[1.]Maša BUKUROV, [1.]Nikola OLUŠKI

THE DEVELOPMENT OF MATHEMATICAL MODELS OF VISCOUS FLUID FLOW AND ARISING ISSUES

[1.]University of NoviSad, Faculty of Technical Science, Novi Sad, SERBIA

Abstract: Although the greatest names in science have been grappling with one of the most complex problems for the past three centuries, the problem of turbulent viscous fluid flow has not been represented by a closed system of equations to this day. The attempts of Euler, Navier, Stokes, Bussinesque, Reynolds, Prandtl and others are getting in the last 50 years new support, such as computational fluid dynamics i.e., numerical modeling but also artificial intelligence with its tools such as deep learning and neural networks which are supposed to provide turbulence closure modeling. A historical overview and efforts of modern science and modern techniques are presented in this paper as well as some arising issues.
Keywords: viscous fluids flow, turbulence modeling, deep learning

INTRODUCTION

The development of fluid mechanics can be traced through historical data related to the great names of science, primarily mathematicians and physicists, but also engineers, inventors and lovers of the rich field of fluid studies. The first written document about fluid behavior was left to us 250 years B.C. by Archimedes entitled "On floating bodies". Despite the two–millennium study of fluids, to this day we do not have a closed mathematical model that describes the turbulent flow of a viscous fluid.

In 1755 Leonhard Euler (1707–1783) formed his famous equation for ideal fluid flow. After Euler, it appears that only Navier was motivated to formally tackle this problem and to succeed in solving it in 1822. He expanded Euler`s equation by introducing the viscous forces. Many investigators had put effort into solving the equation of motion for viscous flows as developed by Navier, and like him, Stokes had a very clear intention on the practicality of his efforts by confronting theory with experiments [1] in the 1830s and 1840s. This may be a reason why he and Navier became associated with the equation of motion for viscous flows. However, it would be fair to call the equation Euler–Navier–Stokes if it is necessary to include in the name the contribution of Stokes which was not fundamental, [1].

The Euler equation is given as:

$$\frac{d\vec{v}}{dt} = \vec{f} - \frac{1}{\rho}\mathrm{grad}p \tag{1}$$

where are:

$\vec{v}$– velocity vector, $\vec{f}$– body forces, p – pressure, ρ – density.

Hence the Navier–Stokes (N–S) equations being an extension of Euler's given in vector form also represents nonlinear partial differential equations:

$$\frac{d\vec{v}}{dt} = \vec{f} - \frac{1}{\rho}\mathrm{grad}p + \nu\Delta\vec{v} + \frac{1}{3}\nu\mathrm{grad}\,\mathrm{div}\vec{v} \tag{2}$$

where ν is kinematic viscosity. The continuity equation which is coupled with Navier–Stokes equations reads:

$$\frac{\partial\rho}{\partial t} + \mathrm{div}(\rho\vec{v}) = \rho\bar{\varepsilon} \tag{3}$$

where $\bar{\varepsilon}$ is specific yield of source or sink is also a partial differential equation.

MATERIAL AND METHODS

The aim of this paper is to present the development of a mathematical model that describes the flow of a viscous fluid, future tendencies and arising issues. In doing so, literary data were used that followed the development of fluid mechanics in the last two and a half centuries. The literature data are in abundance. It takes a lot of time to establish a solid path which can be followed through the labyrinth of investigations, research and experiments conducted by many famous and less famous scientists.

In this paper the main question will be if it is possible to solve one of the hardest problems in fluid mechanics, as well as in computational science, the problem of turbulence modeling, or turbulence closure modeling using modern techniques of deep learning or deep neural networks. Also, what does the future bring in this field from the standpoint of an educator.

