NUMERICAL COMPUTATION OF COMPRESSIBLE LAMINAR FLOW WITH HEAT TRANSFER IN THE ENTRANCE REGION OF A PIPE

M. Ziaei Rad\textsuperscript{1,2}, A. Nouri- Broujerdi\textsuperscript{2}, J. Seume\textsuperscript{3}

\textsuperscript{1,2} School of Mechanical Engineering, Sharif University of Technology, P.O. Box: 11365-9567, Tehran, Iran
Email: ziaeirad@mech.sharif.edu

\textsuperscript{1,3} Institute of Turbomachinery and Fluid Dynamics-TFD
Leibniz University Hannover, Appelstraße 9, 30167, Hannover, Germany
Email: seume@tfd.uni-hannover.de

ABSTRACT

The authors' research work on pressure drop along gas transmission pipelines raised questions regarding the development length of the corresponding compressible flow and the effect of heat transfer in the entrance region on the pressure drop along the whole length of the pipe. In this paper, laminar, viscous, compressible flow in the entrance region of a pipe is investigated numerically in two dimensions. The numerical procedure is a finite-volume based finite-element method applied on unstructured grids. This combination together with a new method applied for boundary conditions allows accurate computation of the variables in the entrance region. The method is applied to some incompressible cases in order to verify the results. The results are confirmed by previous numerical and experimental research on the developing length in incompressible flow.

INTRODUCTION

There are many applications of pipe flow in which the duct length may be too short for establishing fully developed conditions, such as connecting pieces, short pipes or short length heat exchangers. Although in the majority of the mentioned applications the fluid flow is incompressible, knowledge of the entrance region flow parameters are necessary in the compressible regime, especially when the heat transfer effect on density change and boundary layer development is considered. This change of parameters in the entrance region may also affect some parameters such as pressure drop along the whole length of the duct. An important application is the increase in capacity of natural gas transport and distribution networks, which depends on gas pressure loss. The flow is compressible due to the significant change in density and the local Mach number increase towards the pipe exit, possibly reaching critical conditions if the pipe length is sufficiently long.

In the study of external flow over a body, the relationship between the wall heating and the change of skin friction drag, which is caused by the difference in viscosity and density of a fluid when it is heated, can easily be seen to be proportional to the temperature ratio taken to the power of \(-2/3\) \cite{1}. On the other hand, simple calculations on the momentum equation of incompressible gas flow through a pipe show that for a constant pressure drop, the mass flow rate is a function of inflow temperature taken to the power of \(-2.5\). This means that by increasing the inflow temperature, the mass flow rate will be decreased considerably.

When the pressure drop due to the flow of the gas is large enough, causing a considerable decrease in density, then the flow may be considered to be compressible and appropriate formulas that take into consideration changes in both density and velocity must be used to describe the flow.

Many attempts have been made by researchers to reduce the pressure loss along the gas pipelines in order to reduce the cost of transportation, such as lining the gas pipelines with smooth coatings to reduce the frictional pressure drop \cite{2}; using chemicals, for example in cold weather when the pipeline capacity needs to be increased for a period of time \cite{3}; applying condensation systems when a liquid fraction is present; using new surfaces, i.e. microscopic structures (e.g. ‘riblets’ \cite{4}) on pipeline walls in some cases to simulate natural surfaces,
resulting in pressure loss reduction. However, one of the main effects on pressure loss are the effects of environmental heat transfer or the temperature of the gas injected into the pipe from a compressor in the compressor station. On the other hand, since the entire length of transport pipelines cannot be studied with a two-dimensional analysis and also since radial variations are important only in the developing region of flow field, most of the research done on long pipelines uses one-dimensional analysis [5-7]. Two-dimensional analyses, however, are necessary to model the development length.

A large number of investigators have attempted to analyze the flow in the entrance of a pipe or channel analytically in the past. Sparrow [8] presented a method for calculating the laminar flow development and pressure drop in the entrance regions of ducts of any cross section including the application to a circular tube. Mohanty and Asthana [9] solved the Navier-Stokes equations in the inlet region and the with an order-of-magnitude analysis using fourth-degree velocity profiles obtained the total length of the entrance region. Their results were for laminar supersonic flows and used boundary layer assumptions. Gupta [10] presented an integral method for solving the flow in the inlet region of a tube which consists in replacing the term representing the shear stress at the tube wall in the momentum integral equation by integrals involving unknown velocity distribution in the inlet region.

