Linearization of boundary conditions in solving problems by the methods of computational aerodynamics

F A Maksimov

Institute of Computer Aided Design of the RAS, 19/18, 2-ya Brestskaya ul., Moscow 123056, Russian Federation

E-mail: f_a_maximov@mail.ru

Abstract. To simplify the calculations of flow about wing with different possible deflections of aerodynamic surfaces, the algorithm with linearization of a boundary condition of impermeability is offered and implemented. The solution of the Euler equations is obtained by time marching to a steady state. The computational grid for the aerodynamic configuration is made in the variant with surfaces without deflection but when setting a normal vector to the surface it is necessary to take into account not only the direction of the normal towards the local surface but also an angle of rotation (or deflection) of this part of the surface for the calculated variant.

1. Introduction

When solving aerodynamic designing problems by methods of computational mathematics, it is necessary to construct a model of an aircraft and a computational grid near it. Deflection of aerodynamic surfaces changes geometry and it is required to remake the computational grid. To simplify the research into aerodynamic properties with different possible deflections of aerodynamic surfaces, the algorithm with linearization of a boundary condition of impermeability is offered and implemented.

Linearization of boundary conditions is widely used in the theoretical aerodynamics [1, 2]. When solving a problem about the flow past a wing by the methods of the linear theory, boundary conditions as well as equations are linearized. According to the approximation of the thin wing, the boundary condition of impermeability that includes the direction cosines of the wing surface but not the coordinates of this surface is set not on the real surface of the wing but on the nearby azimuthal surface. In fact, it notably simplifies the geometry of the problem’s definition range.

This way is also efficient in solving problems by the methods of the computational aerodynamics. For example, it is used to compute a flow past an aircraft with an axisymmetric body with profiled aerodynamic surfaces by the marching method [3], to compute a rotating body with the empennage of skewed plates [4], to compute an aircraft with an axisymmetric body with a set of aerodynamic surfaces and deflections for the creation of the pitching moment [5]. In those cases, linearization of the impermeability condition on aerodynamic surfaces allows to use a simple algebraic computational grid.
and to implement the algorithm of solving the problem about the flow of rather complicated geometries consisting of an axisymmetric body and a set of thin aerodynamic surfaces.

The comparison of calculations by these methods with theoretical and experimental results demonstrates the high efficiency and validity. The efficiency is secured by the use of linearization of a boundary condition of impermeability on the area of aerodynamic surfaces while the validity is based on the use of a rather complete model of the Euler equations. Surely, when using linearization, one has to understand that the application area for this method is limited. First, it is limited by the range of angles of attack with which the results on the inviscid gas model especially with a linearized boundary condition can truly describe a real flow. When angles of attack in a real flow are rather big, the flow separation with the vortex structure formation is usually implemented on some elements of an aircraft that tends to drastically change aerodynamic characteristics. Capabilities to describe such flows within the Euler equations are limited. It is enough to consider only small angles of attack for the solution of many practical problems. Problems about the aerodynamics of the flight with big angles of attack are to be solved with the application of more complete models if necessary.

This work offers to use linearization of the boundary condition of impermeability for the research into flows past wings with deflectable (pivoting) parts. The solution of the problem is based on the Euler equations by the pseudoviscosity method. The computational grid for the aerodynamic configuration is made in the variant with surfaces without deflection but when setting a normal vector to the surface it is necessary to take into account not only the direction of the normal towards the local surface but also an angle of rotation (or deflection) of this part of the surface for the calculated variant.

2. The mathematical model

The nonstationary three-dimensional Euler equations of the compressible gas in the dimensionless form in the $X = (x, y, z)$ cartesian coordinate system look as follows:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} = 0,$$

$$\mathbf{U} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e \end{pmatrix}, \quad \mathbf{E} = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ (e + p)u \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho vw \\ (e + p)v \end{pmatrix}, \quad \mathbf{G} = \begin{pmatrix} \rho w \\ \rho uw \\ \rho vw \\ \rho w^2 + p \\ (e + p)w \end{pmatrix}.$$