In order to design certain objects such as aircraft, ships, submarines or turbine blades it is needed to estimate certain quantities with which the fluid flow field interacts with the objects, such as lift or drag. The starting point in fluid dynamics is the Navier–Stokes equations. They are time and space–dependent conservation of momentum equations. Navier–Stokes equations together with continuity equation represent a set of nonlinear partial differential equations which can be solved analytically only for a certain number of examples for laminar fluid flow in 2D, but solving these equations in 3D is, even nowadays, if not impossible, then extremely difficult, [1]. Previously, engineers made further approximations and simplifications to the equation set until they had a group of equations that they could solve. Contemporarily, high–speed computers have been used to solve approximations to equations using a variety of techniques, e.g., finite difference, finite volume, finite element, and spectral methods, [2].

ANALYTICAL APPROACH

Today N–S equation represents one of the seven most important unsolved problems established by the Clay Institute of Mathematics. A fundamental problem in analysis is to decide whether such smooth, physically reasonable solutions exist for the Navier–Stokes equations. [3]. There are analytical solutions for the laminar fluid flow examples, but the turbulent flow is much more complicated. It is represented with Reynolds equations which are time–averaged Navier–Stokes equations. By doing so, an unknown term – Reynolds stresses is introduced. They represent the impact of the turbulent fluctuations on the mean flow. The Reynolds equations are partial differential *equations of second order and of elliptical type that do not* possess analytical solutions. In order to solve these equations, it is necessary to introduce not only time averaging but also some additional hypothesis which will establish a connection between turbulent stresses and averaged velocities.

Flow over an obstacle produces turbulent separated structures over a wide range of scales with existing patterns. Large–scale structures (eddies, vortices) are mainly responsible for either drag, lift or mixing efficiency. It would be an extremely expensive simulation if it would characterize every single degree of freedom. Instead, a reduced order model would be sufficient to present how the big energy–containing structures work together to change the property of interest (e.g. drag on a boundary layer). Kolmogorov turbulent energy cascade shows length scales of existing eddies or vortices in a turbulent flow, Figure 1.

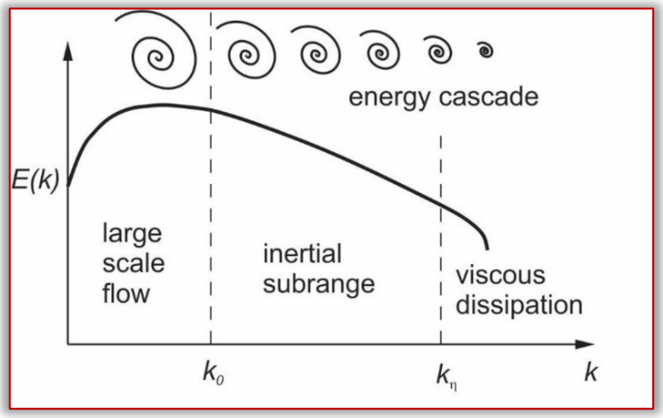

Figure 1. Kolmogorov turbulent energy cascade

The exact solutions of the Euler equation and Navier–Stokes equation are proposed by different authors using different methods. Among the most effective are Lie group theory and Baecklund transformation, symmetry reduction method [4], or transformation into the linear diffusion equations on a different basis [4–6]. Moreover, the N–S equations are solved by introducing some simplifications, e.g. the Cole–Hopf transformation is applicable for an incompressible flow and allows reducing the Navier–Stokes equation to the Einstein – Kolmogorov equation, [7].

Some authors [8] proposed the conversion of Navier–Stokes equations to ~~a~~ one linear diffusion equation based on the proposed linear velocity operator concept where the velocity operator is formulated in terms of a generalized new physical parameter.

The examples of flows for which analytical solutions are possible to find, with certain restraints, are those through ducts, pipes, coaxial gaps, between two parallel plates, etc. [9]. However, analytical solutions to even the simplest turbulent flows do not exist, [10].

Computational Fluid Dynamics (CFD)

In order to calculate how the object interacts with the fluid, and vice versa, it is necessary to simulate fluid flow to estimate quantities of interest. One way to do that is by using Computational Fluid Dynamics (CFD). CFD is a science that, with the help of digital computers, produces quantitative predictions of fluid–flow phenomena based on the conservation laws (conservation of mass, momentum, and energy) governing fluid motion and it complements experimental and theoretical fluid dynamics [11]. CFD enables analyses of complex problems involving fluid–fluid, fluid–solid or fluid–gas interaction, minimizes the planning time and saves costs of experiments. The results of CFD simulations are numerical solutions of the governing equations of fluid dynamics.

