International Information and Engineering Technology Association
Advancing the World of Information and Engineering

International Journal of Heat and Technology
Vol. 42, No. 4, August, 2024, pp. 1139-1148
Journal homepage: http://iieta.org/journals/ijht

Numerical Investigation of Confined Flow Which Occurs Within a Conduit with Isothermal Walls and Complex Cross Section

Fatma I. Belheouane, Salim Bennoud*, Fatma Z. Halfaya

Faculty of Technology, University of Blida 1 (Saad Dahlab), Blida 09000, Algeria

Corresponding Author Email: bensalimen2006@yahoo.fr

https://doi.org/10.18280/ijht.420404

Received: 30 January 2024
Revised: 30 May 2024
Accepted: 16 June 2024
Available online: 31 August 2024

Keywords:
Internal flow, entropy number, thermal system, FEM analysis, non-circular cross-section

ABSTRACT

Viscous incompressible fluid which flows through a cross-section of a circular or non-circular form with uniformly heated walls can be observed in various engineering applications such as ventilation, air conditioning and thermal comfort. These applications particularly concern the buildings construction, and interior design of aircraft and other vehicles. However, the contemporary design evolution of various technological constructions requires the introduction of more and more conduits with noncircular form. Numerical solutions are strongly required to analyze and understand the behaviour of flows, especially, in the presence of complex cross-section geometries and the limitations of analytical solutions related to this flow. Problems describing these specific flows are modelled. Solutions based on the finite elements method are investigated to establish the fluid parameters and also the evolution of entropy generation. These remarkable data are important to ameliorate the performance of a thermal system. The significant parameter (entropy generation number Ns) can be determined once the governing equations are solved to obtain temperature and velocity distributions. The results clearly show how the considered parameters evaluate transversely and also in the longitudinal direction of the desired conduit. They indicate that Ns decreases transversely from heated walls towards the conduit centreline, where the minimum value is located. Additionally, Ns increases as the Prandtl number and the fluid viscosity are important.

1. INTRODUCTION

Internal flow is the flow that occurs within a conduit. The fluid body is of finite dimensions, and it is completely confined by the conduit walls (inner surfaces of a conduit). Diverse engineering applications used in energy conversion technologies, environmental control, and chemical processing, are influenced by these flows. This influence is related to pressure loss, heat flow, and mass transfer.

Particular attention will be focused on specific exploitations related to airflow distribution within indoor environments, ventilation, indoor climate, air conditioning and thermal comfort. The development of previous applications is directly connected with the improvement of energy efficiency and environmental protection. These aims present a primary concern for various fields such as the building construction and interior design of aircraft and other vehicles (see Figure 1) [1-3].

Conduits with isothermal walls can be observed in various thermal systems and components, such as ducts of hot or cold air, thermal exchangers, bends, diffusers, etc. [1, 3].

Due to the increase in applications related to flow distribution and energy consumption, the effective exploit of thermal systems has become an important topic in current engineering studies. Developing the performance of these systems is crucial for appropriate designs concerning modern engineering constructions, including thermal devices.

Internal flow is concerned with fluids flowing in pipes, passages, conduits, culverts, tunnels and other systems and components. The cross-section of all the above cited elements can be simple or complex.

The cross-section form, especially if it is complex, can have a significant impact on the performance of thermal systems and components, and can considerably affect pressure loss, heat transfer, and mass transfer. Moreover, it is very practical to determine parameters that help us to understand how the flow behavior is affected by variations in the cross-section form and thermal conditions.

In recent years, computational research on internal flow has mainly focused on the fluid behavior within a conduit to determine the effects of numerous physical and geometrical parameters. Also, the pressure losses in a conduit and flow rate have been considered in various recent papers. In several studies, the impacts of a change in the conduit form and the input heat power were delved. Triveni and Panua [4] presented numerical analysis and investigated the problem of natural convection applied in a cavity of triangular form with various forms of hot wall. Menni et al. [5] investigated the airflow inside a channel of rectangular form with an isothermal wall by means of FLUENT software and simplification of the problem geometry to the configuration of shell-and-tube heat exchangers. Bennoud [6] numerically studied the pressure and temperature distributions of the fluid which flows inside an isothermal wall conduit of circular form.

Previous papers solved problems of internal flow, giving special interest to the problem of the coupling of the pressure and velocity fields. But it is always interesting to investigate the problem of performance in thermal systems and energy conversion technologies. Ambethkar and Kushawaha [7] and Wang et al. [8] studied the performance improvement in thermal systems and gave an analysis concerning fundamental heat transfer problems. Indeed, various methods were developed and used to predict the performance of engineering applications concerning thermal systems and energy conversion technologies [9, 10].

Furthermore, it is known that all engineering systems subjected to thermal gradients and friction impacts are subjected to energy loss. This loss results in the generation of entropy in this system. Thus, the optimization of the performance of thermal systems can be achieved by applying the principle of minimizing entropy production.

Thermodynamics laws, especially the second law analysis, are applied to investigate and research the sources of irreversibility in terms of the entropy generation number. From this number, the active zones encouraging entropy generation can be focused. This way is important to understand and improve the system performance. Also, it is important to determine the importance of changes in viscosity, and heat convection in this improvement.

The principle of minimizing the entropy production based on the second law of thermodynamics was first introduced in the studies of Bejan [11, 12].

Narusawa [13] gave an interesting analysis of this principle for fluid flow and heat transfer within a rectangular duct. Additionally, Mahmud and Fraser [14] carried out this principle to convective heat transfer problems related to a non-Newtonian flow inside a conduit with two parallel plates.

Makinde and Aziz [15] investigated entropy generation related to a viscous incompressible flow. Basant and Taiwo [16] and Bouabid et al. [17] presented an analytical study of the effect of the channel inclination on the entropy generation in viscous incompressible fluid. Their results show that the channel inclination increases the entropy generation.

Several other investigations related to the above principle are reported in various papers. The application of this principle is established, for example, to industrial heat transfer problems, and diffusers and nozzles optimization [18-20].

