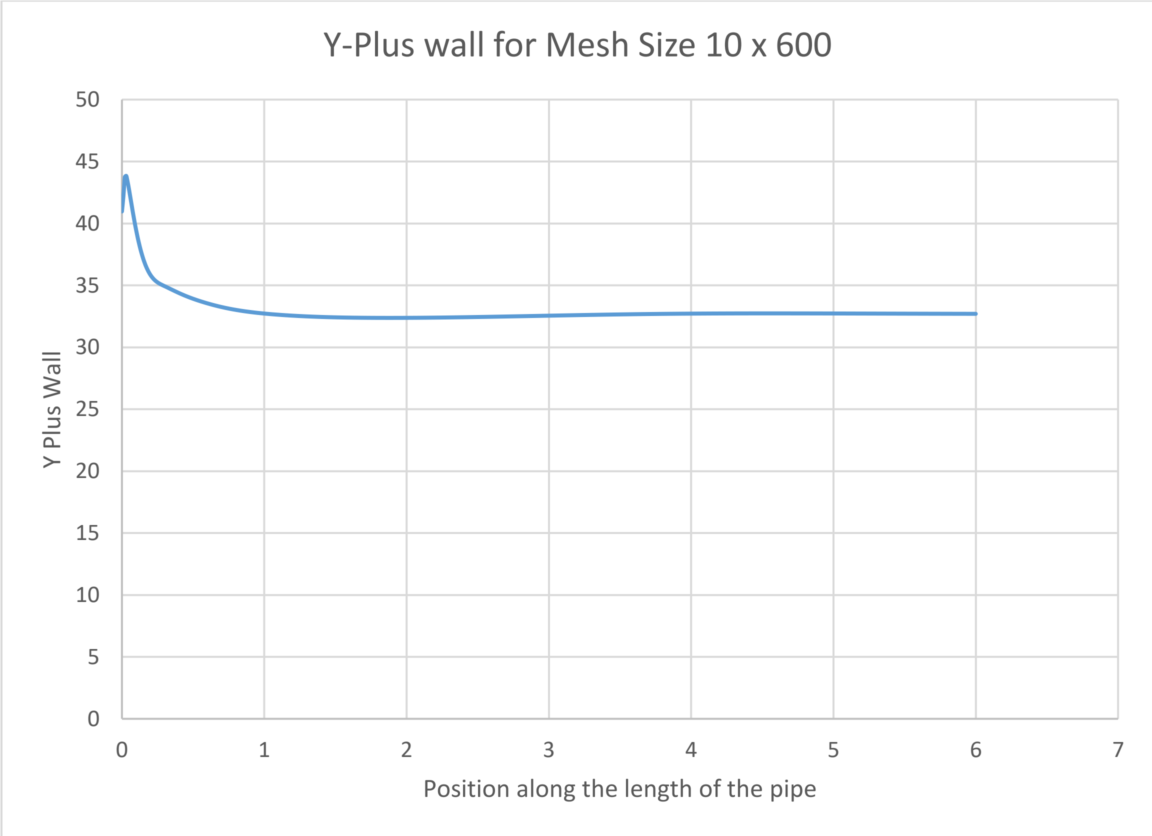


Fig. (16) Y-Plus Wall on Mesh Size 10 x 600 (Re=100000)

