Universitat Politècnica de Catalunya

Escola Superior d'Enginyeries Industrial, Aeroespacial i Audiovisual de Terrassa

Numerical resolution of the Navier - Stokes equations

Computational Engineering

220027

Professor

Carlos David Pérez Segarra

Francesc Xavier Trias Miquel

Author

Sergio Gutiérrez Sánchez

Saturday 22nd January, 2022

Abstract

In this report, it is presented a brief introduction to the numerical resolution of Navier - Stokes equations. First, it is shown a very short explanation of the theoretical basis behind these mathematical expressions and the physical meaning of each. Secondly, it is presented a brief description of the most important aspects of the resolution of incompressible flows such as Fractional Step Method or Checkerboard problem. And finally, study case Flow Around a Square Cylinder is briefly explained alongside a short explanation of the simulation process.

1 Introduction

Navier Stokes equations are probably the most famous fluid dynamics mathematical expressions. These were postulated by Claude-Louis Navier and George Gabriel Stokes.

The 3 equations, Mass, Momentum and Energy Conservation govern the behaviour of any fluid, and can be used to predict or simulate them.

Next up is presented a brief theoretical description of each equation.

1.1 Mass Conservation equation.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

Mass Conservation equation describes how the density of the fluid varies due to the flow motion on the domain. In case of incompressible flows, this equation can be simplified to the divergence term.

1.2 Momentum Conservation equation.

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \tau + \rho f \tag{2}$$

The Momentum Conservation equation can be considered as the formulation of Newton's second law to fluid dynamics. It relates the forces over the fluid, pressure gradients, viscous stresses and volumetric forces, with the change on its velocity.

In this equation, there can be remarked the convective term $\nabla \cdot (\rho \mathbf{u}\mathbf{u})$ which is specially important due to the development of convective schemes. Additionally, the viscous stresses are also a vital part, and the pressure gradients, which will be explained lately in a more detailed way.

1.3 Energy Conservation equation.

$$\frac{\partial \rho h}{\partial t} + \nabla \cdot (\rho h \mathbf{u}) = -\frac{DP}{Dt} - \nabla q + \tau : \nabla \mathbf{u} \tag{3}$$

Finally, the Energy Conservation equation consists basically on applying thermodynamic's basis, which states that the rate of change of energy in a fluid particle must be equal to the energy received by heat and work transfers.

2 Description of numerical methods

Once the theoretical basis of the Navier - Stokes equations have been presented, the numerical resolution of them is explained next up.

For this brief report, there will be shown the resolution of incompressible Navier - Stokes equations.

2.1 Fractional Step Method

The numerical technique to solve incompressible Navier - Stokes equation is known ad "Fractional Step Method". It is one of the most common solving techniques in incompressible fluid dynamics. In addition, it is relatively simple from a coding point of view. It is used to solve Mass and Momentum Conservation equations.

This techniques is mainly based on taking advantage of fluid's incompressibility. First, discretizing the Momentum Conservation equation leads to the following.

$$\mathbf{u}^{n+1} = \mathbf{u}^n + \Delta t \cdot F(\mathbf{u}) + \Delta t \cdot S(p^{n+1}) \quad (4)$$

Where:

$$\begin{aligned} F(\mathbf{u}) &= -\nabla \cdot (\mathbf{uu}) + \frac{\mu}{\rho}\nabla^2\mathbf{u} \\ &\quad + g \cdot (1 - \beta \cdot (T - T_0)) \end{aligned} \quad (5)$$

$$S(p) = -\frac{1}{\rho}\nabla p \quad (6)$$

Then, it is defined a "predictor's velocity", $\mathbf{u}^*$ as follows.

$$\mathbf{u}^* = \mathbf{u}^n + \Delta t \cdot F(\mathbf{u}) \quad (7)$$

And now, applying the nabla operator to each side of equation (7), leads to the following.

$$\nabla \cdot \mathbf{u}^{n+1} = \nabla \cdot \mathbf{u}^* - \frac{\Delta t}{\rho}\nabla \cdot (\nabla p^{n+1}) \quad (8)$$

Finally, there must be taken into account that the fluid is incompressible, which means its divergence is zero ($\nabla \cdot \mathbf{u} = 0$).

$$\frac{1}{\Delta t} \cdot \nabla \cdot \mathbf{u}^* = \frac{1}{\rho}\nabla^2 p^{n+1} \quad (9)$$

Equation (9) is also known as "Poisson equation". It can be easily solved with an iterative solver such as Gauss - Seidel, or any other algebraic system solver.

2.2 Predictor's Velocity

As it has been said previously, when applying FSM solving technique, it is necessary to define what's known as Predictor's Velocity.

The definition of this new variable comes due to the application of the Helmholtz-Hodge theorem. This states as following [1].

