Finite element solution for velocity and temperature in developing laminar pipe flow

G. Lorenzini ¹, O. Saro ²

¹Dipartimento di Ingegneria Energetica Nucleare e del Controllo Ambientale, Università degli Studi di Bologna, Viale Risorgimento, 2, 40136 Bologna, Italy
Email: giulio.lorenzini@mail.ing.unibo.it

²Facoltà di Ingegneria, Università degli Studi di Lecce, Via per Monteroni, 73100 Lecce, Italy
Email: onorio.saro@uniud.it

Abstract

Pipe flow has been widely investigated in the last forty years from an analytical and numerical point of view. However, an accurate numerical solution of the developing flow and temperature field, based on the finite element method, is absent in related literature. This paper realises a numerical approach, carried out with a finite element code applied to a two-dimensional axisymmetric formulation of the problem, to the dynamic and thermal problem of the entrance region in laminar pipe flow, with the changing of a few significant parameters. Two different wall boundary conditions have been alternatively introduced: constant temperature and constant specific heat flux. Numerical results are in good agreement with the analytical ones available in literature.
1 Introduction

The problem of simultaneous development of velocity and temperature fields in a pipe flow has been deeply investigated and many analytical results of the subject are already available in scientific literature [1, 2, 3, 4]. If \( R \) is the radius of the pipe, \( r \) the radial co-ordinate, \( x \) the axial co-ordinate, \( p \) pressure and \( \mu \) the dynamic viscosity we have:

\[
\nu = -\frac{R^2}{4\mu}\left(\frac{dp}{dx}\right)
\left[1 - \left(\frac{r}{R}\right)^2\right]
\]  

(1)

which represents the Hagen-Poiseuille’s parabolic velocity distribution profile for a fully developed flow, and starting from the Energy Equation written for constant specific heat flux at the wall and for constant temperature we have, respectively:

\[
Nu_D = 4.36
\]  

(2)

\[
Nu_p = 3.66
\]  

(3)

where \( Nu_D \) is the Nusselt number referred to the diameter of the duct. More specifically, the transient heat transfer for laminar flow inside a flat or a circular duct following a step change in the wall temperature or heat flux has been presented analytically by the method of characteristics [5, 6, 7, 8]. Lin [9] obtained a direct numerical solution to the transient laminar heat transfer in a tube subjected to a step change in ambient temperature. Chen et al. [10] performed a direct numerical solution to the transient laminar heat transfer inside a circular duct subjected to a step change in either the wall temperature or heat flux. About this subject, the goal has now become to investigate for easy to use and accurate at the same time analytical solutions, performing then a numerical validation. The aim of this paper is to numerically investigate developing flow and temperature fields for laminar, incompressible and Newtonian pipe flow, at changing of three parameters: Reynolds number, Prandtl number and pipe wall thermal boundary conditions. In particular, two Reynolds number values, two Prandtl number values and two pipe wall boundary conditions have been considered. Results have been obtained with a finite element code for the solution, based on the axisymmetric formulation of the problem, of the Navier-Stokes and Energy equations in primitive variables pressure-velocity.
2 Description of the problem and numerical model

