CHAPTER 2

NUMERICAL MODELING AND ANALYSIS OF LAMINAR FLOW FOR VALIDATION OF THE NUMERICAL TOOL

2.1 Introduction to Mathematical Modeling

Any theoretical analysis in fluid dynamics involves mathematical representation for the physical principles. These principles are essentially the conservation laws for mass and momentum of the flowing fluid in terms of pressure and velocity fields. Following continuum approach, these governing equations emerge in the form of partial differential equations. The conservation equation for mass is referred to as continuity equation, and the conservation equation for momentum, for a Newtonian fluid, is called the Navier-Stokes equation. The nature of the momentum variable, expressed in terms of velocity, imparts non-linearity to the Navier-Stokes equation. This makes these equations along with the problem-specific initial and boundary conditions amenable to exact analysis only for a countable number of cases. This calls for the application of Computational Fluid Dynamics, or CFD in short.

CFD capitalizes the power of computer in two ways. The solution of a continuous physical domain is obtained in terms of the solution at a large number of discrete points, known as nodes or grid points. The process of posting the grid points in the solution domain is called the domain discretisation. In this way, the large memory size of the computer is utilized. The fast computing power of the computers is utilized in order to solve a large number of linear simultaneous equations. The process of transforming a governing partial differential equation to a set of linear simultaneous equations is known as discretisation of governing equation. With the rapid growth in computer technology, CFD has emerged as complementary to experimental techniques in fluid mechanics.

The development of the tools in CFD came in a big way in minimizing overall cost and time of development. Increasingly involved problems in fluid mechanics having possible coupling with other areas like heat and mass transfer, electro-dynamics, magneto-dynamics and chemical reaction are being chosen as candidates for CFD-based solution. Typical examples are turbulent flow and heat transfer, multi-phase and dispersed-phase flows, combustion and fluid-structure interaction. Solution of such problems is posing a challenge to the existing computing power. In order to make these problems amenable to present-day computing capability, a judicious sophistication in the representation of the governing equations is often employed. The physical configuration of the spool valve is described in Figure 1.2. For analyzing axi-symmetric flow through the valve, the cylindrical coordinate system provides the easiest means to represent both the governing equations and the boundary conditions.
Chapter 2

\[ \frac{\partial}{\partial x_i}(\rho u_i) = 0 \]  \hspace{1cm} (2.1)

\[ \text{In the above equations, } p \text{ stands for pressure, and the subscripts } i \text{ and } j \text{ represent three directions 1, 2 and 3. With the unit vectors denoted by } \hat{e}_i, \text{ the velocity } u_i \text{ and gradient in terms of the spatial coordinates } x_i \text{ can be written in generalized orthogonal curvilinear coordinate system as} \]
Chapter 2

\[ u_i = \hat{e}_i u_1 + \hat{e}_2 u_2 + \hat{e}_3 u_3, \quad (2.3) \]

and \[ \frac{\partial}{\partial x_i} = \hat{e}_1 \frac{\partial}{\partial h_1 x_1} + \hat{e}_2 \frac{\partial}{\partial h_2 x_2} + \hat{e}_3 \frac{\partial}{\partial h_3 x_3}, \quad (2.4) \]

where \( h_i \) is the scale factor in \( i \)th direction, the scale factor being the ratio of change actual length in a coordinate direction to the change in coordinate value. The use of repeated subscript in the symbolic writing of above equations denotes summation over all coordinate directions. The outermost derivative-index \( j \) in first and third term of in Equation 2.2 represents a simple summation for all values equal to 1, 2 and 3. However, either \( i \) in Equation 2.1 or \( j \) in viscosity related normal stress term Equation 2.2 corresponds to a dot product involving all coordinate values of 1, 2 and 3. Thus, the right hand side of in Equation 2.1 can be expanded as

\[ \left( \hat{e}_1 \frac{\partial}{\partial h_1 x_1} + \hat{e}_2 \frac{\partial}{\partial h_2 x_2} + \hat{e}_3 \frac{\partial}{\partial h_3 x_3} \right) \cdot (\hat{e}_1 \rho u_1 + \hat{e}_2 \rho u_2 + \hat{e}_3 \rho u_3). \quad (2.5) \]

It is apparent from the above that the differences in the expanded forms of Equations 2.1 and 2.2 in different coordinate systems are due to the forms of gradients of unit vectors in respective coordinate directions. In rectangular Cartesian coordinate system, of course, these gradients are zero, as pure translation of any vector of constant magnitude does not introduce any change to the vector.

For modeling flow that is steady and laminar, it is sufficient to consider Equations 2.1 and 2.2 along with the necessary boundary conditions. If the flow is incompressible, then the velocity boundary conditions need to be specified all along the domain boundary.

The inlet conditions are usually posed as with known velocity conditions.

For long flow passages, at the outlet the extrapolation boundary conditions are posed. In case of axial flow, the axial gradient of axial velocity is posed as zero. The companion normal velocity component is obtained as zero from the continuity equation. For radial flow, the radial gradient of product of radius vector and radial velocity is posed as zero. The companion normal velocity is yet again obtained as zero from the continuity equation.

On the solid wall, both the components of velocity are zero because of no-slip and impregnability conditions on the solid surface.