Numerical simulation of internal compressible flow was also the goal of many researchers over the years and several algorithms have been presented. Hornbeck [11] used a simple finite difference method to analyze the laminar flow of an incompressible fluid in the inlet of a pipe numerically, achieving good agreement with the results of solutions of more simplified sets of equations used in previous investigations. Schmidt and Zeldin [12] also tried to solve the set of Navier-Stokes equations using a finite difference technique, but they used the stream function and the vorticity to reduce the number of equations and eliminate the pressure terms. In a recent numerical study by Durst et al. [13] investigating the development length of steady, laminar pipe and channel flows, the diffusion transport in axial direction was considered, and a review of relevant work available in the literature was presented. Chant [14] developed an analytical skin friction model for compressible, turbulent, internal flow involving adiabatic and non-adiabatic flows by extending the incompressible law-of-the-wall relation to compressible cases, but only fully developed flow was considered. Gato and Henriques [15] studied the numerical modeling of the dynamic behavior of high-pressure natural-gas flow in pipelines by one-dimensional compressible flow analysis. The occurrence of pressure oscillations in natural gas pipelines was studied as a result of the compression wave originated by the rapid closure of downstream shut-off valves. Keith and Crowl [16] presented a review and analysis of sonic gas flow in pipelines, particularly for long pipelines in one-dimension. Their results show that the mass flow rate is asymptotic as the velocity head pipe loss increases and the asymptotic value is identical for both adiabatic and isothermal conditions. Mary et al. [17] proposed a second-order accurate algorithm for the simulation of unsteady viscous stratified compressible flows, with a capability of dealing with a broad range of subsonic Mach numbers, including nearly incompressible flows with a single modeling approach, but the application to developing flows and heat transfer effects were not considered. Solution of choked flow of air through a narrow parallel plate channel was investigated by means of finite-difference numerical calculation by Shi et al. [18], but it was for low-density gases and only adiabatic wall condition was taking into account. Toutopoulos and Ekaterinaris [19] investigated the application of second- and fourth-order accurate discontinuous Galerkin finite element method for the numerical solution of the Euler and the Navier-Stokes equations with triangular meshes, but without special considerations on internal flows and heat transfer. Xu et al. [20] presented a finite volume formulation for large eddy simulation of turbulent pipe flows based on the compressible time-dependent three-dimensional Navier-Stokes equations in Cartesian coordinates, only for fully developed flows and focused on momentum equations. Aydin [21] studied specially the effects of viscous dissipation on heat transfer in thermally developing laminar forced convection in a pipe with both, constant heat flux and the constant wall temperature boundary.

There are also some experimental studies in the previous literature in the field of compressible flow. Mekebel and Loraud [22] performed experimental research on unsteady flows and pressures in a long gas transmission pipeline and concluded that heat transfer was a necessary inclusion in the theoretical analysis, contrary to the common assumptions of isothermal or adiabatic flow. In further research on experimental pipe flow [23], existing theoretical analysis was compared with some experiments on compressible pipe flow subjected to wall friction, but this research focused on an experimental setup for special geometries such as a converging-diverging nozzle while the effects of heating in various flow regimes was not widely considered. Another numerical and experimental study was made on internal compressible flow at T-type junctions, also without considering the heat transfer effects [24]. In an experimental and numerical investigation of subsonic, compressible flow in long micro conduits [25], it was suggested that the locally fully developed approximation could be used to interpret the experimental data for low and moderate Mach number flow.

In all of the above mentioned papers and other previous work, consideration was focused on laminar incompressible flows in analytical studies and in numerical and experimental works. Only hydrodynamic developing length was considered and most of the results were presented for incompressible flow in the entrance region while no research could be found on the details of heat transfer effects on pressure loss in compressible developing pipe flow. The lack of such numerical and experimental research in long pipes requires that numerical methods are applied to this problem and their results be verified experimentally. The first of these steps is attempted in the present paper.

