UDC 517.958:533.7; 532.517.4
IRSTI 27.35.17; 30.17.27

https://doi.org/10.55452/1998-6688-2025-22-1-211-222

[1]*Manapova A.,
Master of Applied Mathematics and Computer Science,
ORCID ID: 0000-0003-1548-7061,
*e-mail: manapova.a.k.math@gmail.com
[2]Beketayeva A.,
Doctor of Physical and Mathematical Sciences,
ORCID ID: 0000-0003-4360-3728,
e-mail: azimaras10@gmail.com
[3,4]Makarov V.
Candidate of Technical Sciences,
ORCID ID: 0000-0003-4874-5418,
e-mail: makfone@mail.ru

[1]Civil Aviation Academy, Almaty, Kazakhstan
[2]Institute of Mathematics and Mathematical Modeling CS MSHE RK, Almaty, Kazakhstan
[3]Institute of Control Sciences RAS, Moscow, Russia
[4]National Research Nuclear University «MEPhI», Moscow, Russia

NUMERICAL MODELING OF SPHERE AND CONE STREAMLINE BY SUPERSONIC COMPRESSIBLE FLOW

Abstract

This paper deals with numerical modelling of supersonic flow of cone and sphere bodies using the penalty function method. The main objective of the study is to evaluate the effectiveness of the penalty function method, also known as the immersed boundary method, for solving compressible gas dynamics problems. We apply modified Navier-Stokes equations considering streamlined bodies and use the ENO scheme for the numerical solution. The simulation results demonstrate that the proposed approach successfully describes the physical processes occurring in the supersonic flow of a cone and sphere, including the formation of shock waves, pressure, temperature and density distributions. The obtained data are compared with experimental results, confirming the adequacy and accuracy of the developed numerical model. The presented work contributes to the development of methods for numerical modelling of compressible supersonic flows and demonstrates the promising use of the penalty function method for solving a wide class of gas dynamics problems.

Key words: flow past a cone, flow past a sphere, supersonic flow, turbulent flow, compressed gas.

Introduction

Numerical modelling of compressible flow around moving solids is an important problem in many engineering applications such as rocket engines. The solution of such problems is often related to the study of the interaction between gas flows and solid bodies. For example, the task may be to find the pressure distribution on the surface of a streamlined body, to determine the forces acting on the body or the flow velocity field. Creation of a mathematical model of gas-body interaction requires correct setting of boundary conditions at the interface of two media. The most popular approach for this purpose is the use of boundary meshes. All surfaces are defined by grid nodes, and the required boundary conditions are given by algebraic relations in these nodes. For a complex geometrical body, the construction of a mesh with boundary correction is a very resource-intensive and time-consuming process.

Considering the described difficulties, an alternative approach based on immersed boundary methods [1–2], which ensures fulfilment of boundary conditions in mathematical models without using boundary-controlled meshes, has been rapidly developed in recent decades. The method can perform modelling on simple Cartesian meshes and impose gas immersed boundary conditions. Two important advantages of this idea are its simple implementation and the relatively easy extension from a stationary body to a moving body in general without mesh reconstruction. The immersed boundary method (IBM) was first formulated by Peskin [3], where the interfaces between two media were modelled by adding initial conditions to the basic equations of gas dynamics. Based on this idea, various formulations of the immersed boundary method have now appeared [1], which can be divided into two classes:

The boundary condition is determined by adding external forces or sources that take into account the original gas dynamics equation and the influence of the boundary (Continuous Effect Method, e.g., Brinkman Penalty Method and Feature Based Volume Penalty Method [4] and Discrete Effect Method [5]),

Methods for spatial discretisation of the original equation that vary near an immersed boundary (discrete action methods such as the phantom cell method [6–7] and the cut cell method [8]).

Immersed boundary methods were originally proposed for modelling the flow of incompressible fluid and have recently been applied to modelling the flow of compressible fluid [14–17]. The penalty function method applied is universal because an ordered body is added to the differential equation in the form of an initial function, and the resulting new equation is discretised and solved in the usual way. An extension of the penalty function has been proposed that imposes characteristic-based boundary conditions using its hyperbolicity to establish homogeneous and inhomogeneous Neumann and Robin boundary conditions. This characteristic-based penalty method (CBVP) is flexible and can be applied to parabolic and hyperbolic evolution equations. In this paper, CBVP is considered for the fully compressible Navier-Stokes equation. This method provides strict error control with pre-selected parameters for all boundary conditions [4].