In case an axis of symmetry exists on the flow domain, then the velocity normal to the axis is considered to be zero. The normal gradient of the other component of velocity is set equal to zero.

For pressure, it is sufficient to prescribe its value at a convenient point in the flow field.
The governing equations along with the boundary conditions form the closure of the laminar modeling.

### 2.3 Governing Equations in Cylindrical and Rectangular Cartesian Coordinates

We consider a steady, laminar, incompressible, Newtonian and axi-symmetric flow in cylindrical coordinate system. Following the assumption of axi-symmetry of flow, use of cylindrical coordinate system \((r, \theta, z)\) is most convenient for analyzing the flow through spool valve. Rotation of unit vectors needs to be understood for working out the governing equations in cylindrical coordinate system. Non-zero gradients of unit vectors arise only due to rotation of radial and tangential unit vectors in tangential direction. This is illustrated in Figure 2.2.

![Figure 2.2: Derivatives of Unit Vectors in Cylindrical Coordinate System](image)

It is clear from Figure 2.2 that the derivatives of unit vectors undergoing rotation can be expressed as

\[
\frac{\partial \hat{e}_r}{\partial \theta} = \lim_{\delta \theta \to 0} \frac{\hat{e}_r|_{\theta + \delta \theta} - \hat{e}_r|_{\theta}}{\delta \theta} = \lim_{\delta \theta \to 0} \frac{\partial \hat{e}_r}{\partial \theta} = \lim_{\delta \theta \to 0} \frac{(1.\delta \theta)\hat{e}_\theta|_{\theta + \delta \theta}/2}{\delta \theta} = \hat{e}_\theta, \quad (2.6a)
\]

and

\[
\frac{\partial \hat{e}_\theta}{\partial \theta} = \lim_{\delta \theta \to 0} \frac{\hat{e}_\theta|_{\theta + \delta \theta} - \hat{e}_\theta|_{\theta}}{\delta \theta} = \lim_{\delta \theta \to 0} \frac{\partial \hat{e}_\theta}{\partial \theta} = \lim_{\delta \theta \to 0} \frac{(1.\delta \theta)(-\hat{e}_r|_{\theta + \delta \theta}/2)}{\delta \theta} = -\hat{e}_r. \quad (2.6b)
\]

For axi-symmetric flow, using the above two equations along with Equations 2.1 and 2.2 results in the following expressions.

\[
\frac{1}{r} \frac{\partial}{\partial r} (\rho u_r) + \frac{\partial}{\partial z} (\rho u_z) = 0 \quad (2.7a)
\]

\[
\rho \left( u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_z}{\partial z} \right) = \frac{\partial}{\partial r} \left[ p + \frac{2}{3} \mu \left( \frac{1}{r} \frac{\partial (\rho u_r)}{\partial r} + \frac{\partial (\rho u_z)}{\partial z} \right) \right]
\]
Using Equation 2.7a for incompressible flow in Equations 2.7b and 2.7c, we obtain the equation structure for the geometry indicated in Figure 2.1 (b) as

$$\begin{align*}
\frac{1}{r} \frac{\partial}{\partial r} (ru_r) + \frac{\partial}{\partial z} (u_z) &= 0 \\
\frac{1}{r} \frac{\partial (ru_r)}{\partial r} + \frac{\partial (u_z u_r)}{\partial z} &= -\frac{\partial p}{\partial r} + \frac{1}{\text{Re}} \left( \frac{\partial}{\partial r} \left( \frac{1}{r} \frac{\partial (ru_r)}{\partial r} \right) + \frac{\partial^2 u_r}{\partial z^2} \right) \\
\frac{1}{r} \frac{\partial (ru_r)}{\partial r} + \frac{\partial (u_z u_z)}{\partial z} &= -\frac{\partial p}{\partial r} + \frac{1}{\text{Re}} \left( \frac{1}{r} \frac{\partial u_z}{\partial r} \left( \frac{r}{\partial z} \right) + \frac{\partial^2 u_z}{\partial z^2} \right)
\end{align*}$$

In the similar manner, the equation structure for the geometry indicated in Figure 2.1 (a) can be obtained as

$$\begin{align*}
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\
\frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} &= -\frac{\partial p}{\partial x} + \frac{1}{\text{Re}} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \\
\frac{\partial (uv)}{\partial x} + \frac{\partial (vv)}{\partial y} &= -\frac{\partial p}{\partial y} + \frac{1}{\text{Re}} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right).
\end{align*}$$

Here length, velocity and difference of pressure, with respect to the inlet pressure, are non-dimensionalised with the inlet width or diameter, inlet velocity and dynamic pressure corresponding to inlet velocity. Obviously, in the above equations Reynolds number \( \text{Re} \) is defined in terms of the hydraulic diameter and velocity, both at inlet, and the kinematic viscosity of the fluid. In case of the validation problems of flow between parallel plates and through the circular duct, the length scales are the distance between the plates and the diameter respectively. In case of flow through the valve, the twice of axial width at the inlet has been taken as the length scale.