`

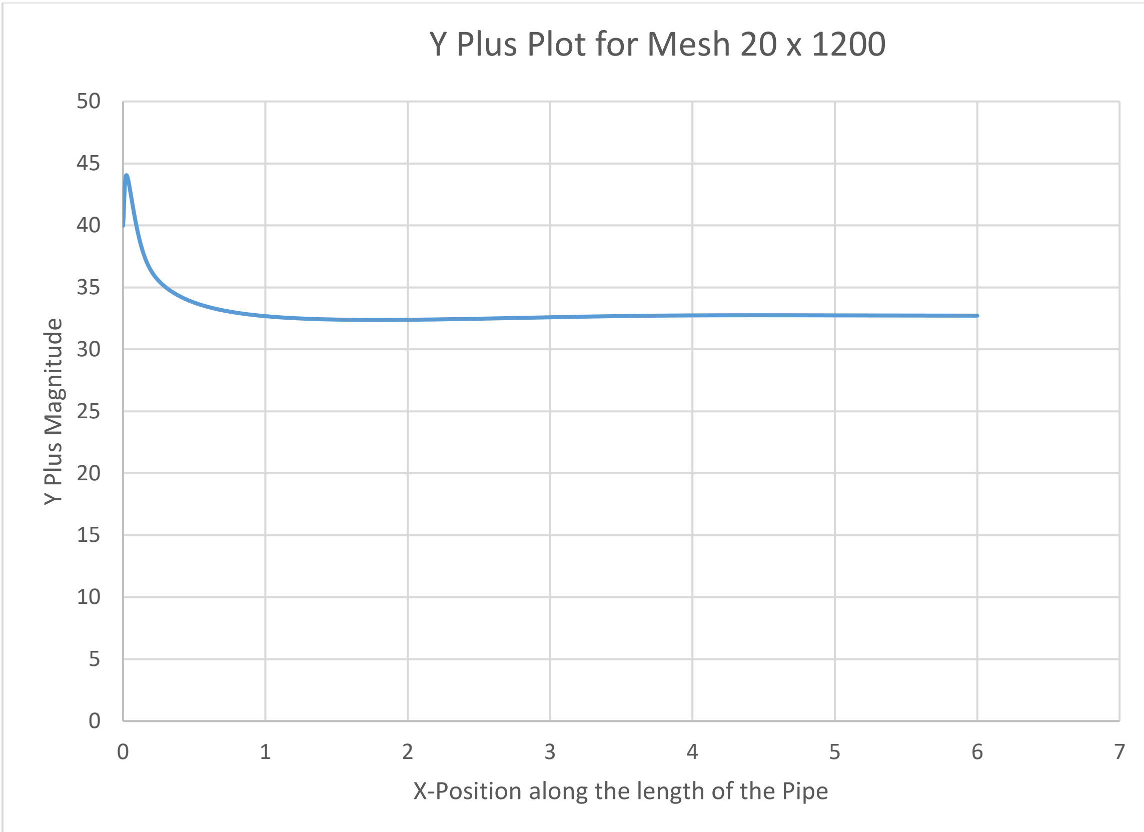


Fig. (16) Y-Plus Wall on Mesh Size 20 x 1200 (Re=100000)

`

Velocity Profile Empirical Velocity using Log Law Vs CFD		
Radial -Position	**Velocity**	**CFD Mesh size 20 x 2000**
0.05	0	0
0.045	91.959	90.9642
0.04	97.8371	96.3437
0.035	102.278	99.4905
0.03	104.812	101.7231
0.025	106.354	103.455
0.02	107.897	104.87
0.015	109.075	106.0663
0.01	109.712	107.1026
0.005	110.349	108.0168
0	110.415	108.8345
-0.005	110.349	108.0168
-0.01	109.712	107.1026
-0.015	109.075	106.0663
-0.02	107.897	104.87
-0.025	106.354	103.455
-0.03	104.812	101.7231
-0.035	102.278	99.4905
-0.04	97.8371	96.3437
-0.045	91.959	90.9642
-0.05	0	0

