Scientific Visualization, 2025, volume 17, number 3, pages 124 - 132, DOI: 10.26583/sv.17.3.11

Comparative Analysis of the Accuracy of OpenFOAM Solvers in Simulating High-Velocity Flow Around a Double Wedge

A.E. Bondarev[1,A], A.E. Kuvshinnikov[2,A], S.V. Andreev[3,A], I.G. Ryzhova[4,A]

Keldysh Institute of Applied Mathematics RAS, Moscow, Russia

[1] ORCID: 0000-0003-3681-5212, bond@keldysh.ru
[2] ORCID: 0000-0003-1667-6307, kuvsh90@yandex.ru
[3] ORCID: 0000-0001-8029-1124, esa@keldysh.ru
[4] ORCID: 0000-0003-1613-3038, ryzhova@gin.keldysh.ru

Abstract

This article is devoted to a comparative analysis of the accuracy of various solvers of the OpenFOAM software package in numerical modeling of supersonic inviscid gas flow past a double wedge. A two-dimensional steady-state problem of flow formation with the intersection of oblique shock waves is considered. A comparison of standard solvers is made rhoCentralFoam and sonicFoam, as well as third-party developments pisoCentralFoam and QGDFoam . The results obtained in tabular form are visualized as error surfaces. The analysis showed the best accuracy of solvers rhoCentralFoam and QGDFoam for both the density field and the velocity modulus field. The results obtained can be used both in fundamental research and in engineering calculations that require high reliability of complex flow modeling.

Keywords: OpenFOAM, generalized computational experiment, double wedge, oblique jump.

Introduction

With the development of computing technologies and the increase in available computing resources, problems previously solved exclusively by analytical or experimental methods are increasingly moving into the field of numerical modeling. This allows us to significantly expand the range of parameters under study, speed up the process of finding optimal solutions and reduce the costs of conducting expensive experiments. However, the accuracy and reliability of numerical calculations largely depend on the approximation algorithms used, the choice of the computational grid and the correctness of the implementation of boundary conditions.

Solving gas dynamics problems involving the intersection of oblique shock waves is of key importance for the development of computational aerodynamics and modeling of complex flows. Oblique shock waves are widely encountered in supersonic flows, in various devices where sharp pressure and velocity changes are realized. Correct numerical reproduction of these phenomena requires the use of efficient and accurate methods for solving the Euler and Navier–Stokes equations. One of the popular platforms for numerical modeling of such problems is the OpenFOAM software package [1], which provides a rich set of solvers and tools for solving gas dynamics equations.

However, despite the wide choice of implemented in OpenFOAM solvers, the question of comparing their accuracy in problems related to the intersection of oblique shocks remains open. The complexity of modeling is aggravated by the presence of sharp gradients of parameters arising at the shock fronts, as well as their interaction, which requires high-resolution numerical methods. It is known that different solvers can cope with these difficulties in different ways, which directly affects the reliability of the results obtained and the possibility of their use in engineering practice. OpenFOAM , as one of the most flexible and actively developed open source platforms, provides researchers with ample opportunities for customizing numerical schemes and modifying algorithms for specific problems. However, the abundance of available solvers and settings often makes it difficult to choose the optimal approach, especially for problems with high sensitivity to numerical errors, such as the intersection of oblique shocks. In this regard, a systematic comparative analysis of various solvers becomes relevant OpenFOAM in order to identify their advantages and limitations when solving problems of the corresponding class. Such an analysis will not only increase the reliability of calculations, but also optimize computing resources, which is especially important when conducting large-scale or multiparameter studies.

Previous works

This work is a continuation of a number of works by the authors. In the works of the previous period, various classes of gas-dynamic problems with reference solutions were considered. All problems were considered for supersonic flows.

Studies devoted to a comparative analysis of the accuracy of solvers for flow around a circular cone at an angle of attack are presented in [2].

Also considered were problems of the formation of an oblique shock wave when a supersonic flow falls at a certain angle onto a plate [3], problems of the formation of a rarefaction wave formed when flowing around a plate at a certain angle [4], problems of flowing around a cone with a spherical blunting [5].

All results were obtained by constructing a generalized computational experiment. The key features and components of the generalized computational experiment are discussed in detail in [6-8].