The boundary conditions for the validation problems have been indicated in Figure 2.1. On the walls, a zero velocity implies the no-slip and no penetration conditions together. The inlet velocity has been assumed as uniform. The assumed exit boundary condition is basically the fully developed flow condition. On the plane or axis of symmetry, the symmetry boundary conditions have been used.
Chapter 2

The governing equations and boundary conditions are solved following the SIMPLE or SIMPLER technique [Patankar (1980), Versteeg and Malalasekera (1995)]. The third order upwind difference scheme (TUDS) is often preferred over simple upwind difference scheme, since simple UDS induces numerical diffusion in the simulation that imparts stability at the cost of accuracy of the solution. Especially in turbulent flow simulation, the numerical diffusion could alter the solution behavior by a great deal. On this count, the TUDS behaves better. The discretization scheme in cylindrical coordinate system is described below.

2.4 Discretization

The governing equations are solved numerically using integral approach of the finite volume method on a staggered grid. The physical domain is subdivided into small volumes (or areas for 2-D case), and dependent variables are evaluated at the center of the volumes (cells). With respect to the scalar variables, the variables that are vector components are given a half control volume lag in the direction of the vector under consideration. In Figure 2.3, such a grid arrangement is schematically shown, representing scalar variables by filled dots and the vector components by directional arrows. With respect to the control volumes for pressure, the two control volume systems for the axial and radial velocities have staggering in the direction of the velocity components. These control volumes have been marked by bold dashed lines, dots, and thin dashed lines respectively. The finite volume formulation for the equations in cylindrical coordinates is described below. In Figures 2.3 to 2.5, the control volume is been shown only about the variable discretized in each case.

![Figure 2.3: Control Volume Discretization for Radial Velocity](image)

However, in the form of a key, the staggering of the variables are shown in each of these figures. In the bigger diagram, the center of the control volume location is marked as P. The one grid-length away neighboring grid points for the same variable in north, south, east and west are indicated by N, S, E and W respectively. The two grid-length away neighboring grid points for the same variable in north, south, east
Chapter 2

and west are indicated by the symbols NN, SS, EE and WW respectively. The locations indicated in small cases are for the other system variables, represented by the corresponding dot or arrow.

2.4.1 Discretization of r-Component of Momentum Equation

For the representative control volume (cell) shown in Figure 2.3, the finite volume \(dV\) is given by

\[
dV = 2\pi r_c \delta r \delta z,
\]

where \(\delta r = (r_e - r_w)\) \(\delta z = (z_n - z_s)\) \(\text{(2.10)}\)

and \(r_c\) is radial distance of the centre of the control volume.

Now, integrating the Equation 2.8b over the finite volume about cell center (P) gives

\[
\int_{cv} \int \int \left[ \frac{1}{r} \frac{\partial (ru_r)}{\partial r} + \frac{\partial (u_z u_r)}{\partial z} \right] dV
\]

\[
= \int_{cv} \int \int \left[ -\frac{\partial p}{\partial r} + \frac{1}{Re} \left( \frac{\partial}{r} \left( \frac{1}{r} \frac{\partial (ru_r)}{\partial r} \right) \right) + \frac{\partial^2 u_r}{\partial z^2} \right] dV \quad \text{(2.11)}
\]

Clearly, the above equation is non-linear. We consider an iterative linearized scheme for its solution. Representing last iterative values for \(u_r\) and \(u_z\) with \(u_{ro}\) and \(u_{zo}\) respectively, we obtain by combining Equations 2.10 and 2.11

\[
\int_{cv} \int \int \left[ \frac{1}{r} \frac{\partial (ru_{ro})}{\partial r} + \frac{\partial (u_{zo} u_r)}{\partial z} \right] 2\pi r_c \delta r \delta z
\]

\[
= \int_{cv} \int \int \left[ -\frac{\partial p}{\partial r} + \frac{1}{Re} \left( \frac{\partial}{r} \left( \frac{1}{r} \frac{\partial (ru_{ro})}{\partial r} \right) \right) + \frac{\partial^2 u_r}{\partial z^2} \right] 2\pi r_c \delta r \delta z
\]

Or,

\[
\left[ \frac{(ru_{ro})_e - (ru_{ro})_w}{r_c \delta r} + \frac{(u_{zo} u_r)_n - (u_{zo} u_r)_s}{\delta z} \right] r_c \delta r \delta z = -\left( \frac{p_e - p_w}{\delta r} \right) r_c \delta r \delta z
\]

\[
+ \left[ \frac{1}{Re} \left( \frac{\partial (ru_{ro})}{\delta r} \right)_e - \frac{1}{Re} \left( \frac{\partial (ru_{ro})}{\delta r} \right)_w + \frac{1}{Re} \left( \frac{\partial u_r}{\delta z} \right)_n - \left( \frac{\partial u_r}{\delta z} \right)_s \right] r_c \delta r \delta z
\]

Or,

\[
(r_e u_{ro} u_{re} - r_w u_{row} u_{rw})(z_n - z_s) + (u_{zon} u_{rn} - u_{zos} u_{rs})(r_e - r_w) r_c
\]

\[
= -\left( p_e - p_w \right)(z_n - z_s) r_c + \frac{1}{Re} \left( \frac{1}{r} \frac{\partial (ru_r)}{\partial r} \right)_e - \left( \frac{1}{r} \frac{\partial (ru_r)}{\partial r} \right)_w \left( z_n - z_s \right) r_c
\]
where the convective coefficients \( C \) and the diffusive coefficients \( D \) are designated with appropriate subscripts for adjacent neighbors. Clearly, these coefficients are as given below.

