Simulation of transonic low-Reynolds jets using quasi-gas dynamics equations

M V Kraposhin1,2,4*, Epikhin A S1,3, Elizarova T G4 and Vatutin K A1,5

1Ivannikov Institute for System Programming of the RAS, Russia, Moscow, Solzhenitsyna str., 25
2The Department of Aeromechanics and Flight Engineering, Moscow Institute of Physics and Technology, Russia, Moscow Region, Jukovskiy, Gagarin str., 16
3Bauman Moscow State Technical University, Russia, Moscow, 2nd Bauman str., 5, building 1
4Keldysh Institute of Applied Mathematics of the RAS, Russia, Moscow, Miusskaya Sqr., 4
5Faculty of Mechanics and Mathematics, Lomonosov Moscow State University, Russia, Moscow, GSP-1, Leninskiye gory, 1

*E-mail: m.kraposhin@ispras.ru

Abstract. In this paper application of the OpenFOAM solver QGDFoam for numerical simulation of transonic viscous flows is considered. The developed solver, which implements regularized or quasi-gas dynamics (QGD) algorithms, is validated using the transonic low-Re jet flow case (Ma=0.9, Re=3600). The conducted numerical simulations allow the assessing applicability of the solver for modelling hydrodynamic instabilities and their interaction with transonic flow. Results of the numerical simulations are compared with experimental observations and Navier-Stokes-based code simulation. Results of the present study formulate a guideline for choosing (values of) QGD-algorithm tuning parameters.

1. Introduction

Accurate resolution of interaction between high-speed flow, hydrodynamic instabilities, and free-stream field is crucial for numerical investigations of mixing processes in trans- or supersonic jets. These processes arise in many industrial applications, such as evaluation of mechanical and heat loads due to the interaction between jets and obstacles during rocket lift-off, prediction of acoustic noise from free jets, mixing layer influence on scattering cross section analysis, etc.

The numerical methods employed for such cases should be applicable both for high-speed flows, usually described by Euler equations, and for viscous flows at sub-sonic speed.

Many general-purpose industrial programs employing explicit methods are based on approximate Riemann solvers of Godunov type (Rusanov scheme, Harten-Lax-Leer (HLL), HLL-Contact (HLLC) and others) and have limitations coming from the restriction on the Mach number range. Due to this restriction, the flows, where Ma is significantly less than 1, are not well resolved. At the same time, projection type methods such as SIMPLE, PISO and other, which are known to be effective for incompressible viscous fluids, are not applicable for compressible flows.
Application of mentioned methods is often tangled by the necessity of proper TVD limiter choice, which introduces (in a model) an additional numerical diffusion of an unknown amount and reduces parallel performance due to implementation issues.

The alternative to described methods has been developed in the Keldysh Institute of Applied Mathematics more than 30 years ago [1,2]. This approach utilizes approximation of regularized or quasi- gas dynamics (QGD) equations. The associated numerical algorithms (or sometime QGD algorithms) are distinguished with the homogeneity of approximation expressions and the simplicity of usage (only 2 tuning parameters) and with physically justified numerical viscosity.

The joint team of Keldysh Institute of Applied Mathematics and Ivannikov Institute of System Programming is developing a family of OpenFOAM solvers based on QGD-algorithms [3,4] and is studying their application to various cases, both academic and industrial.

2. Numerical setup

The considered case [5] is the transonic (Ma=0.9) perfect gas viscous turbulent low-Re (Re=3600) jet flow from the exhaust of circular shape (see figure 1). As the jet propagates in the axial direction from the nozzle exhaust, the Kelvin-Helmholtz instability waves are developed, resulting in momentum waves in radial directions. These waves grow exponentially until they transform into large eddies, which then dissipate into smaller vortexes.

This process is known as turbulence and it is necessary to implement a modern accurate numerical model to resolve all structures from largest to smallest ones or to some cutting scale (if Large Eddy Simulation technique is used).

In order to assess the quality of code properly, the selected validation case should have regimes with laminar or transitional Reynolds number. When Re number is small, it is possible to simulate flows without additional assumptions and models (such as LES models), and thus additional uncertainty related to sub-grid scale turbulence modeling is avoided.