Setting the task and organizing calculations

In this paper, for comparison of solvers a two-dimensional inviscid problem of the formation of a steady flow obtained by flowing past a double wedge with angles α and β by a supersonic gas flow with a Mach number at a zero angle of attack is used. The variable parameters here are the Mach number and the wedge angles β. The ranges of change of the variable parameters and the change step were selected as follows: the Mach number from 1.8 to 2.0 with a step of 0.1, the double wedge angles α /β = 5°/10°, 5°/15°, 10°/15°. The general flow diagram is shown in Fig. 1. For the calculation, a system of Euler equations closed by the equation of state of an ideal gas was taken. We also note that the problem has an analytical solution [8, 9].

The study considered four solvers: two standard ones – rhoCentralFoam and sonicFoam , and two created by third-party authors – pisoCentralFoam and QGDFoam . The latter were developed by specialists from the Institute of System Programming of the Russian Academy of Sciences and the Keldysh Institute of Applied Mathematics of the Russian Academy of Sciences. All the compared solvers implement numerical methods of different nature [10-13], i.e. the work does not compare different software implementations of the same method.

The scheme of the calculation area for a wedge with angles of 10°, 20° is shown in Fig. 2. It is worth noting that in the indicated image, for clarity, the grid is larger than in real calculations.

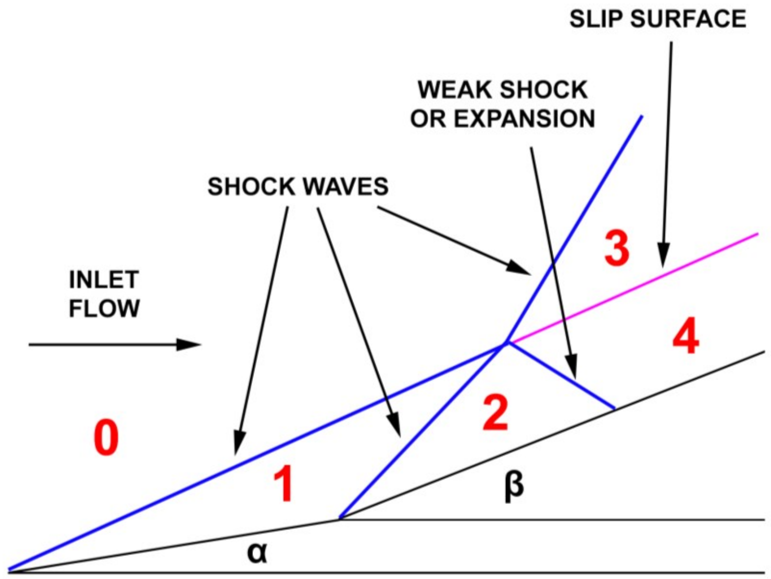

Fig. 1. Flow diagram

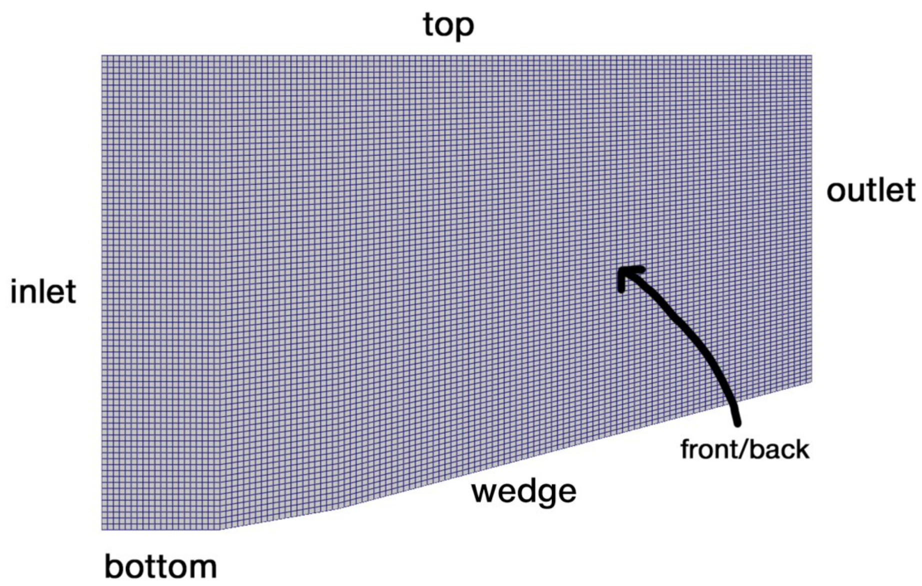