**NOMENCLATURE**

\[ A \quad \text{cell area, Jacobian matrix} \]

\[ b \quad \text{heat flux plus work of frictional forces} \]

\[ C \quad \text{triangle area} \]
DESCRIPTION OF THE PROBLEM AND THE GOVERNING EQUATIONS

The problem of internal flow development in a circular pipe is modeled as shown in Fig. 1. The fluid is assumed to be an ideal gas. For gas flow with variable properties such as density and viscosity, the compressible Navier-Stokes equations are applicable even if a low-speed case is dealt with. For the present investigations, a steady laminar and developing flow was considered without swirl \( (u_\theta = 0) \) and fluid properties were assumed to be constant, except the viscosity which can change with temperature.

The governing equations are made dimensionless using the inflow bulk velocity \( V_0 \), density \( \rho_0 \), viscosity \( \mu_0 \) and pipe diameter \( D \).

\[
\begin{align*}
\frac{z^*}{D} &= z/D, & \frac{r^*}{D} &= r/D, \\
\frac{u^*}{V_0} &= u/V_0, & \frac{\rho^*}{\rho_0} &= \rho/\rho_0, \\
\frac{t^*}{tV_0/D} &= tV_0/D, & \frac{\mu^*}{\mu_0} &= \mu/\mu_0, \\
\frac{q^*}{(\rho_0 V_0^2)} &= q^*/(\rho_0 V_0^2)
\end{align*}
\]

where \( \text{Re} = \rho_0 V_0 D / \mu_0 \) is the Reynolds number. The superscript '*' here denotes the dimensionless variables and the subscript '0' denotes values at a reference state. We drop the superscript '*' for simplicity after this, so all of the parameters used in this paper will be dimensionless, except those specifically mentioned. Therefore, the conservative form of the dimensionless governing equations in the axi-symmetric coordinates system are as follows.

Conservation of mass:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_z)}{\partial z} + \frac{1}{r} \frac{\partial (\rho u_r)}{\partial r} = 0
\]
Conservation of momentum:

\[
\frac{\partial (\rho u_z)}{\partial t} + \frac{\partial (\rho u_z^2 + p)}{\partial z} + \frac{1}{r} \frac{\partial (r \rho u_z u_r)}{\partial r} = \frac{\partial \tau_{zz}}{\partial z} + \frac{1}{r} \frac{\partial (r \tau_{rz})}{\partial r}
\]  

(3)

\[
\frac{\partial (\rho u_r)}{\partial t} + \frac{\partial (\rho u_z u_r)}{\partial z} + \frac{1}{r} \left[ \frac{\partial r (\rho u_z^2 + p)}{\partial r} \right] = \frac{\partial \tau_{rz}}{\partial z} + \frac{1}{r} \frac{\partial (r \tau_{rr})}{\partial r} - \tau_{\theta\theta}/r
\]  

(4)

Conservation of energy:

\[
\frac{\partial (\rho e_z)}{\partial t} + \frac{\partial [(\rho e_z + p) u_z]}{\partial z} + \frac{1}{r} \left[ \frac{\partial r (\rho e_z + p) u_r}{\partial r} \right] = \frac{\partial b_z}{\partial z} + \frac{1}{r} \frac{\partial (r b_r)}{\partial r}
\]  

(5)

In the above equations, \( u_z, u_r \) are the axial and radial velocity components respectively and \( e_z \) is the total internal energy and is defined based on the perfect gas law as:

\[
e_z = \frac{p}{\rho (\gamma - 1)} + \frac{1}{2} (u_z^2 + u_r^2)
\]  

(6)

Heat flux and surface work done by frictional forces terms in energy equation are defined as:

\[
b_z = \left( \frac{\mu}{Pr} \right) \frac{\partial T}{\partial z} + u_z \tau_{zz} + u_r \tau_{zr}
\]

\[
b_r = \left( \frac{\mu}{Pr} \right) \frac{\partial T}{\partial r} + u_z \tau_{rz} + u_r \tau_{rr}
\]  

(7)

The stress terms for the tensor components are:

\[
\tau_{zz} = \mu \left[ \frac{2}{3} \left( \nabla \cdot \vec{V} \right) \right]
\]  

(8)

\[
\tau_{zr} = \mu \left[ \frac{2}{3} \left( \nabla \cdot \vec{V} \right) \right]
\]  

(9)

\[
\tau_{rr} = \mu \left[ \frac{2}{3} \left( \nabla \cdot \vec{V} \right) \right]
\]  

(10)

\[
\tau_{\theta\theta} = \mu \left[ \frac{2}{3} \left( \nabla \cdot \vec{V} \right) \right]
\]  

(11)

where

\[
\nabla \cdot \vec{V} = \frac{\partial u_z}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} (ru_r)
\]  

(12)

The temperature-dependent laminar viscosity is based on the Sutherland law:

\[
\mu = \mu_0 \left( \frac{T}{T_0} \right)^{3/2} \left( \frac{T_0 + T_s}{T + T_s} \right)
\]  

(13)

where \( T_s \) is the constant Sutherland temperature.

