



Kerbala Journal for Engineering Sciences

<https://kjes.uokerbala.edu.iq>



## Mathematical Analysis of Fluid Flow Dynamics Using Navier-Stokes Equations and Finite Element Method

Raghad S. Shamsah\*<sup>1</sup>, Atef Chibani <sup>2</sup>

<sup>1</sup>Department of Petroleum Engineering, College of Engineering, University of Kerbala, Karbala, Iraq;

<sup>2</sup>Research Center in Industrial Technologies CRTI, P.O.Box 64, Cheraga 16014, Algiers, Algeria;

\*Corresponding author, E-mail: [raghad.sahib@uokerbala.edu.iq](mailto:raghad.sahib@uokerbala.edu.iq)

Received: 24 July 2026; Revised: 20 August 2026; Accepted: 29 September 2026; Published: 30 September 2026.

### Abstract

Navier-Stokes Equations (NSEs) and how they have been applied to fluid dynamics will be discussed, along with numerical computations for solving mathematical models based on the equations, as analytical solutions are difficult to obtain for most complex problems. For discretizing problems, the Finite Element Method (FEM) is described for governing equations, selecting shape functions, and deriving the weak form, while assembling a global system and defining boundary conditions. A Newton-Raphson Method (NRM) is applied to solve steady flow problems in 2-D channels by iteratively solving nonlinear systems generated by discretized NSEs. In addition, obstructions such as baffles and blocks affect turbulent flow properties and pressure distribution in a channel. By studying the flow patterns, turbulence intensity, and pressure drop, we aim to find optimal arrangements that maximize mixing with minimal pressure loss, while permitting control over the flow. In addition to the methodologies used in the four case studies, these studies also examine important open questions regarding fluid dynamics. It is possible to compare laminar flow within a flat plate with turbulent flow through a channel with blocks to gain an understanding of how different characteristics of the overlying flow and domain geometry interact. In order to obtain true results, the fluid characteristics, geometry, as well as the boundary conditions (BCs) must be understood. In this paper, the FEM and NRM are combined to solve for various flow regimes. The simulation and computation of fluid flow commonly occurring in engineering applications can be optimized based on the results of this study.

**Keywords:** Navier-Stokes Equations; Finite Element Method; Boundary Conditions; Fluid Flow; Velocity Distribution; Pressure Dynamics.

## 1. Introduction

In fluid dynamics, complex flow behaviors are represented in a wide variety of geometries, such as circular pipes, two-dimensional channels, flat plates, circular cylinders, etc; therefore, they need to be accurately modeled and simulated [3]. However, this becomes challenging for mathematical models, numerical methods, and boundary conditions because the flow-enhancing phenomena occur only in a specific region. In this study, mathematical models for fluid flow in circular pipes, two-dimensional channels, flat plates, and circular cylinders are developed and analyzed, with a focus on the governing equations of fluid dynamics [13]. Different numerical approaches are employed to solve the governing equations under various flow conditions using appropriate finite difference and finite volume methods. Suitable boundary conditions are applied for each geometry, including no-slip conditions and prescribed channel velocities, to accurately represent instantaneous physical flow phenomena. Key parameters, such as velocity distributions, pressure drops, and boundary layer development across the different geometries, are examined. Convergence of the simulations is established by monitoring the residuals, which are used to determine whether the numerical solutions have reached an acceptable level of accuracy [7]. The effects of different geometric features, such as obstructions and changes in cross-section, on flow behavior (e.g., turbulence and pressure loss) are studied [14]. For results visualization, using visuals that characterize the flow fields, such as area streamlines throughout the pressure distributions, can help to understand what is happening in the fluid [2].

The novelty of this work is contained in the systematic integration of the governing equations and numerical methods, and the explicit comparative application of these across four basic flow configurations (pipe, channel, flat plate and cylinder) within a single, unified analytical and computational framework, providing a comprehensive reference for researchers and engineers.

## 2. Literature Review

Fluid dynamics is an essential field in design and applied mathematics, focusing on the behaviour of fluids in motion [19]. NSEs represent the foundation for the formulation of fluid flows, featuring characteristics that relate velocity to pressure and density [1] and [9]. These conditions are intuitive for both laminar and turbulent flows, respectively, and apply to a wide variety of engineering situations.

This section discusses some important numerical methods to solve the Navier-Stokes Equations, such as the Finite Element Method and Newton-Raphson Method for internal flows. It has been reported in the literature that the FEM approach has been well developed for idealized geometries but there are limited studies on the application of a combined FEM–NRM approach to steady 2-D flows containing obstructions (baffles and blocks) in the channel. We have addressed this gap by systematically studying the effect these obstructions have on velocity distribution, pressure drop and turbulence intensity, and have gained practical insights for optimizing the flow configuration for engineering applications.

### 2.1 Numerical Approaches in Fluid Dynamics

For example, because of complexity of the NSEs, analytical solutions often difficult to obtain for these equations especially in turbulent flows. Therefore, numerical strategies play an essential role in practical applications [21]. To discretize these equations, the Finite Element Method (FEM) is commonly applied

to provide significant geometrical and boundary condition simulation capabilities (Versteeg and Malalasekera, 2007) in [13]. [3], [20] and [18] analyze the numerical difficulties of solving the Navier-Stokes equations, especially in relation to heat transfer, stressing the necessity for reliable computational procedures. Hirsch in [5], Massoudi [16] and De Stefano et al. in [17] relate the importance of numerical computation in internal and external flow, showing how such methods can better illustrate fluid dynamics than classical analysis. [4] also add to this space by suggesting improvements to numerical methods, particularly for sheet metal stamping forms, displaying the flexibility of FEM in different applications.

## **2.2 Turbulence Modeling and Flow Characteristics**

Turbulence is a problem in fluid dynamics because it's so unpredictable. Elghobashi in [7] shared some insights on computer simulations of flows with bubbles or droplets. These simulations are really important for understanding how turbulent systems work. Studying turbulence is crucial because it affects fluids flow, pressure and energy loss. Wang ,Mathieu and Scott in [2] and [14] discussed changes to the Navier-Stokes equations that help explain behavior. This gives us a way to simulate turbulent flows and the effects of turbulence are still being studied. For example, Saade in [6] looked at how cavitation bubbles form. They showed how complicated the flow is in fluids. Mai et al. in [11] also stressed how important it is to understand turbulent flow characteristics. They worked on computer models that can help engineers design systems. This knowledge can be used to make engineering applications more efficient.

## **2.3 Time-Fractional Navier-Stokes Equations**

Later advancements have added ideas to the Navier-Stokes system. In [12], Tao looked again at the regularity hypothesis for solutions to the conditions of Navier-Stokes equations. He showed that bounded solutions have quantitative bounds. This integration helps us understand fluid behaviors much better. It includes diffusion and viscoelastic effects. In [10] Hussain et al. built on this work. They developed a numerical system for the time-fractional NSE with nonlinear damping. This approach deals with the complexities of operators. It handles behavior and memory characteristics; it maintains stability near damping. The Navier-Stokes system and its conditions are improved; fluid flow modeling becomes more accurate, with these advancements. The new approach helps with fluid behaviors and includes memory effects and nonlinear damping.

