V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal,14-17 June 2010

ALE METHOD FOR UNSTEADY FLOW COMPUTATIONS

Petr Furmánek*, Jiří Fürst† and Karel Kozel†

*Aeronautical Research and Test Institute,
Beranových 130, 199 05 Praha - Letňany
petr.furmanek@gmail.com
†Czech Technical University in Prague
Faculty of Mechanical engineering
Department of Technical Mathematics
Karlovo náměstí 13, 121 35 - Praha 2
jiri.furst@fs.cvut.cz, karel.kozel@fs.cvut.cz

Key words: ALE, FVM, Unsteady Flow, Turbulence Modelling, TVD, WLSQR, SST

Abstract. *The aim of this work is to summarize results of numerical simulations of steady and unsteady transonic flow obtained by two different modern finite volume schemes in combination with Arbitrary Lagrangian-Eulerian method (computation on moving meshes). The simulations were carried out both in 2D and 3D and the unsteady effects were presented by forced oscillations of the profile/wing around given reference point/axis. Implemented schemes were the so called Modified Causon's scheme (based on TVD form of classical MacCormack scheme) and implicit WLSQR scheme (based on the WENO approach) combined with AUSMPW+ numerical flux in 2D and HLLC flux in 3D. As a 2D test case both inviscid and turbulent flow around the NACA 0012 profile wing have been simulated and the numerical results have been compared with experimental data. Both schemes were extended also for the 3D steady computations and tested on the transonic flow around the ONERA M6 wing. The computational area was discretized with two different types of finite volume meshes (H and C type). Comparison of the numerical results (both in-between and with experimental data) is satisfactory. The Modified Causon's scheme in 3D form was also adapted for unsteady computation with the use of ALE method and was tested on inviscid transonic flow around the ONERA M6 wing (forced oscillation around a given axis). Experimental data for this case are unfortunately not available. However, the numerical results show all the characteristics as expected.*

1 INTRODUCTION

The unsteady effects play very important role in the aircraft industry (as well as in many other technical disciplines) and have a huge impact on the flow-field (sometimes even with fatal consequences, e.g. flutter). Investigation of unsteady flows may be done generally in two ways. Either by experimental measurements or by numerical simulations. One of possible approaches is the Arbitrary Lagrangian-Eulerian method 3, which combines the Lagrangian and Eulerian way of moving fluid investigation, i.e. both the fluid and its reference frame move. The motion is in our case presented by prescribed oscillations of profile/wing around the reference point/axis. The chosen schemes were tested with a very good results for a number of steady test cases before used for numerical solution of unsteady flow.

2 NUMERICAL METHODS

Turbulent flow of the compressible fluid is described by the set of Reynolds-averaged Navier-Stokes equations[1]. In the case of its inviscid simplification, the set of Euler equations is used. Numerical solution of the chosen problems was realized with use of the finite volume method. Both 2D and 3D problems were simulated using inviscid and turbulent computation in order to obtain better understanding of the effectivity of the chosen turbulence models. In the case of 2D unsteady flow the Kok's TNT turbulent model, which usually serves well for steady flow simulations, was chosen. For 3D steady flow, two different turbulent models were employed and compared as in-between as with inviscid results and experimental data. These were namely the Spalart-Allmaras[12] and SST models[13]. The numerical methods developed by the authors were following:

1. **Modified Causon's Scheme**. This scheme is derived from the classical explicit MacCormack predictor-corrector scheme in TVD form, which is able to achieve very good results. However, it also entails disadvantageous demands for both computational memory and power. Therefore a simplification saving approximately 30% of computational time was proposed by Causon[11] by introducing a special type of pressure-gradient dependent self-controling artificial dissipation. This new scheme was still TVD, but the influence of artificial dissipation turned out to be too strong. The authors on the other hand proposed another modification based on Causon's scheme (refered to as the Modified Causon's scheme), which is no more TVD, but keeps the advantages of the Causon's scheme while clearing out its drawbacks in the same time.

