NEW OPENFOAM SOLVERS FOR TRANSONIC AND INCOMPRESSIBLE FLOW SIMULATIONS

MATVEY V. KRAPOSHIN[1,2], DANIIL A. RYAZANOV[1,3], KIRILL A. VATUTIN[1,3], TATIANA G. ELIZAROVA[4]
[1]*Ivannikov Institute for System Programming of the RAS*
[2]*The Department of Aeromechanics and Flight Engineering of Moscow Institute of Physics and Technology*
[3]*Lomonosov Moscow State University*
[4]*Keldysh Institute of Applied Mathematics RAS*

Keywords: *Regularized gas dynamic equations, Quasi hydrodynamic equations, transonic flows, incompressible flows, Finite Volume Method, QGDFoam, QHDFoam*

Introduction

Regularized or quasi-gas dynamics equations (QGD) and their extensions for transonic and incompressible flows have found applications for numerical simulations of different types of viscous flows [1]: subsonic, transonic, super- and hypersonic flows of viscous perfect gas, flows of binary mixtures, low-compressible fluids flows, etc. Our implementation of QGD numerical algorithm as OpenFOAM solver *QGDFoam* has proved to operate properly in wide range of Mach and Reynolds numbers for several prominent practical 1D & 2D cases, such as laminar subsonic flows in channels, subsonic flows with separation, compressible jet flows, supersonic inviscid flows and others [2]. Perfect scalability for medium-sized High Performance Computing clusters (HPC) with number of computational cores up to 96 has been achieved.

In this work we continue to develop OpenFOAM implementations of numerical algorithms for regularised gas and hydrodynamic equations for the next particular cases:

- *TQGDFoam* — a solver for transonic and subsonic viscous gas flow with possibility of extension to arbitrary equation of state;
- *QHDFoam* — a solver for incompressible viscous fluid simulation.

Developed solvers are tested for well-known 2D and 3D cases on large HPC cluster with the number of used computational cores up to 1500.

Quasi hydrodynamic equations for TQGDFoam and QHDFoam solvers

Quasi hydrodynamic (QHD) equations were proposed by Yu. V. Sheretov in 1996 as a special variant of QGD system e.g., [3]. This system has more simple mathematical form comparing to QGD system that simplifies its numerical implementation. It allows directly usage of arbitrary equation of state $p = p(\rho, T)$. Compared with QGD system, QHD equations have a limited range of applicability in terms of Mach numbers, namely, from subsonic to transonic regimes.

QHD system of equations includes continuity(1), momentum (2) and total energy equations (3), accomplished with closure relations for Navier-Stokes viscous stress tensor Π_{NS} and heat flux vector q_{NS}.

$$\frac{\partial \rho}{\partial t} + div\ (\rho\vec{u}) = div\ (\rho\vec{w}) \tag{1}$$

$$\frac{\partial(\rho\vec{u})}{\partial t} + div\ (\rho\vec{u}\otimes\vec{u}) + \vec{\nabla}p = \rho\vec{F} + div\,\Pi_{NS} + div\ [(\rho\vec{w}\otimes\vec{u}) + (\rho\vec{u}\otimes\vec{w})] \tag{2}$$

$$\begin{aligned}\frac{\partial}{\partial t}\left[\rho\left(\frac{\vec{u}^2}{2}+\varepsilon\right)\right] + div\ \left[\rho\vec{u}\left(\frac{\vec{u}^2}{2}+\varepsilon\right) + p\vec{u}\right] + div\ q_{\vec{NS}} = \rho\vec{F}\cdot(\vec{u}-\vec{w}) + \\ div\ (\Pi_{NS}\cdot\vec{u}) + div\ \left[\rho\vec{w}\left(\frac{\vec{u}^2}{2}+\varepsilon\right) + p\vec{w} + \rho\vec{u}(\vec{w}\cdot\vec{u})\right]\end{aligned} \tag{3}$$

with

$$\vec{w} = \tau\left[(\vec{u}\cdot\vec{\nabla})\vec{u} + \frac{1}{\rho}\vec{\nabla}p - \vec{F}\right] \tag{4}$$

In numerical algorithms the regularisation parameter τ has the form $\tau = \alpha\Delta_h/C_s$, where α is a tuning parameter, Δ_h is a local computational grid step and C_s is the speed of sound.

Similarly to Navier-Stokes system, it is possible to construct QHD system for incompressible flows. If the density changes can be neglected, the QHD system of equations reduces to its incompressible form, which can be used for deep subsonic flows simulation: (5) — (7). This system of equations can also be used for stratified flows simulation within Boussinesq assumption with $\vec{F} = -\beta\vec{g}T$.

$$div\,\vec{u} = div\,\vec{w}, \tag{5}$$

$$\frac{\partial\vec{u}}{\partial t} + div\,(\vec{u}\otimes\vec{u}) + \frac{1}{\rho}\vec{\nabla}p = \frac{1}{\rho}div\,\Pi_{NS} + div\,[(\vec{w}\otimes\vec{u}) + (\vec{u}\otimes\vec{w})] - \beta\vec{g}T, \tag{6}$$

$$\frac{\partial T}{\partial t} + div\,(\vec{u}T) = div\,(\vec{w}T) + \chi\Delta T, \tag{7}$$

