Numerical simulation of supersonic flow around a cone in a three-dimensional formulation in the software package OpenFOAM

A E Kuvshinnikov

Keldysh Institute of Applied Mathematics RAS, Miusskaya sq., 4, Moscow, 125047, Russia

E-mail: kuvsh90@yandex.ru

Abstract. Three solvers of the OpenFOAM package are chosen to numerically solve the problem of supersonic flow around a cone. The cone half-angle and the angle of attack varied. The obtained results are compared with the well-known numerical solution of this problem. It has been established that the rhoCentralFoam solver provides minimal error for the pressure field and pisoCentralFoam solver for velocity field.

1. Introduction

This work is part of a series of articles [1, 2], aimed at creating a new method for calculating the flow around elongated bodies of rotation (EBR). Such developments are important due to the fact that currently there are many software packages that solve such problems. Thus, the researcher has a question, which package is better suited for such calculations. The authors have proposed to recreate at the level of modern high-performance computing tools computing technology developed earlier in the Keldysh Institute of Applied Mathematics A.E. Bondarev and V.A. Cherkashin under the direction of A.V. Zabrodin. The essence of this technology is that the aerodynamic drag coefficient $C_x$ is considered as a sum of three other coefficients: $C_p$ – coefficient for inviscid flow, $C_f$ – coefficient for viscous friction and $C_d$ – coefficient for near wake pressure. Such approach was widely used in the problems of mass industrial analysis of the aerodynamic properties of an EBR and proved to be very effective [3].

To calculate the aerodynamic characteristics of the non-viscous flow around EBR, the OpenFOAM [4] software package was used. It is a freely distributed software product created for solving problems of hydro and gas dynamics. It is used in many areas of science and technology, both in commercial and in academic organizations. OpenFOAM contains a number of solvers with different computational properties.

It is necessary to clarify that some solvers were previously created by developers of the OpenFOAM package [5-7], but users can create their own solvers [8].

Tabular solutions from [9] were used as in the previous works of the authors. These tables were obtained using finite-difference methods in a wide range of Mach numbers and cone half-angles with variation of the angle of attack.

In past works [1, 2], the case of a flow around EBR at a zero angle of attack was investigated. Now our main goal is to calculate the problem when the angle of attack varies.
It should be noted that similar comparisons of solvers were carried out in [10-12]. However, these comparisons were made on other examples and do not give clear recommendations on the choice of a solver for the class of problems in question.

2. **Formulation of the problem**

The statement of the problem is presented in full accordance with the work [9], where the results of inviscid flow around cones with different cone half-angles and angles of attack at different Mach numbers are considered.

We study the flow of an EBR placed in uniform supersonic flow of an ideal gas at an angle of attack $\alpha = 0^\circ$, $5^\circ$, $10^\circ$ with a Mach number $M = 3$. The investigated body is a cone with a half-angle $\beta = 20^\circ$. The incoming flow conditions at the input are denoted by the index "$\infty$", and at the output by the index $\xi$, since the solution is self-similar and depends on a dimensionless variable. The flow pattern is shown in figure 1.

For calculation, the Euler system of equations is used. The system is supplemented by the ideal gas equation of state.

![Flow scheme](image)

**Figure 1.** Flow scheme.

3. **OpenFOAM solvers**

For comparison, 3 solvers were selected from the OpenFOAM software package:

- **rhoCentralFoam** – is based on a central-upwind scheme, which is a combination of central differential and upwind schemes [5, 6]. The main idea of central-upwind schemes is a special choice of a control volume containing two types of domains: the first type around the boundary points, the second type around the center point. The advantage of these schemes is that, using the appropriate technique of reducing the numerical viscosity, it is possible to achieve good solvability for discontinuous solutions (shock waves in gas dynamics) and for solutions where viscous phenomena play a major role.

- **sonicFoam** is based on the PISO (Pressure Implicit with Splitting of Operator) algorithm [7]. The basic idea of the PISO method is that two differential equations are used to calculate the pressure to correct the pressure field obtained from discrete analogs of the moment and continuity equations. This approach is due to the fact that the velocities corrected by the first correction may not satisfy the continuity equation; therefore, a second corrector is introduced that allows one to calculate velocities and pressures that satisfy the linearized equations of momentum and continuity.

- **pisoCentralFoam** is a combination of a central-upwind scheme with the PISO algorithm [8]. Calculations for all solvers were carried out using the OpenFOAM version 2.3.0.
4. Organization of calculations

4.1. Grid, initial and boundary conditions
Figure 2 shows the calculated area. At the boundaries indicated in the figure 2 as “top”, “bottom”, “front”, “back” and “outlet” a zero gradient condition is set for gas-dynamic functions. On the left border, denoted by "inlet", the parameters of the incoming flow are set: pressure $P = 101325$ Pa, temperature $T = 300$ K, speed $U = 1041.75$ m/s. On the boundary of the cone “cone”, the pressure and temperature conditions are set to zero gradient, and the speed conditions are set to “slip”, which corresponds to the non-flow condition for the Euler equations. The number of grid cells is 336000.

![Computational domain](image.png)

**Figure 2.** Computational domain.

4.2. Solvers parameters
In the OpenFOAM package, there are two options for changing the approximating of differential operators: directly in the solver code or using the fvSchemes and fvSolution configuration files. In order for the comparison to be correct, we used the same parameters where it was possible.