2. **WLSQR scheme**[4] (Weighted Least-Square Reconstruction scheme), which is based on the WENO approach[14]. The interpolating polynomial is hereby obtained by the least square method. Convective fluxes through the cell interface are approximated by either the AUSMPW+ numerical flux[5] (2D flow) and by the HLLC[6] flux (3D flow). The high order accuracy in time is obtained in a standard way by using the interpolated values at the cell faces. The interpolation is obtained by using the weighted least-square approach, which usually gives better convergence to steady state than the methods with Barth's

limiter. Advancing in time is realized by the non-linear implicit dual-time backward Euler method. Resulting sparse system of linear equations is solved by GMRES with ILU(0) preconditioning. Dimension of the Krylov subspace is chosen between 10–40 and maximum number of iteration is set to 10–50. If the steady solution is not found in prescribed number of iterations the computation proceeds in the next time step.

The unsteady effects were simulated by the Arbitrary Lagrangian-Eulerian[3] method using moving meshes.

3 2D UNSTEADY FLOW AROUND THE NACA 0012 PROFILE

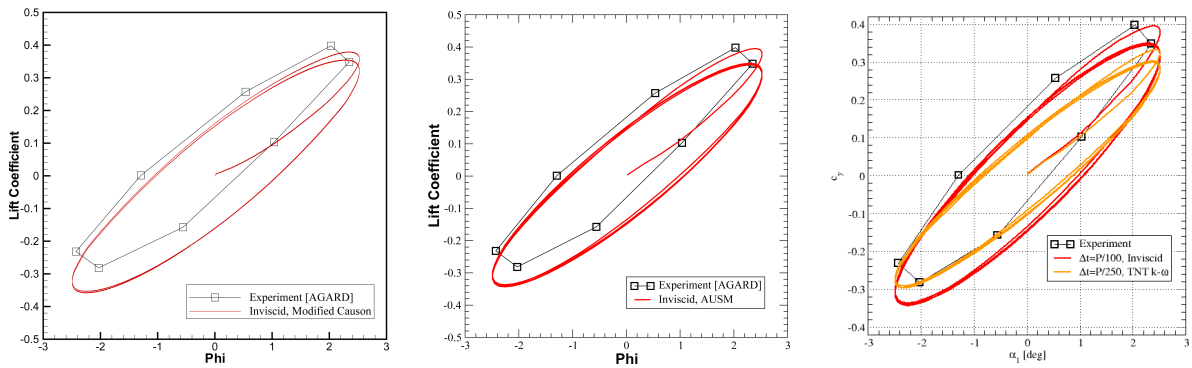
A standard test case described in AGARD Advisory Report No. 702[2] was chosen for the unsteady flow simulation. It is transonic unsteady flow over the NACA 0012 characterized by the inlet Mach number $M_\infty = 0.755$. The oscillatory motion of the profile around the reference point $x_{ref} = [0.25, 0.00]$ is given by the pitching angle $\alpha_1(t) = 0.016^\circ + 2.51^\circ \sin(\omega t)$. The angular velocity is defined as $\omega = \frac{2kU_\infty}{c}$, where U_∞ is the free-stream velocity (since the non-dimensional form of Navier-Stokes equations is considered and angle of attack $\alpha = 0^\circ$ then $U_\infty = M_\infty$), $c = 1$ is the chord length and the reduced frequency $k = 0.0814$. The unsteady state development was observed on the behaviour of the lift coefficient (c_l) given as

$$c_l = \frac{\oint_{\Gamma_{prof}} p\, dx}{\frac{1}{2} U_\infty^2 \rho_\infty},$$

where $\rho_\infty = 1$ and Γ_{prof} is the curve defining the profile. The used computational schemes and meshes were

- Modified Causon's scheme - structured C-mesh with 15096 elements (124 cells around profile),
- WLSQR scheme with AUSMPW+ flux - unstructured mesh with 6720 quadrilateral cells (120 cells around profile). For the turbulent flow simulation the Kok's TNT turbulence model was used.