\[
C_{re} = (z_n - z_s) r_c \hat{u}_{roe}, \quad C_{rw} = (z_n - z_s) r_c \hat{u}_{row}, \\
C_{rn} = (r_e - r_w) r_c \hat{u}_{zon}, \quad C_{rs} = (r_e - r_w) r_c \hat{u}_{zos}, \\
D_{rE} = (z_n - z_s) r_c \frac{r_p}{Re_c(r_E - r_p)}, \quad D_{WW} = (z_n - z_s) r_c \frac{r_w}{Re_w(r_p - r_w)}, \\
D_{rrp} = (z_n - z_s) r_c \frac{r_p}{Re_c(r_E - r_p)} + \frac{r_p}{Re_w(r_p - r_w)}, \\
D_{rN} = (r_e - r_w) r_c \frac{1}{Re(z_N - z_p)}, \quad D_{rS} = (r_e - r_w) r_c \frac{1}{Re(z_p - z_S)},
\]
Chapter 2

\[ D_{r_p} = (r_e - r_w) r_c \left( \frac{1}{\text{Re} (z_N - z_p)} + \frac{1}{\text{Re} (z_p - z_S)} \right), \]  
and \[ D_{r_p} = D_{r_p} + D_{r_p}. \]

Now, old facial velocities are calculated using average values of the adjacent nodes as follows:

\[ u_{roe} = \left( u_{roE} + u_{rop} \right)/2, \quad u_{row} = \left( u_{rop} + u_{row} \right)/2, \]
\[ u_{zon} = \left( u_{zonw} + u_{zone} \right)/2, \quad \text{and} \quad u_{zos} = \left( u_{zosw} + u_{zose} \right)/2. \]

The facial values of the variable (namely, \( u_{re}, u_{rw}, u_{rn} \) and \( u_{rs} \)) are calculated using central differencing scheme (CDS) or upwind differencing scheme (UDS) depending upon the magnitude of the cell Peclet number (Pe) at each face of the control volume. The central differencing scheme is employed for small Peclet numbers (Pe < 2) and the upwind scheme is employed for large Peclet number (Pe ≥ 2). The Peclet number is defined by the ratio of convective flux per unit mass \( F \) to diffusive conductance \( D \).

For example, for west face,

\[ Pe_w = \frac{F_w}{D_w} = \frac{(\rho u)_w}{(\mu/\delta r)_w} = \frac{\rho_w u_w}{u_w/\delta r_{wp}}. \]

Therefore, if Pe < 2, then, from the central difference scheme, the current facial \( u_r \) - component of velocity will be as follows:

\[ u_{re} = \left( u_{re} + u_{rp} \right)/2, \quad u_{rw} = \left( u_{rp} + u_{rw} \right)/2, \]
\[ u_{rn} = \left( u_{rn} + u_{rp} \right)/2, \quad \text{and} \quad u_{rs} = \left( u_{rp} + u_{rs} \right)/2. \]

The upwind scheme is sensitive to direction of flow. Hence, for the first order upwind scheme the facial values of the radial velocity is represented as

\[ u_{re} = u_{rp}, \quad u_{rw} = u_{rw}, \quad u_{rn} = u_{rp}, \quad u_{rs} = u_{rs} \]

when \( u_{row} > 0, u_{row} > 0, u_{zon} > 0, u_{zos} > 0 \)

or,

\[ u_{re} = u_{re}, \quad u_{rw} = u_{rp}, \quad u_{rn} = u_{rn}, \quad u_{rs} = u_{rp} \]

when \( u_{row} < 0, u_{row} < 0, u_{zon} < 0, u_{zos} < 0 \).

For the third order upwind scheme (TUDS) the facial values of the variable are expressed following a three-point upstream-weighted quadratic interpolation for cell-face values. The face value of variable is obtained from a quadratic function passing through two neighboring nodes on each side of the face and an additional neighboring node on the upstream side.

For example, when \( u_w > 0 \) a quadratic fit through WW, W and P is used to evaluate the variable (say, \( \phi_w \)). Thus, we write

\[ \text{if} \quad u_{roe} > 0 \quad \text{then} \quad u_{re} = f'_{eE} u_{re} + f'_{eP} u_{rp} - f'_{eW} u_{nw}, \]
\[ \text{else} \quad u_{re} = f'_{eP} u_{rp} + f'_{eE} u_{re} - f'_{eEE} u_{rEE}. \]
Chapter 2

if \( u_{row} > 0 \) then \( u_{rw} = f_{w_p} u_{rp} + f_{w_p} u_{rw} - f_{w_w} u_{rw} \),
else \( u_{rw} = f'_{w_w} u_{rw} + f'_{w_p} u_{rp} - f'_{w_e} u_{re} \);

if \( u_{zon} > 0 \) then \( u_{rn} = f_{n_p} u_{rp} + f_{n_p} u_{rn} - f_{n_s} u_{rs} \),
else \( u_{rn} = f'_{n_p} u_{rp} + f'_{n_p} u_{rn} - f'_{n_n} u_{rN} \);

if \( u_{zos} > 0 \) then \( u_{rs} = f_{s_p} u_{rp} + f_{s_s} u_{rs} - f_{s_s} u_{rs} \),
else \( u_{rs} = f'_{s_s} u_{rs} + f'_{s_p} u_{rp} - f'_{s_n} u_{rN} \),

where \( f \) and \( f' \) are cell weights.

