Numerical simulation of turbulent unsteady compressible pipe flow with heat transfer in the entrance region

This article has been downloaded from IOPscience. Please scroll down to see the full text article.
2008 Phys. Scr. 2008 014052

View the table of contents for this issue, or go to the journal homepage for more

Download details:
IP Address: 81.31.171.152
The article was downloaded on 30/08/2010 at 07:52

Please note that terms and conditions apply.
Numerical simulation of turbulent unsteady compressible pipe flow with heat transfer in the entrance region

Masoud Ziaei-Rad and Ali Nouri-Broujerdi

School of Mechanical Engineering, Sharif University of Technology, PO Box 11365-9567, Tehran, Iran
E-mail: ziaeiad@mech.sharif.edu

Received 16 March 2008
Accepted for publication 30 May 2008
Published 17 December 2008
Online at stacks.iop.org/PhysScr/T132/014052

Abstract
In this paper, the compressible gas flow through a pipe subjected to wall heat flux in unsteady condition in the entrance region is investigated numerically. The coupled conservation equations governing turbulent compressible viscous flow in the developing region of a pipe are solved numerically under different thermal boundary conditions. The numerical procedure is a finite-volume-based finite-element method applied to unstructured grids. The convection terms are discretized by the well-defined Roe method, whereas the diffusion terms are discretized by a Galerkin finite-element formulation. The temporal terms are evaluated based on an explicit fourth-order Runge–Kutta scheme. The effect of different thermal conditions on the pressure loss of unsteady flow is investigated. The results show that increase in the inflow temperature or pipe-wall heat flux increases the pressure drop or decreases the mass flow rate in the pipe.

1. Introduction

The capacity of natural-gas transport and distribution networks depends on gas pressure loss. When the pressure increases in a gas power station and gas enters the pipeline, the boundary layers grow and pressure is going to be lost due to friction between the gas flow and pipe walls. On the other hand, the flow is compressible due to the significant change in density.

There have been many studies on the flow of a compressible gas in a pipeline in textbooks, papers and technical documents. Two limiting cases, adiabatic and isothermal, are often considered. Adiabatic flow conditions assume flow through an insulated pipe. These conditions are usually valid for short pipelines since there is little heat transfer to or from the gas. Isothermal flow conditions assume flow through a pipe held at a uniform temperature; these conditions are commonly assumed when studying the flow of a gas in a non-insulated pipeline. Most natural gas pipelines are considered isothermal. Especially the analysis of flows and pressure drops in piping systems has been studied by many workers and is usually based on the consideration of steady-state conditions.

Many attempts have been done by researchers to reduce the pressure loss along the gas pipeline in order to reduce the costs of transportation. One of the main steps in pressure loss reduction techniques is lining the gas pipelines with smooth coatings to reduce the frictional pressure drop [1]. Another method is the use of chemicals [2], for example in cold weather and when the pipeline capacity needs to be increased within a period of time. New surfaces is another technique concerning the making of special patterns on pipeline walls, in some cases to simulate natural surfaces, resulting in pressure loss reduction. Condensate systems are also used to find out whether the pressure-loss reduction effects in natural gas pipelines also apply when a liquid fraction is present. Experimental results on external flows [3] show that heating the gas can reduce the skin friction. Very significant drag reduction will be achieved if the beginning part of the model is heated.

Numerical simulation of unsteady internal compressible flows has been the goal of many researchers over the years and several algorithms have already been presented. Mary et al [4] proposed a second-order accurate algorithm for the simulation of unsteady viscous stratified compressible flows. The advantage of their method is its capability to deal with a broad range of subsonic Mach numbers, including nearly incompressible flows with a single modeling, based on the
fully compressible Navier–Stokes equations. The solution of a choked flow of low-density air through a narrow parallel-plate channel with adiabatic walls was investigated by means of finite-difference numerical calculation by Shi et al [5]. Toulopoulos and Ekaterinaris [6] 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. Xu et al [7] 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 with non-Cartesian control volumes. Aydin [8] studied the effects of viscous dissipation on heat transfer in thermally developing laminar forced convection in a pipe with both constant heat flux and constant wall temperature boundary conditions and obtained the distributions for the developing temperature and local Nusselt number in the entrance region and the influence of Brinkman number \((Br)\) and the thermal boundary conditions. Martineau and Berry [9] presented a new, implicit continuous Eulerian scheme for simulating a wide range of transient and steady, inviscid and viscous compressible flows on unstructured finite elements, which is developed as a predictor–corrector scheme by performing a fractional-step splitting of the semi-implicit temporal discretization of the governing equations into an explicit predictor phase and a semi-implicit pressure correction phase coupled with a pressure Poisson solution. Gato and Henriques [10] studied the numerical modeling of the dynamic behavior of high-pressure natural-gas flow in pipelines, for one-dimensional compressible flow. The occurrence of pressure oscillations in natural gas pipelines as a result of the compression wave that originated due to the rapid closure of downstream shut-off valves was studied.