The objective of this work is to present a numerical investigation of a flow within a conduit with isothermal walls. This flow is viscous and incompressible, and the conduit is of a complex cross-section. The production of entropy must be reduced to the maximum and the influence of various parameters will be studied and deduced.

The principal idea of the developed approach is a generalization of the steady Poiseuille flow for a laminar incompressible viscous flow. This flow is developed between two horizontal parallel plates of a conduit with constant wall temperatures. The distributions of velocity, pressure and temperature of the studied physical domain are determined by solving numerically governing equations. These equations are the continuity, momentum and energy equations, which are known as Navier-Stokes equations. The Finite Elements Method (FEM) was applied to obtain a numerical result of these equations. Subsequently, the dimensionless quantities, such as the entropy generation number (Ns) and irreversibility ratio are derived. These quantities are used to interpret the consequence of the viscous dissipation parameter. The analytical results as well as simulations based on the FEM method are carried out using the 'in-house-code' developed during this study. The present results have been discussed by numerical analysis and graphical representation.

Figure 1. Application examples concerning building construction and interior design of aircraft and other vehicles

2. PROBLEM MODELING AND MATHEMATICAL FORMULATION

2.1 Problem modeling

The studied problem consists of a conduit with a fluid that flows steadily in the x-direction. The spacing of the conduit is considered in the z-direction taken normally to the x-direction. The necessary physical properties of the fluid, including viscosity and density, are assumed to be constant. The transversal section of the conduit can have a complex form and the conduit walls are uniformly heated.

The model of the chosen configuration in this work is illustrated in Figure 2.

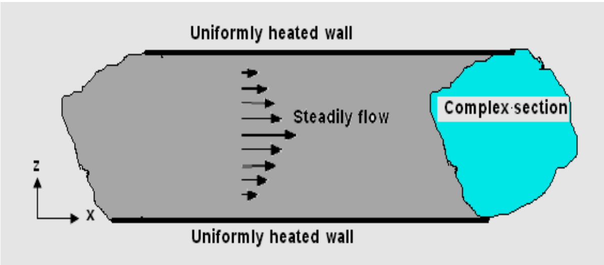


Figure 2. Description of the problem geometry

The classical fluid equations, so-called Navier–Stokes equations, can be used to describe the internal flow of a viscous and incompressible fluid inside a conduit with isothermal walls.

The differential form of these equations (conservation of mass, conservation of momentum, and conservation of energy) is given in the general case as:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{V}) = 0 \quad (1)$$

$$\frac{\partial \rho \vec{V}}{\partial t} = -\rho(\vec{V}.\nabla)\vec{V} - \nabla P + \mu\,\Delta\vec{V} + F \quad (2)$$

By considering the fact that the density is constant for incompressible flows, the equation of energy can be given in different ways, such as the one given below.

$$\frac{\partial \rho h}{\partial t} + \rho(\vec{V}.\nabla)h = \frac{\partial P}{\partial t} + k\,\Delta T + \Phi \quad (3)$$

where, ρ, V, P, μ, F, h, K, and Φ denote the density, velocity vector, total pressure of the fluid, dynamic viscosity of the fluid, volume force, specific enthalpy, thermal conductivity and dissipation function, respectively.

On the other hand, ∇ is the gradient operator; (∇.) is the divergence operator, and Δ is the Laplace operator.

However, the analytical solution of these equations is not always easy due to various difficulties which essentially arise from the nonlinearity of equations and complexity of geometries.

Indeed, the complexity and analysis difficulties of studied problems induce that the exact solution is only available for restricted simple cases. Therefore, the use of approximate numerical solutions is suitable and indispensable.

The subsequent hypotheses and assumptions will be used to simplify the previous equations:

The fluid properties, such as the density and viscosity are taken as constants, and the flow is the same throughout the cross-sections.

Furthermore, the body force and heat source are explicitly given, the velocity is constant along the adjacent layers of fluid, and the movement of the flow is uniform and stationary.

Eq. (3) decouples from Eq. (1) and Eq. (2) based on the assumption that the flow is assumed to be isothermal. The temperature will only be introduced through the viscosity parameter.

In the absence of thermal interaction, one must solve the continuity and momentum equations to obtain velocity and pressure distributions.

The Navier-Stokes system can be simplified as follows, assuming the above simplicities.

$$\nabla.\vec{V} = 0 \quad (4)$$

$$(\vec{V}.\nabla)\vec{V} + \frac{1}{\rho}\nabla P - f = \nu\,\Delta\vec{V} \quad (5)$$

where, υ is the kinematic viscosity parameter υ = μ/ρ.

(Eq. (4) is known as the incompressibility constraint, ρ = Const).

The terms "(V. ∇) V", ΔV and ΔP in Eq. (5) denote the convective term, diffusion term, and pressure gradient.

The convective impact may be dropped for diffusion dominated flows. ΔP is taken as a constant value. The body forces have been assumed to act perpendicularly to the flow direction.

Eventually, Eq. (5) returns to a simplified form given by Eq. (6) as:

$$\Delta\vec{V} = \frac{1}{\mu}\nabla P = C^t \quad (6)$$

Eq. (6) is known as Poisson's equation, which is a differential equation of second order with a constant second term.

The equations of continuity and momentum for the steady incompressible viscous flow are given as follows:

$$\nabla.\vec{V} = 0 \quad (7)$$

$$\Delta\vec{V} = \frac{1}{\mu}\nabla P \quad (8)$$

The complete solution of Eq. (7) and Eq. (8) is required to obtain results and discussion.

Adequate initial and boundary conditions for the problem geometry and fluid variables are imposed to close the system.

The pressure varies in the x-direction only due to the effects of friction forces and viscosity on walls. It is possible to treat the flow as fully developed because it is essentially axial ($V_x \neq 0$, $V_y = V_z = 0$). The velocity at conduit walls equals zero because the boundary conditions are of non-slip type.