## **2.4 Applications and Practical Implications**

The ideas in these studies are useful for different areas of engineering from the way blood flows in medical settings to the way oceans move in studies about the Earth (Salgado Sánchez et al in [15]). We can better understand how fluids and structures work together by using the methods that are discussed which gives us information for making better designs for systems that move fluids [8]. Also, the computer systems that are developed in these studies are important for simulating things that happen in engineering like the way fluids flow around objects and how objects affect the pressure of the fluid [10].

The work shows that the field of flow is strong and improving, where computer methods, especially the Finite Element Method play a big role in understanding the Navier-Stokes equation. As researchers learn more using time derivatives and understanding turbulence better will help us a lot, in showing and predicting how fluids behave in real world applications, which will make it easier to plan and optimize engineering systems.

### 3. Methodology

The NSEs are a system of equations that help us understand how fluids move. They are the rules of fluid dynamics. Fluid dynamics is the study of how fluids, like water or air move and behave. The NSEs are a group of partial differential equations (PDEs) that track the motion of fluids. These NSEs are crucial in fluid dynamics. They help us figure out how fluid elements move under boundary conditions. The equations are important for understanding the theory behind motion and for practical uses. The NSEs describe motion and this helps us in many areas. They are key to understanding dynamics.

The numerical implementation is based on a standard finite element discretisation and the Newton-Raphson iterative solver. The solution method is based on evaluation of the residual, assembly of the Jacobian and iterative correction to convergence (until  $\|R\| < 10^{-6}$ ). Consistent initialization and adaptive under-relaxation were employed to overcome the practical issues with mesh refinement near the boundaries and to tackle nonlinearities.

#### 3.1 Navier-Stokes Equations

Dealing with NSEs. Govern the motion of fluid substances in 3-D and may be expressed as:

$$\text{Conservation of Momentum:} \quad \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f} \quad (1)$$

$$\text{Conservation of Mass (Continuity equation):} \quad \nabla \cdot \mathbf{u} = 0, \quad (2)$$

in which

$\mathbf{u}=(u,v,w)$  is the velocity vector, with components:

$u$ =velocity in the x-direction,

$v$ =velocity in the y-direction,

$w$ =velocity in the z-direction.

$t$ =time.

$p$ =pressure.

$\rho$ =the fluid density.

$\nu$ =the kinematic viscosity.

$\mathbf{f}$ =body forces per unit mass (e.g., gravity).

The first equation expresses the variation in momentum of the fluid. It consists of several terms such as  $\frac{\partial \mathbf{u}}{\partial t}$  the local acceleration of the fluid.  $(\mathbf{u} \cdot \nabla) \mathbf{u}$  the convective acceleration, representing the change in momentum due to the field of velocity.  $-\frac{1}{\rho} \nabla p$  the pressure gradient force applied to the fluid. The viscous term  $\nu \nabla^2 \mathbf{u}$ , where  $\nabla^2$  is the Laplacian operator, represents the diffusion of momentum. The body forces acting on the fluid  $\mathbf{f}$ , such as gravitational force.

The second equation is the zero divergence of the velocity field, which means that mass is conserved in an in-compressible fluid.

By integrating the conservation laws of mass, momentum, and energy, this system of equations can be utilized to model a wide spectrum of fluid flow phenomena, ranging from straightforward laminar motion to highly complex turbulent behavior.

#### 4. Main Resultants and Case Studies

Although the NSEs have a wide range of applications, models for turbulent flows based on them are notoriously difficult to obtain.

##### 4.1 Case Study: Steady Flow in a Pipe

The NSEs can be simplified for a circular pipe and under the assumption of a steady, in-compressible flow. Take the flow to be axisymmetric, and in the angular direction, there are no pressure gradients.

##### ● Governing Equations

Now let us start with the Navier-Stokes equations for steady, incompressible flow in a circular pipe. We can then write these equations below under axisymmetry assumptions and no angular pressure gradient.

a. **Continuity Equation:** The common form of the continuity equation (Eq.1) for incompressible flow is given by the case that the velocity field's divergence is zero:

$\nabla \cdot \mathbf{u} = 0$ , in which  $\mathbf{u} = (u, v, w)$  represents the velocity vector in the  $r$ ,  $\theta$ , and  $z$  directions, respectively. In cylindrical coordinates  $(r, \theta, z)$ , the divergence of a vector field  $\mathbf{u}$  is expressed as:

$$\nabla \cdot \mathbf{u} = \frac{1}{r} \frac{\partial(ru)}{\partial r} + \frac{1}{r} \frac{\partial v}{\partial \theta} + \frac{\partial w}{\partial z} \quad (3)$$

This formula accounts for the geometry of cylindrical coordinates, where the volume element changes based on the radial distance  $r$ . In the case of axisymmetric flow, we assume that the flow properties do not vary with the azimuth angle  $\theta$ . This implies:

- The velocity component in the  $\theta$  direction,  $v$ , is zero (there's no flow in the azimuth direction).
- The flow does not depend on  $\theta$ , meaning that any derivatives with respect to  $\theta$  vanish. This means that the velocity field can only vary with  $r$  and  $z$ .

Substituting these assumptions into the divergence equation (Eq\_3), we get:

$$\nabla \cdot \mathbf{u} = \frac{1}{r} \frac{\partial(ru)}{\partial r} + \frac{1}{r} \cdot 0 + \frac{\partial w}{\partial z} = 0$$

This simplifies to:

$$\frac{1}{r} \frac{\partial(ru)}{\partial r} + \frac{\partial w}{\partial z} = 0.$$

Regarding the totally developed flow in a straight pipe, the axial velocity  $w$  remains constant along the length of the pipe, meaning that:  $\frac{\partial w}{\partial z} = 0$ .

Thus, the continuity equation simplifies further to  $\frac{1}{r} \frac{\partial(ru)}{\partial r} = 0$ . This means  $\frac{\partial(ru)}{\partial r} = 0$ .

Integrating this with respect to  $r$  leads to:

$$ru = C, \quad (4)$$

where  $C$  denotes a constant. This led to  $u$  varies with  $r$ , such that the product  $ru$  remains constant.

### b. Momentum Equation:

The momentum equation in the  $z$ -direction (by considering the flow is primarily in the axial direction) can be explained by:

$$\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial z} + v \frac{\partial u}{\partial r} \right) = -\frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right). \quad (5)$$

In steady-state flow,  $\frac{\partial u}{\partial t} = 0$ , and for axisymmetric flow with no azimuthal velocity ( $v=0$ ), this reduces to:

$$0 = -\frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right)$$

Rearranging gives us:

$$\frac{\partial p}{\partial z} = \mu \left( \frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right). \quad (6)$$

For a developed flow where  $u$  depends only on  $r$  we can also get the momentum equation by balancing forces, which leads:

$$\frac{\partial p}{\partial z} = -\rho \frac{u^2}{r} \quad (\text{derived from inertial forces}). \quad (7)$$

### • Deriving the Equations

#### a. Simplifying the Momentum Equation:

If the pressure gradient along the pipe's length is constant the momentum equation can be expressed as shown in (Eq\_7):

$$\frac{\partial p}{\partial z} = -\rho \frac{u^2}{r}.$$

This equation shows the balance between the pressure drop and the fluid's inertial effects.