Looking at the conservative form of the governing equations, we note that they all have the same generic form, given by

\[
\frac{\partial Q}{\partial t} + \nabla \cdot F(Q) = \nabla \cdot N(Q) + S(Q)
\]  

(14)

\( Q, F(Q), N(Q) \) and \( S(Q) \) are the conservative variables, the inviscid flux vector, the viscous flux vector, and the source vector respectively, with

\[
Q = [\rho \quad \rho u_z \quad \rho u_r \quad \rho e_z]^T
\]  

(15)

The inviscid flux vector \( F(Q) \) includes components \( F_z(Q) \) and \( F_r(Q) \) with the following expressions

\[
F_z(Q) = [\rho u_z \quad \rho u_z^2 + p \quad \rho u_z u_r \quad (\rho e_z + p) u_z]^T
\]

\[
F_r(Q) = [\rho u_r \quad \rho u_z u_r \quad \rho u_r^2 + p \quad (\rho e_z + p) u_r]^T
\]  

(16)

The viscous flux vector \( N(Q) \) also includes \( N_z(Q) \) and \( N_r(Q) \) with the following forms.

\[
N_z(Q) = [0 \quad \tau_{zz} \quad \tau_{zr} \quad b_z]^T
\]

\[
N_r(Q) = [0 \quad \tau_{rz} \quad \tau_{rr} \quad b_r]^T
\]  

(17)

The source vector for axi-symmetric flow is written as

\[
S(Q) = [0 \quad 0 \quad -\tau_{\theta\theta}/r \quad 0]^T
\]  

(18)

Having solved the set of governing equations, the distribution of the thermodynamic variables of gas flow in the pipeline is found.

**NUMERICAL TECHNIQUE**

The Navier-Stokes equations for the compressible viscous flow are solved by a finite-volume Galerkin upwind technique using the Roe [26] Riemann Solver for the convective part and the standard Galerkin technique for the viscous terms. This combination enables us to simulate a wide range of flow fields going from subsonic to supersonic flow for two-dimensional axi-symmetric compressible flows. As different computational grid sizes are required in each direction and at each position in the entrance, unstructured meshes appear to be the best tool to
capture reality and are also suitable for future developments of the code for turbulent cases.

Assume \( \Omega = k_j C_j = k_i A_i \) to be a discretization of the computational domain \( \Omega \) by triangles (or hexagonal cells) where \( C_j \) is the triangle area and \( k_j \) is the number of triangles. \( A_i \) is also the cell area and \( k_i \) is the number of cells. We suppose that \( F \) vector (Eq. 16) varies linearly from one side of each triangle to the other one.

Figure 2 shows the triangulation of computational domain and computational nodes. The variables will be computed on nodes denoted by subscript \( h \). If they are the vertices of triangle elements, these nodes are on the finite element grid. Otherwise, they are related to the control volume in the center of the hexagonal finite volume grid.

![Computational nodes](image.png)

**FIGURE 2: TRIANGULATION OF COMPUTATIONAL DOMAIN \( \Omega \)**

The weak finite element formulation of governing equation (14) without the source term can be written as

\[
\int_{\Omega} \frac{\partial Q_i}{\partial t} \phi_i dA + \int_{\Omega} \nabla (F_i - N_i) \phi_i dA = 0
\]

(19)

where \( \phi \) is the shape function which is set to be one in finite volume calculations, however, in the finite element formulation it is computed from the geometry and is used to compute the derivatives which appear in the viscous terms on each triangle.

Changing the integral form of the viscous term \( N \) using the partial integration method and retaining the convective term \( F \) in its original form with shape function set to one for this term, results in

\[
\int_{\Omega} \frac{\partial Q_i}{\partial t} \phi_i dA + \int_{\Omega} \nabla F_i \phi_i dA + \int_{\Omega} N_i \nabla \phi_i dA - \int_{\Omega} n \phi_i dC = 0
\]

(20)

which is called the weak form of Navier-Stokes equations in finite-element formulation. The first term is the time-dependent one. The second and third terms respectively show the variation of inviscid and viscous parts on the cell. The fourth term is associated with the boundary treatment.

