Bulgarian Academy of Sciences. Space Research Institute.
Aerospace Research in Bulgaria. 18. 2003. Sofia

EULER COMPUTATIONS OF AN AIRFOIL USING FINITE VOLUME METHOD AND RUNGE - KUTTA TIME STEPPING SCHEME

Konstantin Metodiev

Space Research Institute - Bulgarian Academy of Sciences

Abstract.

Finite volume method for spatial discretization of two-dimensional Euler equations is used to describe flow of an inviscid gas among structural elliptic grid. A four step Runge-Kutta scheme is also applied for time integration. A non-linear artificial viscosity is added to suppress numerical oscillation of solution. The boundary conditions at inflow and outflow are based on the method of characteristics. The results for some subsonic cases are presented and collated with experimental data.

Introduction.

For the several decades past a wide number of methods for simulation of viscous compressible flow have been developed. One of them, a finite volume method, is proved simple and efficient for calculation of such cases. The choice of this method is determined by these factors.

It is possible to apply the method for various cases of space discretization. This fact determines two forms of numerical flux implementation – the so called cell and node centering. It is useful to apply cell centering while structural quadrilateral grid is used. Otherwise, in case of triangular meshes, both ways of centering are permitted but in case of node centering an overlapping of two adjacent cells occurs. In the case when the grid is generated by solving a system of Laplace equations, a solution among curves extremely close to the flow equipotential and stream lines is asked.

When the spatial discretization is done, the considered partial differential equation is reduced to ordinary one. There are different numerical methods for solving space – discretized Euler equations but most frequently applied are Runge – Kutta schemes. These schemes use

information from only one previous iterative stage. Their advantage is the increased accuracy. Actually, when high accuracy is not required, it is possible to adjust the current iterative step size. This fact allows acceleration of the numerical process.

Theoretical background.

Conservative system equations, describing compressible gas flow among Cartesian coordinates (2D), is consisting of continuity, Euler and energy equations. It has the form:

$$(1)\qquad \frac{\partial}{\partial t}\vec{q}(x,y,t)+\frac{\partial}{\partial x}\vec{f}\left[\vec{q}(x,y,t)\right]+\frac{\partial}{\partial y}\vec{g}\left[\vec{q}(x,y,t)\right]=$$

$$=\frac{\partial}{\partial t}\begin{Vmatrix}\rho\\ \rho u\\ \rho v\\ E\end{Vmatrix}+\frac{\partial}{\partial x}\begin{Vmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ (E+p)u\end{Vmatrix}+\frac{\partial}{\partial y}\begin{Vmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ (E+p)v\end{Vmatrix}=0$$

where ρ is density, u, v are Cartezian velocities, p is static pressure and E is the total internal energy per unit mass. Then the following problem is formulated: find out the monotonous scalar functions ρ, u, v, E and p, initially defined in space of states, with initial conditions

$$\vec{q}(x,y,0)=\vec{q}_0(x,y)$$

and boundary conditions

$$B\left(\vec{q}(x,y,t)\right)=0$$

Here B is operator for determination

$$\vec{q}(x,y,t)=\left\|\rho \quad \rho u \quad \rho v \quad E\right\|^T$$

and p is replaced with the following expression:

$$p(\rho,e)=\rho RT=\rho R\frac{e}{c_v}=\rho\frac{c_p-c_v}{c_v}e=(\kappa-1)\rho e=$$

$$=(\kappa-1)\left[E-\frac{1}{2}\rho\left(u^2+v^2\right)\right].$$

Here κ is the Poisson adiabatic constant, R is universal gas constant and c_p, c_v are the specific heats. The unknown quantities are included in the governing equations with their dimensionless values

$$p=\frac{p}{p_0}\qquad T=\frac{T}{T_0}\qquad u,v=\frac{u,v}{\sqrt{RT_0}}\qquad \rho=\frac{\rho}{p_0/RT_0}$$

where the subscript $(\ )_0$ denotes stagnation parameter.

Numerical solution.

