

Navier Stokes Equations for Weather Applications

Brendan Meyler, Joshua Sharpes

Abstract

This poster seeks to explore the Navier-Stokes Equation (NSE) and its applications in Computational Fluid Dynamics (CFD). In particular, we derive and explain the steady incompressible form of the Navier-Stokes Equation in $\mathbb{R}^2$. This version of NSE is then solved using the Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) for an overhead, 2D view of a channel flow at a low initial velocity. We briefly explain the SIMPLE algorithm and describe the results in the context of weather modeling. We conclude by identifying avenues for further research and the challenges involved with NSE and weather modeling in general.

Background

Navier-Stokes Equations (NSE) are typically used in applications of fluid dynamics to characterize fluid flow through a pipe and pipe fittings such as orifices. Since air is a fluid, NSE can also be used to model the behavior of weather over time for a given area. These equations allow for monitoring conditions and making predictions. There are several variations of NSE, some more complex than others. We will study the case where the fluid is incompressible to neglect differences in fluid properties which NSE accounts for.

The Foundation of Navier Stokes Equations

The basis for modeling dynamical flow in the atmosphere are the NSEs, which represent the flow of fluids such as water and air. Mathematically, a fluid is represented by three functions: $\rho(x, t)$, $p(x, t)$, $\mathbf{u}(x, t)$ where ρ is density and p is pressure which are scalar functions of the position vector $\mathbf{x} = (x_1, x_2) \in \Omega$ in which Ω is a region of 2D space occupied by the fluid. The velocity of a particle of fluid at a point $\mathbf{x}$ and time t is given by the vector valued function $\mathbf{u}(\mathbf{x}, t) = (u_1(\mathbf{x}, t), u_2(\mathbf{x}, t))$. Given the fluid in question is Newtonian, the three functions ρ , $\mathbf{u}$, and p are governed by the 2D momentum conservation equation:

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \sum_{i=1}^2 u_i \frac{\partial \mathbf{u}}{\partial x_i} \right) = \mu \Delta \mathbf{u} + (2\lambda + \mu) \nabla (\nabla \cdot \mathbf{u}) - \nabla p + \mathbf{f}. \quad (1)$$

This is the formulation of Newton's second law applied to fluid mechanics. Here μ and λ are physical parameters and $\mathbf{f}(x, t)$ is the density of force per volume ($\mathbf{f} = \frac{\mathbf{F}}{v}$). In addition the functions ρ , $\mathbf{u}$, and p also obey the continuity equation:

$$\frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0. \quad (2)$$

This says the mass of the fluid is conserved over time and since the fluid is incompressible, ρ is independent of position and time so equation (1) reduces to:

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \sum_{i=1}^2 u_i \frac{\partial \mathbf{u}}{\partial x_i} \right) = \mu \Delta \mathbf{u} - \nabla p + \mathbf{f}, \quad (3)$$

$$\nabla \cdot \mathbf{u} = 0. \quad (4)$$

Navier Stokes Equations

Realistic atmospheric modeling characterizes the air as compressible. We are concerned with the incompressible case which is a safe assumption for low pressure changes. The convention is to let $\rho = 1$ and to set $\mu = \nu$, and write (3) as follows:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = \mathbf{f} + \nu \Delta \mathbf{u} - \nabla p. \quad (5)$$

Equation (4) plugged into equation (5) results in the nondimensional incompressible NSE in 2D. Setting up a boundary problem to model the flow over a fixed area, we create an initial condition (6) and boundary condition (7) over a single variable, y :

$$u(y, 0) = u_0(y) \quad y \in \Omega \quad (6)$$

$$u(0, t) = 0 = u(1, t) \quad y \in \Gamma \quad (7)$$

Where Ω represents the area of interest and Γ represents the boundary of that area. Equations (6) and (7) can be written in terms of x but the boundary condition will be equal to $\psi(x, t)$ and not zero since fluid through the channel is flowing from left to right. Ω is bounded in our model but can be unbounded for another model in which:

$$u(y, t) \rightarrow \psi(y, t) \quad \text{as } |y| \rightarrow +\infty \quad (8)$$

Considerations

- Flow was considered incompressible to avoid the complexity of an additional variable changing with time. This would have required extra iterations and subroutines in our model.
- Valid solutions of such a system must possess smoothness across them which is defined as:

$$A : X \rightarrow Y \quad (9)$$

Where A is an operator acting on 2D space of X to produce Y .