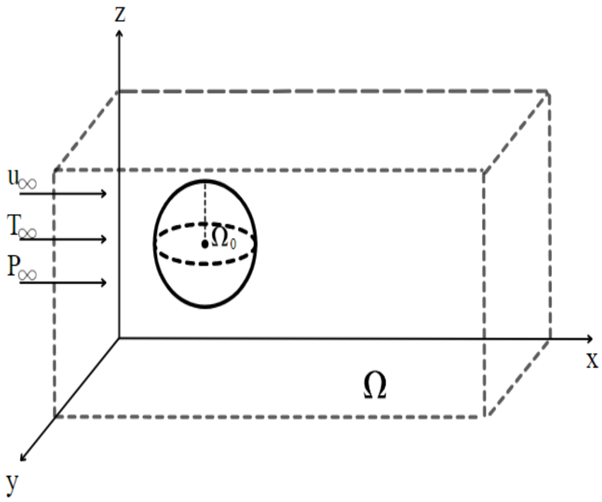
The method of immersed boundaries is considered in many research works [9–13]. When studying the method in this work, the conditions of cone flowing with compressible gas were taken into account. To solve this problem, the method of penalty functions on characteristics was used. The aim of this study is to model the flow of a fully compressible gas past a cone. The study starts with the Navier-Stokes equations with penalty functions for a compressible medium with initial terms, including the continuity equation written by the penalty function.

The results obtained allow us to gain a deeper understanding of the physical processes occurring in the supersonic flow of a cone-sphere-type body and to evaluate the effectiveness of the method of introducing a penalty function in this context. Research work on the supersonic flow problem V.A. This was evaluated by comparing the experimental data with the article by Bashkin [18].

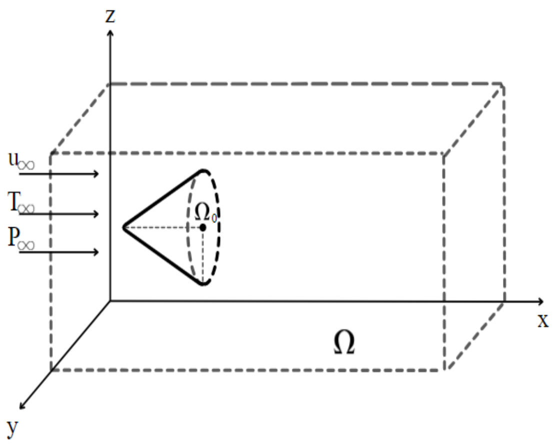
The aim of the work is to solve a numerical method for modelling the supersonic flow of a cone and a sphere. The study aims to evaluate the effectiveness of the penalty function method.

Materials and Methods

The application of the penalty function method to the numerical solution of the flow of a cone and a sphere by a supersonic flow of compressible ideal gas is considered. The numerical calculation of the basic equations will be performed by a through calculation, for which modified equations will be proposed taking into account the body flow (see Fig. 1).



a)



b)

Figure 1 – a) Sphere flow diagram (Problem 1),
b) Cone flow diagram (Problem 2)

The equations of a compressible ideal gas:

$$\frac{\partial \rho}{\partial t} = -\frac{\partial \rho u_i}{\partial x_i} = RHS_\rho, \qquad i = \overline{1,3} \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} = \frac{\partial(\rho u_i u_j)}{\partial x_j} - \frac{\partial \rho}{\partial x_j} + \frac{1}{Re} \cdot \frac{\partial \tau_{ij}}{\partial x_j} = RHS_{u_j}, \tag{2}$$

$$\frac{\partial E_t}{\partial t} = -\frac{\partial (E_t + P)u_j}{\partial x_j} + \frac{1}{Re} \cdot \frac{\partial (u_i \tau_{ij})}{\partial x_j} + \frac{1}{(\gamma - 1)RePr} \cdot \frac{\partial}{\partial x_j}\left(\mu \frac{\partial T}{\partial x_j}\right), \tag{3}$$

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right). \tag{4}$$

This tensor describes the viscous stresses in the fluid.

$$\mu = \frac{1+S_1}{T+S_1} \cdot T^{3/2}, \tag{6}$$

here S_1 is Sutherland constant.

$$E_t = \rho e + \frac{1}{2}\rho \vec{V}^2, \tag{7}$$

$$e = c_V T, \tag{8}$$

$$c_V = \frac{1}{\gamma(\gamma-1)M_\infty^2}, \tag{9}$$

here γ is adiabatic constant.

$$P = (\gamma - 1)\rho e = (\gamma - 1) \cdot \left[E_t - \frac{1}{2}\rho \vec{V}^2\right], \tag{10}$$

$$T = \frac{1}{\rho c_V}\left[E_t - \frac{1}{2}\rho \vec{V}^2\right]. \tag{11}$$

Parameters in equations (1) – (3) are de-measured as in [19].

Boundary conditions for gas dynamics:

1) at the inlet $u_\infty, v_\infty, w_\infty, \mu_\infty, T_\infty, \rho_\infty$;
2) at the outlet non-reflective boundary conditions;
3) on the lateral borders non-reflective boundary conditions;
4) at the upper and lower boundaries there are non-reflection conditions.

The penalty function method is used for this problem, the application of which is described in detail in [20].

Results and Discussion

The process of supersonic compressible flow of a spherical body (problem 1) is considered. Supersonic conditions: flow velocity, Mach number $M_\infty = 1.3$ and Reynolds number $Re = 10^3$. Adiabatic boundary conditions and slip conditions were set on the sphere surface.