The above mentioned papers and other studies on unsteady compressible flow have not focused significantly on the effect of thermal boundary conditions on pressure loss. In this paper, a combined FV–FE unsteady procedure has been developed for the numerical solution of compressible viscous flow based on a general class of cell-centered FV Roe method for the discretization of inviscid parts together with the discretization of viscous terms by the FE method over a triangular grid and the effects of heating on pressure drop in unsteady pipe flow have been studied in detail.

2. Governing equations

For gas flows with property variations (such as density and viscosity), the compressible Navier–Stokes equations are applicable even if a low-speed case is dealt with. The set of dimensional Navier–Stokes equations has been non-dimensionalized by the following dimensional reference quantities:

\[
\begin{align*}
    z^* &= z/D, & T^* &= C_v T/V_0^2, & \mu^* &= \mu/\mu_0 R e_0, \\
    r^* &= r/D, & u^* &= u/V_0, & c_e^* &= c_e/V_0^2, \\
    q^* &= q/\rho_0 V_0^3, & \rho^* &= \rho/\rho_0, & r^* &= V_0/D, \\
    R e_0 &= \rho_0 V_0 D/\mu_0, & k^* &= k/\kappa_0, & p^* &= \rho/\rho_0 V_0^2,
\end{align*}
\]

where the superscript '*' denotes dimensional variables and the subscript '0' denotes values at a reference state. \(D, V_0, \rho_0\) and \(\mu_0\) are the pipe diameter, inflow bulk velocity, density, temperature and viscosity, respectively, at the reference state. After dropping the superscript '*', the non-dimensionalized governing equations in a cylindrical frame of reference are

\[
\begin{align}
    \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial z} + \frac{1}{r} \frac{\partial (r \rho u)}{\partial r} &= 0, \\
    \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2 + p)}{\partial z} + \frac{1}{r} \frac{\partial (r \rho u u)}{\partial r} &= \frac{\partial \tau_{zz}}{\partial z} + \frac{1}{r} \frac{\partial (r \tau_{rz})}{\partial r}, \\
    \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u u)}{\partial z} + \frac{1}{r} \frac{\partial (r \rho u u)}{\partial r} &= \frac{\partial \tau_{rr}}{\partial r} + \frac{1}{r^2} \frac{\partial (r \tau_{rr})}{\partial r} = 0,
\end{align}
\]

Conservation of mass momentum and energy

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

\[
\begin{align}
    b_z &= k \frac{\partial T}{\partial z} + u_z \tau_{zz} + u_r \tau_{rz}, & b_r &= k \frac{\partial T}{\partial r} + u_z \tau_{rz} + u_r \tau_{rr}.
\end{align}
\]

The dimensionless viscous stress terms are defined for the tensor components as

\[
\begin{align}
    \tau_{zz} &= \mu \left( \frac{\partial u_z}{\partial z} - \frac{2}{3} \left( \nabla \cdot \vec{V} \right) \right), \\
    \tau_{rr} &= \mu \left( \frac{\partial u_r}{\partial r} + \frac{\partial u_r}{\partial z} \right), \\
    \tau_{\theta\theta} &= \mu \left( \frac{u_r}{r} - \frac{2}{3} \left( \nabla \cdot \vec{V} \right) \right),
\end{align}
\]

where

\[
\nabla \cdot \vec{V} = \frac{\partial u_z}{\partial z} + \frac{1}{r} \frac{\partial}{\partial r} (r u_r).
\]
are computed by solving two transport equations as follows:

\[
\frac{\partial (\rho \kappa)}{\partial t} + \nabla \cdot (\rho u \kappa) = \nabla \cdot [(\mu + \mu_l) \nabla \kappa] + \frac{1}{r} \frac{\partial}{\partial r} \left[ r (\mu + \mu_l) \frac{\partial \kappa}{\partial r} \right] + S_{\kappa}, \tag{9}
\]

