

# Lecture 08

## CFD: Numerical Modeling Issues

J.D. Lavers

Electrical & Computer Engineering  
University of Toronto

# Motivation

- **Previous Lectures** – The previous lectures have largely been devoted to the development of models appropriate for turbulent fluid flow.
  - The flow equations, and associated turbulence models, are highly nonlinear and tightly coupled.
  - Little attention has been given to the numerical issues associated with the solution of these systems of equations and models.
- Practical issues associated with the numerical solution of flow equations are addressed in this lecture.

# Objectives in Lecture 08

- The primary objective in this lecture is to review several numerical modeling issues that have an impact on fluid flow simulation.
- The focus will be entirely on the solution of RANS flow models.
- Attention will be given to the special issues associated with flow modeling, either laminar or turbulent:
  - The use of the Finite Volume Method.
  - The treatment of the nonlinear convection (or inertial) term.
  - The treatment of the pressure term.
  - Grid related issues.
  - Wall related issues

# Presentation Outline

- Motivation, Objectives, References
- Finite Volume Method – An Overview
- Finite Volume Method – Details
- Three Factors that Complicate CFD
  - Grid Issues
  - Convection
  - Pressure
- The SIMPLE Algorithm
- Some Other Numerical Issues in CFD
- Summary

# References

- S.V. Patankar, *Numerical Heat Transfer and Fluid Flow*, New York: Hemisphere Publishing Corp., 1980.
- E. Dick, “Introduction to Finite Volume Techniques in Computational Fluid Dynamics”, in J.F. Wendt (ed.), *Computational Fluid Dynamics, An Introduction, 2<sup>nd</sup> ed.*, Chapter 2, Berlin: Springer, 1996.
- H. Lomax, T.H. Pulliam, D.W. Zingg, *Fundamentals of Computational Fluid Dynamics*, Berlin: Springer, 2001.
- V.K. Garg (ed.), *Applied Computational Fluid Dynamics*, Chapter 3, New York: Marcel Dekker, 1998.

# Computational Methods in CFD

## Overview - Finite Volume Method

# Numerical Solution Methods for CFD

- **Finite Difference (FD)**. (see Chapters 5 & 7, Wendt)
  - Widely used in much of early work.
  - Discretization of partial differentials on relatively structured grid.
  - Very good representation of flux related quantities.
- **Finite Element (FE)**. (see Chapter 10, Wendt)
  - Strong and rigorous foundation but less widely used for CFD.
  - Unstructured grids excellent for complex problems.
  - Solution obtained in weighted residual sense; conservation not explicitly satisfied at each element.
- **Finite Volume (FV)**. (see Chapter 11, Wendt)
  - Most widely used numerical method for complex CFD problems.
  - Has advantages of both FDM and FEM.
  - Flux related, and can use unstructured grids.

# Finite Volume Methods – General I

- **Finite Volume (FV) Methods** are applied to the **integral form** of the governing equations for a particular physical process. For example, the conservation law is:

$$\frac{\partial}{\partial t} \int_{\Omega} \phi d\Omega + \oint_S \vec{F} \cdot \hat{n} dS = \int_{\Omega} Q d\Omega$$

- FV Methods are widely used to model **Computational Fluid Dynamic (CFD)** problems:
  - They ensure that the discretization is **conservative**.
  - This property is important and is natural to the FV discretization.
  - A co-ordinate transformation is NOT required when the FVM is applied to irregular meshes.
    - Complex shapes can be modeled with unstructured meshes (arbitrary polyhedra in 3D, arbitrary polygons in 2D).

# Finite Volume Methods – General II

## ■ FEM-like Finite Volume Techniques.

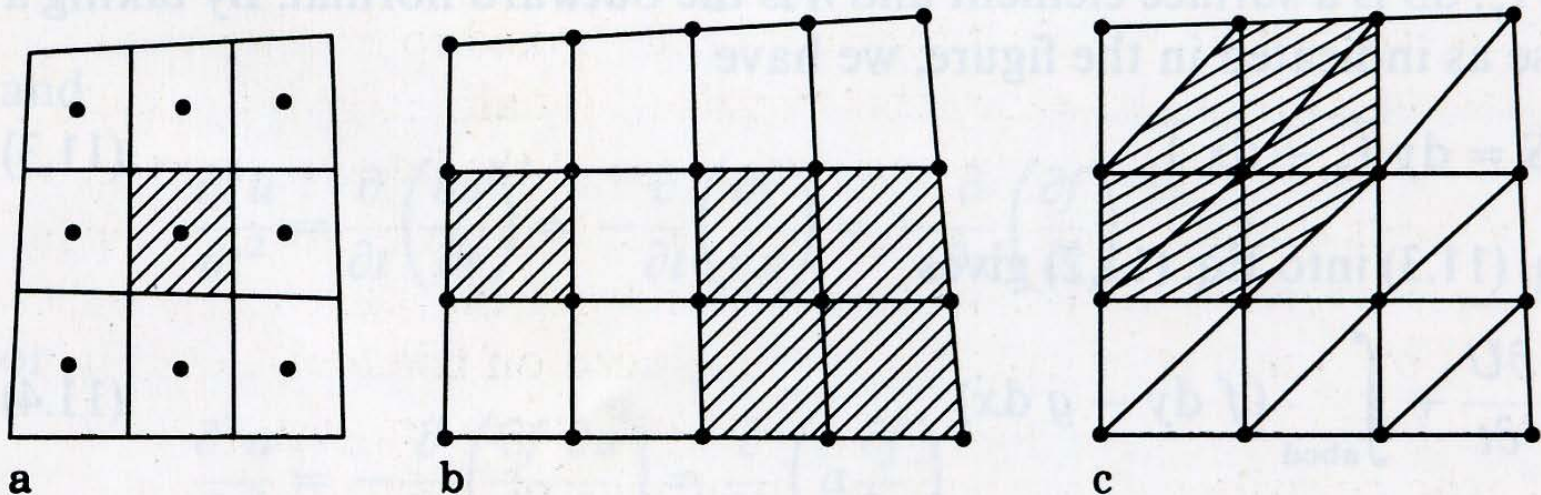
- Use cells to which an interpolation structure is associated.
  - Piecewise constant (*cell centered*) interpolation.
    - Values of dependent variable “stored” at center of cell.
  - Piecewise linear (*cell vertex*) interpolation.
    - Values of dependent variable stored at grid vertices.

## ■ FDM-like Finite Volume Techniques.

- Nodes are explicitly at vertices of grid.
- Useful in treating boundaries (particularly with regard to pressure term).
- Considerable freedom in definition of flux through use of overlapping grids.

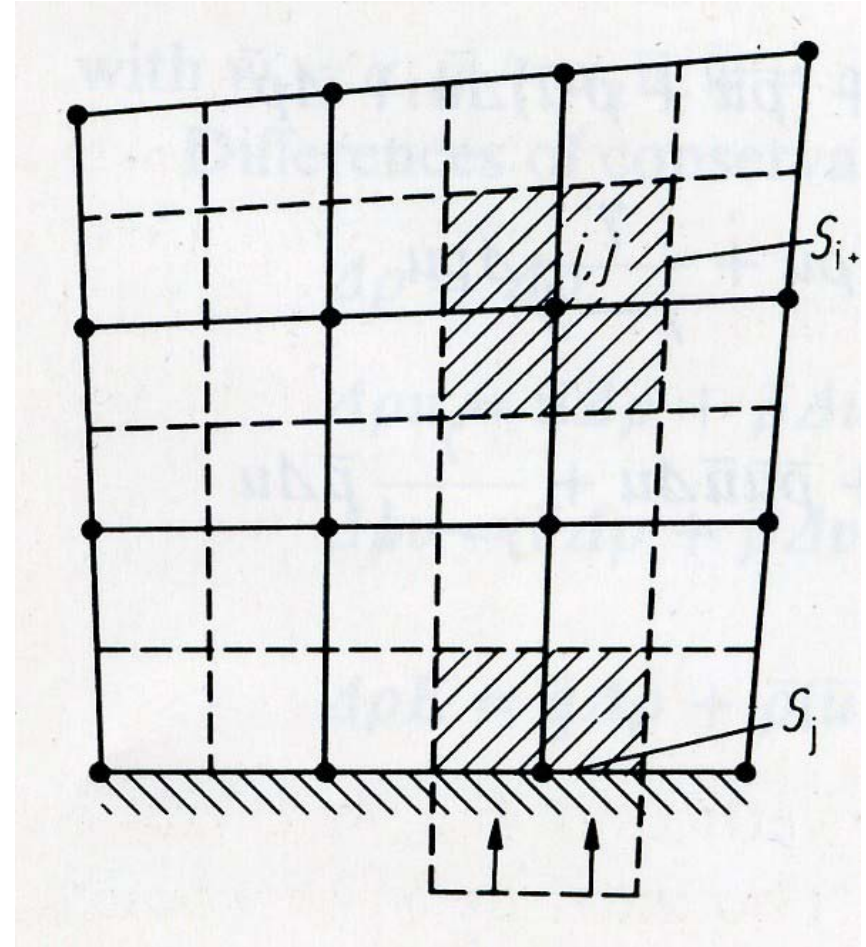
# FEM-like Finite Volumes

- Figure below shows (a) cell centered FV, (b) cell vertex FV on quadrilateral cells, (c) cell vertex FV on triangular cells.