Figure 2 shows the Mach number contour. When the flow is turbulent, it can be found that the shock wave is not destroyed by turbulence, and even when the turbulence is intense, the shock wave is still clearly visible at the top of the particle. Therefore, the wave structure around the particle is not significantly different from laminar flow. In the forebody region, the Mach number induced by the shock wave starts to decrease (to about 0.7).

One can observe the formation of a boundary layer in the front part of the sphere, as well as an increase in the flow velocity as the flow bypasses the body. In the region where the flow detaches from the body, the velocity changes due to vortices and turbulence zones. After passing the area, the flow stabilises.

Figure 3 shows the temperature increase due to compression and heating of the gas during collision. As one moves away from the body, a sparse vortex region appears, and hence the temperature starts to fall.

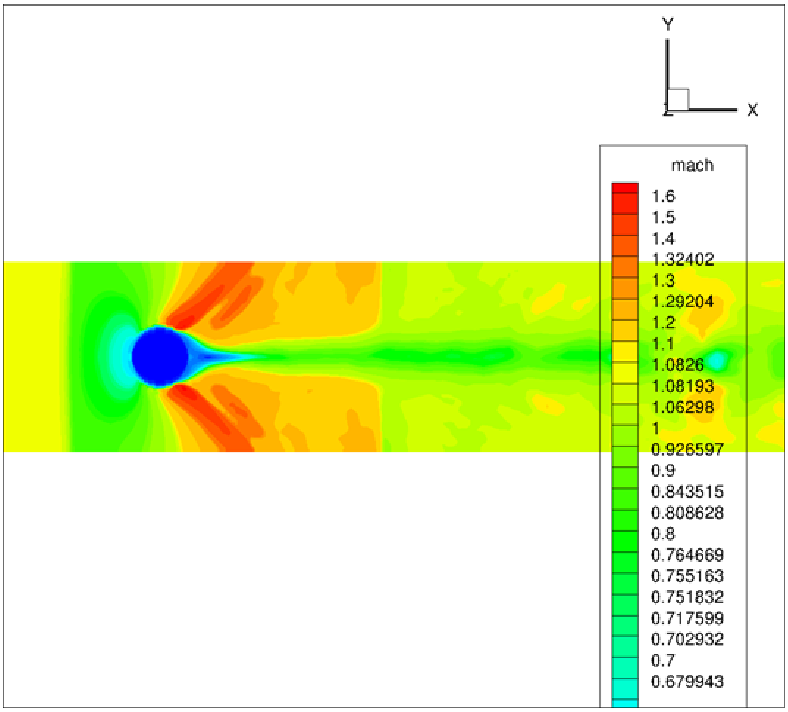


Figure 2 – Variation of the Mach parameter $M_{\infty} = 1.3$ at YX

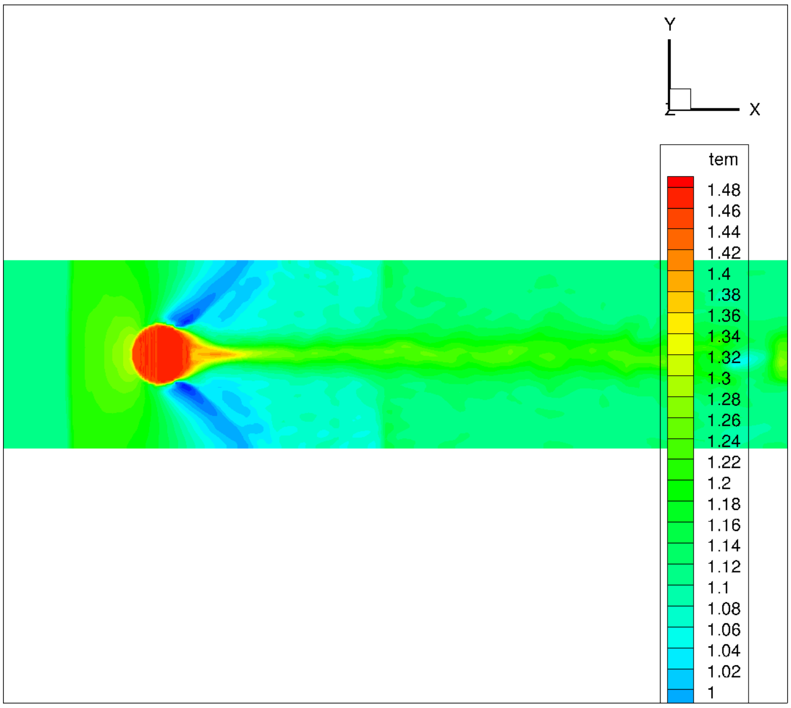


Figure 3 – Temperature T change at ZY

As shown in Figure 4, due to the formation of the shock wave, there is an increase in density in the frontal region of the body. Along the body surface, the density gradually decreases. This is due to the change of flow characteristics along the body, redistribution of density in the boundary layer.

