Simulation of viscous flows with boundary layers within multiscale model using generalized hydrodynamics equations

Alexander I. Fedoseyev\textsuperscript{a,1,*}, Boris V. Alexeev\textsuperscript{b}

\textsuperscript{a}CFD Research Corporation (CFDRC), Huntsville, Alabama, USA
\textsuperscript{b}Moscow Academy of Fine Chemical Technology, Moscow, Russia

Abstract

The multiscale method for computational fluid dynamics (CFD) is proposed to solve viscous flow problems at high Reynolds numbers with a thin boundary layer. This method is a physics-based model, uses generalized hydrodynamic equations proposed by Alexeev (1994), and can be interpreted as a regularization of the Navier-Stokes equations. Numerical solutions using this approach compare favorably with experimental data for the cases we considered for different flow problems in the range of Reynolds number from \(Re = 3200\) to \(1,000,000\). The method is discussed and numerical solutions are compared with the experimental data for a 3D driven cavity flow at \(Re = 3200\) and 10,000, 2D backward facing step flow at \(Re = 44,000\), 2D channel flow at \(Re\) number up to \(10^6\), and a 3D thermal convection in a cylinder at \(Ra = 1000\) to 150,000. Comparison with the analytical asymptotic solution is provided for a thermal convection, in the electrically conducting fluid suppressed by a strong magnetic field at Hartman numbers \(Ha\) up to 20,000. This multiphysics model is not a turbulence model, and no additional equations are introduced. Kinetic effects (small flow scales) are successfully captured with new terms introduced into the governing equations, and the derived small scale of turbulence compares well with observed in the experiments by Koseff and Street (1984).

Keywords: High Reynolds number flows, Alexeev equations, Regularization, Multiscale, Kinetic Effects.

PACS: 47.85.-g, 02.60.-x, 47.11.-j

1. Introduction

Difficulties with the numerical solutions of Navier-Stokes (NS) equations for high \(Re\) number flows have usually been referenced to insufficient mesh resolution, complicated flow physics, turbulence, etc. In series of papers, Shishkin et al. (1995-1997, see e.g. [1]) demonstrated theoretically that grid methods perform poorly when dealing with the boundary layer and provided an estimation for numerical solution error as \(O(1)\) for uniform meshes [1]. As a remedy for these difficulties, the construction of special meshes was proposed, with more optimistic error estimation, as \(O(h^{1/m})\), \(h\) is the mesh size, with \(m = 7\) or more. During last decades the failure of numerical solutions to obtain good agreement with experimental data confirms Shishkin estimates (lid-driven cavity flow: experiments [2].

\textsuperscript{*}Email address: \texttt{ai@cfdr.com} (Boris V. Alexeev)
\textsuperscript{1}Corresponding author

1877-0509 © 2012 Published by Elsevier Ltd.
doi:10.1016/j.procs.2010.04.071
versus simulations [3]; thermal convection in a vertical slot: experiments [4] versus simulations [5], etc.). Successful results implementing Shishkin-type strategy are obtained in [6]. The boundary layer thickness, which is not known in advance, is used, to construct Shishkin mesh. Meshless methods have similar drawbacks [7].

We have developed an alternate approach, that is more accurate, less mesh-dependent and can be interpreted as a regularization of the Navier-Stokes equations, which captures all important scales of the flow, resulting in multiscale resolution of flow physics. It is based on the simplified mathematical model obtained the generalized hydrodynamic equations by Alexeev [8], [9]. The numerical solutions agree well with the experimental measurements for a set of flow problems at high Reynolds number and for flows with thin boundary layer [10], [11], [12], [13], [14]. This is not a turbulence model, and no additional equations are introduced.

The model was successfully compared to: (1) 3D driven cavity flow data by Koseff and Street (1982) at Re = 3200 and Re = 10,000; (2) 2D backward facing step flow by Kim et al. (1980) at Re = 44,000; (3) 3D thermal convection in a cylinder at Ra = 1000 to 150,000; and (4) asymptotic solution for a thermal convection in the semiconductor melt suppressed by the magnetic field at Hartman numbers Ha up to Ha = 20,000. Numerical results for 2D channel flow at Re number up to 10⁶ are also presented.

