Computational Fluid Dynamics

Bachelor's Degree in Aeronautical and Space Sciences

Emanuel A. R. Camacho

emanuel.camacho@iseclisboa.pt
earc@earc96.com

Instituto Superior de Educação e Ciências (ISEC Lisboa)

Latest Update: *28th October, 2026, 11:37*

Bibliography

- White, F. M. (2015). Fluid Mechanics (8th ed.). McGraw-Hill Education.
- Batchelor, G. K. (1967). An Introduction to Fluid Dynamics. Cambridge University Press.
- John David Anderson Jr. (1995). Computational Fluid Dynamics: The Basics with Applications, McGraw-Hill.
- Patankar, S. V. (1980). Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation.

Computational Fluid Dynamics

- 45 hours
- 3 hours per week (classes of 3 hours)
  - First part: 1h:25min
  - Break: 15min
  - Second part: 1h:20min
- Active Learning

Computational Fluid Dynamics is a field that uses numerical methods to solve the governing equations of fluid mechanics and predict the behaviour of fluid flows.

Evaluation

Computational Fluid Dynamics (100% [20/20])

Frequency (40%) + Project (60%)

- Frequency (40% [8/20]) (DD/MM/2027)
- Project (60% [12/20]) (DD/MM/2027)

or

Exam (100% [20/20])

- Exam (100% [20/20])

[10/20] is required to pass.

Ansys Student

- **Terms of Use:** Free student downloads are for educational use only and may only be used for self-learning, student instruction, student projects, and student demonstrations.
- `https://ansys.synopsys.com/academic/students/ansys-student`

Be aware of

- System and Browser Requirements
- Problem Size Limits
- License Duration

Are we ready?

- Suffering is a fundamental human experience.
- Do you like Fluid Mechanics?
- Do you like Calculus?
- Do you like Computational Mathematics?
- Do you tolerate frustration?
- Do you enjoy solving problems that create more problems?
- Can you survive when things do not work on the tenth try?
- Do you have patience?
- **Good. You are going to need all of it.**

Table of Contents

1. Outline
2. Continuum Mechanics and Fluid Description
3. Governing Equations
4. From Navier-Stokes to RANS
5. Solving Turbulent Flow
6. Discretization
7. Pressure-Velocity Coupling
8. Boundary Conditions
9. Geometry
10. Meshing

Outline

1. Outline
2. Continuum Mechanics and Fluid Description
3. Governing Equations
4. From Navier-Stokes to RANS
5. Solving Turbulent Flow
6. Discretization
7. Pressure-Velocity Coupling
8. Boundary Conditions
9. Geometry
10. Meshing

Table of Contents

1 Outline

2 Continuum Mechanics and Fluid Description

3 Governing Equations

4 From Navier-Stokes to RANS

5 Solving Turbulent Flow

6 Discretization

7 Pressure-Velocity Coupling

8 Boundary Conditions

9 Geometry

10 Meshing

Continuum Mechanics: The Basic Idea

Fluid mechanics describes the motion and interaction of matter at a macroscopic scale.

Instead of tracking individual molecules, we assume that the fluid can be treated as a continuous medium.

$$\boxed{\text{Molecular description} \quad \longrightarrow \quad \text{Continuum description}} \tag{1}$$

Under this assumption, physical quantities such as

- density ρ,
- pressure p,
- temperature T,
- velocity $\mathbf{u}$,

are defined at every point in space and time.

The Continuum Hypothesis

At the molecular level, a fluid consists of discrete molecules.

However, for most engineering flows, we consider a sufficiently large number of molecules inside a small volume.

$$\boxed{\text{Small compared with the flow scale} \quad \text{but} \quad \text{large compared with molecular scales}} \tag{2}$$

This allows fluid properties to be represented by smooth fields:

$$\rho = \rho(x, y, z, t) \tag{3}$$

$$p = p(x, y, z, t) \tag{4}$$

$$T = T(x, y, z, t) \tag{5}$$

$$\mathbf{u} = \mathbf{u}(x, y, z, t) \tag{6}$$

Fluid Properties as Fields

A fluid is described by physical quantities that vary with position and time.

For example, the velocity field is

$$\mathbf{u}(x, y, z, t) = \begin{bmatrix} u(x, y, z, t) \\ v(x, y, z, t) \\ w(x, y, z, t) \end{bmatrix} \tag{7}$$

where

- u is the velocity component in the x direction,
- v is the velocity component in the y direction,
- w is the velocity component in the z direction.

Similarly,

$$\rho = \rho(x, y, z, t), \qquad p = p(x, y, z, t), \qquad T = T(x, y, z, t). \tag{8}$$

Eulerian Description

In the **Eulerian description**, we observe what happens at fixed locations in space.

We define the flow variables as functions of

$$(x, y, z, t). \tag{9}$$

For example,

$$\boxed{\mathbf{u} = \mathbf{u}(x, y, z, t)} \tag{10}$$

describes the velocity at a particular position and time.

The observer remains fixed while the fluid moves through the observation point.

This is the description most commonly used in CFD.

Lagrangian Description

In the **Lagrangian description**, we follow individual fluid particles as they move.

The position of a particle is

$$\mathbf{x} = \mathbf{x}(t) \tag{11}$$

and its velocity is

$$\mathbf{u} = \frac{d\mathbf{x}}{dt}. \tag{12}$$

The two descriptions answer different questions:

Eulerian	What is happening here?
Lagrangian	What happens to this fluid particle?

CFD based on finite volumes is naturally formulated using the Eulerian viewpoint.

From Particle Motion to the Material Derivative

Although CFD usually uses the Eulerian description, we may still want to know how a fluid particle experiences a change in a property.

Consider a scalar field

$$\phi = \phi(x, y, z, t). \tag{13}$$

Following a fluid particle gives

$$\frac{D\phi}{Dt} = \frac{\partial \phi}{\partial t} + u\frac{\partial \phi}{\partial x} + v\frac{\partial \phi}{\partial y} + w\frac{\partial \phi}{\partial z}. \tag{14}$$

Therefore,

$$\boxed{\frac{D\phi}{Dt} = \frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi} \tag{15}$$

Material Derivative: Physical Meaning

The material derivative contains two contributions:

$$\boxed{\frac{D\phi}{Dt} = \underbrace{\frac{\partial \phi}{\partial t}}_{\text{Local change}} + \underbrace{\mathbf{u} \cdot \nabla \phi}_{\text{Convective change}}} \tag{16}$$

Local change

- Change observed at a fixed point.

Convective change

- Change experienced because the particle moves through regions with different values of ϕ.

For $\phi = \mathbf{u}$,

$$\boxed{\frac{D\mathbf{u}}{Dt} = \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u}} \tag{17}$$

which represents the acceleration of a fluid particle.

System and Control Volume

Two useful viewpoints are used when deriving conservation laws.

System
A fixed collection of fluid particles is followed through time.

$$\boxed{\text{Follow the matter}} \tag{18}$$

Control Volume
A region in space is selected and fluid is allowed to flow through its boundaries.

$$\boxed{\text{Observe the flow through a region}} \tag{19}$$

CFD commonly uses the **control-volume viewpoint**.

This leads naturally to the finite volume method:

$$\boxed{\text{Control volume} \quad \longrightarrow \quad \text{Conservation over each cell}} \tag{20}$$

From Fluid Description to CFD

We can now connect the physical description to CFD.

$$\boxed{\text{Fluid} \rightarrow \text{Fields} \rightarrow \text{Conservation Laws} \rightarrow \text{Discretization} \rightarrow \text{Numerical Solution}} \tag{21}$$

The fluid is represented by fields such as

$$\rho, \quad p, \quad \mathbf{u}, \quad T.$$

The governing equations describe how these fields evolve.

The finite volume method then converts the continuous equations into algebraic equations over discrete control volumes.

$$\boxed{\text{Continuum} \rightarrow \text{Control Volumes} \rightarrow \text{Algebraic Equations}} \tag{22}$$

Table of Contents

1 Outline

2 Continuum Mechanics and Fluid Description

3 Governing Equations

4 From Navier-Stokes to RANS

5 Solving Turbulent Flow

6 Discretization

7 Pressure-Velocity Coupling

8 Boundary Conditions

9 Geometry

10 Meshing

Vector Calculus: Operator Summary

Gradient	$\nabla\phi$	:	scalar $\rightarrow$ vector
Gradient	$\nabla\mathbf{A}$	:	vector $\rightarrow$ tensor
Divergence	$\nabla \cdot \mathbf{A}$	:	vector $\rightarrow$ scalar
Curl	$\nabla \times \mathbf{A}$	:	vector $\rightarrow$ vector
Laplacian	$\nabla^2\phi$	:	scalar $\rightarrow$ scalar

The operator ∇ acts differently depending on the field and the operation being performed.

Vector Calculus: Operator Summary

Gradient	$\nabla\phi = \begin{bmatrix} \partial\phi/\partial x \\ \partial\phi/\partial y \\ \partial\phi/\partial z \end{bmatrix}$	:	scalar $\rightarrow$ vector
Gradient of a vector	$\nabla\mathbf{A} = \begin{bmatrix} \partial A_x/\partial x & \partial A_x/\partial y & \partial A_x/\partial z \\ \partial A_y/\partial x & \partial A_y/\partial y & \partial A_y/\partial z \\ \partial A_z/\partial x & \partial A_z/\partial y & \partial A_z/\partial z \end{bmatrix}$	:	vector $\rightarrow$ tensor
Divergence	$\nabla \cdot \mathbf{A} = \dfrac{\partial A_x}{\partial x} + \dfrac{\partial A_y}{\partial y} + \dfrac{\partial A_z}{\partial z}$	:	vector $\rightarrow$ scalar
Curl	$\nabla \times \mathbf{A} = \begin{bmatrix} \partial A_z/\partial y - \partial A_y/\partial z \\ \partial A_x/\partial z - \partial A_z/\partial x \\ \partial A_y/\partial x - \partial A_x/\partial y \end{bmatrix}$	:	vector $\rightarrow$ vector
Laplacian	$\nabla^2\phi = \dfrac{\partial^2\phi}{\partial x^2} + \dfrac{\partial^2\phi}{\partial y^2} + \dfrac{\partial^2\phi}{\partial z^2}$	:	scalar $\rightarrow$ scalar

Differential Fluid Volume

Consider an infinitesimal fluid volume

$$dV = dx\,dy\,dz$$

bounded by the six faces

$$x,\ x+dx, \qquad y,\ y+dy, \qquad z,\ z+dz.$$

The velocity field is

$$\mathbf{u} = (u, v, w),$$

with

$$\rho = \rho(x, y, z, t).$$

General Conservation Statement

$$\boxed{\text{Accumulation} + \text{Net Outflow} = \text{Net Generation}}$$

or, equivalently,

$$\boxed{\text{Accumulation} + \text{Net Outflow} - \text{Net Generation} = 0}$$

For a general conserved quantity per unit volume ϕ:

$$\boxed{\frac{d}{dt}\int_V \phi \, dV + \oint_S \phi \, \mathbf{u} \cdot \mathbf{n} \, dA = \int_V S_\phi \, dV}$$

where S_ϕ represents the volumetric source or sink of the conserved quantity.

Mass Inside the Element

The mass per unit volume is

$$\rho.$$

The mass contained in the infinitesimal element is

$$dm = \rho \, dV = \rho \, dx \, dy \, dz.$$

Therefore, the rate of mass accumulation is

$$\boxed{\frac{\partial dm}{\partial t} = \frac{\partial \rho}{\partial t} \, dx \, dy \, dz}$$

Mass Flux in the x Direction

At the face x, mass enters at the rate

$$(\rho u)_x \, dy \, dz.$$

At $x + dx$, mass leaves at

$$(\rho u)_{x+dx} \, dy \, dz.$$

Taylor expansion:

$$(\rho u)_{x+dx} = (\rho u)_x + \frac{\partial(\rho u)}{\partial x} dx.$$

Thus the net mass outflow is

$$\dot{m}_{x,\text{out}} = \left[(\rho u)_x + \frac{\partial(\rho u)}{\partial x} dx \right] dy \, dz - (\rho u)_x dy \, dz$$

$$= \boxed{\frac{\partial(\rho u)}{\partial x} \, dx \, dy \, dz}.$$

Mass Flux in the y Direction

At y:

$$(\rho v)_y \, dx \, dz.$$

At $y + dy$:

$$(\rho v)_{y+dy} \, dx \, dz.$$

Using

$$(\rho v)_{y+dy} = (\rho v)_y + \frac{\partial (\rho v)}{\partial y} dy,$$

the net outflow is

$$\boxed{\dot{m}_{y,\text{out}} = \frac{\partial (\rho v)}{\partial y} \, dx \, dy \, dz}$$

Mass Flux in the z Direction

Similarly,

$$(\rho w)_{z+dz} = (\rho w)_z + \frac{\partial(\rho w)}{\partial z} dz.$$

Therefore,

$$\boxed{\dot{m}_{z,\text{out}} = \frac{\partial(\rho w)}{\partial z}\, dx\, dy\, dz}$$

Total Net Mass Outflow

Adding the three directions,

$$\dot{m}_{\text{out}} = \dot{m}_{x,\text{out}} + \dot{m}_{y,\text{out}} + \dot{m}_{z,\text{out}}.$$

Therefore,

$$\boxed{\dot{m}_{\text{out}} = \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} \right] dx\, dy\, dz}$$

Conservation of Mass

Mass conservation states

$$\boxed{\text{Accumulation} + \text{Net Outflow} = 0}$$

Hence,

$$\frac{\partial \rho}{\partial t} dx\, dy\, dz + \left[\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} \right] dx\, dy\, dz = 0.$$

Since $dx\, dy\, dz \neq 0$,

$$\boxed{\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0}$$

Differential Conservation Form

Using vector notation,

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

or

$$\boxed{\underbrace{\frac{\partial \rho}{\partial t}}_{\text{Accumulation}} + \underbrace{\nabla \cdot (\rho \mathbf{u})}_{\text{Net mass outflow}} = 0}$$

This is the differential conservation equation for mass.