Here $t$ is time, $\rho$ is density, $(u, v, w)$ are the velocity $\mathbf{V}$ vector components in the $(x, y, z)$ directions, $p$ is pressure, $e$ is the internal energy of the perfect gas that can be determined in accordance with the formula:

$$e = \rho \left( \frac{u^2 + v^2 + w^2}{2} \right),$$

where $e = \frac{1}{\gamma - 1} \frac{p}{\rho}$ is the internal gas energy, $\gamma$ is the adiabat parameter. The dimensionless variables are determined via the «$\prime$»-marked dimensional variables in accordance with the formula:

$$t = \left[ \frac{p}{\rho L'} \right]', \quad X = \left[ \frac{X'}{L'} \right]', \quad V = \left[ \frac{V'}{L'} \right]', \quad \rho = \left[ \frac{\rho p'}{p o} \right], \quad p = \left[ \frac{p'}{p o} \right].$$

The «$\prime$» low index is the value of the parameter in the undisturbed flow. $L'$ is the scale length; $X = (x, y, z); \ V = (u, v, w).$
The use of the Cartesian coordinate system to describe flows with the complicated topology in solving problems by the finite-difference methods is inconvenient. So let’s move to the arbitrary curvilinear coordinate system [6]. We will set the uniform mesh for difference approximation of input equations in the coordinate system:

\[ \tau = t; \quad \xi = \xi(x, y, z); \quad \eta = \eta(x, y, z); \quad \zeta = \zeta(x, y, z). \]

The use of the generalized transformation makes it possible to keep the strictly conservative form of equations [6]. So the equations look as follows:

\[
\frac{\partial U}{\partial \tau} + \frac{\partial E}{\partial \xi} + \frac{\partial F}{\partial \eta} + \frac{\partial G}{\partial \zeta} = 0
\]

Here: \( \overline{E} = \xi E + \xi F + \xi G; \quad \overline{F} = \eta E + \eta F + \eta G; \quad \overline{G} = \zeta E + \zeta F + \zeta G \). The coefficients of the transformation matrix and the jacobian (J) are determined by the well-known formulas:

\[
\xi_x = J(y \eta \zeta - z \eta \zeta); \quad \xi_y = J(z \eta x - x \eta x); \quad \xi_z = J(x \eta y - y \eta y);
\]

\[
\eta_x = J(y \zeta \eta - z \zeta \eta); \quad \eta_y = J(z \zeta x - x \zeta x); \quad \eta_z = J(x \zeta y - y \zeta y);
\]

\[
\zeta_x = J(y \zeta \eta - z \zeta \eta); \quad \zeta_y = J(z \zeta x - x \zeta x); \quad \zeta_z = J(x \zeta y - y \zeta y);
\]

\[
J^{-1} = x \eta \eta \eta \eta + y \zeta \zeta \zeta \zeta + z \xi \xi \xi \xi - x \zeta \zeta \zeta \xi - x \xi \xi \xi \zeta - x \xi \xi \xi \xi
\]

The use of the generalized transformation permits to make the uniform mesh in the shape of the unit cube. In the given node distribution in the physical calculation area, the coefficients of the transformation matrix and the jacobian are computed with the help of the symmetric difference formulas.

When solving problems about the ultrasonic flow past the surface of an aircraft via the Euler equations, the conditions of impermeability are set. When using the pseudoviscosity method, the implementation of the given boundary condition is fulfilled after each step in time in the form of the correction of the velocity vector in each node placed on the wing to make the velocity vector parallel to the surface of the body. Conditions in the incident flow are set on the entrance outer boundary; while mild boundary conditions are set on the exit boundary. The correct placement of the outer boundary is the necessary condition for the adequate solution of the problem. The outer boundary is to be placed in the area of the undisturbed flow.

The two-step difference scheme of the predictor–corrector type is used for the computational modeling [7, 8]. To suppress oscillations on the fronts of the shock waves, the smoothing such as the artificial viscosity is introduced. It can conveniently be presented as follows:

\[
U_{i,j,k}^{n+1} = (1 - 6\varepsilon)U_{i,j,k}^{n+1} + \varepsilon(U_{i-1,j,k}^{n} + U_{i+1,j,k}^{n} + U_{i,j-1,k}^{n} + U_{i,j+1,k}^{n} + U_{i-1,j-1,k}^{n} + U_{i+1,j-1,k}^{n} + U_{i-1,j+1,k}^{n} + U_{i+1,j+1,k}^{n})
\]

where \( \varepsilon \approx 0.001 \) is a small parameter.

