

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible Newtonian Fluids using FreeFem++ based on Finite Element Method

M. M. Rhaman¹ and K. M. Helal²

¹Department of EEE, Southeast University, Dhaka, Bangladesh
Email: mahfujst59@yahoo.com

²Department of Mathematics, Comilla University, Comilla, Bangladesh
Email: mailtohelal@yahoo.com

Received 13 March 2014; accepted 30 March 2014

Abstract. The goal of this paper is concerned to numerical approach of the unsteady Navier-Stokes equations for incompressible Newtonian fluids based on finite element method and we present here the numerical simulations implemented with FreeFem++. We first give the constitutive formulation of these equations. The unknowns are \mathbf{u} the velocity and p the pressure. The constitutive equations lead to a non-linear elliptic system of partial differential equations for (\mathbf{u}, p) . We find the variational formulation of the unsteady Navier-Stokes equations and obtain the results of numerical simulations through a programming code developed in FreeFem++. The approximation of the velocity and pressure are P_2 continuous and P_1 continuous finite element respectively.

Keywords: Navier-Stokes Equations, Finite Element Method, FreeFem++

AMS Mathematics Subject Classification (2010): 76Dxx, 76D99, 76E09

1. Introduction

In this paper we study the numerical solutions of the unsteady Navier-Stokes equations for incompressible Newtonian fluids based on finite element method (FEM) and we use FreeFem++ (see Hecht [10]) to obtain the numerical simulations. We deduce the constitutive equations of unsteady Navier-Stokes problem. These constitutive equations consist of highly non-linear system of partial differential equations of elliptic type. The velocity \mathbf{u} and the pressure p are the unknowns. We assume that the solution (\mathbf{u}, p) is regular enough. The approximate velocity and pressure are respectively P_2 continuous and P_1 continuous finite element. We first introduce the conservation of laws (see Shaughnessy, Katz and Schaffer (2005) [14], Chorin and Marsden (2000) [6], Quarteroni and Valli (1994) [12]) and formulate the constitutive equations of unsteady Navier-Stokes for incompressible Newtonian fluids. Then the variational formulation of these

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible Newtonian Fluids using FreeFem++ based on Finite Element Method

constitutive equations is derived and finite element approximation of this problem is introduced (see [5, 11, 12, 16]). All meshes and simulations are done in FreeFem++. Using the variational formulation we develop a programming code in FreeFem++ to find (\mathbf{u}, p) from the Navier-Stokes equations. We consider a well known benchmark flow problem, namely the Kim-Moin model problem whose exact solution is known, to validate the code. Finally, some conclusions and perspective of future works are discussed.

2. The conservation laws

Conservation laws states the physical principles governing the fluid motion. Taking into account the Lavoisier law: “*in nature nothing is created, nothing is lost, everything is transformed*”, we can deduce the basic principles of conservation. According to the conservation laws, a particular measurable property of an isolated physical system does not change as the system evolves. We consider flows of an incompressible Newtonian homogenous fluid in a bounded domain $\Omega \subset \mathbb{R}^2$ with boundary $\partial\Omega$. The mathematical formulations of these conservation laws are as follows:

Conservation law of mass: Conservation of mass is a fundamental principle of classical mechanics. This means that “*mass is neither created nor destroyed*”. This way, during motion the mass of the body remain unchanged. In a fixed region, the total time rate of change of mass is identically zero.

The differential equation expressing conservation of mass is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (1)$$

where ρ is the density of the fluid, \mathbf{u} is the velocity vector. This equation is also called the continuity equation.

If the density is a constant, then the flow of the fluid is incompressible and the conservation of mass is expressed as

$$\nabla \cdot \mathbf{u} = 0 \quad (2)$$

Conservation law of momentum: The consequences of body motion cannot be described only by a velocity, they also depend on the mass. So, we use the *momentum of mass* ($mass \times velocity$) to relate them. The conservation law of momentum is the extension of the famous Newton’s second law of motion, “*force= mass×acceleration*”. For a moving flow field this law describe that the total time rate of change of linear momentum or acceleration of a fluid element is equal to the sum of externally applied forces on a fixed region. The equation of conservation of momentum is given by

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = \nabla \cdot \mathbf{T} + \rho \mathbf{f} \quad (3)$$

where \mathbf{T} is the symmetric tensor field, called Cauchy stress tensor and \mathbf{f} is an external force.

3. The constitutive law and problem formulation

The constitutive law relates the Cauchy stress tensor with the kinematics of different quantities, in particular, the velocity field. These relations allow us to characterize the mechanic behavior of fluid. In this work we are concerned with fluids, obeying a Newtonian behavior. The Newtonian fluids are a subclass of isotropic (direction independent) viscous fluids to which the stress tensor \mathbf{T} is the sum of the tension caused by the thermodynamic pressure in the fluid, the tension that causes deformation fluid and the tension due to volumetric expansion. These fluids are called Stokesians Fluids. Newtonian fluids are isotropic viscous fluids to which the stress tensor \mathbf{T} is given by

$$\mathbf{T} = -p\mathbf{I} + \eta\nabla \cdot \mathbf{u} + 2\mu\mathbf{D}(\mathbf{u})$$

where η , the volumic viscosity, multiplies the tension due to volumetric expansion and μ , the hydrodynamic viscosity multiplies the tension which contributes to the motion of the fluid. These viscosities verify the relations $3\eta + 2\mu \geq 0$ and $\mu \geq 0$. In a Newtonian incompressible fluid, the Cauchy stress tensor is a linear function of the strain tensor. The Cauchy stress tensor can be written in the form

$$\mathbf{T} = -p\mathbf{I} + 2\mu\mathbf{D}(\mathbf{u}) = -P\mathbf{I} + \mu[\nabla\mathbf{u} + (\nabla\mathbf{u})'] \quad (4)$$

where the term $2\mu\mathbf{D}(\mathbf{u})$ is often referred as viscous stress component of the stress tensor.

As example of compressible Newtonian fluids, we refer the following gases: oxygen, hydrogen, air, methane and ammonia. As example of incompressible Newtonian fluids we refer the following liquids: water, gasoline, olive oil.