Real flow structures might have many orders of magnitude of scales both in space and time and instead of modeling all of them which is very expensive for computers, it is possible to approximate how small scales affect the big energy–containing scales that are actually of the main interest since they are mostly responsible for a lift and drag. This field is called closure modeling.

The turbulence modeling should enable avoidance simulation of a wide range of turbulent scales and provide closure of turbulence modeling. This field is rapidly progressing with a constant flow of results in literature, and recently the support of artificial intelligence and its tools, machine learning and more advanced deep neural networks, provide a better understanding of turbulence and the possibility of optimizing real fluid flow.

Direct Numerical Simulation (DNS)

Direct Numerical Simulation (DNS) is a research tool of CFD; it does not provide exact solutions to the Navier–Stokes equations for engineering problems. The aim of DNS is to get a detailed, both in spatial and temporal scales, model of a flow field, e.g., flow around an airfoil, or a turbine blade. But typically, it is too expensive, and lasts too long; even with Moore's law of exponentially growing computer power, it will still take too much time to simulate the largest scale turbulent problems at all resolutions in space and time, not to mention optimization process which would take even more time. Because of that it is a must to do the turbulence modeling.

Problem of Turbulence Modeling

One of the most demanding and intriguing problems of fluid mechanics is the problem of turbulence. The wide range of scales of time and space in turbulent flows demands significant resources, both in time and computer configuration to model turbulence.

Turbulent flow is characterized with oscillatory behavior of physical properties; hence they can be represented such as averaged value plus fluctuation, Figure 2.

$$\mathrm{v} = \bar{\mathrm{v}} + \mathrm{v}' \qquad (4)$$

In addition to velocity, other turbulent flow properties also show oscillatory characteristics. In order to model a turbulent flow, it is necessary to approximate turbulent stresses, which are too demanding in time and computer power requirements to model.

The engineering computation of turbulent flows therefore relies on simpler descriptions with introduction of the statistical consideration of the flow.

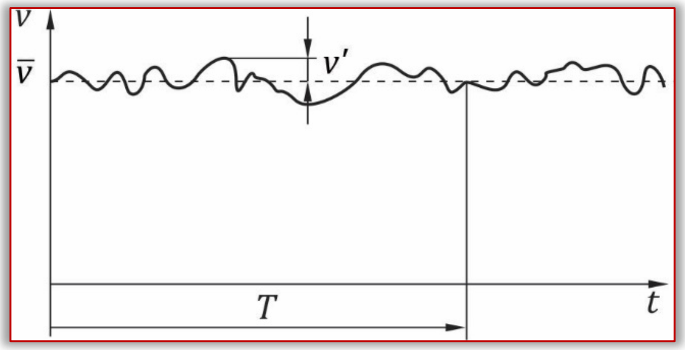

Figure 2. Instantaneous velocity in turbulent flow $\mathbf{v}$, averaged velocity $\bar{\mathbf{v}}$ and fluctuation $\mathbf{v}'$

Bussinesque proposed methods for presenting viscous stresses, and Reynolds contributed. The work of Prandtl, Kolmogorov, Taylor and von Karman [12] was aimed to characterize turbulence. With the growth of computer power, possibilities of numerical simulations increased, but simplified engineering approximations continue to remain popular and widespread, [13].

There are many approaches [14,15], but the two most common approaches are RANS (Reynolds Averaged Navier–Stokes) and LES (Large Eddy Simulation). The RANS approach is based on time–averaged Reynolds equations and requires closures to represent the turbulent stresses and scalar fluxes emerging from the averaging process. The discipline of turbulence modeling has evolved using a combination of intuition, asymptotic theories and empiricism, while constrained by practical needs such as numerical stability and computational efficiency, [13].