The finite volume method is applied for solving of equations (1). For this purpose the physical space around the rigid body is discretized to quadrilateral cells (finite volumes - fig. 2). As explained in Ref. [1] the basic idea of the method is to satisfy the integral form of equations (1) for each control volume. It is necessary to find out total flux of vectors $\vec{f}(\vec{q})$ and $\vec{g}(\vec{q})$ on each face of the cell separately and so on along the entire grid. Then, as proposed in Ref. [2], for the present time step, the system (1) is discretized for the current cell like this:

$$
\begin{aligned}
&\frac{\partial}{\partial t}(J\rho)+\sum_{k=1}^{4}(U_k\rho_k)=0\\
&\frac{\partial}{\partial t}[J(\rho u)]+\sum_{k=1}^{4}\left[U_k(\rho u)_k+(-1)^k\Delta y_k p_k\right]=0\\
&\frac{\partial}{\partial t}[J(\rho v)]+\sum_{k=1}^{4}\left[U_k(\rho v)_k+(-1)^{k+1}\Delta x_k p_k\right]=0\\
&\frac{\partial}{\partial t}(JE)+\sum_{k=1}^{4}\left[U_k(E+p)_k\right]=0
\end{aligned}
\qquad (2)
$$

The J symbol means the cell area and

$$U_k=(-1)^k\Delta y_k u_k+(-1)^{k+1}\Delta x_k v_k$$

corresponds to a contravariant velocity component. The sign replacement in front of an individual additive is conformed to normal vector sign variation while shifting to the adjacent cell. Each quantity, in system (2), is evaluated as the average of the values in the cells on the two sides of the cell's face (fig. 1), for example:

$$U_1(\rho u)_1=\frac{1}{2}\left[U_{i,j}(\rho u)_{i,j}+U_{i,j-1}(\rho u)_{i,j-1}\right]$$

Thus the scheme is an analog of a central difference scheme on a Cartezian grid.

Time – depending derivative discretization scheme, for space-discretized equations

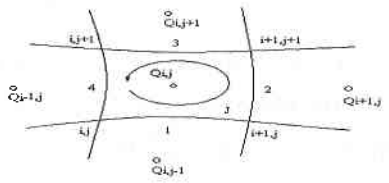


Fig. 1. Flux calculation sequence about cell i, j.

$$\frac{dq_i}{dt} + \frac{1}{J_i} Q(q_i) = 0,$$

$i = 1,2,3,...$

is 4th order Runge – Kutta

$$q^{(0)} = q^n$$

$$q^{(1)} = q^{(0)} - \frac{\Delta t}{2} Q\left(q^{(0)}\right)$$

$$q^{(2)} = q^{(0)} - \frac{\Delta t}{2} Q\left(q^{(1)}\right)$$

$$q^{(3)} = q^{(0)} - \frac{\Delta t}{2} Q\left(q^{(2)}\right)$$

$$q^{(4)} = q^{(0)} - \frac{\Delta t}{6} Q\left(q^{(0)}\right) - \frac{\Delta t}{3} Q\left(q^{(1)}\right) - \frac{\Delta t}{3} Q\left(q^{(2)}\right) - \frac{\Delta t}{6} Q\left(q^{(3)}\right)$$

$$q^{(n+1)} = q^{(4)}$$

Here the flux $Q(q_i)$ is computed over again at each time step where the following initial conditions are considered:

$$\kappa = 1.4 \quad \rho_0 = 1 \quad p_0 = 1 \quad a_0 = \sqrt{\frac{\kappa p_0}{\rho_0}}$$

$$u_0 = M_\infty a_0 \cos\alpha \quad v_0 = M_\infty a_0 \sin\alpha$$

Here symbol a denotes speed of sound, α means the angle of attack and M denotes Mach number.