Table (4) Turbulent Velocity Profile for $Re=10^7$

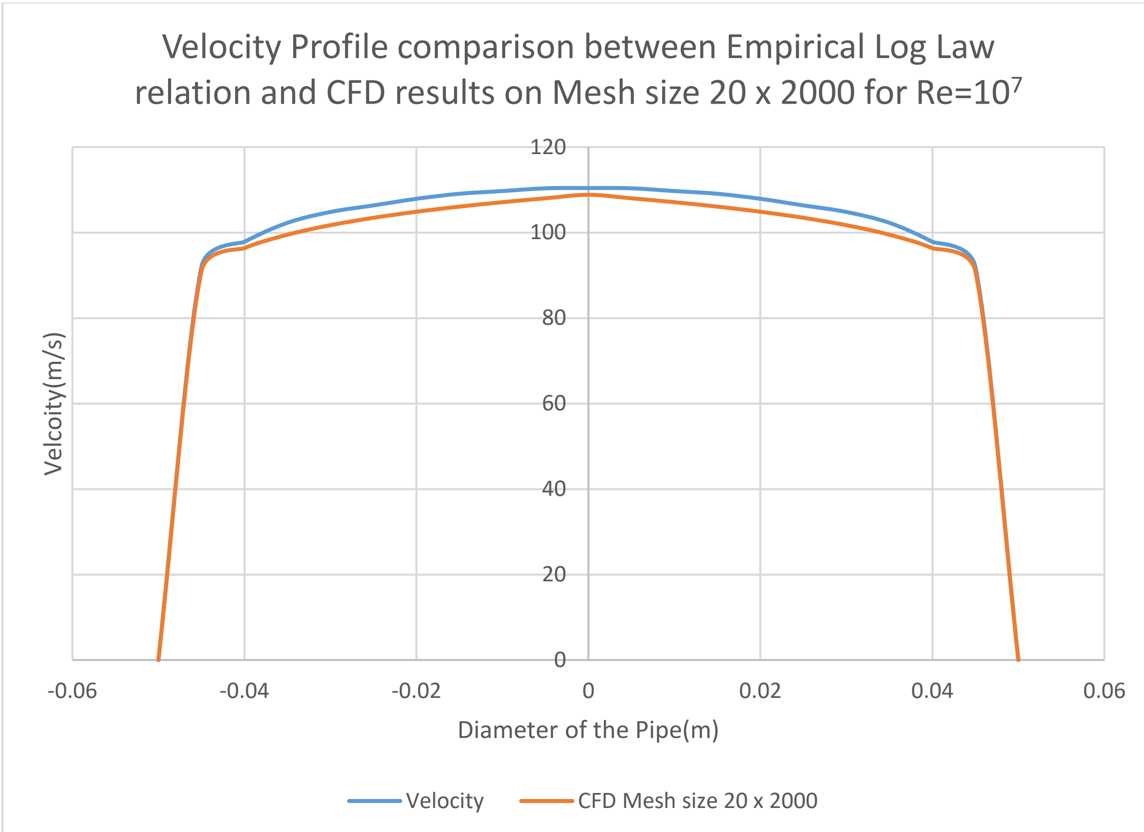


Fig. (17) Velocity profile Size 20 x 2000 (Re=10^7)