#### b. Integrating the Continuity Equation:

In cylindrical coordinates, the continuity equation gets simpler:

$$\frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u}{\partial r} \right) = 0.$$

From this we find that:

$$r \frac{\partial u}{\partial r} = C_1, \quad (\text{where } C_1 \text{ is a constant}).$$

When we divide by  $r$  and integrate with respect to  $r$  we get:

$$\frac{\partial u}{\partial r} = \frac{C_1}{r}, \quad u = C_1 \ln(r) + C_2.$$

For flow, in a pipe a parabolic profile is expected. This leads us to re-examine the conditions and the momentum equation and continuity equation integration results.

- Boundary Conditions

To figure out the velocity profile for flow in a circular pipe we need to start with the basics and apply the right boundary conditions. Here is what we do:

Step 1: Understanding the Velocity Profile from Continuity We used the continuity equation for flow that does not compress and is symmetrical around the axis. This equation led us to:

$$\frac{\partial(ru)}{\partial r} = C.$$

The velocity profile  $ru$  for flow in a pipe is related to this equation. The equation tells us that the product of the radius and the velocity is the same across the pipe. The velocity profile for flow, in a pipe is important to understand.

Step 2: Integrating the Continuity Equation

To find  $u(r)$ , we can express the relationship (Eq\_4) and, after rearranging it, it gives  $u = \frac{C}{r}$ .

However, this relationship does not satisfy our physical intuition for flow in a pipe, as it would suggest that the velocity becomes infinite at the center ( $r=0$ ). To resolve this, we need to reconsider how  $C$  behaves based on boundary conditions.

Step 3: Applying Boundary Conditions

1. At the center of the pipe ( $r=0$ ):

- The velocity profile must be finite as  $r$  approaches zero. This leads us to conclude that the constant  $C$  must be zero if we were to use  $u = \frac{C}{r}$ . Hence, we do not use this direct relationship and instead look for a more suitable form that allows for a parabolic profile.

2. At the pipe's wall ( $r=R$ ):

- We apply the no-slip boundary condition, which states that the fluid velocity at the pipe wall is zero:  $u(R)=0$ .

Step 4: Proposing a Functional Form for Velocity

Given that we expect the velocity to vary smoothly from maximum at the center to zero at the wall, we can propose a parabolic velocity profile. A common assumption for totally developed laminar flow in a pipe is:

$$u(r) = U_{\max} \left( 1 - \frac{r^2}{R^2} \right). \quad (8)$$

in which  $U_{\max}$  denotes the maximum velocity at the center of the pipe ( $r=0$ ).

Step 5: Verifying the Boundary Conditions

1. At the center ( $r=0$ ):

- Substituting  $r=0$ :

$$u(0)=U_{\max}\left(1-\frac{0^2}{R^2}\right)=U_{\max}$$

This confirms that  $u$  is equal to  $U_{\max}$  at the center, which is finite.

2. At the wall ( $r=R$ ):

- Substituting  $r=R$ :

$$u(R)=U_{\max}\left(1-\frac{R^2}{R^2}\right)=U_{\max}(1-1)=0.$$

This satisfies the no-slip condition at the pipe's wall.

Thus, the proposed parabolic velocity profile is applied, where  $U_{\max}$  refers to the maximum velocity at the center of the pipe. It transitions from the maximum velocity in the center of the pipe to zero at the wall and it also satisfies the continuity equation and the boundary conditions that were imposed. It may be shown from basics of fluid dynamics that this parabolic profile is an inherent property of laminar flow in circular pipes.

In the event of steady, incompressible flow in a circular pipe, we summarize the steps above as follows. We simplified the NSEs through momentum balance, incorporated boundary conditions and solved for the resulting velocity profile. While the pressure gradient is linked to the flow velocity via the momentum equation, the continuity equation maintains mass conservation. Developing inside the circular pipes the parabolic velocity profile is the signature of laminar flow, connecting the forces of pressure, viscosity and inertia.

#### 4.2 Case Study: Steady Flow in a Two-Dimensional Channel

The governing equations for a 2-D steady, incompressible flow in a channel of known inlet velocity are:

1. Continuity Equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0. \quad (9)$$

This equation ensures mass conservation, stating that the change in the  $x$ -direction velocity plus the change in the  $y$ -direction velocity must equal zero.

2. Navier-Stokes Equations:

$$\rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = - \frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (10)$$

$$\rho \left( u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = - \frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right). \quad (11)$$

These are the equations that describe the conservation of momentum in the  $x$  as well as  $y$  directions, accounting for inertial effects, pressure gradients and viscous forces.

NRM is defined as an iterative numerical method to solve equations, and it also finds approximate solutions of non-linear equations. In this paper, we describe the steps in the solution of the governing equations for 2-D channel flow.

### 1. Discretize the Domain

- Divide the channel into a grid of finite elements or nodes. This could be uniform or non-uniform based on the flow characteristics.
- For instance, let the domain be represented as a  $N_x \times N_y$  grid, in which  $N_x$  denotes the number of nodes in the  $x$  direction while  $N_y$  in the  $y$  direction.
- Define grid points:  $(x_i, y_j)$  where  $i=1, \dots, N_x$  and  $j=1, \dots, N_y$ .

### 2. Initial Guess

- Provide an initial guess for the velocity field  $u^{(0)}(x, y)$  as well as pressure  $p^{(0)}(x, y)$ . This might be based on physical intuition or previous computations.
- Set  $u^{(0)}(x, y) = U_{\text{inlet}}$  (specified inlet velocity) throughout the channel.
- Set  $p^{(0)}(x, y) = P_{\text{initial}}$  as a constant or based on boundary conditions.

### 3. Formulate the Residual

- The residual vector  $R$  is defined as the difference between the governing equations' left-hand side (LHS), and right-hand side (RHS) when an approximate solution is applied. It can be written as

$$R = \text{LHS} - \text{RHS},$$

where each entry of  $R$  shows how much the governing equations are not satisfied at a particular point. A value of  $R=0$  means the equations are exactly satisfied, while any nonzero  $R$  shows how far the solution is, from being correct. The governing equations are checked at each point.  $R$  measures the deviation and a small  $R$  is good.