Now, substituting the interfacial velocities in terms of the adjacent nodal velocities in appropriate form - namely CDS, UDS or TUDS - Equation 2.12 can be written in the following fashion

\[
a_{rp} u_{rp} = a_{rw} u_{rw} + a_{re} u_{re} + a_{rs} u_{rs} + a_{rN} u_{rN} + (p_w - p_c) (z_n - z_s) r_c \\
(2.13)
\]

where the coefficients \( a \) with subscripts can be expressed in terms of the coefficients defined in this sub-section as \( C \) and \( D \) with appropriate subscripts.

The above equation has also been referred later in the compact form as

\[
a_{rp} u_{rp} = \sum a_{rb} u_{rb} + (p_w - p_c) (z_n - z_s) r_c .
\]  

2.4.2 Discretization of \( z \)-Component of Momentum Equation

Proceeding in the same manner as described in the above sub-section, the Equation 2.8c along with Equation 2.8a can be integrated over the finite volume about cell center (P) as shown in Figure 2.4. This gives

\[
\iiint_{cv} \left( \frac{1}{r} \frac{\partial (ru_z)}{\partial r} + \frac{\partial (u_z u_z)}{\partial z} \right) dV = \iiint_{cv} \left( -\frac{\partial p}{\partial z} + \frac{1}{Re} \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_z}{\partial r} \right) + \frac{\partial^2 u_z}{\partial z^2} \right) \right) dV . \\
(2.14)
\]

Upon linearization, the above equation reduces to

\[
\iiint_{cell} \left( \frac{1}{r} \frac{\partial (ru_{zp} u_z)}{\partial r} + \frac{\partial (u_{zo} u_z)}{\partial z} \right) dV = \iiint_{cell} \left( -\frac{\partial p}{\partial z} + \frac{1}{Re} \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_z}{\partial r} \right) + \frac{\partial^2 u_z}{\partial z^2} \right) \right) dV .
\]

and finally into,
Figure 2.4: Control Volume Discretization for Axial Velocity

\[
\begin{align*}
    u_{roe}(z_n - z_s) r_c u_{ze} - u_{row}(z_n - z_s) r_w u_{zw} + u_{zon}(r_e - r_w) r_c u_{zn} - u_{zos}(r_e - r_w) r_c u_{zs} \\
    &= -(p_n - p_S)(r_e - r_w) r_c \\
    &+ (z_n - z_s) \frac{r_e}{\text{Re}(r_E - r_P)} u_{rE} + (z_n - z_s) \frac{r_w}{\text{Re}(r_p - r_w)} u_{rW} \\
    &- (z_n - z_s) \left( \frac{r_e}{\text{Re}(r_E - r_P)} + \frac{r_w}{\text{Re}(r_p - r_w)} \right) u_{rP} \\
    &+ (r_e - r_w) r_c \frac{1}{\text{Re}(z_N - z_P)} u_{rN} + (r_e - r_w) r_c \frac{1}{\text{Re}(z_P - z_S)} u_{rS} \\
    &- (r_e - r_w) r_c \left( \frac{1}{\text{Re}(z_N - z_P)} + \frac{1}{\text{Re}(z_P - z_S)} \right) u_{rP}
\end{align*}
\]

Or,

\[
\begin{align*}
    C_{ze} u_{ze} - C_{zw} u_{zw} + C_{zn} u_{zn} - C_{zs} u_{zs} &= D_{ze} u_{ze} + D_{zw} u_{zw} - D_{zp} u_{zp} \\
    &+ D_{zn} u_{zn} + D_{zs} u_{zs} - D_{zp} u_{zp} - (p_n - p_S)(r_e - r_w) r_c
\end{align*}
\]

Or,

\[
\begin{align*}
    C_{ze} u_{ze} - C_{zw} u_{zw} + C_{zn} u_{zn} - C_{zs} u_{zs} &= D_{ze} u_{ze} + D_{zw} u_{zw} + D_{zn} u_{zn} + D_{zs} u_{zs} \\
    &- D_{zp} u_{zp} + (p_n - p_S)(r_e - r_w) r_c
\end{align*}
\]

where

\[
\begin{align*}
    C_{ze} &= (z_n - z_s)r_e u_{roe}, & C_{zw} &= (z_n - z_s)r_w u_{row} \\
    C_{zn} &= (r_e - r_w)r_c u_{zon}, & C_{zs} &= (r_e - r_w)r_c u_{zos} \\
    D_{ze} &= (z_n - z_s) r_e / \text{Re}(r_E - r_P), & D_{zw} &= (z_n - z_s) r_w / \text{Re}(r_p - r_w) \\
    D_{zp} &= (z_n - z_s) r_e / \text{Re}(r_E - r_P) + (z_n - z_s) r_w / \text{Re}(r_p - r_w) = D_{ze} + D_{zw} \\
    D_{zn} &= (r_e - r_w) r_c / \text{Re}(z_N - z_P), & D_{zs} &= (r_e - r_w) r_c / \text{Re}(z_P - z_S)
\end{align*}
\]
$D_{zz_p} = (r_e - r_w) r_c / \text{Re}(z_N - z_p) + (r_e - r_w) r_c / \text{Re}(z_p - z_S) = D_{z_N} + D_{z_S}$

$D_{zp} = D_{zp} + D_{zz_p}$

$u_{row} = (u_{row} + u_{row}) / 2$,

$u_{zoz} = (u_{zoz} + u_{zoz}) / 2$.