However, most experimental and analytic cases for laminar and transitional viscous flows consider subsonic (Ma < 0.3) regimes or even incompressible flows. This restriction is natural: keeping constant Reynolds number while increasing the value of Mach number usually requires specific conditions: small characteristic length, small pressure or high temperature. Therefore, experimental data [5] are of special value, because this allows studying the behavior of numerical model without empirical turbulence modeling.

In the considered case the computational domain is a box, which extends 60 nozzle diameters ($D_n$) in X direction, 21 $D_n$ in Y direction and 21 $D_n$ in Z direction. The jet propagates in the X direction starting from inlet plane, submerged in the computational domain, see figure 1 and figure 2.

External boundaries of the computational domain consist of the next parts (see figure 1):
- the inlet of the gas jet of circle section with diameter of 7.9 mm;
- external walls of the nozzle, submerged in the domain to resolve flow separation of the jet from tips of the nozzle;
- outlet boundaries of the domain, representing the far field of undisturbed media.

To save computational resources, the cone-shaped region of the computational domain around the jet flow was refined, figure 1. This region extended 25 $D_n$ from the nozzle exhaust (inlet) in the X direction.

Meshes with various resolutions were used for calculations. Results are presented only for the final mesh with 32 CPD, which corresponds to the best affordable match to experimental data. Following boundary and initial conditions were imposed for gas dynamic state variables:
- pressure: fixed value at the outlet (1823 Pa), the constant gradient at the inlet, ensuring experimental mass flow rate (see [4] for details) and zero gradient at the walls, 1823 Pa as the starting condition;
- temperature: fixed value at the inlet (256 K), zero gradient for the outlet and walls and 297 K as the starting condition;
• velocity: fixed value at the **inlet** (286 m/s for axial component, 0 m/s for other components), zero gradient for the **outlet**, fixed value (0 m/s for all components) at the **walls** and zero speed as the starting condition.

The $\tau$-coefficient of the model was chosen to be proportional to the spatial grid resolution $\Delta x$ and inverse sound speed $c$:

$$\tau = \alpha^{QGD} \frac{\Delta x}{c}.$$ 

Since no shocks are expected in the solution, the QGD viscosity coefficient was set to 0, $\alpha^{QGD}$ was varied in the range from 0.05 to 0.3. Two conditions were imposed on the model time step $\Delta t$ (where CFL is a Courant Friedrichs Lewy number):

1. $\Delta t \leq \tau$;
2. $CFL = (|U| + c) \frac{\Delta t}{\Delta x} \leq 0.3$.

Time-averaged values of the local Mach number were probed along the jet axis and compared to experimental data [5].

Figure 1. Sketch of the computational domain for low-Re jet flow simulation.

Figure 2. Grid lines at the clip of computational domain in XoY plane.

3. Results
Computations were carried out for a number of meshes and 3 values of the QGD tuning coefficient $\alpha^{QGD}$. Preliminary studies showed that Navier-Stokes based solver [6] allows reproducing unsteady behavior qualitatively and quantitatively only for meshes with 32 cells per diameter (CPD) or finer. Computations with QGD solver showed similar behavior, when Kelvin-Helmholtz instabilities start to emerge on 32 CPD grids. Results are presented only for the finest mesh (Figure 3). This mesh counts 20 million cells, simulation time was approximately 18 hours on 504 CPUs.

Qualitatively the results of simulations are in good agreement with the previous studies and modern description of the turbulent jet flow. The evolution of the jet from initial conditions of media at rest until the quasi-steady state is reached involves the next stages:

1. Propagation of momentum in the axial direction with symmetric development of vortexes in the shear layer between the jet and surrounding air (figure 4a).
2. Disruption of large vortexes at the tail of the jet into the smaller ones (figure 4b).
3. Loss of symmetry in the core of the jet (figure 4c).
4. Formation of asymmetric KH instabilities and associated hydrodynamic structures at the end of the laminar shear layer (figure 4d).

When the stage 4 of the developed flow is reached, the averaging of the hydrodynamic fields over time is started to obtain mean values (figure 4e) for comparing with the experiment. The averaging window was set equal to 3 times of passing the domain in the axial direction with the inlet speed of the jet.

Next values of the QGD tuning coefficient $\alpha^{QGD}$ were used for simulation: 0.3, 0.15 and 0.05. It is known from both theoretical and empirical studies that the value of $\alpha^{QGD}$ coefficient has a non-linear and non-monotonic influence on the solution [7,8]: maximum value recommended for stable
simulations is estimated as 0.5 and optimal value, giving a balance between stability, accuracy and performance is about 0.3.