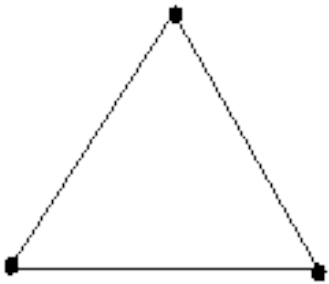
# FDM-like Finite Volumes

- Primary grid: defined by node vertices shown as •.
- Cell-centers define secondary grid, interwoven with primary grid.
  - Cells of secondary grid can serve as control volumes for primary nodes inside them.
  - Control volume establishes flux balance.
  - Primary and secondary grids can be used for different variables.
  - See next slide.

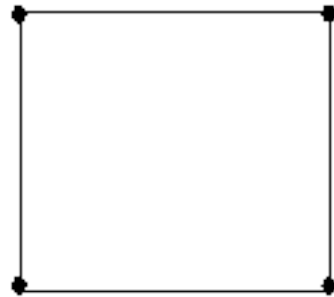


# Representative Cell Types

## 2D Cell Types

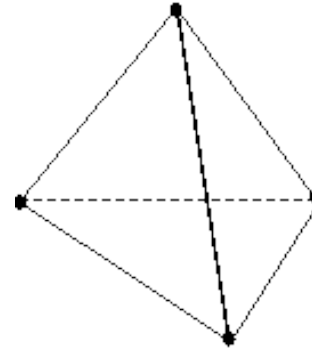


**Triangle**

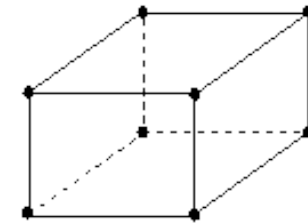


**Quadrilateral**

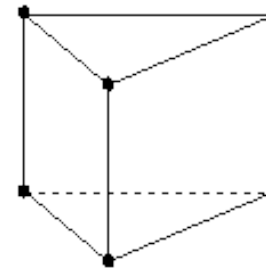
## 3D Cell Types



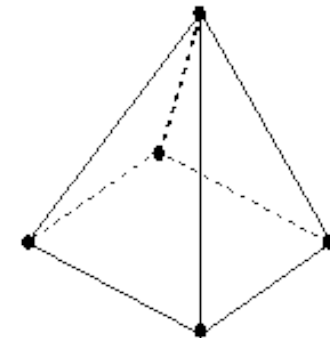
**Tetrahedron**



**Hexahedron**

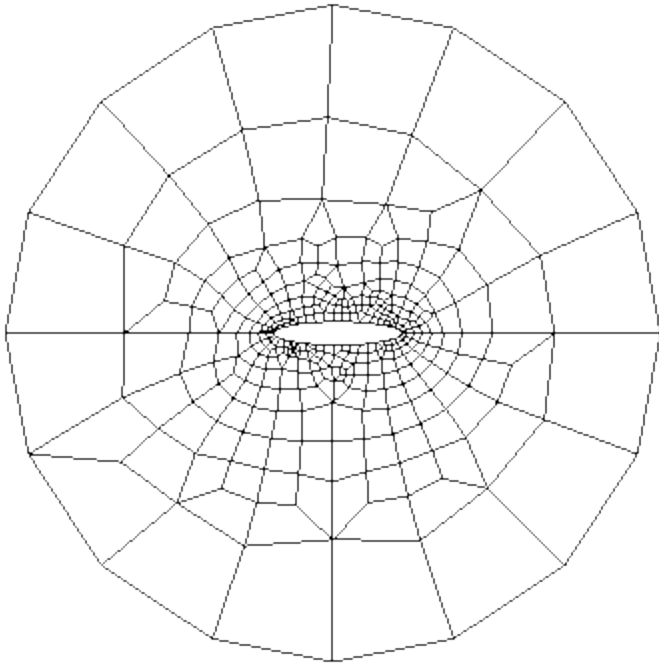


**Prism/Wedge**

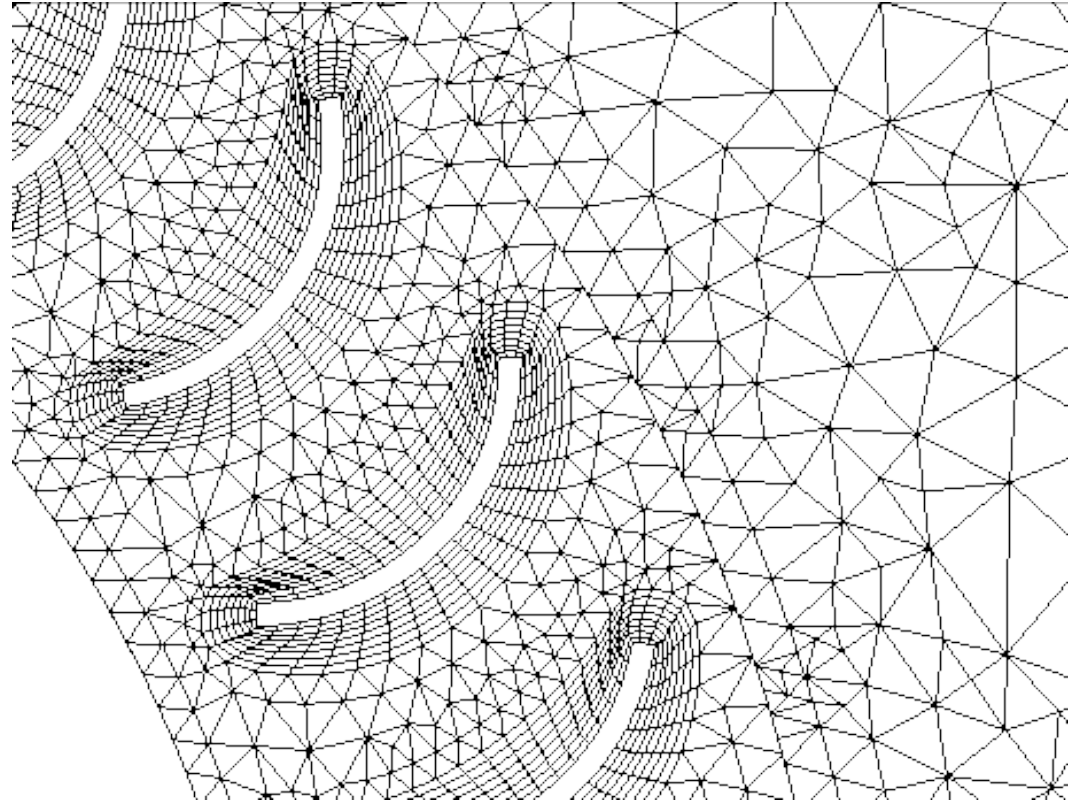


**Pyramid**

# Representative Solution Grids



Unstructured quadrilateral



Hybrid grid

# Computational Methods in CFD

## Some Details – Finite Volume Method

# Finite Volume Methods – Basic Concepts I

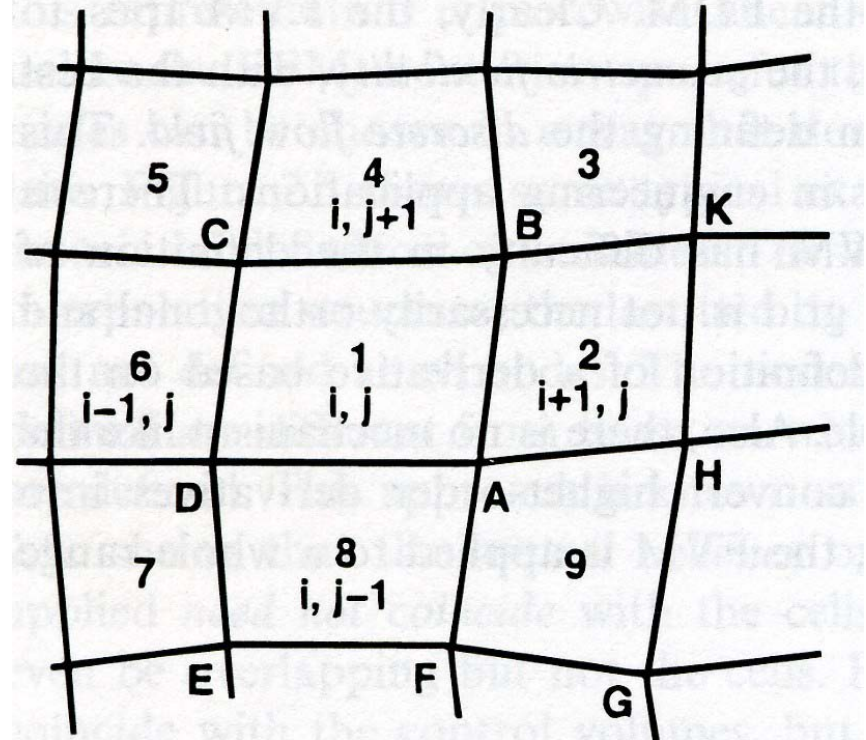
- Integral form of conservation law for variable  $\phi$  is:

$$\frac{\partial}{\partial t} \int_{\Omega} \phi d\Omega + \oint_S \vec{F} \cdot \hat{n} dS = \int_{\Omega} Q d\Omega$$

