Toward a New Simple Analytical Formulation of Navier-Stokes Equations

Gunawan Nugroho[1], Ahmed M. S. Ali[2] and Zainal A. Abdul Karim[3]

Abstract—Incompressible Navier-Stokes equations are reviewed in this work. Three-dimensional Navier-Stokes equations are solved analytically. The Mathematical derivation shows that the solutions for the zero and constant pressure gradients are similar. Descriptions of the proposed formulation and validation against two laminar experiments and three different turbulent flow cases are reported in this paper. Even though, the analytical solution is derived for non-reacting flows, it could reproduce trends for cases including combustion.

Keywords—Navier-Stokes Equations, potential function, turbulent flows.

I. Introduction

THE importance of Navier Stokes equations comes from their wide applicability for different kind of flows, ranging from thin films to large scale atmospheric even cosmic flows. However, Navier-Stokes equations are nonlinear in nature and it is difficult to solve these equations analytically. In order to perform this task, some simplifications are elucidated, such as linearization or assumptions of weak nonlinearity, small fluctuations, discretization, etc.

Despite the concentrated research on Navier Stokes equations, their universal solution is not achieved. The full solution of the three-dimensional Navier-Stokes equations remains one of the open problems in mathematical physics. Computational Fluid Dynamics (CFD) approaches discritise the equations and solve them numerically. Although such numerical methods are successful, they are still expensive and there must be approximation errors associated with them.

The development of high speed computers eventually makes discretization methods more advanced and it enables the numerical treatment of turbulent flow. Solution of turbulent flows mainly depends on solving Navier Stokes equations and using ad-hoc models to close the solution.

The numerical approaches are Reynolds Averaged Navier Stokes (RANS) which provides averaged solution of the flow, Large Eddy Simulation (LES) which solves the big scales and model the small ones, and Direct Numerical Simulations (DNS) which solve all the flow scales.

All authors are with the Department of Mechanical Engineering, Universiti Teknologi Petronas, Bandar Seri Iskandar, 31750 Tronoh, Perak Darul Ridzuan, Malaysia.
[1]gunawan@utp.edu.my, gunawan@ep.its.ac.id
[2]amahersali@petronas.com.my, amaher98@yahoo.com
[3]ambri@petronas.com.my

With respect to the computational cost, DNS is the most expensive numerical approach and it is still limited to small scale research problems. LES guarantees more economical computational time as compared to DNS and the results are not much different than DNS results when appropriate subgrid scale (SGS) models are used [1]. The cost of computation depends also on the dimension of the case and on the coupling with other equation as well, like the case of turbulent reacting flow. On top of the high cost of the numerical approaches, the necessary models play a major role on right predictions and can form a weakness to the solutions.

There is a large number of research concentrating on formulating efficient numerical schemes to solve Navier Stokes equations, such as the recent work as described in [1,2]. However, the computational costs are still high for handling accurate numerical simulations except for simple problems in engineering limited to small scale. In particular, it is known that in finite time interval, the solution of the Navier-Stokes equations may either be blown up or split up, losing its regularity, and beginning to form branches [3,4]. In fact, depending on the values of the relevant parameters, a stationary boundary value problem can have a unique solution, several solutions, or even no solutions at all.

In order to tackle this problem, the existence and smoothness theorem is widely applied to the mathematical analysis of Navier-Stokes equations as describe in many literatures. For example, nonstationary Navier-Stokes equations in the entire three-dimensional space are observed by Panel and Pokorny [5]. Some criteria on certain components of gradient velocity are given to ensure global smoothness in time [6].

On the other hand, the best way to overcome the above described numerical difficulties is to find classes of exact solutions to the full Navier-Stokes equations. Therefore, searching some classes of exact solutions of full Navier-Stokes equations is highly demanded from practical viewpoint [7]. Exact solutions will also provide theoretical understanding in paving the way to full global solutions. They may contribute to the global smoothness in time.

Unfortunately, only a few analytical works are currently present in literature. One of them is the transformation of Navier-Stokes equations to Schrodinger equation by application of Riccati equation [8]. It has good prospects since Schrodinger equation is linear and has well defined solutions. The method of Lie group theory is also applied in order to transform the original partial differential equations into ordinary differential systems [9]. It is concluded that an approximate series solution is obtained. The same route is performed, to transform the Navier-Stokes equations to solvable linear systems [10,11].

In this work, the objective is to find an analytical solution to the three-dimensional incompressible Navier Stokes equations by utilizing transformation coordinate. A potential function is proposed, and the three velocity vector are transformed into a single equation thorough the sum product of gradient and curl of the potential function. The mathematical derivation is carried out in two parts; first is by considering the cases in which the effect of the pressure gradient term is neglected and second by implementing constant pressure gradient in Navier-Stokes equations. The results are also validated with some experimental data for various cases.

