Nile Basin capacity Building Network (NBCBN)

River morphology Research Cluster

Specialised Training Course On Modelling For River Engineering

Applications On : SSIIM Program

(25th -29th September, 2011), Egypt

By : Dr: Ahmed Musa Siyam
Eng: Elnazir Saad Ali

1. INTRODUCTION TO CFD

- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

➢ What is CFD

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as sediment flow or chemical reactions by means of computer-base simulation. The technique is very powerful and spans wide range of industrial and non-industrial areas of applications.

There are different types of numerical models availble for one two and three dimensional anaalysis with varing degree of sophistication and reliability.
The science of numerical modeling is advancing fast.

Generally there are two type of Computation Fluid Dynamic' (CFD) programs. One are general purpose programs and other are exclusively made for River Engineering Among the first type PHOENICS. STAR-CD,CFX, FLUENT and FLOW-3D and in second type TELEMAC, MIKE3, DELFT-3D, CH3D, TABS and SSIIM are prominent

In recent years multi-dimensional, computer programs for computing several different processes, for example water surface profiles etc has been developed,
These also exist width-averaged Two-dimensional models, but these are mostly used mainly for research purposes.

1. INTRODUCTION TO CFD

- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

Areas of Applications

- Aerodynamics of aircraft and vehicles: (Lift and drag)
- Hydrodynamics of ships
- Power plant: (Combustion engines &gas turbine)
- Turbo-machinery: (Flows inside rotating passages, diffusers, etc..).
- Electrical and electronic engineering: (Cooling of equipment)

Areas of Applications cont'd

- Chemical process engineering : (mixing and separation)
- External and internal environment of buildings: (wind load, heating ,ventilation).
- Meteorology: (weather prediction).
- Marine engineering: (loads on off-shore structures))
- Hydrology and oceanography: (flows in rivers, Estuaries and oceans)
- Biomedical Engineering: (blood flows through arteries and veins)

1. INTRODUCTION TO CFD

- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

Advantages of CFD

- Substantial reduction of lead times and costs of new designs.
- Ability to study systems under hazardous conditions at and beyond their normal performance limit.
- Practical unlimited level of detail of result.
- Ability to study systems where control experiments are difficult to perform.

1. INTRODUCTION TO CFD

- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

How does a CFD works

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems.

In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine results. Hence all codes contain three main elements:

1- A pre-processor

2- A solver

3- A post processer

(1) A pre-processor

Pre-processing consists of the input of a flow problem to CFD program by means of an operator- friendly interface and the subsequent transformation of this input into a form suitable for use by solver.

The user activities at the pre-processing stage involve:

(1) Definition of the geometry of the region of interest: the computational domain.

(2) Grid generation the sub-division of the domain into a number of smaller, non-overlapping sub-domains: a grid (or mesh) of cells(or control volumes or elements).

(3) Selection of the physical and chemical phenomena that need to be modeled.

(4) Definition of fluid properties.

(5) Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary.

The solution to a flow problem (velocity, pressure, temperature etc ..) is defined at nodes inside each cell.

The accuracy of CFD solution is governed by the number of cells in the grid. In general the large number of cells the better solution accuracy.

Optimal meshes are often non-uniform: finer in areas where large variations occur and coarser in regions with relatively little change.

Currently some CFD codes have developed self-adaptive meshing capabilities. i.e. automatically refines grid in area of rapid variations.

At present it is still up to the skills of the CFD user to design a grid that is a suitable compromise between desired accuracy and solution cost.

Over 50% of the time spent in industry on a CFD project is devoted to definition of the domain geometry and grid generation.

To maximize the productivity of CFD personnel, currently all major CFD codes incorporates their own CAD-style interface and/or facilitates to import data from other grid generation models

(2) A solver

There are three distinct streams of numerical solution techniques: finite difference , finite element and spectral methods.

In outline the numerical methods that form the basis of the solver perform the following three steps:

1. Approximation of the unknown flow variables by means of simple functions.

2. Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.

3. Solution of the algebraic equations.

Finite difference methods :

Describe the unknowns φ of the flow problem by means of point samples at the node points of grid of co-ordinate lines.

Truncated Taylor series expansions are often used to generate finite difference approximation of derivatives of grid of φ in term of point samples of φ at each grid point and its immediate neighbours.