As can be seen from figures 1 and 2 the numerical results obtained by both schemes in the case of inviscid flow are very good. For the c_l comparison the results correspond qualitatively, but experimental data show a bit higher c_l values (Fig. 1). Considering symmetry of the problem, also the behaviour of the c_l should be symmetric with the center of symmetry in the point $[0,0]$. The experimental data however do not have this characteristic and therefore the suspicion of their systematical error comes in mind. Important characteristics, e.g. the position and intensity of the shock wave (minimal and maximal reached value of c_p), are however in a very good correspondence, which is unfortunately not the case of the turbulent computation, where both the c_p and c_l coefficient differ significantly (Fig 3). It is therefore necessary to use another turbulence model (EARSM) or large eddy simulation (LES).

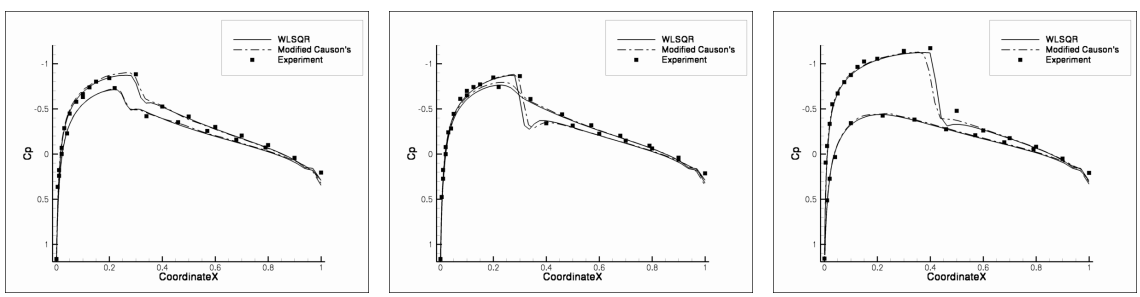


(a) Modified Causon's scheme, inviscid computation.

(b) WLSQR scheme, AUSMPW+, inviscid computation.

(c) WLSQR scheme, AUSMPW+, comparison of inviscid (line) and turbulent(dashed) computation.

Figure 1: NACA 0012, lift coefficient behaviour, comparison of numerical (black line) and experimental (colored lines) results.



$\alpha = 1.09°$ $\alpha = -1.25°$ $\alpha = 2.34°$

Figure 2: c_p coefficient during the 5th period of forced oscillatory motion, inviscid flow, comparison of experimental (dots) and numerical (lines) results (Modified Causon's scheme, WLSQR scheme with AUSMPW+ flux).

4 3D FLOW OVER THE ONERA M6 WING

4.1 Steady Flow

Another standard test case (mentioned in AGARD AR 138[7]) was chosen for 3D computation. The transonic flow over the ONERA M6 wing is in this case characterizes byt the inlet Mach number $M_\infty = 0.8395$, angle of attack $\alpha_0 = 3.06°$ and Reynolds number $Re = 11.72 \times 10^6$. Both inviscid and turbulent flow were simulated in this case, using the above mentioned schemes (Modified Causon's and WLSQR). The inviscid computation was carried out using the following computational meshes

- structured C-mesh with 467313 hexahedral elements (Modified Causon's scheme),
- unstructured mesh with 306843 pyramidal elements (WLSQR scheme with HLLC flux).