Please follow these instructions as carefully as possible so all articles within a conference have the same style to the title page. This paragraph follows a section title so it should not be indented.

3. The construction of the mesh near the wing

The easy way of the mesh construction to research into the flow past the sharp-edged wings is implemented. The quasi-3D mesh is made by the set of the sections of perpendiculars to the central chord of the wing. In each section, the mesh is constructed with the help of the Zhukovsky function. The surface of the wing is placed in the plane of the section. The exemplary computational grid near the rectangular wing planform is given on figure 1.
The Zhukovsky function makes it possible to construct the mesh near the zero-thick segment. To shift to the non-zero-thick wing, we take each coordinate line made by the normal to the median surface of the wing to find the point of the intersection between this line and the surface of the considered wing. Then the nodes along this line are redistributed from the surface of the wing to the given outer boundary of the calculating area. As a result, we get the mesh exactly near the considered wing.

The final mesh is not conformal as the angles of the cells are deformed over the different node displacement along the adjacent coordinate lines. Coordinate lines can also be non-perpendicular to the surface of the wing. But all the deformations of the computational grid are conditional on the considered geometry of the wing, so we can view all the deformations of the grid’s cells as permissible because the wing is considered within the linearized boundary conditions.

![Figure 1. Construction of mesh near the wing.](image)

Figure 1 shows the wing and the mesh in two sections: in the symmetry plane and in the output cross-section. The meshes in each of the sections perpendicular to the chord of the wing are similar. The surface of the wing is placed near the section. If the node of the section gets to the surface of the wing, so the condition of the impermeability on windward and leeward sides respectively is set on it. If the node doesn’t belong to the surface of the wing, the solutions on windward and leeward sides are united – the gas-dynamic properties in the section are averaged.

This way doesn’t require the strict separation of the wing edges but when setting the position of the boundary of the section it can be chosen in the way to conform to the front edge of the swept wing or to the side edge of the wing. When studying the rectangular wing, edges of the wing will conform to some coordinate lines. The mesh is made not only in the wing area but also in some area in front of and behind the wing. The mesh behind the wing is made to compute the trail. However, applying the concentration, the main part of the sections is placed in the area where the wing is positioned.

The normal to the surface of the wing is determined after making the mesh in each node with the help of the vector product of two vectors placed tangentially to the surface of the wing:

\[
R_{i,k}^1 = X_{i+1,k} - X_{i-1,k}, \quad R_{i,k}^2 = X_{i,k+1} - X_{i,k},
\]
here $i$ is the index of the mesh’s node in the circular direction, $k$ is the index of the section for a longitudinal coordinate. So the formula to describe the normal to the surface of the body looks as follows: $\mathbf{n} = \frac{R^1 \times R^2}{|R^1 \times R^2|}$.

4. Test computation
To check the method and the implemented program, the computation of the flow of the rectangular wing by the extension 2 with the trapezoid profile was made. The results of the experimental research into this wing are given on [9]. The rather thick wing with the 30-degree angle between the windward and leeward sides near edges is considered. Figure 2a shows the comparison of the distribution of the $C_p$ pressure coefficient along the central chord with the $M=2.86$ Mach number and the $\alpha=10^\circ$ angle of attack. The thick line shows the result of the computation while markers show experimental data [9].

![Figure 2](image-url)

**Figure 2.** Pressure distribution along central chord of the wing (a) and character of the flow near rear edge of the wing within Euler (b) and Navier-Stokes (c) equations.

The results of the computation and the experiment on the windward side (the upper curve and the appropriate set of markers) are well-coincided. As for the leeward side (the lower curve), the results of the computation and the experiment are also well-coincided near the front edge ($x=0$) and in the center up to $x=0.75$; however, they differ near the rear edge.