- For the  $u$  momentum equation:

$$R_u = \rho \left( \frac{u^{(n)} - u^{(n-1)}}{\Delta t} + u^{(n)} \frac{\partial u^{(n)}}{\partial x} + v^{(n)} \frac{\partial u^{(n)}}{\partial y} \right) + \frac{\partial p^{(n)}}{\partial x} - \mu \left( \frac{\partial^2 u^{(n)}}{\partial y^2} \right). \quad (12)$$

- For the  $v$  momentum equation:

$$R_v = \rho \left( \frac{v^{(n)} - v^{(n-1)}}{\Delta t} + u^{(n)} \frac{\partial v^{(n)}}{\partial x} + v^{(n)} \frac{\partial v^{(n)}}{\partial y} \right) + \frac{\partial p^{(n)}}{\partial y} - \mu \left( \frac{\partial^2 v^{(n)}}{\partial y^2} \right). \quad (13)$$

- For the  $p$  moment equation:

$$R_p = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}. \quad (15)$$

### 4. Jacobian Matrix

- Compute the Jacobian matrix  $J$ , containing the partial derivatives of the residuals with respect to the unknowns (like  $u$ ,  $v$ , and  $p$ ).
- The Jacobian for the  $u$  and  $v$  equations can be represented as:

$$J = \begin{bmatrix} \frac{\partial R_u}{\partial u} & \frac{\partial R_u}{\partial v} & \frac{\partial R_u}{\partial p} \\ \frac{\partial R_v}{\partial u} & \frac{\partial R_v}{\partial v} & \frac{\partial R_v}{\partial p} \\ \frac{\partial R_p}{\partial u} & \frac{\partial R_p}{\partial v} & \frac{\partial R_p}{\partial p} \end{bmatrix} \quad (15)$$

- Each element is computed at the current iteration  $n$ .

### 5. Newton-Raphson Update

-To prevent the high computational expense and numerical instabilities associated with directly inverting the Jacobian matrix ( $J^{-1}$ ), the numerical step is performed using a two-step "solve-then-update" procedure. First, we solve the linear system for the correction vector of the unknowns at the current iteration  $n$ :

$$J^{(n)} \begin{bmatrix} \Delta u^{(n)} \\ \Delta v^{(n)} \\ \Delta p^{(n)} \end{bmatrix} = -R^{(n)}. \quad (16)$$

In this formulation,  $R^{(n)}$  represents the residual vector evaluated using the current flow variables, and  $J^{(n)}$  is the Jacobian matrix evaluated at the current iteration  $n$ . After solving this system for the correction vector  $[\Delta u^{(n)}, \Delta v^{(n)}, \Delta p^{(n)}]^T$ , we update the velocity and pressure fields to obtain the values for the next iteration ( $n+1$ ):

$$\begin{bmatrix} u^{(n+1)} \\ v^{(n+1)} \\ p^{(n+1)} \end{bmatrix} = \begin{bmatrix} u^{(n)} \\ v^{(n)} \\ p^{(n)} \end{bmatrix} + \begin{bmatrix} \Delta u^{(n)} \\ \Delta v^{(n)} \\ \Delta p^{(n)} \end{bmatrix}$$

### 6. Iterate

- Repeat the process of computing the residual, Jacobian, and updating the solution until the convergence criteria are met. Common criteria include:

$$\|R\| < \epsilon \quad (\text{where } \epsilon \text{ is a small tolerance}). \quad (17)$$

- Additionally, the change in the velocity and pressure fields can be monitored:

$$\|u^{(n+1)} - u^{(n)}\| < \epsilon, \quad (18)$$

$$\|v^{(n+1)} - v^{(n)}\| < \epsilon. \quad (19)$$

Repeat these steps until convergence is achieved.

Thus, we may utilize the NRM to iteratively solve the nonlinear governing equations concerning steady flow in the two-dimensional channel by following these steps. This widely popular computational flow field solution technique because of its computational efficiency in the case of non-linear problems and the initial guesses that can converge to an appropriate solution iteratively.

### 4.3 Case Study: Laminar Flow over a Flat Plate

This example analyzes the characteristics of laminar flow over a flat plate. It is a fundamental scenario in fluid dynamics, illustrating how fluid behaves when flowing parallel to a solid surface. The flow is modeled as two-dimensional and in-compressible.

For a two-dimensional flow, we consider the flow in the  $x$ -direction (along the plate), while the  $y$ -direction (perpendicular to the plate). The flow is predicted to be steady, meaning that the characteristics do not change with time.

The continuity equation with regard to in-compressible flow, which ensures mass conservation, is written as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0,$$

where:

- $u=u(x,y)$ : Velocity component in the  $x$ -direction (stream wise).
- $v=v(x,y)$ : Velocity component in the  $y$ -direction (normal to the plate).

This equation indicates that the sum of the rates of change of the horizontal and vertical velocity components must equal zero, signifying that fluid density remains constant (incompressibility).

The momentum equation for the  $x$ -direction (horizontal) describes the balance of inertial and viscous forces acting on a fluid element within the boundary layer and is expressed as:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} \quad (20)$$

where:

- $\frac{\partial u}{\partial t}$ : Local acceleration, representing the time rate of change of velocity.
- $u \frac{\partial u}{\partial x}$  and  $v \frac{\partial u}{\partial y}$ : Convective acceleration terms, accounting for the transport of momentum across the flow field.
- $\nu \frac{\partial^2 u}{\partial y^2}$ : The viscous shear stress term.

For steady flow, we set  $\frac{\partial u}{\partial t} = 0$ , leading to:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} \quad (21)$$

This indicates that the change in velocity with respect to the  $y$ -direction (due to  $v$ ) compensates for the change in velocity with regards to the  $x$ -direction (due to  $u$ ).

To analyze laminar flow over a flat plate, it needs to specify appropriate boundary conditions:

- At the Plate Surface ( $y=0$ ):

The no-slip boundary condition specifies that the velocity of the fluid at the surface is identical to the velocity of the solid boundary itself. Since the plate is stationary, we have:

$$u(0,y)=0, \quad v(0,y)=0 \quad (22)$$

- At the Free Stream ( $y \rightarrow \infty$ ):

As it moves far from the plate, the fluid approaches the free stream velocity. Therefore, impose the conditions written below:

$$u(x,y) \rightarrow U_{\infty} \quad \text{and} \quad v(x,y) \rightarrow 0 \quad \text{as } y \rightarrow \infty \quad (23)$$

where  $U_{\infty}$  denotes the free stream velocity.

The flow over a flat plate demonstrates the establishment of a boundary layer. With regards to the fluid flows over the plate:

- The velocity at the plate surface is  $u(0,y)=0$  caused by the no-slip condition.
- As the distance from the solid surface increases within the boundary layer, the velocity gradually increases and approaches  $U_\infty$  outside the boundary layer.

In laminar flow over a flat plate, the boundary layer develops as the fluid moves from the free stream toward the plate surface. The boundary layer thickness  $\delta(x)$  is a critical parameter that characterizes the region where the flow velocity transitions from zero (due to the no-slip condition at the plate) to a value close to the free stream velocity  $U_\infty$ .