2.2 Thermal effects

In the case of incompressible flow, fluid properties do not change with temperature, and they are taken as constant. If fluid properties are functions of temperature, all equations become coupled, as in the case of compressible flows.

It will be noted that for an incompressible flow, it is impossible to talk about the specific thermodynamic equation, so-called equation of state, which is always used in the case of compressible flows.

In the case of a viscous incompressible fluid within a conduit with isothermal walls, the equations system consists of the continuity and momentum equations.

Equations governing thermal effects could be taken and thermodynamic relations (specific relation of enthalpy for example) must be used to construct the system of fluid-mechanical equations.

It was mentioned above for incompressible flows that the energy equation is decoupled from the two other equations to obtain the unknown velocity and pressure fields without knowing the temperature.

Using the relation of enthalpy given as (h = Cp T), the equation of energy (Eq. (3)) can be rewritten as:

$$\frac{\partial \rho \mathrm{T\,Cp}}{\partial \mathrm{t}} + \rho(\vec{\mathrm{V}}.\nabla)T\,Cp = \frac{\partial \boldsymbol{P}}{\partial \mathrm{t}} + \mathrm{k}\,\Delta\mathrm{T} + \boldsymbol{\Phi} \tag{9}$$

Eq. (9) can be simplified using the following assumptions:

-The density ρ is taken as constant (incompressible flow).

-The pressure term is usually neglected, ∂P/∂t≈0.

-The dissipation function is irreversibly converted into internal energy. The radiative heat transfer and internal heat generation are neglected, Φ≈0.

-The fluid movement is uniform and stationary, ∂/∂t≈0.

Eq. (9) takes the following form:

$$(\vec{\mathrm{V}}.\nabla)\mathrm{T} = \boldsymbol{\alpha}\,\Delta\mathrm{T} \tag{10}$$

where, α denotes the thermal diffusivity (α = k/ (ρ Cp)).

In the present problem, it is assumed that the temperature at the conduit walls is constant (Tw = Cte).

Applying the boundary conditions T(0, z) = T_0 (inlet condition), T(x, wall) = Tw (walls temperature) and ∂T/∂x(x, 0)=0 (symmetric temperature condition).

2.3 Entropy generation

Indeed, it is essential to establish parameters responsible for generating entropy in the case of internal flows (more details can be found in studies [14, 17-20]). By applying some simplifying assumptions, the formula of the entropy generation rate is obtained and given as:

$$E_G = \frac{K}{T_0^2}(\nabla T)^2 + \frac{\mu}{T_0}\varphi \tag{11}$$

Also, the characteristic entropy generation rate is defined as:

$$E_c = \frac{K(T_w - T_0)^2}{h^2 T_0^2} \tag{12}$$

The terms, on the left hand of Eq. (11), represent the sources of irreversibility. They represent the energy losses by heat transfer (due to conduction effect), and by fluid friction (due to viscosity effect), respectively.

Eq. (11) must be divided by Eq. (12). The dimensionless Ns number is determined as:

$$N_S = \frac{h^2 T_0^2 E_G}{K(T_w - T_0)^2} \tag{13}$$

It is possible to expand Eq. (13) in non-dimensional form. It can be written using the dimensionless velocity and temperature with the application of appropriate assumptions and boundary conditions.

The pressure gradient, fluid viscosity and density are taken as constant, (the velocity is a function of y only, Vx≠0 = f(y), Vy = Vz = 0, ∂V/∂x = 0, Vx = 0 at walls), (see Section 2.1).

Dimensionless variables and parameters are introduced, and the entropy generation number becomes in terms of these dimensionless variables:

$$N_S = \frac{1}{Pe^2}\left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \frac{Br}{\Omega}\left(\frac{\partial V}{\partial y}\right)^2 = Nx + Nz + Nf \tag{14}$$

where, Pe, Br and Ω are the *Peclet* number, Brinkman number and temperature difference parameter respectively (Ω = (Tw − T_0)/T_0).

Nx, Nz and Nf are dimensionless entropy numbers due to both axial and transverse heat conduction, and to viscosity (fluid friction) respectively.

The entropy generation number Ns can simply be determined once the velocity and temperature distributions are obtained.

From the number Ns, an important analysis can be performed in order to know which parameter dominates the studied model (the fluid changes or the heat transfer).

The performance of the engineering processes regarding thermal systems can be predicted by the analysis based on the thermodynamics second law.

3. SOLUTION OF THE PROBLEM

The system of equations (Eq. (1)-(3)) governs the desired flow. These equations are generally complex and nonlinear (the term "(V. ∇) V" in Eq. (2) presents nonlinear convective term). With suitable assumptions, a simplified form of this system can be obtained.

It is very usual to write these governing equations in dimensionless form. To do this, various dimensionless variables and parameters must be introduced to select the characteristic quantities that describe the flow problem.

The dimensionless governing equations and their relative dimensionless quantities must be defined and obtained. These equations are related to a steady, viscous and incompressible flow, and associated with the appropriate boundary conditions.

Note that the form of these dimensionless equations is not unique and can also be expressed in several ways [18-20].

Indeed, dimensionless equations are very similar to their dimensional counterparts. Several additional dimensionless numbers such as Reynolds number (shown in the momentum and energy equations), Eckert, Prandtl and Peclet numbers (shown in the energy equation) must be added and used to obtain a simplified system of governing equations.

The general procedure for solving the problem described above involves the following steps:

Obtain and define the governing equations of the considered problem: give the mathematical model of governing equations in the simplified form most possible.

Recall that the present study is a CFD-problem which concerns a viscous and incompressible fluid. The fluid flows in a conduit of complex form with isothermal heated walls (see Section 2.1).

The system of governing equations can be delivered in the simplified dimensional form or also in the dimensionless form with their relative dimensionless quantities.

Obtain distributions and profiles of the velocity and pressure: solve the above system of equations.