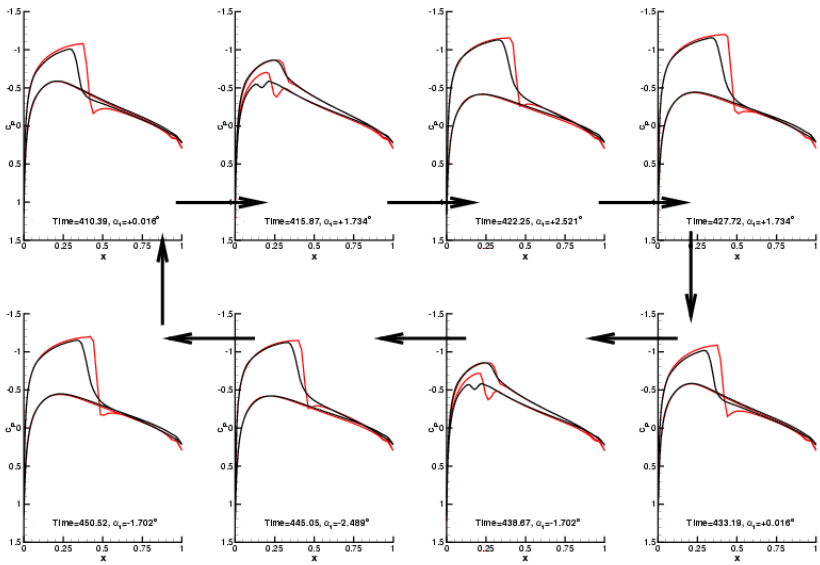


Figure 3: c_p coefficient behaviour during the 5th period of forced oscillatory motion, WLSQR scheme, comparison of inviscid (dashed) and turbulent (line) model (Kok's TNT).

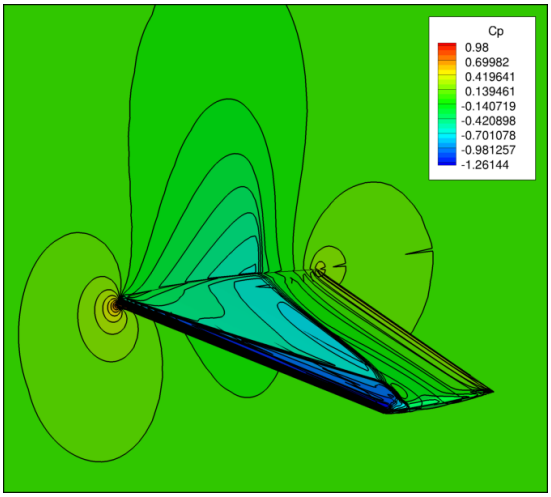
Considering the turbulent simulation needs, the unstructured mesh was condensed by layers of prismatic elements in the close proximity of the wing in order to capture flow behaviour in the boundary layer. The turbulent effects were modelled by

- the Spalart-Allmaras one-equation model[12],
- the SST $k-\omega$ model[13].

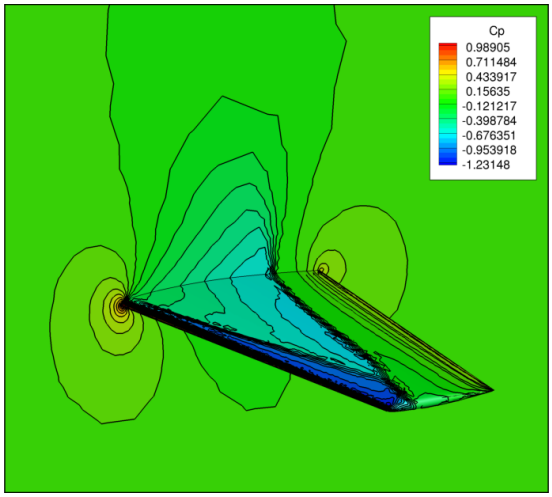
As can be seen in figures 4 and 5, considering both inviscid and turbulent computations, all the schemes give very good results. Differences between experimental and numerical data are naturally greater in the case of inviscid computation, but even so the correspondence is more than satisfactory. The turbulent results are closer to the real flow, especially in the regions with strong shock-waves, but the difference between SST and Spalart-Allmaras models are nearly negligible (the SST model seams to be slightly better). The greatest difference can be odserved in the 80% cut, region, where two shock-waves interact. This interaction is a very complex phenomena and difficult to capture.