Expand the divergence:

$$\frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \nabla \rho + \rho \nabla \cdot \mathbf{u} = 0.$$

Therefore,

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{u} = 0.$$

For constant density,

$$\boxed{\nabla \cdot \mathbf{u} = 0}$$

or

$$\boxed{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0}$$

General Conservation Statement

$$\boxed{\text{Accumulation} + \text{Net Outflow} = \text{Net Generation}}$$

or, equivalently,

$$\boxed{\text{Accumulation} + \text{Net Outflow} - \text{Net Generation} = 0}$$

For a general conserved quantity per unit volume ϕ:

$$\boxed{\frac{d}{dt}\int_V \phi \, dV + \oint_S \phi \, \mathbf{u} \cdot \mathbf{n} \, dA = \int_V S_\phi \, dV}$$

where S_ϕ represents the volumetric source or sink of the conserved quantity.

Momentum Conservation

We use the same infinitesimal fluid element:

$$dV = dx\,dy\,dz$$

with

$$\mathbf{u} = (u, v, w).$$

Newton's second law gives

$$\boxed{\text{Rate of momentum accumulation} + \text{Net momentum outflow} = \text{Sum of forces}}$$

The forces acting on the fluid element are:

- pressure forces,
- viscous stresses,
- body forces.

We first derive the x-momentum equation.

Momentum Inside the Element

The x-momentum per unit volume is

$$\rho u.$$

Therefore, the total x-momentum inside the element is

$$\rho u \, dx \, dy \, dz.$$

Its rate of accumulation is

$$\boxed{\frac{\partial(\rho u)}{\partial t} \, dx \, dy \, dz}$$

Similarly,

$$\rho v \qquad \text{and} \qquad \rho w$$

are the y- and z-momentum densities.

Convective Transport of x-Momentum: x Direction

The x-momentum flow rate entering through the x-face is

$$(\rho u^2)_x \, dy \, dz.$$

At $x + dx$:

$$(\rho u^2)_{x+dx} \, dy \, dz.$$

Taylor expansion:

$$(\rho u^2)_{x+dx} = (\rho u^2)_x + \frac{\partial(\rho u^2)}{\partial x} dx.$$

Thus,

$$\boxed{\dot{P}^{(x)}_{x,\text{out}} = \frac{\partial(\rho u^2)}{\partial x} \, dx \, dy \, dz}$$

Convective Transport of x-Momentum: y Direction

The x-momentum flow rate entering through the y-face is

$$(\rho u v)_y \, dx \, dz.$$

At y:

$$(\rho u v)_y \, dx \, dz.$$

At $y + dy$:

$$(\rho u v)_{y+dy} \, dx \, dz.$$

Hence,

$$\boxed{\dot{P}^{(y)}_{x,\text{out}} = \frac{\partial(\rho u v)}{\partial y} \, dx \, dy \, dz}$$

Convective Transport of x-Momentum: z Direction

The x-momentum flow rate entering through the z-face is

$$(\rho u w)_z \, dx \, dy.$$

Using the same Taylor expansion,

$$\boxed{\dot{P}^{(z)}_{x,\text{out}} = \frac{\partial(\rho u w)}{\partial z} \, dx \, dy \, dz}$$

The total convective x-momentum outflow is therefore

$$\boxed{\nabla \cdot (\rho u \mathbf{u}) \, dV}$$

Pressure Forces in the x Direction

For x-momentum, only the two x-faces contribute.
At x:

$$F_{p,x}^{(x)} = p_x \, dy \, dz.$$

At $x + dx$, pressure acts in the opposite direction:

$$F_{p,x}^{(x+dx)} = -p_{x+dx} \, dy \, dz.$$

Using

$$p_{x+dx} = p_x + \frac{\partial p}{\partial x} dx,$$

we obtain

$$F_{p,x} = p_x dy \, dz - \left(p_x + \frac{\partial p}{\partial x} dx \right) dy \, dz = \boxed{-\frac{\partial p}{\partial x} \, dx \, dy \, dz}.$$

Viscous Stresses

The viscous stress tensor is

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix}.$$

For the x-momentum equation, the relevant stresses are

$$\boxed{\tau_{xx}, \qquad \tau_{xy}, \qquad \tau_{xz}}$$

because these are the stresses acting in the x direction.

Viscous Force in the x Direction

On the x-faces:

$$\tau_{xx}(x)\, dy\, dz.$$

$$\tau_{xx}(x + dx)\, dy\, dz.$$

$$\tau_{xx}(x + dx) = \tau_{xx}(x) + \frac{\partial \tau_{xx}}{\partial x}\, dx$$

The net force is

$$\boxed{\frac{\partial \tau_{xx}}{\partial x}\, dx\, dy\, dz}$$

Viscous Force in the x Direction

On the y-faces:

$$\tau_{xy}\, dx\, dz.$$

Therefore,

$$\boxed{\frac{\partial \tau_{xy}}{\partial y}\, dx\, dy\, dz}$$

On the z-faces:

$$\tau_{xz}\, dx\, dy.$$

Therefore,

$$\boxed{\frac{\partial \tau_{xz}}{\partial z}\, dx\, dy\, dz}$$

Total Viscous Force in x

Adding the three contributions:

$$\boxed{F_{\tau,x} = \left(\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right) dV}$$

or, using tensor notation,

$$\boxed{F_{\tau,x} = (\nabla \cdot \boldsymbol{\tau})_x \, dV}$$

Body Force

Suppose the body force per unit mass is

$$\mathbf{f} = (f_x, f_y, f_z).$$

The body force acting on the element is

$$\boxed{\mathbf{F}_b = \rho \mathbf{f}\, dV}$$

Therefore, the x-component is

$$\boxed{F_{b,x} = \rho f_x\, dx\, dy\, dz}$$

For gravity,

$$\mathbf{f} = \mathbf{g}.$$

Complete x-Momentum Balance

Newton's second law:

$$\boxed{\text{Accumulation} + \text{Net convection} = \text{Surface forces} + \text{Body forces}}$$

Therefore,

$$\begin{aligned}&\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z}\\ &= -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + \rho f_x.\end{aligned}$$

$$\boxed{x\text{-momentum conservation equation}}$$

The y-Momentum Equation

The conserved quantity is

$$\rho v.$$

The convective fluxes are

$$\rho uv, \qquad \rho v^2, \qquad \rho vw.$$

The relevant viscous stresses are

$$\tau_{yx}, \qquad \tau_{yy}, \qquad \tau_{yz}.$$

Pressure contributes

$$-\frac{\partial p}{\partial y}.$$

Thus,

$$\boxed{\begin{aligned}&\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}\\&=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{yx}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}+\rho f_y\end{aligned}}$$

The z-Momentum Equation

The conserved quantity is

$$\rho w.$$

The convective fluxes are

$$\rho uw, \qquad \rho vw, \qquad \rho w^2.$$

The relevant viscous stresses are

$$\tau_{zx}, \qquad \tau_{zy}, \qquad \tau_{zz}.$$

Pressure contributes

$$-\frac{\partial p}{\partial z}.$$

Therefore,

$$\boxed{\begin{aligned}&\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}\\&=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{zx}}{\partial x}+\frac{\partial \tau_{zy}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z\end{aligned}}$$

Conservative Momentum Equation

Combining the three momentum equations gives

$$\boxed{\frac{\partial(\rho\mathbf{u})}{\partial t} + \nabla \cdot (\rho\mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho\mathbf{f}}$$

where

$$\mathbf{u} \otimes \mathbf{u} = \begin{bmatrix} u^2 & uv & uw \\ uv & v^2 & vw \\ uw & vw & w^2 \end{bmatrix}.$$

This is the **conservative form of the momentum equations**.

Total Stress Tensor

Define the total stress tensor as

$$\boxed{\boldsymbol{\sigma} = -p\mathbf{I} + \boldsymbol{\tau}}$$

Then

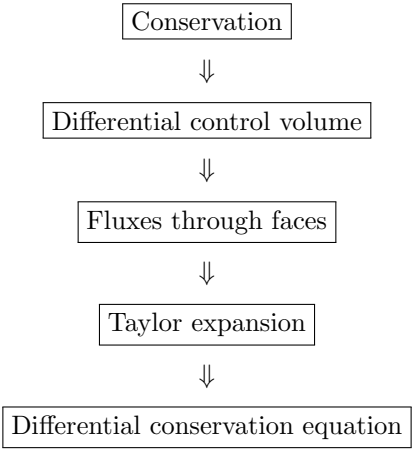
$$-\nabla p + \nabla \cdot \boldsymbol{\tau} = \nabla \cdot \boldsymbol{\sigma}.$$

Therefore,

$$\boxed{\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = \nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{f}}$$

From the Differential Equation to CFD

The derivation followed exactly the structure used by the finite-volume method:



Navier–Stokes: Conservation of Momentum

The momentum equation can be written as

$$\boxed{\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{f}}$$

where

- $\rho \mathbf{u}$ is the momentum per unit volume,
- p is the thermodynamic pressure,
- $\boldsymbol{\tau}$ is the viscous stress tensor,
- $\rho \mathbf{f}$ represents body forces.

At this stage, the equation is **not yet closed**.

Navier–Stokes: x-Momentum Equation

Let

$$\mathbf{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}.$$

The x-component of the momentum equation is

$$\boxed{\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + \rho f_x}$$

The viscous contribution is therefore

$$\boxed{(\nabla \cdot \boldsymbol{\tau})_x = \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z}}$$

We now need a **constitutive relation** for the stresses.

The Viscous Stress Tensor

For a general fluid,

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix}.$$

For a Newtonian fluid, the viscous stresses depend **linearly** on the velocity gradients.
The rate-of-deformation tensor is

$$\boxed{\mathbf{D} = \frac{1}{2}\left[\nabla \mathbf{u} + (\nabla \mathbf{u})^T\right]}$$

and therefore

$$\boxed{\boldsymbol{\tau} = 2\mu \mathbf{D} + \lambda(\nabla \cdot \mathbf{u})\mathbf{I}}$$

where

$$\mu = \text{dynamic viscosity}, \qquad \lambda = \text{second viscosity coefficient}.$$

Newtonian Constitutive Law: Components

Expanding $\mathbf{D}$ gives

$$\mathbf{D} = \begin{bmatrix} \dfrac{\partial u}{\partial x} & \dfrac{1}{2}\left(\dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x}\right) & \dfrac{1}{2}\left(\dfrac{\partial u}{\partial z} + \dfrac{\partial w}{\partial x}\right) \\ \dfrac{1}{2}\left(\dfrac{\partial v}{\partial x} + \dfrac{\partial u}{\partial y}\right) & \dfrac{\partial v}{\partial y} & \dfrac{1}{2}\left(\dfrac{\partial v}{\partial z} + \dfrac{\partial w}{\partial y}\right) \\ \dfrac{1}{2}\left(\dfrac{\partial w}{\partial x} + \dfrac{\partial u}{\partial z}\right) & \dfrac{1}{2}\left(\dfrac{\partial w}{\partial y} + \dfrac{\partial v}{\partial z}\right) & \dfrac{\partial w}{\partial z} \end{bmatrix}.$$

Therefore, for the x-momentum equation,

$$\boxed{\tau_{xx} = 2\mu\frac{\partial u}{\partial x} + \lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)}$$

$$\boxed{\tau_{xy} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)} \qquad \boxed{\tau_{xz} = \mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)}$$

Substituting the Newtonian Stresses

The viscous term in the x-momentum equation is

$$\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z}.$$

Substituting the Newtonian constitutive relations,

$$\begin{aligned}\nabla \cdot \boldsymbol{\tau}\big|_x = &\frac{\partial}{\partial x}\left[2\mu \frac{\partial u}{\partial x} + \lambda \nabla \cdot \mathbf{u}\right] \\ &+ \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] \\ &+ \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right].\end{aligned}$$

This is the point where the familiar second derivatives begin to appear.