Proceed to the heat transfer analysis to obtain the temperature distributions: solve the energy equation subject to the appropriate boundary and interface conditions of the thermal domain. The solution of the Eq. (3), the equation of energy, can be generally solved analytically.

Find the entropy generation distribution through the conduit. The entropy generation function is obtained using the temperature and velocity distributions. IT is used to limit the entropy production rate through the conduit.

The analysis of entropy generation minimization EGM is an approved strategy to optimize the amelioration of thermal fluid devices by assessing both fluid parameters and properties.

Finally, investigate effectively the problem: the physical and thermodynamic quantities must be illustrated in curves and graphs.

3.1 Analytical solutions for a simple conduit

A famous example of an internal laminar flow is the Poiseuille flow, which is illustrated in Figure 3. This flow can be adapted for conduits having a circular section.

In the next section, an analytical algorithm with appropriate boundary conditions will be developed and presented to resolve of Eqs. (7) and (8).

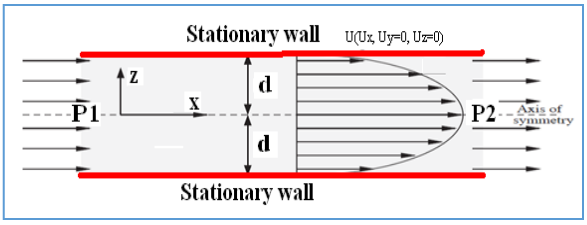


Figure 3. Physical model of the Poiseuille flow

The flow remains identical to the bi-dimensional flow with revolution symmetry. It stays same to it even in all plans passing by the conduit axis. The normal and tangential components of the velocity at the conduit walls are null.

The flow is viscous, the density is constant and walls surfaces are smooth. The heat transfer and frictions impact are negligible.

The studied flow is considered as a fully developed flow. The flow parameters and properties must be obtained without including the conduit entrance region in the calculations.

The equation of continuity with a constant value of ρ is given as follows:

$$\frac{\partial V}{\partial x} = 0 \tag{15}$$

The differential equation of motion for this type of a Newtonian fluid with constant μ (fluid viscosity) is given in Cartesian form as:

$$\mu \frac{\partial^2 V}{\partial z^2} - \frac{\partial p}{\partial x} = 0 \tag{16}$$

A constant value, non-zero value, must be given to the pressure gradient ∂p/∂x (if ∂p/∂x = 0, then V = 0 and there is no flow.) and the conduit half-width is d in the Z-direction. Using the no slip boundary conditions V = 0 at Z_1 = d and at Z_2 = -d, the expression for the fluid velocity profile is given by:

$$V = \frac{1}{(2\,\mu)} \frac{\partial p}{\partial x} \left(d^2 - z^2\right) \tag{17}$$

The viscous and stationary flow within a cylindrical tube is an interesting extended case of the above flow. This flow is illustrated in Figure 4.

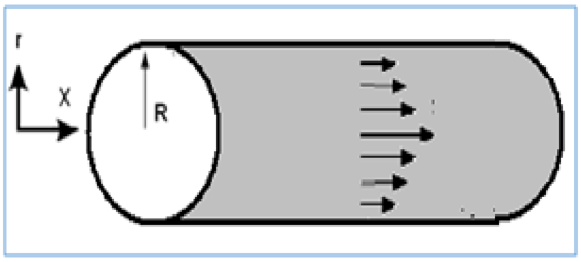


Figure 4. The stationary viscous flow in a cylindrical tube

To resolve this problem, physical properties of the flow (such as viscosity and density), and the pressure gradient ∂p/∂x are taken as constant. V = 0 at r = R, and ∂Vx/∂r = 0 at r = 0 are applied as adequate boundary conditions.

The expression of velocity distributions, in cylindrical coordinates, is given as:

$$Vx = \frac{1}{(4\,\mu)} \frac{\partial p}{\partial x} \left(R^2 - r^2\right) \tag{18}$$

3.2 Numerical solution

Due to the complex form of the conduit section, numerical solutions are needed to solve the governing equations of the studied problem. Numerical solutions for velocity, pressure function and dimensionless temperature are obtained using a developed code based on the FEM method. This in house-code is written and performed in the *Fortran language*.

The governing equations are discredited by applying appropriate schemes based on a discrete technique.

Geometrical dimensions and selection of the mesh element type as well as the input of fluid parameters, and boundary conditions are necessary to obtain the mesh of the considered domain. The mesh generation is the first important step in the formulation by FEM method.

For the problem analysis by the FEM method, various discrete formulations can be applied. In this work, the *weak formulation of Galerkin* is chosen to be used.

Using the Galerkin's formulation, the domain of the studied configuration is meshed into a number of sub-domains. The nodal shape functions defined on the mesh element are the basis functions which are used to numerically discretizate Eq. (8).

The unknown functions of each mesh element are approximated by these shape functions as:

$$V = \sum_{i=1}^{Ke} N_i V_i \tag{19}$$

where, *Ke* denotes the number of nodes attached to the mesh element. N_i and V_i are the nodal interpolation function and potential function of the i^{th} node of the element, respectively.

The Galerkin formulation permits the calculation of elemental values on each element of the mesh and the generation of a global system of equations.

So, the FEM formulation transfers the algebraic system of equations into an equivalent matrix form that allows the resolution of the problem and calculation of the unknown values.

The global system is written as:

$$[k]\{V\}=\{F\} \tag{20}$$

where, [K] is the (N × N) positive, banded and symmetric, final matrix. {V} is the (N × 1) vector of velocity distributions and {F} is the (N × 1) vector of sources (N is the total number of nodes of the meshed domain).

The Gauss elimination method is used to solve the algebraic system (20). This algorithm is integrated using a computational developed code.

Indeed, the mathematical approach and its appropriate algorithms are integrated in a computational code.

For numerical integration, the governing equations are numerically solved using a developed Fortran code. This iterative solution enables to obtain values for velocity, pressure, and temperature at mesh nodes.