`

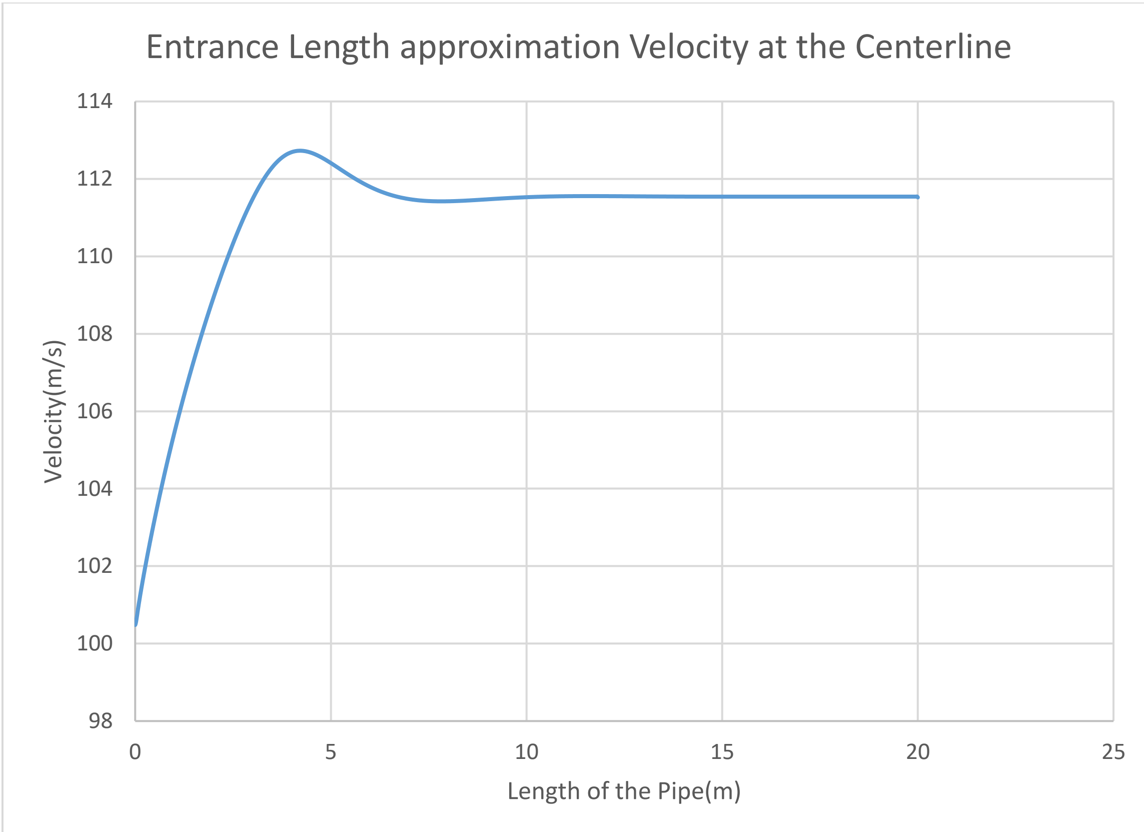


Fig. (18) Entrance Length Size 20 x 2000 (Re=10^7)