Hence those derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation for the values of φ at each grid point.

Finite element method:

Uses simple piecewise functions (e.g. linear or quadratic) valid on elements to describe the local variations of unknown flow variables φ. The governing equation is precisely satisfied by the exact solution φ.

If the piecewise approximating functions for φ is substituted into the equations it will not hold exactly an a residual is defined to measure the errors.

Next the errors are minimised in some sense by multiplying them by a set of weighted functions and integrating.

As a result we obtain a set of algebraic equations for the un-known coefficients of the approximating functions. The theory of finite elements has been developed initially for structural stress analysis.

o Spectral methods:

Spectral methods approximate the unknowns by means of truncated Fourier series or series of Chebyshev polynomials. Again the unknowns in the governing equation are replaced by the truncated series.

The constraint that leads to the algebraic equations for the coefficients of the Fourier or Chebyshev series is provided by a weight residuals concept similar to the finite element method or by making the approximate function coincide with the exact solution at a number of grid points.

(3) Post processor

As in pre-processing a huge amount of development work has recently taken place in the post-processing field . owing to the increased popularity of engineering workstations , many of which have outstanding graphics capabilities, the leading CFD package are now equipped with visualization tools include :

- **Domain geometry and grid display**
- **2D and 3D surface plots**
- **Vector plots**
- **Line and shaded contour plots**
- **Colour postscript output**
- **Particle tracking**
- **View manipulation (translation, rotation, scaling etc.)**

More recently these facilities may also include animation for dynamic result display and in addition to graphics all codes produce trusty alphanumeric output and have data export facilities for further manipulation external to the code . As in many other branches of CAE the graphics capabilities of codes have revolutionized the communication of ideas to the non-specialist.

1. INTRODUCTION TO CFD

- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

➢ Problem Solving With CFD

In solving fluid flow problems we need to be aware that the underlying physics is complex and the results generated by a CFD code are at best as good as the physics(and chemistry)embedded in it and at worst as good as its operator.

Prior to setting up and running a CFD simulation there is:

(1) a stage of identification and formulation of the flow problem in terms of physical phenomena that need to be considered. Such as:

- Whether to model a problem in 2D or 3D.
- To exclude effect of pressure or temperature variation on density of air flow
- To neglect effect of small air bubbles in dissolved in tap water.
- To choose to solve the turbulent flow equations.

(2) A good understanding of the numerical solution algorithm

There are three mathematical concepts that are useful in determining the success or other wise of the solution Algorithm

Convergence Consistency Stability

Convergence: is the property of a numerical method to produce a solution which approaches the exact solution as the grid spacing, control volume size or element size reduced to zero.

Consistency: is the capability of the numerical scheme to produce systems of algebraic equations which can be demonstrated to be equivalent to the original governing equations as the grid spacing tend to zero.

Stability: is associated with damping of errors as the numerical method proceeds. If a technique is not stable even round-off errors in the initial data can cause wild oscillations and divergence.

1. INTRODUCTION TO CFD

- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

The Convection - Diffusion Equation

The movement and dispersion of suspended sediments temperature, a pollutant etc. in a water body is described by its convection-diffusion equation is in general the same for almost water quality parameters,

The convection-diffusion equation for steady sediment transport is:

$$U_j \frac{\partial c}{\partial x_j} + w \frac{\partial c}{\partial z} = \frac{\partial}{\partial x_j}\left(\Gamma_T \frac{\partial c}{\partial x_j}\right)$$

Where Γ is the turbulent mixing coefficient defined as the sediment flux per unit area divided by the concentration gradient and given by:

$$\Gamma = \frac{\left(\frac{F}{A}\right)}{\left(\frac{dc}{dx}\right)}$$

For three-dimensional flow this means that the equation can be written:

$$U\frac{\partial c}{\partial x} + V\frac{\partial c}{\partial y} + W\frac{\partial c}{\partial z} + w\frac{\partial c}{\partial z}$$

$$= \frac{\partial}{\partial x}\left(\Gamma_T\frac{\partial c}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma_T\frac{\partial c}{\partial y}\right) + \frac{\partial}{\partial z}\left(\Gamma_T\frac{\partial c}{\partial z}\right)$$