The large eddy simulation (LES) technique of turbulence modelling reduces the complexity of simulation by focusing on turbulence on larger length scales and larger time scales, while the smaller scale flow behavior can be described using a subgrid model. The LES technique is an exact method which is still computationally tractable, while the RANS is a less precise method which is more computationally efficient than LES.

Machine Learning (ML)

Fluid mechanics, with massive amounts of data increasing daily, either from experiments or simulations, is a field with massive potential for machine learning, rapidly becoming an integral part of everyday life.

Simply put, machine learning is building models from data using optimization. More precisely, machine learning algorithms are a growing set of data–intensive optimization and regression techniques ideal for these types of high–dimensional, non–linear, non–convex, and constrained optimizations [16].

The essential tasks in fluid dynamics are connected to reduced–order modeling, experimental data processing, shape optimization, turbulence closure modeling, and control [17]. Machine learning can be used for three main objectives:

— to accelerate direct numerical simulations
— to improve modeling basically in the context of LES and RANS
— to obtain more robust reduced order models, [18]

Machine learning application in fluid dynamics encounters many obstacles, as mentioned in [17]. But this is a very fast–growing field with constant advance which can be seen in papers produced recently in the field of reduced order modeling [19], or for detecting interface between turbulent and non–turbulent flow [20]. One of the most developed segments of ML is image processing. It is also an aspect of ML applicable to improve flow visualization, what is done in [21], where was conducted super–resolution analysis of grossly under–resolved turbulent flow field data.

A group of authors [22] used ML to stabilize fluid flow in the wake of a fluidic pinball, and in [23], to stabilize an open cavity flow experiment. In order to improve Reynolds–averaged Navier Stokes (RANS) turbulence models, ML is applied in the paper [24] using a data–driven approach. In [13] is presented how machine learning and data–driven methods are being used to tackle the closure problem and how machine learning can make a practical impact on everyday industrial flows.

Optimization problems are also solved increasingly well with the aid of machine learning, and instead of using the full Navier–Stokes equations, which are far too demanding in computer power and time, it is possible to build surrogate models with the aid of ML that are accurate and fast enough to use in real–time for feedback control.

Another mighty tool of ML is deep neural networks (DNN), the dominant data mining tool for big data applications [25], using an artificial neural network with multiple layers between the input and output layers.

In case of RANS modeling in [26] is presented how a custom deep neural network with additional tensor input based on prior physical knowledge can be improved compared with a generic neural network architecture that does not embed this invariance property.

RESULTS AND DISCUSSION

In the future, mechanics of fluids will take a central role in many fields of human activities, including energy sector, transportation, utility sector, etc. Most of these activities will be enabled by advanced fluid mechanics models and controls, and these tasks can generally be written as challenging optimization problems. These optimizations are nonlinear, non–convex, multi–scale, very high dimensional, and that is exactly where machine learning is significantly advancing.

In the 21st Century, computational methods and software tools are put on another level. An increase in computer power has made engineering and scientific computations more available and economically viable.

Modeling has become a mainstream step in engineering analysis and design of products, processes, and systems. However, the required training that engineering and science students often receive is not at the adequate level. Therefore, they may not have all the background training required to use software packages. This has created a challenge for industry to have trained professionals who can create “reliable” models and fully utilize commercially available software packages.

On the other hand, students, engineers, and scientists may not have the luxury of time and training to learn all the necessary technical subjects like physics, mathematical modeling, numerical methods, and programming languages.

Therefore, some arising thoughts and questions are:

— The theory is chasing the praxis, but the experiment remains the primary tool in fluid flow analysis, even though CFD has gone a long way with a lot of data and is constantly advancing. But, we need both experiment and computation [27].
— There is a transition from first principles to data–driven techniques.
— The abundance of data from experimental research and simulations provided asolid base for machine learning. Application of deep learning and neural networks provided additional advancement in closure problems. However, the physical involvement is logical and should be prioritized overthe mathematical approach.