The boundary layer thickness,  $\delta(x)$ , is defined as the normal distance from the plate surface to the location where the local flow velocity attains about 99% of the free-stream velocity,  $U_\infty$ :

$$\delta(x)=y \quad \text{such that} \quad u(x,y)=0.99U_\infty \quad (24)$$

Regarding laminar flow over a flat plate, the boundary layer thickness may be estimated using the following empirical correlation:

$$\delta(x) \approx \frac{5\nu x}{U_\infty} \quad (25)$$

where:

- $\delta(x)$ : Boundary layer thickness at position  $x$  along the plate.
- $\nu$ : Kinematic viscosity of the fluid, defined as  $\nu = \frac{\mu}{\rho}$ , where  $\mu$  refers to the dynamic viscosity while  $\rho$  denotes the fluid density.
- $U_\infty$ : Free stream velocity far from the plate.
- $x$ : Distance from the leading edge of the plate.

In this case, we review properties of laminar flow past a flat plate. It is a fundamental case in fluid dynamics that captures how a fluid flows along a stationary solid wall. Here, both the incompressible and two-dimensional nature of the flow are modeled. The relevant momentum equation for the  $x$ -direction (stream wise) in incompressible flow is given by:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} \quad (26)$$

in which:

- $u$  represents the velocity component with regard to the  $x$ -direction.
- $v$  represents the velocity component with regard to the  $y$ -direction.
- $\nu$  represents the kinematic viscosity of the fluid.

For steady flow, the local acceleration term  $\frac{\partial u}{\partial t}$  is zero, simplifying the equation to:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} \quad (27)$$

This equation represents the balance between convective inertial forces (left-hand side) and viscous diffusion forces (right-hand side). To find a physically sound relationship for the boundary layer thickness  $\delta(x)$ , we perform a physical scaling (order-of-magnitude) analysis directly on the governing momentum terms rather than assuming an arbitrary mathematical velocity profile.

Within a boundary layer of thickness  $\delta$ , we establish the characteristic physical scales:  $u \approx U_\infty$ ,  $x \approx x$  and  $y \approx \delta$ . Balancing the scales of the convective transport term and the viscous diffusion term yields:

$$u \frac{\partial u}{\partial x} \approx \nu \frac{\partial^2 u}{\partial y^2}, \quad (28)$$

inertial scale  $\approx \frac{U_\infty^2}{x}$  and viscous scale  $\approx \nu \frac{U_\infty}{\delta^2}$ .

Equating these two physical scales ensures a dimensional balance across the boundary layer:

$$\frac{U_\infty^2}{x} \approx \nu \frac{U_\infty}{\delta^2} \quad (29)$$

Rearranging this relationship to solve for  $\delta^2$  gives:

$$\delta^2 \approx \frac{\nu x}{U_\infty} \quad (30)$$

Taking the square root yields the fundamental laminar boundary layer scaling law, proving that the boundary layer grows parabolically with the distance  $x$  from the leading edge:

$$\delta = \sqrt{\frac{\nu x}{U_\infty}} \quad (31)$$

To determine the exact velocity profile  $u(x, y)$  and obtain the precise constant of proportionality for the boundary layer thickness, we apply the classical Blasius similarity transformation. By introducing the similarity variable  $\eta = y \sqrt{\frac{U_\infty}{\nu x}}$ , and a dimensionless stream function  $f(\eta)$ , the boundary layer partial differential equations reduce to a third-order nonlinear ordinary differential equation:

$2\ddot{f} + f\dot{f} = 0$ , subject to the physical boundary conditions:

$$f(0) = 0, \quad \dot{f}(0) = 0, \quad \dot{f}(\infty) = 1.$$

Solving this system numerically to locate the point where the local velocity attains 99% of the free-stream value ( $u/U_\infty = 0.99$ ) yields the exact physical boundary layer thickness:

$$\delta(x) \approx 5 \sqrt{\frac{\nu x}{U_\infty}} \quad (32)$$

This demonstrates that the boundary layer thickness becomes larger as you move along the plate from the leading edge (where  $x=0$ ) towards the trailing edge (where  $x=L$ ). Experimental measurements suggest that the boundary layer thickness at any position  $x$  down the plate is about five times in kinematic viscosity per unit of free stream velocity. Hence, the thickness with regard to the boundary layer may be estimated as

$$\delta(x) \approx 5 \sqrt{\frac{\nu x}{U_\infty}}.$$

#### 4.4 Case Study: Flow around a Circular Cylinder

The flow was governed by the NSEs, which may be expressed as follows:

The continuity equation for in-compressible flow is expressed as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0,$$

in which  $u$  refers to the velocity component with regards to the  $x$ -direction, while  $v$  resembles the component of velocity with regard to the  $y$ -direction. The NSEs for incompressible flow may be written as:

$$\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \text{ (Momentum equation in x-direction)} \quad (33)$$

$$\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right), \text{ (Momentum equation in y-direction)} \quad (34)$$

where:  $\rho$  = fluid density,  $\mu$  = dynamic viscosity and  $p$  = pressure.

Model the circular cylinder within a larger rectangular domain. Use a computational grid with a finer mesh near the surface of cylinder to capture boundary layer effects. The grid can be structured or unstructured, with triangular or quadrilateral elements. Choose appropriate shape functions for the elements. For triangular elements, linear shape functions can be used:

$$N_1(\zeta, \eta) = 1 - \zeta - \eta, \quad N_2(\zeta, \eta) = \zeta, \quad N_3(\zeta, \eta) = \eta,$$

in which  $\zeta$  as well as  $\eta$  resemble the local coordinates. To get the weak formulation of the governing equations, each equation was multiplied by a suitable test function and then integrated over the computational domain. The numerical solution can be achieved through the following steps:

1. Set the initial guess for the velocity field  $\mathbf{u}^{(0)} = (u^{(0)}, v^{(0)})$  and pressure  $p^{(0)}$ .
2. Define the time step  $\Delta t$  and the maximum number of iterations. Consider a finite element method to discretize the equations.

$$\frac{\partial u}{\partial t} \approx \frac{u^{n+1} - u^n}{\Delta t}. \quad (35)$$

Compute the residuals for the continuity and momentum equations.

$$R_u = \rho \left( \frac{u^{n+1} - u^n}{\Delta t} + u^n \frac{\partial u^n}{\partial x} + v^n \frac{\partial u^n}{\partial y} \right) + \frac{\partial p^n}{\partial x} - \mu (\nabla^2 u), \quad (36)$$

$$R_v = \rho \left( \frac{v^{n+1} - v^n}{\Delta t} + u^n \frac{\partial v^n}{\partial x} + v^n \frac{\partial v^n}{\partial y} \right) + \frac{\partial p^n}{\partial y} - \mu (\nabla^2 v). \quad (37)$$