2. A regularization approach to solving Navier-Stokes equations

Governing equations. To consider a flow of incompressible viscous fluid, in a closed 2D or 3D domain Ω, Navier-Stokes momentum and continuity equations are:

\[
\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} - Re^{-1} \nabla^2 \mathbf{V} + \nabla p - \mathbf{F} = 0 \tag{1}
\]

\[
\nabla \cdot \mathbf{V} = 0 \tag{2}
\]

where \( Re = \frac{V_0 L}{\nu} \) is the Reynolds number, \( V_0 \) is the velocity scale, \( L \) is the hydrodynamic length scale, \( \nu \) is the kinematic viscosity, and \( \mathbf{F} \) is a body force. In the case of thermal convection the body force is \( \mathbf{F} = GrRe^{-1} \cdot \Theta \cdot \mathbf{e}_g \) (Boussinesq approximation), where \( \Theta \) is a nondimensional temperature, \( Gr \) is the Grashoff number and \( \mathbf{e}_g \) is the unit vector in the direction of gravity. The energy equation is:

\[
\frac{\partial \Theta}{\partial t} + (\mathbf{V} \cdot \nabla) \Theta = Pr^{-1} Re^{-1} \nabla^2 \Theta \tag{3}
\]

where \( \Theta \) represents nondimensional temperature, scaled by \( \Theta = (T - T_{cold})/\Delta T \) with \( \Delta T = T_{hot} - T_{cold} \). The Prandtl, Grashoff and Rayleigh numbers are respectively \( Pr = \nu/k \), \( Gr = Ra/Pr \), and \( Ra = \beta \Delta T g L^3 k^{-1} \nu^{-1} \), where \( \beta, g, k \) are the coefficients of thermal expansion, gravitational acceleration, and of thermodiffusivity.

We have analyzed the generalized hydrodynamic equation, proposed in [8] (a review in [9]), for the case of incompressible viscous flow and kept only a few main order terms, spatial fluctuations, in the continuity equation. This may be interpreted as a regularization of the Navier-Stokes equations.

A proposed regularization involves modifying the continuity equation (2) to become

\[
\nabla \cdot \mathbf{V} = \tau^* \nabla \cdot (\nabla p - \mathbf{F}) \tag{4}
\]

where \( \tau^* \) is a small regularization parameter. For \( \tau^* \to 0 \), eq. (4) approaches the continuity equation (2). The boundary condition for pressure on a wall is

\[
(\nabla p - \mathbf{F}) \cdot \mathbf{n} = 0 \tag{5}
\]

where \( \mathbf{n} \) is a unit vector normal to the wall. Equations (4) and (5) present the basis of our method, which takes into account only a few of many additional terms of the generalized hydrodynamic equations, called fluctuations (temporal and spatial), in [8]. Equation (5) is a condition of absence of the fluctuations on walls, according to [8].

Preliminary results with a more complicated model are presented in [10] and [11]. Further numerical experiments have shown that satisfactory results can be obtained with the simpler model presented above [12]. Numerical solutions of eqs. (1), (3), and (4), called regularized Navier-Stokes equations (RNS), with the boundary condition (5), give a
Figure 1: Lid-driven cavity problem, $Re = 3200$: (a) horizontal velocity profiles (1-4) for numerical solution (solid and dashed lines): 1-NS, 2 - standard $k-\varepsilon$ model, 3-RNS (2D), 4-RNS (3D); squares - experimental data by Koseff and Street (1982) in the symmetry plane $Y = 1.5$. (b) residuals of numerical solution in the momentum equation (1) for the 2D NS and RNS solutions at $x = 0.5$. Residuals are estimated by differentiation of numerical solutions, using interpolating polynomials. (c) $Re = 10^4$, numerical horizontal velocity profiles (1-3) and experimental data (triangles). A $k-\varepsilon$ model solution is obtained with commercial code CFD2000.
better agreement with the experimental measurements for high Reynolds number flows than the traditional solution of Navier-Stokes eqs. (1) and (2), (NS), with the finite element method (FEM).

Note that numerical formulations, containing extra terms in the momentum and continuity equations, have been proposed in the frame of kinetically consistent numerical schemes, developed by Elizarova and Chetverushkin [17], [18]. The justification of introducing extra terms into hydrodynamic equations is discussed in [19] from a physical kinetics viewpoint. The pressure Laplacian and other terms in the discretized continuity equation, have also been proposed by Löhrner as a result of consistent treatment of the time-advancement for the continuity eq. (2) [16].

Further, eqs. (1), (3), and (4) with the boundary condition (5) are treated as mathematical model, having a control (regularization) parameter $\tau^*$. This regularization parameter $\tau^*$ is expressed dimensionally as $\tau^* = \tau l^{-1} V_0$, where $\tau$ is time. The dimension of $\tau v$ is a length squared. We introduce the regularization length scale $l$ with $l^2 = \tau v$ as $\tau^* = l^2 L^{-2} Re = K \cdot Re$, where $K = l^2 / L^2$. The optimal value of $\tau$ (or $l$) is not known in advance. We found that, for problems with smooth boundary conditions, the numerical solution only slightly depends on the value of $\tau^*$, with $\tau^*$ in the range of $10^{-8}$ to $10^{-2}$. This proposed regularization has an additional useful feature for the FEM. It uses the same order finite element approximation for velocity and pressure for RNS.