Figures 2b and 2c show the wing and the calculated lines of flow in the plane going through the central chord of the wing and obtained within the Euler and Navier-Stokes equations respectively. It can be seen that when using the Euler equations, it is possible to make the solution without the flow separation near the rear edge. Within the Navier-Stokes equations, the separation with the formation of the circulating zone appears on the leeward side. As a result, the flow practically doesn’t turn on the bend of the profile in contrast to the inviscid solution. It results in the fact that the pressure on the leeward side near the rear edge of the wing practically copies that near the central part.

Figure 2a demonstrates the calculation result within the Navier-Stokes equations with the dash-and-dot line. This calculation result is well-conformed with the experiment [9]. The calculation based on the Navier-Stokes equations is made by the method [8] with the relatively small Reynolds number ($Re \approx 7 \times 10^4$). The presence of the separation and the area of the circulation flow near the rear edge on the leeward side of the wing within the local angle of attack $\alpha_\infty \approx 10^\circ + 15^\circ = 25^\circ$ is in fact expected.
The results of the comparison of the computation via the given method with the experiment and the computation based on the Navier-Stokes equations allow to draw the conclusion that the developed modelling method secures the adequate forecast of the flow parameters in the conditions when the flow without separation is implemented. The method can be applied to research into the thin wings.

5. Computation results

Figure 3 demonstrates the computation of the flow past the rectangular wing planform with extension 2. In this case, the wing is rather thin – the angle between windward and leeward sides near edges amounts to ≈11.4 degrees. The computation is made with the \( M=2 \) Mach number and the \( \alpha=10^\circ \) angle of attack.

![Figure 3a](image1.png) ![Figure 3b](image2.png) ![Figure 3c](image3.png)

**Figure 3.** The example of the computation of the flow past the wing.

Figure 3a demonstrates the general view of the flow past the leeward side in the form of the pressure distribution on the wing and in the plane of the flow symmetry as well as the flow lines drawn through the points near side and rear edges.

Figures 3b and 3c demonstrate the pressure distribution on windward and leeward sides of the wing (the part of the wing from the central chord to the side edge is drawn). The wing profile is diamond-shaped which reflects in the pressure distribution. The local normal to the surface of the wing in the central part of the wing is bidirectional: one direction is near the front edge (the bottom of the figures) and the other is near the rear edge. Two pressure levels are implemented respectively – on the windward side of the wing and on the leeward side. The typical sphere with the loss of the lifting area is formed near the edge. Figure 3a demonstrates the formation of the tip vortex.

This computation is made without linearization of a boundary condition. The local normal is set for each point of the surface of the wing in accordance with the geometry, and this normal is calculated in accordance with the described-above algorithm.

To compute the flow near the wing with its bend or a bend of a part of the wing in accordance with linearization of boundary conditions, it is enough to change the normal vector to the surface of the wing in the nodes belonging to the pivoting part.

The bend of the wing relative to the straight line perpendicular to the plane of the OXY angle of attack is considered. The projection of the normal in the form of the two-dimensional vector \((n_x, n_y)\) is turned by the given angle \(\delta\). The turn is implemented via the shift from \((n_x, n_y)\) to polar coordinates \((r, \varphi)\), and then the formula for the components of the changed normal looks as follows:

\[
 n_x^* = r \cdot \cos(\varphi + \delta), \quad n_y^* = r \cdot \sin(\varphi + \delta).
\]

It should be said that the position of the rotation axis isn’t taken into account for the turn in the linearized approach.
To check the possibility and the correctness of this method, the test computations in $\alpha=5$ and $10^\circ$ conditions without a turn and with the turn of the whole wing by $\delta=+5$ and $-5^\circ$ respectively without changing the angle of attack were made. In this case, the bend of the wing is made in accordance with the linearization method, i.e. the normal to the surface of the wing is changed in the respective way but the shift of the surface of the wing isn’t taken into account, and the computational grid is not remade.