Considering that μ is constant and \mathbf{T} as in (4), for the Newtonian incompressible fluid, conservation law of momentum (3) can be written as

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = \nabla \cdot \mathbf{T} + \rho \mathbf{f}$$

$$\Rightarrow \rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = \nabla \cdot [-p\mathbf{I} + 2\mu\mathbf{D}(\mathbf{u})] + \rho \mathbf{f}$$

After simplifying we get

$$\Rightarrow \rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + 2\mu\nabla \cdot \mathbf{D}(\mathbf{u}) + \rho \mathbf{f}$$

Considering ρ as a constant, we define the kinematic viscosity $\nu = \frac{\mu}{\rho}$ (m^2/s) and the

scaled pressure $\frac{p}{\rho}$ (m^2/s^2) still denoted by p and we obtain

$$\Rightarrow \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p - 2\nu\nabla \cdot \mathbf{D}(\mathbf{u}) = \mathbf{f} \quad (5)$$

The Navier-Stokes equations for incompressible fluids is the system of equations formed by the partial differential equations of the law of conservation of mass (2) and the momentum equations (5)

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible
Newtonian Fluids using FreeFem++ based on Finite Element Method

$$\begin{cases} \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p - \nu \Delta \mathbf{u} = \mathbf{f} & \text{in } \Omega, \\ \nabla \cdot \mathbf{u} = 0 & \text{in } \Omega, \end{cases} \quad (6)$$

If $\nabla \cdot \mathbf{u} = 0$ (from (2)), then $2\nabla \cdot \mathbf{D}(\mathbf{u}) = 2\nabla \cdot \frac{1}{2} [\nabla \mathbf{u} + (\nabla \mathbf{u})^t] = \Delta \mathbf{u}$.

So, the conservation of momentum can be written as

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p - \nu \Delta \mathbf{u} = \mathbf{f} \quad (7)$$

Using (7), we can rewrite the Navier-Stokes equations for the incompressible fluids as follows:

$$\begin{cases} \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p - \nu \Delta \mathbf{u} = \mathbf{f} & \text{in } \Omega, \\ \nabla \cdot \mathbf{u} = 0 & \text{in } \Omega. \end{cases} \quad (8)$$

Here Ω is a bounded domain of \mathbb{R}^d , $d = 2, 3$ with Lipschitz continuous boundary $\partial\Omega$. To close mathematical formulation and obtain a well-posed problem, the above equations need to be supplemented by some boundary conditions. For simplicity, we consider the case in which the system of differential equations (8) is equipped with the Dirichlet boundary conditions $\mathbf{u} = \mathbf{g}$ on $\partial\Omega$ (adherence conditions). The condition $\mathbf{g} = 0$ is called the homogeneous Dirichlet boundary conditions (or no-slip boundary conditions) i.e., $\mathbf{u} = 0$ on $\partial\Omega$, which describes a fluid confined into a domain with fixed boundary (the boundary is at rest).

So, with the homogeneous Dirichlet boundary conditions defined over Ω , we can write the unsteady Navier-Stokes problem as follows:

$$\begin{cases} \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p - \nu \Delta \mathbf{u} = \mathbf{f} & \text{in } \Omega, \\ \nabla \cdot \mathbf{u} = 0 & \text{in } \Omega, \\ \mathbf{u} = 0 & \text{on } \partial\Omega. \end{cases} \quad (9)$$

4. Variational formulation

We use different function spaces with different notations details of which can be found in (Adams and Fournier (2003)[1], Brezis(2011) [4]). Without loss of generality, we consider an incompressible fluid confined into a domain with fixed boundary. Mathematically, for each $t \in [t_0, T]$ (to simplify, we take from now $t_0 = 0$), we write the unsteady Navier-Stokes equations as

Given \mathbf{f} , find (\mathbf{u}, p) such that

$$\begin{cases} \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p - \nu \Delta \mathbf{u} = \mathbf{f} & \text{in } [0, T] \times \Omega, \\ \nabla \cdot \mathbf{u} = 0 & \text{in } [0, T] \times \Omega, \\ \mathbf{u} = 0 & \text{on } [0, T] \times \partial \Omega, \\ \mathbf{u}(t = 0, \mathbf{x}) = \mathbf{u}_0(\mathbf{x}), \forall \mathbf{x} \in \Omega. \end{cases} \quad (10)$$

where \mathbf{f} is a given external force field per unit mass, \mathbf{u} is the velocity field, \mathbf{u}_0 is the known initial velocity field, p is the rate between the pressure and the density and ν is the constant kinematic viscosity.

The variational or weak formulation of Navier-Stokes equation consists of the integral equations over Ω obtained by integration, after multiplying the momentum equation and continuity equation by appropriate test functions. Let us suppose that $\mathbf{u} \in C^2([0, T] \times \overline{\Omega})$ and $p \in C^1([0, T] \times \overline{\Omega})$ are the classical (or strong) solution of (10). Consider two Hilbert spaces $\mathbf{V} = \mathbf{H}_0^1(\Omega)$ and $Q = L_0^2(\Omega)$ and take $\mathbf{v} \in \mathbf{V}$ and $q \in Q$ be two arbitrary test functions. Applying the Green's formula for the integration by parts and taking into account that \mathbf{v} vanishes on the boundary and after simplifying we get the variational formulation of the Navier-Stokes problem as:

$\forall t \in [0, T]$, given $\mathbf{f} \in L^2(0, T; \mathbf{H}^{-1}(\Omega))$ and $\mathbf{u}_0 \in \mathbf{H}_0^1(\Omega)$ with $\nabla \cdot \mathbf{u}_0 = 0$, find $(\mathbf{u}, p) \in L^2(0, T; \mathbf{V}) \times L^2(0, T; Q)$ such that

$$\begin{cases} \int_{\Omega} \frac{\partial \mathbf{u}}{\partial t} \cdot \mathbf{v} + \int_{\Omega} (\mathbf{u} \cdot \nabla \mathbf{u}) \cdot \mathbf{v} - \int_{\Omega} p \nabla \cdot \mathbf{v} + 2\nu \int_{\Omega} \mathbf{D}(\mathbf{u}) : \mathbf{D}(\mathbf{v}) = \int_{\Omega} \mathbf{f} \cdot \mathbf{v} \\ \int_{\Omega} q \nabla \cdot \mathbf{u} = 0 \\ \mathbf{u}(0) = \mathbf{u}_0 \end{cases} \quad (11)$$

for all $(\mathbf{v}, q) \in \mathbf{H}_0^1(\Omega) \times L_0^2(\Omega)$. Here $\mathbf{D}(\mathbf{u}) = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$ is the deformation tensor.