Once the velocity, temperature, and pressure values are obtained, other parameters of this flow can be calculated and deduced, such as total surface Atotal, hydraulic diameter DH, perimeter of meshed section Pm, maximum velocity Vmax, mean velocity Vmoy, pressure losses, friction K and dimensionless numbers.

4. RESULTS AND DISCUSSIONS

Initially, the computational code was tested to validate the reliability of the used method. The results are compared with some cases of conduits that have a simple cross-section and exact analytical solutions. The validation verifies code results by recovering the analytical solution of Hagen-Poiseuille [1, 10]. A comparison between analytical and numerical results permits the validation of calculus concerning velocity and temperature profiles for configurations of a simple geometry (circular section).

The velocity components are computed at the grid nodes of the considered mesh, while the other variables and parameters are derived from the obtained velocity values.

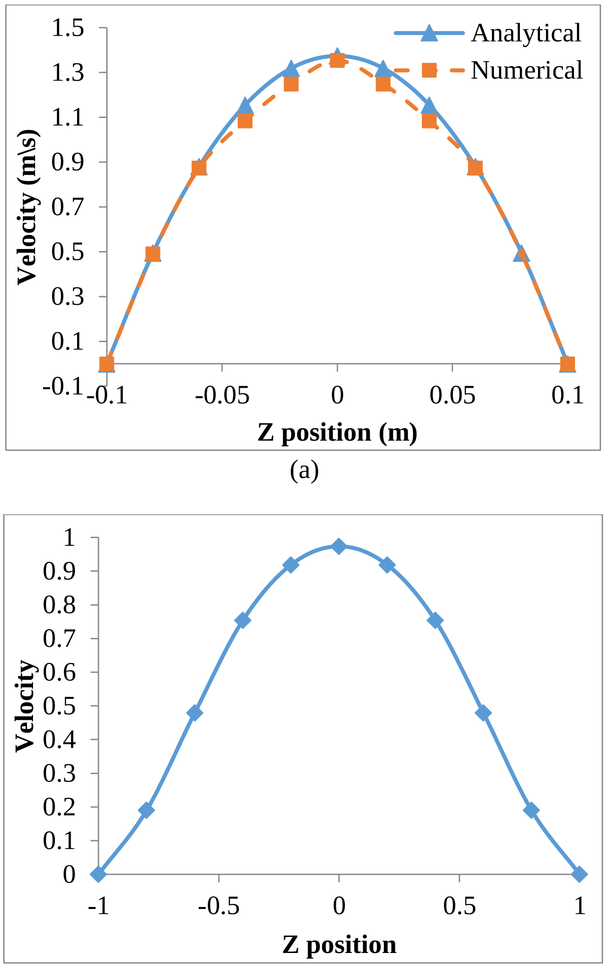
An application related to a cross-section of a cylindrical conduit having a circular form was performed. The mesh was realized for this cross-section with 289 nodes and 512 triangular elements, and the circle radius is r=0.1m.

The fluid density, dynamic viscosity and Pe are taken as ρ=1.205 kg m^{-3}, μ=1.82 kg m^{-1}s^{-1}, Pe=7.1, respectively.

The velocity profile is plotted in Figure 5(a), which presents a comparison between analytical and computational results. On the other hand, Figure 5(b) shows the distribution of dimensionless velocity.

The illustrated values are displayed for a specific range (z [-0.1, 0.1] m) and for different positions of x in the Z-axis direction. The fluid velocity profile in Figure 5(a) is parabolic, and it increases transversely to the position of the circle centreline. Lower values are observed near the walls until a null value at (r=R). The maximum value is obtained along the circle centreline. Present results show a good agreement with the analytical results and an average error of 0.03 is obtained between both solutions

Figure 6 shows that the temperature reduces transversely. The maximum value is obtained at the heated walls, the minimum value is presented along the centreline of the conduit, and an average error of 0.04 is obtained for this time.



(a)

(b)

Figure 5. (a) Velocity profiles; (b) Dimensionless velocity profile

The analytical and computational results of the temperature are plotted in Figure 6.

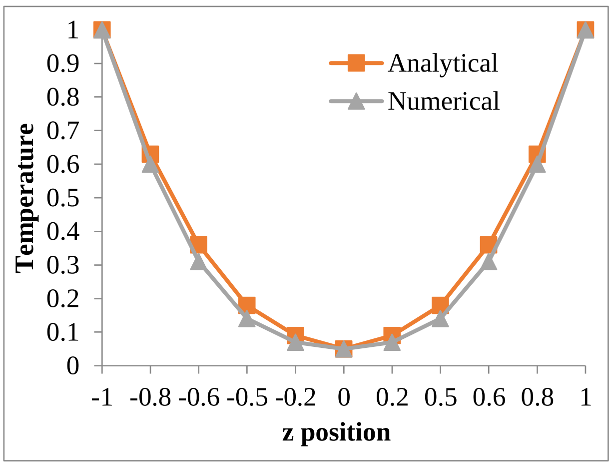


Figure 6. Dimensionless temperature profiles

Figures 5 and 6 illustrate a comparison between analytical and numerical results for a conduit of a circular section. This comparison is used to validate the developed numerical code.

The results present a good agreement between analytical and present results. The obtained average error is of 0.04 for temperature profiles and 0.03 for velocity profiles.

To exploit the performance of numerical analysis, other parameters having a more significant indication can be determined and calculated.

Figure 7 presents the evolution of the dimensionless temperature profile for different positions in the longitudinal direction (x=0.4, 0.6, 0.8).

It can be observed that there is a variation and change of temperature profile in the longitudinal direction (x-direction) of the conduit, the fluid temperature decreases from the inlet to the outlet.