3. Numerical experiments with RNS

In this section, we present the numerical results of a few flow problems, and compare these with experimental data. The FEM was used to discretize the governing equations. Algebraic equations for momentum and continuity were solved simultaneously using the iterative CNSPACK solver [20], with preconditioning by a high order incomplete decomposition. Iteration termination criticet was a convergence of relative residuals to $10^{-8}$ (or $10^{-12}$).

**Driven cavity problem.** Shown in Fig. 1 are the numerical solutions, with RNS employed [10],[12], against the 2D and 3D driven cavity flow data at $Re = 3,200$ and $10,000$ [2], and results by other methods. We used 81x81 node uniform triangular mesh for the RNS. The same mesh, with quadratic for velocity and linear for pressure finite elements, was used in a standard FEM solution of eqs. (1, 2). These results are labeled as NS in Fig. 1(a) and 1(c). The RNS model parameter, $\tau^*$ (or $l^2$ in $l^2 L^{-2}$), was varied to match one of the experimental velocity profiles. A value, $l^2 L^{-2} \approx 1.5 \cdot 10^{-5}$, resulted in good agreement for all the velocity profiles and both flow regimes, $Re = 3,200$ and $10,000$. To our surprise, the dimensional value of $l \approx 0.58 \text{mm}$ was a good approximation to the experimentally observed “Kolmogorov microscale” $l_{exp} \approx 0.5 \text{mm}$ (see [2], p. 398).

For 3D flow we solved the RNS at $Re = 3200$ for half of the cavity with a mesh of 41x41x33 nodes (221,892 unknowns), refined near the walls. The symmetry condition was used on the vertical symmetry plane $y = 1.5$. The same 2D value of $K = 1.5 \cdot 10^{-5}$ was used in the 3D computations, the modeling results are presented in Fig. 1 as well. One can see that the velocities obtained for both the 2D and 3D profiles are close to the experimental data, except in
one region. We did not obtain the 3D stationary solution at $Re = 10^4$. The solution went unstationary at $Re \geq 8,450$. According to Baggett and Trefethen [22], the stationary solution exists, but the basin of attraction of this solution may be extraordinarily narrow, having a width of $O(Re^{\alpha})$ for some $\alpha < -1$.

**Mass conservation** in the continuity equation was thoroughly analyzed. The local values of $\text{div} \mathbf{v}$ for the numerical solution were examined. The conclusion was that a numerical solution for the RNS model has the same order of error in the mass conservation as the NS solution with the same number of mesh nodes [12].

The residuals of numerical solutions in the momentum equation (1) are presented in Fig. 1b. It shows one to two order reductions for the RNS solution residual in the boundary layer, compared to the NS solution residual.

**Flow over a backward facing step.** The numerical results for a 2D flow over a backward facing step of height $H$, $H = L/3$ ($L$ is a channel height) at $Re = 4.4 \times 10^4$ (or $Re_L = 1.32 \times 10^5$ in [24]) were obtained and compared with the experimental measurements of [23]. We used a 110x60 mesh, refined near the walls, and started the computations at a low Reynolds number. We raised $Re$ in small increments until reaching 44,000. Fig. 2(a) presents the computed velocity profile, at $x = 5.33H$ ($x = 0$ at the edge of the step), and the experimental mean velocity measurements [23].

The RNS model output satisfactorily agrees with the experimental data for both the velocity profile and the recirculation zone length $X_r$. We computed $X_r / H = 7.50$, while $X_r^{exp} / H \approx 7 \pm 0.5$ was obtained experimentally. The value of $\tau^*$ used in the computations was in the range of $10^{-2}$ to $10^{-4}$. This did not noticeably influence the results. For smaller values of $\tau^*$, it was more difficult to reach the steady state solution at $Re = 44,000$; more and smaller increments in $Re$ had to be used. If the increments in the $Re$ number were large, the flow pattern bifurcated to an unsteady flow, with vortices periodically originating from the recirculation zone and flowing downstream.

The solution with a standard $k-\varepsilon$ model shows the velocity profile at $x = 5.33$ that has no backward flow. A standard $k-\varepsilon$ model underpredicts the recirculation zone length $X_r$ by a substantial amount, 20-25% according to [24], where more sophisticated turbulence models have been proposed for this problem.