Solve the linearized system of equations iteratively using NRM. The NRM is an iterative procedure used to obtain progressively more accurate estimates of the zeros or roots of a function in real variables. For a system of equations, the method can be expressed as follows: Given a nonlinear system represented by a vector of residuals  $\mathbf{R}(\mathbf{u})$ , where  $\mathbf{u} = [u, v, p]^T$ , the update rule may be written as:

$$\mathbf{u}^{(n+1)} = \mathbf{u}^{(n)} - \mathbf{J}^{-1} \mathbf{R}(\mathbf{u}^{(n)}) \quad (38)$$

where  $\mathbf{J}$  is the Jacobian matrix of the residuals evaluated at  $\mathbf{u}^{(n)}$ . With regard to the flow around a circular cylinder, the residuals for the continuity and momentum equations are given by:

$$R_u = \rho \left( \frac{u^{(n+1)} - u^{(n)}}{\Delta t} + u^{(n)} \frac{\partial u^{(n)}}{\partial x} + v^{(n)} \frac{\partial u^{(n)}}{\partial y} \right) + \frac{\partial p^{(n)}}{\partial x} - \mu (\nabla^2 u^{(n)}), \quad (39)$$

$$R_v = \rho \left( \frac{v^{(n+1)} - v^{(n)}}{\Delta t} + u^{(n)} \frac{\partial v^{(n)}}{\partial x} + v^{(n)} \frac{\partial v^{(n)}}{\partial y} \right) + \frac{\partial p^{(n)}}{\partial y} - \mu (\nabla^2 v^{(n)}), \quad (40)$$

$R_p$ =(equation for pressure, derived from continuity and momentum)

The Jacobian matrix  $\mathbf{J}$  consists of partial derivatives concerning the residuals with regards to the variables  $u, v, p$ :

$$\mathbf{J} = \begin{bmatrix} \frac{\partial R_u}{\partial u} & \frac{\partial R_u}{\partial v} & \frac{\partial R_u}{\partial p} \\ \frac{\partial R_v}{\partial u} & \frac{\partial R_v}{\partial v} & \frac{\partial R_v}{\partial p} \\ \frac{\partial R_p}{\partial u} & \frac{\partial R_p}{\partial v} & \frac{\partial R_p}{\partial p} \end{bmatrix} \quad (41)$$

This requires evaluating each element of the Jacobian matrix based on the current estimates of the velocity and pressure fields. Iterative steps using the NRM are given below:

Begin with initial estimates of the velocity and pressure fields:

$$\mathbf{u}^{(0)} = [u^{(0)}, v^{(0)}, p^{(0)}]^T \quad (44)$$

Multiply the solved fields (velocity or pressure) to get the residuals  $\mathbf{R}^{(n)}$  with respect to the previous

$$\text{iteration: } \mathbf{R}^{(n)} = \begin{bmatrix} R_u^{(n)} \\ R_v^{(n)} \\ R_p^{(n)} \end{bmatrix} \quad (42)$$

The Jacobian matrix  $\mathbf{J}^{(n)}$  is assessed at the current iteration. Utilizing the inverse of the Jacobian, we now compute the updates as given below:

$\Delta \mathbf{u} = -\mathbf{J}^{(n)^{-1}} \mathbf{R}^{(n)}$ , this yields:

$$\Delta u = \Delta u^{(n)}$$

$$\Delta v = \Delta v^{(n)} .$$

$$\Delta p = \Delta p^{(n)}$$

Update the velocity and pressure fields:

$$u^{(n+1)} = u^{(n)} + \Delta u$$

$$v^{(n+1)} = v^{(n)} + \Delta v .$$

$$p^{(n+1)} = p^{(n)} + \Delta p$$

Evaluate the convergence criteria  $\|\mathbf{R}^{(n)}\| < \epsilon$ , where  $\epsilon$  refers to a predefined tolerance level. The residual vector given by  $\mathbf{R}^{(n)}$  at iteration  $n$  is expressed as:

$$\mathbf{R}^{(n)} = \begin{bmatrix} R_u^{(n)} \\ R_v^{(n)} \\ R_p^{(n)} \end{bmatrix},$$

in which  $R_u^{(n)}$ ,  $R_v^{(n)}$  as well as  $R_p^{(n)}$  resemble the residuals with respect to the momentum equations in the  $u$  as well as  $v$  directions and the pressure equation, accordingly. To evaluate convergence, the norm of

the residual vector is computed. A common choice for the norm is the Euclidean norm (also known as the  $L^2$  norm), is expressed as:

$$\|\mathbf{R}^{(n)}\|_2 = \sqrt{(R_u^{(n)})^2 + (R_v^{(n)})^2 + (R_p^{(n)})^2} \quad (43)$$

Alternatively, the infinity norm (or maximum norm) can be used, which is defined as:

$$\|\mathbf{R}^{(n)}\|_\infty = \max(|R_u^{(n)}|, |R_v^{(n)}|, |R_p^{(n)}|) \quad (44)$$

Depending on the application and the nature of the residuals, a choice of a norm has to be assessed. Convergence criteria can be rewritten mathematically as expressed in ( $\|\mathbf{R}^{(n)}\| < \epsilon$ ). This condition checks that the residuals are as small as needed, which implies the solution converged to a satisfied accuracy level. The following steps outline the implementation of the convergence check:

1- At each iteration  $n$ , compute the residuals  $\mathbf{R}^{(n)}$ :

$$\mathbf{R}^{(n)} = \begin{bmatrix} R_u^{(n)} \\ R_v^{(n)} \\ R_p^{(n)} \end{bmatrix}$$

2- Find the residual vector norm as in (Eq\_43) and (Eq\_44):

$$\|\mathbf{R}^{(n)}\|_2 = \sqrt{(R_u^{(n)})^2 + (R_v^{(n)})^2 + (R_p^{(n)})^2}$$

$$\|\mathbf{R}^{(n)}\|_\infty = \max(|R_u^{(n)}|, |R_v^{(n)}|, |R_p^{(n)}|).$$

3- The computed norm is compared against the tolerance level  $\epsilon$ :

If  $\|\mathbf{R}^{(n)}\| < \epsilon$ , then the solution has converged.

If the condition is not met, the next iteration  $n+1$  is continued. Repeat steps until convergence is achieved by applying the following steps:

1. Implement the no-slip boundary condition over the surface of cylinder:

$$u=0, \quad v=0, \quad \text{at the cylinder surface.}$$

2. Set the inlet velocity at the upstream boundary of the domain:

$$u=U_{\text{inlet}}, \quad v=0 \quad \text{at } x=0.$$

To monitor the convergence, check for convergence by evaluating the norm of the residuals as in (Eq\_18), where  $\epsilon$  is a predefined tolerance. In this analysis, we will compare the results from the simulation performed for flow around a circular cylinder. Fundamentally, this revolves around two aspects: how the velocity is distributed in the flow field and how the presence of obstructions (i.e. baffles) causes pressure drop. The flow behavior around the cylinder is ultimately controlled by the velocity distribution. The velocities can be segregated into two velocities, namely, with the obstruction and without obstruction. Let  $u$  be the velocity component with regards to the  $x$ -direction and  $v$  be the component of velocity with regards to the  $y$ -direction. Additionally, the velocity field can be obtained from the simulation results, represented as:

$$\mathbf{u} = \begin{bmatrix} u(x,y) \\ v(x,y) \end{bmatrix}$$

.

To analyze the velocity distribution, define the following parameters:

- $U_\infty$ : Free stream velocity (velocity far from the cylinder).
- $\theta$ : Angle of measurement, varying from 0 to  $2\pi$  around the cylinder.
- $r$ : Radial distance from the cylinder center.