Fig. 2. Scheme of the computational domain

inlet boundary, the parameters of the unperturbed incident flow are set: pressure P = 101325 Pa, temperature T = 300 K, the x-component of the velocity Ux varies in the previously specified range, the y-component of the velocity Uy is taken to be 0 m/s. At the outlet boundary, as well as at the upper (top) and lower (bottom) boundaries, the zero gradient conditions are set for all quantities. For the wedge boundary (wedge), the zero gradient condition is set for pressure and temperature, and the slip condition is used for the velocity, which corresponds to the no-flow condition in the Euler equations. At the front (front) and back (back) boundaries, a special boundary condition, empty, is applied, which is used in cases where calculations in a given direction are not performed. The number of grid cells is 90000. The initial conditions correspond to the boundary conditions on the inlet face, that is, the incident flow parameters are used as the initial conditions. In the solver QGD-Foam parameter α_{QGD}, which affects the dissipative properties, was set to 0.1 (by default it is 0.5).

Experimental results

The flow patterns are shown in Fig. 3 and Fig. 4 as pressure and density distributions in the computational domain. The presented distributions were obtained using the solver rhoCentralFoam. The solution does not collapse for any of the solvers, which indicates high stabilizing properties of all solvers participating in the study.

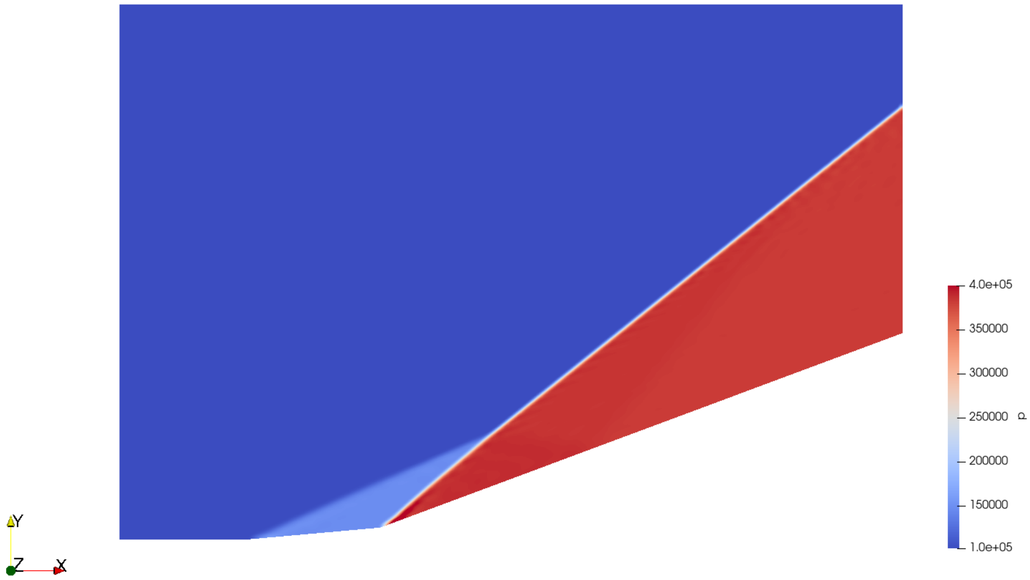

Fig. 3. Steady-state flow pressure field for the solver rhoCentralFoam, α = 5°, β = 20°