- When applied to a discretized control volume, as shown in figure to the right:

$$\frac{\partial}{\partial t} (\phi_a \Omega_a) + \sum_{\text{sides}} (\vec{F} \cdot \hat{n} dS) = Q_a \Omega_a$$

- $\Omega_a$  is the discrete control volume (e.g. the volume labeled 1 in figure).
- $\phi_a = \phi_{ij}$  for the control volume selected in the figure.



# Finite Volume Methods – Basic Concepts II

- To illustrate the most basic of FV formulations, consider consider the 2D heat transfer equation:

$$\rho c_p \frac{\partial T}{\partial t} = \vec{\nabla} \cdot k \vec{\nabla} T + w$$

- Averaging over a control volume  $\Omega_a = \Omega_{ij}$ , centered about the node (i,j), and assuming the  $\Omega_{ij}$  is sufficiently small that  $T, \rho, c_p$  and  $w$  are constant within the control volume:

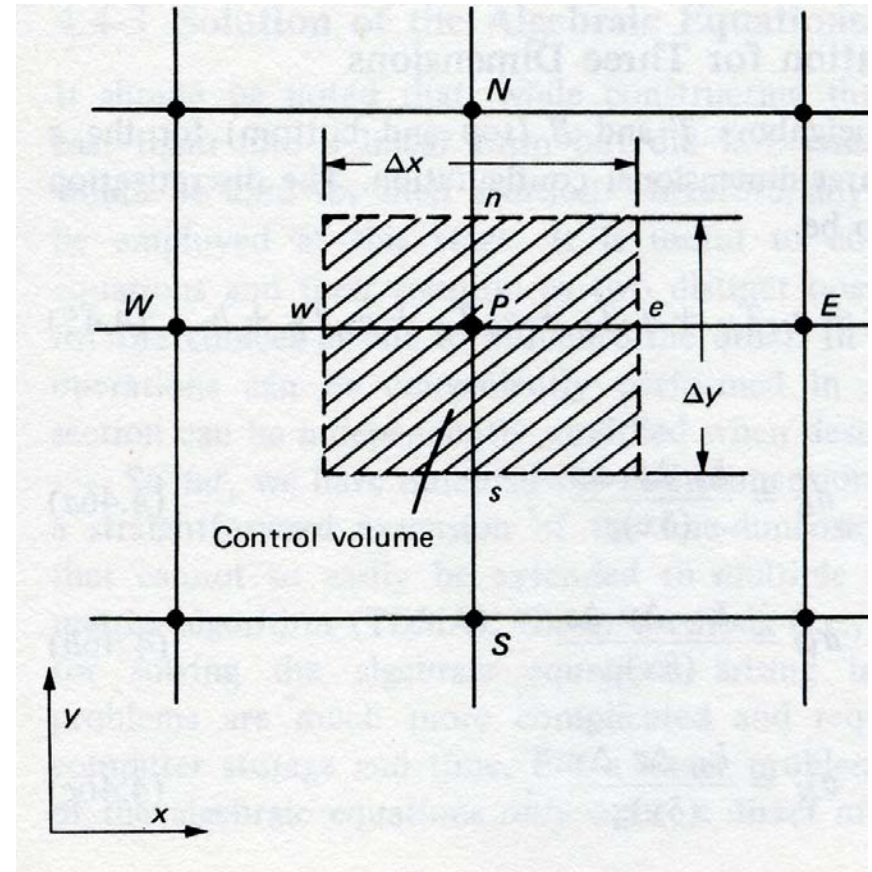
$$\rho c_p \frac{\partial T_{ij}}{\partial t} \Omega_{ij} = \int_{\Omega_{ij}} (\vec{\nabla} \cdot k \vec{\nabla} T) d\Omega + (w_{ij} \Omega_{ij}) = \oint_{S_{ij}} k \vec{\nabla} T \cdot \hat{n} dS + (w_{ij} \Omega_{ij})$$

- This is clearly in the stand FV form.

# Finite Volume Methods – Basic Concepts III

- Consider the shaded FV shown in the figure:
  - T is constant in the shaded area.
  - Material parameters are constant in each of the volumes surrounding node P.
- Relative to the node P:

$$\rho c_p \frac{\partial T_P}{\partial t} \Omega_P = \frac{\partial T_P}{\partial t} \frac{1}{4} [(\rho c_p)_{P,NE} + (\rho c_p)_{P,NW} + (\rho c_p)_{P,SW} + (\rho c_p)_{P,SE}] \Delta x \Delta y$$



# Finite Volume Methods – Basic Concepts IV

- Similarly, the heat flux through the FV surface between node P and node E is:

$$\int k \vec{\nabla} T \cdot \hat{n} dS = \left( \frac{k_{P,NE} + k_{P,SE}}{2} \right) \left( \frac{T_E - T_P}{\Delta x} \right) \Delta y$$

- It is a simple matter to extend the formulation to a non-uniform rectangular grid.
- By considering all sides of the FV, together with the source term, and introducing a time discretization, and algebraic equation of the following form is obtained:

$$a_P T_P - a_E T_E - a_W T_W - a_N T_N - a_S T_S = b_P$$

# Three Factors that Complicate CFD

Grid Related Issues

The Nonlinear Convection Term

The Role of Pressure

The SIMPLE Algorithm

# Reminder: Forms of the Momentum Eq<sup>n</sup>

- Before discussing numerical issues, it is worthwhile to recall the forms of the momentum equation:

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

$$\rho \frac{\partial \vec{u}}{\partial t} + \vec{\nabla} \cdot (\rho \vec{u} \vec{u}) = -\vec{\nabla}p + \vec{\nabla} \cdot \vec{\tau} + \vec{f}$$

- Note, in particular, the inertia and the pressure terms.

# Reminder: The Transport Equation

- More generally, the transport equation for any quantity  $\Phi$  can be written as:

$$\frac{\partial(\rho\Phi)}{\partial t} + \vec{\nabla} \cdot (\rho\vec{u}\Phi) = C_{\Phi} \vec{\nabla} \cdot \Gamma_{\Phi} \vec{\nabla}\Phi + G_{\Phi} - S_{\Phi}$$

- $C_{\Phi}$  is a general constant (equal to unity in the Equation of Motion).
  - $\Gamma_{\Phi}$  is a material property (equal to viscosity  $\mu$  in the equation of Motion).
- This basic equation was used to describe energy, as well as turbulence energy, turbulence dissipation and Reynolds stresses.