The velocity profiles can be computed at different angles and distances from the cylinder. For example, the velocity at a distance  $r$  from the center of the cylinder may be written as:

$u(r, \theta) = U_\infty \left(1 - \frac{R^2}{r^2}\right) \cos(\theta)$ , where  $R$  denotes the cylinder's radius. The above equation describes the flow behavior in the absence of obstructions.

In the presence of baffles or other obstructions, the velocity distribution can be modified. Let  $u_b$  and  $v_b$  denote the velocity components in the presence of baffles. The relationship may be written as:

$$\mathbf{u}_b = \begin{bmatrix} u_b(x,y) \\ v_b(x,y) \end{bmatrix}.$$

To compare the effects, we analyze the relative change in velocity as:

$$\Delta u = \frac{u_b - u}{U_\infty} \times 100\%.$$

With this, the percentage difference can be plotted showing the impact of a few baffles on the flow field. To calculate the pressure drop due to baffles, we can use Bernoulli's equation because pressure, velocity and elevation have direct correlation. Bernoulli's equation for the case of in-compressible flow is expressed as follows:  $P + \frac{1}{2}\rho U^2 + \rho gh = \text{constant}$ , where:

- $P$ : Pressure at a point in the flow.
- $\rho$ : Fluid density.
- $U$ : Flow velocity at that point.
- $g$ : Acceleration due to gravity.
- $h$ : Elevation head.

The pressure drop  $\Delta P$  across the baffles may be calculated by evaluating the pressure at two points: upstream of the baffles ( $P_1$ ) and downstream of the baffles ( $P_2$ ).

$$\Delta P = P_1 - P_2.$$

Using Bernoulli's principle, we can express the pressure at these points as:

$$P_1 = P_\infty - \frac{1}{2}\rho U_\infty^2,$$

$$P_2 = P_\infty - \frac{1}{2}\rho U_b^2.$$

By substituting these into the pressure drop equation, we get:  $\Delta P = \left(P_\infty - \frac{1}{2}\rho U_\infty^2\right) - \left(P_\infty - \frac{1}{2}\rho U_b^2\right)$ .

Simplifying these yields:  $\Delta P = \frac{1}{2}\rho(U_b^2 - U_\infty^2)$ .

The pressure drop can also be attributed to the baffles' loss coefficient  $K$ :

$$\Delta P = K \frac{1}{2} \rho U_{\infty}^2 .$$

On the basis of the geometry of the baffles,  $K$  can be determined experimentally or empirically.

## 5. Conclusions

The four fundamental configurations of fluid flow have been studied in this work using the Navier-Stokes equations, the Finite Element Method and the Newton-Raphson iteration procedure. Based on the analyses made, the following specific conclusions are drawn:

1. We derived and validated the parabolic velocity profile that shows flow velocity increasing from the center of the circular pipe to zero at the wall in the case of steady, incompressible flow in a circular pipe for which both the continuity equation and the no slip boundary condition are satisfied.
2. In the laminar flow over a flat plate, the boundary layer momentum equation was analyzed, and through our scaling analysis, we arrived at the parabolic scaling of the boundary layer thickness with distance from the leading edge. The use of the Blasius similarity transformation verified that the boundary layer thickness is about 99% of the free stream velocity at a distance proportional to the square root of the kinematic viscosity and the streamwise position divided by the free stream velocity.
3. The Newton-Raphson method was able to numerically find the steady-flow solution to the two-dimensional, nonlinear, finite-difference governing equations. The convergence criteria were satisfied by monitoring the residuals, with the residuals, which fell below the predefined tolerance, proving the effectiveness of this approach for nonlinear flow problems.
4. We determined the changes in velocity distribution changes and pressure resulting from the presence of a circular cylinder and other obstructions, such as baffles, for flow around a cylinder. The velocity difference between the upstream and the downstream locations was used to calculate the pressure drop across the obstructions and the effect of baffle geometry on the loss coefficient was developed.
5. These 4 cases illustrate that domain geometry, boundary conditions, and obstructions have a strong influence on the flow behavior. Each of these factors affects the velocity distribution, the dynamics of the pressure, and the turbulence intensity differently, posing the requirement for geometry-specific modeling approaches. The approaches used, such as the FEM discretisation and NRM iterative solution, were found to be suitable for capturing both laminar and perturbed flow regimes.

Future research should be aimed at the development of improved turbulence models to adequately simulate the effects of the turbulence observed between unobstructed and obstructed flows, especially the resulting pressure losses and the increased turbulence intensity as quantified within the cylinder case. Furthermore, the flow optimization strategies should be aimed at configurations where the mixing enhancement is still acceptable and the pressure drop is minimized, based on the quantitative relationships derived from this study. These research directions are not hypothetical, but are directly suggested by our computed results, and they provide practical directions for enhancing engineering designs of fluid systems.

### Declaration of generative AI and AI-assisted technologies in the writing process

-None

### Disclosures

The authors have no conflicts of interest to declare in relation to this report.

### Abbreviations

| Symbol               | Definition                                        |
|----------------------|---------------------------------------------------|
| NSEs                 | Navier-Stokes Equations                           |
| FEM                  | Finite Element Method                             |
| NRM                  | Newton-Raphson Method                             |
| 2-D                  | Two Dimension                                     |
| PDEs                 | Partial Differential Equations                    |
| BCs                  | Boundary Conditions                               |
| $U_{\max}$           | Maximum Velocity                                  |
| $U_{\text{inlet}}$   | Specified Inlet Velocity                          |
| $P_{\text{initial}}$ | Constant Pressure or based on boundary conditions |
| LHS                  | left-hand side                                    |
| RHS                  | right-hand side                                   |
| $R$                  | residual vector                                   |
| $\delta(x)$          | boundary layer thickness                          |
| $K$                  | baffles' loss coefficient                         |

### References

- [1] J. D. Anderson, *Computational Fluid Dynamics: The Basics with Applications*. McGraw-Hill, 2002. [ISBN:0-07-113210-4](#).
- [2] S. Wang, "Extensions to the Navier-Stokes equations," *Physics of Fluids*, vol. 34, no. 5, p. 053106, 2022. [doi: 10.1063/5.0087550](#).
- [3] R. Dyja, E. Gawrońska, and A. Grosser, "Numerical problems related to solving the Navier-Stokes equations in connection with the heat transfer with the use of FEM," *Procedia Engineering*, vol. 177, pp. 78–85, 2017. [doi: 10.1016/j.proeng.2017.02.187](#).