1. Transport Processes

There are two main transport processes: Convection and diffusion

Convection: The convection is a movement by the average water velocity. The transport because of the fall velocity of the sediment particles is also a type of convective transport. When calculating the flux, F, through a given surface with area A, the following formula is used:

$$F = c * U * A$$

Diffusion

The other process is the turbulent diffusion of sediments. This is due to turbulent mixing and concentration gradients. The turbulent mixing process is usually modelled with a turbulence mixing coefficient, Γ.

$$\Gamma = \frac{\left(\frac{F}{A}\right)}{\left(\frac{dc}{dx}\right)}$$

Normally, the convective transport will be dominating. But in some cases, the diffusive transport is important. An example is the reduced settling in a sand trap because of turbulence.

2. Discretization:

The point of discretization is to transform the partial differential equation into a new equation where the variable in one cell is a function of the variable in the neighbour cells

The new function can be thought of as a weighted average of the concentration in the neighbouring cells. For a two-dimensional situation , the following notation is used, according to directions north, south, east and west:

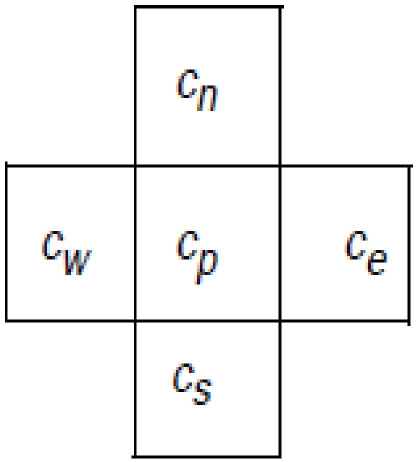


Figure 3.2.1 Discretization molecule. Computation of concentration, c, in the center cell, p, as a function of the concentration in the neighbouring cells n, s, e and w.

:

a_e : weighting factor for *cell e*
a_w : weighting factor for *cell w*
a_n : weighting factor for *cell n*
a_s : weighting factor for *cell s*
$a_p = a_e + a_w + a_n + a_s$

The formula becomes:

$$c_p = \frac{a_w c_w + a_e c_e + a_n c_n + a_s c_s}{a_p} \qquad \text{Eq. 1.}$$

The weighting factors for the neighbouring cells a_e, a_w, a_n and a_s are often denoted a_{nb}

What we want to obtain are formulas for a_{nb}

There are a number of different discretization methods available for the control-volume approach. The difference is in *how the concentration on a cell surface is calculated.* Some methods are described in the following.

In a three-dimensional computation, the same principles are involved. But two more neighbouring cells are added: t (top) and b (bottom), resulting in six neighbour cells. The simple extension from 2D to 3D is one of the main advantages of the finite volume method.

3. Discretization Using the First-Order Upstream Scheme:

Development of CFD algorithms was initially done in aeronautics. The fluid was air, and the methods were then called upwind instead of upstream. Both expressions are used, meaning the same.