Constant Viscosity

If μ and λ are constant,

$$\begin{aligned}\nabla \cdot \boldsymbol{\tau}\big|_x = 2\mu\frac{\partial^2 u}{\partial x^2} + \lambda\frac{\partial}{\partial x}\left(\nabla \cdot \mathbf{u}\right) \\ + \mu\frac{\partial^2 u}{\partial y^2} + \mu\frac{\partial^2 v}{\partial y\,\partial x} \\ + \mu\frac{\partial^2 u}{\partial z^2} + \mu\frac{\partial^2 w}{\partial z\,\partial x}.\end{aligned}$$

Collecting the terms involving u,

$$\boxed{\begin{aligned}\nabla \cdot \boldsymbol{\tau}\big|_x = \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \\ + (\mu+\lambda)\frac{\partial}{\partial x}\left(\nabla \cdot \mathbf{u}\right).\end{aligned}}$$

Incompressibility: The Simplification

For an incompressible flow,

$$\boxed{\nabla \cdot \mathbf{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0}$$

and therefore

$$\frac{\partial}{\partial x}\left(\nabla \cdot \mathbf{u}\right) = 0.$$

The viscous term reduces to

$$\boxed{\nabla \cdot \boldsymbol{\tau}\big|_x = \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)}$$

or, using the Laplacian,

$$\boxed{\nabla \cdot \boldsymbol{\tau}\big|_x = \mu \nabla^2 u.}$$

Incompressible Newtonian x-Momentum

Substituting the viscous term into the x-momentum equation,

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + \rho f_x.$$

Thus, the second derivatives of velocity are not introduced arbitrarily.

They arise from the divergence of the Newtonian viscous stress.

Returning to Vector Form

The same result applies to the y- and z-momentum equations.

For constant μ and incompressible flow,

$$\boxed{\nabla \cdot \boldsymbol{\tau} = \mu \nabla^2 \mathbf{u}}$$

and therefore

$$\boxed{\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{f}}$$

together with

$$\boxed{\nabla \cdot \mathbf{u} = 0.}$$

These are the incompressible Navier–Stokes equations.

Table of Contents

1. Outline
2. Continuum Mechanics and Fluid Description
3. Governing Equations
4. From Navier-Stokes to RANS
5. Solving Turbulent Flow
6. Discretization
7. Pressure-Velocity Coupling
8. Boundary Conditions
9. Geometry
10. Meshing

Navier-Stokes Equations: General Form

The governing equations of fluid motion are obtained from the fundamental conservation laws.

Mass conservation

$$\boxed{\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0} \tag{23}$$

Momentum conservation

$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + \rho f_x. \tag{24}$$

Similarly, the y- and z-momentum equations are obtained by applying the momentum conservation law in the other directions.

Navier–Stokes Equations: Momentum Components

x-momentum

$$\boxed{\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}+\rho f_x} \tag{25}$$

y-momentum

$$\boxed{\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{yx}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}+\rho f_y} \tag{26}$$

z-momentum

$$\boxed{\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{zx}}{\partial x}+\frac{\partial \tau_{zy}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z} \tag{27}$$

Energy Conservation

In some situations, we also have to conserve energy.

The conservation of total energy is

$$\boxed{\frac{\partial(\rho E)}{\partial t} + \nabla \cdot [(\rho E + p)\mathbf{u}] = \nabla \cdot (\boldsymbol{\tau} \cdot \mathbf{u}) - \nabla \cdot \mathbf{q} + \rho \mathbf{f} \cdot \mathbf{u}} \tag{28}$$

where

$$E = e + \frac{1}{2}\left(u^2 + v^2 + w^2\right)$$

is the total specific energy.

$$\boxed{\text{Mass} + \text{Momentum} + \text{Energy} \quad \longrightarrow \quad \text{Navier–Stokes system}}$$

For an incompressible Newtonian fluid with constant properties:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{29}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\nabla^2 u \tag{30}$$

where

$$\nabla^2 u = \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}.$$

Reynolds Decomposition

The instantaneous velocity and pressure are decomposed into a mean component and a fluctuating component:

$$u = U + u', \tag{31}$$
$$v = V + v', \tag{32}$$
$$w = W + w', \tag{33}$$
$$p = P + p'. \tag{34}$$

Here,

- U, V, W are the mean velocity components,
- u', v', w' are the turbulent fluctuations,
- P is the mean pressure,
- p' is the pressure fluctuation.

Reynolds Decomposition

By definition of the Reynolds average,

$$\overline{u'} = \overline{v'} = \overline{w'} = \overline{p'} = 0. \tag{35}$$

Therefore,

$$\overline{u} = \overline{U + u'}, \qquad \overline{v} = \overline{V + v'}, \qquad \overline{w} = \overline{W + w'}, \qquad \overline{p} = \overline{P + p'}.$$

$$\overline{u} = \overline{U} + \overline{u'}, \qquad \overline{v} = \overline{V} + \overline{v'}, \qquad \overline{w} = \overline{W} + \overline{w'}, \qquad \overline{p} = \overline{P} + \overline{p'}.$$

$$\overline{u} = U, \qquad \overline{v} = V, \qquad \overline{w} = W, \qquad \overline{p} = P.$$

Useful Reynolds-Averaging Rules

For a Reynolds average,

$$\overline{a+b} = \overline{a} + \overline{b}, \tag{36}$$

$$\overline{Ca} = C\overline{a}, \tag{37}$$

$$\overline{\frac{\partial a}{\partial x}} = \frac{\partial \overline{a}}{\partial x}, \tag{38}$$

$$\overline{a'} = 0. \tag{39}$$

However, in general,

$$\boxed{\overline{ab} \neq \overline{a}\,\overline{b}}$$

because the product of two fluctuating quantities has a non-zero average.
For example,

$$\overline{uv} = \overline{(U+u')(V+v')}.$$

Start with the x-Momentum Equation

The instantaneous x-momentum equation is

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\nabla^2 u. \tag{40}$$

Substitute

$$u = U + u', \qquad v = V + v', \qquad w = W + w', \qquad p = P + p'.$$

Thus,

$$\frac{\partial (U+u')}{\partial t} + (U+u')\frac{\partial (U+u')}{\partial x} \tag{41}$$

$$+ (V+v')\frac{\partial (U+u')}{\partial y} \tag{42}$$

$$+ (W+w')\frac{\partial (U+u')}{\partial z}$$

$$= -\frac{1}{\rho}\frac{\partial (P+p')}{\partial x} + \nu\nabla^2 (U+u'). \tag{43}$$

Expand the Convective Terms

Expanding the x-direction convective term:

$$(U+u')\frac{\partial(U+u')}{\partial x} = U\frac{\partial U}{\partial x} + U\frac{\partial u'}{\partial x} \tag{44}$$

$$+ u'\frac{\partial U}{\partial x} + u'\frac{\partial u'}{\partial x}. \tag{45}$$

Similarly,

$$(V+v')\frac{\partial(U+u')}{\partial y} = V\frac{\partial U}{\partial y} + V\frac{\partial u'}{\partial y} \tag{46}$$

$$+ v'\frac{\partial U}{\partial y} + v'\frac{\partial u'}{\partial y}, \tag{47}$$

$$(W+w')\frac{\partial(U+u')}{\partial z} = W\frac{\partial U}{\partial z} + W\frac{\partial u'}{\partial z} \tag{48}$$

$$+ w'\frac{\partial U}{\partial z} + w'\frac{\partial u'}{\partial z}. \tag{49}$$

Take the Reynolds Average

Now take the Reynolds average of the entire equation.
Terms containing only one fluctuation vanish:

$$\overline{u'} = \overline{v'} = \overline{w'} = \overline{p'} = 0.$$

For example,

$$\overline{U\frac{\partial u'}{\partial x}} = U\frac{\partial \overline{u'}}{\partial x} = 0, \quad \text{and} \quad \overline{u'\frac{\partial U}{\partial x}} = \frac{\partial U}{\partial x}\overline{u'} = 0.$$

The important terms are the products of fluctuations:

$$\overline{u'\frac{\partial u'}{\partial x}} = \frac{\partial \overline{u'^2}}{\partial x}, \tag{50}$$

$$\overline{v'\frac{\partial u'}{\partial y}} = \frac{\partial \overline{u'v'}}{\partial y}, \tag{51}$$

$$\overline{w'\frac{\partial u'}{\partial z}} = \frac{\partial \overline{u'w'}}{\partial z}. \tag{52}$$

Averaged x-Momentum Equation

After averaging,

$$\begin{aligned}\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} + W\frac{\partial U}{\partial z} \\ + \frac{\partial \overline{u'^2}}{\partial x} + \frac{\partial \overline{u'v'}}{\partial y} + \frac{\partial \overline{u'w'}}{\partial z} \\ = -\frac{1}{\rho}\frac{\partial P}{\partial x} + \nu\nabla^2 U.\end{aligned} \tag{53}$$

Rearranging the fluctuation terms,

$$\boxed{\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} + W\frac{\partial U}{\partial z} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + \nu\nabla^2 U - \frac{\partial \overline{u'^2}}{\partial x} - \frac{\partial \overline{u'v'}}{\partial y} - \frac{\partial \overline{u'w'}}{\partial z}} \tag{54}$$

This is the Reynolds-averaged x-momentum equation.

Where Did the Reynolds Stresses Come From?

The new terms are

$$\overline{u'^2}, \qquad \overline{u'v'}, \qquad \overline{u'w'}.$$

They can be written as

$$-\frac{1}{\rho}\frac{\partial(\rho\overline{u'^2})}{\partial x}, \qquad -\frac{1}{\rho}\frac{\partial(\rho\overline{u'v'})}{\partial y}, \qquad -\frac{1}{\rho}\frac{\partial(\rho\overline{u'w'})}{\partial z}.$$

These terms represent the transport of mean momentum by turbulent velocity fluctuations. They are called the **Reynolds stresses**. For example,

$$\boxed{\tau^R_{xx} = -\rho\overline{u'^2}}$$

$$\boxed{\tau^R_{xy} = -\rho\overline{u'v'}}$$

$$\boxed{\tau^R_{xz} = -\rho\overline{u'w'}}$$

General Reynolds Stress Tensor

The complete Reynolds stress tensor is

$$\boxed{\tau_{ij}^R = -\rho\,\overline{u_i' u_j'}}$$

or

$$\boldsymbol{\tau}^R = -\rho \begin{pmatrix} \overline{u'^2} & \overline{u'v'} & \overline{u'w'} \\ \overline{u'v'} & \overline{v'^2} & \overline{v'w'} \\ \overline{u'w'} & \overline{v'w'} & \overline{w'^2} \end{pmatrix}.$$

Therefore the RANS momentum equations can be written compactly as

$$\boxed{\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \nabla^2 U_i - \frac{\partial \overline{u_i' u_j'}}{\partial x_j}}$$

Now Apply the Same Procedure to Mass Conservation

The instantaneous incompressible continuity equation is

$$\boxed{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0}$$

Substitute the Reynolds decomposition:

$$\frac{\partial (U + u')}{\partial x} + \frac{\partial (V + v')}{\partial y} + \frac{\partial (W + w')}{\partial z} = 0. \tag{55}$$

Separate mean and fluctuating components:

$$\left(\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} \right) + \left(\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} \right) = 0. \tag{56}$$

Average the Continuity Equation

Taking the Reynolds average gives

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} + \overline{\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z}} = 0. \tag{57}$$

Since

$$\overline{u'} = \overline{v'} = \overline{w'} = 0,$$

we obtain

$$\boxed{\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} = 0}$$

Therefore, the Reynolds-averaged velocity field is also divergence-free.

The transformation can therefore be summarized as

$$\boxed{\text{Instantaneous flow} \quad \longrightarrow \quad \text{Reynolds decomposition} \quad \longrightarrow \quad \text{Reynolds averaging}}$$

which leads to

$$\boxed{\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \nabla^2 U_i - \frac{\partial \overline{u_i' u_j'}}{\partial x_j}} \quad \text{with} \quad \boxed{\frac{\partial U_i}{\partial x_i} = 0.}$$

The price of averaging the nonlinear Navier–Stokes equations is the appearance of the unknown Reynolds stresses:

$$\boxed{-\overline{u_i' u_j'}}$$

which creates the **closure problem**.