\[
\frac{\partial (\rho v)}{\partial t} + \frac{\partial (\rho u v)}{\partial x} = \frac{1}{r} \frac{\partial}{\partial r} \left[ r (\mu + C_l \mu_l) \frac{\partial v}{\partial r} \right] + S_v, \tag{10}
\]

with \(c_e = 0.07\). The right-hand sides of (9) and (10) contain the production and the destruction terms for \(\rho \kappa\) and \(\rho v\):

\[
S_{\kappa} = \mu_P - \frac{2}{3} \rho \kappa D - \rho \varepsilon, \tag{11}
\]

\[
S_v = c_l \rho_k P - \frac{2 c_l}{3 c_{\mu}} \rho \varepsilon D - c_2 \rho \varepsilon^2 - \frac{\kappa}{\rho}, \tag{12}
\]

where \(c_1 = 0.129\) and \(c_2 = 1.83\). By definition, the following expressions for \(D\) and \(P\) in 2D can be used:

\[
D = \frac{\partial u_y}{\partial z} + \frac{\partial u_z}{\partial r}, \quad P = \left( \frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z} \right)^2. \tag{13}
\]

The classical \(\kappa-\varepsilon\) model is valid under the hypothesis that the local Reynolds number is high. Therefore, it is not adequate to describe regions close to a solid wall. The idea is to use a two-layer approach, i.e. to couple the \(\kappa-\varepsilon\) model to a one-equation model automatically. The method enables us to compute the flow up to the wall with no empirical work. Of course, this requires more computational resources as a finer mesh should be used. This method comprises introducing a local Reynolds number, \(y^+\) as \(y^+ = \sqrt{\rho \mu w_y/yh_y}\), where subscript \(w\) means computed at the point closest to the wall and \(y\) is the distance between the current point and this point.

To compute for \(y^+ < 200\), \(\kappa\) using the following transport equation:

\[
\frac{\partial (\rho \kappa)}{\partial t} + \nabla \cdot (\rho u \kappa) - \nabla \cdot [(\mu + \mu_l) \nabla \kappa] = \mu_l P - D_{vis}, \tag{14}
\]

where \(D_{vis} = \rho \kappa^{3/2} / l_r\), and the dynamic viscosity of turbulence will be obtained from \(\mu_l = c_\mu \rho / \sqrt{\kappa} l_h\) in which \(l_h\) and \(l_r\) are two length scales containing the damping effects in the near wall regions and are defined as:

\[
l_h = c_4 c_\mu^{-3/4} \left[ 1 - \exp \left( -\frac{y^+}{c_3} \right) \right], \tag{15}
\]

\[
l_r = c_4 c_\mu^{-3/4} \left[ 1 - \exp \left( -\frac{y^+}{2 c_4 c_\mu^{-3/4}} \right) \right],
\]

where \(c_3 = 70\) and \(c_4 = 0.41\). The governing equations can be described in a flux vector form as:

\[
\frac{\partial Q}{\partial t} + \nabla \cdot F(Q) = \nabla \cdot N(Q) + S(Q). \tag{16}
\]

The vector of conservative variables, \(Q\), for the continuity, momentum and energy equations is defined as:

\[
Q = \begin{bmatrix} \rho & \rho u_x & \rho u_y & \rho v & \lambda & \rho \kappa & \rho \varepsilon \end{bmatrix}^T. \tag{17}
\]

The inviscid flux vector \(F(Q)\), which includes components \(F_x(Q)\) and \(F_y(Q)\), and the viscous flux vector \(N(Q)\), which includes \(N_x(Q)\) and \(N_y(Q)\), have the following expressions:

\[
F_x(Q) = \begin{bmatrix} \rho u_x & \rho u_x^2 + p & \rho u_x u_y & (\mu + e) u_x u_y & \rho u_x \varepsilon \end{bmatrix}^T, \tag{18}
\]

\[
F_y(Q) = \begin{bmatrix} \rho u_y & \rho u_x u_y & \rho u_y^2 + p & (\mu + e) u_x u_y & \rho u_y \varepsilon \end{bmatrix}^T, \tag{19}
\]

\[
N_x(Q) = \begin{bmatrix} 0 & \tau_{xz} & \tau_{xr} & b_x & (\mu + \mu_l) \frac{\partial \rho \varepsilon}{\partial \rho} \end{bmatrix}^T, \tag{20}
\]

\[
N_y(Q) = \begin{bmatrix} 0 & \tau_{xz} & \tau_{xr} & b_y & (\mu + \mu_l) \frac{\partial \rho \varepsilon}{\partial \rho} \end{bmatrix}^T. \tag{21}
\]