For convective velocities, if $Pe < 2$, then, from the central difference scheme, the current facial $u_x$ - component of velocity will be as follows:

$u_{ze} = (u_{ze} + u_{ze}) / 2$, $u_{zw} = (u_{zp} + u_{zw}) / 2$, $u_{zn} = (u_{zn} + u_{zp}) / 2$, $u_{zn} = (u_{zp} + u_{zs}) / 2$.

For the first order upwinding, the facial values of the axial velocity is

$u_{ze} = u_{zp}, u_{zw} = u_{zp}, u_{zn} = u_{zp}, u_{zs} = u_{zs}$

when $u_{roe} > 0, u_{row} > 0, u_{zon} > 0, u_{zos} > 0$.

or,

$u_{ze} = u_{zp}, u_{zw} = u_{zp}, u_{zn} = u_{zp}, u_{zs} = u_{zp}$

when $u_{roe} < 0, u_{row} < 0, u_{zon} < 0, u_{zos} < 0$.

For the third order upwind scheme (TUDS) the facial values of the variable will be as follows.

if $u_{roe} > 0$ then $u_{ze} = f_e u_{zp} + f_e u_{zp} - f_e u_{zw}$,

else $u_{ze} = f'_e u_{zp} + f'_e u_{zp} - f'_e u_{zw}$.

if $u_{row} > 0$ then $u_{zw} = f_w u_{zp} + f_w u_{zp} - f_w u_{zw}$,

else $u_{zw} = f'_w u_{zp} + f'_w u_{zp} - f'_w u_{zw}$.

if $u_{zon} > 0$ then $u_{zn} = f_n u_{zp} + f_n u_{zp} - f_n u_{zs}$,

else $u_{zn} = f'_n u_{zp} + f'_n u_{zp} - f'_n u_{zn}$.

if $u_{zos} > 0$ then $u_{zs} = f'_w u_{zp} + f'_w u_{zp} - f'_w u_{zn}$,

else $u_{zs} = f'_w u_{zp} + f'_w u_{zp} - f'_w u_{zn}$.

where $f$ and $f'$ are cell weights.

Finally, Equation 2.14 can be written as

$a_{zp} u_{zp} = a_{zw} u_{zw} + a_{ze} u_{ze} + a_{z} u_{z} + a_{zn} u_{zn} + (p_S - p_n) (r_e - r_w) r_c$ (2.15)

or,

$a_{zp} u_{zp} = \sum a_{nb} u_{nb} + (p_S - p_n) (r_e - r_w) r_c$

where the coefficients $a$ with subscripts can be expressed in terms of the coefficients defined in this sub-section as $C$ and $D$ with appropriate subscripts.

2.4.3 Discretization of Continuity Equation

Proceeding in the same manner as discussed in the preceding two sub-sections, Equation 2.8a is integrated over the finite volume about cell centre (P) as shown in Figure 2.5. This gives
\[ \int \left[ \frac{1}{r} \frac{\partial (ru_r)}{\partial r} + \frac{\partial u_z}{\partial z} \right]_{\text{cell}} 2\pi r c \delta r \delta z = 0 \]

or,

\[ \frac{(ru_r)_e - (ru_r)_w}{r \delta r} r \delta r \delta z + \frac{(u_z)_n - (u_z)_s}{\delta z} r_c \delta r \delta z = 0 \]

or,

\[ (r_e u_{re} - r_w u_{rw})(z_n - z_s) + (u_{zn} - u_{zs})(r_e - r_w) r_c = 0. \quad (2.16) \]

Now introducing velocity correction, facial velocities are written as

\[ u_{re} = u^*_{re} + (p'_p - p'_E)(z_n - z_s) r_c / a_{rp} \]
\[ u_{rw} = u^*_{rw} + (p'_w - p'_P)(z_n - z_s) r_c / a_{rp} \]
\[ u_{zn} = u^*_{zn} + (p'_p - p'_N)(r_e - r_w)r_c / a_{zp} \]
\[ u_{zs} = u^*_{zs} + (p'_s - p'_N)(r_e - r_w)r_c / a_{zp} \]

where, \( u^* \) are the velocities obtained from the solution of momentum equations.