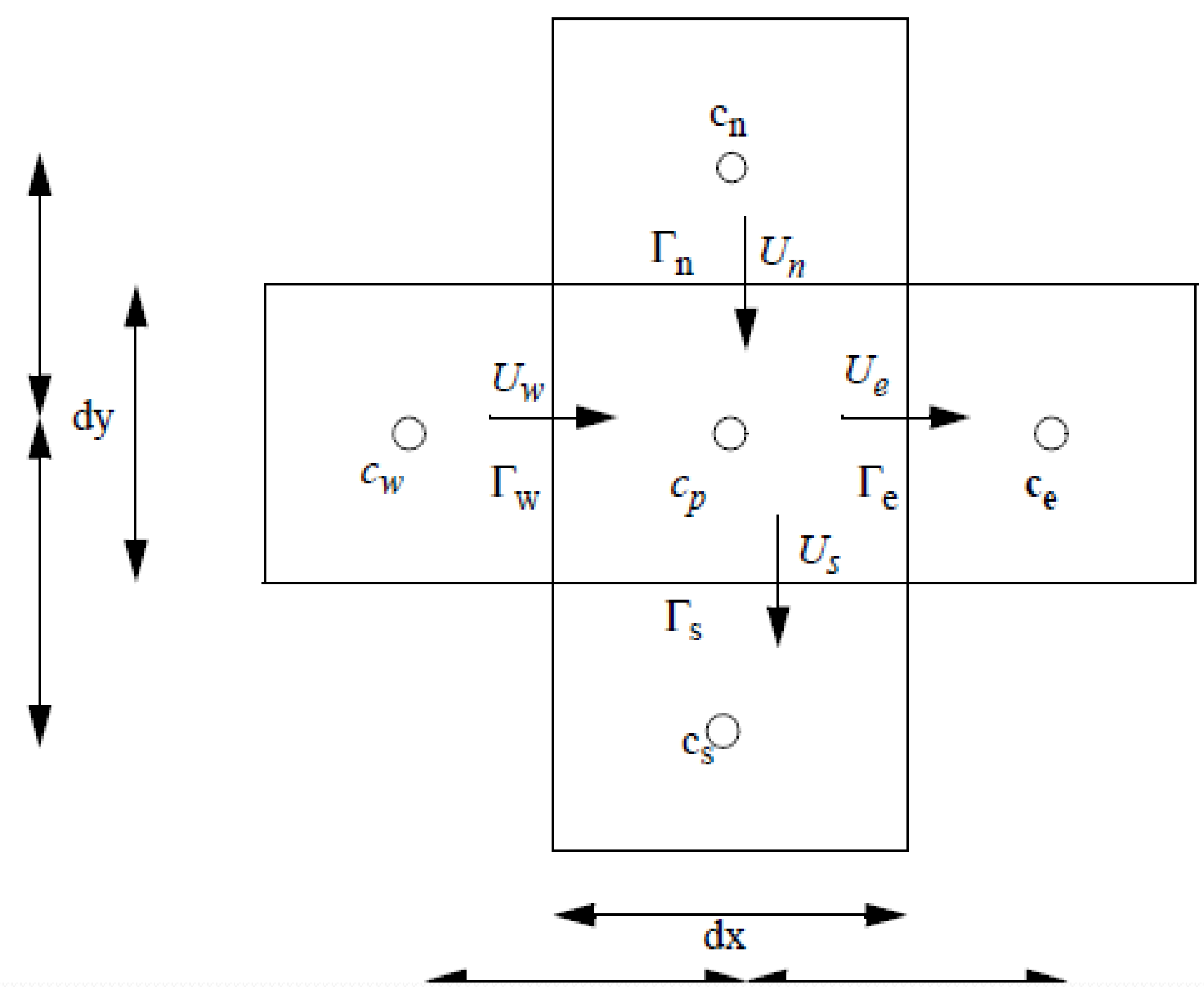
For a non-staggered grid, the values of the variables are given in the center of the cells. Using the finite volume method, it is necessary to estimate variable values on the cell surfaces. The main idea of the upstream methods is to estimate the surface value from the *upstream* cell. The first order method uses information in only one cell upstream of the cell surface. In other words: the concentration at a cell surface for the first-order upstream method is the same as the concentration in the cell on the upstream side of the cell side.

The control volume method is based on continuity of sediments. The basis of the calculation is the fluxes on a cell surface. The surface area is denoted *A*; the velocity at the surface, normal to it, is denoted *U*; c is the concentration at the surface, and Γ is the turbulent diffusion at the surface.

The convective flux is calculated as: $U * A * c$
The diffusive flux is calculated as: $\Gamma * A * dc / dx$

The term *dc/dx* is calculated as the concentration difference between the cells on each side of the surface, divided by the distance between the centres of the cells.

Fluxes through the walls of the center cell in computational molecule. The cells have a width dx and a height dy.



The flux, F_W, through the west side of cell P then becomes:

$$F_w = U_w A_w c_w + \Gamma_w \frac{A_w(c_w - c_p)}{dx}$$

where A_W is the area of the cell wall on the west side, equal to Δy times the height of the wall.

