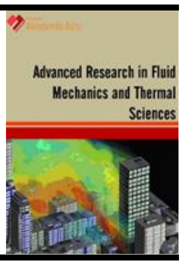
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Numerical Simulations and Analysis of Viscous Incompressible Flow in Conduit With a Complex Transversal Section

Open Access

Bennoud Salim[1,*], Fatma I Belheouane[2]

[1] Laboratory of Aircrafts, University of Saad Dahlab (Blida 1), Blida, Algeria
[2] Faculty of Technology, University of Saad Dahlab (Blida 1), Blida, Algeria

ARTICLE INFO

Article history:
Received 23 September 2018
Received in revised form 12 December 2018
Accepted 5 January 2019
Available online 11 January 2019

ABSTRACT

The paper presents simulations of a viscous incompressible flow which through conduits having a complex section. The finite element method was applied because of its adaptation for complex geometries. The solution of chosen system gives a distribution of flow velocity profile through the studied section, and from the velocity values, others parameters can be deduced such as: the maximal velocity, Reynolds number, the volume of considered flow, the friction force to exert on the wall, as well as the shear stress. The validation was made with a comparison between obtained solutions and results given by analytical solutions with reference to cases of a simple section such as the circular section. Developed code and analysis can be used, as helpful tool, in many technological and engineering applications related to construction and installation of conduits, and in particular when users are interested in modifications of conduits section. In this study, special attention has been given to effects of section form on the distribution of flow velocity profile, to have results and necessary data according to the geometry of the selected conduits which permit to determinate the performances of desired system.

Keywords:
Conduits, Finite elements method, Flow simulation, Viscous incompressible flow.

1. Introduction

The viscous incompressible flow phenomena occur and appear in various disciplines in science and engineering. The simplest viscous flow problems involve just one fluid in the laminar regime. The governing equations, in this case, are those so-called Navier–Stokes's equations [1-3].

Several recent works have investigated a numerical study to understand the effect of section form and geometry orientation on fluid displacements and also on heat and mass transfer [4-6].

The design and realization of apparatuses and aerodynamic or hydraulic machines require the exploitation of vital elements for the connection between various bodies that are conduits and piping. All technical fields are, directly or indirectly, concerned by the conduits and their problems [7-9].

* *Corresponding author.*
E-mail address: bensalimen2006@yahoo.fr (Bennoud Salim)

The phenomenon of the studied flow is governed by incompressible Navier-Stocks equations. In the general case, these equations are complex and nonlinear, but with suitable assumptions employed for the present study (such as incompressibility, flow permanence, viscosity, *etc*), a simplified model of equations is obtained, and it is important to note that this model was always used by authors [10-15]. The theoretical analysis of this model is offered to support the construction of numerical methods, and often computational examples are used to illustrate theoretical results [16-18].

Indeed, the complexity of studied conduits sections induced that the analytical solution does not exist, with the exception of some solutions for simple geometries, and obviously our interest is directed to search for approximate numerical solutions, by employing the finite element method which adapts with any selected section [19-23].

In this paper, the finite element method (FEM) is chosen and used to compute primitive variables such as velocity and pressure profiles. This special computational method is based on a variational formulation of Navier-Stokes equations in appropriate function spaces and also used to determinate approximate values in certain finite dimensional subspaces.

The purpose of this paper is to present a computational solution, using a developed code, to obtain more efficient and accurate numerical tools to compute viscous flows, and specially to get solutions for cases of flows in conduits with a complex section.

The results retain the advantages of computational analysis and enable to have necessary data to determinate the system performances according to the selected conduits geometry.

2. Flows Through Conduits With A Complex Section

The flow in conduits is a fundamental problem in aeronautic, hydraulic as in aerodynamic. One of the challenges that must be met to enable the realization of the technologies of scientific and/or industrial applications is the transport and pumping of various quantities of fluids [24, 25]. The conditions of flow through conduits, with or without heat transfer, depend primarily on geometrical and dynamic parameters (dimensions and forms of conduit, pressure, *etc*). And all flows in the closed pipes, conduits and channels are considered as internal flows [9, 12].