Taking into the definitions of the following bilinear and trilinear forms:

$$a(\mathbf{u}, \mathbf{v}) = 2\nu(\mathbf{D}(\mathbf{u}), \mathbf{D}(\mathbf{v})) = 2\nu \int_{\Omega} \mathbf{D}(\mathbf{u}) : \mathbf{D}(\mathbf{v}),$$

$$b(\mathbf{v}, p) = -(p, \nabla \cdot \mathbf{v}) = - \int_{\Omega} p \nabla \cdot \mathbf{v},$$

and
$$c(\mathbf{w}, \mathbf{u}, \mathbf{v}) = ((\mathbf{w} \cdot \nabla) \mathbf{u}, \mathbf{v}) = \int_{\Omega} (\mathbf{w} \cdot \nabla) \mathbf{u} \cdot \mathbf{v}$$

we can reformulate the variational formulation of the Navier-Stokes problem as follows:

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible
Newtonian Fluids using FreeFem++ based on Finite Element Method

$\forall t \in [0, T]$, given $\mathbf{f} \in L^2(0, T; \mathbf{H}^1(\Omega))$ and $\mathbf{u}_0 \in \mathbf{H}_0^1(\Omega)$ with $\nabla \cdot \mathbf{u}_0 = 0$, find $(\mathbf{u}, p) \in L^2(0, T; \mathbf{V}) \times L^2(0, T; Q)$ such that

$$\begin{cases} \left(\frac{\partial \mathbf{u}}{\partial t}, \mathbf{v} \right) + a(\mathbf{u}, \mathbf{v}) + c(\mathbf{u}, \mathbf{u}, \mathbf{v}) + b(\mathbf{v}, p) = (\mathbf{f}, \mathbf{v}) \\ b(\mathbf{u}, q) = 0 \\ \mathbf{u}(0) = \mathbf{u}_0 \end{cases} \quad (12)$$

for all $(\mathbf{v}, q) \in \mathbf{H}_0^1(\Omega) \times L_0^2(\Omega)$.

It can be proved [9] that the problem (12) is well-posed and equivalent to (10). The existence and uniqueness of theorem for the solutions of Navier-Stokes system can be found in (Galdi (1994) [7], Girault and Raviart (1986) [8], Temam (1984) [15], Girault and Raviart (1979) [9]).

5. Finite element approximation

We consider finite element method (FEM) to approximate the numerical solutions of Navier-Stokes problem (12). The FEM is a method which approaches the solution of partial differential equations (PDEs) and is a general technique for constructing approximate solutions to boundary value problems in dimension $d, (d \leq 3)$. All results will be presented here for the two-dimensional case, where we will do the application of these concepts and presentation of numerical simulations. Although there are several types of finite elements, in the following, we deal only with the discretization of the Navier-Stokes problem, using a Lagrange Finite Element of type $P_2 - P_1$. The solution (\mathbf{u}, p) of the problem (12) lives in a space of infinite dimension. In this circumstance, it is generally impossible to calculate the exact solution. Then we determine an approximation of \mathbf{u} and p , respectively \mathbf{u}_h and p_h , each one defined in finite dimensional appropriate spaces V_h , such that $\dim V_h = I(h) (\lim_{h \rightarrow 0} I(h) = +\infty)$ and dependent on a parameter $h > 0$. These spaces are formed by polynomials and for all function v_h in V_h (in particular \mathbf{u}_h and p_h for the appropriate spaces) we have

$$v_h = \sum_{i=1}^I \alpha_i \varphi_i, \alpha \in \mathbf{IR}, i = 1, \dots, I, \text{ where } \{\varphi_1, \varphi_2, \dots, \varphi_I\} \text{ is a basis of } V_h.$$

This is the principle of the Finite Element Method. The FEM can be studied in details in (Brenner and Scott (1994) [3], [12], Becker et al. (1981) [2], Girault and Raviart (1979) [9]).

We use classical Galerkin method to find the solution. We consider Galerkin's method for constructing approximate solutions to the variational boundary-value problem (11) or its abstract formulation (12). Galerkin's method consists of seeking an approximate solution (11) in a finite-dimensional subspace V_h of the space of

admissible functions where the solution lies in this subspace rather than in the whole space. The natural Galerkin approximation for problem (10) is a mixed method which is based on Lagrange multiplier formulations of constrained problems. We refer to mixed approximation methods as those associated to the approximation of saddle point problems, in which there are two bilinear forms and two approximation spaces satisfying a compatibility condition (see [12]).

We discretize problem (12). Let $\{\mathbf{T}_h\}_{h>0}$ be a family of triangulations and h denotes a discretization parameter and let \mathbf{V}_h and Q_h be two finite dimensional spaces such that $\mathbf{V}_h \in \mathbf{H}^1(\Omega)$ and $Q_h \in L^2(\Omega)$. We let $\mathbf{V}_h^0 := \mathbf{V}_h \cap \mathbf{H}_0^1(\Omega)$ and $M_h := Q_h \cap L_0^2(\Omega)$. In these spaces, the discrete finite element approximation problem of (12) can be written as follows:

for each $t \in [0, T]$, $\mathbf{u}_{0,h} \in \mathbf{V}_h^0$, find $(\mathbf{u}_h, p_h) \equiv (\mathbf{u}_h(t, \cdot), p_h(t, \cdot)) \in \mathbf{V}_h^0 \times M_h$ such that

$$\begin{cases} \frac{d}{dt}(\mathbf{u}_h, \mathbf{v}_h) + a(\mathbf{u}_h, \mathbf{v}_h) + c(\mathbf{u}_h, \mathbf{u}_h, \mathbf{v}_h) + b(\mathbf{v}_h, p_h) = (\mathbf{f}, \mathbf{v}_h) & \forall \mathbf{v}_h \in \mathbf{V}_h^0, \\ b(\mathbf{u}_h, q_h) = 0 & \forall q_h \in M_h, \\ \mathbf{u}_h(0) = \mathbf{u}_{0,h} \end{cases} \quad (13)$$

which can be written as

for each $t \in [0, T]$, $\mathbf{u}_{0,h} \in \mathbf{V}_h^0$, find $(\mathbf{u}_h, p_h) \equiv (\mathbf{u}_h(t, \cdot), p_h(t, \cdot)) \in \mathbf{V}_h^0 \times M_h$ such that

$$\begin{cases} \int_{\Omega} \frac{\partial \mathbf{u}_h}{\partial t} \cdot \mathbf{v}_h + \int_{\Omega} (\mathbf{u}_h \cdot \nabla \mathbf{u}_h) \cdot \mathbf{v}_h - \int_{\Omega} p_h \nabla \cdot \mathbf{v}_h + 2\nu \int_{\Omega} \mathbf{D}(\mathbf{u}_h) : \mathbf{D}(\mathbf{v}_h) = \int_{\Omega} \mathbf{f} \cdot \mathbf{v}_h & \forall \mathbf{v}_h \in \mathbf{V}_h^0 \\ \int_{\Omega} q_h \nabla \cdot \mathbf{u}_h = 0 & \forall q_h \in M_h \\ \mathbf{u}_h(0) = \mathbf{u}_{0,h} \end{cases} \quad (14)$$