Table of Contents

1 Outline

2 Continuum Mechanics and Fluid Description

3 Governing Equations

4 From Navier-Stokes to RANS

5 Solving Turbulent Flow

6 Discretization

7 Pressure-Velocity Coupling

8 Boundary Conditions

9 Geometry

10 Meshing

How Can We Solve Turbulent Flow?

After Reynolds averaging, the momentum equations contain the unknown Reynolds stresses

$$-\rho\overline{u_i' u_j'}. \tag{58}$$

These additional unknowns require a turbulence treatment.

Three major approaches are:

1. **DNS – Direct Numerical Simulation**
2. **LES – Large Eddy Simulation**
3. **RANS – Reynolds-Averaged Navier–Stokes**

Method	**What is resolved?**
DNS	All turbulent scales
LES	Large scales; small scales are modeled
RANS	Mean flow; turbulence is modeled

DNS, LES and RANS

DNS

- Resolves all spatial and temporal turbulent scales.
- No turbulence model is required.
- Computational cost increases dramatically with Reynolds number.

LES

- Resolves the large, energy-containing turbulent structures.
- Models the smaller scales using a subgrid-scale model.
- More expensive than RANS, but provides much more information about unsteady turbulent structures.

RANS

- Solves equations for the mean flow.
- All turbulent fluctuations are represented through modeled Reynolds stresses.
- Computationally much cheaper than DNS and LES.

For engineering CFD, RANS is often the practical choice. (59)

Why Does RANS Need a Turbulence Model?

The Reynolds-averaged momentum equation is

$$\boxed{\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \nabla^2 U_i - \frac{\partial \overline{u_i' u_j'}}{\partial x_j}} \tag{60}$$

The Reynolds stresses

$$\overline{u_i' u_j'}$$

are additional unknowns.

The original equations have therefore produced **more unknowns than equations**.

This is the turbulence closure problem.

We need a physical model for the Reynolds stresses.

Boussinesq Hypothesis

A common approach is to relate the Reynolds stresses to the mean rate of strain.
The **Boussinesq hypothesis** assumes

$$\boxed{-\overline{u_i' u_j'} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij}} \tag{61}$$

where

$$S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \tag{62}$$

is the mean strain-rate tensor. The turbulent kinetic energy is

$$\boxed{k = \frac{1}{2}\overline{u_i' u_i'} = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)} \tag{63}$$

and ν_t is the **turbulent (eddy) viscosity**.

What Does the Boussinesq Hypothesis Mean?

The molecular viscous stresses are related to the mean strain rate through the viscosity ν.

Boussinesq assumes that turbulent momentum transport can be represented in a similar way:

$$\boxed{\text{Turbulent momentum transport} \sim \text{turbulent viscosity} \times \text{mean strain rate}} \tag{64}$$

Thus,

$$\boxed{\nu \quad \longrightarrow \quad \nu_t} \tag{65}$$

where

$$\nu = \text{molecular kinematic viscosity}$$

and

$$\nu_t = \text{turbulent kinematic viscosity.}$$

The key problem has therefore changed from

$$\overline{u_i' u_j'} \quad \text{to} \quad \nu_t.$$

Boussinesq Hypothesis: x-Momentum

For the x-momentum equation,

$$-\overline{u'^2} = 2\nu_t \frac{\partial U}{\partial x} - \frac{2}{3}k \tag{66}$$

and

$$-\overline{u'v'} = \nu_t \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right). \tag{67}$$

Similarly,

$$-\overline{u'w'} = \nu_t \left(\frac{\partial U}{\partial z} + \frac{\partial W}{\partial x} \right). \tag{68}$$

Therefore, the Reynolds stresses can be expressed using mean velocity gradients and ν_t.

Model Reynolds stresses $\rightarrow$ determine ν_t

The Remaining Question: How Do We Find ν_t?

The Boussinesq hypothesis gives

$$-\overline{u_i' u_j'} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij}. \tag{69}$$

But ν_t is still unknown.

Therefore, we need another model that provides the turbulent viscosity.

Two common approaches are:

$$\boxed{k-\varepsilon} \qquad \text{and} \qquad \boxed{k-\omega} \tag{70}$$

Both introduce additional transport equations to determine the turbulence quantities required to calculate ν_t.

$$\boxed{\text{Reynolds stresses} \rightarrow \nu_t \rightarrow \text{Turbulence Model}}$$

The $k - \varepsilon$ Model

The standard $k - \varepsilon$ model solves two additional transport equations:

$$\boxed{\text{Transport equation for } k} \quad \text{and} \quad \boxed{\text{Transport equation for } \varepsilon} \tag{71}$$

The turbulent viscosity is then modeled as

$$\boxed{\nu_t = C_\mu \frac{k^2}{\varepsilon}} \tag{72}$$

where

$$k = \text{turbulent kinetic energy}$$

and

$$\varepsilon = \text{turbulent kinetic energy dissipation rate.}$$

Thus,

$$\boxed{k, \varepsilon \quad \longrightarrow \quad \nu_t \quad \longrightarrow \quad -\overline{u_i' u_j'}} \tag{73}$$

The $k-\omega$ Model

The $k-\omega$ model also solves two additional transport equations:

$$\boxed{\text{Transport equation for } k} \quad \text{and} \quad \boxed{\text{Transport equation for } \omega} \tag{74}$$

The turbulent viscosity is related to

$$\boxed{\nu_t \sim \frac{k}{\omega}} \tag{75}$$

where

$$k = \text{turbulent kinetic energy}$$

and

$$\omega = \text{specific dissipation rate.}$$

Therefore,

$$\boxed{k, \omega \quad \longrightarrow \quad \nu_t \quad \longrightarrow \quad -\overline{u_i' u_j'}} \tag{76}$$

From Navier–Stokes to a Turbulence Model

The complete chain is

$$
\begin{aligned}
\text{Navier–Stokes} &\longrightarrow \text{Reynolds decomposition} \\
&\longrightarrow \text{RANS equations} \\
&\longrightarrow \underbrace{-\overline{u_i' u_j'}}_{\text{unknown Reynolds stresses}} \\
&\longrightarrow \text{Boussinesq hypothesis} \\
&\longrightarrow \nu_t \\
&\longrightarrow \begin{cases} k-\varepsilon \\ k-\omega \\ \text{etc.} \end{cases}
\end{aligned}
$$

Why Are Walls Special?

At a solid wall, the no-slip condition gives

$$\boxed{\mathbf{U}_{wall} = 0} \tag{77}$$

while away from the wall the velocity approaches the outer-flow value.

Therefore, the velocity changes rapidly close to the wall:

$$\boxed{\text{Large velocity gradients} \quad \Rightarrow \quad \text{strong viscous effects}} \tag{78}$$

The near-wall region contains important physical information about

- wall shear stress,
- momentum transfer,
- boundary-layer development,
- separation,
- heat and mass transfer.

$$\boxed{\text{The near-wall flow must therefore be treated carefully.}}$$

The Turbulent Boundary Layer Near a Wall

The flow close to a wall can be divided into different regions.

Region	Dominant behaviour
Viscous sublayer	Molecular viscosity dominates
Buffer layer	Viscous and turbulent effects are important
Logarithmic region	Turbulent transport dominates

Very close to the wall,

$$\boxed{u^+ \approx y^+} \tag{79}$$

while in the logarithmic region,

$$\boxed{u^+ = \frac{1}{\kappa}\ln(y^+) + B} \tag{80}$$

where

$$u^+ = \frac{U}{u_\tau}, \qquad y^+ = \frac{u_\tau y}{\nu}.$$

The location of the first computational point therefore matters.

What Is y^+?

The dimensionless wall distance is defined as

$$\boxed{y^+ = \frac{u_\tau y}{\nu}} \tag{81}$$

where

$$\boxed{u_\tau = \sqrt{\frac{\tau_w}{\rho}}} \tag{82}$$

is the friction velocity.
Here,

$$\tau_w = \text{wall shear stress}, \qquad \rho = \text{density}, \qquad y = \text{distance from the wall}.$$

Therefore, y^+ measures the distance from the wall relative to the viscous length scale.

$$\boxed{y^+ \text{ is not a mesh-quality metric}}$$

It is a dimensionless measure of the near-wall resolution.

Two Ways to Treat the Near-Wall Region

There are two broad strategies for dealing with the flow close to a wall.

1. Resolve the near-wall region

- The viscous sublayer is explicitly resolved.
- The first computational point is placed very close to the wall.
- Typically requires

$$y^+ \approx 1.$$

- Requires a fine near-wall mesh.

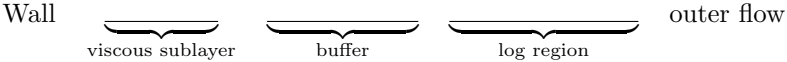
2. Use wall functions

- The viscous sublayer is not fully resolved.
- The behaviour between the wall and the first computational point is modeled.
- The first computational point is placed farther from the wall.
- Typically requires a larger y^+.

Wall treatment determines the required near-wall resolution.

Wall-Resolved Approach

In a wall-resolved simulation, the mesh extends sufficiently close to the wall to resolve the viscous region.



The first computational point is typically located at

$$\boxed{y^+ \approx 1} \tag{83}$$

so that the strong velocity gradients near the wall are resolved directly.

This requires:

- small first-cell height,
- several cells across the boundary layer,
- gradual growth of the near-wall layers,
- increased computational cost.

Wall Functions

Wall functions avoid resolving the complete viscous sublayer.

Instead, the velocity between the wall and the first computational point is represented using an empirical or semi-empirical relationship. In the logarithmic region,

$$\boxed{u^+ = \frac{1}{\kappa}\ln(y^+) + B} \tag{84}$$

can be used to relate the velocity at the first computational point to the wall shear stress.

Therefore:

$$\boxed{\text{Wall} \rightarrow \text{wall function} \rightarrow \text{first computational point}} \tag{85}$$

The near-wall behaviour is modeled rather than fully resolved.

Wall Treatment and Turbulence Models

The appropriate near-wall treatment depends on the turbulence model and the way the near-wall region is represented.

Approach	**Near-wall resolution**
Wall-resolved	Typically $y^+ \approx 1$
Wall functions	First point located farther from wall

For example, turbulence models may be used with different near-wall treatments depending on the formulation and solver.

Turbulence model	+	wall treatment	→	near-wall requirements

Therefore, y^+ should not be selected independently of the turbulence model and wall treatment.

From Wall Treatment to the Mesh

The wall treatment determines how close the first computational point must be to the wall.

The chain is

$$\boxed{\text{Turbulence model}} \rightarrow \boxed{\text{Wall treatment}} \rightarrow \boxed{\text{Target } y^+} \rightarrow \boxed{\text{First-cell height}}$$

Thus, mesh design is not simply:

"make the cells smaller".

Instead,

$$\boxed{\text{The mesh must be designed to resolve the physics that the model requires.}}$$

$k - \varepsilon$ and the Near-Wall Region

The standard $k - \varepsilon$ model is commonly used together with **wall functions**.

The near-wall region is therefore not normally resolved all the way to the wall.
Instead, the wall function provides a relationship between the wall and the first computational point:

$$\boxed{\text{Wall} \quad \longrightarrow \quad \text{Wall Function} \quad \longrightarrow \quad \text{First Computational Point}} \tag{86}$$

The first computational point is therefore placed outside the viscous sublayer, in a region where the wall-function formulation is applicable.

$$\boxed{\text{Standard } k - \varepsilon \quad \Rightarrow \quad \text{typically wall-function based}}$$

This allows a relatively coarse near-wall mesh compared with a fully wall-resolved approach.

$k-\omega$ and the Near-Wall Region

The $k-\omega$ formulation is designed to describe the turbulence behaviour much closer to the wall.

The model can be integrated towards the wall, allowing the near-wall region to be resolved rather than completely replaced by a wall function.

This makes the first-cell position particularly important:

$$\boxed{\text{Wall} \quad \longrightarrow \quad \text{Near-Wall Cells} \quad \longrightarrow \quad \text{Outer Flow}} \tag{87}$$

For a wall-resolved treatment, the first computational point is typically placed at

$$\boxed{y^+ \approx 1} \tag{88}$$

Thus,

$$\boxed{k-\omega \quad \Rightarrow \quad \text{can resolve the near-wall region}} \tag{89}$$

and the mesh must be designed accordingly.

Table of Contents

1. Outline
2. Continuum Mechanics and Fluid Description
3. Governing Equations
4. From Navier-Stokes to RANS
5. Solving Turbulent Flow
6. Discretization
7. Pressure-Velocity Coupling
8. Boundary Conditions
9. Geometry
10. Meshing

Discretization

The Navier–Stokes equations are differential equations defined over a continuous domain. To solve them numerically, the differential equations must be transformed into a finite set of algebraic equations.

$$\boxed{\text{Differential equations} \quad \longrightarrow \quad \text{Algebraic equations}} \tag{90}$$

Three classical approaches are:

1. **Finite Difference Method (FDM)**
2. **Finite Volume Method (FVM)**
3. **Finite Element Method (FEM)**

Other approaches include spectral methods and discontinuous Galerkin methods.