# Three Factors that Complicate CFD

- CFD problems are characterized by the following three complicating factors:
  - The convection of momentum is nonlinear in velocity and requires careful attention.
    - Convection, in general, can lead to instability when modeling many physical problems; e.g. moving conductors in B-fields.
  - The momentum equation contains a term that is derived from the pressure  $p$ , and there is no explicit relationship to determine  $p$ .
  - Due to pressure, plus the continuity equation, overlaying grids must be used to maintain consistent solution accuracy

# Three Factors that Complicate CFD

## Grid Related Issues

The Nonlinear Convection Term

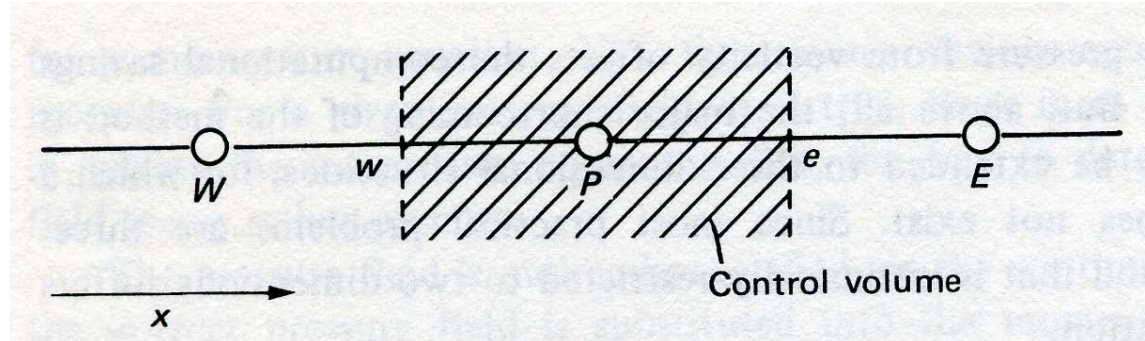
The Role of Pressure

The SIMPLE Algorithm

# Grid Related Issues I

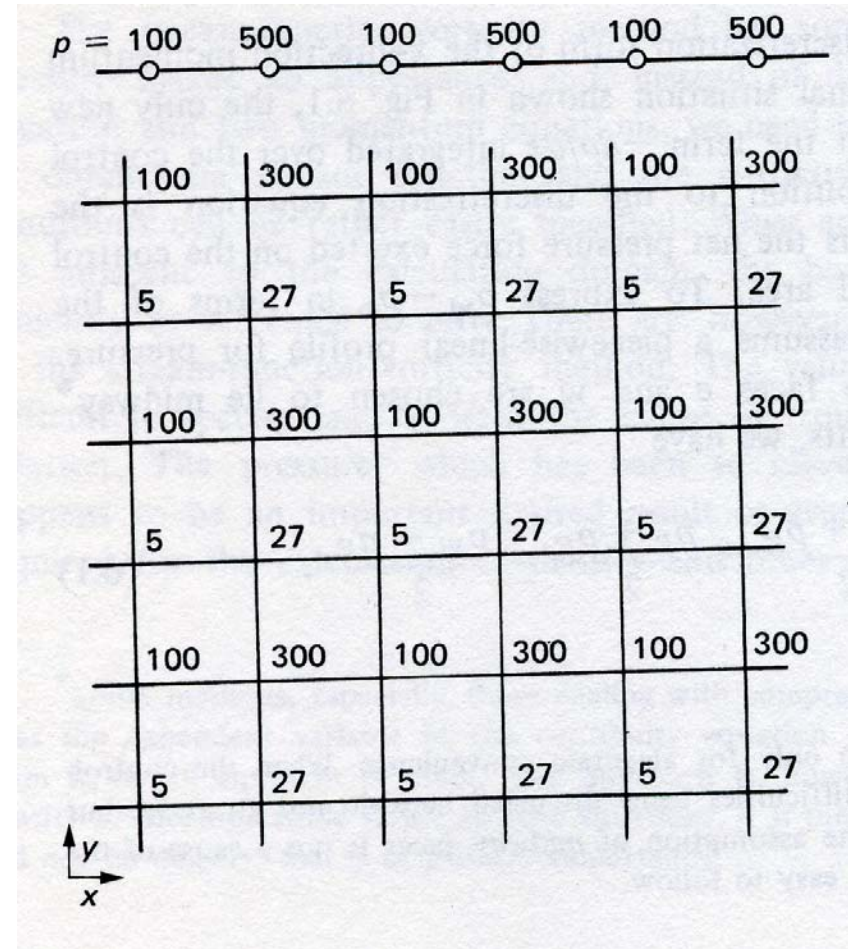
- With both FV and FD, use of a single regular grid can result in loss of accuracy for certain variables and/or equations.
- Consider the pressure gradient  $-dp/dx$  for the simple 1D control volume shown below.

$$\int_{\Omega_p} -\frac{dp}{dx} d\Omega = -\left( \frac{(p_E + p_P)/2}{\Delta x} - \frac{(p_P + p_W)/2}{\Delta x} \right) \Delta x \Delta y = -\frac{(p_E - p_W)}{2} \Delta y$$



# Grid Related Issues II

- Associating pressure with the same nodes that will be used for velocity means that the differential is NOT taken with respect to adjacent nodes.
  - A single fixed grid implies loss of accuracy when dealing with pressure gradients.
- Figures to right show serious implication when this differencing scheme is used.
  - Pressure field highly irregular.
  - Gradient is zero, so this pressure has no impact on momentum!



# Grid Related Issues III

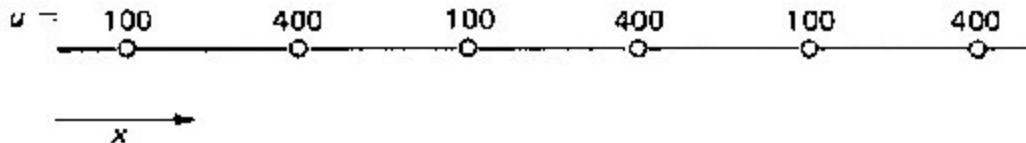
- The same grid problem arises with the FV approximation to the incompressible continuity equation:

$$\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} = 0$$

- In the simple 1D case, continuity implies:

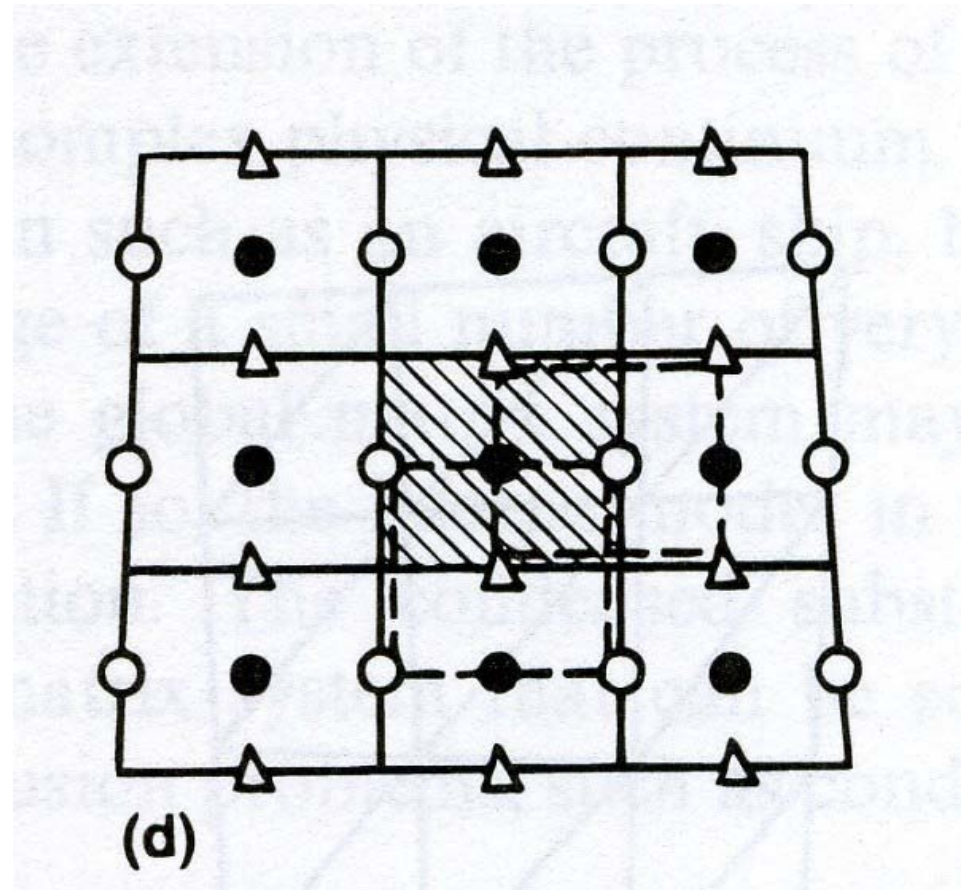
$$u_{x,E} - u_{x,W} = 0$$

- In other words, in 1D continuity imposes equality of velocities at alternate nodes, not at adjacent nodes as the physics would impress.



# Solution – The “Staggered Grid”

- Not all variables need be computed at the same set of grid points – see figure to right.
- The velocity components are calculated on the faces of the control volume.
- The pressures are calculated at the control volume centers.
- With a staggered grid, first order differences apply where they are needed;
  - e.g. pressure differentials act at the surface of the control volume.