— How should the students be thought?There can be two kinds of schools: one deep and broad, and the other will treat the application of CFD and ML as black box. That would lead to two kinds of engineers: engineers with wide knowledge, capable of thinking broadly and capable of introducing new concepts, and engineers who will specialize in a narrow field, with less ability to provide some general solutions. The first approach requires longer education, more devotion and more abundant resources.

CONCLUSION

Under the umbrella of authority such as Euler, Navier, Stokes, Reynolds, Prandtl, etc., it is difficult to stand out and deviate from the established path. And that is exactly the step that should be taken: to step away from the problem and try to look at it from another angle, by possible expansion of the system boundaries in order to, at least, get nearer to the solution of the turbulence closure problem.

The existing transition from first principles (such as the Navier–Stokes equation) to data–driven techniques is exactly such a step which leads us to the solution using another way: ML. What is important is that machine learning should not become a black box and must be connected to physically interpretable and generalizable models which are clear, trustworthy, and repeatable under different circumstances and can be interacted with. The most important issue is the future education of engineering and science students and to which extent it should be provided to them.

Educational institutions must embrace a dynamic approach to equip the next generation of engineers with the skills and mindset needed to tackle these complex problems. This involves fostering a deep understanding of both traditional principles and cutting–edge techniques like machine learning. Thus, the paramount challenge ahead is to seamlessly integrate machine learning while reshaping the education of future engineers.

References

[1] Bistafa, S. R., On the development of the Navier–Stokes equation by Navier, Revista Brasileira de Ensino de F´ısica, Vol. 40, No. 2, p.2603, 2018

[2] Bell, J.B., Colella, P. and Glaz, H.M., A second–order projection method for the incompressible Navier–Stokes equations. J. Comput. Phys., Vol. 85, pp. 257–283, 1989.

[3] Carlson, J. Jaffe, A. and Wiles, A. Editors: The Millennium Prize Problems, Clay Mathematics Institute American Mathematical Society, 2006.

[4] Liu, M., Li, X., Zhao, Q.: Exact solutions to Euler equation and Navier–Stokes equation. ZeitschriftfuerangewandteMathematik und PhysikZAMP, Z. Angew. Math. Phys., Vol. 70, pp. 43, 2019.

[5] Gunawan Nugroho, Ahmed M.S. Ali, Zainal A. Abdul Karim: A class of exact solutions to the three–dimensional incompressible Navier_Stokes equations. Applied Mathematics Letters., 23, 1388_1396 (2010).

[6] Mohammadein, S. A.1, Gad El–Rab, R.A., and Ali, M.S.: The Simplest Analytical Solution of Navier–Stokes Equations, Inf. Sci. Lett. 10, No. 2, 159–165, 2021

[7] Vladimir V. KULISH* & José L. LAGE: Exact solutions to the Naiver–Stokes equation for an incompressible flow from the interpretation of the Schroedinger wave function. School of Mechanical & Aerospace Engineering, Nanyang Technological University, Singapore 639798, Source arXiv, Jan 2013.

[8] Mohammadein, S. A. New Treatment of Fluid Mechanics with heat and Mass Transfer: Theory of Diffusion. Submitted to Appl. Math & Info Sci Lett.

[9] Bukurov, Ž., Cvijanović, P.: Mehanikafluidazbirkazadataka, Univerzitet u Novom Sadu, Fakultettehničkihnauka, Novi Sad, 1987.

[10] Moin, P.,Mahesh, K.: Direct numerical simulation: A Tool in Turbulence Research, Annu. Rev. Fluid Mech. Vol. 30, pp.539–78, 1998.

[11] [Hu, H.H.: Computational Fluid Dynamics, Fluid Mechanics, Fifth Edition, Elsevier Inc., 2012.

[12] Darrigol, O.: Worlds of flow: A history of hydrodynamics from the Bernoullis to Prandtl, Oxford University Press, 2005.

[13] Duraisamy, K., Iaccarino, G., Xiao, H.: Turbulence Modeling in the Age of Data, Annual Review of Fluid Mechanics, No. 51, pp. 1–23, 2019.