Figure 4 demonstrates the comparison of the computation results in the form of the pressure distribution along the central chord. Lines on figures 4a and b show data for $\alpha=5$ and $10^\circ$ angles of attack respectively. The angle of attack is determined by external flow conditions. Markers show the results for $\alpha=10^\circ$, $\delta=-5^\circ$ and $\alpha=5^\circ$, $\delta=+5^\circ$ angles of attack respectively. The respective calculations obtained in fact with the one total angle of attack are well-conformed. This result proves that linearization of boundary conditions permits to turn the wing in the flow without remaking the mesh.

![Figure 4](image)

**Figure 4.** Pressure distribution along the central chord with $\alpha=5$ (a) and $10^\circ$ (b).

As for the practical application, the given example of computation has no meaning as the calculation with the turn of the wing on the whole is usually easier to implement via the change of external flow conditions. Examples of calculation when only a part of the surface of the wing turns are given below. In fact, in this case the normal changes not for the whole surface of the wing but only for those nodes of the computational grid that belong to the pivoting part. The area of the pivoting part is determined by geometric requirements.

Figures 5a and b demonstrate the examples of calculation ($M=2$, $\alpha=5^\circ$) for rectangular and swept-shaped wings with the turn of the part of the wing along the rear edge by the 5-degree angle. The deflectable part of the wing in the form of the parallelogram is well-marked by isolines.

The developed modelling method of the flow past a wing with the use of the computational grid with a rather big number of nodes permits to research into aerodynamic characteristics of the wings with a variety of planforms.

Figure 6 demonstrates the computation for the wing with the circle planform with the control surfaces placed behind and with $M=2$, $\alpha=5^\circ$. Figure 6a demonstrates the general view of the wing with the pressure distribution in the form of isolines in the trail of the wing, Figure 6b demonstrates the pressure distribution on the windward part. This wing is formed of two aerodynamic surfaces: one is a main part in the shape of two parts of the ball surface and the other is the surface of the wing with the diamond-shaped profile. In this case, the pressure change on the smooth surface of the wing is gradual.

The given examples of computation are made on the mesh 125-51-111 (in the circular direction, by the normal, along the chord of the wing) which makes it possible to calculate the variant of the wing on a PC within an hour.
Figure 5. Example of computing wings with turn of a part of a wing for rectangular (a) and swept-shaped (b) wings.

Figure 6. The example of computation of the flow past the wing of the complicated shape.
6. Conclusions
The method of computation of the supersonic flow past sharp-edged wings is developed and implemented. To compute the flow past wings with deflectable parts (aerodynamic control surfaces) on the basis of the inviscid gas model to secure a possibility of computation on the one fixed mesh, one can use linearization of boundary conditions.

7. References
[1] Karman T 1948 *Supersonic Aerodynamics. Principles and Applications* (Moscow: IL) p 99
[2] Krasilschikova E A 1952 *Wing of Finite Span in Compressible Flow* (Moscow: Gostekhizdat) p 158
[3] Wardlaw A B, Solomon J M, Baltakis F P 1981 Supersonic Inviscid Flow Field Computations of Missile Bodies *AIAA Journal* **19** pp 899-906
[4] Rozin A V 1983 Space Flow Past Finned Body by Supersonic Stream of Ideal Gas *Izvestiya AN the USSR. Mechanics of Liquid and Gas* **1** pp 185-187
[5] Dunaev V A and Maksimov F A 2006 *Simulation of Supersonic Flows of Inviscid Gas* (Tula: Tula State University Press) p 186
[6] Vivand H 1974 Conservative Forms of Gas Dynamics Equations *La Recherche Aerospatiale* **1** pp 65-68
[7] MacCormack R W 1969 The Effect of Viscosity in Hypervelocity Impact Cratering *AIAA paper* pp 69-354
[8] Maksimov F A, Churakov D A and Shevelev Yu D 2011 Development of Mathematical Models and Numerical Methods for Solving Problems of Aerodynamic Designing on Multiprocessor Computing Machinery *Computational Mathematics and Mathematical Physics* **51** pp 284-307
[9] Stallings R L Jr and Lamb M 1981 Wing-Alone Aerodynamic Characteristics for High Angles of Attack at Supersonic Speeds *NASA TP* 1889