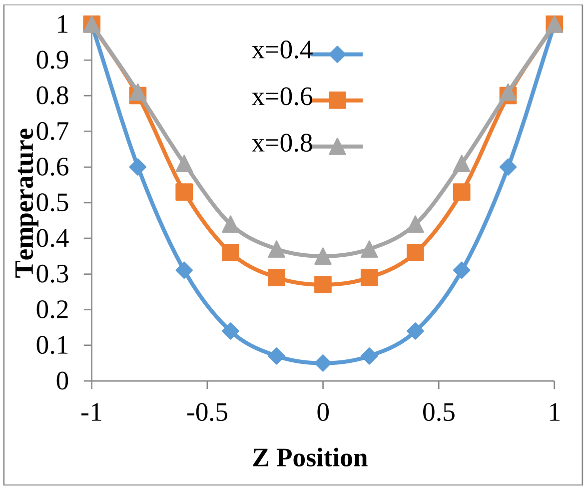


Figure 7. Temperature evolution in a longitudinal direction

The entropy generation number (dimensionless number) decreases transversely. The maximum value is obtained on the heated walls and the minimum value is focalized along the centreline of the cross-section (see Figure 8).

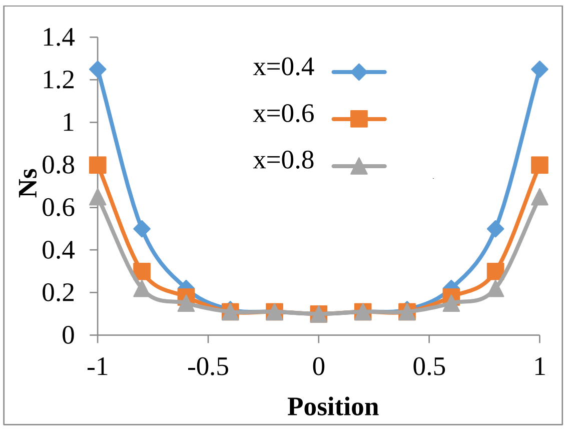


Figure 8. Evolution of the entropy generation number for different positions

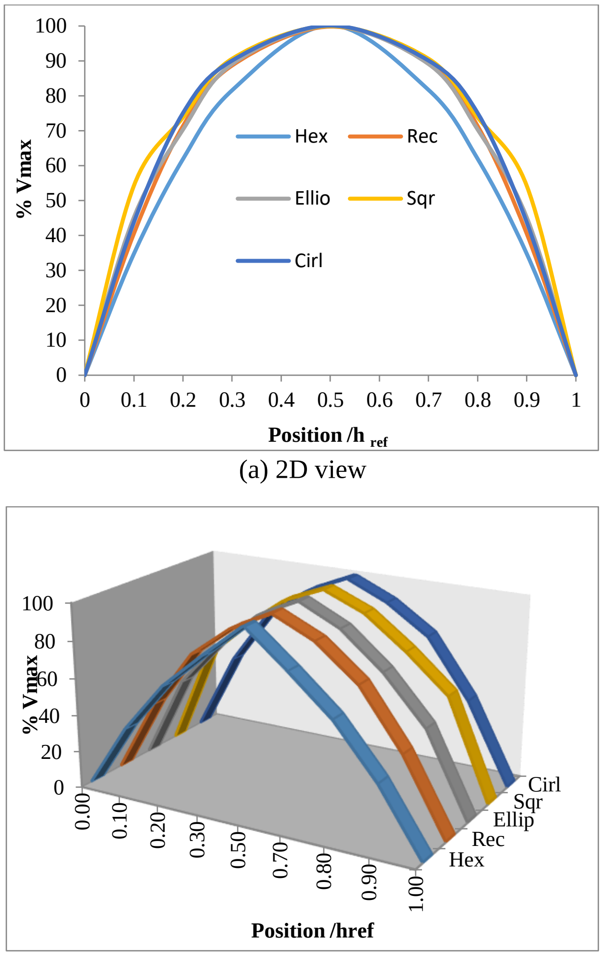
In Figure 8 also, the evolution of entropy in the longitudinal direction is presented for different positions (x=0.4, x=0.6, x=0.8) and it can be shown that this number decreases in the longitudinal direction.

After validation, simulations can be carried out for various geometries. The developed computational code can be used to solve similar cases to the above CFD problem. This means any flow problem of incompressible viscous fluid in a conduit with uniformly heated walls and complex geometry of the cross-section.

Initially, several geometries were selected so that the forms of their cross-section are different, but their relative surfaces are roughly equal.

This choice is justified by the reason to keep the same value of the pressure gradient and the same properties of the desired fluid which are mentioned in the above paragraphs.

Figure 9 shows the profiles of dimensionless velocity for different geometries with perfect symmetry.



(a) 2D view

(b) 3D view

Figure 9. Dimensionless velocity profile for different geometries 2D and 3D view

Figure 9 shows that the velocity profile is always parabolic in nature for the different axisymetric configurations. It increases transversely to the position of the centreline of the selected conduit with a null value on the walls. The maximum value is always observed along the conduit centreline for geometries having a perfect symmetry.

Table 1 shows the various geometrical characteristics of the selected geometries.

The selected geometries present perfect symmetry when compared to the plans or the axes.

The illustrated values are displayed for a specific range (z [0.0, href] m) and a fixed position in the longitudinal direction (x=0.4). It is worth noting that the z-axis is parallel to the yz-plan and regular to the x-axis.

The bottom wall is taken as the z-axis central value (z=0) and the conduit half-width is href/2 in the z-direction which is also the conduit centreline (y=0, z= href/2).

Table 1. Characteristics of the selected geometries

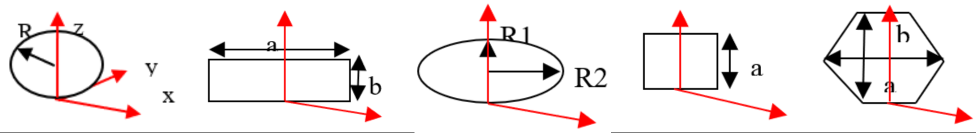
Selected geometry					
Dimensions (m)	R=0.14	a=0.32 b=0.2	R1=0.1 R2=0.2	a=0.25	a=0.30 b=0.075
Area (m^2)	A=0.0615	A=0.064	A=0.0623	A=0.0625	A=0.0675

Figure 10 shows the change in the temperature profile for different geometries with perfect symmetry.