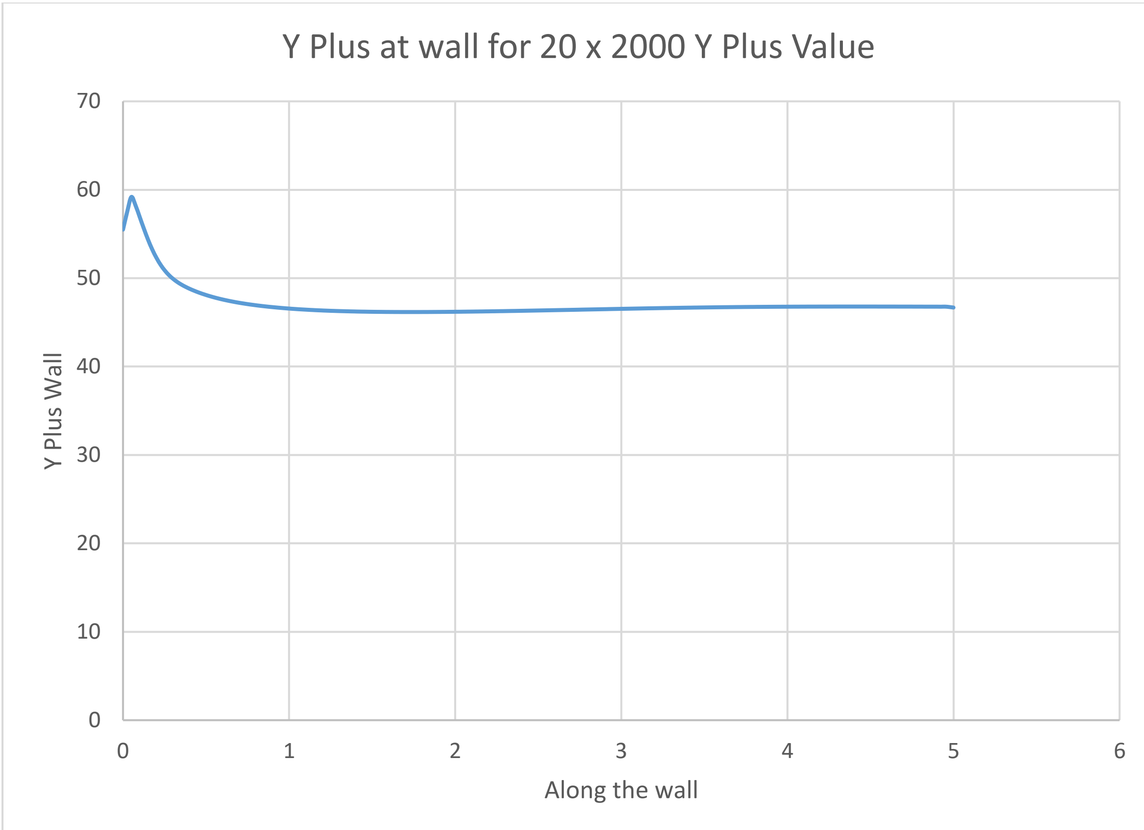


Fig. (19) Y Plus wall 20 x 2000 (Re=10^7)

`

Problem 2:

The second problem involves numerical calculation of the velocity and temperature of turbulent flow through 2-D pipe when Reynolds number is 10^6 with the following boundary conditions:

d) T_e=300 degree Celsius (Temperature of the fluid while entering the pipe)
e) T_0=25 degree Celsius (Uniform Temperature of the surface)
f)

Properties of the Pipe:

c) Diameter of the pipe= 0.1 Meter
d) Length of the pipe= 5 Meter

From the given statement, we can calculate the average velocity of the flow from the Reynold number

$$R_e = \frac{\rho v d}{\mu}$$

v=8.92 m/s

Analytical Entrance length:

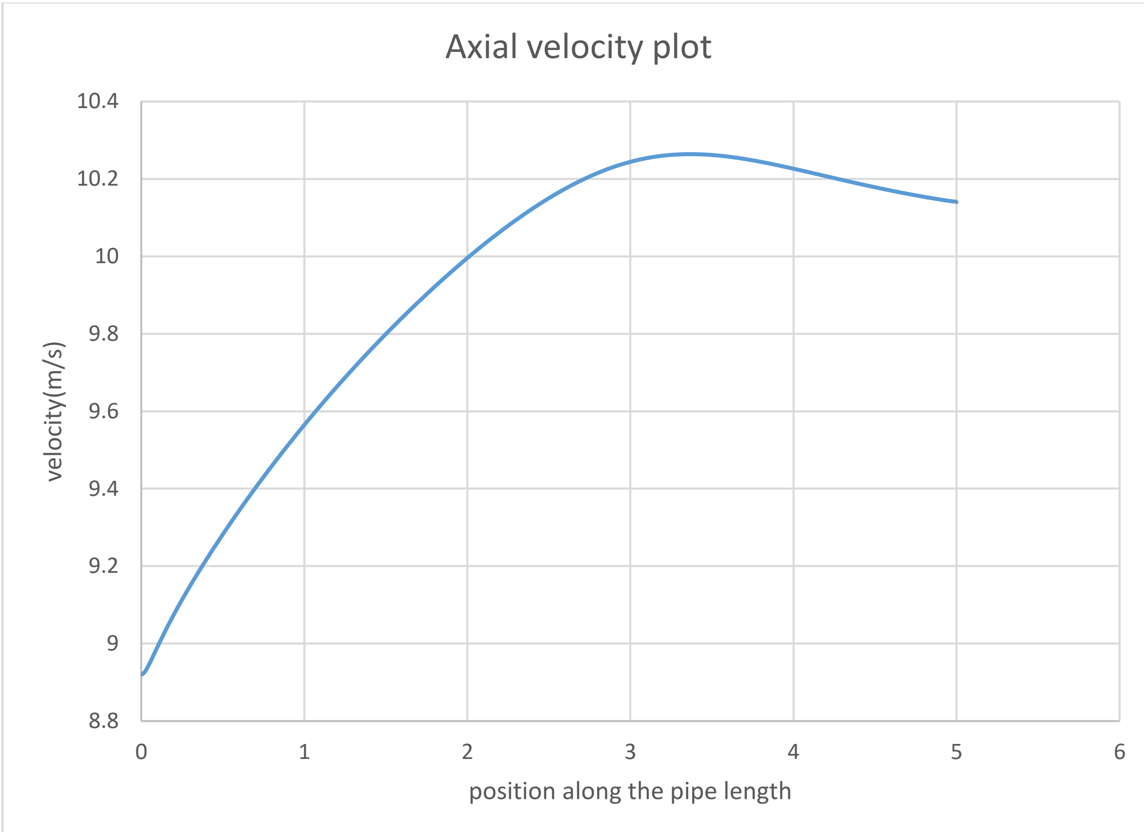
$$L_e = 4.4 * R_e^{\frac{1}{6}} * D$$
$$L_e = 4.4\ m$$

