Modifed Method of Calculation of the Flows in Areas with Moving Boundaries

I. P. Boychuk[1], A. V. Grinek[2], I. M. Dantsevich[3]
Admiral Ushakov Maritime State University
Novorossiysk, Russia
[1]igor_boichuk@mail.ru, [2]grinyokann@gmail.com, [3]dantsevich65@mail.ru

***Abstract*— The method of calculation of flows in areas with moving boundaries is considered in the article. Solid body in a flow is modeled as a continuously distributed volume force field. The reaction of the flow to a solid is found from the solution of the problem of the interaction of a homogeneous gas flow with a barrier. A mapping method of guiding properties of smooth boundaries for increasing the accuracy of calculations on rough Cartesian grids is proposed. The possibility of using this approach for simulation of flow past bodies with moving boundaries is demonstrated.**

Key words— Cartesian grids; moving boundaries; guiding properties; numerical method; flow calculations

I. Introduction

Finding solutions in areas with complex geometry is one of the most important problems in computational fluid dynamics. Most methods for calculating hydrodynamic flows assume a computational grid, in which solids are monitored with means of boundary grid nodes. Curvilinear boundaries are represented as structured or unstructured grids. The construction of a qualitative grid itself is not an easy task, the solution of which is often formalized as a separate grid generator, and the quality of this stage performance affects on the accuracy of the final results.The time for constructing the grid can be commensurate with the calculation time in those cases when solids have a complex shape. For problems of optimizing the geometry of the body, when adaptive grids are also used [1], the time spent on building and reconstructing the grid can be especially critical.

Alternative methods based on the use of adaptive grids are finite-difference methods that allow to solve equations geometrically in complex areas of Cartesian grids [2]. The construction of Cartesian grids is quite economical. In addition, the methods of Cartesian grids make it possible to solve problems with moving boundaries and change of the geometry of the body without their multiple rebuilding. The main disadvantage of this approach is the difficulty of approximating rather complex calculation areas with rectangular boundaries. In this case Cartesian grids due to local degeneracy can either not be constructed in principle, or they are far from optimal, which leads to large errors.

The authors propose to use the developed and proven approach for the calculation of flows in turbomachines for modeling the flow past moving bodies with curvilinear surfaces [3], so-called " actuator disc," which consists in replacing the turbine in the simulation by a continuous force field.

II. Methodology of research

A. Statement of the problem

Three-dimensional flow of a viscous compressible flow in area Ω_f (fig. 1) is described by an equation system expressing the laws of conservation of mass and momentum:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho w_j)}{\partial x_j}=\delta_1, \tag{1}$$

$$\frac{\partial(\rho w_j)}{\partial t}+\frac{\partial(\rho w_j w_k)}{\partial x_k}+\frac{\partial p}{\partial x_j}=\delta_{2j}\,,\ \ \mathrm{j}=\overline{1{,}3}\,,\ \text{в}\ \Omega_f\,, \tag{2}$$

$$\begin{gathered}\alpha w_j+\beta\frac{\partial w_j}{\partial n}=\gamma\ \text{на}\ \Sigma_f\,,\\ w_i(\vec{x},t_0)=w_{0i}\,,\\ w_i(\vec{x},t)\big|_{\Sigma_b}=U_i(t)\,,\end{gathered} \tag{3}$$

where Ω_f – calculation area with boundary Σ_f, Ω_b – streamline body with boundary Σ_b, w_j – components of the flow velocity vector, p – pressure, x_j – coordinates, ρ – density, δ_1 – density source, δ_{2j} – impulse sources, α,β,γ – coefficients depending on the type of known boundary conditions, U_i – components of the velocity vector of the body. The system (1) – (2) is closed by the equation of state of the medium.

To solve system (1) – (3) on a curvilinear grid, it is necessary to sample the boundary Σ_b of the body Ω_b and then construct a grid of the space Ω_f.

According to [4], the influence of the body Ω_b on the flow can be taken into account by adding the volume force $f(x_1,x_2,x_3,t)$ to the equations of motion. Then the momentum equation has the following form

$$\frac{\partial(\rho w_j)}{\partial t}+\frac{\partial(\rho w_j w_k)}{\partial x_k}+\frac{\partial p}{\partial x_j}=\delta_{2j}+f_j(x_1,x_2,x_3,t). \tag{4}$$

978-1-5386-5824-6/18/$31.00 ©2018 IEEE

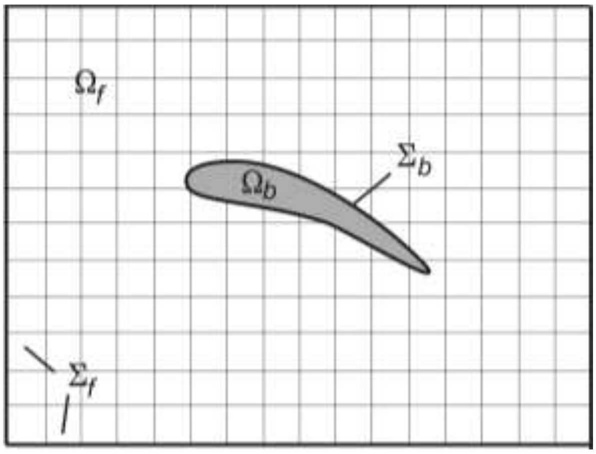

Fig. 1. Geometry of the computational domain

Instead of the condition for the impermeability of the body surface Ω_b, a force field continuously distributed over the volume equivalent to the initial one at each moment of time is introduced. The boundary of the area occupied by the force field coincides with the boundary of the considered body Ω_b. The expression for force $f(x_1, x_2, x_3, t)$ is determined on the basis of the physical properties of a particular problem [5-15].