Now, incorporating above velocity expressions in Equation 2.16, it can be written that

\[ a_{cp} p'_p = a_{cw} p'_w + a_{cE} p'_E + a_{cS} p'_S + a_{cN} p'_N + b_c \quad (2.17) \]

where

\[ a_{cE} = r_e r_c (z_n - z_s)^2 / a_{rp}, \quad a_{cw} = r_w r_c (z_n - z_s)^2 / a_{rp}, \]
\[ a_{cN} = r_c^2 (r_e - r_w)^2 / a_{zp}, \quad a_{cS} = r_c^2 (r_e - r_w)^2 / a_{zp}, \]
\[ a_{cp} = a_{cN} + a_{cS} + a_{cE} + a_{cw} \]
and

\[ b_c = [r_w(z_n - z_s) u_{re}^* - r_e(z_n - z_s) u_{re}^*] + [r_e(r_e - r_w) u_{zn}^* - r_c(r_e - r_w) u_{zn}^*]. \]

Following the discretization, we now briefly present the solution algorithm.
2.4 Solution Algorithm

The numerical solution of the problem demands the solution of the conservation equations. In the last section the treatment of conservation equations have been described in detail. This has been coupled with the SIMPLE or SIMPLER technique of solving the transport equations. The algorithm for the code is provided.

![Flow Diagram for Laminar CFD Solution (SIMPLE)](image)

Figure 2.6: Flow Diagram for Laminar CFD Solution (SIMPLE)
below. A flow diagram for the algorithm is presented in Figure 2.6. In terms of this figure, the algorithm is outlined below.

**Algorithm**

1. Read input, generate grid and set boundary conditions.
2. Guess initial pressure field and velocity field.
3. Solve r-momentum equation to obtain $u_r$ velocity.
4. Solve z-momentum equation to obtain $u_z$ velocity.
5. Solve of pressure correction equation.
6. Correct pressure and velocity field.
7. Test the level of convergence. If convergence is obtained in terms of pressure and velocity, then go to step 11 else go to step 10.
8. Set new guess pressure field and velocity field using last iteration values through under relaxation. Next go to step 4.
9. If solution converges, then show results and stop.

**2.5 Code Validation**

As a first validation case, the flow between two parallel plates is solved. The geometry and the boundary conditions are indicated in Figure 2.1 (a). Equations 2.9a to 2.9c corresponding to 2-dimensional planer geometry are solved.

**Table-2.1: Validation Data for Flow through Parallel Plates**

<table>
<thead>
<tr>
<th>Re</th>
<th>Fully Developed Pressure Gradient $[\partial p/\partial z]$</th>
<th>Entrance length</th>
<th>Fully Developed $u_{axl}/U_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Theoretical</td>
<td>Numerical</td>
<td>Existing</td>
</tr>
<tr>
<td>10</td>
<td>-1.20</td>
<td>-1.180</td>
<td>1.50</td>
</tr>
<tr>
<td>50</td>
<td>-0.24</td>
<td>-0.236</td>
<td>3.55[**]</td>
</tr>
<tr>
<td>100</td>
<td>-0.12</td>
<td>-0.119</td>
<td>7.10[***]</td>
</tr>
<tr>
<td>200</td>
<td>-0.06</td>
<td>-0.058</td>
<td>14.20[**]</td>
</tr>
<tr>
<td>250</td>
<td>-0.05</td>
<td>-0.047</td>
<td>17.75[**]</td>
</tr>
<tr>
<td>300</td>
<td>-0.04</td>
<td>-0.039</td>
<td>21.30[***]</td>
</tr>
</tbody>
</table>

[*] Cebeci and Bradshaw (1977) show from their numerical solution that the entrance length in a duct for laminar flow can be determined as, $L_e = 0.071 \times D \times Re$.

[**] Granger (1995) predicted that for very Low Reynolds Number $L_e = 0.080 \times D \times Re + 0.7 \times D$.

Flow through this configuration is analyzed at different Reynolds number. Tables 2.1 and 2.2 present the validation of the results obtained from the code. The code is run for $h=0.5$ and $U_0=1.0$. Results are shown up to Re=300, for which developing length and pressure gradient are compared with the theoretically obtained values. The maximum centerline velocity at fully developed regime should be 1.5
times the average velocity \([U_0]\). The values obtained from the code are also tabulated in Table-2.1. All the data are in good agreement with the theoretical values and ensure validation of the code.

Table 2.2 shows the fully developed horizontal velocity profile for flow through parallel plates at Different Reynolds numbers. In Table 2.3, the same is shown corresponding to a circular duct at Reynolds number of 200. Numerically obtained velocity is compared with the theoretical velocity at different distance from the plane or axis of symmetry. The tables show good agreement between numerical and theoretical velocity values.

<table>
<thead>
<tr>
<th>Re 10</th>
<th>Re 100</th>
<th>Re 300</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>y</td>
<td>(u_{nu})</td>
<td>(u_{th})</td>
</tr>
<tr>
<td>0.5000</td>
<td>1.480</td>
<td>1.500</td>
</tr>
<tr>
<td>0.4593</td>
<td>1.480</td>
<td>1.489</td>
</tr>
<tr>
<td>0.3798</td>
<td>1.400</td>
<td>1.411</td>
</tr>
<tr>
<td>0.3078</td>
<td>1.270</td>
<td>1.276</td>
</tr>
<tr>
<td>0.2434</td>
<td>1.100</td>
<td>1.102</td>
</tr>
<tr>
<td>0.1866</td>
<td>0.902</td>
<td>0.908</td>
</tr>
<tr>
<td>0.1374</td>
<td>0.704</td>
<td>0.708</td>
</tr>
<tr>
<td>0.0957</td>
<td>0.514</td>
<td>0.517</td>
</tr>
<tr>
<td>0.0616</td>
<td>0.343</td>
<td>0.345</td>
</tr>
<tr>
<td>0.0351</td>
<td>0.201</td>
<td>0.202</td>
</tr>
<tr>
<td>0.0162</td>
<td>0.094</td>
<td>0.095</td>
</tr>
<tr>
<td>0.0048</td>
<td>0.028</td>
<td>0.029</td>
</tr>
<tr>
<td>0.0005</td>
<td>0.003</td>
<td>0.003</td>
</tr>
<tr>
<td>0.0000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