In the file fvSchemes: ddtSchemes — Euler, gradSchemes — Gauss linear, divSchemes — Gauss linear, laplacianSchemes — Gauss linear corrected, interpolationSchemes — vanLeer. In the file fvSolution: solver — smoothSolver, smoother — symGaussSeidel, tolerance — $10^{-9}$, nCorrectors — 2, nNonOrthogonalCorrectors — 1.

5. Flow calculation
Figure 3 shows the steady-state flow field for pressure using the interpolation of the tabular solution from [9].

Tables 1–3 show the results of calculations in the form of an analog of the $L_2$ norm:
\[
\sqrt{\sum_{m} |y_m - y_m^{\text{exact}}|^2 V_m} / \sqrt{\sum_{m} |y_m^{\text{exact}}|^2 V_m},
\]

where \(y_m\) are the velocity component \(U_x\), pressure \(p\) and density \(\rho\) in the cell, \(V_m\) is the cell volume of the cone half-angles \(\beta = 10^\circ\)–\(20^\circ\). Mach number \(M = 3\), angle of attack \(\alpha = 0^\circ\), \(5^\circ\), \(10^\circ\). The minimum values are highlighted in bold. Here, \(y_m^{\text{exact}}\) values are obtained by interpolating tabular values from [9] onto grid cells. It should be noted that the authors of the tables [9] indicate the admissibility of interpolation for all parameters and table values.

Further we will use abbreviations for solvers: rCF (rhoCentralFoam), pCF (pisoCentralFoam), sF (sonicFoam).

**Figure 3.** Pressure field.

**Table 1.** Deviation from the exact solution, \(\beta = 10^\circ\).

| Angle of attack | Pressure \(p\) | Velocity component \(U_x\) |
|-----------------|---------------|---------------------------|
|                 | rCF           | pCF           | sF       | rCF               | pCF               | sF       |
| 0               | 0.026174      | 0.032426      | 0.046394 | 0.005242          | 0.004086          | 0.004455 |
| 5               | 0.030636      | 0.037702      | 0.067424 | 0.005573          | 0.004241          | 0.005796 |

**Table 2.** Deviation from the exact solution, \(\beta = 15^\circ\).

| Angle of attack | Pressure \(p\) | Velocity component \(U_x\) |
|-----------------|---------------|---------------------------|
|                 | rCF           | pCF           | sF       | rCF               | pCF               | sF       |
| 0               | 0.046490      | 0.058198      | 0.091404 | 0.010903          | 0.009031          | 0.012235 |
| 5               | 0.050298      | 0.060187      | 0.116237 | 0.011135          | 0.008931          | 0.014591 |
| 10              | 0.060519      | 0.069622      | 0.145829 | 0.011158          | 0.009040          | 0.016186 |
Table 3. Deviation from the exact solution, $\beta = 20^\circ$.

| Angle of attack | Pressure $p$ | Velocity component $U_x$ |
|-----------------|-------------|--------------------------|
|                 | rCF         | pCF                      | sF |
| 0               | 0.060614    | 0.069128                 | 0.123338 |
| 5               | 0.065373    | 0.075543                 | 0.149785 |
| 10              | 0.072673    | 0.081022                 | 0.172609 |

It is clearly seen that the error increases both with increasing cone half-angle and with increasing angle of attack.

6. Conclusion

Obtained results show that the rhoCentralFoam solver has a minimum field error rate for the pressure field and the pisoCentralFoam for the velocity field. The only drawback of rhoCentralFoam is the appearance of oscillations near the surface at the head of the cone. And solver pisoCentralFoam does not have such oscillations. The results are qualitatively similar to the axisymmetric case.

Solver sonicFoam has oscillations at the shock wave front. This adversely affects the error rate. Thus, it can be argued that the rhoCentralFoam and pisoCentralFoam solvers provide better accuracy for the class of problems and can be used in the construction of the computational flow calculation technology for the elongated bodies of rotation.

Acknowledgments

This work was supported by RFBR grant № 18-31-00320.

References

[1] Bondarev A E and Kuvshinnikov A E 2018 Analysis of the accuracy of OpenFOAM solvers for the problem of supersonic flow around a cone Computational Science – ICCS 2018 221–30
[2] Bondarev A E and Kuvshinnikov A E 2017 Keldysh Institute Preprints 12 1–16
[3] Krasil'shchikov A P and Gur'yashkin L P 2007 Eksperimental'nye Issledovaniya Tel Vrashcheniya v Giperzvukovykh Potokakh (Moscow: FIZMATLIT) [In Russian]
[4] OpenFOAM Available at: http://www.openfoam.org (Accessed 25 April 2019)
[5] Kurganov A and Tadmor E 2000 J. Comp. Phys. 160 241–82
[6] Greenshields C, Weller H, Gasparini L and Reese J 2009 Int. J. Num. Meth. Fluids 63 1–21
[7] Issa R 1986 J. Comp. Phys. 62 40–65
[8] Kraposhin M, Bovtrikova A and Strijhak S 2015 Proc. Comp. Science 66 43–52
[9] Babenko K I, Voskresenskii G P, Lyubimov A N and Rusanov V V 1984 Prostranstvennoe Obtekanie Gladkikh Tel Ideal'nym Gazom (Moscow: Nauka) [In Russian]
[10] Karvatskii A Ya, Pulineni I V, Lazarev T V and Pedchenko A Yu 2015 Kosm. Nauka Tehnol. 21 47–52 [In Russian]
[11] Gutierrez L F, Tamagno J P and Elaskar S A 2012 Mecanica Computacional XXXI 2939–59
[12] Lorenzon D and Elaskar S A 2015 Rev. Fac. Cien. Exactas Fis. Nat. 2 65–76 [In Spanish]