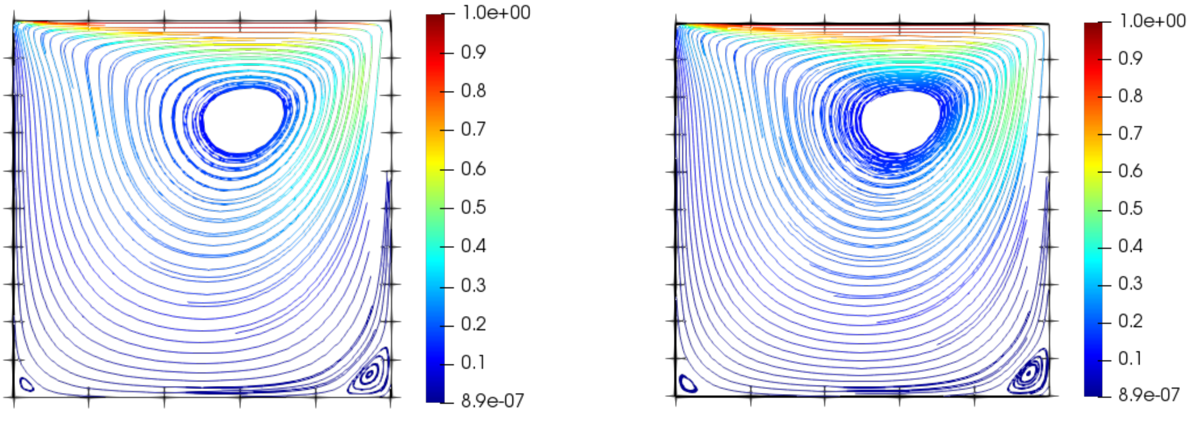
where $\rho = const > 0$ — average value of density, $\vec{u} = \vec{u}(\vec{x}, t)$ - hydrodynamic velocity, $p = p(\vec{x}, t)$ - static pressure, $T = T(\vec{x}, t)$ - deviation of temperature from its average value T_0, $\vec{g}$ — acceleration due to gravity, β — temperature coefficient of expansion, coefficients of dynamic viscosity μ and thermal diffusivity χ are assumed to be constant and $\nu = \mu/\rho$ is kinematic viscosity. In contrast to compressible simulations, the smoothing parameter can be calculated using system Re number, e.g.: $\tau = \alpha\cdot\Delta_h^2/\nu$, or $\tau = \alpha\Delta_h/U_0$, where U_0 is the characteristic flow velocity.
The approximations of the both systems were implemented in OpenFOAM as *TQGDFoam* and *QHDFoam* solvers. For demonstration purposes here *TQGDFoam* uses only perfect gas equation of state, but it can be modified effortless to use in conjunction with arbitrary EoS.

Conclusion

Several cases were selected for assessing developed solvers properties:

1. Laminar unsteady flow over cylinder.
2. Laminar flow in the quadrangular cavity.
3. Laminar and transitional flow over bacward-facing step.

Numerical solution was compared with the standard OpenFOAM solvers. The solvers was tested in parallel regime using HPC cluster with up to 1500 cores. Preliminary results of comparison between *QHDFoam* solver and *icoFOam* solver are presented on figures below.



a) b)

Figure 1: Visualization of streamlines and the velocity field magnitude for the cavity case at Re=100 calculated on 100x100 grid using a) icoFoam solver and b) *QHDFoam* solver

Two new solvers based on regularized hydro-dynamic equations for viscous subsonic (*TQGDFoam*) and incompressible *QHDFoam* flows have been implemented in OpenFOAM library. First order Euler scheme for time derivatives and second order scheme for spatial derivaties are used in the implementation. Solvers were tested on classical 2D and 3D cases.

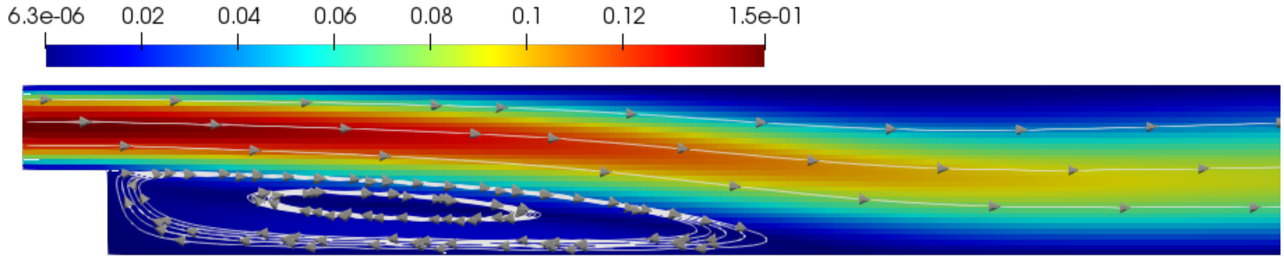


Figure 2: Visualization of velocity field and streamlines for backward step case at Re=100 calculated on 100x100 grid using *QHDFoam*

Acknowledgments

Calculations of this research has been carried out using computing resources of the federal collective usage center Complex for Simulation and Data Processing for Mega-science Facilities at NRC Kurchatov Institute, `http://ckp.nrcki.ru/`
This study was supported by the Program of Fundamental Research of the Presidium of the Russian Academy of Sciences No. 26

References

[1] Tatiana G. Elizarova. *Quasi Gas Dynamic equations*. Springer; 2009 . ISBN 978364200295.

[2] Matvey V. Kraposhin,Elena V. Smirnova, Tatiana G. Elizarova, Maria A. Istomina. Development of a new OpenFOAM solver using regularized gas dynamic equations, *Computers and Fluids*, 166(2018), pp. 163175

[3] T. G. Elizarova, Y. V. Sheretov. Theoretical and numerical analysis of quasi-gasdynamic and quasi-hydrodynamic equations, *J. Computational Mathematics and Mathematical Physics* 41 (2) (2001) 219234.