This Problem involves Turbulent flow with heat transfer so the numerical will be solved by K-E model and Energy equation. The upwind second order numerical scheme is used for both the governing equation for robustness and accuracy.

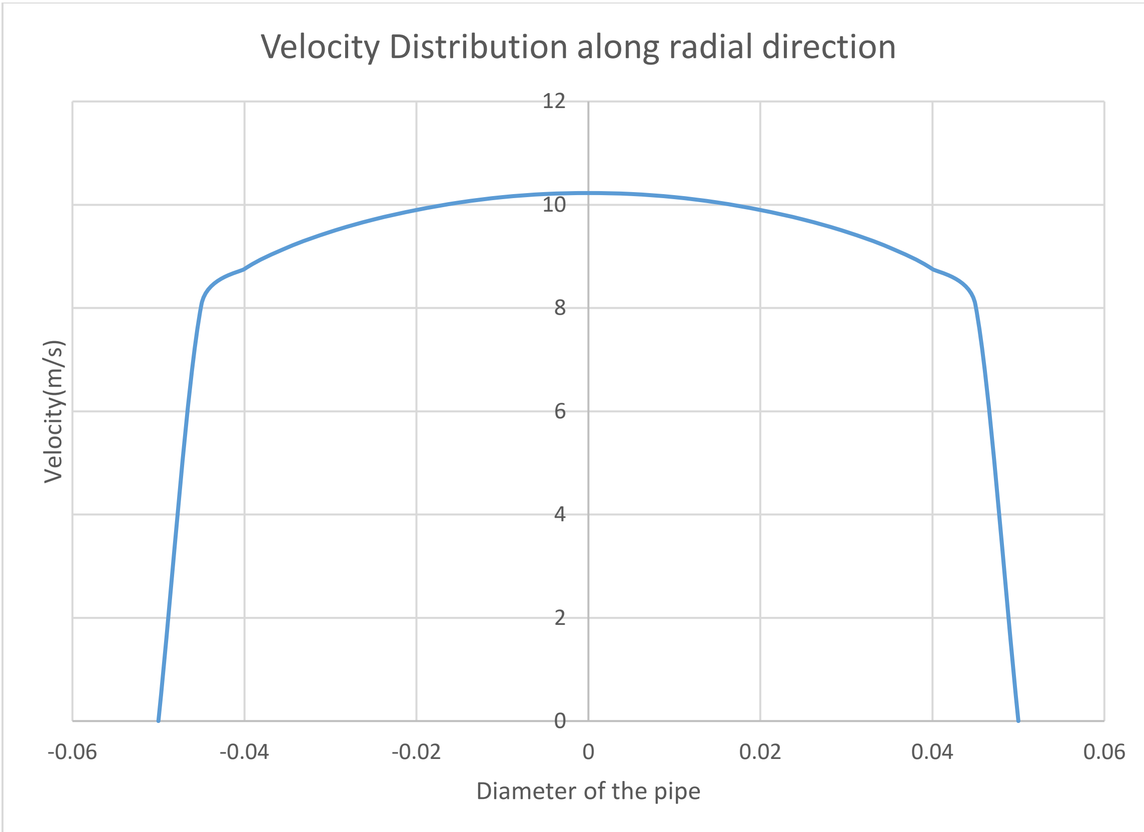
Boundary conditions:

a) T_e=300 degree Celsius (Temperature of the fluid while entering the pipe)
b) T_0=25 degree Celsius (Uniform Temperature of the surface)
c) Wall No Slip Boundary condition
d) Inlet Velocity 8.92 m/s
e) Outlet: outflow type