Using explicit time integration and introducing divergence theorem for convective part, the final formulation is obtained as follows.

\[
[A] Q^{n+1} - Q^n \frac{\Delta t}{\Delta t} + \int_{\Omega} F_i \cdot n dC = -\int_{\Omega} N_i \nabla \phi_i dA + \int_{\Omega} n \phi_i dC
\]

(21)

where the superscripts \( n \) and \( n+1 \) denotes the old and new time steps respectively. A centered scheme is used to compute the viscous part on each cell. The source term is computed explicitly.

The difference between the cylindrical and Cartesian coordinates’ terms in weak form of the Navier-Stokes equation (20) comes from the differential area in the integrals (\( rdrdz \) replaced by \( drdz \)). So we multiply the cell areas, \( |A| \), the triangle areas, \( |C| \), and the edge lengths of the computational domain by some radius \( r \) obtained from the radius of the nodes \( r_i \). The other modifications come from the additional source terms occurring in the governing equation. Hence, if we transfer the - compared to the Cartesian form - additional terms of the cylindrical operator to the right hand side of the governing equation, we can solve the left hand side of the equation based on a solution of the two dimensional Cartesian coordinate algorithm.

The second term on the right-hand side of equation (21) is related to boundary condition which is set to be zero herein and the boundary condition will be applied in the finite volume formulation of convective terms. The convection term on the left hand side is evaluated by finite volume Roe method [26] on control volume surfaces which are the sides of hexagonal shape. In this way the flux vector across these planes is computed from

\[
F = \frac{1}{2} (F_L + F_R) - \frac{1}{2} R |A| \Delta Q
\]

(22)

where \( R \) is the eigenvector matrix of the Jacobian matrix \( A = \frac{\partial F}{\partial Q} \). \( F_L \) and \( F_R \) are the flux vectors computed from the right and left states of \( Q_L \) and \( Q_R \), and \( \Delta Q = Q_R - Q_L \). \( |A| \) is a diagonal matrix of eigenvalues of the Jacobian matrix which includes three characteristics with speeds \( \hat{u}_L - \hat{c} \), \( \hat{u}_R \), \( \hat{u}_L + \hat{c} \), where \( \hat{c} \) is the speed of sound. The hats refer to the Roe-average of a quantity computed for \( \hat{u}_L \), \( \hat{u}_R \) and \( \hat{H} \) by weighting with \( \sqrt{\rho} \), as follows:

\[
\hat{u} = \frac{u_L \sqrt{\rho_L} + u_R \sqrt{\rho_R}}{\sqrt{\rho_L} + \sqrt{\rho_R}}
\]

(23)

The other quantities with hats in eigenvector arrays are not averaged independently, but are the basic Roe-averaged quantities by their normal functional relation. More details of Roe method can be found in [26].

Finally the governing equation (14) can be re-written as

\[
\frac{\partial Q}{\partial t} = \text{RHS}(Q)
\]

(24)
where $RHS(Q)$ contains all convective, viscous, and source terms. The time integration procedure has been done explicitly using a fourth order Runge-Kutta scheme as follows:

$$Q_h = Q^i$$
$$Q_i = Q_i + \alpha_i \frac{\Delta t}{4} RHS(Q_{i-1}) \quad \text{for } i = 1, \ldots, 4$$
$$Q^{i+1} = Q_i$$

where $RHS(Q_{i-1})$ consists of convective, diffusive and source terms in the previous time step.

**BOUNDARY CONDITIONS**

The governing equations require the specification of the boundary conditions at the wall, inlet, and outlet due to the elliptic nature of the equations. The classical boundary condition for velocity on solid walls is the no-slip condition, namely $\vec{V} = 0$. On the symmetry line of the pipe, symmetry boundary condition ($\vec{V} \cdot \hat{n} = 0$) is also necessary.

For thermal boundary conditions, two separate cases are considered: constant wall heat flux and constant wall temperature. For a constant wall heat flux, $N_u$ of the fourth term in equation (20) should be set to the value of heat flux added to the computational nodes on the wall. Under the constant wall temperature, since the velocity vector is also zero on the wall and $N_u$ will be replaced by $e_i = C_i T_w$.