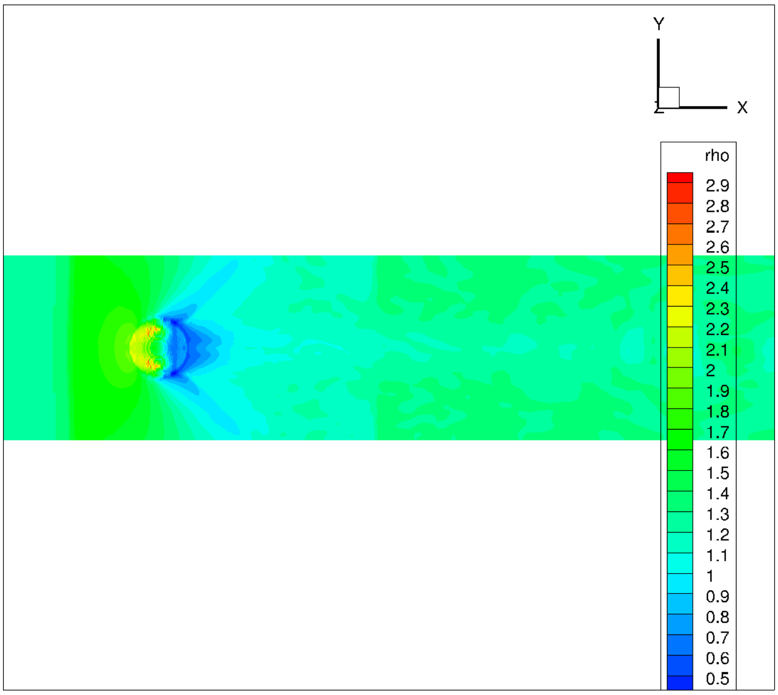


Figure 4 – Density ρ change at YX

Figure 5 shows the pressure increase under the influence of the shock wave. The pressure gradually decreases along the surface of the sphere. And in the rear region of the sphere there will be a sharp pressure drop.

The process of flowing of a cone-shaped body by a supersonic compressible body (Problem 2) is considered. Supersonic conditions: flow velocity, Mach number $M_{\infty} = 1.3$ and Reynolds number $Re = 10^3$. Adiabatic boundary conditions and slip conditions were set on the sphere surface.

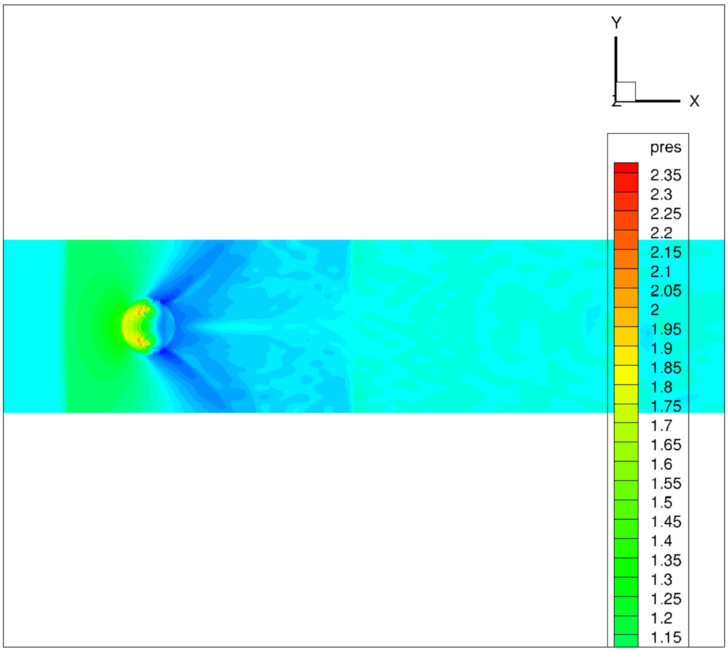


Figure 5 – Pressure P change at YX

In Figure 6, the distribution of Mach number $M_\infty = 1.3$ when moving around a cone-type body is shown. In the front part of the body, due to the impact of the shock wave, one can observe a decrease in the flow velocity and a gradual acceleration of the flow velocity as the body is enveloped.