For the present course, we will focus on the **Finite Volume Method**, which is the approach commonly used in engineering CFD codes such as Fluent.

Finite Difference Method

The Finite Difference Method approximates derivatives using values of the variables at discrete points. For example, the first derivative can be approximated by

$$\boxed{\left.\frac{\partial \phi}{\partial x}\right|_i \approx \frac{\phi_{i+1} - \phi_{i-1}}{2\Delta x}} \tag{91}$$

and the second derivative by

$$\boxed{\left.\frac{\partial^2 \phi}{\partial x^2}\right|_i \approx \frac{\phi_{i+1} - 2\phi_i + \phi_{i-1}}{\Delta x^2}} \tag{92}$$

The differential equation is therefore replaced by an algebraic equation involving neighboring points.

$$\boxed{\text{Derivatives} \quad \rightarrow \quad \text{Differences between neighboring points}} \tag{93}$$

FDM is particularly convenient on structured grids.

Example: 2-D Transient Heat Conduction

Consider transient heat conduction in a two-dimensional solid:

$$\boxed{\frac{\partial T}{\partial t} = \alpha \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)} \tag{94}$$

where

$$\alpha = \frac{k}{\rho c_p} \tag{95}$$

is the thermal diffusivity.

We discretize the domain using a Cartesian grid:

$$(x_i, y_j)$$

and denote the temperature at a grid point by

$$T_{i,j}^n = T(x_i, y_j, t^n).$$

Example: Discretizing the Derivatives

For the time derivative, use a forward difference:

$$\boxed{\left.\frac{\partial T}{\partial t}\right|_{i,j}^{n} \approx \frac{T_{i,j}^{n+1} - T_{i,j}^{n}}{\Delta t}} \tag{96}$$

For the second derivative in the x direction:

$$\boxed{\left.\frac{\partial^2 T}{\partial x^2}\right|_{i,j}^{n} \approx \frac{T_{i+1,j}^{n} - 2T_{i,j}^{n} + T_{i-1,j}^{n}}{\Delta x^2}} \tag{97}$$

Similarly, in the y direction:

$$\boxed{\left.\frac{\partial^2 T}{\partial y^2}\right|_{i,j}^{n} \approx \frac{T_{i,j+1}^{n} - 2T_{i,j}^{n} + T_{i,j-1}^{n}}{\Delta y^2}} \tag{98}$$

Example: From PDE to Algebraic Equation

Starting from

$$\frac{\partial T}{\partial t} = \alpha \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) \tag{99}$$

substituting the finite-difference approximations gives

$$\frac{T_{i,j}^{n+1} - T_{i,j}^{n}}{\Delta t} = \alpha \left[\frac{T_{i+1,j}^{n} - 2T_{i,j}^{n} + T_{i-1,j}^{n}}{\Delta x^2} + \frac{T_{i,j+1}^{n} - 2T_{i,j}^{n} + T_{i,j-1}^{n}}{\Delta y^2} \right] \tag{100}$$

Therefore,

$$\boxed{\begin{aligned} T_{i,j}^{n+1} = T_{i,j}^{n} &+ \Omega_x \left(T_{i+1,j}^{n} - 2T_{i,j}^{n} + T_{i-1,j}^{n} \right) \\ &+ \Omega_y \left(T_{i,j+1}^{n} - 2T_{i,j}^{n} + T_{i,j-1}^{n} \right) \end{aligned}} \tag{101}$$

where

$$\Omega_x = \frac{\alpha \Delta t}{\Delta x^2}, \qquad \Omega_y = \frac{\alpha \Delta t}{\Delta y^2}.$$

Example: The 5-Point Stencil

For a uniform grid,

$$\Delta x = \Delta y = \Delta$$

and therefore

$$\Omega_x = \Omega_y = \Omega = \frac{\alpha \Delta t}{\Delta^2}.$$

The update equation becomes

$$\boxed{T_{i,j}^{n+1} = T_{i,j}^{n} + \Omega \left(T_{i+1,j}^{n} + T_{i-1,j}^{n} + T_{i,j+1}^{n} + T_{i,j-1}^{n} - 4T_{i,j}^{n} \right)} \tag{102}$$

The temperature at (i, j) therefore depends on its four nearest neighbors.

East	West	North	South

Finite Volume Method

The Finite Volume Method starts from the **conservation equations**.
The governing equation is integrated over a control volume:

$$\boxed{\int_V \frac{\partial(\rho\phi)}{\partial t}\,dV + \int_V \nabla\cdot(\rho\mathbf{u}\phi)\,dV = \int_V \nabla\cdot(\Gamma\nabla\phi)\,dV + \int_V S_\phi\,dV} \tag{103}$$

Using the divergence theorem,

$$\boxed{\int_V \nabla\cdot\mathbf{F}\,dV = \oint_A \mathbf{F}\cdot\mathbf{n}\,dA} \tag{104}$$

the volume integrals become surface fluxes.
Therefore,

$$\boxed{\text{Accumulation} + \text{Net flux through faces} = \text{Sources}} \tag{105}$$

This produces one algebraic equation for each control volume.

Finite Element Method

The Finite Element Method divides the domain into a set of finite elements. Instead of satisfying the governing equations exactly at every point, a **weak formulation** is constructed.

$$\boxed{\text{Strong form} \quad \longrightarrow \quad \text{Weak form} \quad \longrightarrow \quad \text{Finite-element equations}} \tag{106}$$

The solution is approximated using shape functions:

$$\boxed{\phi(\mathbf{x}) \approx \sum_{i=1}^{N} N_i(\mathbf{x})\phi_i} \tag{107}$$

where

$$N_i = \text{shape functions}$$

and

$$\phi_i = \text{unknown value at a node.}$$

FEM is widely used in structural mechanics and is also used for fluid-flow problems, particularly in multiphysics applications.

From the PDE to the Finite Volume Equation

Consider a generic transport equation:

$$\frac{\partial(\rho\phi)}{\partial t} + \nabla \cdot (\rho \mathbf{u}\phi) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi. \tag{108}$$

Integrating over a control volume gives

$$\int_V \frac{\partial(\rho\phi)}{\partial t}\, dV + \oint_A \rho \mathbf{u}\phi \cdot \mathbf{n}\, dA = \oint_A \Gamma \nabla\phi \cdot \mathbf{n}\, dA + \int_V S_\phi\, dV. \tag{109}$$

After approximation,

$$\boxed{a_P \phi_P = \sum_{nb} a_{nb}\phi_{nb} + b} \tag{110}$$

where P is the current cell and nb denotes its neighboring cells.

The PDE has now become a system of algebraic equations.

Finite Volume Discretisation of Continuity

The continuity equation is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0. \tag{111}$$

Integrating over a control volume V_P,

$$\int_{V_P} \frac{\partial \rho}{\partial t}\, dV + \int_{V_P} \nabla \cdot (\rho \mathbf{u})\, dV = 0. \tag{112}$$

Using the divergence theorem,

$$\boxed{\frac{d}{dt} \int_{V_P} \rho\, dV + \oint_A \rho \mathbf{u} \cdot \mathbf{n}\, dA = 0} \tag{113}$$

For a fixed control volume, the first term represents the **rate of mass accumulation**, while the surface integral represents the **net mass flux**.

Continuity: Discretisation at the Faces

Consider a two-dimensional cell with four faces:

$$e,\ w,\ n,\ s$$

The integral continuity equation becomes

$$\frac{d}{dt}(\rho_P V_P) + [(\rho u A)_e - (\rho u A)_w + (\rho v A)_n - (\rho v A)_s] = 0. \tag{114}$$

Define the mass flux through a face as

$$\dot{m}_f = \rho_f (\mathbf{u}_f \cdot \mathbf{n}_f) A_f. \tag{115}$$

Therefore,

$$\boxed{\frac{d}{dt}(\rho_P V_P) + \dot{m}_e - \dot{m}_w + \dot{m}_n - \dot{m}_s = 0} \tag{116}$$

For steady flow,

$$\boxed{\dot{m}_e - \dot{m}_w + \dot{m}_n - \dot{m}_s = 0} \tag{117}$$

Continuity: The Discrete Conservation Equation

For a steady two-dimensional flow,

$$\dot{m}_e - \dot{m}_w + \dot{m}_n - \dot{m}_s = 0. \tag{118}$$

Using

$$\dot{m}_f = \rho_f u_{n,f} A_f, \tag{119}$$

we obtain

$$\boxed{F_e - F_w + F_n - F_s = 0} \tag{120}$$

where

$$F_f = \rho_f u_{n,f} A_f \tag{121}$$

is the mass flux through face f.

Important:

The finite volume method enforces mass conservation **cell by cell**.

Finite Volume Discretisation of x-Momentum

The x-momentum equation is

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho \mathbf{u} u) = -\frac{\partial p}{\partial x} + \nabla \cdot (\mu \nabla u) + S_u. \tag{122}$$

Integrating over a control volume,

$$\int_V \frac{\partial(\rho u)}{\partial t}\, dV + \int_V \nabla \cdot (\rho \mathbf{u} u)\, dV = -\int_V \frac{\partial p}{\partial x}\, dV + \int_V \nabla \cdot (\mu \nabla u)\, dV + \int_V S_u\, dV. \tag{123}$$

Applying the divergence theorem,

$$\iiint_V (\nabla \cdot \mathbf{F})\, \mathrm{d}V = \oiint_S (\mathbf{F} \cdot \hat{\mathbf{n}})\, \mathrm{d}S$$

$$\boxed{\frac{d}{dt}\int_V \rho u\, dV + \oint_A \rho u (\mathbf{u} \cdot \mathbf{n})\, dA = -\int_V \frac{\partial p}{\partial x}\, dV + \oint_A \mu \nabla u \cdot \mathbf{n}\, dA + \int_V S_u\, dV} \tag{124}$$

X-Momentum: Face Fluxes

For a two-dimensional cell,

$$\frac{d}{dt}(\rho_P u_P V_P) + [(\rho u u A)_e - (\rho u u A)_w + (\rho v u A)_n - (\rho v u A)_s] \tag{125}$$

$$= -(p_e A_e - p_w A_w) + \left[\mu A \frac{\partial u}{\partial n}\right]_e - \left[\mu A \frac{\partial u}{\partial n}\right]_w \tag{126}$$

$$+ \left[\mu A \frac{\partial u}{\partial n}\right]_n - \left[\mu A \frac{\partial u}{\partial n}\right]_s + S_u V_P. \tag{127}$$

The equation contains three physical contributions:

- **Convection** of x-momentum
- **Pressure forces**
- **Diffusion** of x-momentum

X-Momentum: Diffusive Flux

Consider the east face.
The diffusive flux is approximated using a central difference:

$$\left(\mu A \frac{\partial u}{\partial n}\right)_e \approx \mu_e A_e \frac{u_E - u_P}{\delta x_{PE}}. \tag{128}$$

Similarly, at the west face,

$$\left(\mu A \frac{\partial u}{\partial n}\right)_w \approx \mu_w A_w \frac{u_P - u_W}{\delta x_{WP}}. \tag{129}$$

Therefore,

$$D_e(u_E - u_P) - D_w(u_P - u_W) \tag{130}$$

where

$$D_e = \frac{\mu_e A_e}{\delta x_{PE}}, \qquad D_w = \frac{\mu_w A_w}{\delta x_{WP}}. \tag{131}$$

The same procedure is applied to the n and s faces.

X-Momentum: Convective Flux

Consider the east face. The convective flux of x-momentum is

$$(\rho u u A)_e = (\rho u A)_e \, u_e. \tag{132}$$

Define the mass flux through the east face as

$$F_e = \rho_e u_e A_e. \tag{133}$$

Therefore,

$$(\rho u u A)_e = F_e u_e. \tag{134}$$

Similarly, at the west face,

$$(\rho u u A)_w = F_w u_w. \tag{135}$$

The net convective contribution in the x direction is therefore

$$F_e u_e - F_w u_w. \tag{136}$$

The same procedure is applied to the n and s faces.

X-Momentum: Upwind Approximation

The face velocity u_f is not known directly.
For the first-order upwind scheme, the face value is taken from the upstream cell.
At the east face,

$$u_e = \begin{cases} u_P, & F_e > 0, \\ u_E, & F_e < 0. \end{cases} \tag{137}$$

At the west face,

$$u_w = \begin{cases} u_W, & F_w > 0, \\ u_P, & F_w < 0. \end{cases} \tag{138}$$

Therefore, the convective contribution becomes

$$\boxed{F_e u_e - F_w u_w} \tag{139}$$

where the face values are determined by the flow direction.