Similarly the fluxes through the other sides can be obtained

$$F_e = U_e A_e c_p + \Gamma_e \frac{A_e(c_p - c_e)}{dx}$$

$$F_s = U_s A_s c_p + \Gamma_s \frac{A_s(c_p - c_s)}{dy}$$

$$F_n = U_n A_n c_n + \Gamma_n \frac{A_n(c_n - c_p)}{dy}$$

Sediment continuity means the sum of the fluxes is zero, in other words:

$$F_w - F_e + F_n - F_s = 0$$

This gives the following equation:

$$\left(U_w A_w + U_e A_e + \Gamma_e \frac{A_e}{dx} + U_s A_s + \Gamma_s \frac{A_s}{dy} + \Gamma_n \frac{A_n}{dy}\right) c_p$$

$$= \left(U_w A_w + \Gamma_w \frac{A_w}{dx}\right) c_w + \left(\Gamma_e \frac{A_e}{dx}\right) c_e +$$

$$\left(U_n A_n + \Gamma_n \frac{A_n}{dy}\right) c_n + \left(\Gamma_s \frac{A_s}{dy}\right) c_s$$

Rearranging and compare with Eq. 1 we can get formulas for a_{nb} as follows

$$a_p = \Gamma_w \frac{A_w}{dx} + U_e A_e + \Gamma_e \frac{A_e}{dx} + U_s A_s + \Gamma_s \frac{A_s}{dy} + \Gamma_n \frac{A_n}{dy}$$

$$a_w = U_w A_w + \Gamma_w \frac{A_w}{dx}$$

$$a_e = \Gamma_e \frac{A_e}{dx}$$

$$a_s = \Gamma_s \frac{A_s}{dy}$$

$$a_n = U_n A_n + \Gamma_n \frac{A_n}{dy}$$

Further the water continuity for the cell can be written as:

$$U_e A_e - U_w A_w + U_n A_n - U_s A_s = 0$$

or:

$$U_e A_e + U_n A_n = U_w A_w + U_s A_s$$

If the above equation is inserted into the expression for a_p, the equation

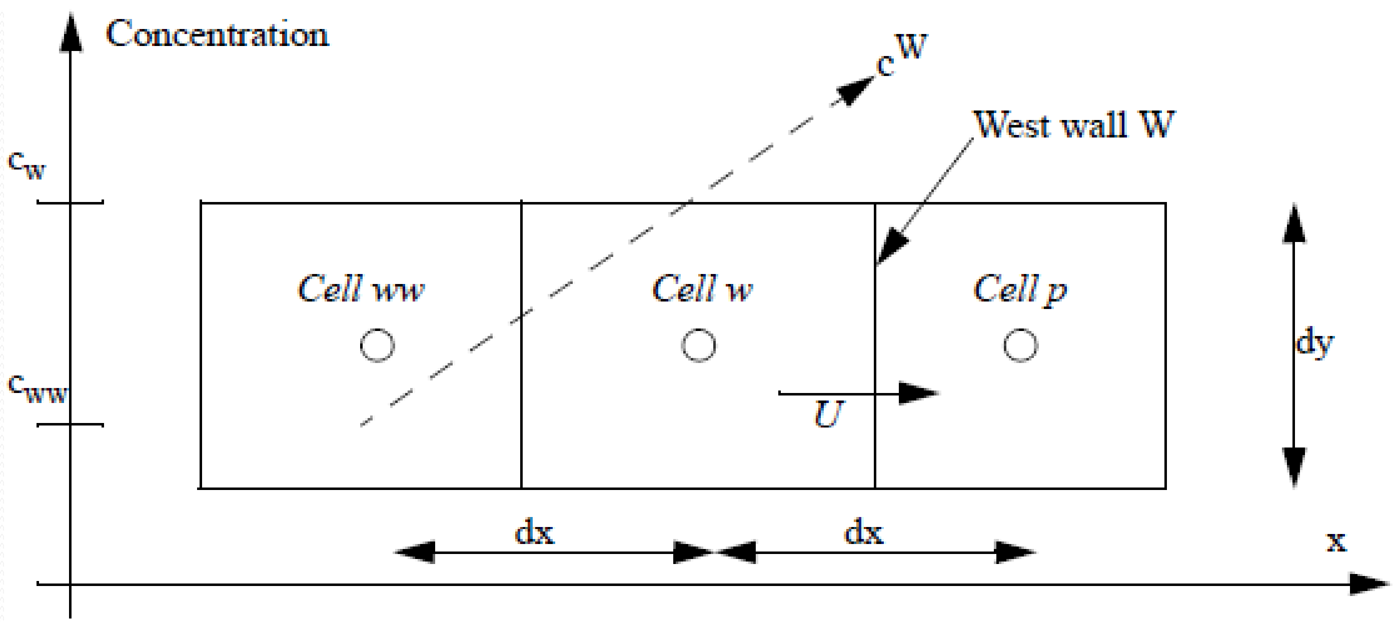
$$a_p = a_e + a_w + a_s + a_n$$

is verified to be correct.