The investigation began with the definition of the numerical domain simulating the problem and with the setting of the appropriate boundary conditions. A two-dimensional axisymmetric geometry was utilised thanks to the nature of the problem itself, thermally and dynamically symmetrical with respect to the symmetry axis of the pipe. A nondimensional domain was built, choosing the pipe diameter as the reference parameter, set equal to one. The co-ordinates were included between 0 and 0.5 in the radial direction and between 0 and 125 in the axial direction. This choice allows to retain as dynamically fully developed the outlet flow. A nondimensional description of the problem implies also that the thermal carrier fluid inside the pipe doesn’t need to be specifically defined except by its appropriate Prandtl number, allowing then for a general purpose study. The flow was set to be laminar and two different Reynolds number values were chosen for the numerical simulations, 100 and 500, together with two different Prandtl numbers, 1 and 5. Two different boundary conditions were defined, alternatively, on the pipe wall: constant temperature or constant specific heat flux. A plug inlet velocity distribution profile was defined. This study comprises a total number of eight cases which have been analysed. A general view of the problem boundary conditions is showed in figure 1. In particular, if \( u \) is the velocity vector component in axial direction and \( v \) that in radial direction, the natural condition for the finite element method, \( \frac{\partial w}{\partial x} = \frac{\partial v}{\partial x} = 0 \), can be applied to the outlet surface together with a condition of known pressure in the last node on the axis of symmetry. A condition \( \frac{\partial u}{\partial y} = 0 \) and \( v = 0 \) can be set on the symmetry axis. The non-slip conditions, \( u = v = 0 \), are then chosen for the pipe wall. Thermal boundary conditions were defined at the pipe wall, \( t=0 \) for constant temperature and \( q''=q''_w=-1 \) for constant specific heat flux, and at the inlet section, \( t=1 \) for constant temperature condition on the wall and \( t=0 \) for constant specific heat flux. At the outlet section a thermal boundary condition \( q''=4 q''_w \) has to be set in the cases with constant specific heat flux at the wall because of the energetic balance to be respected; in the cases with constant temperature at the wall a thermal boundary condition \( \frac{\partial t}{\partial x} = 0 \) was utilised. Last pre-processing operation was the grid generation. First the numerical domain was divided into 7800 linear four nodes elements with 2 Gauss points in each direction, singled out by 8127 nodes. In the axial direction 300 subdivisions were set and 26 in the radial one, making the grid finer at the beginning of the pipe and close to the heated wall. So the length of the domain has been determined in order to obtain a fully developed flow at the outlet of the duct, for the whole range of Re chosen. In a second time an up to ten times finer grid in the entrance region has been adopted, to better analyse this part of the domain. In all the cases considered convergence, that is steady state, was considered as reached when the Euclidean norm of the variation of relative velocity and temperature between two consecutive time steps became lower than \( 10^{-7} \).
3 Procedure of solution

Resorting to the Boussinesq approximation, the problem, referring to two-dimensional axisymmetric geometries, can be described by the vectorial momentum equation

$$\rho \frac{\partial \mathbf{w}}{\partial t} + \rho \mathbf{w} \cdot \nabla \mathbf{w} = \mu \nabla^2 \mathbf{w} - \nabla p$$

and the continuity equation

$$\nabla \cdot \mathbf{w} = 0$$

In the previous equations $\mathbf{w}$ is the velocity vector, $t$ is time, $\rho$ is density, $\mu$ is the dynamic viscosity and $p$ the pressure evaluated with reference to the hydrostatic conditions at the reference temperature. In the absence of volumetric heating and neglecting the effects of viscous dissipation, the energy equation can be written as

$$\rho c \frac{\partial \theta}{\partial \tau} + \rho c \mathbf{w} \cdot \nabla \theta = k \nabla^2 \theta$$

In the previous equation $\theta$ is time. The finite element procedure utilised to solve the above reported equations has been described in [11] with a validation reported in [12]. It is based on a primitive variables algorithm of the kind SIMPLER, in which the momentum and continuity equations are solved in transient state, integrating in time until a steady solution is reached. In the solution process, at every step the pressure field is first assessed, based on the velocity field computed at the previous step. Then, utilising the assessed
pressure distribution, the momentum equations are solved and used to determine a new velocity field that, generally, does not respect local mass conservation. Finally, superimposing the continuity condition, the pressure and velocity components correlations are determined. The finite elements formulation is obtained with the Galerkin method in the classic form, applying Green’s formula to the diffusive terms of the equations and utilising interpolating functions of the same order for all the variables [11]. The non-symmetric systems of algebraic equations resulting from the discretisation of the momentum equation were solved based on a CGS (Conjugate Gradient Squared) method, while the symmetric ones resulting from the discretisation of Poisson’s equation for pressure were solved based on a CR (Conjugate Residuals) method. In both cases pre-conditioning matrices were employed, obtained with incomplete LU decomposition (ILU).

