Aug 6th, 9:30 AM - 12:00 PM

Efficient simulation of fluid flow

David Hannasch
University of Nevada Las Vegas

Monika Neda
University of Nevada Las Vegas, Department of Mathematical Sciences, Mentor

David Hannasch
Mentor – Dr. Monika Neda
University of Nevada Las Vegas – Department of Mathematical Science

We are computationally investigating fluid flow models for physically correct predictions of flow structures. Models based on the idea of filtering the small scales/structures and also the Navier-Stokes equations which are the fundamental equations of fluid flow, are numerically solved via the continuous finite element method. Crank-Nicolson and fractional-step theta scheme are used for the discretization of the time derivative, while the Taylor-Hood and Mini elements are used for the discretization is space. The effectiveness of these numerical discretizations in time and space are examined by studying the accuracy of fluid characteristics, such as drag, lift and pressure drop.

Efficient Simulation of Fluid Flow

David Hannasch, Dr. Monika Neda

University of Nevada Las Vegas NSF EPSCoR UROP 2009

Introduction

Fluid dynamics is the study of the motion of fluids such as air and water. It is important to know how air may be expected to flow over a windmill's blades, or for that matter how liquid sodium flows through a nuclear reactor. **Computational Fluid Dynamics (CFD)** puts theory into practice and tries to simulate fluid flow in a computer.

A CFD primer

What we want out of CFD is the ability to predict the velocity ($\mathbf{u}$) and pressure (p) of a fluid at any given point in space and time. First, we need to know the shape of the channel (Ω), the kinematic viscosity (ν) of the fluid, and what force ($\mathbf{f}$) is acting on the body of the fluid (e.g. gravity). Incompressible fluid flow is then governed by the **Navier-Stokes equations** given below:

$$\begin{aligned} \mathbf{u}_t - \nu\Delta\mathbf{u} + (\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p &= \mathbf{f} \text{ in } (0,T]\times\Omega \\ \nabla\cdot\mathbf{u} &= 0 \text{ in } [0,T]\times\Omega \end{aligned}$$

Fluid flow can be predicted by solving these equations for $\mathbf{u}$ and p.

Intuitively, the flow of molasses is easier to predict than the flow of water. Meanwhile, large, high-speed flows are more difficult to predict than smaller, slower flows. This suggests the use of the ratio of average velocity ($\mathbf{u}_{ave}$) and characteristic scale (L) to viscosity as a rough measure of how chaotic or turbulent a flow is. This number is called the **Reynolds number** (Re) of the flow.

$$\mathrm{Re} = \frac{\mathbf{u}_{ave}\cdot L}{\nu}$$

The higher the Reynolds number, the more expensive it is to simulate the flow. Many applications involve very high Reynolds numbers, and accurate simulation can tax the abilities of the world's largest supercomputers. Because of this, we are looking for ways to eke out more accuracy without increasing simulation time.

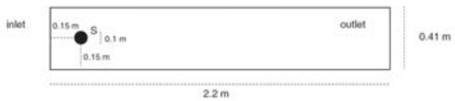
One option is to tweak the formulation of the differential equations. The version shown above is the simplest and most compact, but another form may better represent the true physical forces at work. We can reform the viscous term using a symmetric gradient:

$$\begin{aligned} \mathbf{u}_t - 2\nu\nabla\cdot\nabla^S\mathbf{u} + (\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p &= \mathbf{f} \text{ in } (0,T]\times\Omega \\ \nabla\cdot\mathbf{u} &= 0 \text{ in } [0,T]\times\Omega \end{aligned}$$

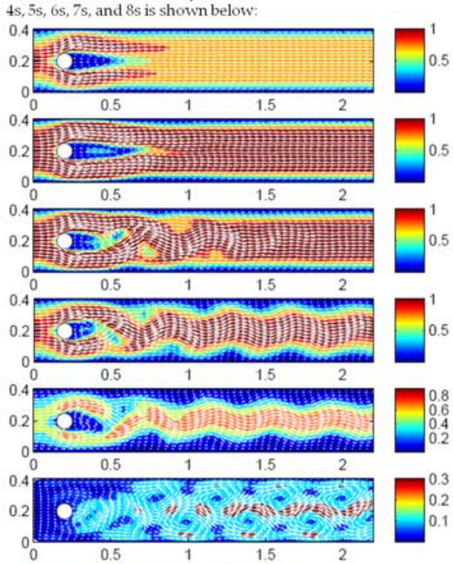
$$-2\nu\nabla\cdot\nabla^S\mathbf{u} = -\nu\nabla\cdot 2(\frac{\nabla\mathbf{u}+\nabla\mathbf{u}^T}{2}) = -\nu\Delta\mathbf{u} - \nu\nabla(\nabla\cdot\mathbf{u}) = -\nu\Delta\mathbf{u}$$

The Test Problem

We use as a benchmark the problem of two-dimensional flow through a channel pierced by a cylinder (figure from Volker [1]):



This problem is useful because of the vortices that form when the flow is accurately simulated. The channel at t = 2s, 4s, 5s, 6s, 7s, and 8s is shown below:



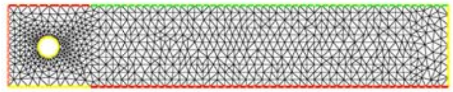
Pressure Drop, Drag and Lift

The appearance of the vortex street is a good, simple criterion for success of a simulation, but another important one is accurate calculation of the pressure drop (Δp) across the cylinder, as well as the drag (c_d) and lift (c_l). The pressure drop is defined simply as the difference between the pressures at the right and left edges of the cylinder. For the drag and lift, we integrate around the cylinder C:

$$\begin{aligned} c_d(t) &= \frac{2}{\rho L U_{max}^2}\int_C \hat{n}\cdot(p\mathbf{I} - \nu\nabla\mathbf{u})\cdot\hat{a}_d \\ c_l(t) &= \frac{2}{\rho L U_{max}^2}\int_C \hat{n}\cdot(p\mathbf{I} - \nu\nabla\mathbf{u})\cdot\hat{a}_l \end{aligned}$$

The Mesh

We find numerical solutions to the Navier-Stokes equations using the Finite Element Method. First, we use a mesh to divide the channel into many triangular regions ("elements"):



The mesh shown is the coarsest mesh that still showed the vortex street (it is designated mesh level 1 for this reason). With the mesh in hand, we can define hat functions for each element. These are functions that rise to 1 in the center of their corresponding element and fall to 0 on all non-adjacent elements.

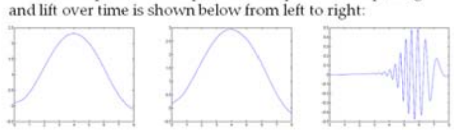
We approximate $\mathbf{u}$ and p as linear combinations of hat functions and solve for the coefficients. We use hat functions that are piecewise linear (P1), piecewise quadratic (P2), or piecewise linear with an additional internal bubble (P1b). The velocity/pressure pairs tested were P2/P1 and P1b/P1.

Results

Schäfer and Turek [2] give us reference intervals for the final pressure drop and for the maximum drag and lift:

$\Delta p^{ref}(8s) \in [-0.115, -0.105]$ $\quad c_{d,max}^{ref} \in [2.93, 2.97]$, $c_{l,max}^{ref} \in [0.47, 0.49]$

The expected development of the pressure drop, drag and lift over time is shown below from left to right:



The results obtained using the first form of the Navier-Stokes equations and P2/P1 hat functions are shown below:

Level	Δt	$t(c_{d,max})$	$c_{d,max}$	$t(c_{l,max})$	$c_{l,max}$	$\Delta p(8s)$
1	0.04	3.92	2.75017	6.28	0.210088	-0.0980916
1	0.02	3.94	2.75031	6.12	0.251092	-0.108209
1	0.01	3.93	2.75033	6.08	0.262798	-0.103207
1	0.005	3.925	2.75032	6.075	0.265685	-0.102197
1	0.0025	3.925	2.75031	6.07	0.266305	-0.102085
1	0.00125	3.92375	2.75031	6.0675	0.266393	-0.102116
2	0.04	3.96	2.92506	5.68	0.382033	-0.103618
2	0.02	3.94	2.92535	5.98	0.456821	-0.105039
2	0.01	3.94	2.92543	5.94	0.474071	-0.110534
2	0.005	3.94	2.92544	5.925	0.478015	-0.111308
2	0.0025	3.9375	2.92544	5.9225	0.478875	-0.11156
2	0.00125	3.9375	2.92544	5.9225	0.478962	-0.111681
3	0.04	3.96	2.94438	6.12	0.393596	-0.102093
3	0.02	3.94	2.94469	5.96	0.458516	-0.106851
3	0.01	3.94	2.94478	5.93	0.479811	-0.110915
3	0.005	3.94	2.94480	5.915	0.483938	-0.111205
3	0.0025	3.9375	2.94480	5.9125	0.484778	-0.111338
3	0.00125	3.9375	2.94480	5.91125	0.484849	-0.111429

The alternative symmetric form yielded results very similar to these, neither appreciably better nor worse.

P2/P1 vs. P1b/P1

Simulations using P1b/P1 hat functions ran faster than P2/P1 simulations, but with less accuracy. With P1b/P1, neither form of the equation showed the vortex street at mesh level 1 (both still did at mesh level 2).

For a given mesh level and Δt, P2/P1 gave greater accuracy. The shorter running time of P1b/P1 allowed finer discretizations to be used for the same cost, but even comparing simulations of equal cost P2/P1 came out ahead.

Acknowledgements

Our thanks go out to our predecessors, and most particularly to Drs. John, Schäfer and Turek for providing detailed material on this benchmark problem.

This research would not have been possible without the generous support of NSF EPSCoR RII Award EPS-0814372 and the University of Nevada Las Vegas. Special thanks to the UNLV Office of Information Technology and the Nevada Radiological Computational Center for providing much-needed computational power. Thanks also to Shipra De and Jung Eun Kim for their advice and support.

Literature Cited

[1] V. John. Reference values for drag and lift of a two-dimensional time-dependant flow around a cylinder. *International Journal for Numerical Methods in Fluids, 44:777–788, 2004.*

[2] M. Schäfer and S. Turek. The benchmark problem 'flow around a cylinder'. *Notes on Numerical Fluid Mechanics, 52:547–566, 1996. In Flow Simulation with High-Performance Computers II.*

For further information

If you have any questions or would like more information, the authors may be reached at davidh@egr.unlv.edu and monika.neda@unlv.edu.