Simulation with $\alpha^{QGD}=0.5$ produces the jet flow with almost no instabilities, resulting in the non-decaying jet. Decreasing $\alpha^{QGD}$ from 0.5 to 0.3 produced solution with jets instabilities starting at approximately $x/D=9$. Further elaboration of solution could be made by increasing mesh density or by decreasing $\alpha^{QGD}$ coefficient. With alpha=0.15, instability emerging starts at $x/D=6$, which is very close to experimental observations. Next diminution of $\alpha^{QGD}$ to the value 0.05 did not improve solution significantly, however, it produced numerical oscillations, creating a non-physical reduction in time-average velocity magnitude at the jet centerline. For reference, the QGD solution was compared to Navier-Stokes (NS) solver [6] with the Minmod Total Variation Diminishing (TVD) flux limiter and Smagorinsky LES (Large Eddy Simulation) model, figure 3. Curves for $\alpha^{QGD}=0.3$ and NS LES are very close to each other, showing results similar to [9].

Simulations with different values of $\alpha^{QGD}$ show possibility to study QGD numerical solution by tuning parameter without consuming additional computational resources due to mesh refinement. The solution at the next mesh density level can be predicted by simulating with smaller value of $\alpha^{QGD}$. And if results change with $\alpha^{QGD}$ significantly, then the next grid level could be used for simulation.

![Figure 3. Time-averaged jet centreline Mach number distribution:](image)

1 – QGDFoam with $\alpha^{QGD}=0.3$, 2 – QGDFoam with $\alpha^{QGD}=0.15$, 3 – QGDFoam with $\alpha^{QGD}=0.05$, 4 – Navier-Stokes solver [6] with LES, 5 – experimentally measured values.
Figure 4. Jet velocity distribution in the plane $X_0Y$ at different time steps computed with the QGD algorithm and $\alpha^{QGD}=0.3$; a) time 5 ms, b) time 10 ms, c) time −15 ms, d) −30 ms, e) time average over period 5 ms.

Conclusion
Numerical simulation of turbulent unsteady jet flow at $Re=3600$ and Mach number 0.9 was carried out. The numerical algorithm is based on explicit approximation of regularized gas dynamics equations. Results of numerical simulation are compared to experimentally measured values. Good
qualitative and quantitative agreement is observed for the QGD tuning coefficient $\alpha_{QGD}$ value 0.15. These results are very close to those obtained in [10], where laminar-turbulent transition using direct numerical simulation of compressible gas was studied. Corresponding mesh resolution is 32 cells per diameter of the nozzle. Qualitative analysis of QGD tuning coefficient $\alpha_{QGD}$ influence was performed. Results of comparative study allow formulation of a guideline for selecting QGD parameters when simulating subsonic and transonic jet flows at low and moderate Re number values (less than 4000).

Acknowledgements
This work was supported by the RSF under the grant No. 19-11-00169. The results of the work were obtained using computational resources of MCC NRC «Kurchatov Institute», http://computing.nrcki.ru/.

References
[1] Elizarova T and Chetverushkin B 1984 Dokl. Akad. Nauk SSSR 279 (1) 80–3
[2] Elizarova T and Shirokov I 2017 Regulyarizovannye uravneniya i primery ikh ispolzovaniya pri modelirovanii gasodinamicheskikh techeniy (M.: MAKS Press) p 136 (in Russian)
[3] https://github.com/unicfdlab/QGDsolver
[4] Kraposhin M, Smirnova E, Elizarova T and Istomina M 2018 Computer&Fluids 166 163–75
[5] Stromberg J, McLaughlin D and Troutt T 1980 J. Sound Vib. 72 159–76
[6] Kraposhin M, Banholzer M, Pfitzner M and Marchevsky I 2018 Int. J. for Num. Meth. In Fluids 88 (2) 79–99
[7] Elizarova T 2009 Quasi-gas dynamic equations (Springer)
[8] Zlotnik A and Lomonosov T 2019 Mathematical Modelling and Analysis 24 (2) 179–94
[9] Kraposhin M, Ryazanov D, Smirnova E, Elizarova T and Istomina M 2018 Proceedings of Ivannikov ISP RAS Open Conference 117–23
[10] Shirokov I and Elizarova T 2014 Journal of Turbulence 15 (10) 707–30