The source vector for axisymmetric flow is written as:

\[
S(Q) = \begin{bmatrix} 0 & 0 & -\tau_{\theta \theta}/r & 0 & S_x & S_z \end{bmatrix}^T. \tag{22}
\]

3. Numerical technique

Navier–Stokes equations for the compressible viscous flow are solved by a finite-volume-Galerkin upwind technique using the Roe [11] Riemann solver for the convective part and standard Galerkin technique for the viscous terms. If \(\Omega = K_j C_j\) is a discretization by triangles of computational domain \(\Omega\), where \(C\) is the triangle area and \(K\) is the number of triangles and \(\Omega = K_j A_i\) is its partition in cells, where \(A_i\) is the cell area, thus we suppose that \(F\) varies linearly on each triangle. We move the additional terms of the cylindrical operator of equation (15) compared with the Cartesian operator to the right-hand side of the equation as a source term to use a 2D Cartesian-Coordinates-Form algorithm.

Figure 1 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 related to a finite-element grid. Otherwise, they are related to the control volume in the center of the hexagonal finite volume grid. The weak finite element formulation of governing equation (15) without the source term can be written as:

\[
\int_{\Omega} \frac{\partial Q_h}{\partial t} \phi_h dA + \int_{\Omega} \nabla \cdot (F_h - N_h) \phi_h dA = 0, \tag{23}
\]
where $\phi$ is the shape function. Changing the integral form of the viscous term $N$ using the part by part method, retaining the convective term $F$ in its original form with shape function set to 1 for this term and using explicit time integration and introducing divergence theorem for the convective part, we obtain

$$
\frac{Q^{n+1} - Q^n}{\Delta t} + \int_{\Omega_t} F_d \cdot n \, dC
= - \int_{\Omega_n} N_h \nabla \phi_h \, dA + \int_{\Omega_t} N_h \cdot n \, \phi_h \, dC.
$$

(21)

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 boundary treatment. The superscripts $n$ and $n+1$ denote 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 second term on the right-hand side of equation (21) is related to boundary condition, which is set to zero herein, and the boundary condition will be applied to finite volume formulation of convective terms. The convection term on the left-hand side of equation (21) is evaluated by the finite-volume Roe method \cite{11} on control volume surfaces. Finally, the governing equation (15) can be rewritten as

$$
a \frac{Q}{\partial t} = \text{Rhs} (Q),
$$

(22)

where Rhs $(Q)$ contains all convective, viscous and source terms. The time integration procedure has been done explicitly using a fourth-stage Runge–Kutta scheme. This scheme enables us to investigate the treatment of time-dependent pipe flow.

### 3.1. Boundary conditions

The governing equations require specification of 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 no-slip condition, namely $V = 0$. On the symmetry line of the pipe, symmetry boundary condition ($V \cdot n = 0$) is also necessary. At the pipe inlet, the flow is subsonic and two primitive variables should be specified there, where for subsonic outflow only one variable is required. Inflow and outflow boundaries are treated by a new characteristics technique. Along these boundaries, the fluxes are split into positive and negative parts following the sign of the eigenvalues for the Jacobian $A = \partial F / \partial Q$ of the convective operator $F$ \cite{12}. The heat addition to the gas flow from the pipe wall is subjected to a uniform heat flux. In order to apply this condition on the pipe wall in the numerical method, the last term in equation (21) will not be zero when a constant heat flux is imposed on the wall. Thus, the heat flux should be added to the fourth term of conservative variables vector in each iteration, $Q_4^a = Q_4^f + \int_{\Omega_t} q'' \cdot n \, dC$.

