ZÁPADOČESKÁ UNIVERZITA V PLZNI

FAKULTA STROJNÍ

KATEDRA ENERGETICKÝCH STROJŮ A ZAŘÍZENÍ

ZÁPADOČESKÁ UNIVERZITA V PLZNI

POWER SYSTEM ENGINEERING, THERMODYNAMICS & FLUID FLOW

2012

JEDNOTLIVÝ PŘÍSPĚVEK ZE SBORNÍKU

evropský sociální fond v ČR

EVROPSKÁ UNIE

MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY

OP Vzdělávání pro konkurenceschopnost

INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

NUMERICAL SOLUTION OF COMPRESSIBLE STEADY FLOWS AROUND THE NACA 0012 AIRFOIL

KRYŠTŮFEK Pavel, KOZEL Karel

The article presents results of a numerical solution of subsonic and transonic flows described by the system of Euler equations in 2D flows around the NACA 0012 airfoil. Authors used Runge-Kutta method to numerically solve the flows around the NACA 0012 airfoil.

Keywords: numerical, solution, NACA 0012 airfoil, Euler, Runge-Kutta

1. Introduction

A numerical code has been developed for simulating transonic flow field around the NACA 0012 airfoil. In these simulations the meshes type C has been used, which was created by mesh generator and has been described in [4]. In this case mesh has been created for numerical solution over profile NACA 0012.

2. Mathematical models

The 2D flow of an inviscid compressible fluid is described by the system of Euler equations.

$$W_t + F(W)_x + G(W)_y = 0 \tag{1}$$

where

$$W = \begin{bmatrix} \rho \\ \rho w_1 \\ \rho w_2 \\ e \end{bmatrix},\; F(W) = \begin{bmatrix} \rho w_1 \\ \rho w_1^2 + p \\ \rho w_1 w_2 \\ (e+p) w_1 \end{bmatrix},\; G(W) = \begin{bmatrix} \rho w_2 \\ \rho w_1 w_2 \\ \rho w_2^2 + p \\ (e+p) w_2 \end{bmatrix}, \tag{2}$$

$$p = (\kappa - 1)\left[e - \frac{1}{2}\rho\left(w_1^2 + w_2^2\right)\right]. \tag{3}$$

In the above equations, W is conservative variable, F, G are function of inviscid physical fluxes, ρ denotes density, w_1, w_2 are components of velocity in the direction of axis x, y, p is pressure, e is total energy per unit volume. The parameter $\kappa = 1.4$ is the adiabatic exponent.

3. Specification of test case

We selected for numerical solution a structured mesh formed by quadrilateral finite volumes. The grid around profiles (wing) usually consists of a C-grid in the flow direction. In the case of the C-topology the aerodynamics body is enclosed by one family grid lines, which also form the wake region. The situation is sketched in Fig. 1.

The initial grid is generated algebraically by using the linear **TFI method** [8 - pg. 382], [1]. Afterwards, **elliptic PDE's** are employed to produce boundary-orthogonal grid with specific wall spacing. The **NACA0012** airfoil contour is approximated by a **Bezier spline** [8 - pg. 384], [5].

The grid with 512x64 elements created by ours program for this simulation were presented in Fig. 2.

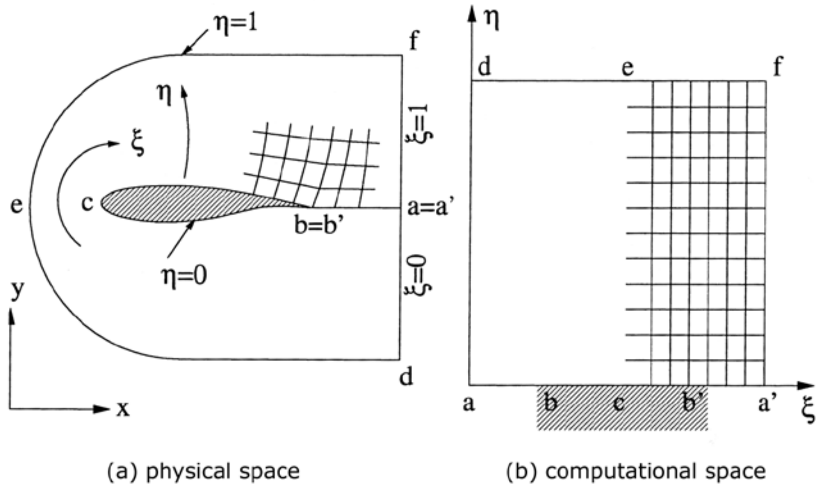

Fig. 1: C-grid topology in 2D (by Blazek [1])