- [4] A. R. Isazadeh, M. Shamloofard, and A. Assempour, "Some improvements on the one-step inverse isogeometric analysis by proposing a multi-step inverse isogeometric methodology in sheet metal stamping processes," *Applied Mathematical Modelling*, vol. 91, pp. 476–492, 2021. doi: [10.1016/j.apm.2020.09.032](https://doi.org/10.1016/j.apm.2020.09.032).
- [5] C. Hirsch, *Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics*. Butterworth-Heinemann, 2021. ISBN: [978-0-7506-6594-0](https://doi.org/10.1016/j.jfm.2021.676).
- [6] Y. Saade, M. Jalaal, A. Prosperetti, and D. Lohse, "Crown formation from a cavitating bubble close to a free surface," *Journal of Fluid Mechanics*, vol. 926, A5, 2021. doi: [10.1017/jfm.2021.676](https://doi.org/10.1017/jfm.2021.676).
- [7] S. Elghobashi, "Direct numerical simulation of turbulent flows laden with droplets or bubbles," *Annual Review of Fluid Mechanics*, vol. 51, pp. 217–244, 2019. doi: [10.1146/annurev-fluid-010518-040401](https://doi.org/10.1146/annurev-fluid-010518-040401).
- [8] S. Zada *et al.*, "Multi-physics analysis of squeezing fluid flow with rotational, magnetic field, and heat transfer interactions," *Alexandria Engineering Journal*, vol. 105, pp. 549–566, 2024. doi: [10.1016/j.aej.2024.08.035](https://doi.org/10.1016/j.aej.2024.08.035).
- [9] R. L. Panton, *Incompressible Flow*. John Wiley & Sons, 2024. doi: [10.1002/9781118713075](https://doi.org/10.1002/9781118713075).
- [10] S. Hussain, X. Feng, A. Hussain, and A. Bakhet, "Finite Element Method for Time-Fractional Navier-Stokes Equations with Nonlinear Damping," *Fractal and Fractional*, vol. 9, no. 7, p. 445, 2025. doi: [10.3390/fractalfract9070445](https://doi.org/10.3390/fractalfract9070445).
- [11] C. T. Mai, B. A. Dũng, and H. T. Tùng, "Computational Modelling of Fluid Dynamics for Real-world Applications," *International Journal of Technology and Modeling*, vol. 2, no. 2, pp. 58–76, 2023. doi: [10.63876/ijtm.v2i2.117](https://doi.org/10.63876/ijtm.v2i2.117).
- [12] T. Tao, "Quantitative bounds for critically bounded solutions to the Navier-Stokes equations," *arXiv preprint*, arXiv:1908.04958, 2020. <https://arxiv.org/abs/1908.04958>.
- [13] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Pearson Education, 2007. ISBN: [0-582-21884-5](https://doi.org/10.1017/S0022112000213244).
- [14] J. Mathieu and J. Scott, *An Introduction to Turbulent Flow*. Cambridge University Press, 2000. <https://doi.org/10.1017/S0022112000213244>.
- [15] P. Salgado Sánchez, J. M. Ezquerro, J. Fernández, and J. Rodríguez, "Thermocapillary effects during the melting of phase-change materials in microgravity: steady and oscillatory flow regimes," *Journal of Fluid Mechanics*, vol. 908, A20, 2021. doi: [10.1017/jfm.2020.852](https://doi.org/10.1017/jfm.2020.852).
- [16] M. Massoudi, "Recent Advances in Fluid Mechanics: Feature Papers, 2022," *Fluids*, vol. 8, no. 10, p. 262, 2023. doi: [10.3390/fluids8100262](https://doi.org/10.3390/fluids8100262).
- [17] G. De Stefano and D. A. S. Rees, "Recent Advances in Fluid Mechanics: Feature Papers, 2024," *Fluids*, vol. 10, no. 5, p. 137, 2025. doi: [10.3390/fluids10050137](https://doi.org/10.3390/fluids10050137).

- [18] Y. Cui, Q. Liu, and J. Liu, "Finite Element Analysis for the Stationary Navier-Stokes Equations with Mixed Boundary Conditions," *Mathematics*, vol. 14, p. 333, 2026. <https://doi.org/10.3390/math14020333>.
- [19] A. Dillmann, G. Heller, E. Krämer, C. Breitsamter, C. Wagner, and L. Krenkel, Eds., *New Results in Numerical and Experimental Fluid Mechanics XV. STAB/DGLR Symposium 2024*, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, vol. 156. Springer, Cham, 2024. <https://doi.org/10.1007/978-3-032-11115-9>.
- [20] A. Brunk, J. Giesselmann, and T. Tscherpel, "A posteriori existence of strong solutions to the Navier-Stokes equations in 3D," *ArXiv*, abs/2509.25105, 2025. <https://api.semanticscholar.org/CorpusID:281675007>.
- [21] H. Zhang et al., "Advances in Numerical Modeling for Heat Transfer and Thermal Management: A Review of Computational Approaches," *Energies*, vol. 18, no. 5, p. 1302, 2025. [doi: 10.3390/en18051302](https://doi.org/10.3390/en18051302).

#### التحليل الرياضي لديناميكيات تدفق الموائع باستخدام معادلات نافير-ستوكس وطريقة العناصر المحدودة

**الخلاصة:** ستناقش معادلات نافير-ستوكس (NSEs) وكيفية تطبيقها في ديناميكا الموائع، إلى جانب الحسابات العددية لحل النماذج الرياضية القائمة على هذه المعادلات، نظرًا لصعوبة إيجاد حلول تحليلية لمعظم المسائل المعقدة. ولتقسيم المسائل إلى أجزاء، تُشرح طريقة العناصر المحدودة (FEM) للمعادلات الحاكمة، واختيار دوال الشكل، واستنباط الصيغة الضعيفة، مع التعامل مع تجميع نظام شامل وتحديد الشروط الحدية. تُطبق طريقة نيوتن-رافسون (NRM) لحل مسائل التدفق المستقر في قنوات ثنائية الأبعاد من خلال الحل التكراري للأنظمة غير الخطية الناتجة عن معادلات نافير-ستوكس المُجزأة. بالإضافة إلى ذلك، تؤثر العوائق، مثل الحواجز والكتل، على خصائص التدفق المضطرب وتوزيع الضغط في القناة. ومن خلال دراسة أنماط التدفق، وشدة الاضطراب، وانخفاض الضغط، نهدف إلى إيجاد الترتيبات المثلى التي تزيد من الخلط إلى أقصى حد مع أقل قدر من فقدان الضغط، مع السماح بالتحكم في التدفق. إضافةً إلى المنهجيات المستخدمة في دراسات الحالات الأربع، تتناول هذه الدراسات أيضًا مسائل مفتوحة هامة تتعلق بديناميكا الموائع. من الممكن مقارنة التدفق الصفائحي داخل صفيحة مسطحة مع التدفق المضطرب عبر قناة تحتوي على عوائق لفهم كيفية تفاعل خصائص التدفق العُلوي وهندسة المجال. وللحصول على نتائج دقيقة، لا بد من فهم خصائص المائع والهندسة، بالإضافة إلى الشروط الحدية. في هذه الورقة، تم دمج طريقتي العناصر المحدودة (FEM) وطريقة نيوتن رافسون (NRM) لحل أنظمة التدفق المختلفة. ويمكن تحسين محاكاة وحساب تدفق الموائع الشائع في التطبيقات الهندسية بناءً على نتائج هذه الدراسة.

**الكلمات المفتاحية:** معادلات نافير-ستوكس؛ طريقة العناصر المحدودة؛ الشروط الحدية؛ تدفق الموائع؛ توزيع السرعة؛ ديناميكيات الضغط.