### 4. Results and discussion

The computational domain for the gas flow in the entrance region of a pipe is depicted in figure 2. The half-section of the pipe was selected as a computational domain, because of the pipe symmetry. An unstructured triangular grid in the $z$ and $r$ directions was employed. In order to get a better resolution on the boundary layer, inflow and outflow, the mesh was made fine in these positions. Several mesh sizes were tested to insure that the solution is not mesh-dependent and, finally, a mesh containing 2121 nodes and 4000 elements was used for a pipe of $z/D = 25$. Since the pressure drop between two gas stations is usually specified according to the pumping power in stations, the studied parameter is the effect of heating on gas flow rate crossing the pipe in a determined pressure difference between inflow and outflow. In our numerical method of boundary condition implementation, the outflow pressure was specified, but at the inflow the mass flow was set; therefore an iterative procedure was used to obtain the prescribed pressure at the inflow which concedes the mass flow rate.

In order to evaluate the flow variables in a 1D manner, a reference set of variables is chosen to prevent computational errors and scale all variables in an acceptable range of variation. The reference quantities used in this paper, which have been denoted by index ‘0’ in the governing equations, are as follows:

$$
Re_0 = 2 \times 10^6, \quad M_0 = 0.3, \quad T_0 = 300 \text{ K},
$$

$$
\mu_0 = 1.1 \times 10^{-5} \text{ N s}^{-1} \text{ m}^{-2}.
$$

The first results are for the development of velocity and temperature profiles in the pipeline for different heat fluxes on the pipe wall, which are depicted in figures 3 and 4. The coupling of momentum and energy equations in compressible flow will cause the change in velocity profiles with change of temperature profile, which is observed in figure 3. Increasing the heat flux on the pipe wall leads to increase in velocity profile steepness near the wall and therefore the profile curve will be completed faster than in the case of an adiabatic wall. This means that the heating will cause the hydraulic boundary layers to grow and the developing region will be shorter in this case.

The development of the thermal boundary layer is also shown in figure 3. The results show that in the case of constant heat flux on the pipe wall, generally, the thermal...
boundary layers grow faster than in the case of adiabatic flow. In figure 4, the variations of inflow pressure versus time in unsteady flow starting from a uniform initial condition are plotted. This initial condition was the steady solution of adiabatic flow. This figure shows the inflow variations with time toward the steady-state solution.

It can be seen that in the case of lower heating, there is more time needed to reach a steady-state solution, whereas increasing the heat flux on a pipe wall leads to lower steady-state time duration. On the other hand, the oscillations of pressure are higher when the heat flux is increased.

In figure 5, variation of the mass flow rate in unsteady flow for different cases of heat flux is plotted versus time. As can be seen, the final steady flow rate is decreased with heating. However, in unsteady flow, the inflow mass reduces first and then increases toward the steady-state value.

Figure 6 shows the variation of inflow temperature with time corresponding to different thermal conditions on a pipe wall. The results were obtained for a constant pressure drop of $\Delta p^* = 4$ along the pipe. It can be seen, as expected, that as in the case of the pressure variations, the inflow temperature will also increase with the heat flux. Also the steady-state temperature profile will be formed when the heat flux is increased on the pipe wall.

The variation of mass flow rate with the pressure ratio under different thermal conditions is shown in figure 7. In this graph, it is clear that heating will reduce the mass rate from the pipe for a specified pressure ratio. Therefore, in order to increase the rate of gas flow from the pipeline between two stations, one should reduce the rate of heat transfer from the pipe wall.

Effect of change in inflow pressure on mass flow rate from the pipe at different inflow temperatures is plotted in
Figure 6. Inlet gas temperature versus time for different heat fluxes.

Figure 7. Mass flow rate versus pressure ratio corresponding to different wall heat fluxes.

Figure 8. Mass flow rate versus inflow pressure at different inflow temperatures for an adiabatic pipe.

5. Concluding remarks

Unsteady compressible flow of gas from a circular pipe with friction and heat addition was studied using the 2D numerical method. The numerical procedure was a finite-volume-based finite-element method applied to unstructured grids. The figures presented in this paper show the effect of wall heating and inflow gas temperature on the pressure loss and mass flow rate in the pipe. The results indicate that increasing the heat flux on a pipe wall leads to an increase in pressure loss. In other words, in order to decrease the pressure loss, one should reduce the heat transfer to the gas flow. Furthermore, the results show that the change in the gas viscosity has a considerable effect on the flow quantities such as pressure loss or the variation of friction factor is dominant.

References

[1] Zamorano R 2002 Internal coating total gas transport cost reduction study Pipeline Gas J.