# Three Factors that Complicate CFD

Grid Related Issues

**The Nonlinear Convection Term**

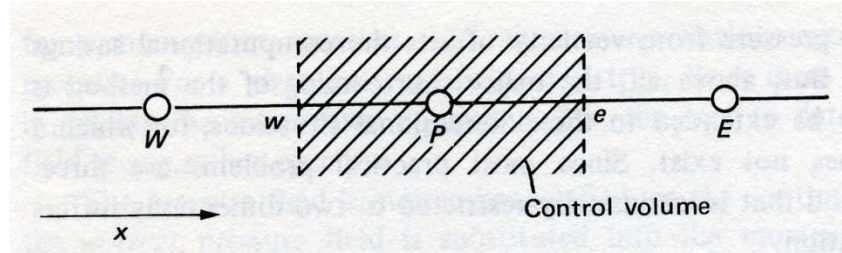
The Role of Pressure

The SIMPLE Algorithm

# Convection Related Issues I

- The presence of the **convection term** can lead to an unacceptable discretization.
- To illustrate, consider a very simple 1D convection-diffusion problem: 
$$\frac{d}{dx}(\rho u_x \phi) = \frac{d}{dx} \left( \Gamma \frac{d\phi}{dx} \right)$$
- Using the grid shown below, the following FV discretization is obtained:

$$(\rho u_x \phi)_e - (\rho u_x \phi)_w = \left( \Gamma \frac{d\phi}{dx} \right)_e - \left( \Gamma \frac{d\phi}{dx} \right)_w$$



# Convection Related Issues II

- Assuming  $\phi_e = (\phi_E + \phi_P)/2$ ,  $\phi_w = (\phi_W + \phi_P)/2$ , the discrete FV equation at node P is:

$$F = \rho u_x \quad D = \Gamma / \Delta x$$

$$a_P = a_E + a_W$$

$$a_E = D_e - F_e / 2 \quad a_W = D_w + F_w / 2$$

$$a_P = D_e + F_e / 2 + D_w - F_w / 2 = a_E + a_W + (F_e - F_w)$$

- It is a simple matter to show that, with this equation,  $\phi$  exhibits nonphysical behaviour; e.g. when  $D = F = 1$ :
  - If  $\phi_E = 200$  and  $\phi_W = 100$ , then  $\phi_P = 50$ .
  - If  $\phi_E = 100$  and  $\phi_W = 200$ , then  $\phi_P = 250$ .
  - Neither is possible in a purely convective-diffusive process.
- If  $|F| > 2D$ ,  $a_E$  or  $a_W$  negative  $\rightarrow$  solution may diverge.
  - $|F| < 2D$  limits this approach to low Reynolds Number flow.

# Upwinding I

- Weak point of previous FV scheme: assumption that (convected)  $\phi_e, \phi_w$  were equally weighted averages.

- **Simple Upwind Scheme:**

$$u_x > 0 \Rightarrow F_e, F_w > 0 \Rightarrow \phi_e = \phi_P \text{ and } \phi_w = \phi_W$$

$$u_x < 0 \Rightarrow F_e, F_w < 0 \Rightarrow \phi_e = \phi_E \text{ and } \phi_w = \phi_P$$

- With this scheme,  $a_E$  and  $a_W$  always positive and solution is physically realistic.
- **Simple Physical Interpretation:** Formalizes fact that a convected fluid should retain memory of conditions where it has been, but has no fore-knowledge of conditions where it is heading.

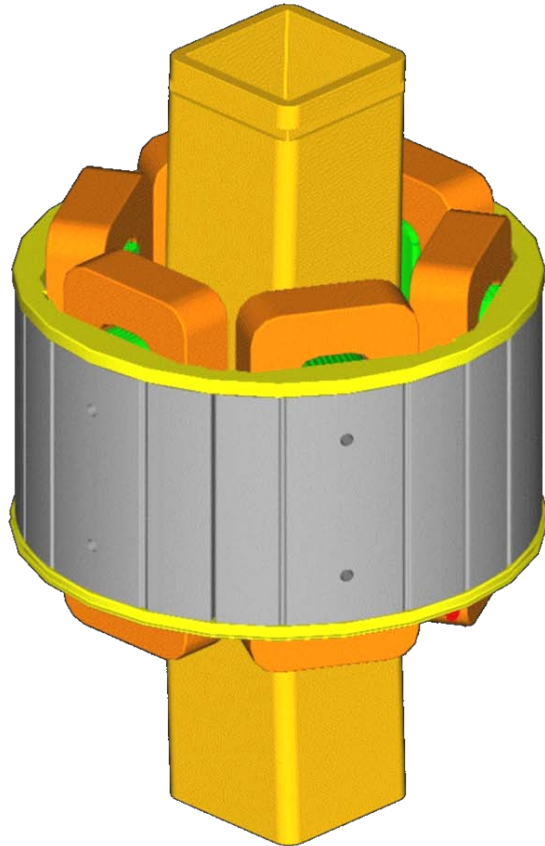
# Upwinding II

- Simple upwinding fails at large Peclet Number  $P$  (ratio of convection to diffusion).
  - $P \gg 1 \Rightarrow$  diffusion almost absent; simple upwinding over estimates diffusion effects.
- Many other upwind schemes exist (e.g. see Patankar, Chapter 5):
  - **The Exponential Scheme** – FV coefficients weighted exponentially on basis of  $F/D$  ratio.
    - In 1D case, guaranteed to produce an exact solution.
    - Not widely used – not exact in 2D or 3D.
  - **The Hybrid Scheme** – No upwinding when  $|P| \leq 2$ ; upwinding with diffusion effects set to zero for  $|P| > 2$ .

# Upwinding III

- **The Power Law Scheme** – Hybrid upwinding fails for  $|P| \cong 2$ . Power Law provides better approximation in this range.
- Similar schemes (first order and high order) have been developed for 2D and 3D problems.

# Impact of Upwindng – M-EMS I



- Consider the In-mold EMS system to the left.
- Steady angular velocity predicted for mid-plane and meniscus:
  - Linear upwinding
  - High Order upwinding.
- Impact on flow also examined for choice of:
  - Reynolds Stress
  - $k-\varepsilon$