B. Motion of the flat plat

Let us consider the motion of a thin plate immediately immersed in a stream (Fig. 2). The plate moves with velocity U. Gas dynamic load $\delta P = p_+ - p_-$ appears on the body contour in the process of flowing around the body, where $p_\pm$ is the pressure on the contour. The change in pressure on the contour depends on the parameters of the unperturbed flow and on the mutual orientation of the flow and body. In this case, the direction of the force action coincides with the external normal ($\vec{f} \parallel \vec{n}_\pm$) to the profile at the corresponding point of force application.

The pressure values on the contour are determined by the solution of the self-simulated problem [16] on the interaction of a homogeneous gas flow with an obstacle [3]. As a result a force which is determined by the pressure gap in front of and behind the profile is obtained. This makes it possible to obtain velocity and pressure field with allowance for the effect on the flow of a solid in the considered flow area.

Flow in front of the profile. The flow must stop before the plate. Obviously, the braking process must be carried out in a compression wave.

Flow behind the profile. The flow is obviously connected with the appearance of a rarefaction wave behind the plate.

Flow characteristics (velocity and pressure) are determined according to [3].

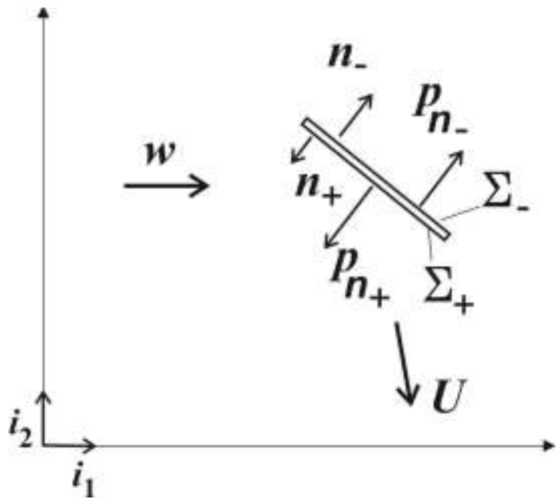

Fig. 2. Model of the force interaction of a contour with a flow of a compressible medium

C. Display of directing surface properties

Generally speaking, the constructed flow occurs along streamlines that do not coincide with the contours of the body, i.e. there is a flow of mass through the body. An isentropic interaction model taking into account that the surface forces do not perform work in the reference system associated with the streamlined object is used to take into account the directing properties of the surfaces of bodies on relatively coarse grids: $(\vec{f}, \vec{w}) = 0$. Such model operates on the concept of the force action on a Lagrangian particle, which acquires a given direction of motion from the initial position after a short period of time. This model allows us to find the velocity vector taking into account the rotation of the flow (Fig. 3), which makes it possible to further determine the pressure and temperature.

Using the interaction model in the area adjacent to the body we obtain the velocity vector taking into account the profile guiding properties [5]:

$$\begin{cases} w_1^{(v)} = w^{(v_m)} \cos\alpha_{2,3} \sin\alpha_{1,3} \\ w_2^{(v)} = w^{(v_m)} \cos\alpha_{1,3} \sin\alpha_{2,3} \\ w_3^{(v)} = w^{(v_m)} \cos\alpha_{1,3} \cos\alpha_{2,3} \end{cases}$$

where $\alpha_{i,j}$ – the cosines of the angles between the velocity vector and the unit vectors of the coordinate system associated with the profile.

A smooth flow pattern of the body which is constructed with a detailed account of the curvature of the surface (Fig. 4) is obtained as a result of modeling.

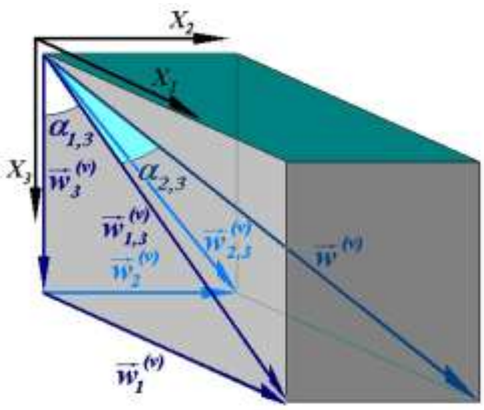

Fig. 3. Components of the velocity vector in a given direction field

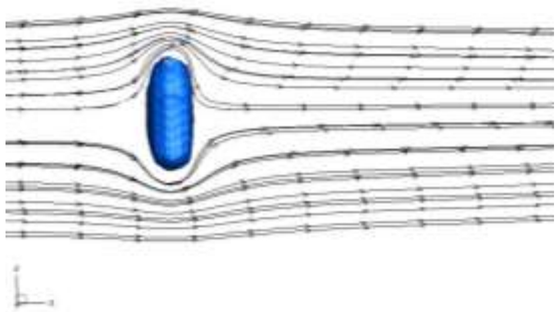

Fig. 4. Flow around the body with concidering the directing properties of the surface