For subsonic flow, the inflow and outflow boundary conditions require three and one primitive variables respectively and they should be specified. The inflow and outflow conditions require three and one primitive variables on the wall and constant wall temperature, since the velocity vector is also zero on the wall and $N_u$ will be replaced by $e_i = C_i T_w$.

For subsonic flow, the inflow and outflow boundary conditions require three and one primitive variables respectively and they should be specified. The inflow and outflow conditions are treated by a characteristics technique. Along these boundaries, the fluxes are split in positive and negative parts following the sign of the eigenvalues for the Jacobian $A = \partial F / \partial Q$ of the convective operator, $F$ [27]. So the second term in equation (21) on the wall could be written as

$$\int_{\Omega_w} F_c \cdot n dC = \int_{\Omega_w} (A^t Q_{in} + A^t Q_{out}) n dC$$

where $Q_{in}$ is the computed internal value at the previous iteration and $Q_{out}$ the external value given by the flow configuration. Using this boundary condition method, the code can easily switch between subsonic and supersonic conditions and properly chose the related positive or negative eigenvalues for each configuration, without further consideration by the user.

**RESULTS AND DISCUSSION**

Before proceeding to consider the computational results, the quantities to be specified will be discussed. Since we solved dimensionless equations, the flow configuration can also be specified with two dimensionless parameters i.e. the Mach number and Reynolds number. Choosing the reference condition at the inlet of the pipe, these dimensionless parameters are also known at the inlet. Furthermore, a reference temperature should also be specified in which the reference viscosity of flow can be determined. This values are set to $M = 0.3$, $Re = 100$ and $T_0 = 300 K$ for all the calculations, except where a different value is mentioned. The viscosity, however, changes with of the fluid temperature in the pipe. Finally, in order to study the effect of wall heating, a value for the heat flux rate on pipe wall must be specified which is made dimensionless using equation (1).

Because of the pipe symmetry, a half section of the pipe is selected as a computational domain. An unstructured triangular grid in the $z$ and $r$ directions is employed. In order to get better resolution at the wall and the inflow, the mesh is refined in these positions. Several mesh sizes were tested to insure that the solution is not mesh dependent. However, the number of mesh points in each direction is considered depend on output results and the parameters are going to be presented. For some computations, uniform grid was chosen in the cross-stream direction. For all cases, numerical computations were carried out for different grid spacings starting from 100 (axial direction)×50 (radial direction) to 300×150 for various output parameters. The computational domain for 200×70 mesh size, for example, consists in 14271 vertices and 28000 triangles. A grid independence study is presented below.

The physical properties of the air used in this study are $\gamma = 1.4$ and $Pr = 0.71$. The considerable change in properties with temperature is only accounted for in viscosity which is calculated from equation (13) using a reference value of $T_0 = 300 K$.

Before considering the results for compressible developing flow with heat transfer effects, some comparisons are provided to verify the accuracy of the numerical method - with incompressible studies available in the literature. In Fig. 3 we see the entrance pressure drop coefficient presented by Schmidt and Zeldin [12] and more previously by Sparrow [8] both for air. As the value of $z / D Re$ increases, the pressure variation in the transverse direction approaches zero for all Reynolds numbers. Due to the presence of this transverse pressure gradient in the entrance region, a modification in the expression for the pressure drop was necessary as follow

$$\frac{p_h - p_{in}}{\rho_{in} V_w^2 / 2} = C_{f,ld} z + K$$

where $K$ is the entrance pressure drop coefficient and $p_{in}$, $p_h$, and $C_{f,ld}$ are the area integral average pressure at a specified axial direction, the centerline pressure at the pipe entrance, and the fully developed friction factor; $C_{f,ld} = 64 / Re$. $p_{in}$ and $V_w$ are average density and velocity at each cross section where for incompressible flow, $\rho_{in}$ remains constant. The results presented in this graph are for $Re = 100$, however the value of $K$ also depends upon the Reynolds number. The fully developed value for $K$ presented by Schmidt [12] was 1.4, while it was 1.24 in Sparrow [8] paper.
The differences in results may arise since in the first paper, stream function and vorticity assumptions were used, while in the second one an analytical approach used mathematical series expansions and did not consider the Reynolds number effects. An obvious difference between incompressible and compressible flow can be seen at the entrance region. For the subsonic compressible case, the outflow pressure has an important role in determining the fluid flow and mass flow rate. So the inflow pressure profile does not remain constant in the inlet cross section and therefore, the centerline pressure $p_0$ is not equal to the area-averaged pressure at this section because of the change in transverse pressure at the inflow, the pressure does not have a uniform profile like the velocity at this section.

