Computational Fluid Dynamics

Lecture 27
May 3, 2017
Grétar Tryggvason

Grading
Projects 60%
Homework 25%
Final 15%

Project and HW due NOW

What did we learn?
Coarse goals and brief outline
More detailed list of topics
Grading/Exam
Input

Coarse Goals:
Learn how to solve the Navier-Stokes and Euler equations for engineering problems.
Hear about various concepts to allow continuing studies of the literature.

Ways:
Detailed coverage of selected topics, such as: simple finite difference methods, accuracy, stability, etc.
Rapid coverage of other topics, such as: multigrid, monotone advection, unstructured grids.

What are the current trends?

Course outline

Part I
Introduction, what is CFD, examples, computers, elementary numerical analysis, course administration
Elementary numerical analysis, accuracy, stability, partial differential equations
Review of fluid mechanics: the governing equations
Finite Difference solution of the Navier-Stokes Equations in vorticity/streamfunction form
Finite Volume Approach, Solving the Navier-Stokes Equations in Primitive Variables, the MAC Method

Part II
Numerical Analysis of partial differential equations, cumulating in solution techniques for the Navier-Stokes equations

Part III
Advanced topics in CFD
First order Partial Differential Equations (PDF's).
Characteristics. Classification of Second Order PDF's.
Algorithms for Hyperbolic equations. The Euler equations.
Algorithms for Parabolic equations.
Algorithms for Elliptic equations.
Advection-diffusion equation
More Volume Approach, Solving the Navier-Stokes Equations in Primitive Variables, the MAC Method, Colocated grids

Complex Domains. Body fitted Coordinates.
Complex Domains. Grid Generation
Visualization
Other Methods. Spectral, Finite Element, Vortex, etc
Software

Introduction to Complex Flows: Turbulence, Multiphase flow, and Combustion
Direct Numerical Simulations of Multiphase Flows
Current trends (predictivity, multiscale, large systems)
Parallel Computations,
Computer Science Issues

The Navier Stokes Equations:
The basic equations of fluid mechanics in integral and differential form. The difference between a conservative and non-conservative form. Different formulations of the basic equations: Primitive (velocity-pressure) form; stream function-vorticity, and velocity-vorticity form in two- and three-dimensions. The representation of advection, viscous friction, and incompressibility, in the equations of motion. The pressure equation for the primitive formulation. Physical boundary conditions.

Partial Differential Equations:

Elementary Numerical Concepts:
Numerical Solutions of PDE's:


Parabolic equations:

Elliptic equations:

Numerical Solutions of the Navier-Stokes equations:

Vorticity-streamfunction form. Order in which the equations must be solved. Boundary conditions for the vorticity.


Complex domains:

Complex flows:
Additional considerations for stratified and turbulent flows. The $k$-$\epsilon$ model for turbulent flows.

Computations of multiphase flows. Lagrangian and Eulrian modeling of disperse flow

Combustion modeling: Diffusion flames versus Premixed flames

Predictivity: Method of manufactured Solutions and UQ

Multiscale and Large Systems

Parallel computing
Serial versus parallel, Shared versus distributed memory, Domain decomposition, Message Passing Interface (MPI)
To compute the mixing as a liquid jet enters a large rectangular domain it is often necessary to cluster the grid points near the entrance region. Assume that the domain has a width W and length L, and the jet enters through the middle of the left boundary and has a diameter D. The domain is very large and it is impractical to use a uniform grid. You should solve for the flow using the Navier-Stokes equations in the pressure-velocity form, but you can assume that the flow is two-dimensional.

(a) Describe your gridding strategy and sketch what you would like the grid to look like.
(b) Propose a mapping function that clusters the grid points near where the jet enters the domain.
(c) Describe the flow solver that you will use.