III. Results

Two-bladed propeller working in a compressible fluid flow was simulated using the proposed modified method. The results of the calculation obtained with the aid of the developed method shown in Fig. 5 showed good qualitative and quantitative agreement with the experimental data.

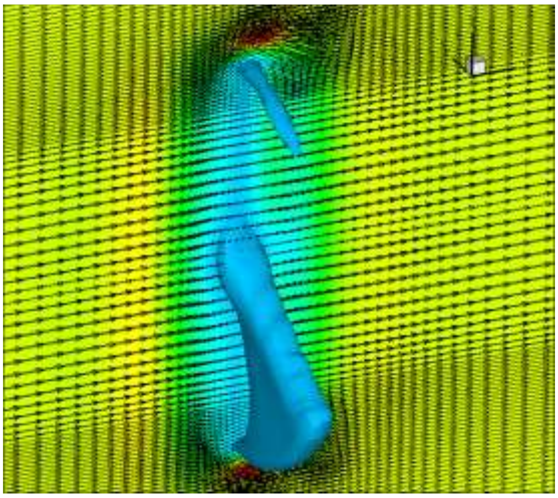

Fig. 5. Isosurface of velocity, velocity vectors and velocity field in the computational domain

The proposed method allowed modeling mobile objects with curvilinear boundaries without multiple rebuilding of the computational grid. At the same time the calculation speed has increased [17].

Acknoledgments

The authors are grateful to Professor Ambrozhevich A.V. for ideological inspiration and useful criticisms in the performance of this work.

References

[1] Weizhang H., Russell R. D. Adaptive Moving Mesh Methods. Springer-Verlag New York. 2011.

[2] Vinnikov V.V., Reviznikov D.L. Application of Cartesian grids to solve the Navier-Stokes equations in domains with curvilinear boundaries. Matem. modeling. 2005. Vol. 17. No. 8. Pp. 15-30.

[3] Ambrozhevich A.V. Numerical modeling of a complex of non-stationary gas-dynamic processes in thermal engines. Kharkov. Publ. of the Kharkov State Technical University. 1999. 77 p.

[4] Mortikov E.V. Application of the Immersed Boundary Method to Solve the Navier-Stokes Equation System in Domains of Complex Configuration. Computational Methods and Programming. 2010. Vol. 11. Pp. 32-42.

[5] Conlisk A.T. Progress in Aerospace Sciences. 37. 2001. Pp. 419–476.

[6] Mikkelsen R. Actuator Disc Methods Applied to Wind Turbines. MEK-FM-PHD 2003-02, 2003.

[7] Mikkelsen R., Sørensen J.N. Yaw Analysis Using a Numerical Actuator Disc Model. Proc. 14th IEA Symp. on the Aerodynamics of Wind Turbines. Boulder, Col, USA. 2000. 53.59.

[8] Mikkelsen R, Sørensen J.N., Shen W.Z. Yaw Analysis Using a 3D Actuator Line Model. European Wind Energy Conf. Copenhagen, Danmark. 2001. 478.480

[9] Verweij A.P. An investigation in actuator disc CFD solution applicability for aeroacoustics analysis ofpropellers and rotors. Internship report. CIRA. 2010.

[10] Renaud T., O'Brien D., Smith M., Potsdam M. Evaluation of Isolated Fuselage and Rotor-Fuselage Interaction Using Computational Fluid Dynamics. Journal of the American Helicopter Society. 2008. No. 53. Pp. 3-17. 10.4050/JAHS.53.3.

[11] Houlsby G.T., Draper S., Oldfield M.L.G. Application of Linear Momentum Actuator Disc Theory to Open Channel Flow. Report No. 2296/08, OUEL.

[12] Simisiroglou N., Breton S-P., Ivanell S. Validation of the actuator disc approach using small-scale model wind turbines Wind Energ. Sci. 2. Pp. 587–601. 2017. https://doi.org/10.5194/wes-2-587-2017

[13] Conway J.T. Analytical solutions for the actuator disc with variable radial distribution of load. J. Fluid Mech. 1995. 297. 327.355.

[14] Conway J.T. Exact actuator disc solutions for non-uniform heavy loading and slipstream contraction. J. Fluid Mech. 1998. 365. 235.267.

[15] Pope S.B. Turbulent Flows. Cambridge University Press. 2000.

[16] Boychuk I.P. Complex aerodynamic modeling of transport systems with direct reaction engines. Belgorod. Publ. of BSTU. 2016. 230 p.