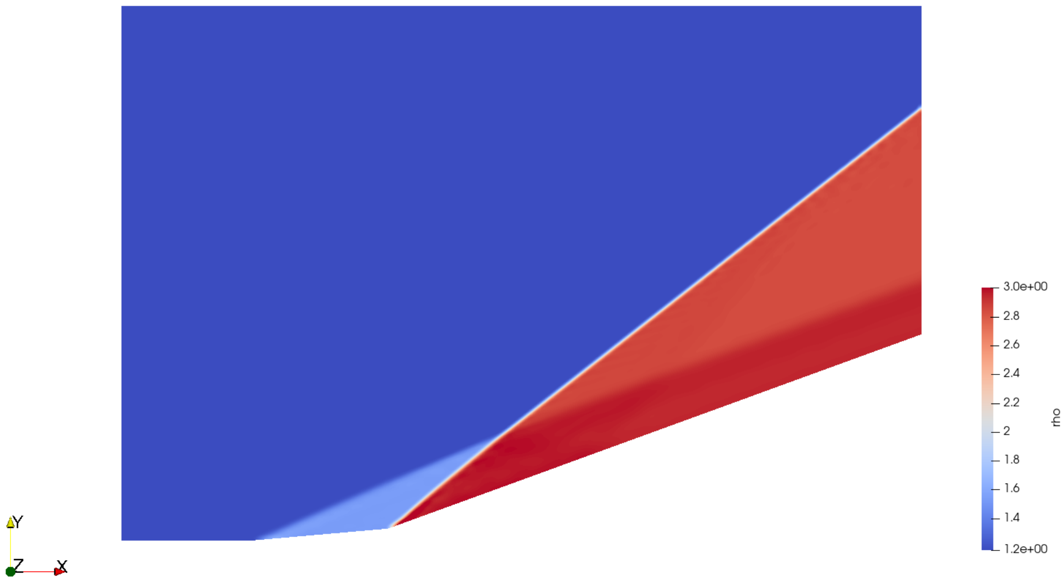

Fig. 4. Density field of steady flow for solver rhoCentralFoam , α = 5°, β = 20°

Let us construct estimates of the deviation from the exact solution for the entire computational domain in the $L_{2\ \mathrm{norm}}$. To do this, we define the relative error *Err* for the $L_{2\ \mathrm{norm}}$ as follows:

$$Err = \sqrt{\sum_m |y_m - y_m^{exact}|^2 S_m} \Big/ \sqrt{\sum_m |y_m^{exact}|^2 S_m} \qquad (1)$$

where y_m are the studied quantities (the modulus of the velocity vector and the density) obtained by numerical calculation in the cell m, V_m is the cell volume. The values of y_m^{exact} are obtained by interpolation of the analytical solution of the problem. Solvers were used in the comparative accuracy analysis sonicFoam, QGDFoam, rhoCentralFoam and pisoCentralFoam. The values of deviation from the exact solution for the entire computational domain are given in Tables 1–3. The smallest values in each row are shown in bold. The table uses abbreviated notations for solvers: rCF (rhoCentralFoam), pCF (pisoCentralFoam), sF (sonicFoam), QGDF (QGDFoam). The smallest values in each row are shown in bold.

Table 1. Errors for M = 1.8

Size	**Angles α / β**	***rCF***	***pCF***	***sF***	***QGDF***
Speed module	5/10	**0.006476**	0.006831	0.009934	0.006510
	5/15	**0.009169**	0.009417	0.013579	0.009830
	10/15	0.007309	0.007734	0.012018	**0.006619**
Density	5/10	**0.009679**	0.009915	0.015727	0.010157
	5/15	0.010035	0.010582	0.014361	**0.009777**
	10/15	0.008180	0.008527	0.015914	**0.007864**

Table 2. Errors for M = 1.9

Size	**Angles α / β**	***rCF***	***pCF***	***sF***	***QGDF***
Speed module	5/10	**0.006283**	0.006579	0.009431	0.006478
	5/15	0.009247	0.009625	0.012744	**0.009120**
	10/15	0.007709	0.008042	0.009118	**0.006568**
Density	5/10	**0.009600**	0.009903	0.015304	0.010341
	5/15	0.011428	0.011852	0.015710	**0.010675**
	10/15	0.009540	0.009837	0.013656	**0.009179**

Table 3. Errors for M = 2.0

Size	**Angles α / β**	***rCF***	***pCF***	***sF***	***QGDF***
Speed module	5/10	**0.006336**	0.006598	0.009035	0.006557
	5/15	0.008936	0.009217	0.012018	**0.008562**
	10/15	0.007502	0.007924	0.008734	**0.006628**
Density	5/10	**0.010110**	0.010587	0.015884	0.011495
	5/15	0.011604	0.012043	0.015914	**0.010864**
	10/15	**0.010062**	0.010514	0.014583	0.010274

To analyze the tables, we visualize the data as error surfaces. The result is shown in Fig. 5 and Fig. 6.