Table 2.3: Fully Developed Velocity Profile for Flow through Circular Duct

<table>
<thead>
<tr>
<th>Radius</th>
<th>0.015</th>
<th>0.075</th>
<th>0.135</th>
<th>0.195</th>
<th>0.255</th>
<th>0.315</th>
<th>0.375</th>
<th>0.435</th>
<th>0.485</th>
<th>0.498</th>
</tr>
</thead>
<tbody>
<tr>
<td>(U_{th}/U_0)</td>
<td>1.998</td>
<td>1.955</td>
<td>1.854</td>
<td>1.696</td>
<td>1.480</td>
<td>1.206</td>
<td>0.875</td>
<td>0.486</td>
<td>0.118</td>
<td>0.019</td>
</tr>
<tr>
<td>(U_{nu}/U_0)</td>
<td>1.994</td>
<td>1.951</td>
<td>1.850</td>
<td>1.692</td>
<td>1.477</td>
<td>1.204</td>
<td>0.874</td>
<td>0.486</td>
<td>0.118</td>
<td>0.019</td>
</tr>
</tbody>
</table>

In Figure 2.7 is presented the axial velocity distribution at different cross section in the entrance region of a circular duct of diameter \(D\). The location of the different cross sections are indicated in the figure in terms of the normalized axial length \(L^*\) defined as

\[
L^* = \frac{L}{D} \cdot \frac{0.057}{Re}
\]

The denominator of the last expression is the ratio, as predicted by Langhaar (1942), of entrance length to the pipe diameter. It is seen that instead of a normalized length of 1.0, the velocity profile approaches the analytical prediction of the Poiseilleu
profile at about a value of 1.478. In fact for the normalized length of about 1.0, the prediction of the axial velocity at the axis of the duct is about 97% of the fully developed value of 2.0. All the results in Figure 2.7 have been generated for a non-dimensional duct length of 180 with a uniform axial spacing of 360 nodes in the interior. In order to resolve the wall gradient with optimum number of nodes, a non-uniform mesh with 20 nodes in the radial direction has been adopted with higher nodal density near the wall. Near the axis, inlet and outlet grid size that is half of the adjacent spacing has been used.

![Figure 2.7: Development of Axial Velocity Profile for Flow through circular duct](image)

**Figure 2.7: Development of Axial Velocity Profile for Flow through circular duct**

In Figure 2.8, the normalized values of numerically predicted pressure gradient \( P^{**} \) and wall shear stress \( \tau^{**} \) in a straight circular pipe has been plotted against the normalized axial length. First two variables have been normalized by the respective analytical values at the fully developed region. Thus, we obtain

\[
P^{**} = \frac{\Delta P/\left(0.5 \rho U_0^2\right)}{L/D} = \frac{\Delta P^*/L^*}{32/Re},
\]

(2.18a)
\[ \tau^{**} = \frac{\tau_{\text{wall}}}{16/\text{Re}} \left( 0.5 \rho U_o^2 \right) \]

where the quantities with single * in the superscript are the non-dimensional computational quantities, and \( U_o \) is the mean axial flow velocity in the duct. Figure 2.8 clearly reveals that as the normalized axial location in the duct approaches one, the normalized flow variables expressed by the last two equations also approach 1.0. This is a clear indication of the acceptability of the numerical code. The agreement of the wall shear stress between the analytical and numerical predictions exhibited by the normalized value of 1.0 demonstrates the velocity resolution to be acceptable near the wall. Similarly, the normalized value of pressure gradient equal to 1.0 in the fully developed region indicates that the numerical prediction is accurate with respect to the analytical expression in the region where the later is obtainable. In the SIMPLE algorithm, the continuity equation, which is physically a kinematic equation in terms of only velocity, is reconstructed as the equation to predict pressure, and therefore, poses considerable numerical difficulties. The good prediction of the pressure gradient by the numerical code shown in the figure is a clear demonstration of the successful implementation of the code.

We further mention that in the entrance region, with respect to the axial pressure gradient, the high value of wall shear stress because of thinness of the boundary layer results in a gradual reduction of momentum across the pipe in the axial direction. In terms of the normalized values, the above statement yields that \( 2\tau^{**} > \Delta P^{**} \). Figure 2.8 clearly indicates that the numerical predictions in the entrance region are consistent with the above inequality. All these studies establish the acceptability of the numerical code.

### 2.6 Conclusion

A numerical code has been developed for analyzing the laminar internal flow. Here, the code has been validated against the analytically known results of flow between parallel plates and flow through circular ducts.