Projection method for incompressible flow

MPhil in Scientific Computing
Written assignment

Abstract

This project involves solving the incompressible Navier-Stokes equations in 2D, including consideration of how to extend the implemented method to non-Newtonian fluids.

Report requirements

The report should include:

- A brief introduction to the incompressible Navier-Stokes equations and non-Newtonian fluids.
- A description of the numerical techniques used, including the treatment of boundaries.
- Clear presentation of results; this should include resolution studies of some of the tests to demonstrate convergence to a solution.
- Analysis of the strengths and weaknesses of the method implemented.
- All information obtained from external sources should be appropriately cited, and a reference list included.

Care must be given as to the presentation of two-dimensional results. In order to clearly display results, a combination of full two-dimensional results and one-dimensional slices will be required. Two-dimensional results should be presented using some form of colour coded map in a visualisation program of your choice. As an example, if using Gnuplot, then the commands `set view map`, `set pm3d` and `unset surface` will give the desired view.

Though two-dimensional plots give a good overall description of the behaviour, they do not allow for easy analysis of e.g. convergence. Though the tests in this project do not have analytic solutions, some level of convergence can be examined by investigating one-dimensional slices of the results with multiple resolutions. These results should be plotted along a suitably chosen slice through the data.

You should use your judgement when choosing how many plots to include and how to display solutions. As a guide, the following plots would show a good overview of the solution to a test case:

- One-dimensional slices of $x-$ and $y-$velocity components and pressure for at least three different resolutions (at the final time of the simulation if the simulation is time-dependent). Different resolutions may be shown clearly on the same plot.

- A two-dimensional visualisation of the results for $x-$ and $y-$velocity components and pressure for the highest resolution run (at the final time if the simulation is time-dependent).

- For time-dependent simulations it is useful to show several (at least four) two-dimensional snapshots of the evolution for a single variable (e.g. the velocity magnitude). Again, the highest resolution run should be used for these plots.

Each test case should be described in full detail including the boundary conditions, number of grid points, solver parameters, time step used, etc.

1 Projection method

Projection methods, based on work originally by Chorin, are a class of methods for solving incompressible fluid problems by decoupling the computation of the velocity and pressure fields. There are many variants but the typical idea is to compute an intermediate velocity field which disregards the incompressibility constraint, and then project this onto a divergence-free space.

Implement a simple fractional step projection method such as those described in [3] (section 10.3.1-2), for an incompressible Newtonian fluid at low Reynolds number Re in two dimensions, satisfying the dimensionless Navier-Stokes equations

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \mathrm{Re}^{-1} \nabla^2 \mathbf{u}, \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0. \tag{2}$$

Note that for flows where $\mathrm{Re} \to 0$ a different pressure scaling can be adopted, so that the dimensionless equations are

$$\mathrm{Re}\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right) = -\nabla p + \nabla^2\mathbf{u}, \tag{3}$$

$$\nabla\cdot\mathbf{u} = 0. \tag{4}$$

You are free to choose the precise details of the method you implement, but it should work for about $\mathrm{Re} \leq 1000$.

The classical fractional step projection method involves two steps. Firstly an intermediate velocity field $\widetilde{\mathbf{u}}^{n+1}$ is computed using the advection and viscosity terms, but disregarding the pressure term:

$$\frac{(\widetilde{\mathbf{u}}^{n+1} - \mathbf{u}^n)}{\Delta t} - \mathrm{Re}^{-1}\nabla^2\widetilde{\mathbf{u}}^{n+1} + (\mathbf{u}^n\cdot\nabla)\mathbf{u}^n = 0, \tag{5}$$

$$\widetilde{\mathbf{u}}^{n+1}\Big|_b = \mathbf{u}_b^{n+1}, \tag{6}$$

where $\mathbf{u}_b$ is a Dirichlet boundary condition on $\mathbf{u}$. Secondly the projection is performed, enforcing the incompressibility constraint:

$$\frac{(\mathbf{u}^{n+1} - \widetilde{\mathbf{u}}^{n+1})}{\Delta t} + \nabla p^{n+1} = 0, \tag{7}$$

$$\nabla\cdot\mathbf{u}^{n+1} = 0, \tag{8}$$

$$\mathbf{u}^{n+1}\cdot\mathbf{n}\Big|_b = \mathbf{u}_b^{n+1}\cdot\mathbf{n}, \tag{9}$$

where $\mathbf{n}$ indicates the boundary normal. Numerically this can be solved by taking the divergence of equation 7 and eliminating $\nabla\cdot\mathbf{u}^{n+1}$ using equation 8, leading to an elliptic equation for p^{n+1}.

You may use any spatial discretisation scheme you wish (finite difference, finite volume, ...). The domain should be a square. Pay particular attention to the implementation of the boundary conditions for $\widetilde{\mathbf{u}}^{n+1}$, $\mathbf{u}^{n+1}$ and p^{n+1}, and to numerical stability restrictions on the time step. Implement solid wall boundary conditions with a no-slip condition, where the top wall may move tangentially with a given speed; this forms a lid-driven cavity.

Discretise the viscosity term implicitly in time as shown above (when might this become important?). Therefore both steps of the fractional method require solving a large implicit system. You may do this with any appropriate linear solver software package. In the equations above the advection term has been discretised explicitly in time for simplicity, but you can try an implicit discretisation if you wish.

Use your solver to compute some of the lid-driven cavity test cases in [1], for several different Reynolds numbers (you might find that your solver struggles with the higher Reynolds numbers – try to get to Re = 1000). Show both the transient (impulsively-started) solution as well as the steady-state. Use one-dimensional slices to compare your steady-state results at several different resolutions with some of the data in [1] (tables 1 and 2); this will help you to choose appropriate grid resolutions. Discuss the accuracy of your solutions and the computational efficiency of your method.

2 Non-Newtonian fluids

For a class of simple non-Newtonian fluids, the viscous term $\nabla^2\mathbf{u}$ in equation 3 is replaced by the more general term

$$\nabla \cdot \mu(|\dot{\gamma}|)\dot{\gamma}, \tag{10}$$

where

$$\dot{\gamma} = \nabla\mathbf{u} + \nabla\mathbf{u}^{\mathrm{T}} \tag{11}$$

is the fluid strain rate tensor. In two dimensions,

$$\dot{\gamma} = \begin{pmatrix} 2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y} \end{pmatrix}. \tag{12}$$

The effective viscosity μ is a function only of the strain rate magnitude

$$|\dot{\gamma}| = \sqrt{\frac{1}{2}\sum_{i,j}\dot{\gamma}_{ij}^2}. \tag{13}$$

Explain in detail (including discretisation and algorithmic details) how you might modify your solver to simulate a Bingham plastic fluid at Reynolds number zero and Bingham number B. Such a fluid has an effective viscosity

$$\mu(|\dot{\gamma}|) = 1 + \frac{B}{|\dot{\gamma}|}. \tag{14}$$

Note that the Bingham plastic model has a singularity as $|\dot{\gamma}| \to 0$. Comment on the physical meaning of this. (Hint: for the Bingham model, regions of the flow can exist where the stress $\tau = \mu(|\dot{\gamma}|)\dot{\gamma}$ is non-zero but where the strain rate $\dot{\gamma} = 0$; these are called unyielded regions.) How would this affect your numerical method? Look up regularisation methods as implemented in e.g. [2]. How do these change the physical interpretation of the model?

References

[1] U. Ghia, K. N. Ghia, and C. T. Shin. High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. *Journal of Computational Physics*, 411:387–411, 1982.

[2] E. Mitsoulis and T. Zisis. Flow of Bingham plastics in a lid-driven square cavity. *Journal of Non-Newtonian Fluid Mechanics*, 101(1-3):173–180, 2001.

[3] O. Zikanov. *Essential Computational Fluid Dynamics*. John Wiley & Sons, 2010.