# Impact of Upwinding – M – EMS II

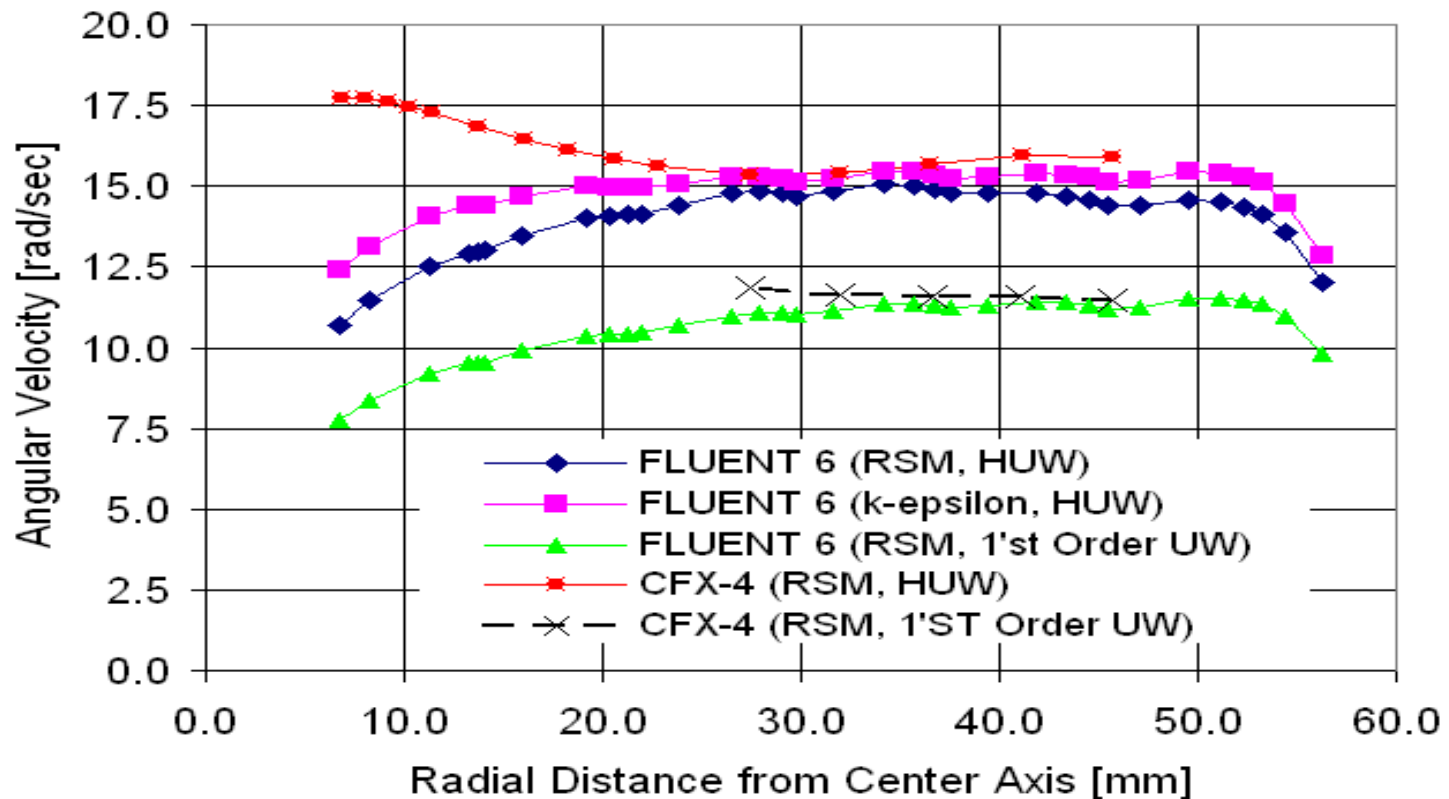
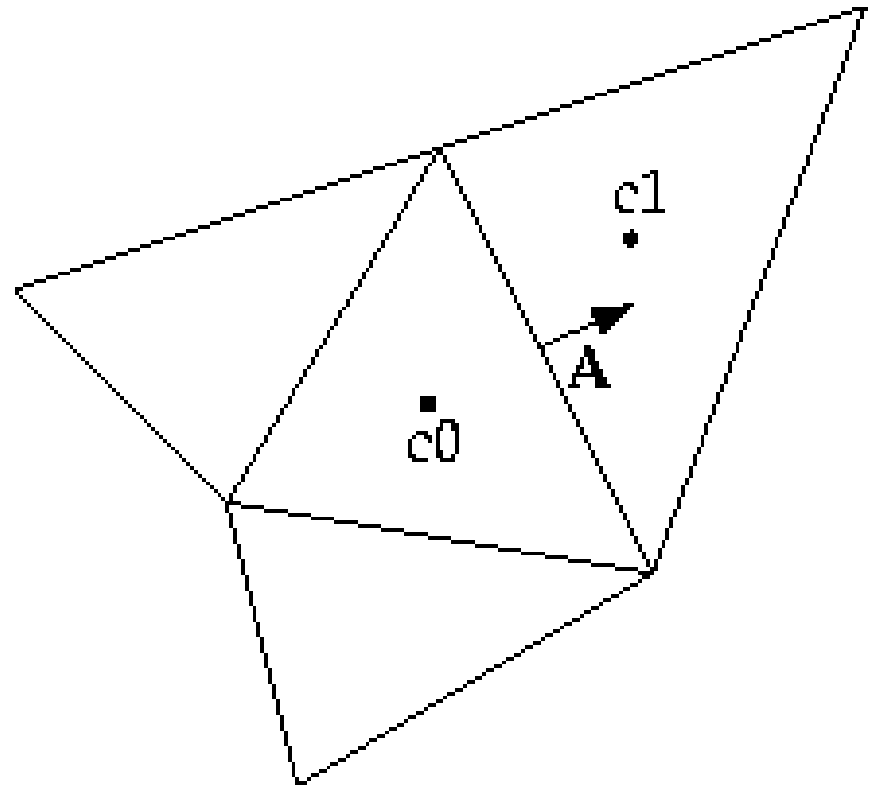


Figure 4. Angular velocity radial distribution predicted at the mid-plane under various model assumptions.

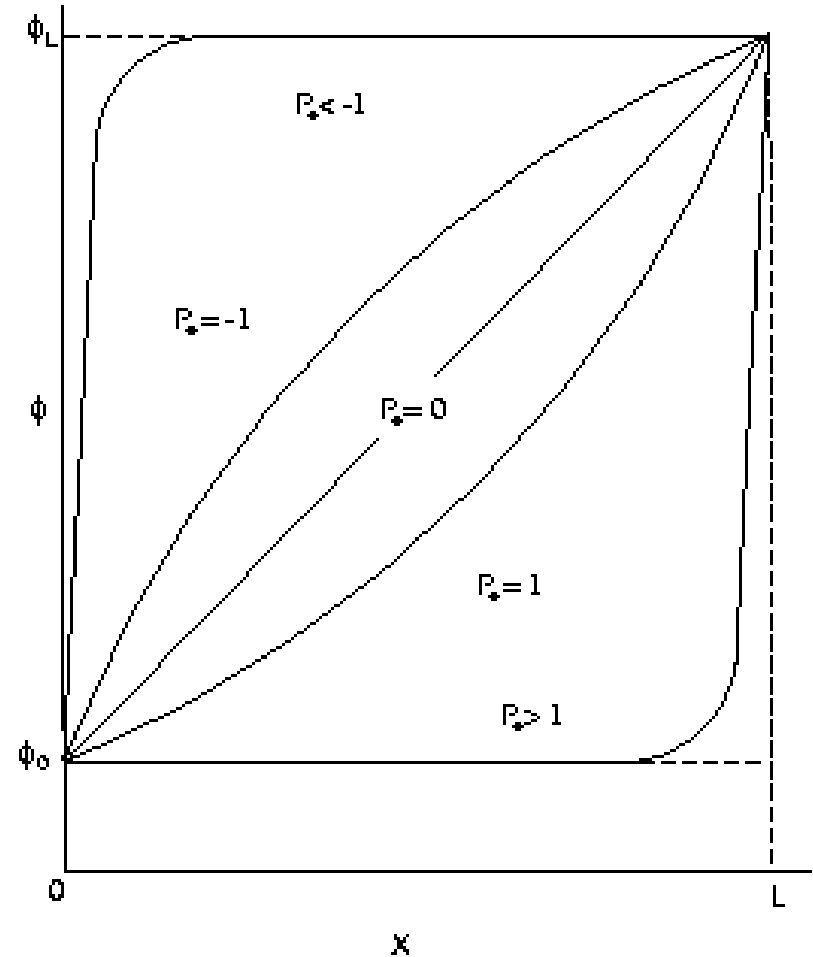
# 2D and 3D Upwinding I

- Consider upwinding for a scalar quantity  $\phi$ . As with 1D:
  - Values of  $\phi$  stored at cell centers ( $c_0$ ,  $c_1$  etc.)
  - Values of  $\phi$  also required at cell faces for convection terms.
  - Required face values **interpolated** from cell centered values using linear or high order upwinding.
  - As with 1D, upwinding means that face value  $\phi_f$  derived from upstream cell; “upwind” w.r.t. normal velocity.



# 2D and 3D Upwinding II

- **First-Order Upwinding:** Face value  $\phi_f$  set equal to cell centered value of  $\phi$  for upstream cell.
- **Power Law Upwinding:** The face value  $\phi_f$  is interpolated using the 1D power law described by Patankar (fig. to right).
  - Pe is the Peclet number (ratio of convection to diffusion).
  - Pe = 0 implies no convection and linear interpolation.



# 2D and 3D Upwinding III

- **Second-Order Upwind:** Face value  $\phi_f$  is obtained by interpolating using a mix of cell centered values and gradients:

$$\phi_f = \phi + \vec{\nabla}\phi \cdot \Delta\vec{\ell}$$

- $\phi$  and  $\nabla\phi$  are cell centered values in upstream cell.
  - $\Delta\vec{\ell}$  is a displacement vector from the upstream cell centroid to the face centroid.
- **Cell-Dependent Schemes:** Many specialized schemes exist that are specifically tailored for particular cells (quadrilaterals, hexahedrals, etc.).

# Numerical or “False” Diffusion I

- An analysis of basic central difference and simple upwind schemes show that:
  - Truncation in central difference scheme  $\rightarrow (\Delta x)^2$ .
  - Truncation in simple upwind scheme  $\rightarrow (\Delta x)$ .
  - Central differences appear to have advantage, **BUT**
  - **Only for very small  $\Delta x$  due to impact of convection/diffusion.**
- **False diffusion:** In simple upwinding, the effective diffusion coefficient is:

$$\Gamma' = \Gamma + (\rho u_x \Delta x) / 2$$

- The second term is referred to as the “**false**” diffusion coefficient.
  - “**False**” diffusion tends to reduce or eliminate certain difficulties at high Peclet number.

# Numerical or “False” Diffusion II

- Clearly, numerical “false” diffusion is most noticeable in convection dominated flow.
- All practical numerical schemes for solving fluid flow problems contain a certain amount of “false” diffusion due to truncation.
- Second order discretization can reduce the “false” diffusion effects.
- Refinement of the mesh also reduces “false” diffusion.
- “False” diffusion reduced when flow is aligned to mesh. Impacts choice of elements: hexahedral vs. tetrahedral.