The governing equations of flow in conduits are functions of properties of conveyed fluids. The flow itself generates pressure losses which have a capital influence on the design of conduits. The correct design requires also more information on temperatures of use and mechanical properties of used materials.

There are several useful applications of conduits. The distribution and transportation systems of fluid in various industries frequently use conduits of a circular or rectangular section. Also, an elliptic or others special sections (complex forms) can be used spreading to required standards (see Figure 1).

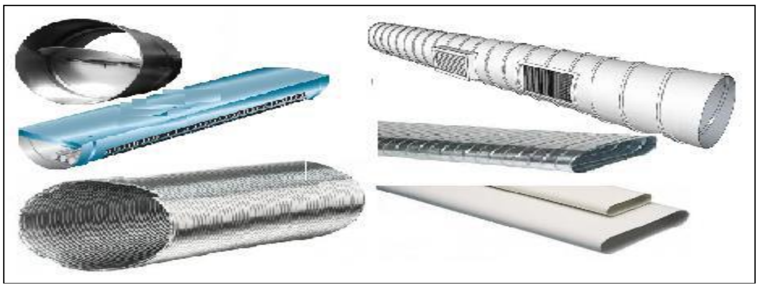


Fig. 1. Application of circular and non-circular sections for various conduits

The circular conduits are used as much as possible; the conduits of rectangular section or complex form are used when the form of these conduits accomplishes better the requirements of the multiple installations (limitation of space, interior design, *etc*).

3. Analysis and Mathematical Modeling

A conduit of finite length l and constant section S is considered and, inside it, a viscous incompressible fluid is moving in steady and laminar state (Figure 2).

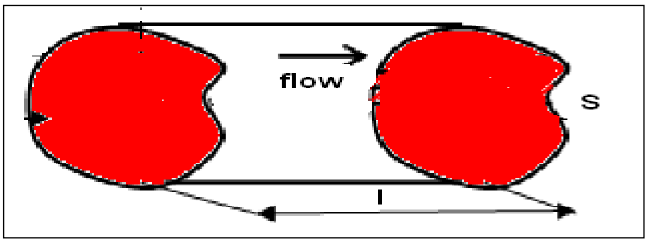


Fig. 2. Problem description

The mathematical model for describing continuous medium of Newtonian viscous incompressible flow is expressed by Navier–Stokes equations which are the equations of conservation of mass, momentum and energy.

For isothermal incompressible flows, energy equation can be dropped and only the momentum and continuity equations are solved to obtain the velocity and pressure fields. These equations are given as

$$\rho \frac{\partial \vec{V}}{\partial t} = -\rho(\vec{V}.\nabla)\vec{V} - \nabla P + \mu \Delta \vec{V} + F_B \tag{1}$$

$$\frac{\partial \rho}{\partial t} + \rho \nabla . \vec{V} = 0 \tag{2}$$

where ρ is the fluid density (taken as constant), V is the velocity vector, p is the fluid pressure, μ is the fluid viscosity, and F_B is the body force, ∇ is the gradient operator; Δ is the Laplace operator and (∇.) is the divergence operator.

The following hypotheses are applied to simplify the above equations:

The flow is the same one in all the cross sections, the speed is constant all along the fluid thread, the movement is uniform and stationary, the flow is two-dimensional and with the revolution symmetry it remains identical to him even in all plans passing by the conduit axis and the density ρ is constant.

The flow is viscous, the speed components (normal and tangential) against wall are null, the heat transfers and heating effects due to frictions are negligible, and the nature of conduit surfaces is smooth.

The Navier-Stokes system can be written in short as

$$(\vec{V}.\nabla)\vec{V} + \frac{1}{\rho}\nabla P - f = \nu \Delta \vec{V} \tag{3}$$

$$\nabla.\vec{V} = 0 \tag{4}$$

with the kinematic viscosity parameter υ = μ/ρ. The term '(V.Δ) V' on the right hand side of Eq. 3 is known as the convective term. ΔV is known as the diffusion term. In the case of diffusion dominated flows the convective term can be dropped.