**Results and discussion**

The simulation results are reported here after in a graphic form utilising nondimensional variables:

\[
T^* = \frac{T}{T_0} \quad \chi^* = \frac{x}{D \ast \text{Re}} \quad u^* = \frac{u}{u_0}
\]

![Graph showing axial velocity along the radius at different locations (Re=100).](image)

Figure 2: axial velocity along the radius at different locations (Re=100).

Figure 2 shows the trend of the axial component of velocity along the radius in correspondence to different locations of the duct for Re=100.
It can be noted that, in a nondimensional location $x^*$ along the axis equal to $x^*=10^{-1}$ times the diameter of the duct for $Re=100$ the fully developed velocity field characterised by a parabolic profile is almost reached.

The thermal results of the present study are highlighted in the figures from 3 to 6. Figures 3 and 4 put in evidence temperature profile along the radius in correspondence to different locations of the duct for $Re=100$ and $Pr=1$ or 5, respectively, for a boundary thermal condition of fixed temperature ($Bi = \infty$).

Figure 3: temperature along the radius at different locations ($Re=100$, $Pr=1$, $Bi = \infty$).

Figure 4: temperature along the radius at different locations ($Re=100$, $Pr=5$, $Bi = \infty$).
In figure 5 it is reported the temperature trend along the radius of the pipe in the case of thermal flux superimposed ($Bi = 0$), for $Pr = 1$.

Figure 5: temperature along the radius ($Bi = 0$, $Pr = 1$).

Figure 6: bulk temperature, $T_b$ ($Bi=\infty$, 0).
Figures 7 and 8 highlight the trend of the Nusselt number $Nu$ versus the non-dimensional co-ordinate $x^*/Pr$ for $Bi = \infty$ and $Bi = 0$, respectively.

Figure 7: $Nu$ versus $x^*/Pr$ ($Bi = \infty$).

Figure 8: $Nu$ versus $x^*/Pr$ ($Bi = 0$).
Temperature trends in the case Bi = ∞ put in evidence how, for higher Pr, the
development of the thermal field occurs at a further location along the axis. The
axial location where a fully developed thermal field is reached can be inferred
looking at the trend of the bulk temperature, $T_b$, reported in figure 6 for the case
considered.
For Bi = 0 figure 5 shows how at $x^* = 5 \cdot 10^{-2}$ temperature profiles already
assume an almost coincident shape with the fully developed ones, parallel one
another.
The values of Nu, obtained in the numerical simulation performed, highlight a
very significant agreement with the analytical and numerical solutions available
in scientific literature, for axial co-ordinates $(x/D*Pe) > 8 \cdot 10^{-4}$ (Pe is Peclet
number) both with a constant wall temperature boundary condition and with a
constant specific thermal flux. Below this value of the axial co-ordinate it is
possible to notice a significant offset especially for lower Pr numbers. This is
probably due to the influence of the flow field as obtained in this paper that, as
shown in figure 2, presents the maximum axial velocity in the vicinity of the
solid wall, for the very close sections to the inlet region.

**Conclusions**

The aim of the paper was to perform an accurate numerical solution of the
developing flow and temperature field utilising the finite element method. A
two-dimensional axisymmetric formulation of the problem was chosen, applied
to the dynamic and thermal problem of the entrance region in laminar pipe flow,
at changing of significant parameters like Re, Pr and the wall thermal boundary
conditions (constant temperature or constant specific heat flux). A general good
agreement with the data in literature was obtained, except, about the Nusselt
number, for values of the axial co-ordinate smaller than $(x/D*Pe) = 8 \cdot 10^{-4}$; these
values cause an evident offset between the numerical data determined in the
present work and the literature data, especially for lower Pr numbers. This can
be caused by the influence of the flow field calculated which is characterised by
the maximum axial velocity next to the solid wall, in the inlet region. Some
more investigation will be requested about this topic. Another future
development of the research will concern wall thermal boundary conditions,
carrying out a numerical simulation in the case $0 < Bi < \infty$.

**Acknowledgements**

The Authors wish to thank Prof. P.G. Berardi and Dr. G. Cuccurullo
(Dipartimento di Ingegneria Meccanica, Università degli Studi di Salerno, Italy)
for their precious advice.
References