# Three Factors that Complicate CFD

Grid Related Issues

The Nonlinear Convection Term

**The Role of Pressure**

The SIMPLE Algorithm

# Pressure Related Issues I

- Recall the continuity and momentum equations for a steady incompressible flow:

$$\vec{\nabla} \cdot \vec{u} = 0$$

$$\rho \vec{u} \cdot \vec{\nabla} \vec{u} = -\vec{\nabla} p + \mu \nabla^2 \vec{u} + \vec{f}_b$$

- In solving these equations, the fluid velocity components ( $u_x, u_y, u_z$ ) **and** the pressure  $p$  must be determined.
- One possibility is to cast the solution in terms of a **vorticity** and **stream function** to eliminate  $p$ .
- BUT, CFD models are almost always solved for the velocity components.
  - The corresponding pressure distribution must be determined.

# Pressure Related Issues II

- There is no specific equation from which pressure can be directly determined.
  - Pressure is indirectly specified by the continuity equation.
  - When the correct pressure is substituted in the momentum equation, the resulting velocity satisfies continuity.

- Velocity and pressure are determined iteratively:

$$\bar{u} = \bar{u}^* + \bar{u}' \quad p = p^* + p'$$

- \* Terms: current iterative value
  - ' Terms: iterative correction
- Several iterative algorithms are available.
    - Patankar's SIMPLE algorithm will be used to illustrate.

# Three Factors that Complicate CFD

Grid Related Issues

The Nonlinear Convection Term

The Role of Pressure

**The SIMPLE Algorithm**

# The SIMPLE Algorithm I

- The SIMPLE algorithm is semi-implicit
- Considering the momentum and continuity equations:

$$\rho \vec{u}^* \cdot \vec{\nabla} \vec{u}^* = -\vec{\nabla} p^* + \mu \nabla^2 \vec{u}^* + \vec{f}_b \qquad \vec{\nabla} \cdot (\vec{u}^* + \vec{u}') = 0$$

$$\rho \vec{u}' \cdot \vec{\nabla} \vec{u}' = -\vec{\nabla} p' + \mu \nabla^2 \vec{u}'$$

- The momentum equation is split into \* and ' components.
- The continuity equation is written in terms of the most recent total estimate - iterative value plus update – of velocity.
- **STEP 1:** Given  $p^*$ , the \* version of the momentum equation can be solved for the \* components of velocity.

# The SIMPLE Algorithm II

- **STEP 2:** When cast in discrete form, the ' version of the momentum equation can be cast in a form that explicitly related the “primed” velocity components an  $p'$ :

$$\bar{u}' = g(p')$$

- **STEP 3:** Given the form of  $g(p')$ , the continuity equation can be used to determine the pressure correction:

$$\vec{\nabla} \cdot g(p') = -\vec{\nabla} \cdot \bar{u}^*$$

- **STEP 4:** The values of velocity and pressure are updated and the iteration proceeds.

$$p^* + p' \rightarrow p^* \quad \bar{u}^* + \bar{u}' \rightarrow \bar{u}^*$$

- The various algorithms to treat pressure primarily differ in the definition of  $g(p')$ , and in the interpolation scheme used.

# Other Numerical Issues in CFD

Choice of Model and Solution  
Near Wall Behaviour

# Choosing a Turbulence Model

- The choice between RANS and LES depends on the application:
  - Is mixing important?
  - Is quantitative information on turbulence important?
  - Resources available.
- In the case of RANS, options range from single (transport) equation models, through the variations of  $k$ - $\epsilon$ , to 7 equation Reynolds stress models.
  - Is the flow confined (any of the induction furnace applications), or does it involve significant momentum transport (as in continuous casting)?
  - Choice guided by application **AND** available experimental data.

# Steady State vs. Transient Solution

- RANS models of turbulent flow are, by definition, steady state models.
  - The computed velocities are the time averaged values of  $\mathbf{u}$ .
- The time derivatives in the momentum and transport equations are effectively zero.
- Solution options available are either “steady”, or “transient”.
  - The steady option iterates directly to the steady state.
  - The transient option iterates/integrates through a “transient” from a prescribed initial state.

# Steady State Solution

- With the “steady” option, the solver attempts to iterate directly to the steady state.
- It may be necessary to adjust the default under relaxation factors.
- It is absolutely necessary to obtain good starting estimates for  $k$  and  $\varepsilon$ .
- A useful strategy to estimate  $k$  is to set the initial turbulent intensity to the 5-10% range.  $K$  can be back calculated.
- The initial estimate of  $\varepsilon$  should be such that the eddy viscosity  $C_\mu k^2/\varepsilon$  is in the order of unity.

# Transient Solution

- Our experience is that the transient option proves to be **much more robust** than the steady solution approach.
- The steady solution is very sensitive to problem configuration changes.
- With the transient option, it is a simple matter of proceeding (through false transient), from a zero state, to the steady.
- The solution is relative insensitive to the assumed time step.
- Total “solution time” should match the actual time it takes the flow to reach steady state.

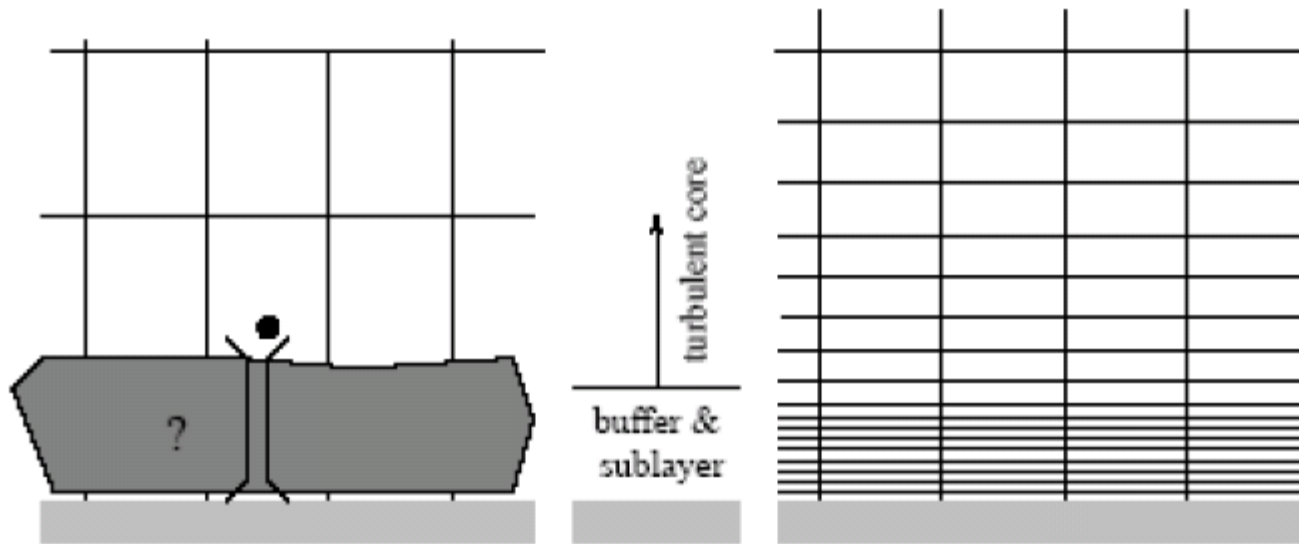
# Convergence

- Typically, convergence criteria on the residuals (velocity,  $k$  and  $\varepsilon$ , the Reynolds stresses, the turbulence intensity) are set to  $10^{-4}$ .
- The resulting solutions seem to have reasonable consistency.
- Typically, in the order of 50 nonlinear iterations are allowed per “time” step.

# Near Wall Treatment

- Fully developed turbulence: rapid velocity variation from turbulent core to no slip behavior at walls.
- Treatment of transition in boundary layer:
  - Use a graded grid.
  - Model the boundary layer.
- Near wall model significantly impacts quality of solution.
- Solution grid, regardless, must be compatible with near wall model that is used.

# Near Wall Modeling Options



## Wall Function Approach

- The viscosity-affected region is not resolved, instead is bridged by the wall functions.
- High-Re turbulence models can be used.

## Near-Wall Model Approach

- The near-wall region is resolved all the way down to the wall.
- The turbulence models ought to be valid throughout the near-wall region.