The body forces 'F' were assumed to act perpendicularly to the flow direction and not appear in Eq. 3. Also, the pressure gradient ∇P is taken as a constant value. Finally, Eq. 3 returns to the following simple form

$$\Delta\vec{V} = \frac{1}{\mu}\nabla P = C^{t} \tag{5}$$

The Eq. 5 is a differential equation of second order called (Poisson's equation) with a constant second term. Considering the presence of friction forces and viscosity to walls, the boundary conditions are of Direchlet type, so that, a null velocity on conduit walls is supposed.

4. Analytical Solutions for a Conduit with a Circular Section

The analytical solutions of Eq. 5 with associated boundary conditions correspond to simplified models of laminar flow, such as the Poiseuille flow which provided exact analytical solutions of this equation for conduits having a circular section, these conduits being most current and simplest to consider.

The flow which runs in conduit having a circular section, and for the reason of symmetry, can be developed and treated by using the cylindrical system (r, θ, z). The equation of continuity is given by

$$\nabla.\vec{V} = \frac{1}{r}\frac{d}{dr}(rVr) + \frac{1}{r}\frac{\partial V\theta}{\partial\theta} + \frac{\partial Vz}{\partial z} = 0 \tag{6}$$

The velocity profile in conduit becomes a function of the component r only, also the velocity V and its derivates ΔV don't depend of θ. So (∂/∂θ=0, Vθ=0) for an axisymmetric flow and (Vr=0) for a parallel flow. Eq. 6 gives

$$(\partial V_z/\partial z=0) \tag{7}$$

Assuming all above assumptions and substituting Eq. 7 in Eq. 5, the cylindrical version of the differential equation of the momentum equation with constant ρ and μ, with no body forces in the z direction, is given by

$$\mu\frac{1}{r}\frac{\partial}{dr}\left(r\frac{dVz}{dr}\right) = \frac{\partial P}{\partial z} \tag{8}$$

Note that the flow is entirely driven by the pressure gradient which is taken as constant. Consequently, velocity profile is given by the following expression

$$Vz(r) = \frac{a}{4\mu} R^2 \left[1 - \frac{r^2}{R^2} \right]. \tag{9}$$

where (∂P/∂z= - a, and a>0).

The velocity distribution is same all along of the conduit for the established mode. In the same way, distribution of the pressures can be determinate along of the conduit.

5. Mathematical Formulation and Analysis

5.1 Finite Elements Formulation

The finite elements method FEM for numerical solution of the incompressible Navier-Stokes equations will be investigate in this part to get simulations and analysis that allow to determinate performances and parameters for conduits with a complex section.

The discretization by FEM method of the Navier-Stokes problem is based on its variational formulation. The considered problem in this formulation is an elliptic problem; of a partial differential equation; presented in a specific form. For the present case, it is given as the Poisson equation in the arbitrary area Ω of an unspecified border (Γ) which is specified by

$$\Delta(W) = f \text{ in } \Omega \tag{10}$$

where W indicates the unknown factors of the problem (in our case, the velocity), (Δ) is the Laplace operator and f is the given function which is taken as constant.

To study this problem mathematically, Galerkin formulation is used to obtain the variational formulation of the problem (note that several variational formulations can be used for the same problem).

In order to adapt the problems into a form which is more useful to practical computation, the unknown functions can be approximated by so-called nodal shape functions (interpolation functions).

The interpolation function is an approximate function of the solution (W) for a typical element which belongs to the area of study. Hence, the simple and basic linear interpolation is still widely used for the incompressible Navier-Stokes equations. An expression for the unknown solution in the element (e) is given by

$$w^e = \sum_{i=1}^{nn} N_i^e w_i^e \tag{11}$$

where N_i and W_i: are respectively the nodal interpolation functions and the value of potential function corresponding to the i^{th} node of the element (e), nn is the number of nodes in the element.