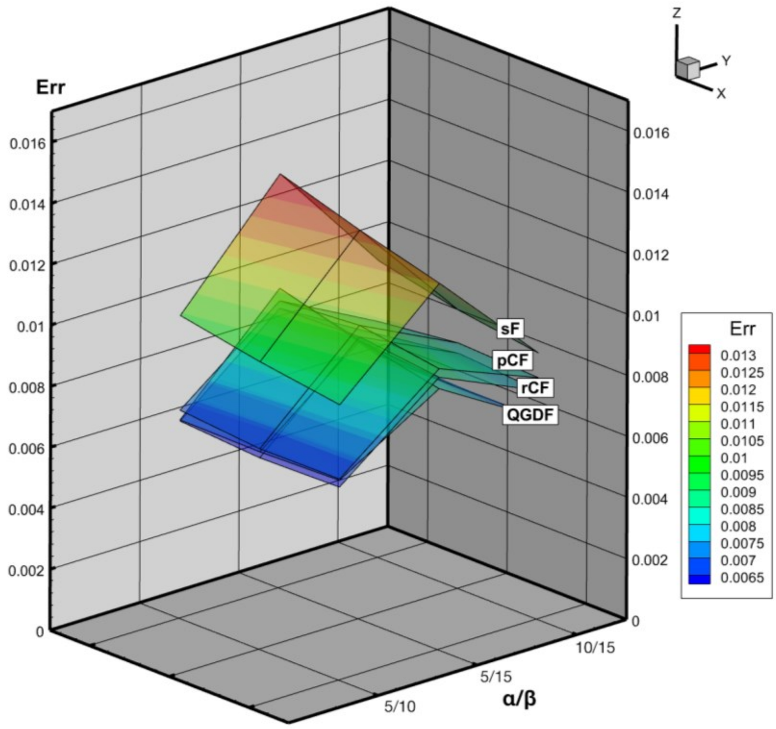

Fig. 5. Error surfaces of the solvers under consideration for the velocity module

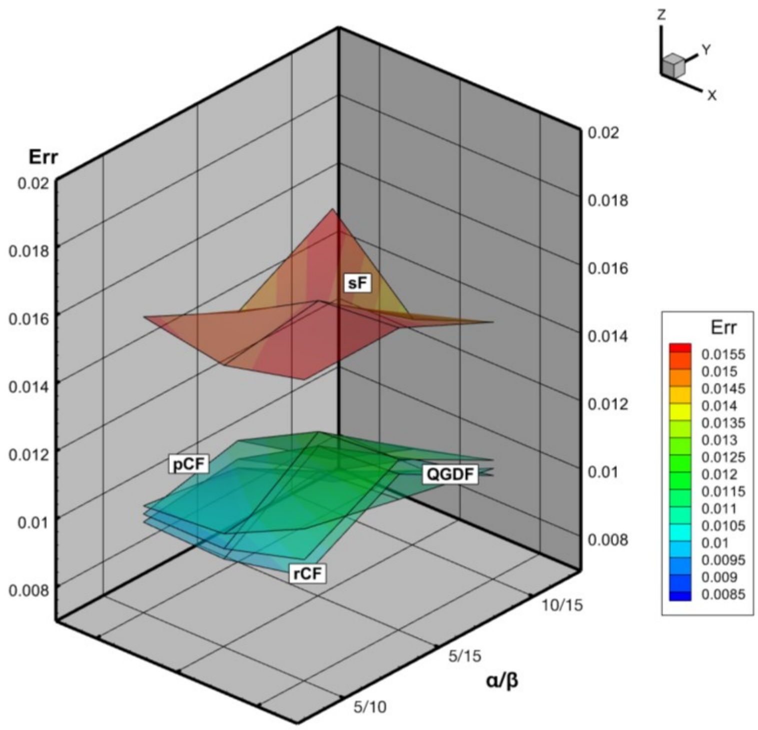

Fig. 6. Error surfaces of the considered solvers for density

Analysis of the presented tables showed that the solvers rhoCentralFoam and QGDFoam give similar and often better results in accuracy, p isoCentralFoam is slightly worse than rhoCentralFoam. The solver has the largest errors sonicFoam, especially for density. For example, for angles of 5° and 10°, the error for sonicFoam is 1.5 times greater than the error of the most accurate rhoCentralFoam in this case. With an increase in the incident flow velocity in this range, the error does not increase. It is also worth noting that the errors for the velocity module for angles of 5° and 15° are 1.3 times greater than the error for angles of 5° and 10°. The errors of angles of 10° and 15° are only slightly greater than the errors of 5° and 10°. This is true for all solvers. The errors for density are practically independent of the angles and the incident flow velocity. However, it is worth noting that for the solver QGDFoam the dissipative parameter plays a major role. It was selected for the range of input parameters used in the current problem, but when going beyond the interval the solver QGDFoam accuracy may deteriorate. Authors recommend solvers rhoCentralFoam and pisoCentralFoam as the most universal, or solver QGDFoam, if there is time to select the optimal value of the dissipative parameter.