As motion is non-stationary we need to discretize the Navier-Stokes equations over time. There are several methods of time discretization. In this paper we use Characteristic Galerkin Method which associates backward Euler scheme of first order defined by

$$\frac{\partial \mathbf{u}}{\partial t}(t^{n+1}, \cdot) = \frac{\mathbf{u}(t^{n+1}, \cdot) - \mathbf{u}(t^n, \cdot)}{t^{n+1} - t^n}$$

The Characteristic Galerkin Method evaluates time derivatives of vector field on Lagrangian frame, appealing to characteristic lines or trajectories described by a material particle when it has been driven by the field at the velocity of the field. We describe the motion of material particle of Newtonian fluid during the time interval $[t_0, t_1] \subset [0, T]$, ($T > 0$), which was in position ξ at instant t_0 by

$$\chi_{t;t_0}: \xi \rightarrow \chi(t; t_0, \xi)$$

and define its characteristics line or trajectory, with the same flow direction, by the only solution of Cauchy problem

$$\begin{cases} \frac{d\chi}{dt}(t; t_0, \xi) = \mathbf{u}(t; \chi(t; t_0, \xi)), t \in]0, T[\\ \chi(t_0; t_0, \xi) = \xi \end{cases}$$

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible
Newtonian Fluids using FreeFem++ based on Finite Element Method

Now taking an uniform mesh of $[0, T]$ defined by $t^n = n\Delta t, n = 0, \dots, \frac{T}{\Delta t}$, Δt being the time step and applying the backward Euler scheme we can write the scheme for the problem (10), denoting $\mathbf{u}^n \circ \chi^n(\mathbf{x}) \approx \mathbf{u}^n(\mathbf{x} - \Delta t \mathbf{u}(t^{n+1}, \mathbf{x})) = \mathbf{u}^n(\mathbf{x}^*)$

$$\begin{cases} \mathbf{u}^{n+1} + \Delta t \nabla p^{n+1} - 2\nu \Delta t \nabla \cdot \mathbf{D}(\mathbf{u}^{n+1}) = \Delta t \mathbf{f}^{n+1} + \mathbf{u}^n(\mathbf{x}^*), \\ \nabla \cdot \mathbf{u}^{n+1} = 0, \\ \mathbf{u}^{n+1} = 0, \quad \mathbf{u}^0 = \mathbf{u}_0 \end{cases} \quad (15)$$

The discrete variational formulation of (15) is as follows:

for each $t^{n+1} = (n+1)\Delta t \in [0, T]$, ($n \in \mathbf{N}_0$) given $\mathbf{u}_h^0 = \mathbf{u}_h(0)$, find

$$\begin{aligned} &(\mathbf{u}_h^{n+1}, p_h^{n+1}) \in \mathbf{V}_h^0 \times M_h \text{ such that} \\ &\begin{cases} (\mathbf{u}_h^{n+1}, \mathbf{v}_h) - \Delta t (p_h^{n+1}, \nabla \cdot \mathbf{v}_h) + 2\nu \Delta t (\mathbf{D}(\mathbf{u}_h^{n+1}), \mathbf{D}(\mathbf{v}_h)) = (\mathbf{g}, \mathbf{v}_h) & \text{in } \Omega, \\ (\nabla \cdot \mathbf{u}_h^{n+1}, p_h^{n+1}) = 0 & \text{in } \Omega. \end{cases} \end{aligned} \quad (16)$$

where $\mathbf{g}^{n+1} = \Delta t \mathbf{f}^{n+1} + \mathbf{u}^n(\mathbf{x}^*)$.

Let $\{\varphi_i\}_{i=1, \dots, N_h}$ and $\{\psi_j\}_{j=1, \dots, m_h}$ be the Lagrange bases of the spaces \mathbf{V}_h^0 and M_h respectively. Given \mathbf{u}_h^n , we express the corresponding approximate solutions $\mathbf{u}_h^{n+1} = (u_{1,h}^{n+1}, u_{2,h}^{n+1})$ and p_h^{n+1} in the basis of \mathbf{V}_h^0 and M_h

$$u_{1,h}^{n+1} = \sum_{j=1}^{N_h} u_{1,j}^{n+1} \varphi_j, \quad u_{2,h}^{n+1} = \sum_{j=1}^{N_h} u_{2,j}^{n+1} \varphi_j, \quad p_h^{n+1} = \sum_{k=1}^{m_h} p_k^{n+1} \psi_k \text{ and with the test functions}$$

$\varphi_i \in \mathbf{V}_h^0$ and $\psi_k \in M_h$, we obtain the following linear algebraic system in matricial form as:

$$\begin{bmatrix} \mathbf{A}_1 & \mathbf{A}_2 & \mathbf{B}_x \\ \mathbf{A}_2^t & \mathbf{A}_1 & \mathbf{B}_y \\ \mathbf{B}_x^t & \mathbf{B}_y^t & 0 \end{bmatrix} \begin{bmatrix} u_1^{n+1} \\ u_2^{n+1} \\ p^{n+1} \end{bmatrix} = \begin{bmatrix} \mathbf{b}_1 \\ \mathbf{b}_2 \\ 0 \end{bmatrix} \quad (17)$$

where $\mathbf{u}_k^{n+1} = [u_{k,1}^{n+1}, \dots, u_{k,N_h}^{n+1}]$ and $p^{n+1} = [p_1^{n+1}, \dots, p_{m_h}^{n+1}]$ are the vectors of unknown degree of freedom and