That is why the value of the pressure drop coefficient is not exactly equal to zero at the inflow. On the other hand, the pressure drop for compressible flow changes more strongly along the pipe than in the incompressible flow. This is caused by the density change along the pipe.

The variations of velocities at different radial and axial positions as a function of axial and radial distance for incompressible flow development are presented in Figs. 4 and 5. In Fig. 4 the development of velocity profiles at various axial locations agrees well with the numerical results of Durst et al. [13], which are again for incompressible developing flow.
Figure 5 was obtained for two different Reynolds numbers and clearly shows that the velocities at the other radial locations attain their fully developed value much earlier than at the centerline. From this figure we can conclude that the development of the velocity profile is complete within dimensionless length greater than 0.16 at Re=10 and greater than 0.1 for Re=100. However this does not mean that the velocity will remain constant after a specific length for compressible flow, because it increases with length as the Mach number will also increase towards unity. A similarity velocity profile can be seen after the developing region of the pipe.

The results of the numerical solution for compressible developing flow in the circular pipe under different wall thermal conditions are illustrated in Figs. 6 through 8. The computations are carried out through the position where the velocity reaches 0.99 of its centerline velocity in each cross section. Heat generation due to viscous dissipation is also taken into account.

In Fig. 6 the variation of non-dimensional pressure drop of compressible flow along the pipe is presented and compared with some previous works for incompressible adiabatic flow. Here $\Delta p^*$ is plotted to consider the effect of heat transfer exactly on pressure drop and also comparing with results for incompressible pressure drop available in the literature. The effect of transverse pressure in the entrance and difference in the value of pressure drop can also be seen in this figure when comparing the lines for different $q^*$ values.

In Fig. 7 the hydraulic boundary layer development for different heat fluxes added to the pipe wall is observed. Also the results of adiabatic case can be seen in this figure.
A change in hydraulic boundary layer development with heat transfer is observed. It can be seen from this figure that growth of boundary layer thickness in the entrance region occurs more quickly when the pipe wall is exposed to a heat flux, i.e. the developing hydraulic length is reduced with heat addition on the pipe wall. The case of wall cooling is also considered in this figure. The boundary layer grows more slowly with cooling and the pressure drop is reduced as shown in Fig. 6.

Furthermore, the change of skin friction factor in the entrance region is investigated for compressible pipe flow in Fig. 8. For this calculation, a much finer grid was used especially through the entrance region for better resolution. The results are shown for varying heat fluxes. The local skin friction factor was normalized with the incompressible fully developed friction factor \(C_{f,\infty} = 64 / Re\).

Despite a very sharp decrease in \(C_f\) in the entrance region it also changes with heat addition of the pipe wall. More heat flux caused the friction factor to decrease in the developing region, however it may not result in less pressure drop along the pipe, because the density will also change and the momentum and energy equations are coupled in compressible flow, so the heat addition increased the pressure drop, as we saw in Fig. 6.

**CONCLUDING REMARKS**

Investigations of the development length of laminar pipe flow have been carried out for different thermal boundary conditions. The numerical results are verified for the case of subsonic flow in the inlet region of a smooth pipe. The numerical procedure and method of boundary condition implementation calculate the change of parameters in the entrance region for compressible flow. The results were compared with various reports of relevant work for incompressible cases available in the literature. Comparisons lend support to the findings of the present investigation for compressible flow.

The results indicate that although the friction factor decreased with heat flux through the pipe wall and the hydraulic boundary layer developed more quickly than for an adiabatic pipe wall, increasing the heat flux through the pipe wall in the entrance region leads to an increase in pressure loss, i.e. in order to increase the mass flow rate from the pipe at a fixed pressure difference driving the flow, one should reduce the rate of heat flux through the pipe wall.

**REFERENCES**


Copyright © 2008 by ASME