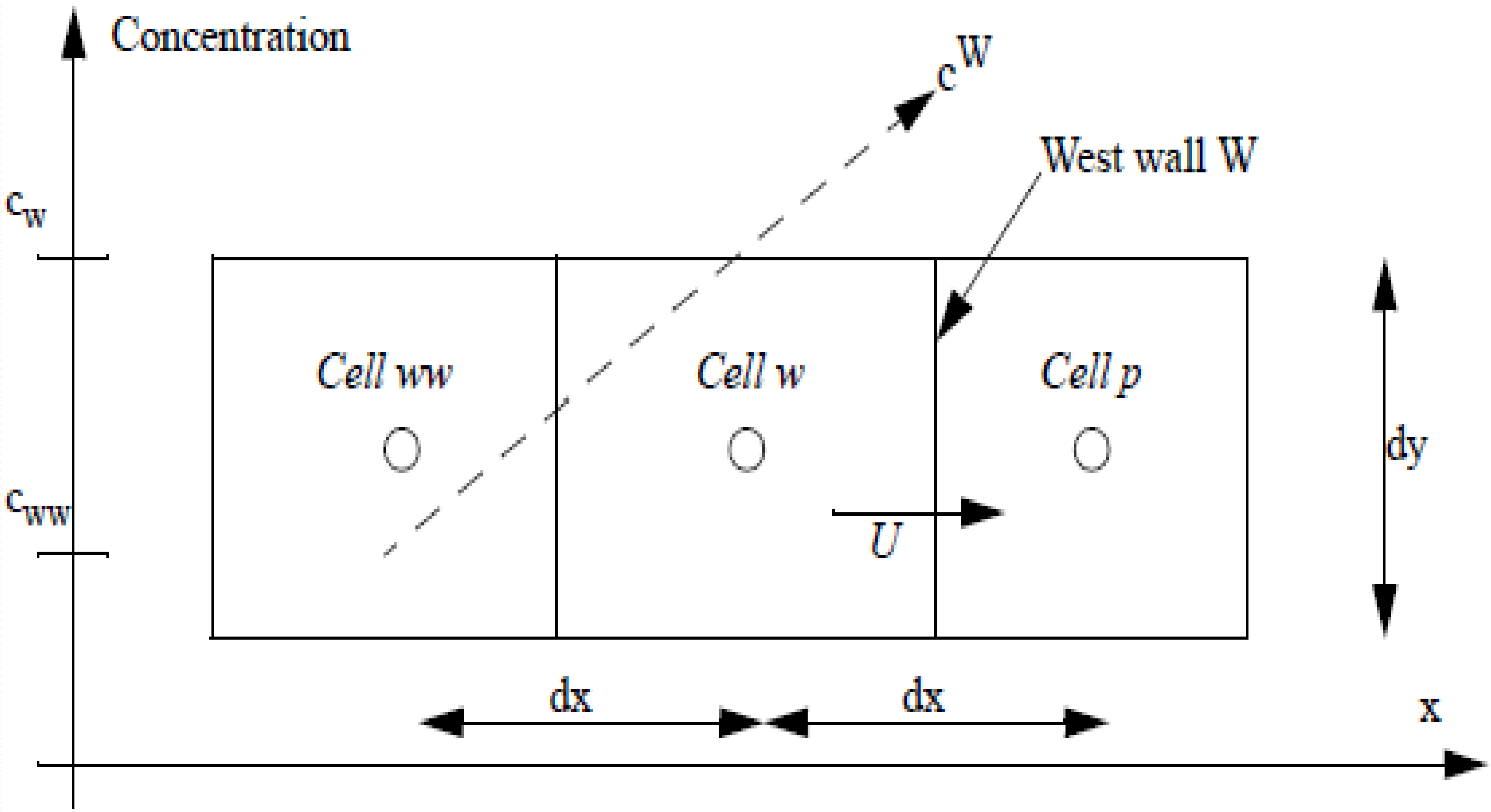
Note that the equations above are only valid if the velocity flows in the same direction as given on the arrows in the figure above.