X-Momentum: Convective Fluxes on All Faces

The four convective fluxes of x-momentum are

$$F_e u_e, \qquad F_w u_w, \qquad F_n u_n, \qquad F_s u_s. \tag{140}$$

Hence, the net convective contribution is

$$\boxed{F_e u_e - F_w u_w + F_n u_n - F_s u_s} \tag{141}$$

where

$$F_f = \rho_f (\mathbf{u}_f \cdot \mathbf{n}_f) A_f. \tag{142}$$

Using the upwind approximation, each u_f is obtained from the upstream cell.

Convection = mass flux $\times$ transported quantity

X-Momentum: Collecting the Discretised Terms

For a steady two-dimensional control volume, the discretised x-momentum equation is

$$F_e u_e - F_w u_w + F_n u_n - F_s u_s = -\left(p_e A_e - p_w A_w\right) \tag{143}$$

$$+D_e(u_E - u_P) - D_w(u_P - u_W) + D_n(u_N - u_P) - D_s(u_P - u_S) + S_u V_P. \tag{144}$$

where

$$D_f = \frac{\mu_f A_f}{\delta_f} \tag{145}$$

and

$$F_f = \rho_f(\mathbf{u}_f \cdot \mathbf{n}_f)A_f. \tag{146}$$

We now substitute the upwind expressions for u_e, u_w, u_n, u_s.

X-Momentum: Substituting the Upwind Fluxes

For first-order upwind,

$$u_e = \begin{cases} u_P, & F_e > 0, \\ u_E, & F_e < 0, \end{cases} \qquad u_w = \begin{cases} u_W, & F_w > 0, \\ u_P, & F_w < 0. \end{cases} \tag{147}$$

Similarly,

$$u_n = \begin{cases} u_P, & F_n > 0, \\ u_N, & F_n < 0, \end{cases} \qquad u_s = \begin{cases} u_S, & F_s > 0, \\ u_P, & F_s < 0. \end{cases} \tag{148}$$

The convective terms can therefore be written in terms of the neighboring cell values.
Using

$$\max(F, 0) = \begin{cases} F, & F > 0, \\ 0, & F < 0, \end{cases} \tag{149}$$

the resulting coefficients can be written compactly.

X-Momentum: Neighbor Coefficients

After substituting the upwind expressions and collecting the coefficients of the neighboring velocities,

$$a_E u_E + a_W u_W + a_N u_N + a_S u_S \tag{150}$$

are obtained, with

$$\boxed{a_E = D_e + \max(-F_e, 0)} \tag{151}$$

$$\boxed{a_W = D_w + \max(F_w, 0)} \tag{152}$$

$$\boxed{a_N = D_n + \max(-F_n, 0)} \tag{153}$$

$$\boxed{a_S = D_s + \max(F_s, 0)} \tag{154}$$

Each coefficient contains two contributions:

$$\boxed{a_{nb} = \underbrace{D_{nb}}_{\text{diffusion}} + \underbrace{\text{upwind convection}}_{\text{mass transport}}} \tag{155}$$

X-Momentum: The Central Coefficient

After collecting all terms containing u_P, the central coefficient is

$$\boxed{a_P = a_E + a_W + a_N + a_S + (F_e - F_w + F_n - F_s)} \tag{156}$$

For a mass-conserving control volume,

$$F_e - F_w + F_n - F_s = 0. \tag{157}$$

Therefore,

$$\boxed{a_P = a_E + a_W + a_N + a_S} \tag{158}$$

for the steady, source-free coefficient form.

The diagonal coefficient is therefore linked directly to the neighboring coefficients through conservation.

X-Momentum: The Central Coefficient

After collecting all terms containing u_P, the central coefficient is

$$\boxed{a_P = a_E + a_W + a_N + a_S + (F_e - F_w + F_n - F_s)} \tag{159}$$

For a mass-conserving control volume,

$$F_e - F_w + F_n - F_s = 0. \tag{160}$$

Therefore,

$$\boxed{a_P = a_E + a_W + a_N + a_S} \tag{161}$$

for the steady, source-free coefficient form.

The diagonal coefficient is therefore linked directly to the neighboring coefficients through conservation.

X-Momentum: Pressure and Source Terms

The pressure contribution is approximated as

$$-\left(p_e A_e - p_w A_w\right). \tag{162}$$

For a Cartesian control volume with constant face area,

$$-\left(p_e - p_w\right) A. \tag{163}$$

The volumetric source contributes

$$S_u V_P. \tag{164}$$

Therefore, the non-neighbor contributions can be collected into a single right-hand-side term,

$$\boxed{b_u = (p_w - p_e)A + S_u V_P} \tag{165}$$

with additional contributions possible from boundary conditions and source linearisation.

What Did We Just Do?

Starting from the differential equation,

$$\boxed{\text{PDE}} \tag{166}$$

we performed

$$\boxed{\text{PDE} \longrightarrow \text{Control-volume integral} \longrightarrow \text{Face fluxes}} \tag{167}$$

$$\boxed{\text{Face fluxes} \longrightarrow \text{Discretisation} \longrightarrow \text{Algebraic equation}} \tag{168}$$

giving

$$\boxed{a_P u_P = \sum_{nb} a_{nb} u_{nb} + b_u} \tag{169}$$

One control volume → one algebraic equation

Temporal Discretisation

The transient term in the x-momentum equation is

$$\frac{\partial(\rho u)}{\partial t}. \tag{170}$$

Integrating over the control volume,

$$\int_V \frac{\partial(\rho u)}{\partial t}\, dV = \frac{d}{dt}(\rho_P u_P V_P). \tag{171}$$

For a constant-density flow and a fixed control volume,

$$\frac{d}{dt}(\rho_P u_P V_P) = \rho_P V_P \frac{\partial u_P}{\partial t}. \tag{172}$$

We now need to approximate the time derivative.

Temporal Discretisation: First-Order Euler

Let

$$u_P^n \tag{173}$$

be the velocity at the previous time level and

$$u_P^{n+1} \tag{174}$$

the velocity at the new time level.
Using a first-order backward difference,

$$\boxed{\frac{\partial u_P}{\partial t} \approx \frac{u_P^{n+1} - u_P^n}{\Delta t}} \tag{175}$$

Therefore,

$$\boxed{\frac{d}{dt}(\rho_P u_P V_P) \approx \frac{\rho_P V_P}{\Delta t}\left(u_P^{n+1} - u_P^n\right)} \tag{176}$$

The new-time value u_P^{n+1} is used in the discretised equation.

Temporal Discretisation: Contribution to the Equation

Expanding the transient term,

$$\frac{\rho_P V_P}{\Delta t} u_P^{n+1} - \frac{\rho_P V_P}{\Delta t} u_P^n. \tag{177}$$

The first term contains the unknown u_P^{n+1}.
Therefore it contributes to the central coefficient:

$$\boxed{a_P^{time} = \frac{\rho_P V_P}{\Delta t}} \tag{178}$$

The second term is known from the previous time level and contributes to the right-hand side:

$$\boxed{b^{time} = \frac{\rho_P V_P}{\Delta t} u_P^n} \tag{179}$$

Thus,

$$\boxed{\text{Transient term} = a_P^{time} u_P^{n+1} - b^{time}} \tag{180}$$

Transient X-Momentum Equation

The spatial discretisation gives

$$a_P^{space} u_P = a_E u_E + a_W u_W + a_N u_N + a_S u_S + b_u. \tag{181}$$

Adding the temporal contribution,

$$\frac{\rho_P V_P}{\Delta t}\left(u_P^{n+1} - u_P^n\right) + \text{spatial terms} = 0. \tag{182}$$

Rearranging,

$$\left(a_P^{space} + \frac{\rho_P V_P}{\Delta t}\right) u_P^{n+1} = a_E u_E^{n+1} + a_W u_W^{n+1} \tag{183}$$

$$+ a_N u_N^{n+1} + a_S u_S^{n+1} + b_u + \frac{\rho_P V_P}{\Delta t} u_P^n. \tag{184}$$

Transient Algebraic Equation

Define the transient contribution

$$a_P^{time} = \frac{\rho_P V_P}{\Delta t}. \tag{185}$$

Then the algebraic equation becomes

$$\boxed{a_P u_P^{n+1} = \sum_{nb} a_{nb} u_{nb}^{n+1} + b^{n+1}} \tag{186}$$

where

$$\boxed{a_P = a_P^{space} + \frac{\rho_P V_P}{\Delta t}} \quad \text{and} \quad \boxed{b^{n+1} = b_u + \frac{\rho_P V_P}{\Delta t} u_P^n.} \tag{187}$$

The previous time level u_P^n is known, while u_P^{n+1} is the new unknown.

What Does the Time Step Do?

The transient term introduces a new contribution to the central coefficient:

$$\boxed{a_P = a_E + a_W + a_N + a_S + \frac{\rho_P V_P}{\Delta t}} \tag{188}$$

and a contribution from the previous solution:

$$\boxed{b \supset \frac{\rho_P V_P}{\Delta t} u_P^n} \tag{189}$$

Therefore, each time step solves

$$\boxed{\mathbf{A}\mathbf{u}^{n+1} = \mathbf{b}(u^n)} \tag{190}$$

The solution at time n becomes the known input for the solution at time $n+1$.

Solve → advance time → solve again

Table of Contents

1 Outline

2 Continuum Mechanics and Fluid Description

3 Governing Equations

4 From Navier-Stokes to RANS

5 Solving Turbulent Flow

6 Discretization

7 Pressure-Velocity Coupling

8 Boundary Conditions

9 Geometry

10 Meshing

Pressure–Velocity Coupling

The discretised momentum equations have the form

$$\boxed{a_P u_P = \sum_{nb} a_{nb} u_{nb} + b_u} \tag{191}$$

and similarly for v.

However, the momentum equations contain the **pressure field**.

At the same time, the pressure must be such that the velocity field satisfies continuity:

$$\boxed{F_e - F_w + F_n - F_s = 0} \tag{192}$$

Therefore, velocity and pressure are **coupled**.

How do we solve for pressure and velocity simultaneously?

The Pressure-Velocity Coupling Problem

For incompressible flow,

$$\nabla \cdot \mathbf{u} = 0. \tag{193}$$

This equation does not contain pressure explicitly.

The momentum equations contain pressure:

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \mu \nabla^2 \mathbf{u}. \tag{194}$$

Therefore:

- Momentum determines velocity from pressure.
- Continuity constrains the velocity field.
- Pressure must adjust so that continuity is satisfied.

Pressure is the field that enforces mass conservation.

The Basic Idea of SIMPLE

The **SIMPLE** algorithm stands for

Semi-Implicit Method for Pressure-Linked Equations

The central idea is to separate the problem into two stages:

1. Guess the pressure field.
2. Use this pressure to calculate a velocity field.
3. Determine how the pressure must change to satisfy continuity.
4. Correct pressure and velocity.
5. Repeat until convergence.

Schematically,

$$\boxed{p^* \longrightarrow \mathbf{u}^* \longrightarrow p' \longrightarrow \mathbf{u}} \tag{195}$$

where * denotes a provisional solution and $'$ denotes a correction.

SIMPLE: Momentum Prediction

Suppose that a pressure field p^* is available. The discretised x-momentum equation is

$$a_P u_P = a_E u_E + a_W u_W + a_N u_N + a_S u_S + b_u. \tag{196}$$

The pressure contribution is evaluated using p^*. Therefore, we obtain a **provisional velocity**

$$\boxed{u_P^*} \tag{197}$$

and similarly,

$$\boxed{v_P^*} \tag{198}$$

from the momentum equations.

However, these provisional velocities do not necessarily satisfy continuity:

$$F_e^* - F_w^* + F_n^* - F_s^* \neq 0. \tag{199}$$

The pressure guess generally produces a mass imbalance.

Velocity and Pressure Corrections

The actual solution is written as a provisional value plus a correction:

$$\boxed{p = p^* + p'} \tag{200}$$

$$\boxed{u = u^* + u'} \tag{201}$$

$$\boxed{v = v^* + v'} \tag{202}$$

where

$$p', \quad u', \quad v'$$

are the pressure and velocity corrections.