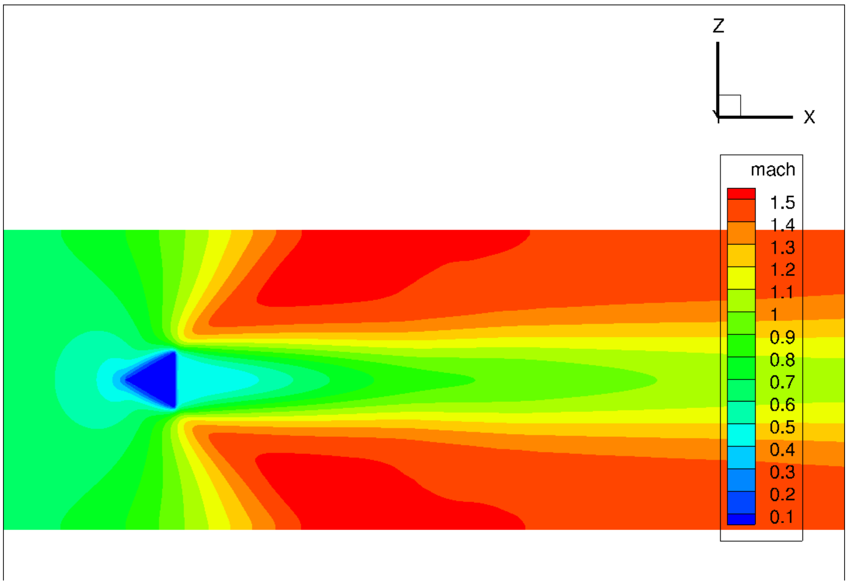


Figure 6 – Mach number $M_\infty = 1.3$ at ZX

In Figure 7 the temperature of the device may cause the device to overheat. Doing so may cause the temperature to drop below the specified value. Do not touch the sphere type with the cone type and the cone type with the cone type and the cone type with the cone type, but also with the cone type with the cone type and the cone type with the cone type and the cone type with the cone type.

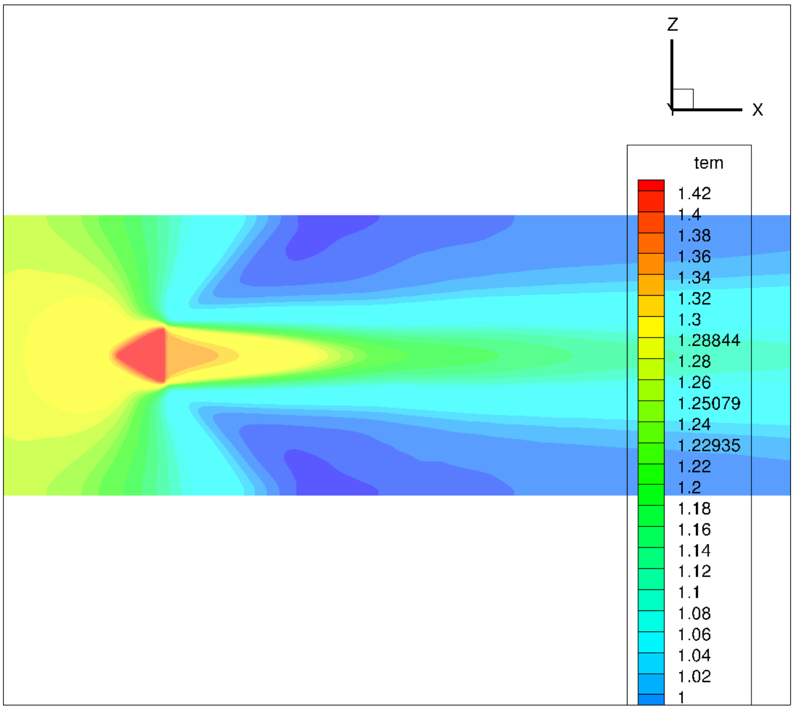


Figure 7 – Temperature T change at ZX

Figure 8 shows the contour of density flux distribution around a cone-type body. There is an increase in density in the region of the shock wave and a decrease in density in the rear region of the body.

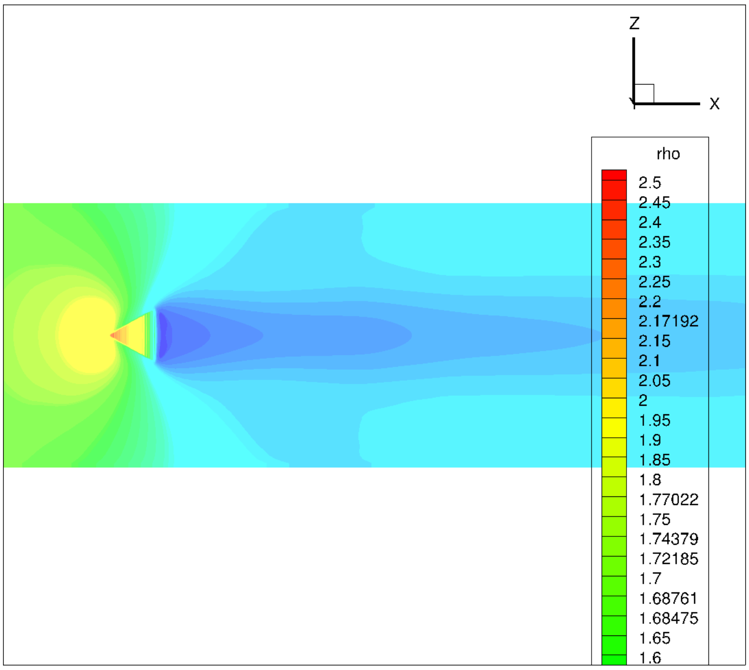


Figure 8 – Density ρ change at ZX

Figure 9 shows the pressure distribution when travelling around a cone-type body. Due to the shock wave in the frontal region of the body, the pressure increases, while the pressure decreases when the flow detaches from the body.