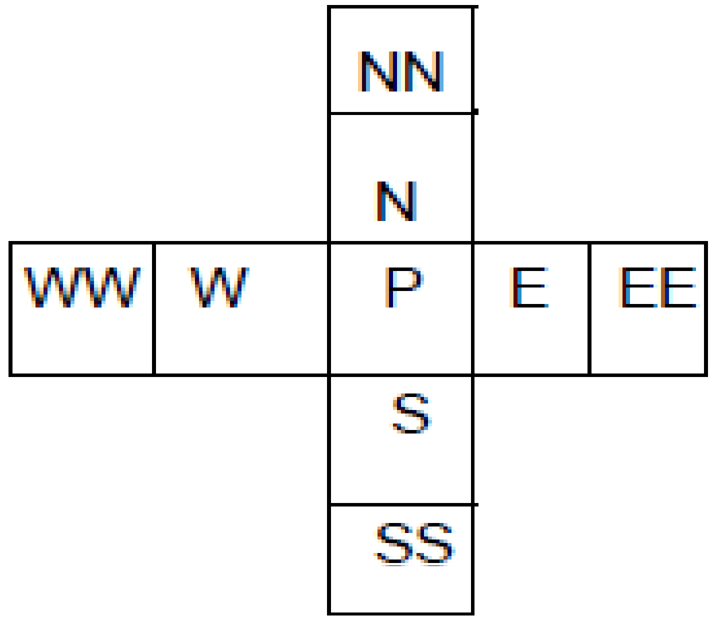
4. Discretization Using the Second-Order Upstream Scheme:

The Second-Order Upstream (SOU) method is based on a second-order accurate method to calculate the concentration on the cell surfaces. The method only involves the convective fluxes. and the diffusive terms are calculated as before



Definition sketch for concentration estimation at the wall for the SOU scheme.





SOU Nine point calculation molecules

Following similar steps as before the weighting factors becomes:

$$
\begin{aligned}
a_w &= \frac{3}{2}U_w A_w + \Gamma_w \frac{A_w}{dx} + \frac{1}{2}U_e A_e \\
a_{ww} &= -\frac{1}{2}U_w A_w \\
a_e &= \Gamma_e \frac{A_e}{dx} \\
a_{ee} &= 0 \\
a_n &= \frac{3}{2}U_n A_n + \Gamma_n \frac{A_n}{dy} + \frac{1}{2}U_s A_s \\
a_{nn} &= -\frac{1}{2}U_n A_n \\
a_s &= \Gamma_s \frac{A_s}{dy} \\
a_{ss} &= 0
\end{aligned}
$$

and $$c_p = \frac{a_w c_w + a_e c_e + a_n c_n + a_s c_s + a_{ww} c_{ww} + a_{nn} c_{nn}}{a_p}$$

The formula is used for a two-dimensional situation. In 3D, the terms for top and bottom is also added, giving four extra coefficients: a_t, a_{tt}, a_b, a_{bb}.

1. INTRODUCTION TO CFD

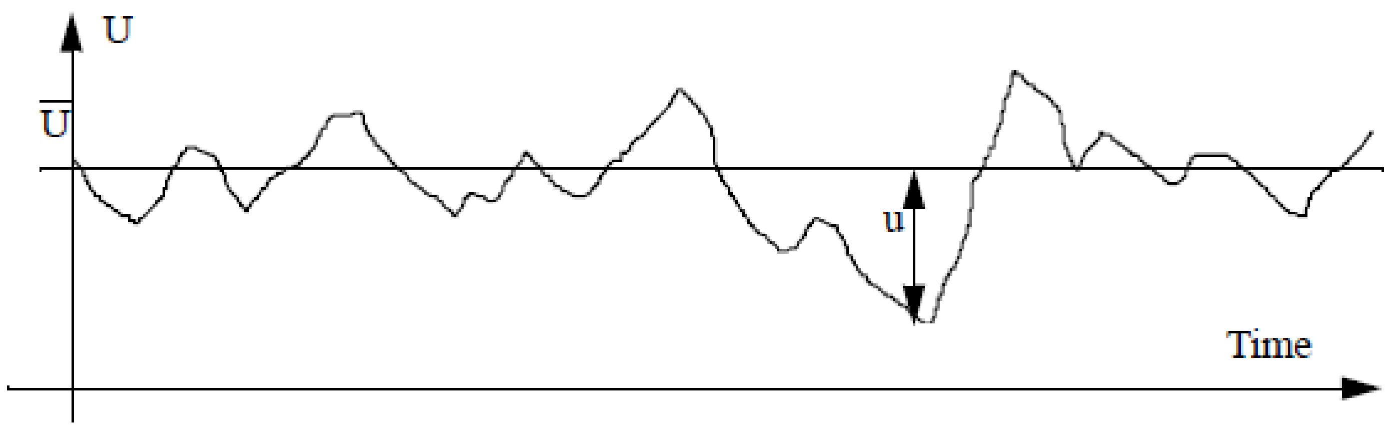
- What is CFD
- Areas of Applications
- Advantages of CFD
- How does CFD Works
- Problem Solving with CFD
- The Convection - Diffusion Equation
- The Navier-Stokes Equations