The objective is to determine p' such that the corrected velocity field satisfies continuity:

$$\boxed{F_e - F_w + F_n - F_s = 0.} \tag{203}$$

Deriving the Velocity Correction

Consider the discretised x-momentum equation:

$$a_P u_P = \sum_{nb} a_{nb} u_{nb} + (p_W - p_E)A + S_u V_P. \tag{204}$$

For the provisional solution,

$$a_P u_P^* = \sum_{nb} a_{nb} u_{nb}^* + (p_W^* - p_E^*)A + S_u V_P. \tag{205}$$

Subtracting the two equations gives an equation for the velocity correction.

$$a_P u_P' = \sum_{nb} a_{nb} u_{nb}' + (p_W' - p_E')A. \tag{206}$$

When deriving the velocity correction from the pressure correction, we neglect their contribution to the equation. Therefore,

$$\boxed{u_P' \approx \frac{A}{a_P}(p_W' - p_E')} \tag{207}$$

Velocity Correction at the Faces

Continuity is written in terms of **face mass fluxes**. For the east face,

$$F_e = \rho_e u_e A_e. \tag{208}$$

The velocity correction gives

$$u_e = u_e^* + u_e'. \tag{209}$$

Therefore,

$$F_e = F_e^* + \rho_e A_e u_e'. \tag{210}$$

Using the momentum correction relation,

$$u_e' \approx \frac{A_e}{a_e}(p_P' - p_E') \tag{211}$$

for the corresponding momentum coefficient, we obtain

$$\boxed{F_e = F_e^* + \rho_e A_e d_e (p_P' - p_E')} \tag{212}$$

Similar expressions are obtained for the west, north and south faces.

The Pressure-Correction Equation

Continuity requires

$$F_e - F_w + F_n - F_s = 0. \tag{213}$$

Substitute the corrected fluxes:

$$F_f = F_f^* + F_f'. \tag{214}$$

Then

$$(F_e^* - F_w^* + F_n^* - F_s^*) + (F_e' - F_w' + F_n' - F_s') = 0. \tag{215}$$

The first term is the **mass imbalance** of the provisional velocity field.

The second term is the correction generated by p'.

After substituting the flux corrections and collecting pressure-correction terms,

$$\boxed{a_P^p p_P' = a_E^p p_E' + a_W^p p_W' + a_N^p p_N' + a_S^p p_S' + b_p} \tag{216}$$

This is the **pressure-correction equation**.

The Pressure-Correction Source

The source term of the pressure-correction equation is the mass imbalance of the provisional solution:

$$\boxed{b_p = -\left(F_e^* - F_w^* + F_n^* - F_s^*\right)} \tag{217}$$

Therefore,

$$\boxed{b_p = -\text{mass imbalance}} \tag{218}$$

This gives SIMPLE a very physical interpretation:

If a cell gains too much mass, pressure is corrected to modify the surrounding velocity field.

When

$$b_p \to 0, \tag{219}$$

the provisional velocity field increasingly satisfies continuity.

Pressure Correction

Once the pressure-correction equation is solved,

$$\boxed{p'_P = p'_P(\text{neighbouring cells})} \tag{220}$$

the pressure field is corrected:

$$\boxed{p_P = p^*_P + p'_P} \tag{221}$$

The velocities are also corrected:

$$\boxed{u_P = u^*_P + u'_P} \tag{222}$$

$$\boxed{v_P = v^*_P + v'_P} \tag{223}$$

and the face mass fluxes become

$$\boxed{F_f = F^*_f + F'_f.} \tag{224}$$

The corrected fluxes are therefore more consistent with the continuity equation.

Why Do We Need Under-Relaxation?

The pressure and velocity corrections obtained from SIMPLE may be too large.
Instead of applying the full correction, we can use under-relaxation.
For pressure,

$$\boxed{p^{new} = p^* + \alpha_p p'} \tag{225}$$

where

$$0 < \alpha_p \leq 1.$$

Similarly, for velocity,

$$\boxed{u^{new} = u^* + \alpha_u u'} \tag{226}$$

Typical interpretation:

- $\alpha = 1$: full correction.
- $\alpha < 1$: only part of the correction is applied.
- Smaller α: generally more damping.

Under-relaxation improves robustness of the iterative solution.

The SIMPLE Algorithm

The complete SIMPLE iteration can be summarized as:

1. Guess pressure p^*.
2. Solve the u-momentum equation to obtain u^*.
3. Solve the v-momentum equation to obtain v^*.
4. Calculate the provisional face mass fluxes F_f^*.
5. Construct the pressure-correction equation.
6. Solve for p'.
7. Correct pressure:
$$p = p^* + \alpha_p p'.$$
8. Correct velocities and face fluxes.
9. Check convergence.

If the solution has not converged:

$$\boxed{\text{repeat the SIMPLE iteration}} \tag{227}$$

SIMPLE and the Algebraic System

The discretisation produced equations such as

$$\boxed{a_P u_P = \sum_{nb} a_{nb} u_{nb} + b_u} \tag{228}$$

and

$$\boxed{a_P v_P = \sum_{nb} a_{nb} v_{nb} + b_v.} \tag{229}$$

SIMPLE does not replace these equations. Instead, SIMPLE provides a strategy for coupling

$$\boxed{\text{momentum equations} \quad \leftrightarrow \quad \text{continuity}} \tag{230}$$

through the pressure-correction equation.

The resulting iterative process is therefore

$$\boxed{\text{Momentum} \rightarrow \text{Pressure correction} \rightarrow \text{Velocity correction} \rightarrow \text{Continuity}} \tag{231}$$

SIMPLE for Transient Flow

SIMPLE is often introduced using the **steady-state** equations. However, it can also be used for transient simulations.

For a transient problem, the time derivative has already been discretised:

$$\boxed{a_P = a_P^{space} + \frac{\rho_P V_P}{\Delta t}} \tag{232}$$

and

$$\boxed{b = b^{space} + \frac{\rho_P V_P}{\Delta t} u_P^n.} \tag{233}$$

At each new time level,

$$t^n \rightarrow t^{n+1}, \tag{234}$$

SIMPLE can be used to obtain a converged solution at t^{n+1}.
Therefore,

$$\boxed{\text{Time step} \quad \rightarrow \quad \text{SIMPLE iterations} \quad \rightarrow \quad \text{next time step}} \tag{235}$$

What Does SIMPLE Actually Do?

The momentum equations alone can produce a velocity field for a given pressure field.
But that velocity field may violate continuity.

SIMPLE therefore asks:

How must pressure change so that mass is conserved?

The answer is obtained through the chain

$$\boxed{p^* \rightarrow \mathbf{u}^* \rightarrow \text{mass imbalance} \rightarrow p' \rightarrow \mathbf{u}' \rightarrow \text{corrected flow}} \tag{236}$$

Repeated iterations drive

$$\boxed{\text{mass imbalance} \rightarrow 0} \tag{237}$$

and therefore

$$\boxed{\nabla \cdot \mathbf{u} \rightarrow 0.} \tag{238}$$

SIMPLE is fundamentally a pressure–velocity coupling procedure.

From Theory to Fluent

The mathematical concepts we have introduced appear directly in the CFD software.

Theory	**CFD software**
Governing equations	Models / Physics
Mesh / control volumes	Mesh
Time discretization	Time-stepping scheme
Convection interpolation	Spatial discretization
Pressure–velocity coupling	SIMPLE / SIMPLEC / PISO
Under-relaxation	Solution controls
Initialization	Initialization
Residuals	Residual monitors
Convergence	Convergence criteria

We now know what the buttons actually mean.

Table of Contents

1 Outline

2 Continuum Mechanics and Fluid Description

3 Governing Equations

4 From Navier-Stokes to RANS

5 Solving Turbulent Flow

6 Discretization

7 Pressure-Velocity Coupling

8 Boundary Conditions

9 Geometry

10 Meshing

Boundary Conditions

After discretization, the governing equations are written for the control volumes in the computational domain.

For an interior cell, the discretized equation contains information from neighbouring cells:

$$\boxed{a_P \phi_P = \sum_N a_N \phi_N + b} \tag{239}$$

At a boundary, however, one or more neighbouring fluid cells do not exist.

Therefore, additional information must be specified:

$$\boxed{\text{Boundary conditions} \rightarrow \text{information required to close the discretized equations}} \tag{240}$$

The boundary conditions come from the **physics of the problem**.

What Must Be Specified at a Boundary?

The required boundary information depends on the governing equations and the physical boundary.

For the Navier–Stokes equations, this may involve:

- velocity,
- pressure,
- velocity gradients,
- mass flow,
- temperature,
- turbulence quantities.

For example, at a stationary solid wall:

$$\boxed{\mathbf{u} = \mathbf{0}} \tag{241}$$

while at an outlet we may prescribe

$$\boxed{p = p_{out}.} \tag{242}$$

The boundary conditions provide the information that is missing from the discretized equations.

Common Boundary Conditions

Typical boundary conditions include:

Boundary	**Typical specification**
Wall	$\mathbf{u} = \mathbf{u}_{wall}$
Inlet	$\mathbf{u} = \mathbf{u}_{in}$
Outlet	$p = p_{out}$
Symmetry	$\mathbf{u} \cdot \mathbf{n} = 0$
Far-field	$\mathbf{u} \to \mathbf{U}_\infty$
Periodic	Repeating solution

The exact specification depends on the physical problem.

$$\boxed{\text{Choose the boundary condition from the physics}} \tag{243}$$

Boundary Conditions in the FVM

The finite volume method enforces conservation over each control volume.

At an interior cell:

$$\boxed{\text{Neighbour information} \rightarrow \text{Fluxes} \rightarrow \text{Discrete equation}} \tag{244}$$

At a boundary cell:

$$\boxed{\text{Boundary condition} \rightarrow \text{Boundary information} \rightarrow \text{Fluxes} \rightarrow \text{Discrete equation}} \tag{245}$$

Therefore, boundary conditions are part of the numerical formulation itself.

They are not simply settings applied after the equations have been solved.

Dirichlet and Neumann Boundary Conditions

Two fundamental types of boundary conditions are:

Dirichlet: prescribe the value of the variable

$$\boxed{\phi = \phi_{BC}} \tag{246}$$

For example, a prescribed velocity at an inlet:

$$\mathbf{u} = \mathbf{u}_{in} \tag{247}$$

Neumann: prescribe the gradient of the variable

$$\boxed{\frac{\partial \phi}{\partial n} = \left(\frac{\partial \phi}{\partial n}\right)_{BC}} \tag{248}$$

For example, a zero-gradient condition:

$$\boxed{\frac{\partial \phi}{\partial n} = 0} \tag{249}$$

Example: Boundary Condition at a Solid Wall

Consider a control volume adjacent to a stationary bottom wall.

The south face of the control volume coincides with the wall.

For a stationary solid wall, the no-slip and impermeability conditions are

$$\boxed{u_s = 0, \qquad v_s = 0} \tag{250}$$

where u_s and v_s are the velocity components at the wall face.

- $v_s = 0$: no fluid crosses the wall.
- $u_s = 0$: the fluid velocity relative to the wall is zero.

Wall Boundary Condition: Convective Flux

Consider the x-momentum equation for the cell:

$$\begin{aligned} F_e u_e - F_w u_w + F_n u_n - F_s u_s = & -(p_e A_e - p_w A_w) \\ & + D_e(u_E - u_P) - D_w(u_P - u_W) \\ & + D_n(u_N - u_P) - D_s(u_P - u_s) + S_u V_P. \end{aligned} \tag{251}$$

The mass flux through the south face is

$$F_s = \rho v_s A_s. \tag{252}$$

At an impermeable wall,

$$v_s = 0$$

and therefore

$$\boxed{F_s = 0} \tag{253}$$

Hence the convective contribution through the wall is

$$\boxed{F_s u_s = 0.} \tag{254}$$

No mass or convective momentum crosses a stationary wall.

Wall Boundary Condition: Diffusive Flux

Although the convective flux is zero, the viscous flux is **not** zero. For the south face,

$$D_s = \frac{\mu_s A_s}{d_{Ps}}, \tag{255}$$

where d_{Ps} is the distance from the cell centre P to the wall face.
The velocity gradient is approximated by

$$\left.\frac{\partial u}{\partial y}\right|_s \approx \frac{u_P - u_s}{d_{Ps}}. \tag{256}$$

Therefore, the diffusive contribution is

$$\boxed{-D_s(u_P - u_s)} \tag{257}$$

For a stationary no-slip wall,

$$\boxed{-D_s(u_P - u_s) = -D_s u_P.} \tag{258}$$

The Final x-Momentum Equation at the Wall

Starting from

$$\begin{aligned} F_e u_e - F_w u_w + F_n u_n - F_s u_s = & -(p_e A_e - p_w A_w) \\ & + D_e(u_E - u_P) - D_w(u_P - u_W) \\ & + D_n(u_N - u_P) - D_s(u_P - u_s) + S_u V_P, \end{aligned} \tag{259}$$

the stationary-wall conditions give

$$\boxed{\begin{aligned} F_e u_e - F_w u_w + F_n u_n = & -(p_e A_e - p_w A_w) \\ & + D_e(u_E - u_P) - D_w(u_P - u_W) \\ & + D_n(u_N - u_P) - D_s u_P + S_u V_P. \end{aligned}} \tag{260}$$

- The **convective south flux disappears**;
- The **viscous south flux remains**;
- The wall velocity $u_s = 0$ enters the diffusion term;
- The wall distance d_{Ps} determines D_s.