The following equation (a nonlinear diffusion equation),
$$\frac{\partial f}{\partial t} + D \frac{\partial^2 f}{\partial x^2} = g(x)$$
is solved on time $0 \leq t \leq T$ for the domain $0 \leq x \leq L$, where T is the final time and L is the length of the domain. The source $g(x)$, which is concentrated at $L/2$ is turned-on for a very short time at $T/2$. For the most part we expect to be able to use a relatively coarse grid, except around the source when it is active. Thus, we want to use a grid that is refined in both time and space around $T/2$ and $L/2$.

(a) Propose a mapping function to refine the grid in space and time around $T/2$ and $L/2$.
(b) Write down the partial differential equation in the new coordinates.
Propose a numerical scheme to solve for the unsteady flow over a rectangular cube in an unbounded domain. The Reynolds number is relatively low, 500-1000. Identify the key issues that must be addressed and propose a solution. Limit your discussion to one page. Do NOT write down the detailed finite difference equations, but state clearly what kind of spatial and temporal discretization you would use.

**Problem 26**
Show that the one fluid formulation contains the "usual" Navier-Stokes equations.

**Problem 25**
(a) Propose a mapping function to transform the problem into a new coordinate system. 
(b) Write down the partial differential equation in the new coordinates.

To evaluate the effectiveness of a new mixing device, you have been asked to write a code to simulate the flow in a channel shaped somewhat like the one sketched below. The Reynolds number is only a few hundreds, so the flow is laminar, and you can take the channel to be two-dimensional. Accuracy is extremely important and you have been asked to accomplish this in only a few weeks. You only have to worry about getting the flow field correct.

Describe how you would accomplish this. You do not need to write down the detailed discrete equations, but your description should be sufficiently detailed so that it is clear what approximations you are going to use, and you should clearly state all critical issues and how you intend to deal with those. Limit the response to less than a page!

It is proposed to use a finite difference method to simulate the stirring and mixing of two fluids by a prescribed velocity field. The fluids are completely identical, except that one is read and the other is blue. The colors can mix, but diffusion is VERY small. The velocity field is unsteady (but GIVEN) and maximum velocity is one. Assume that the mixing takes place in a square, two-dimensional domain (with sides of length one) and that the initial configuration of the fluids is given.

(a) EXPLAIN the main difficulties in solving this problem and (b) PROPOSE a method to allow us to simulate this problem. You do not have to write down the finite difference equations in detail. Limit the response to less than a page!

We can expect that the development of numerical methods for computing fluid flow will continue and we will see methods that are:

- More accurate
- More robust
- More versatile

than current methods.

We are also seeing new trends and a shift in emphasis to topics that in the past were on the sidelines.

**Commercial Solvers:**
- Fluent: http://www.anysys.com/
- StarCD: http://www.cd-adapco.com/

**Miscellaneous Resources**
- OpenFOAM: http://www.openfoam.com/
- Gerris: http://gfs.sourceforge.net/index.php/Main_Page
- Fire Dynamics Simulator: http://fdm.nist.gov/fds4
- Multiphase Flow: https://www.mfix.org/
- Visualization: http://www.paraview.org

And many, many others!
New Directions for Computational Physics:
Computational predictivity
Multiscale/multiphysics
Integrated simulations of complex systems

Section 605 of the America COMPETES Reauthorization Act of 2010 [2] is titled Promoting use of high-end computing simulation and modeling by small and medium sized manufacturers, and states that Congress finds that:

1. the utilization of high-end computing simulation and modeling by large-scale government contractors and Federal research entities has resulted in substantial improvements in the development of advanced manufacturing technologies; and

2. such simulation and modeling would also benefit small- and medium-sized manufacturers in the United States if such manufacturers were to deploy such simulation and modeling throughout their manufacturing chains.

Ensuring American Leadership in Advanced Manufacturing by the President’s Council of Advisors on Science and Technology (PCAST) and the President’s Innovation and Technology Advisory Committee (PITAC) states that “powerful computational tools and resources for modeling and simulation could allow many U.S. manufacturing firms to improve their processes, design, and fabrication.”

PLEASE FILL OUT COURSE INSTRUCTOR FEEDBACK!

http://www.nd.edu/~cif/cif.shtml