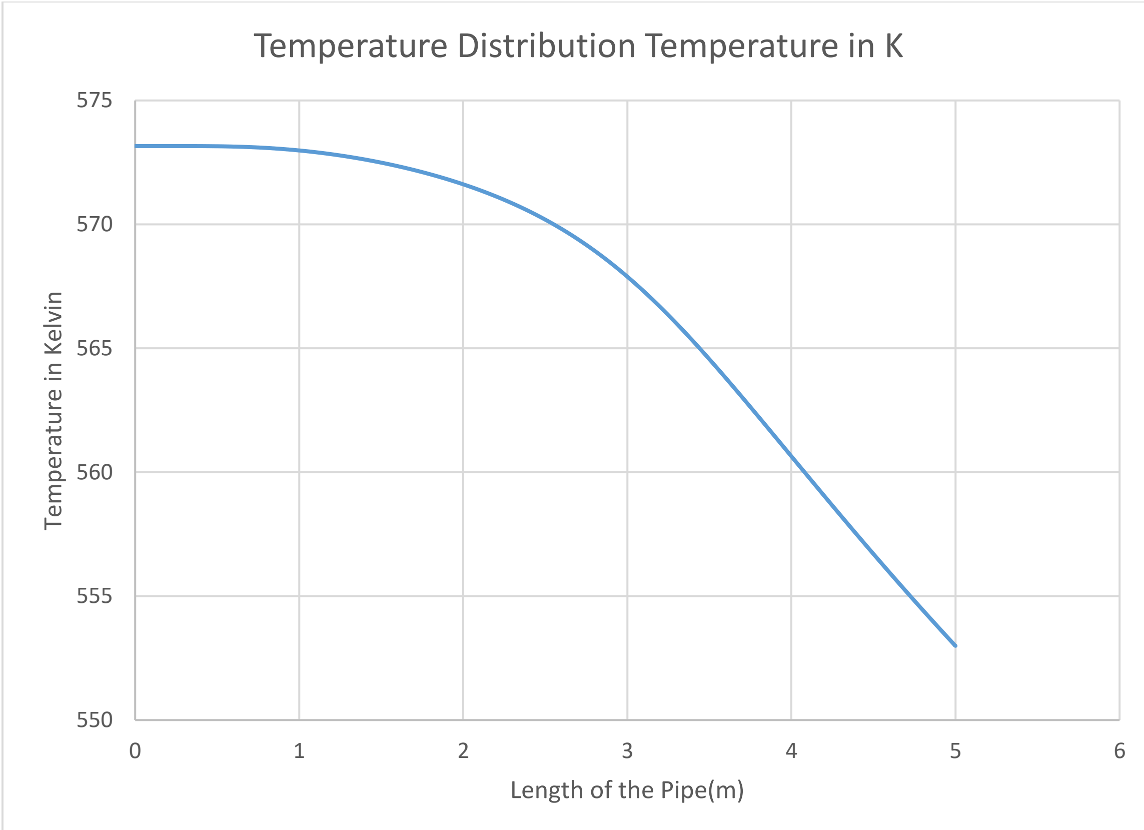
`



The above axial velocity plot gives the information regarding the development of flow inside the pipe after 4.5 m length of the pipe the velocity remains constant which is very close to what we have obtained through analytically. However, to attain full constant value we must increase the length of the pipe.



The above distribution of velocity profile in the radial direction of the pipe looks very similar to velocity profile for turbulent cases that can be calculated by empirical relations. The maximum velocity attained at the centerline of the pipe is **10.26 m/s**



From the above trend of the temperature distribution one can say the temperature is decreasing with the increasing length of the pipe because temperature flows from center to walls. walls are at low temperatures. The minimum temperature is at 5m length around **552 k** while the maximum is at inlet **573.15 k.**

`

Name: ______________________ Assignment NO.________

	MARKS	MAX
General Organization		5
Quality of Presentation		5
Technical Writing		5
Problem Description		5
Mathematic Models		10
Numerical Procedure		10
Grid/Time Step Independent Tests		10
Analysis of Results		20
Discussion of Results		20
Conclusions		5
References		5
TOTAL		100