II. ANALYTICAL SOLUTIONS

Navier Stokes equations in Cartesian form for incompressible fluids are written as,

At x-direction:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon\frac{\partial^2 u}{\partial x^2} + \upsilon\frac{\partial^2 u}{\partial y^2} + \upsilon\frac{\partial^2 u}{\partial z^2} \quad (1a)$$

At y-direction:

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \upsilon\frac{\partial^2 v}{\partial x^2} + \upsilon\frac{\partial^2 v}{\partial y^2} + \upsilon\frac{\partial^2 v}{\partial z^2} \quad (1b)$$

At z-direction:

$$\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + \upsilon\frac{\partial^2 w}{\partial x^2} + \upsilon\frac{\partial^2 w}{\partial y^2} + \upsilon\frac{\partial^2 w}{\partial z^2} \quad (1c)$$

The continuity equation is written as,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1d)$$

where p is static pressure, ρ is fluid density and υ is kinematic viscosity. The solution of the system of equations describes the three velocity components in the three spatial directions, i.e., $u = u(x, y, z, t)$, $v = v(x, y, z, t)$ and $w = w(x, y, z, t)$. The three velocity components are interlinked and coupled together such as the velocity magnitude in vertical sum can be written as, $U = (u^2 + v^2 + w^2)^{1/2}$.

Consider a potential function Φ, in which its derivatives are the velocity components which is expressed in vectorial form as,

$$V = \nabla\Phi + \nabla \times \Phi \quad (2a)$$

with $\nabla = (\partial/\partial x, \partial/\partial y, \partial/\partial z)$. The spatial coordinates are transformed into a single coordinate through the following transformation,

$$\xi = kx + ly + mz - ct \quad (2b)$$

where $k, l.m$, and c are transformation constants. The velocity components in equation (2a) can be rewritten including the new coordinate. Then, substituting to the Navier-Stokes equations and add them all to give,

$$-a\frac{\partial^2\Phi}{\partial\xi^2} + b\frac{\partial^2\Phi}{\partial\xi^2}\frac{\partial\Phi}{\partial\xi} = -\frac{d}{\rho}\frac{\partial P}{\partial\xi} + e\upsilon\frac{\partial^3\Phi}{\partial\xi^3} \quad (3)$$

with a, b, d and e are constant due to coordinate transformation. Dropping the pressure gradient term, and integrating once to have solution for $\partial\Phi/\partial\xi$. By performing integration once more, the expression for Φ is produced as

$$A\ln\left(1 + e^{B\xi}\right) + C\xi + D \quad (4)$$

where A, B, C and D are also constants from integration. Thus, by implementing the coordinate relation (2b), the explicit analytical solution is obtained.

Now equation (3) is recalled back for constant pressure gradient case,

$$-a\frac{\partial^2\Phi}{\partial\xi^2} + b\frac{\partial^2\Phi}{\partial\xi^2}\frac{\partial\Phi}{\partial\xi} = -\frac{d}{\rho}\frac{\partial P}{\partial\xi} + e\upsilon\frac{\partial^3\Phi}{\partial\xi^3}$$

and can be written with consideration of constant pressure gradient. Implementing $Q = \partial\Phi/\partial\xi$ and taking $bQ - a = R$ to get shorter expression. Taking $S = \partial R/\partial\xi$ and differentiating once will yield,

$$\frac{\partial^2 S}{\partial R^2} = \alpha \quad (5)$$

And by integrating equation (5) twice, the following result is obtained,

$$\frac{\partial R}{\partial\xi} = \alpha R^2 + \beta R + \gamma \quad (6)$$

By transforming back to R, and by rearranging $bQ - a = R$, the solution for Q is produced and for potential function. The obtained equality is similar to zero pressure gradient case,

$$\Phi = g\ln\left(1 + ke^{h\xi}\right) + f\xi + j \quad (7)$$

where *g, k, h, f* and *j* are the integration constants. Therefore, by substituting equation (2b), the explicit analytical solution is reproduced.