**Flow in a 2D channel,** of height $H = 1$ and length $L = 4$, was the subject of a few experiments with the RNS at Reynolds numbers $Re = 5 \cdot 10^3$, $10^4$, $10^5$, and $10^6$. An 81x100 mesh refined near the walls was used. Inlet flow profiles were (i) $U = 1$, and (ii) $U = 6(1 - z)$. We were obtained both parabolic and “turbulent” flow profiles for $Re$ up to $10^5$, depending on the inlet flow conditions and the value of $\tau^*$. To obtain a “turbulent” flow profile at $Re = 5,000$, with the inlet condition (ii), we started with $\tau^* = 0.01$, and were then able to keep this “turbulent” profile type at reduced $\tau^*$, down to $10^{-4}$ (Fig. 2b). The boundary layer thickness is about $\delta \sim Re^{-1/2}$ (obtained graphically from Fig. 2b).

**3D Thermal convection in a differentially heated horizontal cylinder.** A linear temperature profile is given assumed on a cylinder wall. Experimental data by Bogatirev et al. [27] and finite volume simulations by Bessonov [26] are used for comparison at Rayleigh numbers in the range of $10^3$ to $1.2 \cdot 10^5$, $Pr = 0.9$ (Fig. 3a). For the linear temperature distribution on a cylinder wall, we obtained good agreement with the numerical results [26]. We used a 17,4357 node hexaedral mesh refined near the walls. Agreement with the experimental data was not good for $Ra > 2000$ (solid line in Fig. 3b). Therefore computations were done for a real, finite wall conductivity [26]. The thermodiffusivity data for stainless steel was used, and the adjoint problem was solved. Thereby, agreement with the experimental data has been significantly improved (Fig. 3b, dashed line). The value of $\tau^*$ used was $10^{-7}$ to $10^{-3}$, which did not noticeably affect the RNS results.

**Magnetic field suppression of the semiconductor melt flow,** modeling with RNS, is considered in [13] and [14]. The application of magnetic fields is a promising approach for reducing convection during directional solidification of electrically conductive melts. Current technology allows for experiments using very strong static fields, for which nearly convection free segregation is expected in melts exposed to stabilizing temperature gradients (vertical Bridgman method) [28]. The governing equations solved for this problem are eqs. (1), (3), and (4), where $\mathbf{F}$, the body force due to the magnetic field (Lorenz force), is given by

$$\mathbf{F} = PrHa^2[(\nabla \times \mathbf{e}_B) \times \mathbf{e}_B].$$

Here $\mathbf{F} = (Pr(\mathit{Ha})^2U, 0)$ for the 2D case of a vertical magnetic field and $U$ is the horizontal component of the velocity.

The Hartmann number, given by $Ha = LB_0 \sqrt{\frac{\sigma}{\rho}}$, is in the range of $Ha = 100$ to $10^4$, for the materials and magnetic fields under investigation. Here $\rho, \sigma$ are the density and electrical conductivity, $B_0$ is the magnetic field intensity, and $\mathbf{e}_B$ is the unit vector in the direction of the magnetic field.

The computations are difficult because of the thin boundary layer, although the velocity of the generated flows is extremely low, $Re \sim 10^{-1}$ to $10^{-6}$. A high value of the Hartmann number results in a relatively small coefficient at the
Figure 3: (a) Thermal convection in a 3D differentially heated cylinder; (b) comparison of the temperature difference $\Delta T$, versus $Ra$ with the experimental data [27] and finite-volume computations: squares - experimental data, solid line - numerical results for perfect wall conductivity [26], triangles - RNS results; dashed line - numerical results when the steel properties for the wall conductivity have been used (adjoint problem); here $\Delta T$ is a temperature difference between locations marked as 1 and 2 in (a). Both the NS and RNS solutions are nearly identical here.
highest derivative of the velocity in the momentum equation. Solutions of such problems exhibit thin boundary layers of thickness $\delta \sim Ha^{-1}$, along with “equivalent” Reynolds number $Re_{eqv} \sim Ha^2$, $Re_{eqv} = 4 \cdot 10^4$ for $B=0.5$ Tesla, and $Re_{eqv} = 4 \cdot 10^6$ for $B=50$ Tesla. Some of the results for 2D models are shown in Fig. 4. The RNS numerical solution is rather smooth even for a very thin boundary layer, with thickness $\delta \sim 10^{-4}$ ($Ha = 2 \cdot 10^4$). Other methods tested in [13], [14] (including industrial code) did not provide acceptable results or failed for $Ha > 100$.

Conclusions

The RNS method demonstrated as an efficient approach for multiscale simulations 2D and 3D flows at high Reynolds number, which resolves all important flow scales including thin boundary layers. The numerical results compared favorably with experimental data for driven cavity flow, flows in channels, thermal convection, and asymptotic solutions for electrically conductive fluid flows under strong magnetic fields. The RNS method is used for modeling 3D thermo-vibrational convection in Bridgman melt configurations [15]. Similar ideas have been used successfully to improve the accuracy of the meshless multiquadratic radial-basis function methods [29], [30]. We consider this approach as multiphysics model for wide range of application to flow simulations with different flow scales and a variety physical fields involved.
References