$$\begin{aligned} \mathbf{A}_1 &= [\mathbf{A}_{1ij}]_{N_h \times N_h} = \left[\int_{\Omega} \left[\varphi_i \varphi_j + \nu \Delta t \left(2 \frac{\partial \varphi_i}{\partial x} \frac{\partial \varphi_j}{\partial x} + \frac{\partial \varphi_i}{\partial y} \frac{\partial \varphi_j}{\partial y} \right) \right] \right]_{N_h \times N_h} \\ \mathbf{A}_2 &= [\mathbf{A}_{2ij}]_{N_h \times N_h} = \left[\int_{\Omega} \frac{\partial \varphi_i}{\partial x} \frac{\partial \varphi_j}{\partial y} \right]_{N_h \times N_h} & \mathbf{B}_x &= [\mathbf{B}_{xjk}]_{N_h \times m_h} = \left[- \int_{\Omega} \frac{\partial \varphi_i}{\partial x} \psi_k \right]_{N_h \times m_h} \\ \mathbf{B}_y &= [\mathbf{B}_{yjk}]_{N_h \times m_h} = \left[- \int_{\Omega} \frac{\partial \varphi_i}{\partial y} \psi_k \right]_{N_h \times m_h} & \mathbf{b}_i &= \left[\int_{\Omega} \mathbf{g}_{i,j}^{n+1} \varphi_i \right]_{N_h \times 1} \end{aligned}$$

We can rewrite the above matricial equation in a more simple way

$$\begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{B}^t & 0 \end{bmatrix} \begin{bmatrix} \mathbf{u} \\ p \end{bmatrix} = \begin{bmatrix} \mathbf{b} \\ 0 \end{bmatrix} \text{ with } \mathbf{A} = \begin{bmatrix} \mathbf{A}_1 & \mathbf{A}_2 \\ \mathbf{A}_2^t & \mathbf{A}_1 \end{bmatrix}, \mathbf{B} = \begin{bmatrix} \mathbf{B}^x \\ \mathbf{B}^y \end{bmatrix}, \mathbf{u} = \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \text{ and } \mathbf{b} = \begin{bmatrix} b_1 \\ b_2 \end{bmatrix}$$

6. Numerical simulations

The solution can be evaluated using a direct method or iterative method applied to symmetric matrices as CG (conjugate gradient method). Details can be found in (Saad (2003) [13]). All meshes and simulations were done in FreeFem++ which is a free software with its own high level programming language based on the finite element method (FEM) to solve partial differential equations. An automatic mesh generator is used in FreeFem++ based on Delaunay-Voronoi algorithm where the number of internal points is proportional to the number of points on the boundaries.

The direct numerical simulation from the variational formulation for the time discretization can be straightforwardly implemented on the general finite element solver FreeFem++ which we use to make numerical experiments. The graphics were generated in FreeFem++ and Mathematica.

We develop the programming code in FreeFem++ from the variational problem and use Crout method and Conjugate Gradient method as solvers to solve the system. Towards the validation of the code, choosing solver and error analysis, we consider the Kim-Moin model problem with known exact solution given by

$$\mathbf{u}(t, \mathbf{x}) = (-\cos(2\pi x) \sin(2\pi y) e^{-8\pi^2 vt}, \sin(2\pi x) \cos(2\pi y) e^{-8\pi^2 vt}) \quad (18)$$

$$p(t, \mathbf{x}) = -\frac{1}{4}(\cos(4\pi x) + \cos(4\pi y)) e^{-16\pi^2 vt} \quad (19)$$

The velocity and pressure field remain in space and decrease monolithically with time. The Kim-Moin model problem is solved on the unit square $\Omega = [0.25, 1.25] \times [0.5, 1.5]$ and prescribes the exact velocity according to (18) and (19) along the boundary of the fluid domain. The calculations have been performed with a kinematic viscosity of $\nu = 0.01$ which results the null external force. The problem has been discretized in space with two meshes with 2048 and 8192 Hood-Taylor elements. For the finer mesh we have 4225 nodes P2 for the velocity and 1089 nodes P1 for the pressure and for the coarse mesh we have 16641 nodes P2 for the velocity and 4225 nodes P1 for the pressure. The time interval $[0, 1]$ was discretized into subintervals, of equal amplitude $\Delta t = 2^{-k}$ with $k = 5, 6, 7, 8$, of type $[t^n, t^{n+1}]$ ($n = 0, \dots, 2^k - 1$).

We use a linear direct and an iterative solver available in FreeFem++: Crout method is a direct method. It is a variant of the factorization method \mathbf{LU} where $\mathbf{U} = [u_{ij}]_{n \times n}$ is an upper triangular matrix with unitary diagonal and $\mathbf{L} = [l_{ij}]_{n \times n}$ matrix is an lower triangular, with its coefficients defined by

$$l_{ij} = a_{ij} - \sum_{k=1}^{j-1} l_{ik} u_{kj}, \quad j \leq i \quad u_{ij} = \frac{a_{ij} - \sum_{k=1}^{i-1} l_{ik} u_{kj}}{l_{ii}}, \quad j > i$$

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible
Newtonian Fluids using FreeFem++ based on Finite Element Method

Being \mathbf{A} a nonsingular matrix, as well as the matrix \mathbf{L} and \mathbf{U} , in this way the diagonal elements are not null. In this case, we can split the system $\mathbf{Ax} = \mathbf{b}$ into triangular systems of simpler resolution as follows

$$\mathbf{Ax} = \mathbf{b} \text{ or } \mathbf{LUx} = \mathbf{b} \text{ or } \begin{cases} \mathbf{LY} = \mathbf{b} \\ \mathbf{Ux} = \mathbf{y} \end{cases}$$

Conjugate Gradient method (CG) is an iterative method that applies to linear systems in which the matrix is symmetric positive definite. This method is normally used in large sparse matrices.