To suppress non-physical oscillations of the results, according to Ref. [2], the last time step is augmented by addition of the filter:

$$q^{(n+1)} = q^{(4)} + \Delta t D_x^+ \mu_x D_x^- q^{(4)} + \Delta t D_y^+ \mu_y D_y^- q^{(4)}$$

In the expression above the superscripts + and – denoted forward and backward difference operators. The coefficients μ_x and μ_y are made proportional to

$$\mu_x \sim \frac{Q_{i+1,j} - 2Q_{i,j} + Q_{i-1,j}}{Q_{i+1,j} + 2Q_{i,j} - Q_{i-1,j}}$$

$$\mu_y \sim \frac{Q_{i,j+1} - 2Q_{i,j} + Q_{i,j-1}}{Q_{i,j+1} + 2Q_{i,j} - Q_{i,j-1}}$$

The boundary conditions, in accordance with placement, are reduced to the following kinds: along the rigid body contour and along the outer boundary of the physical domain. In regard to the first group, it is truth that

all fluxes across contour are identically zero (impermeable condition). Exception is numerical contribution to the momentum flux.

The second kind boundary conditions, computation of which is based upon one-dimensional characteristic method, is divided on inflow and outflow. Because wave propagation normal to the boundary is dominant, variations parallel to the boundary may be neglected and the linearized one-dimensional Euler equations can be written as:

$$\frac{\partial U}{\partial t} + A \frac{\partial U}{\partial x} = 0$$

$$U = \begin{Vmatrix} \rho \\ u \\ v \\ p \end{Vmatrix} \qquad A = \begin{Vmatrix} u & \rho & 0 & 0 \\ 0 & u & 0 & \frac{1}{\rho} \\ 0 & 0 & u & 0 \\ 0 & \rho a^2 & 0 & u \end{Vmatrix}_{ref}$$

The reference state for evaluating the matrix A will be the state on boundary at the old time step. To make the matrix A constant, the average value of the state vector on the boundary will be used to evaluate A .

For subsonic inflow, there are three incoming waves and one outgoing wave. The last one is computed by the algorithm using quantity conservation of corresponding Riemann invariant among the same wave while for the incoming waves the prescribing quantities for nozzle inflow (total enthalpy, entropy and flow angle) are used:

$$H = \frac{\kappa}{\kappa - 1}\frac{p}{\rho} + \frac{1}{2}\left(u^2 + v^2\right)$$

$$s = \ln\left(p\right) - \kappa \ln\left(\rho\right)$$

$$\alpha = arctg \frac{v}{u}$$

For subsonic outflow the linearized analysis shows that there are three outgoing waves and one incoming wave. In such case it is classic to prescribe the static pressure at outflow nodes.

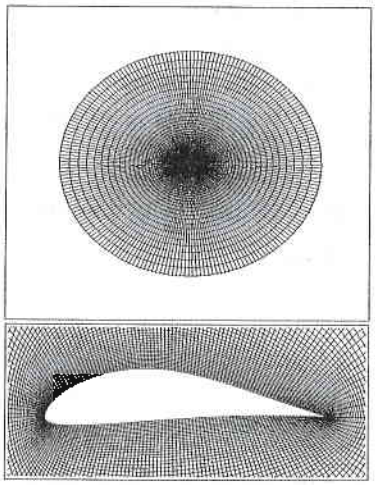


Fig. 2. An "O"-type computational grid (above) and its expanded center (below) surrounding Eppler387 wing section

Computational grid.

As was mentioned above, the physical domain surrounding wing section is discretized into finite cells. In the

current case of study this fact is achieved by curvilinear elliptic coordinate system generation by solving system Laplace equations. As mentioned in Ref. [3] the iterative act of synthesis uses initial coordinates of the grid nodes, which are evaluated previously by interpolating the interior of the physical domain. The grid coordinates at the boundaries of the field import the boundary conditions. The interpolation used is one-dimensional (normally to boundary curves) and includes hyperbolic tangent function. In this way the derived initial "ordinate" axis allocation corresponds well to the Laplace operator "effect of smoothness": the closing in of the "abscissas" curves to the convex boundary. The construction of the interpolation, as proposed in Ref. [4], is made as follows: let arc length s varies from 0 to 1 and arc's points number ξ varies from 0 to I in such way so $s(0) = 0; s(I) = 1$. Initially assigning values of the first derivatives

$$\frac{\partial}{\partial \xi} s(\xi = 0) = \Delta s_1$$

$$\frac{\partial}{\partial \xi} s(\xi = I) = \Delta s_2$$

the following expressions are defined:

$$A = \sqrt{\frac{\Delta s_2}{\Delta s_1}}$$

$$B = \frac{1}{I\sqrt{\Delta s_1 \Delta s_2}}$$

$$B = \frac{sh\ \delta}{\delta}$$

Then these equations are valid:

$$u_1(\xi) = \frac{1}{2}\left\{1 + \frac{th\left[\delta\left(\frac{\xi}{I} - \frac{1}{2}\right)\right]}{th\ \frac{\delta}{2}}\right\}$$

$$s(\xi) = \frac{u_1(\xi)}{A + (1 - A)u_1(\xi)}$$

If the equations above are applied to curve for which $\vec{r}(\xi) \in [\vec{r}(0); \vec{r}(I)]$ then for the curve point distribution follows:

$$\vec{r}(\xi) = \vec{r}(0) + [\vec{r}(I) - \vec{r}(0)]s(\xi), \xi = 0,1,2,3,\ldots, I$$

The derived in the described manner initial grid quantities are "smoothed" by solving the Laplace equations about $\vec{r}$

$$\Delta\xi = 0$$

$$\Delta\eta = 0$$

which are previously transformed into curvulinear basis (Ref. [3]). The new system equations looks like:

$$g_{22}r_{\xi\xi} + g_{11}r_{\eta\eta} - 2g_{12}r_{\xi\eta} = 0$$

$$g_{11} = x_\xi^2 + y_\xi^2$$

$$g_{22} = x_\eta^2 + y_\eta^2$$

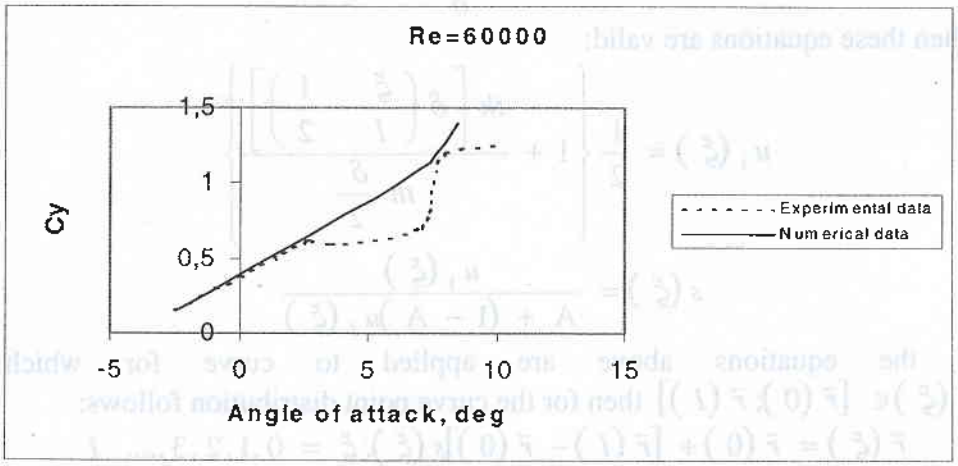
$$g_{12} = x_\xi x_\eta + y_\xi y_\eta$$