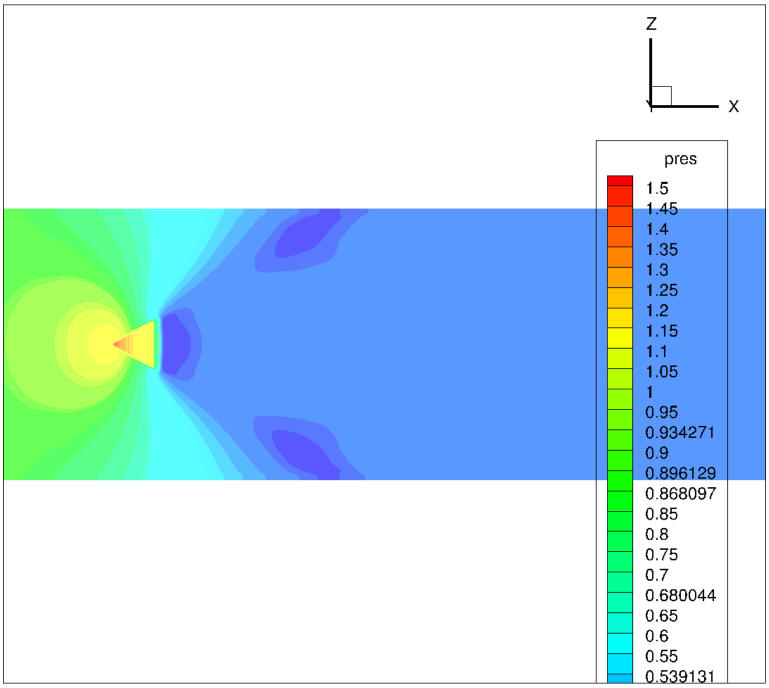


Figure 9 – Pressure P change at ZX (Problem 2)

Figure 10 compares the experimental data for the pressure coefficient C_p on the surface of the sphere (Problem 1). The distribution calculated for $M_\infty = 1.3$ the distribution agrees well with the experimental results; however, there are certain differences. The calculated dependence almost up to the cleavage point agrees with the data of [18].

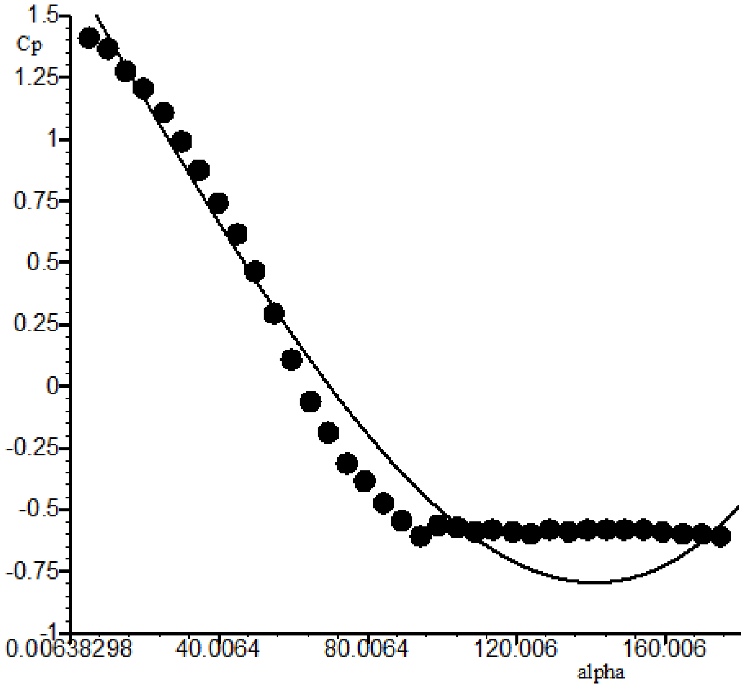


Figure 10 – The pressure coefficient C_p on the sphere surface is a constant solution for $M_\infty = 1.3$ (● ● ● is experiment and ——— is numerical result)

There are some limitations and difficulties in modelling cone and sphere flow by the penalty function method in supersonic flows. The main limitations of the penalty method are sensitivity to parameter selection, possible problems with resolution of sharp nonlinearities, geometric complexity and ensuring numerical stability of the model. When solving such problems, a combined approach including adaptive methods and adjustments of the penalty term model to account for the specificity of supersonic flows is often required.

Conclusion

The cone and sphere flow was analysed using the penalty function method for compressible flows. The analysis includes the flow conditions $Re = 1000$, $M_\infty = 1.3$, at which various physical features are manifested. An ENO scheme is created for the problem. The system of basic equations is reduced to a standard dimensionless form. The results of this analysis successfully demonstrate the physics of the full range of configurations studied by the model of the proposed method. An accurate prediction of the main volumetric quantities is obtained, in particular, the stability characteristics of the method when registering impact traces on the sphere surface and supersonic region are confirmed. As for the flow structure near the particle, a completely different picture was observed than in the incompressible case. It can be determined that the shock wave is not eliminated by turbulence, which means that the shock wave is still clearly visible in front of the particle.

This research work was developed using computational resources within the project. Rigorous tests were carried out considering different test conditions. The streamline images of the investigated cones showed the temperature, density and pressure distribution on the surfaces of the corrected bodies. The results were compared with many experimental and numerical references available in the literature.