The temperature distribution is determined to growth towards the wall. The fluid temperature decreases from the conduit centreline to the heated partitions for all selected geometries.

In Figure 11, the evolution of the entropy generation number in the transversal cross-section (yz plane) is presented for different geometries, for a fixed position in the longitudinal direction (x=0.4). This dimensionless number decreases transversely from the heated walls where the maximum values can be observed on the conduit centreline.

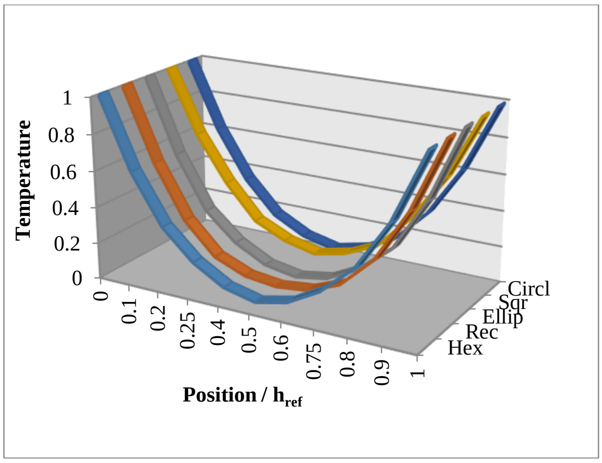


Figure 10. Dimensionless temperature profiles

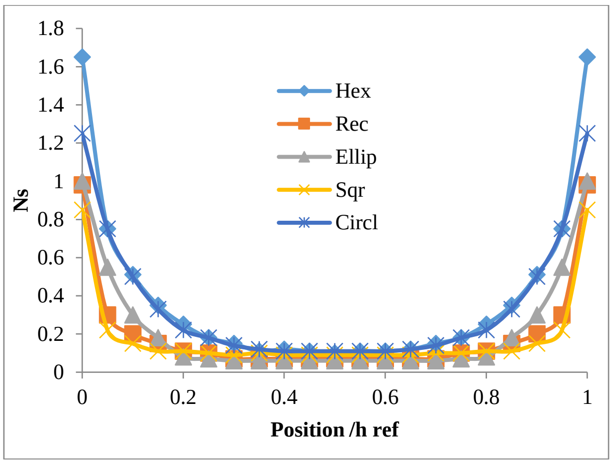


Figure 11. Evolution of the entropy generation number for different geometries (x=0.4)

The value of entropy generation number near the walls is higher than those at the conduit centreline. This is due to the effect of surface friction on the fluid which increases entropy production.

4.1 Effect of the non dimensional parameters

To develop this numerical investigation, the effect of the non dimensional parameters must be studied in order to determine which ones are most dominant.

Several values of the Prandtl number Pr are chosen in accordance with the appropriate fluids (air, water, …). The variations of the Entropy generation number for various fluids are represented in Figure 12.

Figure 12 illustrates that entropy production augments with an increase in Pr. This is due to the accumulation of heat within the conduit resulting in a rise in fluid temperature.

It is worth noting that thermal diffusivity is inversely proportional to Pr, and its increase leads to a rise in the fluid temperature.

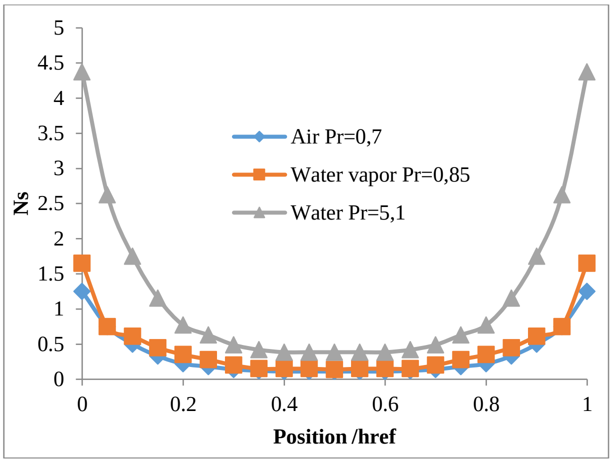


Figure 12. Evolution of the entropy generation number for different fluids

5. CONCLUSION

In this paper, an incompressible viscous fluid within a conduit with isothermal walls is analyzed using the FEM method and thermodynamics laws. The velocity and temperature distributions are used to deduce and calculate the entropy generation number for various cases.

The numerical calculation of various dimensionless parameters and physical quantities such as velocity, temperature, and entropy generation has been carried out using a Fortran code developed during this study. The influences of some parameters on the flow behavior are shown graphically and finally, the active zones where the entropy generation is maximal can be identified.

It is important to obtain an optimal level between the dynamic point of view (minimal velocity) and thermal improvement (better transfer of heat). Thus, it is significant to design a model that allows users to deduce the influence of diverse parameters on the generation of entropy which must be minimized.

The effect of the cross-section and fluid characteristics on the velocity, pressure, temperature distribution as well as the impact of Prandtl number on the entropy production were considered.

Interesting results can be reported, such as:

The velocity decreases transversely from the conduit centreline where the maximum value is focused to the heated walls where null values are observed.

The fluid temperature increases from the conduit centreline to the heated walls for all selected geometries with perfect symmetry.

The dimensionless number Ns (entropy generation) decreases transversely from the heated walls to the conduit centreline where the minimum value is identified.

It can also be seen that the Ns number increases with the increase of Prandtl number Pr. Furthermore, it is confirmed that the entropy generation increases when the viscosity of the fluid is important.

Simulations prove that the thermal conduction irreversibility dominates along the conduit centreline. The augmentations in dimensionless parameters can also motivate the domination of the fluid friction irreversibility near the conduit heated walls.