The elemental contributions of the solution can be calculated and summed into a global equations system. The problem (Eq. 10) is equivalent to the linear algebraic system given by

$$[k]\{W\}= \{F\} \tag{12}$$

where [K] is the (N × N) symmetric, positive definite and banded global matrix “stiffness matrix”, and (F) and (W) are the (N x 1) source vector “load vector” and the (N × 1) vector of unknowns (N is the total number of nodes in the studied domain).

The resolution of obtained system, by the finite elements discretization, allows to determinate the unknown (W), which is the velocity distribution in each point of the field.

5.2 Calculus of Flow Parameters

From the velocity values, various parameters of this flow are calculated. The studied section of conduit is presented by the total surface A_{total}. To evaluate this surface, the elementary surfaces A_{ei} of any meshes must be calculated and summed to obtain it, and summation expression of these surfaces elementary is given by

$$A_{total} = \sum_{i=1}^{NEL} Ae_i \tag{13}$$

The elementary surfaces depend on the mesh type and its form. The characteristic dimension DH (hydraulic diameter) is calculated for various geometries of the studied conduits by the following expression given as

$$D_H = \frac{4A_{total}}{P_m} \tag{14}$$

where Pm is the perimeter of meshed section. It will be calculated in the program for each case using the border nodes. The maximum velocity W_{max} is obtained by numerical comparison between various nodal velocities. The mode of flow is laminar, Reynolds number is defined according to hydraulic diameter and mean velocity of the flow by the following formula

$$\Re_e = \frac{\rho W_{moy} D_H}{\mu} \tag{15}$$

where ρ and μ are constants of considered fluid. However, others parameters can be calculate such as the pressure losses, the friction K.

6. Applications and Discussion

6.1 Model Validation and Performances

The conduit with a circular geometry which has an analytical solution is chosen and validation was performed. Table 1 presents a comparison between the analytical values and the numerical values obtained by the developed code for a mesh which is realized with 121 nodes and 200 triangular elements (the line of scan comparison was mentioned on the table).

The agreement between the analytical solutions and those obtained numerically is observed to be very good.

The obtained results approach towards the analytical solution by an average error of 0.005. The solution to calculate velocity field was validated by comparison with analytical results given for configurations of a simple geometry. The code validation was made. So, the developed approach allows to obtain velocity fields and profiles in various cases.

Several parameters (number of nodes, number of elements, interpolation function) can affect the precision and the reliability of the solution. Different meshes with triangular or quadratic elements can be implicated with a different number of nodes, which gives the possibility to apply various interpolation functions forms.

Table 2 gives results obtained from various applications for different triangular meshes with various numbers of nodes (meshes realized with 121, 961 and 3721 nodes).

Table 1
Evolution of velocity field for a conduit of circular section

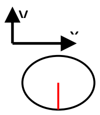
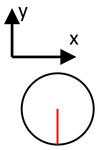
Dp=1 R=0.2 μ=1	COORDONATES:		APPLICATIONS:		
	y	x	num (121)	analytic	error
	-0.20	0	0.00E+00	-8.32E-19	0.00E+00
	-0.16	0	3.57E-003	3.60E-003	0.03E-003
	-0.12	0	6.35E-003	6.40E-003	0.05E-003
	-0.08	0	8.33E-003	8.40E-003	0.07E-003
	-0.04	0	9.53E-003	9.60E-003	0.07E-003
	0	0	9.92E-003	1.00E-002	0.08E-003

Table 2
Influence of the number of the nodes.

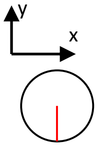
Dp=1 R=0.2 μ=1	COORDONATES:		APPLICATIONS:			
	y(i)	x(i)	1(121)	2(961)	3(3721)	analytic
	-0.2	0	0.00E+0	0.00E+00	0.00E+00	-8.32E-19
	-0.16	0	3.57E-003	3.60E-003	3.60E-003	3.60E-003
	-.12	0	6.35E-003	6.39E-003	6.40E-003	6.40E-003
	-0.08	0	8.33E-003	8.39E-003	8.40E-003	8.40E-003
	-0.04	0	9.53E-003	9.59E-003	9.60E-003	9.60E-003
	0	0	9.92E-003	9.98E-003	1.00E-002	1.00E-002

The approach towards the analytical solution is more fast and precise if the number of used nodes is augmented.