Further studies of the penalty function method for modelling cone and sphere flow in supersonic flows can be developed in several interrelated directions. Particular attention should be paid to the adaptation of this method for high-precision modelling of shock waves and breakaway zones, which will significantly improve the reliability of the results obtained in critical flow regions. At the same time, it is necessary to improve approaches to accounting for viscous effects in the boundary layer, since these effects have a significant influence on the character of flow and thermal loads on the surfaces of bodies.

REFERENCES

1 Mittal R. and Iaccarino G. Immersed boundary methods, Annu. Rev. Fluid Mech., 2005, no. 37, pp. 241–260.

2 Abalakin I.V., Zhdanova N.S., Kozubskaya T.K. Immersed boundary method implemented for the simulation of an external flow on unstructured meshes, Computational Mathematics and Mathematical Physics, 2016, no. 8 3, pp. 219–230.

3 Peskin C.S. Flow patterns around heart valves: A numerical method, J. Comput. Phys., 1972, no. 10 2, pp. 252–271.

4 Brown-Dymkoski E., Kasimov N., and Vasilyev O.V. A characteristic based volume penalization method for general evolution problems applied to compressible viscous flows, J. Comput. Phys., 2014, no. 262, pp. 344–357.

5 Jiang Y., Wang X., Jing X., and Sun X. A study of three-dimensional acoustic scattering by arbitrary distribution multibodies using extended immersed boundary method, ASME. J. Vib. Acoust., 2015, no. 136 3, pp. 034505-7.

6 Fedkiw R. Coupling an Eulerian fluid calculation to a Lagrangian solid calculation with the ghost fluid method. J. Comput. Phys., 2002, no. 175, pp. 200–224.

7 Antoine Michael Diego Jost St´ephane Glockner, Direct forcing immersed boundary methods: Improvements to the Ghost-Cell Method, Journal of Computational Physics, 2021, no. 438 10, p. 110371.

8 Wei Li, Yihui Ma, Xiaopei Liu, Mathieu Desbrun, Efficient kinetic simulation of two-phase flows, ACM Transactions on Graphics, 2021, no. 41 4, pp. 1–17.

9 Clarke D.K., Hassan H.A., and Salas M.D. Euler calculations for multielement airfoils using cartesian grids, AIAA J., 1986, no. 24 3, pp. 353–358.

10 Liu Q. and Vasilyev O.V. Brinkman penalization method for compressible flows in complex geometries, J. Comput. Phys., 2007, no. 227, pp. 946–966.

11 Shun Takahashi, Taku Nonomura, and Kota Fukuda. A Numerical Scheme Based on an Immersed Boundary Method for Compressible Turbulent Flows with Shocks:Application to Two-Dimensional Flows around Cylinder, Journal of Applied Mathematics, 2013, no. 2014, pp. 1–27.

12 Riahi H., Meldi M., Favier Ju., Serre E., Goncalves da Silva E. A pressure-corrected Immersed Boundary Method for the numerical simulation of compressible flows, Journal of Computational Physics, 2018, no. 374, pp. 361–383.

13 Li Wanga, Fang-Bao Tian, John Younga. An immersed boundary method for the fluid–structure–thermal interaction in rarefied gas flow, Journal Physics of Fluids, 2024, no. 36 1.

14 Sambasivan S.K. and H.S. Uday Kumar. Ghost fluid method for strong shock interactions part 2: immersed solid boundaries, AIAA Journal, 2009, no. 47 12, pp. 2923–2937.

15 Hu X.Y., Khoo B.C., Adams N.A., and F.L. Huang. A conservative interface method for compressible flows, Journal of Computational Physics, 2006, no. 219 2, pp. 553–578.

16 de Tullio M.D., de Palma P., Iaccarino G., Pascazio G., and M. Napolitano, An immersed boundary method for compressible flows using local grid refinement, Journal of Computational Physics, 2007, no. 225 2, pp. 2098–2117.

17 Al-Marouf M., Samtaney R. A versatile embedded boundary adaptive mesh method for compressible flow in complex geometry, J. Com. Phys., 2017, vol. 337, pp. 339–378.

18 Bashkin V.A., Vaganov A.V., Egorov I.V., Ivanov D.V., and G.A. Ignatova. Comparison of Calculated and Experimental Data on Supersonic Flow Past a Circular Cylinder, J. Fluid Dynamics, 2002, no. 37 3, pp. 473–483.

19 Manapova A., Beketaeva A., Makarov V. Numerical modeling of essentially subsonic flows of compressible gas, Herald of the Kazakh-British technical university, 2023, vol. 20, no. 4, pp. 85–96. [In Russian]. https://doi.org/10.55452/1998-6688-2023-20-4-85-96.

20 Manapova A., Beketayeva A., Makarov V. Penalty function method for modeling of cylinder flow with subsonic compressible flow, Herald of the Kazakh-British technical university, 2024, vol. 21, no. 4, pp. 107–123. https://doi.org/10.55452/1998-6688-2024-21-4-107-123.