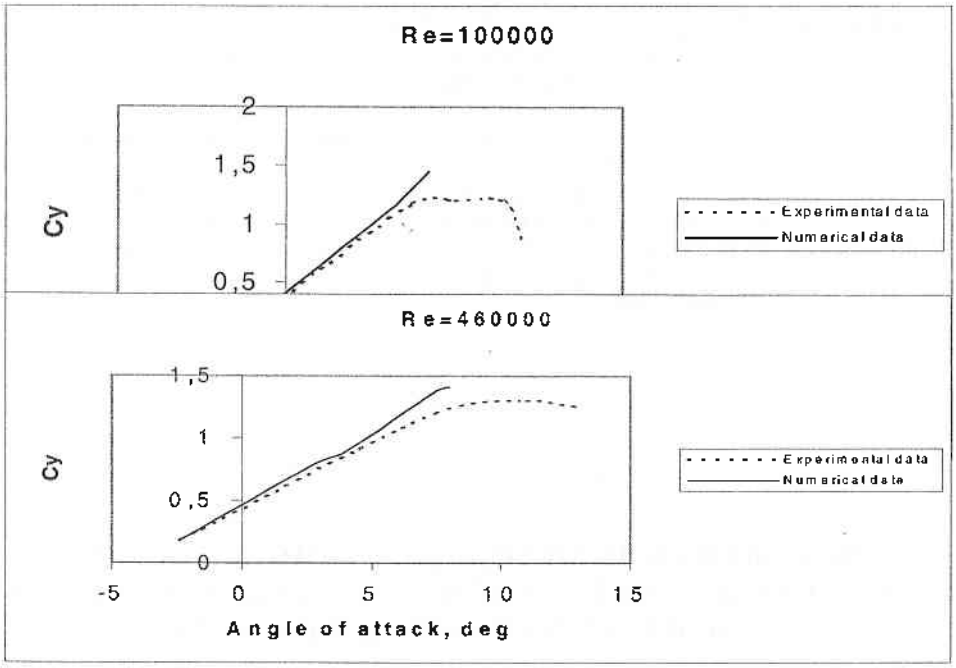
It is solved, after aproximation with finite defferences, by successive overrelaxation method with corresponding coefficient $\omega = 1.8$.

Results.

An "O" - type elliptic grid has been generated with 200 points along tangent and 100 points along normal directions of Eppler 387 wing section. Thus the physical domain around the foil has been discretized with 20000 cells (fig. 2).

With the developed numerical algorithm serial calculations has been implemented with various Reynolds numbers and angles of attack. The results, verified with experimental data (published in Ref. [5]), are shown below. There are not a good coincidence with the experiment at the Reynolds numbers Re~60000. The reason of this is inability of the algorithm to predict transient flows.





Conclusion.

A finite volume method has been used to solve partial differential equations of Euler, describing a motion of an ideal gas in two-dimensional space. A steady-state solution has been achieved using a 4^{th} order Runge – Kutta scheme. The numerical oscillations of solution were suppressed by augmenting artificial viscosity. The boundary conditions were derived using method of characteristics. An elliptic grid has been generated while solving a system of Laplace equations. The ability of the algorithm has been demonstrated to simulate a subsonic flow over a wing section.

References.

1. Lomax, H., T. Pulliam, D. Zingg, *Fundamentals of Computational Fluid Dynamics*, NASA Ames Research Center, 1998
2. Jameson, A., W. Schmidt, E. Turkel, Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge Kutta Time Stepping Schemes, *AIAA Paper 81-1259*, 1981, AIAA 14th Fluid and Plasma Dynamic Conference, Palo Alto, June 1981
3. Thompson, J., Z. Warsi, C. Mastin, *Grid Generation Techniques*, Mississippi State University, 1985
4. Vinokur, Marcel, On One – Dimensional Stretching Functions for Finite – Difference Calculations, *Journal of Computational Physics*, 1983, No 50, p. 215
5. Mc Ghee, R., B. Walker, B. Millard, Experimental Results for the Eppler 387 Airfoil at Low Reynolds Number in the Langley Low – Turbulence Pressure Tunnel, *NASA TM-4062*, 1988

ПРИЛОЖЕНИЕ НА МЕТОДА НА КРАЙНИТЕ ОБЕМИ И СХЕМА РУНГЕ - КУТА ЗА АЕРОДИНАМИЧЕН АНАЛИЗ НА ОБТИЧАНЕТО НА КРИЛЕН ПРОФИЛ

Константин Методиев

Резюме

В работата е използван метода на крайните обеми за дискретизация на уравненията на Ойлер, описващи движението на идеален газ върху структурна елиптична координатна система. Използвана е също схема Рунге–Кута от 4-ти ред за последващо интегриране по време. Добавен е нелинеен изкуствен вискозитет за да се подтиснат нефизически числени осцилации на решението. Определянето на граничните условия е базирано на метода на характеристиките. Резултатите от приложението на алгоритъма върху случай на дозвуково обтичане на двумерно тяло са сверени с експериментални данни.