Table 3 gives results obtained from two applications. The first one consists to use triangular mesh with 121 nodes and 200 elements, and the second uses quadratic mesh with 121 nodes and 100 elements. In this case, the interpolation function must be changed according to the number of nodes of mesh element and mesh element type itself.

Table 3
Influence of the interpolation function

Dp=1 R=0.2 μ=1	COORDONATES:		APPLICATIONS:		
	y(i)	x(i)	Element tri	Element quadr	analytic

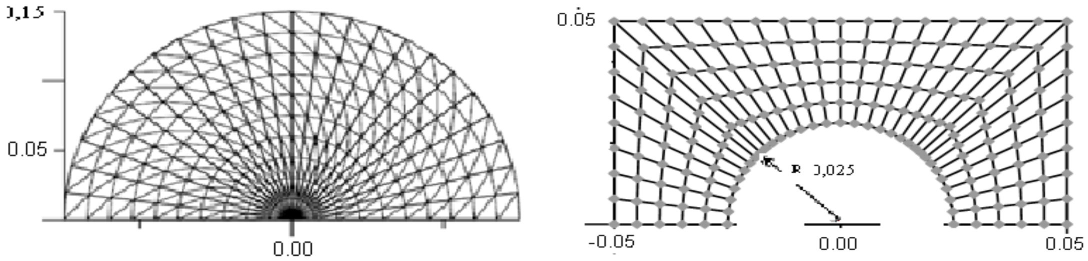
	-0.2	0	0.00E+00	0.00E+00	-8.32E-19
	-0.16	0	3.57E-003	3.60E-003	3.60E-003
	-.12	0	6.35E-003	6.40E-003	6.40E-003
	-0.08	0	8.33E-003	8.40E-003	8.40E-003
	-0.04	0	9.53E-003	9.60E-003	9.60E-003
	0	0	9.92E-003	1.00E-002	1.00E-002

The approach towards the analytical solution is faster and more precise if a good type of the interpolation function is chosen. The analysis illustrates that the number of nodes, the number of elements and the interpolation function have an important influence on the determination of velocity profile. But this advantage must be controlled and well optimized in order to limit run time cost and increase the precision and the reliability of the code results.

6.2 Applications

After the validation and analysis of the applicable model performances, this model can apply for various geometries, it can be useful to performed and calculate the velocity field and its distributions.

Figure 3 presents some meshes output for different geometries and different mesh type. (Figure 3 (a)) presents an example for triangular meshes (251 nodes and 456 elements) applied to a conduit with a section of half-circle and (Figure 3 (b)) presents an example for quadratic meshes (198 nodes and 240 elements) applied to a conduit with a complex form.



(a) (b)

Fig. 3. Meshes examples

Figure 4 presents some simulations that illustrate the velocity distribution for non circular sections. Each case demonstrates the result of solving the viscous incompressible flow problem on a chosen conduit section using the developed approach.

For Figure 4(a), the velocity field distributions are obtained for a non circular conduit with a simple section (application for a rectangular geometry 100mm x 200mm with 1116 nodes and 2100 triangular elements, dp=1, µ=1). And for figure 4b, the velocity field distributions are obtained for a conduit with a complex section (application with 441 nodes and 400 quadratic elements, dp=1, µ=1).

Figure 4(b) shows clearly how the section form can affect the distribution of the velocity field. The uniformity of the velocity field observed through simple geometries is directly affected in the case of complex section.

The velocities near the conduit walls are lower. The maximum velocity is at the centre of the conduit that present a perfect and total symmetric section (circle, rectangle, elliptic).