III. RESULTS AND DISCUSSIONS

The analytical solution is validated at early stage and the validation cases are presented in this paper. The validation procedure started with laminar cases. The first validation case is the laminar free jet experiment due to Symons and Labus [12].
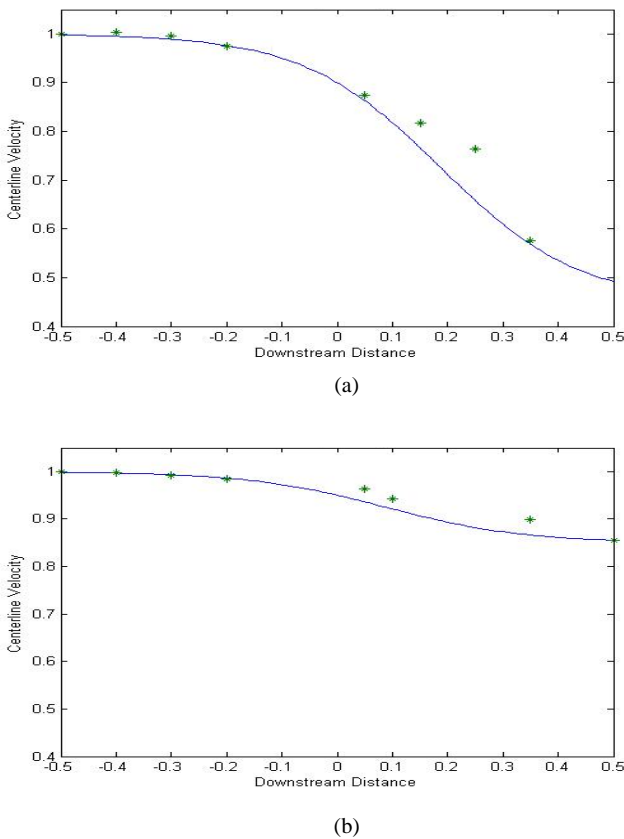
(a)

(b)

Fig. 1 Decay velocity along downstream direction produced by analytical solution (solid line) and the experimental data (points) for laminar free jet flow for (a) Re 255 and (b) Re 1839 [12]

The prescribed data is the normalised downstream velocity. Fig. 1 shows that the analytical solution could reproduce the decay of the measured downstream velocity with longitudinal distance from the nozzle. Here both experimental data and analytical calculations are normalized. It is observed that the comparison for higher velocity (higher Re) is more accurate as depicted in Fig. 1a. It may be due to the characteristic of the solution itself. Analytical solutions are obtained through the simple coordinate transformation. By dimensional analysis, it is clear that contributions of viscous terms are weakened for higher Reynolds number as described below,

$$\frac{\partial U}{\partial \eta} + U.\nabla U = -\nabla P + \frac{1}{Re}\nabla^2 U \qquad (8)$$

with $\Omega = Wt$, $\eta = \Omega/L$, $U = u/W$ and $Re = WL/v$.

The other similar experiment used for validation is the laminar free jet due to Eappen [13]. The transverse velocity is the one used for comparison here. Fig. 2 shows a comparison of the analytical solution predictions with the measured values. The inlet boundary condition is based on parabolic velocity profile to reproduce the profile of the experiment set up. The calculated values follow the same trend as the measured ones with high accuracy. However, at some other locations there is slight deviation which can be attributed to the vortex formation around the longitudinal axis immediately when the flow jets out of the nozzle exit. Different from the decay velocity, comparisons for transverse velocity profile show that calculation for higher velocity is less accurate than the other as shown in Fig. 2a.

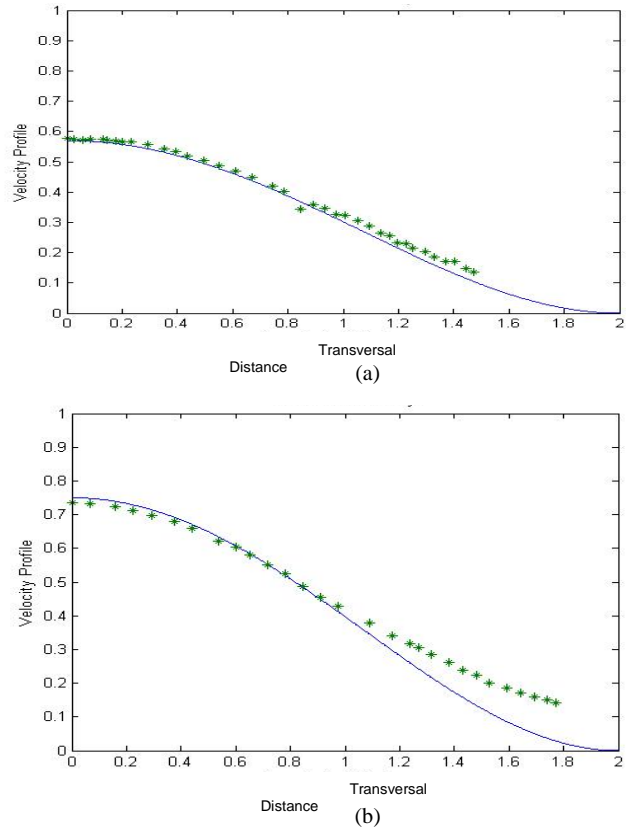Transversal Distance

(a)