Taking the different meshes and using the two solvers (Crout and CG) in FreeFem++ we obtain the following numerical results:

First test case: In this test case, we take the mesh with 2048 elements and $\Delta t = 2^{-k}$ with $k = 5, 6, 7, 8$ and solve the problem with Crout method. The figure 1.1 shows the errors of the fluid velocity field and the pressure for each instant of time

evaluated in the L^2 -norm i.e., $ev_{2048}(t^n) = \|\mathbf{u} - \mathbf{u}_h\|_{L^2(\Omega)} = \left(\sum_{i=1}^2 \int_{\Omega} (u_i - u_{h,i})^2 \right)^{1/2}$ and

$$ep_{2048}(t^n) = \|p - p_h\|_{L^2(\Omega)} = \left(\int_{\Omega} (p - p_h)^2 \right)^{1/2}.$$

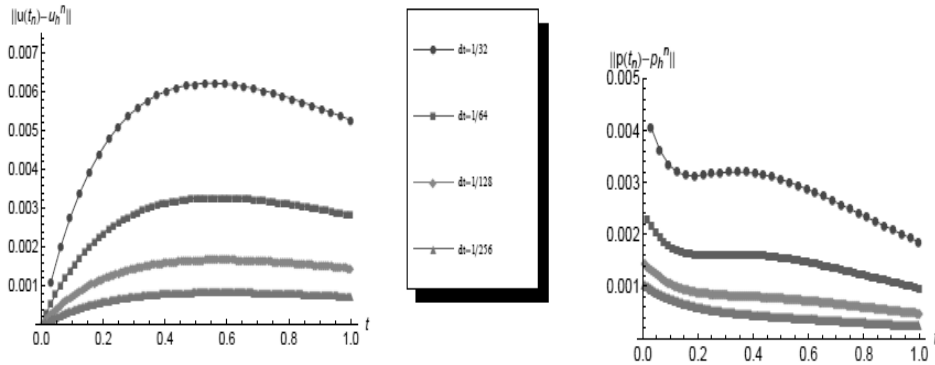


Figure 1.1: Error on the fluid velocity field (on the left) and the pressure (on the right) in L^2 -norm for each instant of time, for different Δt using a mesh with 2048 elements.

Second test case: In this test case, we take the mesh with 8192 elements and $\Delta t = 2^{-k}$ with $k = 5, 6, 7, 8$ and solve the problem with Crout method. The figure 1.2 shows the $ev_{8192}(t^n)$ and $ep_{8192}(t^n)$, the errors of the fluid velocity field and the pressure respectively for each instant of time evaluated in the L^2 -norm.

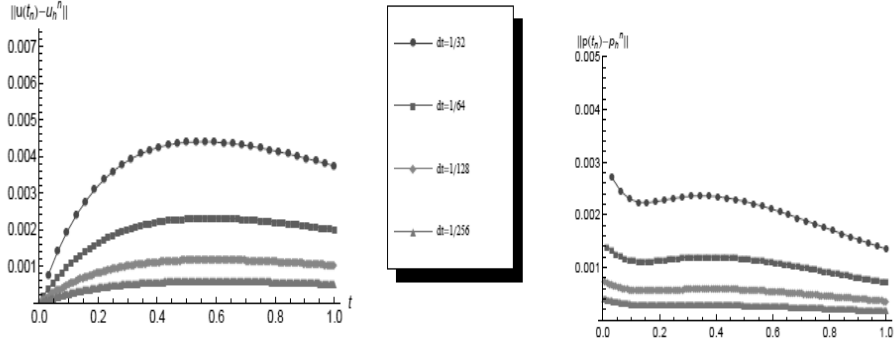


Figure 1.2: Error on the fluid velocity field (on the left) and the pressure (on the right) in L^2 -norm for each instant of time, for different Δt using a mesh with 8192 elements.

Comparison of both tests: Comparing the solutions computed by the two meshes for different time steps Δt , we notice that there is no remarkable difference between both solutions, although the error L^2 is decreasing which can be observed from the figure 1.1 and figure 1.2, which we can easily conclude from the figure 1.3. The figure 1.3 shows the error $ev_{2048}(1)$ and $ev_{8192}(1)$ of the fluid velocity field (on the left) and the error $ep_{2048}(1)$ and $ep_{8192}(1)$ of the pressure (on the right) for the instant of time $t = 1$ evaluated in the L^2 -norm.

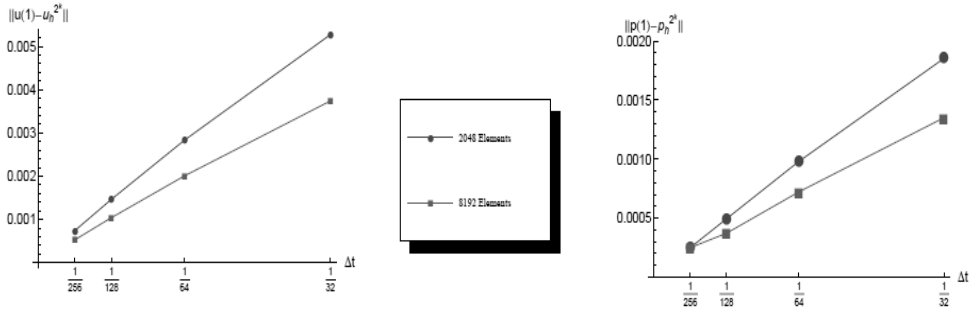


Figure 1.3: Comparison of errors of the fluid velocity field (on the left) and the pressure (on the right) in L^2 -norm for each instant of time $t = 1$, for different Δt using the both meshes.

Given the behavior of the error we can speculate that when $\Delta t \rightarrow 0$ we have $e_{2048}(1) \approx e_{8192}(1)$. Comparing the CPU time for each test, clearly the fine mesh demands for a large CPU time, for different time steps (figure 1.4). From the point of view of CPU time, it is better to employ a mesh little refined and a smaller Δt to achieve the same accuracy level.

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible Newtonian Fluids using FreeFem++ based on Finite Element Method

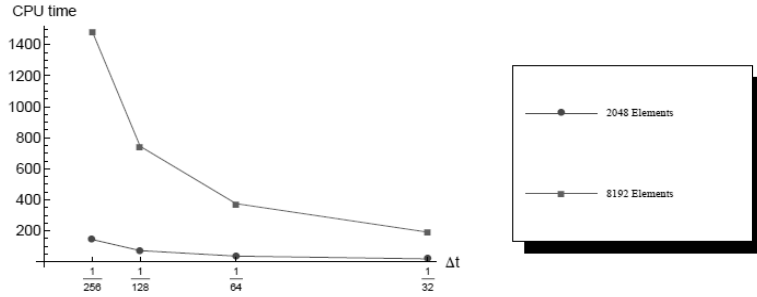


Figure 1.4: Comparison of CPU of times used as a function of Δt in solving the problem by the method of *Crout* for the two meshes.