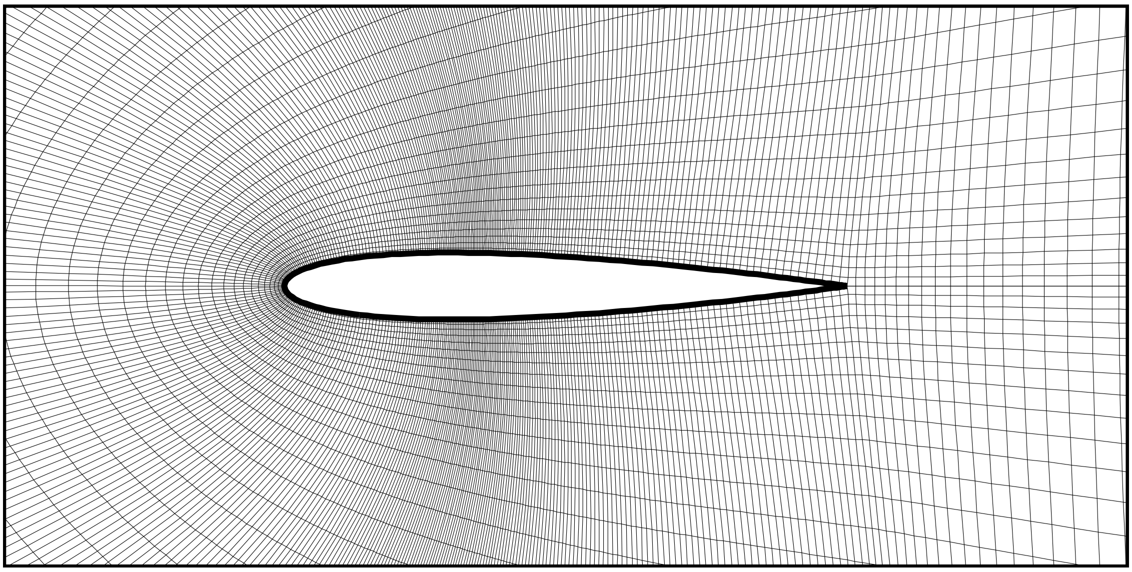

Fig. 2: Inviscid, C-type, NACA 0012 airfoil, 512x64 elements (by Kryštůfek [4])

4. Numerical method

For modeling of the mentioned flow case, numerical scheme of the finite volume method Runge-Kutta (**RK**) is used on non-orthogonal structured grids of quadrilateral cells D_{ij}.

$$W_{i,j}^{(0)} = W_{i,j}^{n}, \tag{4}$$

$$W_{i,j}^{(r+1)} = W_{i,j}^{(0)} - \Delta t \alpha_r \operatorname{Re} z W_{i,j}^{(r)} + AD\left(W_{i,j}^{(r)}\right), \tag{5}$$

$$W_{i,j}^{n+1} = W_{i,j}^{(m)}, \tag{6}$$

$$\operatorname{Re} z W_{i,j}^{n} = \frac{1}{\mu_{i,j}} \sum_{k=1}^{4} \left(F_k^n \Delta y_k - G_k^n \Delta x_k \right), \tag{7}$$

for

$$r = 0,1,\ldots,m-1;\; m = 5\,, \tag{8}$$

$$\alpha_1 = 0,25; \alpha_2 = 0,1667; \alpha_3 = 0,375; \alpha_4 = 0,5; \alpha_5 = 1\,. \tag{9}$$

The Jameson's artificial dissipation AD damps undesirable oscillations and improves the stability of the method. The convergence to the steady state is followed by log L2 residual defined by

$$\mathrm{Re}\,zW_{i,j}^{n} = \frac{1}{N}\sqrt{\sum_{i,j}\left(\frac{W_{i,j}^{n+1} - W_{i,j}^{n}}{\Delta t}\right)^{2}} \tag{12}$$

where N is a number of all elements in the computational domain.

Three characteristic variables are prescribed based on the free stream values at far field subsonic inflow. One characteristic variable is extrapolated from interior of the physical domain. In the case of subsonic outflow, three flow variables (density and the two velocity components) have to be extrapolated from interior of the physical domain. The remaining four variables (pressure) must be specified externally (as a multiple of the input pressure). On wall zero derivatives of velocity vector along normal is considered.

5. Numerical results

For a 2D numerical simulation of flows of an inviscid compressible fluid around NACA 0012 airfoil, the authors applied **RK** numerical schemes on a structured grid with **512 x 64** cells. For free stream value $Ma = 0,8$ and angle of attack $\alpha = 1,25$ results are shown on Fig 3. In the case of angle of attack $\alpha = 1,25$ results (Fig. 3) are in very good agreement with the results from the author Fürst, 2004 [3] (Fig. 4) with computational grid 168 x 40 cells. Positions of the shock wave are the same.