Transversal Distance

(b)

Fig. 2 Velocity profile in transverse coordinate performed by analytical solution (solid line) and the experimental data (points) for laminar free jet for (a) Re 5 and (b) Re 20 [13]

It is well known that turbulent flows are much more complicated than laminar flows. Thus some naive approaches will fail for turbulent flows prediction even if they were successful for simple laminar flows prediction. Therefore, the analytical solution needed to go through a second stage of validation against turbulent flow cases.

The first turbulent case chosen for this validation stage is a boundary layer in atmospheric flow experiment due to Farrel and Iyengar [14]. In this experiment, data were produced in a 1.7 m wide, 1.8 m high and 16 m long test section of the St. Anthony Falls Laboratory tunnel. The experimental simulation technique was based on the use of quarter-elliptic, constant-wedge angle spires with height of 1.2 m and a castellated barrier wall to produce the necessary initial momentum defect in the boundary layer, followed by a fetch of roughness elements representative of the terrain under consideration.

Fig. 3 gives a comparison of the calculated boundary layer velocity profile produced by the analytical solution and the measured profile from the experiment. The analytical results are found to be in good agreement with the experimental data. However, the conformity showed here is the averaged quantity of turbulent flows. The fluctuating parts which are represented in energy transfer between eddies is a matter of further research.
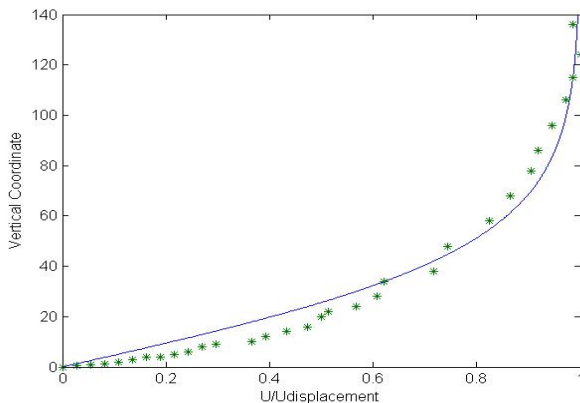

Fig. 3 Trend of boundary layer velocity profile produced by analytical solution (solid line) and measured values (points) for boundary layer flow [14]

Note that the analytical solutions described here are similar to the famous Blasius solution for boundary layer flows. Even though some prefer to use numerical methods for Blasius equation, the analytical solution can be produced easily for rectangular coordinate as follows,

$$2f''' + ff'' = 0 \qquad (9)$$

where all parameters above are non dimensional. Equation (9) is a class of quasi linear differential equation. Following the technique used for constant pressure gradients cases. By integrating (9) once to get,

$$2f'' + ff' = constant \qquad (10)$$

Equation (10) is similar to equation (3) and its solutions resemble previous solutions. Thus, it is not surprising that analytical solutions performed here can describe boundary layer flows.

The second challenging case used for validation in this stage is the recently published combustion experiment due to Cuoci et al. [15]. The fuel is fed in a central tube (3.2 mm internal diameter and 1.6 mm wall thickness), centered in a 15 cm x 15 cm square test section, 1m long, with flat Pyrex windows on the four sides.
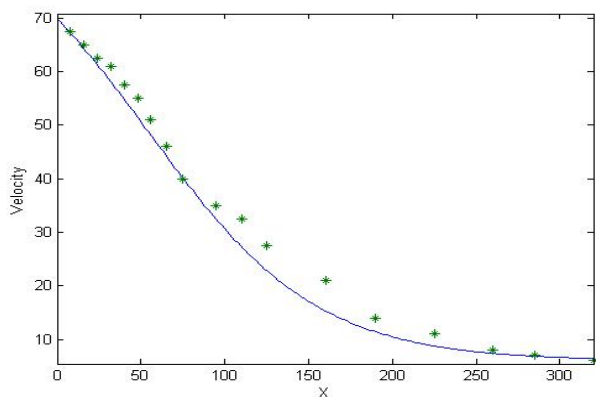

Fig. 4 Measured mean axial velocity along flame centre line in radial direction (points) for combustion [15] against the analytical solution (solid line)

The fuel molar composition is 39.7% CO, 29.9 H_2, 29.7 N_2 and 0.70 CH_4. Ammonia was added in different amounts up to 1.64%; in the absence of ammonia, methane was not included in the fuel mixture. The average fuel flow velocity was 54.6 m/s with a resulting Reynolds number of ~8500; and the inlet flow air velocity was 2.4 m/s. The inlet temperature of both streams is ~300K. Several radial profiles of velocity, temperature and species concentrations are available at different distances from the fuel inlet.