Simulations indicate that the zone of maximum velocity departs from the conduit centerline when the conduit section is complex and moves toward largest zones which present higher difference between their bounded walls. The velocity distribution becomes unsymmetrical with the dominance of effects of walls and viscosity.

Also, from the velocity values, others parameters can be calculated which enables to analyze the influence of some parameters on the flow move that is important to upgrading the system performances.

The numerical examples in the previous figures illustrate the results obtained for the various conduit sections, and also enable to check the velocity profile and the properties of the flow on different cases for which analytical solutions are not available.

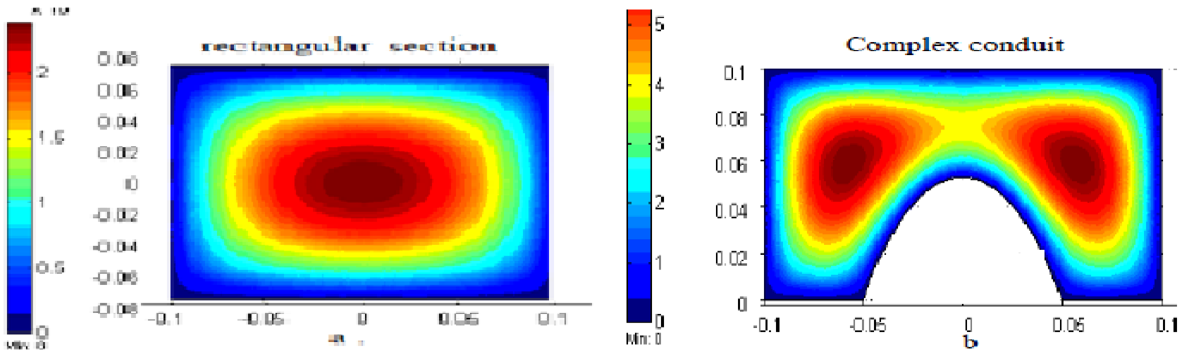


Fig. 4. Velocity distribution for non-circular sections

7. Conclusion

The comprehension of viscous incompressible flow problems and also their solutions is not always obvious, especially when the studied system requires the applicability of complex sections.

In this paper, a computational methodology to simulate viscous incompressible laminar flows was investigated. The description of the numerical approach was accompanied by a theoretical analysis to more understand the performance of the selected method.

The finite element method was implemented through a computational application which makes it more effective and reliable.

The model validation was made for simple two-dimensional cases. Numerical results converge quickly towards the analytical solution with an average error of 0.005 that enables to use a computational code to carry out simulations for more complex geometries.

The velocity field is directly affected by the section form, its distribution becomes unsymmetrical and the zone of maximum velocity departs from the conduit centerline in the case of complex section.

Characteristic results of the viscous incompressible flow parameters are given along the chosen section once the velocity profile is calculated.

The model can be easily extended and adapted to study more proprieties of viscous incompressible flow trough different conduits.

References

[1] Chung, T. J. *Computational fluid dynamics.* Cambridge university press, 2010.

[2] Wesseling, Pieter, and Cornelis W. Oosterlee. "Geometric multigrid with applications to computational fluid dynamics." *Journal of computational and applied mathematics* 128, no. 1-2 (2001): 311-334.

[3] Ronalds. L*, Incompressible Flow*, 2nd Edition: Texas, USA (1996).

[4] Manjunatha. G, Rajshekhar V. Choudhary. "Slip effects on peristaltic transport of Casson fluid in an inclined elastic tube with porous walls", *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 43, no. 1(2018): 67-80.

[5] Bennoud S, Azzazen M. "Simulation of viscous incompressible flow through a conduit with isothermal walls." *International Workshop on the Complex Turbulent Flows (IWCTF2017), Tangier, Morocco, November*, (2017): 27-28

[6] Sorour. M. M, Teamah M. A, Teamah W. M. and Abdel Aziz H. M. "Numerical Simulation of Double Diffusive Laminar Mixed Convection in a Horizontal Rotating Annulus", *CFD Letters* 6, no.1 (2014): 32-45.