- Smoothness, in the context of our model, is assumed since error tolerances (relaxation parameters) were used to ensure that iterations are continuous with respect to each other.

Model Simulation: The SIMPLE Algorithm

The SIMPLE algorithm uses a guess-and-correct method which involves solving a set of algebraic equations at each iteration to obtain a new solution for the flow field of the given problem [2]. The algorithm uses inputs for domain (the dimensions of the area being modeled) and the boundary conditions (Eqn (7)) which define the areas of no-slip regions at $y=0$ and $y=1$ as well as the initial wind speed of 1 m/s. The pressure and velocity fields are calculated and compared against one another and corrected until the solution converges. Results for our model are shown below in Figures 1 and 2.

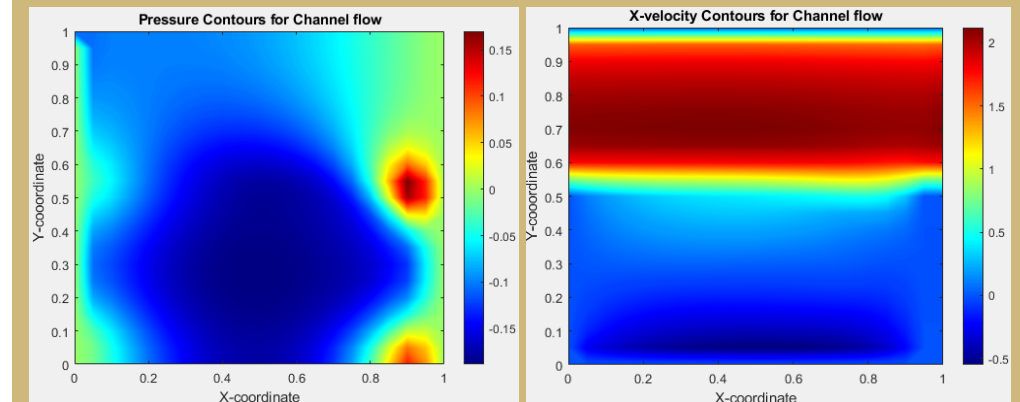


Figure 1: Overhead view of pressure contours Figure 2: Overhead view of X-velocity contours

The lower initial wind speed allowed for a more realistic model as the no-slip boundaries had less of a turbulence-inducing effect on the wind, that is less of a deviation between channel air and the ideal open space (unbounded) model [3]. Still, there are strange deviations in the results such as the elevated channel flow and the stagnation points. We assume that these are a result of the algorithm equation setup.

Conclusions

We conclude that modeling weather for a 2D region of space using numerical methods for the Navier-Stokes equation is a difficult problem. There are several different approaches to the problem, some better than others given the IC and BC of what is being modeled. Essential components of all methods are density, pressure, position, and velocity. In the future we wish to explore more CFD algorithms and techniques to gain more insight into what it is like to solve real world problems with NS equations.

References

- 1 BROOME', S. O. F. I. A., amp; RIDENOUR, J. O. N. A. T. H. A. N. (n.d.). A PDE PERSPECTIVE ON CLIMATE MODELING. A PDE PERSPECTIVE ON CLIMATE MODELING.
- 2 Versteeg, H. K., amp; Malalasekera, W. (2011). An introduction to computational fluid dynamics: The Finite Volume Method. Prentice Hall.
- 3 Navier-stokes solver using simple. MathWorks. (n.d.). <https://www.mathworks.com/matlabcentral/fileexchange/74996-navier-stokes-solver-using-simple>