As shown in Fig. 4, the analytical solution could reproduce the velocity change throughout the axial line with good agreement with the measured values. Detailed analysis for this case needs other equations (energy, species and thermodynamic state) to be solved simultaneously and to describe turbulence-reaction interactions properly. This is of course a very challenging task and less tractable by considering that full mathematical theory for Navier-Stokes equations is not yet complete. However, the comparison here is to show the potentiality of the simple analytical solution to cover complex cases.

IV. CONCLUSION

Three-dimensional incompressible Navier-Stokes equation with continuity equations are solved analytically in this work. Derivations show that the two solutions for the zero and constant pressure gradients are similar. Furthermore, the proposed analytical solution is validated against experimental data for different cases. For the laminar nozzle jet flows, the analytical solutions are able to follow the decay and transversal velocity profiles. They also give good results for boundary layer flows and found to agree with combustion experiment

REFERENCES

[1] J.C. Kalita, A.K. Dass, and N. Nidhi, "An Efficient Transient Navier-Stokes Solver on Compact Nonuniform Space Grids," Journal of Computational and Applied Mathematics 214, 2008, pp. 124 – 162.

[2] Y. He and A. Wang, "A Simplified Two-Level Method for the Steady Navier-Stokes Equations," Comput. Methods Appl. Mech. Engrg. 197, 2008, pp. 1568 – 1576.

[3] J.D. Gibbon, D.R. Moore, and J.D. Stuart, "Exact, Infinite Energy, Blow up Solutions of the Three-Dimensional Euler Equations," Nonlinearity 16, 2003, pp. 1823 – 1931.

[4] P. Constantin, G. Gallavoti, A.V. Kazhikov, Y. Meyer, and S. Ukai, Mathematical Foundation of Turbulent Viscous Flows, Springer-Verlag, Berlin Heidelberg, 2003.

[5] P. Penel and M. Pokorny, "Some New Regularity Criteria for the Navier-Stokes Equations Containing Gradient of the Velocity," Applications of Mathematics 49, No. 5, 2004, pp. 483 – 493.

[6] Y. Zhou, "On a Regularity Criterion in Terms of the Gradient Pressure for the Navier-Stokes Equations in," Z. angew. Math. Phys 57, 2006, pp. 384 – 392.

[7] S.K. Kao, "An Analytical Solution for Three-Dimensional Stationary Flows in the Atmospheric Boundary Layer over Terrain," Journal of Applied Meteorology, Vol. 20, 1980.

[8] V. Christianto and F. Smarandache, "An Exact Mapping from Navier-Stokes Equation to Schrodinger Equation via Riccati Equation," Progress in Physics, Vol. 1, 2008.

[9] F. Kamran, C. Zu-Chi, J. Xiaoda, and Y. Cheng, "Similarity Reduction of a (3+1) Navier-Stokes System," Engineering Computations: International Journal for Computer-Aided Engineering and Software, Vol. 23, No. 6, 2006, pp. 632 – 643.

[10] A.V. Meleshko, "A Particular Class of Partially Invariant Solutions of the Navier-Stokes Equations," Nonlinear Dynamics 36, 2004, pp. 47 – 68.

[11] K. Thailert, "One Class of Regular Partially Invariant Solutions of the Navier-Stokes Equations," Nonlinear Dynamics, 2005.

[12] E.P. Symons and T.K. Labus, "Experimental Investigation of an Axisymmetric Fully Developed Laminar Free Jet," NASA Technical Notes, D-6304, 1971.

[13] T. Eappen, "Exit Region of Submerged Laminar Jets," Thesis, Submitted to the Faculty of Graduate Studies, Department of Mechanical Engineering, University of Windsor, Ontario, Canada, 1991.

[14] C. Farrel and A.K.S. Iyengar, "Experiments on the Wind Tunnel Simulation of Atmospheric Boundary Layers," Journal of Wind Engineering and Industrial Aerodynamics 79, pp. 11 – 35, 1999.

[15] A. Cuoci, Frassoldati, G. Buzzi Ferraris, T. Faravelli, and E. Ranzi, "The Ignition, Combustion and Flame Structure of Carbon Monoxide/Hydrogen Mixtures. Note 2: Fluid Dynamics and Kinetic Aspects of Syngas Combustion," Int. J. Hydrogen Energy, 2007.