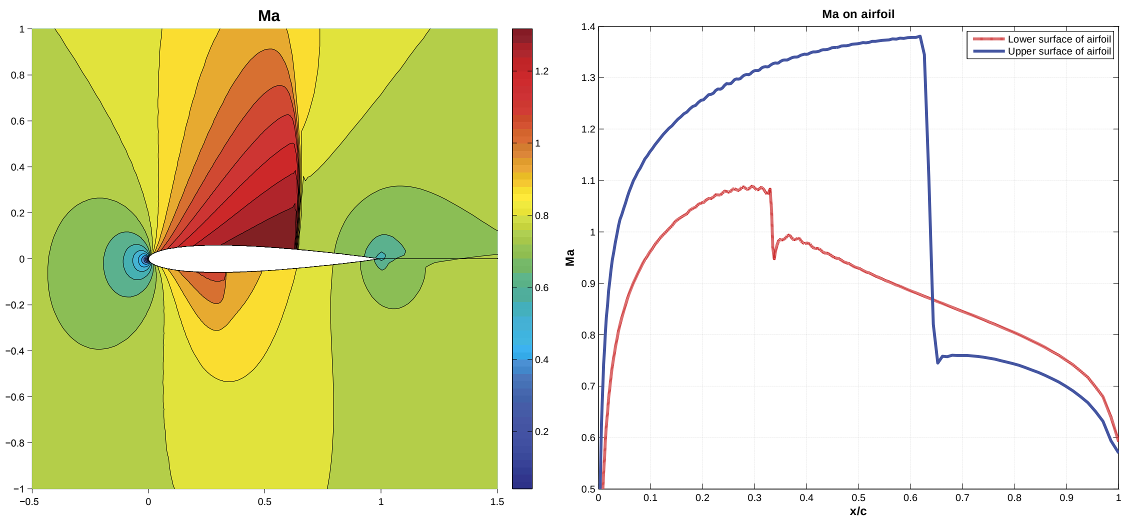

Fig. 3: Inviscid compressible flow around the NACA 0012 airfoil: Mach number isolines (left) and Mach number on bottom and upper wall (right), angle of attack $\alpha = 1,25$.

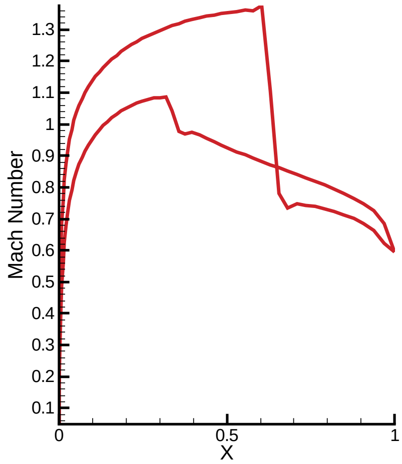

Fig. 4: Mach number distribution for flow around NACA 0012 airfoil: Mach number $Ma_{\infty} = 0{,}8$ and angle of attack $\alpha = 1{,}25$ - Fürst, 2004 [3]

6. Conclusion

Numerical solution has been applied on structured meshes with 512x64 elements. The Runge-Kutta sheme has been used for the NACA 0012 airfoil. Then results were compared with other authors [3]. The mesh generator of the type C for profile with a blunt leading edge has been tested. In the future, will continue with a profile RAE 2822 and the Navier-Stokes equations.

7. References

[1] Blazek, J. – Computational Fluid Dynamics: Principles and Applications, Elsevier, 2005, ISBN: 0-080-44506-3

[2] Dvořák, R. – Kozel, K.: Matematické modelování v aerodynamice, ČVUT, Praha, 1996, ISBN: 80-01-01541-6

[3] Fürst, J.: Numerical solution of inviscid and viscous flows using modern schmes and quadrilateral or triangular mesh, Proceedings of the Czech–Japanese Seminar in Applied Mathematics 2004, Czech Technical University in Prague, August 4-7, 2004

[4] Kryštůfek, P. – Kozel, K.: Numerical solution of compressible steady flows in a 2D channel and DCA 18% profile, Experimental fluid mechanics 2011, Czech Technical University of Liberec, Jičín, pp. 771-777, 2007, ISBN: 978-80-7372-784-0

[5] Sommerfeld, A.J.W.: Partial Differential Equations in Physics, Academie, New York, 1949

Acknowledgments

The work was by the project CZ.1.07/2.2.00/07.0234.

Ing. Kryštůfek Pavel, Czech Technical University of Liberec, Faculty of Mechanical Engineering, Dept. of Power Engineering Equipment, Studentská 2, 461 17 Liberec 1, pavel.krystufek@tul.cz

prof. RNDr. KOZEL Karel DrSc., Czech Technical University in Prague, Faculty of Mechanical Engineering, Karlovo náměstí 13, 121 35 Prague 2; karel.kozel@fs.cvut.