Numerical Simulations Techniques applied to Coastal and Oceanographic Engineering - Computational Fluid Dynamics (CFD)

Lecture 1

1. See syllabus – finalize time/ location
2. See text book – Fletcher Vols. 1 & 2
3. See list of CFD books
4. Standard grading, Undergrad $\rightarrow$ Grad $\rightarrow$ Advanced

Our interest in CFD for Coastal and Ocean Engineering: Encompassing

1. Large scale ocean flows ~ 10,000 km
2. Regional modeling ~ 1,000 km
3. Coastal modeling ~ 100 km
4. Littoral modeling ~ 10 km
5. Near shore modeling ~ 1 km
6. Large eddy simulations ~ 100 m
7. Turbulent layers ~ 1-10 m
8. Fluid – structure interactions 1-100 m
9. Small scale processes 1 mm -1 m

10. Engineering processes
   a. Flow around propellers
   b. Flow in pipes – plants
   c. Ship wakes
   d. Boundary layers
   e. Stream line vehicles/propulsion

11. Wave Dynamics
   a. Air-sea interactions
   b. Wave prediction
   c. Harbor design
   d. Wave – body interaction/drag reduction

12. Research on Physics of Fluids
   a. Effects of stratification
   b. Two phase flows
      i. Sediment/ fluid interaction
      ii. Bubbles – fluid
   c. Acoustic propagation in the ocean
   d. Turbulent dispersion in environment
      i. Pollution

ii. Heat reservoir
iii. Gases - CO_2
iv. Momentum transfer

e. Flow around complex topography

13. Beach Erosion

14. Estuaries and Inlets

a. Tides in harbors
b. Storm surge and shore protection

15. Physical – Biological Coupled Systems

a. Biological productivity
b. Anthropogenic influences

Many of society's problems in transportation, environment, energy production, entertainment, construction, agriculture, defense, and medicine use CFD to develop innovative solutions for the future.

There are many jobs available in C.F.D.

Prerequisites:

- Advanced Engineering Math
- Physical Aspects of Oceanography

  and/or

- Advanced Hydrodynamics 1

Reading Assignment 1

- Chapters 1 & 2 of Fletcher I (next class)
- Chapter 3 of Fletcher I (by next week)

Course Logistics

1. Need a Computer – Unix or windows
2. Will write your own CFD programs in Fortran (or C or Matlab)?
3. Need familiarity with a Scientific Plotting Package. (i.e. Tecplot or Matlab).
4. Help each other with programming/debugging but do majority of assignments individually.
5. Review ~ 5-10 pages of typed notes together, questions, discuss readings, examples, review solutions, and assist with homework.

CFD is a separate discipline distinct from theoretical and experimental fluid dynamics, but more closely associated with experiments.

With the rapid increase in computational power in the last 20 years CFD has gained significant popularity and modeling is often considered more illuminating and cost effective than lab or field experiments.

CFD began around 1910 with L.F. Richardson. He used people doing approximately 2,000 calculations per week to do the first numerical weather prediction. In 1928, Courant, Friedrichs and Lewy (CFL) determined the numerical stability criteria for time marching solutions forward.

Around 1945, John von Neumann developed the theory of numerical stability analysis.

Digital computers were applied to Fluid flow problems from about 1945 onward in increasing numbers.

There are now over 100 scientific journals that report work on CFD in different fields (Math, Physics, Engineering, Atmosphere, Astronomy, Combustion, Ocean……)

The potential for future applications in CFD is large as computing capabilities are expected to continue to increase and more complex physical systems can be modeled realistically.

The practice of CFD is more Engineering (art) than a rigorous mathematical science as many combinations of physical and mathematical approximations must be chosen and there are many equally valid (defensible) models for a given system.

We begin with the Conservation Laws governing Fluid Flow typical of the Ocean.

<u>The Navier – Stokes Equations</u>

1. $$\frac{\partial \vec{u}}{\partial t} + \vec{u} \bullet \nabla \vec{u} = -\frac{1}{\rho} \nabla p + 2\vec{\Omega} \times \vec{u} - g\hat{k} + \nu \nabla^2 \vec{u}$$

2. $$\nabla \bullet \vec{u} = 0$$

Describing the conservation of linear momentum and mass in the continuum approximation, for an incompressible flow.

Despite the relatively large computing capabilities available today, except for very small domains with simple boundary conditions, it is not possible to obtain full and accurate numerical solutions to the full non-linear N.S. equations for a turbulent flow.

The general approach is to attempt to retain the essential physics of the flow and reduce 1 & 2 to a more simplified sub-system that can be more readily solved.

The incompressible equations themselves are one such approximate reduced model from the compressible flow equations, that have filtered out sound waves which is a “good” approximation for flows where sound waves are not of primary interest and $|\vec{u}| \ll C$ say $|\vec{u}| \lesssim 0.3\mathrm{M}$.

Half the <u>Modeling problem</u> done in CFD is deciding on the “appropriate” governing equations for a system. The choice will strongly influence the results you obtain.

We will only spend $\sim \frac{1}{100}$ of our time in this class, considering what the right models to use for different flows are.

We will spend 99% of our time examining mathematical approximations to “solve” the selected equations.

The choice of the numerical methods will also significantly influence our “solutions”, and compromises between cost and performance are made at every level.

Simplified Models Include:

1. Hydrostatic Ocean models (P.O.M., R.O.M.S., Mi com, etc.)
   3D, vertical velocity $\ll$ horizontal velocity

2. Geostrophic Models
   Dominant balance between large scale pressure gradients and Coriolis forces

3. Balance Models
   - Filters out internal waves

4. Shallow water models
   Depth $\ll$ horizontal scales

5. Large Eddy Simulations
   Small scales represented by turbulence model

6. 2-D flow – symmetry invoked to reduce dimensionality

7. Direct Numerical Simulations limited to low Re flows and simple geometries

8. Reynolds Averaged N.S.
   - Solve for mean flows, model turbulent fluctuations

9. Steady flows

The physics (geometry, length and time scales) of a system, usually represented by dimensionless parameters, Re, Fr, Pr, Ri, $\frac{h}{L}$, etc. are analyzed to determine the "optimal" choice of the governing equation set.

This is called Physical modeling and precedes numerical modeling.

The validity and range of applicability of a physical model should be evaluated before and after the numerical model is "run".

Special sets of numerical techniques are often best suited for various subsets of systems of partial differential equations (PDE's).

A second preliminary consideration is when to write a new model and when to use an off the shelf CFD model.

Hundreds of Commercial and Public Domain CFD models are available for free or at modest (less than 6 months pay) cost.

Popular Engineering CFD Commercial models include:

Fluent	Star-CD	ACE
Flow 3D	AVL	Flotran
CFX	Unic CFD	FIDAP
Algor	Simpler	CFD2000
Splash	Maya	CFD++

Popular Ocean related open research CFD models include:

ADCIRC, POM, ROMS, MitOM, MOM, WAM, VOF-3D, Ripple …

Popular CFD web sites include:

www.cfd-online.com

Generally if you are doing engineering you will use a CFD package, if you need a solution in a matter of days or weeks (Models take $\frac{1}{2}$-2 years to write).

If you are an academic researcher, you will use one of the open research models, or if you have more than a year you will write your own.

Open models – give source code clear what was done, easily modified or test influence of approximations (Good Compromise).

Packages – Black boxes, less versatility, don't have to "reinvent the wheel", lower cost.