Third test case: In this test case, we take the mesh with 2048 elements and $\Delta t = 2^{-k}$ with $k = 5, 6, 7, 8$ and solve the problem with CG method. We compare the results with the first test. The figure 1.5 shows the comparison of errors of the fluid velocity field and the pressure respectively for each instant of time evaluated in the L^2 -norm, for the both methods *Crout* and CG. It is evident that they present the same behavior and the same precision. The difference of error values between the *Crout* method and CG method are of the order 10^{-7} .

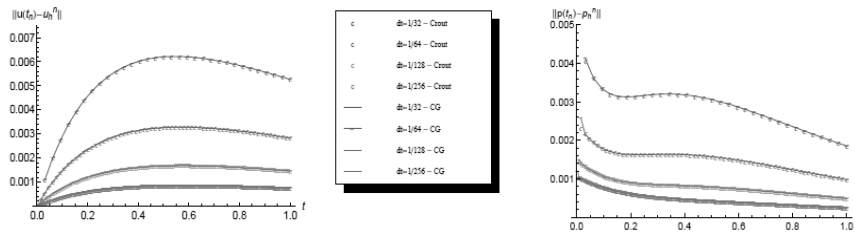


Figure 1.5: Comparison of errors of the fluid velocity field (on the left) and the pressure (on the right) in L^2 -norm for each instant of time, for different Δt using a mesh with 2048 elements, for the both methods: *Crout* and CG.

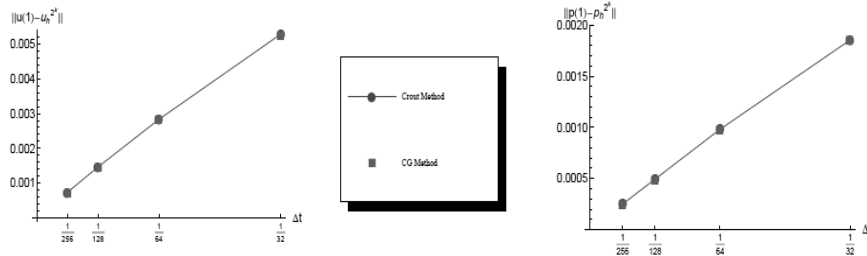


Figure 1.6: Comparison of errors of the fluid velocity field (on the left) and the pressure (on the right) in L^2 -norm for each instant of time $t = 1$, for different Δt using the both methods.

As it was expected, by a-priori estimates for the error in time, we obtain a linear convergence in order to time (figure 1.6).

Fourth test case: In this test case, we take the mesh with 8192 elements and $\Delta t = 2^{-k}$ with $k = 5, 6, 7, 8$ and solve the problem with CG method. We compare the results with the third test. We also notice for this method that there is no remarkable difference between both solutions, although the error L^2 decrease. The figure 1.7 shows the comparison of errors of the fluid velocity field and the pressure respectively for each instant of time evaluated in the L^2 -norm, for the both methods Crout and CG. It is evident that they present the same behavior and the same precision. The difference of error values between the Crout method and CG method are of the order 10^{-7} .

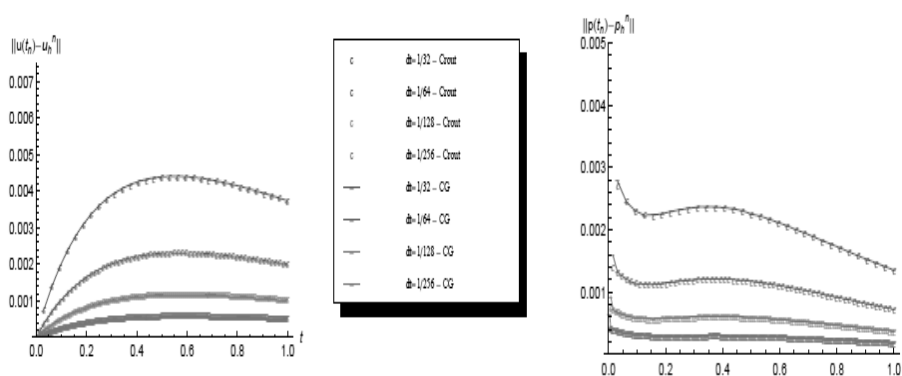


Figure 1.7: Comparison of errors of the fluid velocity field (on the left) and the pressure (on the right) in L^2 -norm for each instant of time, for different Δt using a mesh with 8192 elements, for the both methods: Crout and CG.

Since the both methods have the same accuracy, we have to decide which method to use based on CPU time. As we can see (figure 1.8), the method CG expends much CPU time than the method Crout, in the case of mesh to be refined. In the case of less fine mesh, the time spent by the two methods are approximately equal, although in this case the Crout method is slightly faster (figure 1.9).

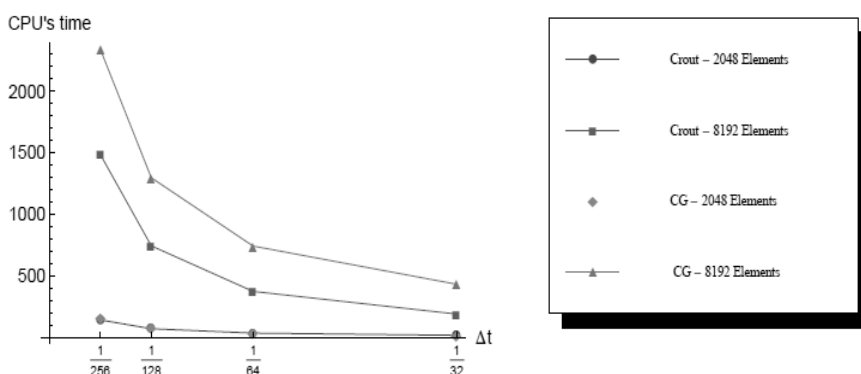


Figure 1.8: Comparison of CPU of times used as a function of Δt , in solving the problem with the two meshes.

Numerical Simulations of Unsteady Navier-Stokes Equations for Incompressible Newtonian Fluids using FreeFem++ based on Finite Element Method

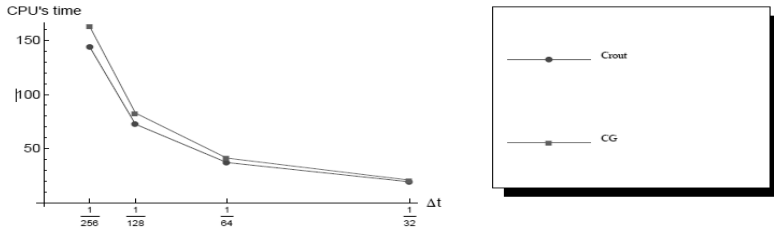


Figure 1.9: Comparison of CPU of times used as a function of Δt , in solving the problem by the method of Crout for the two meshes.