Conclusion

The conducted study allowed us to comprehensively evaluate the accuracy of various solvers OpenFOAM in solving the actual problem of crossing oblique shocks arising when a high-speed inviscid gas flow passes a double wedge. A comparative analysis of four solvers showed that rhoCentralFoam demonstrates the lowest values of numerical errors in the $L_{2\ norm}$, which makes it a preferable choice for problems requiring high accuracy in reproducing complex gas-dynamic structures. It is noted that an increase in the incident flow velocity and wedge angles does not lead to an increase in errors for all solvers for density. For pressure, a change in wedge angles already affects the error. The presented results allow us to recommend the use of rhoCentralFoam and pisoCentralFoam for engineering calculations where the reliability of the reproduced shocks is critical. The data obtained can be useful both for specialists engaged in fundamental research in the field of computational gas dynamics and for engineers implementing numerical methods in the practice of designing aerodynamic devices.

Acknowledgements

The calculations were carried out using the K100 hybrid supercomputer installed at the Supercomputer Center for Collective Use of the Keldysh Institute of Applied Mathematics of the Russian Academy of Sciences.

References

1. OpenFOAM Foundation [Electronic resource]. Access mode: https://openfoam.org (date of access 03/10/2025).
2. Bondarev AE, Kuvshinnikov AE Analysis of the accuracy of OpenFOAM solvers for the problem of supersonic flow around a cone // ICCS 2018, Lecture Notes in Computer Science (LNCS) 10862, 2018. P. 221–230.
3. Alekseev AK, Bondarev AE, Kuvshinnikov AE Comparative analysis of the accuracy of OpenFOAM solvers for the oblique shock wave problem // Matematica Montisnigri. 2019. V. XLV, pp. 95–105.
4. Bondarev AE, Kuvshinnikov AE Analysis and Visualization of the Computational Experiments Results on the Comparative Assessment of OpenFOAM Solvers Accuracy for a Rarefaction Wave Problem // Scientific Visualization. 2021. V. 13. No. 3. P. 34–46.
5. Bondarev AE, Kuvshinnikov AE Integrating Scientific Visualization in the Assessment of Openfoam Solvers for the Flow Around a Spherically Blunted Cone // Scientific Visualization. 2024. V. 16. No. 4. P. 25–36.
6. Bondarev AE On the Construction of the Generalized Numerical Experiment in Fluid Dynamics // Mathematica Montisnigri. 2018. T. XLII. P. 52–64.
7. Bondarev AE, Galaktionov VA Generalized Computational Experiment and Visual Analysis of Multidimensional Data // Scientific Visualization. 2019. V. 11. No. 4. P. 102–114.
8. Alekseev AK, Bondarev AE, Galaktionov VA, Kuvshinnikov AE On the construction of a generalized computational experiment in verification problems // Matematica Montisnigri. 2020. T. XLVIII. pp. 19–31.

9. Arutyunyan G.M. Reflected shock waves / G.M. Arutyunyan, L.V. Karchevsky. Moscow: Mashgiz, 1973. 376 p.
10. Anderson JD Modern Compressible Flow: With Historical Perspective. McGraw-Hill Education, 2021. 778 p.
11. Issa R. Solution of the implicit discretized fluid flow equations by operator splitting // J. Comput. Phys. 1986. Vol. 62. No. 1. P. 40–65.
12. Kurganov A., Noelle S., Petrova G. Semidiscrete central-upwind schemes for hyperbolic conservation laws and Hamilton–Jacobi equations // SIAM J Sci Comput. 2001. Vol. 23 P. 707–740.
13. Kraposhin M., Bovtrikova A., Strijhak S. Adaptation of Kurganov-Tadmor numerical scheme for applying in combination with the PISO method in numerical simulation of flows in a wide range of Mach numbers // Procedia Computer Science. 2015. Vol. 66. P. 43–52.
14. Istomina M.A. On the implementation of a one-dimensional quasi-gasdynamic algorithm in the open software package OpenFOAM // Preprints of the Keldysh Institute of Applied Mathematics. 2018. No. 001.