The proposed approach allows for an efficient and accurate calculation of the velocity and temperature profiles. It can be applied to determine the entropy generation rate for any form of the cross-section related to a conduit with isothermal walls. This conduit confines an incompressible, viscous and stationary flow. Furthermore, the model can be without difficulty adapted to investigate the entropy generation rate of any similar flow in a plan or inclined conduit.

REFERENCES

[1] Çengel, Y., Ghajar, A. (2014). Heat and Mass Transfer: Fundamentals and Applications, 5th ed. NY: McGraw-Hill Higher Education.

[2] Awbi, H.B. (2003). Ventilation of Buildings, 2nd ed. Spon Press, Publisher: Taylor & Francis.

[3] Han, O., Li, A.G., Yin, H.G. (2021). Comparative studies on isothermal attachment ventilation based on vertical walls, square and circular columns. Energy and Buildings, 213: 110634. https://doi.org/10.1016/j.enbuild.2020.110634

[4] Triveni, M.K., Panua, R. (2017). Numerical analysis of natural convection in a triangular cavity with different configurations of hot wall. International Journal of Heat and Technology, 35(1): 11-18. https://doi.org/10.18280/ijht.350102

[5] Menni, Y., Azzi, A., Zidani, C. (2018). CFD Simulation of thermo-aeraulic fields in a channel with multiple baffle plates. Journal of Thermal Engineering, 4(6): 2481-2495. https://doi.org/10.18186/THERMAL.465696

[6] Bennoud, S. (2017). Simulation of viscous incompressible flow through a conduit with isothermal walls. In International Workshop on the Complex Turbulent Flows (IWCTF2017), Tangier, Morocco. https://www.ercoftac.org/ercoftac_news/report_on_the_intern/.

[7] Ambethkar, V., Kushawaha, D. (2017). Numerical simulations of fluid flow and heat transfer in a four-sided lid-driven rectangular domain. International Journal of Heat and Technology, 35(2): 273-278. https://doi.org/10.18280/ijht.350207

[8] Wang, G.H., Dbouk, T., Wang, D.B., Pei, Y.S., Peng, X., Yuan, H.L., Xiang, S. (2020). Experimental and numerical investigation on hydraulic and thermal performance in the tube-side of helically coiled-twisted trilobal tube heat exchanger. International Journal of Thermal Sciences, 153: 106328. https://doi.org/10.1016/j.ijthermalsci.2020.106328

[9] Liberman, M.A. (2008). Introduction to Physics and Chemistry of Combustion: Explosion, Flame, Detonation. Springer Germany.

[10] White, F.M. (2011). Fluid Mechanics, 7th ed. NY: McGraw-Hill Higher Education.

[11] Bejan, A. (1994). Entropy Generation Through Heat and Fluid Flow. John Wiley & Sons, Inc.

[12] Bejan, A. (1996). Entropy Generation Minimization. CRC Press: USA.

[13] Narusawa, U. (2001). The second law analysis of mixed convection in rectangular ducts. Heat and Mass Transfer, 37: 197-203. https://doi.org/10.1007/s002310000173

[14] Mahmud, S., Fraser, R.A. (2002). Thermodynamic analysis of flow and heat transfer inside channel with two parallel plates. Energy, an International Journal, 2(3): 140-146. https://doi.org/10.1016/S1164-0235(02)00062-6

[15] Makinde, O.D., Aziz, A. (2010). Second law analysis for a variable viscosity plane Poiseuille flow with asymmetric convective cooling. Computers & Mathematics with Applications, 60(11): 3012-3019. https://doi.org/10.1016/j.camwa.2010.09.063

[16] Basant, K., Taiwo, S.Y. (2019). Entropy generation in an inclined porous channel with suction/injection. Nonlinear Engineering, 9(1): 103-110. https://doi.org/10.1515/nleng-2018-0116

[17] Bouabid, M., Magherbi, M., Hidouri, N., Brahim, A.B. (2011). Entropy generation at natural convection in an inclined rectangular cavity. Entropy, 13(5): 1020-1033. https://doi.org/10.3390/e13051020

[18] Mburu, Z.M., Mondal, S., Sibanda, P., Sharma, R. (2021). A numerical study of entropy generation on Oldroyd-B nanofluid flow past a Riga plate. Journal of Thermal Engineering, 7(4): 845-866. https://doi.org/10.18186/thermal.930653

[19] Giangaspero, G., Sciubba, E. (2011). Application of the EGM method to a LED-based spotlight: A constrained pseudo-optimization design process based on the analysis of the local entropy generation maps. Entropy, 13(7): 1212-1228. https://doi.org/10.3390/e13071212

[20] Schmandt, B., Herwig, H. (2011). Diffuser and nozzle design optimization by entropy generation minimization. Entropy, 13(7): 1380-1402. https://doi.org/10.3390/e13071380

NOMENCLATURE

A	cross section, m^2
Cp	specific heat, J. kg^{-1}. K^{-1}
d	tube diameter, m
Ec	characteristic entropy generation rate
E_G	entropy generation rate

F	body force, N
h	specific enthalpy
k	thermal conductivity, $W.m^{-1}. K^{-1}$
Ns	entropy generation number
P	pressure, Pa
Pr	Prandtl number
R, r	radius, m
Re	Reynolds number
T	Temperature, K
T_0	inlet temperature, K
Tw	walls temperature, K
V	velocity, $m\ s^{-1}$
Vmax	maximum velocity, $m\ s^{-1}$
Vmoy	average velocity, $m\ s^{-1}$
Vx, Vy, Vz	velocity components
x	x Cartesian axis direction, m
y	y Cartesian axis direction, m
z	z Cartesian axis direction, m

Greek symbols

∇	is the gradient operator
Δ	is the Laplace operator
$\nabla.$	is the divergence operator
α	thermal diffusivity, $m^2. s^{-1}$
μ	dynamic viscosity, $kg. m^{-1}.s^{-1}$
ν	kinematic viscosity parameter
ρ	fluid density, $kg\ m^{-3}$
φ	dissipation function