A given vector field , defined in a bounded domain ω with smooth boundary $\delta\omega$, is uniquely decomposed in a pure gradient field and a divergence-free vector parallel to $\delta\omega$

$$\omega = a + \nabla\varphi \quad (10)$$

Where:

- $\nabla \cdot a = 0$
- $a \in \omega$

Then, as there can be observed, Predictor's velocity fit perfectly on the theorem basis as a term in equation (10).

It can be defined as follows.

$$\mathbf{R}(\mathbf{v^n}) = -(\rho\mathbf{v} \cdot \nabla)\mathbf{v} + \mu\nabla^2\mathbf{v} \quad (11)$$

$$\mathbf{v^p} = \mathbf{v^n} + \frac{\Delta t}{\rho}[\frac{3}{2} \cdot \mathbf{R}(\mathbf{v^n}) - \frac{1}{2}\mathbf{R}(\mathbf{v^{n-1}})] \quad (12)$$

In order to compute the term $\mathbf{R}(\mathbf{v^n})$, it is necessary to calculate the convective term of the equation. To do that, it is possible to implement convective schemes or use meshes such as staggered mesh. In what comes to the second term $\mu\nabla^2\mathbf{v}$, it can e calculates applying the Stokes' theorem and evaluating the gradients of velocity at the control volume walls.

2.3 Pressure Gradient

The last aspect to mentioned about the Fractional Step Method and the resolution of the incompressible Momentum Conservation equation is the pressure gradient $\nabla \cdot (\nabla p^{n+1})$.

The pressure gradient consists on calculating the rate in which pressure is increasing or decreasing in any direction in order to take this into account while computing the acceleration of the flow. However, it is important to evaluate this term carefully due to the possibility of decoupling between pressure and velocity. This is also known as "Checkerboard Problem" (figure 1).

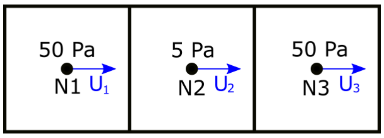

Figure 1: Checkerboard problem scheme. Extracted from [2]

As it is shown in figure 1, the Checkerboard Problem consists on the decoupling of the velocity from the pressure. Naturally, the velocity on node N2 would increase or decrease respectively due to the pressure gradient from the side nodes. However, if this gradient is not computed correctly, the pressure may change between nodes but the gradient may be null.

One option to avoid this problem is to implement an Staggered Mesh. This consists on 3 different meshes, one for pressure and temperature, and the other two for the horizontal and vertical velocities.

Finally, in what comes to the resolution of the Energy Conservation equation there must be said that it consists on a numerical integration of its terms. Since the Fractional Step Method allows to compute pressure and velocities, there is no need for applying a special technique to compute this equation.

Once the theoretical and numerical basis of incompressible Navier - Stokes equations have been explained, net up, it is going to be presented a brief introduction and description of one application example.

3 Study case

The study case selected for this report is the Flow Around a Square Cylinder. It is a widely known CFD problem for incompressible flows.

It consists on a channel with a parabolic inflow and a square cylinder placed on its mid section. To give a better insight of the case, it is presented a scheme in figure 2.

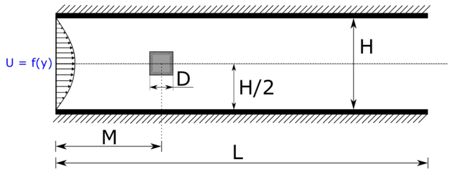

Figure 2: Flow around a Square Cylinder scheme. Extracted from [2]

As there can be seen in figure 2, when going through the channel, the flow is affected by the presence of the cylinder. Depending on the Reynolds number (the relationship between inertial and viscous forces), the flow behave differently near and past the cylinder.

For low Reynolds, the flow goes around it describing a smooth movement with no major perturbations at the outlet. However, when Re is increased, oscillating phenomena appears and "Von - Kárman Vortex" take place behind the cylinder reaching what's known as a periodic oscillating state.

In order to simulate this case, there are several steps that have to be done. First of them is the creation of the mesh. Due to the geometry and problem characteristics, it would be a great option to implement nodal distributions such as hyperbolic tangent or sinusoidal. Then, it is necessary to apply the Fractional Step Method described in previous section until an steady or periodic state is reached.

And finally, once the simulation has been completed, it is possible to compute post-processing parameters such as the lift and drag coefficients of the cylinder, the time period of the flow oscillations, etc...

References

[1] CTTC. “Fractional Step Method”. In: 1 (), pp. 1–41.
[2] Sergio Gutierrez. “Bachelor ’ s Degree Final Project NAVIER-STOKES EQUATIONS FOR LAMINAR AND Author : Director :” in: (2020).