The Navier-Stokes Equations

The Navier-Stokes equations are set of coupled differential equations and could, in theory, be solve for a given flow problem by using methods from calculus. But, in practice, these equations are too difficult to solve analytically. In the past, engineers made further approximation and simplification to the equation set until they had a group of equations that they could solve. Presently, fast computers are being used to solve approximations to the equations using a variety of techniques like finite difference, finite volume, finite element, and spectral methods, This area of study is called Computational Fluid Dynamics or CFD.

The Navier-Stokes Equations describe how the velocity, pressure, and density of a moving fluid are related. The equations were derived independently By G.G Stokes, in England, and M. Navier, in France, in the early 1800‹s. The equation are extensions of the Euler Equation and include the effects of viscosity on the flow.

We are looking at a time series of the velocity at a given location in turbulent flow:



The velocity is divided into an average value U, and a fluctuating value u. The two variables are inserted into the Navier-Stokes equation for laminar flow, and after some manipulations and simplification the Navier-Stokes equation for turbulent flow emerges:

The Navier-Stokes Equation for Turbulent flow

$$\boxed{\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho}\frac{\partial}{\partial_j}(-P\delta_{ij} - \rho\overline{u_i u_j})}$$

P is the pressure and δ_{ij} is the Kronecker delta, which is 1 if i=j and 0 if i≠ j. The last term is the Reynolds stress term, often modelled with the Boussinesq' approximation:

$$\boxed{-\rho\overline{u_i u_j} = \rho\nu_T\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\rho k\delta_{ij}}$$

Where k is the turbulent kinetic energy.

1. Transient term

2. Convective term

$$\boxed{\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[-\left(P + \frac{2}{3}k\right)\delta_{ij} + \nu_T \frac{\partial U_i}{\partial x_j} + \nu_T \frac{\partial U_j}{\partial x_i}\right]}$$

3. Pressure /kinetic energy term

4. Diffusion term

5. Stress term

The Convective and the diffusive terms

The convective and diffusive term are solved with the same methods as the solution of the convection-diffusion equation for sediment transport. The difference is that the sediment concentration is replaced by the velocity.

Also the other difference between the equation above and the convection-diffusion equation for sediment is the diffusion coefficient. Here an eddy-viscosity ν_T is included instead of the diffusion coefficient Γ. The relation between the two variables is:

$$\nu_T = S_c \Gamma$$

where Sc is the Schmidt number. This is usually set to unity, meaning *the eddy-viscosity is the same as the turbulent sediment diffusivity.*

The stress Term

has very little effect and usually neglected in many cases. The reason is that in most hydraulic engineering flow fields there is usually one dominant flow direction

Pressure /kinetic energy term

This term is solved as a pressure term. The kinetic energy is usually very small, and often negligible compared with the pressure.

Several methods exist to solve the pressure term. With the control volume approach, the most commonly used method is the SIMPLE method.

SIMPLE is an abbreviation for Semi-Implicit Method for Pressure-Linked Equations. The purpose of the method is to find the unknown pressure field. The main idea is to guess a value for the pressure and use the continuity defect to obtain an equation for a pressure-correction. When the pressure-correction is added to the pressure, water continuity is satisfied.

The End

Thank you