In large sparse matrix, the Conjugate Gradient method should be more efficient than the method of Crout, but we don't know if the implementation of this solver is optimized. Given the previous study we choose to use the method of Crout.

The following figures illustrate the exact solution and corresponding approximation obtained for a mesh with 2048 elements and $\Delta t = 1/128$ at time $t = 1$.

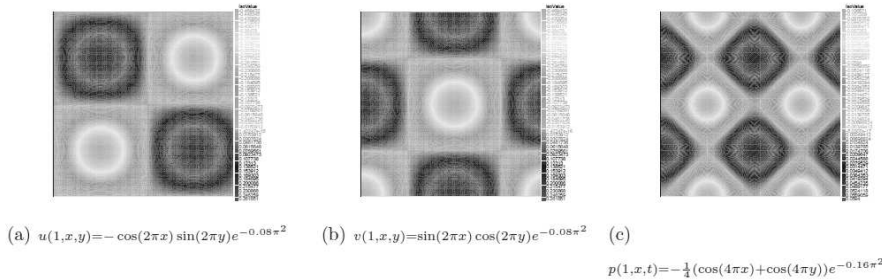


Figure 1.10: Exact solution at $t = 1$. First component of velocity (on the left), second component of velocity (on the centre) and pressure (on the right).

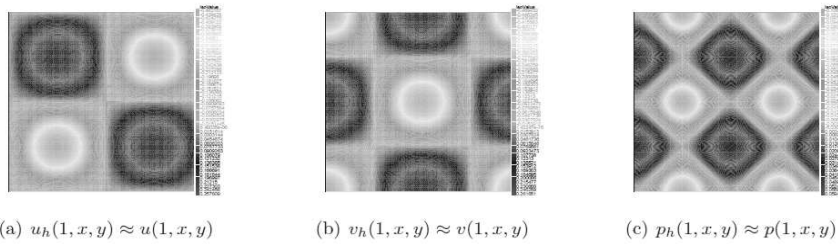


Figure 1.11: Approach solution at $t = 1$. First component of velocity (on the left), second component of velocity (on the centre) and pressure (on the right).

7. Discussion and conclusions

We have presented in this paper the Glerkin finite element method to simulate the motion of fluid particles which satisfies the unsteady Navier-Stokes equations. Time discretization is obtained using Characteristic Galerkin method. We use FreeFem++ to implement the simulations straightforward from the variational formulation. The velocity and pressure, i.e. the solution of Navier-Stokes equation is obtained. Comparing different test cases using different meshes and solvers we conclude that the Crout solver in FreeFem++ is more effective. From figure 1.10 and figure 1.11, we observe that the exact solution and approach solution is approximately same. The approximation of the velocity

M. M. Rhaman and K. M. Helal

and pressure are P_2 continuous and P_1 continuous finite element respectively. All the simulations were done in two dimensional case. We can extend it to the three-dimensional case using 3-D Navier-Stokes solver in FreeFem++ for further work. Furthermore, discontinuous Galerkin finite element method can be used in case of unstructured mesh and simulations can be implemented with FreeFem++.

REFERENCES

1. R.A.Adams and J.F.Fournier, *Sobolev Spaces*, 2ed, Academic Press, NY, 2003.
2. E.Becker, G.Carey and J.Oden, *Finite Elements. An Introduction Vol. 1*, Prentice Hall, Inc., Englewood Cliffs, New Jersey, 1981.
3. S.Brenner and L.R.Scott, *The Mathematical Theory of Finite Element Methods*, Springer-Verlag, New York, 1994.
4. H.Brezis, *Functional Analysis, Sobolev Spaces and PDE*, Springer, 2011.
5. G.F.Carey and J.T.Oden, *Finite elements, Vol.VI. Fluid mechanics*, The Texas Finite Element Series, VI. Prentice Hall, Inc., Englewood Cliffs, New Jersey, 1986.
6. A. J. Chorin and J. E. Marsden, *A Mathematical Introduction to Fluid Mechanics*, 3 Ed., Springer, 2000.
7. G.P.Galdi, An Introduction to the Mathematical Theory of the Navier-Stokes Equations: Nonlinear Steady Problems, *Springer Tracts in Natural Philosophy*, Vol. 38, Springer, New York, 1994.
8. V.Girault and P.A.Raviart, *Finite Element Approximation of the Navier-Stokes Equations, Computational Mathematics*, Springer-Verlag, Berlin, 1986.
9. V.Girault, and P.A.Raviart, *Finite Element Approximation of the Navier-Stokes Equations*, Lecture Notes in Math., 749, Springer-Verlag, Berlin, 1979.
10. F. Hecht, 'FreeFem++', 3rd Edition, Version 3.17, <http://www.freefem.org/ff++>
11. P.Lesaint, and P.A.Raviart, On a finite element method for solving the neutron transport equation, in C. de Boor, editor, *Mathematical Aspects of Finite Elements in Partial Differential Equations*, 89-123, New York, Academic Press, 1974.
12. A.Quarneroni and A.Valli, *Numerical Approximation of Partial Differential Equations*, Springer-Verlag, 1994.
13. Y.Saad, *Iterative Methods for Sparse Linear Systems*, 2ed, Society for Industrial and Applied Mathematics, Philadelphia, 2003.
14. E.J.Shaughnessy, I.M.Katz and J.P.Schaffer, *Introduction to Fluid Mechanics*, Oxford University Press, 2005.
15. R.Temam, *Navier-Stokes Equations, Theory and Numerical Analysis*, 3ed, North-Holland, Amsterdam, 1984.
16. M.Pal, *Numerical Analysis for Scientists and Engineers: Theory and C Program*, Narosa, New Delhi and Alpha Science, Oxford, United Kingdom, 2007.
17. M.A.Rahman, M.A.Alim and Md. Jahurul Islam, thermophoresis effect on MHD forced convection on a fluid over a continuous linear stretching sheet in presence of heat generation and power-law wall temperature, *Annals of Pure and Appl. Math.*, 4(2) (2013) 192-204.
18. N.N.Anika and Md.M.Hoque, Thermal buoyancy force effects on developed flow considering hall and ion-slip current, *Annals Pure and Appl. Math.*, 3(2) (2013) 179-188.