# Recall: Wall Function for Near Wall Model

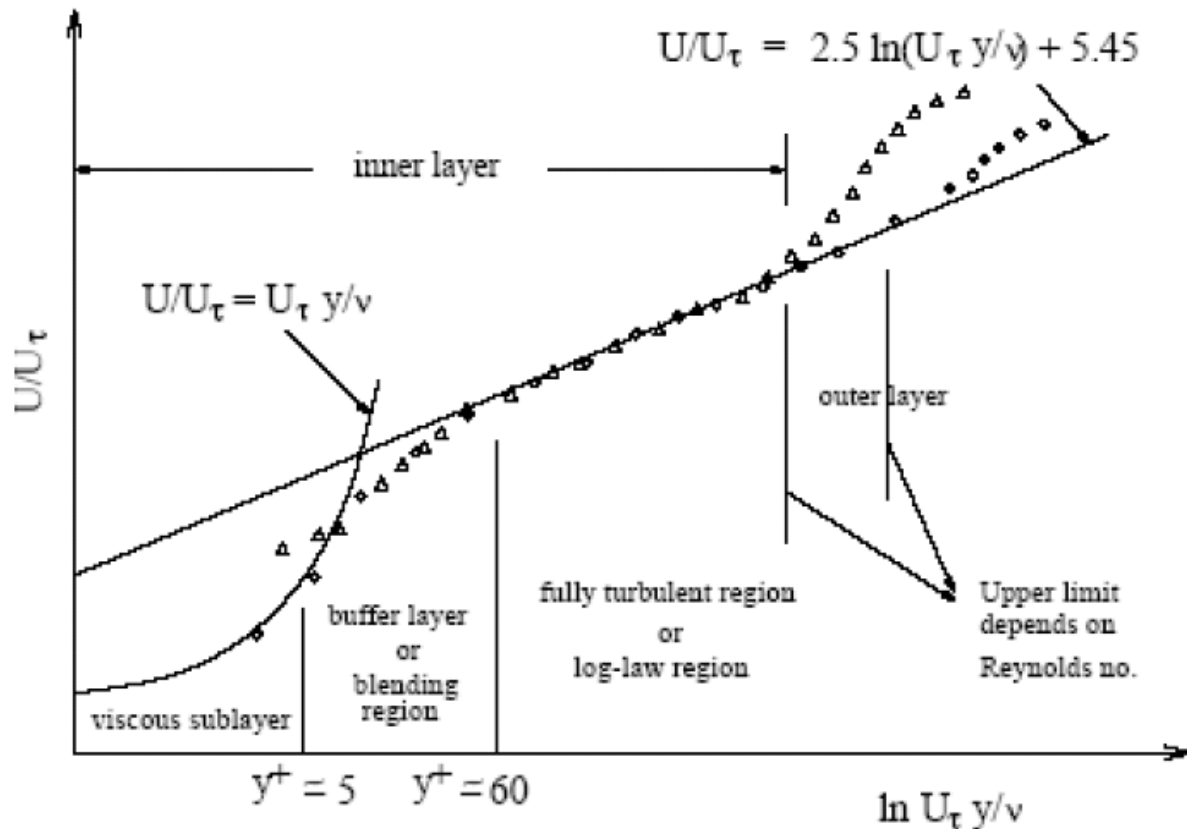


Figure 12.10.1: Subdivisions of the Near-Wall Region

# Recall: Universal Wall Function

- The wall function parameters have been used to estimate a Universal Wall Function for pipe flow (J. Nikuradse, *Ver. Dsch. Ing. Forsch.*, **356**, 1932):

$$u^+ = y^+ \quad \text{for } 0 \leq y^+ \leq 5 \quad (\text{laminar sublayer})$$

$$u^+ = -3.05 + 5.0 \ln y^+ \quad \text{for } 5 \leq y^+ \leq 30 \quad (\text{buffer layer})$$

$$u^+ = +5.50 + 2.5 \ln y^+ \quad \text{for } y^+ > 30 \quad (\text{turbulent core})$$

- The wall function has been widely used to represent near wall behaviour in many turbulent flow applications.
- The wall function is compared with experimental data in the next slide. (taken from Szekely, p. 143).

# Grid Considerations I

- Turbulent flows are much more susceptible to grid dependencies than are laminar flows.
  - The grid must be consistent with turbulence transport considerations and near wall behaviour.
- As with all modeling, fine meshes are used where mean flows change rapidly.
- CFD software usually should have available information related to near wall parameters; e.g.  $y^+$ ,  $y^*$  and boundary layer Reynolds number.
- Note that  $y^+$  and  $y^*$  are primarily solution dependent, with secondary dependence on mesh size.

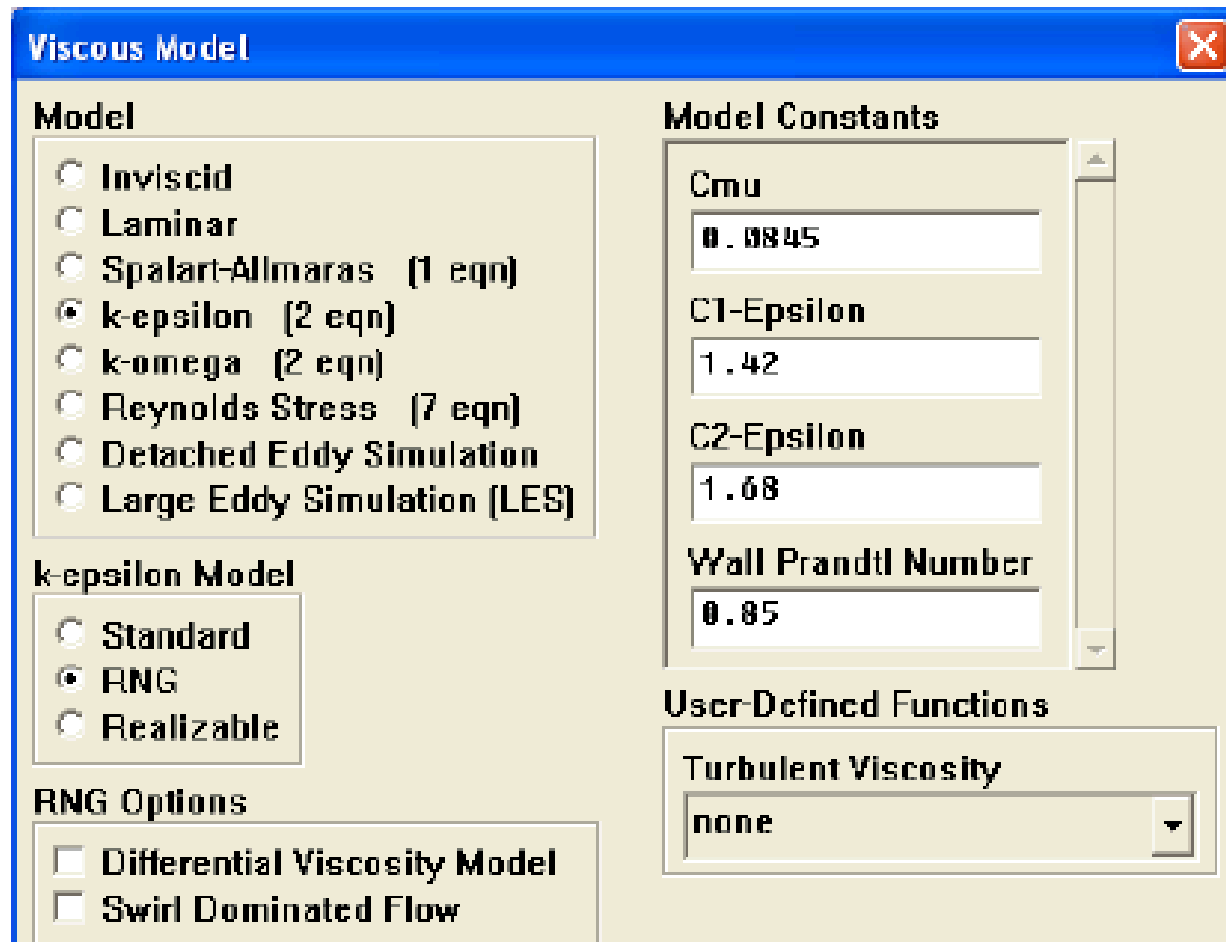
# Grid Considerations II

- When using the standard wall function, the mesh should be such that the centroid of the wall cell is within the log-law layer:

$$30 < y^+ < 300$$

- The closer  $y^+$  is to 30, the better the quality of the solution.
- Wall adjacent cells should not have centroids in the buffer layer:  $y^+ = 5 \rightarrow 30$
- As in all simulations, well shaped elements should be used.

# Choice of Viscous Models in Fluent



# Summary & Conclusions

- This lecture has attempted to provide insight into special issues that arise in CFD:
  - Overlapping solution grids are typically used.
  - Upwinding must be used to overcome difficulties associated with the convection term.
  - The pressure term is accounted for by means of the continuity equation.
- The SIMPLE algorithm for solving the N-S equations was briefly reviewed.
  - This algorithm is at the heart of many of the available codes.
- A number of other issues relating to numerical solution were reviewed.