[1]*Манапова А.К.,
магистр прикладной математики и информатики,
ORCID ID: 0000-0003-1548-7061,
*e-mail: manapova.a.k.math@gmail.com
[2]Бекетаева А.О.,
доктор физико-математических наук,
ORCID ID: 0000-0003-4360-3728,
e-mail: azimaras10@gmail.com
[3, 4]Макаров В.В.
кандидат технических наук,
ORCID ID: 0000-0003-4874-5418,
e-mail: makfone@mail.ru

[1]Академия гражданской авиации, г. Алматы, Казахстан
[2]Институт математики и математического моделирования КН МОН РК, г. Алматы, Казахстан
[3]Институт проблем управления РАН, г. Москва, Россия
[4]Национальный исследовательский ядерный университет «МИФИ», г. Москва, Россия

ЧИСЛЕННОЕ МОДЕЛИРОВАНИЕ ОБТЕКАНИЯ СФЕРЫ И КОНУСА СВЕРХЗВУКОВЫМ СЖИМАЕМЫМ ПОТОКОМ

Аннотация

В данной работе рассматривается численное моделирование сверхзвукового обтекания тел конуса и сферы с использованием метода штрафных функций. Основной целью исследования является оценка эффективности метода штрафных функций, также известного как метод погруженной границы, для решения задач сжимаемой газовой динамики. Применяется модифицированные уравнения Навье-Стокса с учетом обтекаемых тел и используют схему ENO для численного решения. Результаты моделирования демонстрируют, что предложенный подход успешно описывает физические процессы, происходящие при сверхзвуковом обтекании конуса и сферы, включая формирование ударных волн, распределение давления, температуры и плотности. Полученные данные сравниваются с экспериментальными результатами, подтверждая адекватность и точность разработанной численной модели. Представленная работа вносит вклад в развитие методов численного моделирования сжимаемых сверхзвуковых течений и демонстрирует перспективность использования метода штрафных функций для решения широкого класса задач газовой динамики.

Ключевые слова: обтекание конуса, обтекание сферы, сверхзвуковой поток, турбулентное течение, сжимаемый газ.

[1]***Манапова А.Қ.,**
Қолданбалы математика және информатика магистрі,
ORCID ID: 0000-0003-1548-7061,
*e-mail: manapova.a.k.math@gmail.com
[2]**Бекетаева А.О.,**
Физика-математика ғылымдарының докторы,
ORCID ID: 0000-0003-4360-3728,
e-mail: azimaras10@gmail.com
[3,4]**Макаров В.В.**
Техника ғылымдарының кандидаты,
ORCID ID: 0000-0003-4874-5418,
e-mail: makfone@mail.ru

[1]Азаматтық авиация академиясы, Алматы қ., Қазақстан
[2]ҚР БҒМ ҒК Математика және математикалық модельдеу институты,
Алматы қ., Қазақстан
[3]РҒА Басқару мәселелері институты, Мәскеу қ., Ресей
[4]«МИФИ» Ұлттық ядролық зерттеу университеті, Мәскеу қ., Ресей

СФЕРА МЕН КОНУСТЫ ДЫБЫС ЖЫЛДАМДЫҒЫНАН ЖОҒАРЫ ЖЫЛДАМДЫҚТАҒЫ АҒЫНМЕН АЙНАЛЫП ӨТУДІ САНДЫҚ МОДЕЛЬДЕУ

Аңдатпа

Бұл зерттеу конус пен сфера тәрізді денелердің айналасындағы дыбыстан жоғары жылдамдықтағы ағынды сандық модельдеуге арналған. Модельдеу үшін айыппұлдық функция әдісі қолданылып, оның сығылатын газ динамикасы есептерін шешудегі тиімділігі бағаланады. Зерттеудің негізгі мақсаты – батырылған шекаралық әдіс ретінде белгілі жазалау функциясы әдісінің қолдану мүмкіндіктерін талдау. Модельдеу барысында айналып өтетін денелерді ескеретін модификацияланған Навье-Стокс теңдеулері пайдаланылды. Сандық шешім алу үшін жоғары дәлдікті ENO (Essentially Non-Oscillatory) схемасы қолданылды. Алынған нәтижелер ұсынылған әдістің конус пен сфера айналасындағы дыбыстан жоғары ағын кезінде туындайтын физикалық процестерді – соққы толқындарының түзілуін, қысымның, температураның және тығыздықтың таралуын дәл сипаттайтынын көрсетеді. Модельдеу нәтижелері тәжірибелік деректермен салыстырылып, әзірленген сандық модельдің сәйкестігі мен дәлдігі расталды. Зерттеу нәтижелері сығылатын газ динамикасы есептерінің кең класын шешуде айыппұлдық функция әдісінің перспективалы тәсіл екенін дәлелдейді және дыбыстан жоғары ағындарды сандық модельдеу әдістерін жетілдіруге ықпал етеді.

Тірек сөздер: конус арқылы өтетін ағын, сфера арқылы өтетін ағын, дыбыстан жоғары ағын, турбулентті сығылатын газ ағыны.

Article submission date: 25.01.2025