Physical Interpretation

The stationary wall imposes

$$\boxed{u_s = 0, \qquad v_s = 0.} \tag{261}$$

These conditions have two different consequences:

$$\boxed{v_s = 0 \implies F_s = 0} \rightarrow \quad \text{No fluid crosses the wall.} \tag{262}$$

$$\boxed{u_s = 0 \implies \frac{\partial u}{\partial y} \approx \frac{u_P}{d_{Ps}}} \rightarrow \quad \text{A velocity gradient exists near the wall.} \tag{263}$$

This gradient produces the viscous wall shear stress:

$$\boxed{\tau_w = \mu \left.\frac{\partial u}{\partial y}\right|_w} \tag{264}$$

No convective flux does not mean no momentum interaction with the wall.

Example: Velocity Inlet

Consider a control volume adjacent to a prescribed-velocity inlet at the west boundary.

At the inlet, the velocity is prescribed:

$$\boxed{u_w = U_{\text{in}}, \qquad v_w = V_{\text{in}}} \tag{265}$$

For a simple horizontal inlet,

$$\boxed{u_w = U_{\text{in}}, \qquad v_w = 0.} \tag{266}$$

Unlike a solid wall, fluid is allowed to cross the inlet boundary.
The west mass flux is

$$F_w = \rho\, \mathbf{u}_w \cdot \mathbf{n}_w A_w. \tag{267}$$

For a west inlet, the outward normal points to the left:

$$\mathbf{n}_w = (-1, 0).$$

Inlet Boundary: Convective Flux

Therefore,

$$F_w = -\rho U_{\text{in}} A_w. \tag{268}$$

Hence,

$$\boxed{F_w < 0} \tag{269}$$

for an inlet.

The negative sign indicates that mass is entering the control volume.

The convective contribution to the x-momentum equation is

$$-F_w u_w. \tag{270}$$

Since $u_w = U_{\text{in}}$,

$$\boxed{-F_w u_w = \rho U_{\text{in}}^2 A_w.} \tag{271}$$

Inlet Boundary: Diffusive Flux

The west diffusive contribution is

$$-D_w(u_P - u_w). \tag{272}$$

At the inlet, the velocity at the boundary is known:

$$u_w = U_{\text{in}}. \tag{273}$$

Therefore,

$$\boxed{-D_w(u_P - U_{\text{in}})} \tag{274}$$

where

$$D_w = \frac{\mu_w A_w}{d_{Pw}}. \tag{275}$$

The x-Momentum Equation at the Inlet

Starting from

$$\begin{aligned} F_e u_e - F_w u_w + F_n u_n - F_s u_s = & -(p_e A_e - p_w A_w) \\ & + D_e(u_E - u_P) - D_w(u_P - u_W) \\ & + D_n(u_N - u_P) - D_s(u_P - u_S) + S_u V_P, \end{aligned} \tag{276}$$

at the inlet,

$$\boxed{\begin{aligned} F_e u_e - F_w U_{\text{in}} + F_n u_n - F_s u_s = & -(p_e A_e - p_w A_w) \\ & + D_e(u_E - u_P) - D_w(u_P - U_{\text{in}}) \\ & + D_n(u_N - u_P) - D_s(u_P - u_S) + S_u V_P. \end{aligned}} \tag{277}$$

The inlet value appears in both the convective and diffusive contributions.

Physical Interpretation: Inlet

At a prescribed-velocity inlet,

$$\boxed{u_w = U_{\text{in}}, \qquad v_w = V_{\text{in}}.} \tag{278}$$

This provides the velocity information required at the boundary.

Two effects appear in the finite-volume balance:

$$\boxed{F_w u_w \quad \rightarrow \quad \text{convective transport entering the domain}} \tag{279}$$

and

$$\boxed{D_w(u_P - u_w) \quad \rightarrow \quad \text{viscous transport between the boundary and the cell.}} \tag{280}$$

Unlike the wall:

$$\boxed{F_w \neq 0}$$

because fluid crosses the inlet.

The prescribed velocity supplies the missing information for the boundary face.

From Boundary Conditions to Geometry

We now know:

- the governing equations,
- how they are discretized,
- how pressure and velocity are coupled,
- how boundary conditions enter the finite-volume equations,
- how boundary conditions determine the face fluxes.

The next question is therefore:

What computational domain should we create?

The geometry must contain the physical boundaries required by the mathematical problem.

$$\boxed{\text{Physics} \rightarrow \text{Equations} \rightarrow \text{BCs} \rightarrow \text{Computational domain} \rightarrow \text{Mesh}} \tag{281}$$

Next: Geometry and Meshing

Table of Contents

1. Outline
2. Continuum Mechanics and Fluid Description
3. Governing Equations
4. From Navier-Stokes to RANS
5. Solving Turbulent Flow
6. Discretization
7. Pressure-Velocity Coupling
8. Boundary Conditions
9. Geometry
10. Meshing

CFD Workflow

A typical CFD workflow can be organized as:

1. **Geometry**
2. **Mesh**
3. **Solver setup**
4. **Boundary conditions**
5. **Solution**
6. **Convergence and verification**
7. **Post-processing**

$$\text{Geometry} \rightarrow \text{Mesh} \rightarrow \text{Solver} \rightarrow \text{Solution} \rightarrow \text{Results}$$

The quality of the final result depends strongly on the decisions made at each stage.

1. Geometry: What Are We Actually Simulating?

The first step is to define the physical problem and its computational representation.

Before creating the mesh, ask:

- What is the physical object?
- What part of the geometry is relevant to the flow?
- What is the fluid domain?
- What dimensions are important?
- Which geometric details affect the flow?
- Which details can be simplified?

$$\boxed{\text{CAD geometry} \quad \neq \quad \text{CFD geometry}}$$

The geometry should be prepared specifically for the flow simulation.

From CAD Geometry to the Fluid Domain

CFD solves the equations in the **fluid domain**.

For an external-flow problem:

$$\boxed{\text{Fluid domain} = \text{Computational enclosure} - \text{Solid body}} \tag{282}$$

For an internal-flow problem:

$$\boxed{\text{Fluid domain} = \text{Volume occupied by the fluid}} \tag{283}$$

The computational domain must be large enough that the artificial boundaries do not significantly influence the physical solution.

Defining the Computational Boundaries

The geometry must also identify the surfaces where boundary conditions will be applied.

Typical boundaries include:

Boundary	**Typical use**
Inlet	Prescribed velocity, mass flow, turbulence quantities
Outlet	Pressure or outflow condition
Wall	No-slip condition
Symmetry	Symmetry plane
Far-field	External-flow boundary
Periodic	Repeating flow domain

$$\boxed{\text{Geometry} \rightarrow \text{Physical boundaries} \rightarrow \text{Boundary conditions}} \tag{284}$$

The boundary names should be defined clearly before meshing and solver setup.

Geometry Cleanup

Real CAD models often contain geometric features that are irrelevant to the flow solution.

Examples include:

- Tiny fillets
- Small gaps and edges
- Bolts and fasteners
- Cosmetic details

These features can create:

- Very small mesh cells
- Poor cell quality
- Large cell-count increases
- Difficult convergence
- Unnecessary computational cost

Remove details that do not affect the physics

How Much Geometry Should We Keep?

Geometry simplification should be based on the **physics**, not simply on visual appearance.

A feature should be retained if it can significantly affect:

- Separation
- Boundary-layer development
- Vortices
- Pressure distribution
- Drag or lift
- Heat transfer
- Flow recirculation

A useful question is:

"Will this geometric feature change the flow prediction?"

If the answer is no, the feature may not need to be represented in the CFD geometry.

Geometry Checklist

Before moving to the meshing stage:

1. Define the physical problem.
2. Identify the fluid region.
3. Create the computational domain.
4. Define inlet, outlet, walls and other boundaries.
5. Remove unnecessary geometric details.
6. Check for gaps, overlaps and unwanted intersections.
7. Verify dimensions and units.
8. Check that the geometry is suitable for meshing.

Good geometry → good mesh → better CFD

Next: Meshing

Once the geometry is prepared, the fluid domain is discretized into a finite number of control volumes.

$$\boxed{\text{Continuous fluid domain} \quad \longrightarrow \quad \text{Discrete computational mesh}} \tag{285}$$

The mesh determines where the governing equations will be solved and strongly influences:

- Accuracy
- Numerical stability
- Convergence
- Computational cost

$$\boxed{\textbf{Geometry} \rightarrow \textbf{Mesh}}$$

Table of Contents

1 Outline

2 Continuum Mechanics and Fluid Description

3 Governing Equations

4 From Navier-Stokes to RANS

5 Solving Turbulent Flow

6 Discretization

7 Pressure-Velocity Coupling

8 Boundary Conditions

9 Geometry

10 Meshing

2. Meshing: Discretizing the Flow Domain

The governing equations are defined over a continuous fluid domain.
To solve them numerically, the domain is divided into a finite number of smaller control volumes.

$$\boxed{\text{Continuous domain} \quad \longrightarrow \quad \text{Discrete control volumes}} \tag{286}$$

Each cell contains the variables required by the solver, such as

$$p, \qquad u, \qquad v, \qquad w, \qquad T, \qquad k, \qquad \varepsilon, \qquad \omega.$$

The mesh therefore defines:

- Where the governing equations are solved
- The spatial resolution of the solution
- The computational cost
- Part of the numerical accuracy

Types of Computational Meshes

Meshes can be classified according to their topology and cell type.

Structured mesh

- Regular connectivity
- Typically quadrilateral cells in 2D
- Typically hexahedral cells in 3D
- Efficient and highly controllable
- Can be difficult for complex geometries

Unstructured mesh

- Flexible connectivity
- Triangles in 2D
- Tetrahedra in 3D
- Easily adapts to complex geometries
- Often requires more cells

Other common approaches include:

Hexahedral	Tetrahedral	Prismatic	Polyhedral	Hybrid

In practice, many engineering CFD simulations use **hybrid meshes**.

What Makes a Good CFD Mesh?

Important mesh characteristics include:

- **Cell size** – sufficient spatial resolution
- **Skewness** – cells should not be excessively distorted
- **Orthogonality** – faces should be well aligned
- **Aspect ratio** – appropriate for the local flow
- **Smoothness** – avoid abrupt changes in cell size

The mesh should be refined in regions such as:

- Boundary layers
- Shear layers
- Separation regions
- Wake regions
- Vortices

Example: Resolving the Boundary Layer

Near a solid wall, velocity changes rapidly from the wall condition to the external flow.

$$\boxed{\text{Large velocity gradients} \quad \Rightarrow \quad \text{high mesh resolution required}} \tag{287}$$

A typical boundary-layer mesh uses several layers of inflation/prism cells:

$$\boxed{\text{Wall} \rightarrow \text{1st layer} \rightarrow \text{2nd layer} \rightarrow \cdots \rightarrow \text{Outer flow}}$$

For turbulence models, the first-cell height is closely related to the dimensionless wall distance

$$\boxed{y^+ = \frac{u_\tau y}{\nu}} \tag{288}$$

where u_τ is the friction velocity.

The appropriate y^+ depends on the turbulence model and near-wall treatment.

Mesh Independence

A simulation should not depend strongly on the particular mesh resolution.
Consider a sequence of meshes:

$$\boxed{\text{Coarse} \rightarrow \text{Medium} \rightarrow \text{Fine} \rightarrow \text{Very Fine}}$$

For a quantity of interest ϕ,

$$\phi_{\text{coarse}} \neq \phi_{\text{medium}} \neq \phi_{\text{fine}}$$

initially. As the mesh is refined,

$$\boxed{\phi_{\text{fine}} \approx \phi_{\text{very fine}}} \tag{289}$$

indicates that the solution is becoming **mesh independent**. The same idea is applied to the time step when the fluid flow varies over time.

Typical quantities to monitor include:

- Aerodynamic coefficients
- Pressure drop

Meshing Checklist

Before moving to the solver:

1. Choose an appropriate mesh topology.
2. Define the required global resolution.
3. Identify regions requiring local refinement.
4. Resolve boundary layers.
5. Select an appropriate first-cell height and y^+.
6. Check cell quality.
7. Check for negative-volume or invalid cells.
8. Prepare a mesh-independence study.

$$\boxed{\text{Geometry} \rightarrow \text{Mesh} \rightarrow \text{Solver Setup}}$$

Computational Fluid Dynamics

Bachelor's Degree in Aeronautical and Space Sciences

Emanuel A. R. Camacho

emanuel.camacho@iseclisboa.pt
earc@earc96.com

Instituto Superior de Educação e Ciências (ISEC Lisboa)

Latest Update: *28th October, 2026, 11:37*