[7] Rashidi. M. M and Sadri. S. M. "Solution of the Laminar Viscous Flow in a Semi-Porous Channel in the Presence of a Uniform Magnetic Field by Using the Differential Transform Method", *Int. J. Contemp. Math. Sciences*, 15, no 5 (2010): 711-720.

[8] Jung, Seo Yoon, and Yongmann M. Chung. "Large-eddy simulation of accelerated turbulent flow in a circular pipe." International Journal of Heat and Fluid Flow 33, no. 1 (2012): 1-8.

[9] Bennoud. S, Habbouche. H, Belheouane. F. I. "Simulation and calculation of various parameters of a liquid flow in conduits with complex form", *Forth Environmental Symposium of German- Arab Scientific Forum for Environmental Studies, Fez, Morocco*, (2008).

[10] Munson, B. R., and D. D. Joseph. "Viscous incompressible flow between concentric rotating spheres. Part 1. Basic flow." *Journal of Fluid Mechanics* 49, no. 2 (1971): 289-303.

[11] Nigro, F. E. B., A. B. Strong, and S. A. Alpay. "A numerical study of the laminar viscous incompressible flow through a pipe orifice." *Journal of Fluids Engineering* 100, no. 4 (1978): 467-472.

[12] Vasanth. K. R, Hanumagowda. B, Jogyalla. S. K, Raju. B. T. "Combined Effect of Piezo-Viscous Dependency and Non-Newtonian Couple Stresses in Porous Squeeze-Film Circular Plate", *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 51, no 2 (2018): 158-168.

[13] Kracmar, S., and J. Neustupa. "Modelling of flows of a viscous incompressible fluid through a channel by means of variational inequalities." *Zeitschrift für angewandte Mathematik und Mechanik* 74, no. 6 (1994): T637-T639.

[14] Chorin, Alexandre Joel. "Numerical solution of the Navier-Stokes equations." *Mathematics of computation* 22, no. 104 (1968): 745-762.

[15] Chakraborty. S,.Borkakati A.K. "Effect of variable viscosity on laminar convection flow of an electrically conducting fluid in uniform magnetic field", *Theoretical and Applied Mechanics* 27, (2002): 49-61.

[16] Szumbarski, Jacek. "Instability of viscous incompressible flow in a channel with transversely corrugated walls." *Journal of Theoretical and Applied Mechanics* 45, no. 3 (2007): 659-683.

[17] Bristeau, Marie Odile, Roland Glowinski, and Jacques Periaux. "Numerical methods for the Navier-Stokes equations. Applications to the simulation of compressible and incompressible viscous flows." *Computer Physics Reports* 6, no. 1-6 (1987): 73-187.

[18] Brown, David L., Ricardo Cortez, and Michael L. Minion. "Accurate projection methods for the incompressible Navier–Stokes equations." *Journal of computational physics* 168, no. 2 (2001): 464-499.

[19] Becker, Roland. *An adaptive finite element method for the incompressible Navier-Stokes equations on time-dependent domains*. IWR, 1995.

[20] Braack, Malte. "An adaptive finite element method for reactive flow problems." PhD diss., IWR, 1998.

[21] Gresho, Philip M., and Robert L. Sani. "Incompressible flow and the finite element method. Volume 1: Advection-diffusion and isothermal laminar flow." (1998).

[22] Gunzburger, Max D. *Finite element methods for viscous incompressible flows: a guide to theory, practice, and algorithms*. Elsevier, 2012.

[23] Masatak. M. Million M, The finites elements methods and its application, Publishing Company: New York (1986).

[24] Miccal T, Yvonne M, "Lubrication analysis and numerical simulation of the viscous micropump with slip", *International Journal of Heat and Fluid Flow* 33, (2012): 22-34.

[25] Xu, Hang, Zhi-Liang Lin, Shi-Jun Liao, Jie-Zhi Wu, and Joseph Majdalani. "Homotopy based solutions of the Navier–Stokes equations for a porous channel with orthogonally moving walls." *Physics of Fluids* 22, no. 5 (2010): 053601.