[14] Girimaji, S.S.; Partially–Averaged Navier–Stokes Model for Turbulence: A Reynolds–Averaged Navier–Stokes to Direct Numerical Simulation Bridging Method, Journal of Applied Mechanics, Vol. 73 No.3, pp 413–421, 2006.

[15] Spalart, P. R.: Detached–Eddy Simulation, Annual review of Fluid Mechanics, Vol. 41, pp. 181–202, 2009.

[16] Brunton, S., Kutz, J.N., Manohar, K., Aravkin, A.Y., Morgensen, K., Klemisch, J. Goebel, N., Buttrick, J., Poskin, J., Blom–Schieber, A.W., Hogan, T., McDonald, D.: Data–Driven Aerospace Engineering: Reframing the Industry with Machine Learning, Published Online:20 Jul 2021,

[17] Brunton, S. L., Noack B.R., Koumoutsakos, P.: Machine Learning for Fluid Mechanics, Annual Review of Fluid Mechanics, Vol. 52, pp.477–508, 2020.

[18] Vinuesa, R., Brunton, S.: Enhancing computational fluid dynamics with machine learning, Nature Computational Science, Vol. 2, pp. 358–366, 2022.

[19] Loiseau, J.C., Noack, B.R., Brunton, S.L.: Sparse reduced–order modeling: Sensor–based dynamics to full–state estimation, Journal of Fluid Mechanics, Volume 844 , pp. 459 – 490, 2018.

[20] Li, B., Yang, Y., Zhang, X., He, G., Deng, B., Shen, L.: Using machine learning to detect the turbulent region in flow past a circular cylinder, J. Fluid Mech., Vol. 905, A10, 2020.

[21] Fukami, K., Fukagata, K., Taira, K.: Super–resolution reconstruction of turbulent flows with machine learning, J. Fluid Mech., Vol. 870, pp. 106–120, 2019.

[22] Maceda, G., Li, Y., Lusseyran, F., Morzynski, M., Noack, B.R.: Stabilization of the fluidic pinball with gradient–enriched machine learning control, J. Fluid Mech., Vol. 917, A42, 2021.

[23] Maceda, G., Varon, E., Lusseyran, F., Noack, B.R.: Stabilization of a multi–frequency open cavity flow with gradient–enriched machinelearning control, J. Fluid Mech., Vol. 955, A20, 2023.

[24] Grabe, C., Jäckel, F., Khurana, P.: Data–driven augmentation of a RANS turbulence model for transonic flow prediction, International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 33 No. 4, pp. 1544–1561, 2023.

[25] Kutz, J.N.: Deep learning in fluid dynamics, J. Fluid Mech., Vol. 814, pp. 1–4, 2017.

[26] Ling, J., Kurzawski, A., Templeton, J.: Reynolds averaged turbulence modelling using deep neural networks with embedded invariance, Journal of Fluid Mechanics, Vol. 807, pp. 155–166, 2016
[27] Goldstein, S.: Fluid Mechanics in the First Half of this Century, By Annu. Rev. Fluid Mech. Vol. 1, p.1–29. 1969.

Note: This paper was presented at IIZS 2023 – The XIII International Conference on Industrial Engineering and Environmental Protection, organized by Department of Mechanical Engineering and Department of Environmental Protection of the Technical Faculty "Mihajlo Pupin" Zrenjanin, from the University of Novi Sad, in cooperation with partners – University Politehnica Timisoara, Faculty of Engineering, Hunedoara (ROMANIA), University "St. Kliment Ohridski", Technical Faculty, Bitola (MACEDONIA), "Aurel Vlaicu" University of Arad, Faculty Of Engineering, Arad (ROMANIA), University of East Sarajevo, Faculty of Mechanical Engineering East Sarajevo, Sarajevo (BOSNIA & HERZEGOVINA) and University of Giresun, Faculty of Engineering, Giresun (TURKEY) – in Zrenjanin, SERBIA, in 05–06 October, 2023.

ISSN: 2067-3809