4.2 Unsteady Flow

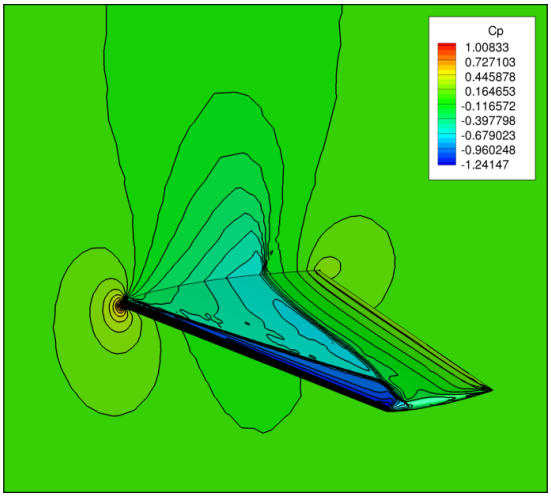
The initial conditions for 3D unsteady inviscid transonic flow were taken from the standard test case mentioned in 7 (same as steady case in previous section). The forced oscillatory motion of the wing around the elastic axis parallel with the axis z and going through the reference point $x_{ref} = [\frac{1}{3}; 0.00; 0.00]$ was given by the same relation for pitching angle as in 2D. The inlet Mach number was considered $M_\infty = 0.8395$, initial deviation $\alpha_0 = 3.06°$, amplitude $\alpha_1 = 1.5°$



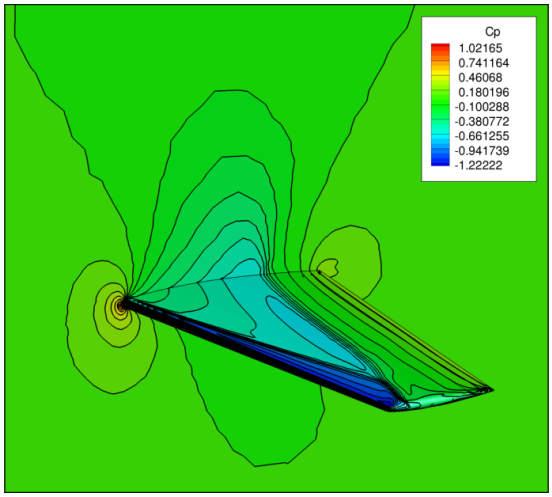
a) Modified Causon's scheme, inviscid



b) WLSQR, HLLC flux, inviscid



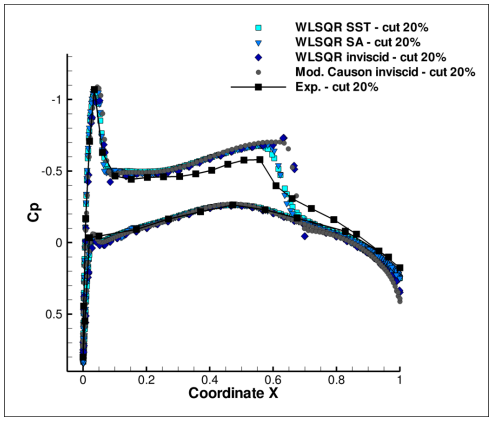
c) WLSQR, HLLC flux, Spalart-Allmaras



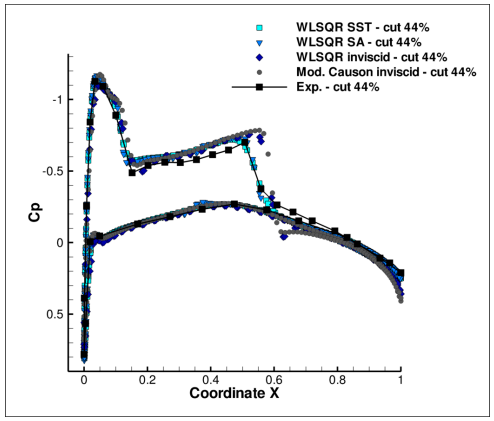
d) WLSQR, HLLC flux, SST

Figure 4: c_p coefficient isolines, ONERA M6, comparison of the numerical results.

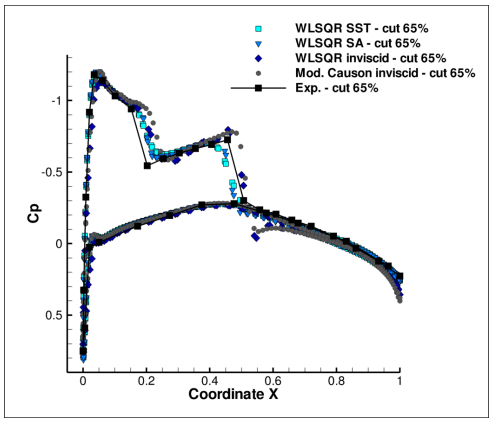
and frequency $f = 10$Hz. The structured computational mesh had 467313 elements. Computation was carried out using the the Modified Causon's scheme, which has proved well - the results (Fig. 6) show that fully periodic state has been achieved at least during the 5th period of oscillatory motion. Pressure coefficient decreases with increasing angle of attack (and vice versa) and the scheme does not produce spurious oscillations. Comparison with experimental data is unfortunately not yet available, but work is in progress at the present time on implementation of wing geometry used in experiments with oscillating wing at the Aeronautical Research and Test Institute in Prague (VZLÚ a. s.).



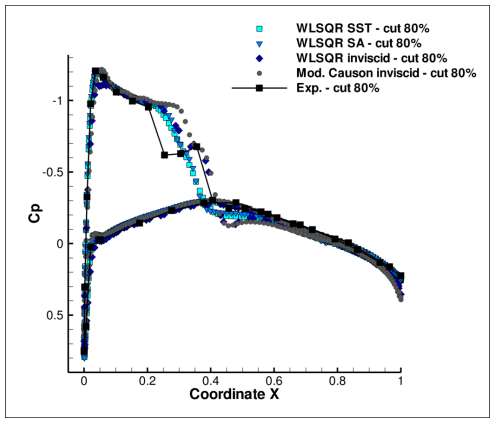
a) 20%



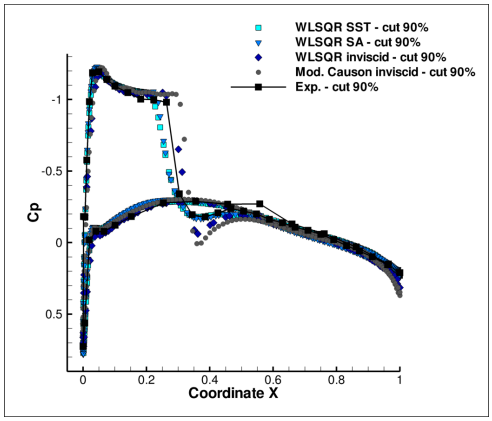
b) 44%



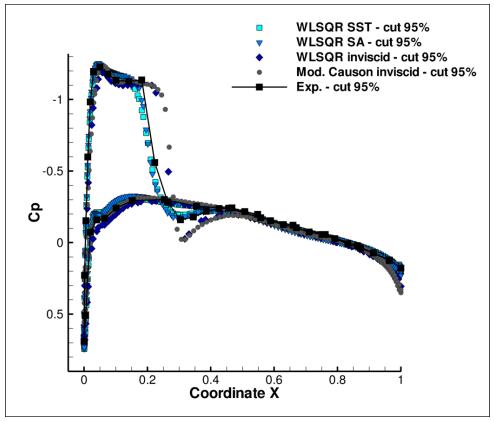
c) 65%



d) 80%

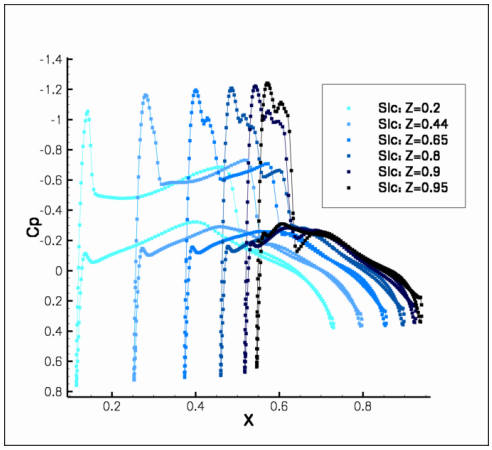


e) 90%

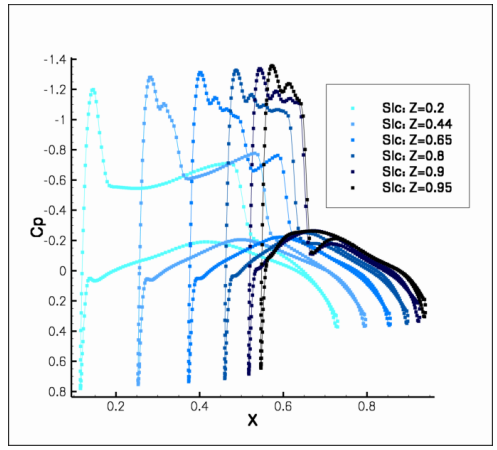


f) 95%

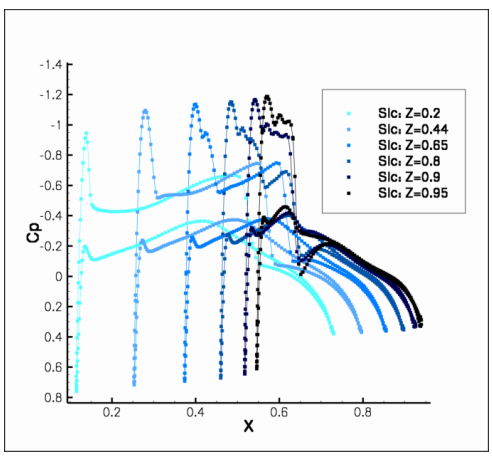
Figure 5: c_p coefficient in cuts alongside the wing, ONERA M6, comparison of experimental and numerical results.



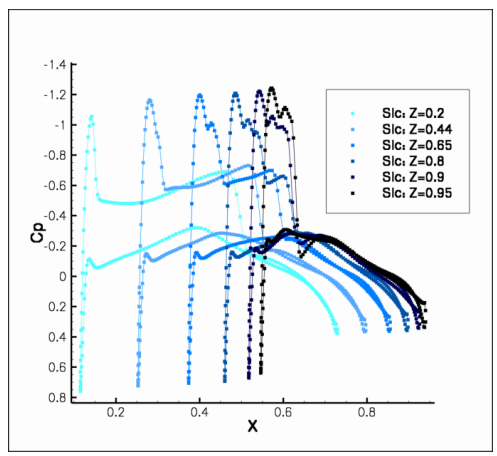
a) $\omega t = 0\pi + 10\pi$, $\alpha = 3.06°$



b) $\omega t = \frac{1}{2}\pi + 10\pi$, $\alpha = 4.56°$



c) $\omega t = \frac{3}{2}\pi + 10\pi$, $\alpha = 1.56°$



d) $\omega t = 2\pi + 10\pi$, $\alpha = 3.06°$

Figure 6: ONERA M6, c_p coefficient behaviour during the 5th period of forced oscillatory motion.

5 CONCLUSIONS

- Proposed FVM schemes for numerical solution of both unsteady 2D and 3D and steady 3D transonic inviscid flow show very good accuracy and efficiency.
- The schemes were able to capture important flow characteristics as the position and intensity of the schockwaves even in the case of inviscid flow.
- The future steps intended are implementation of implicit version of Modified Causon's scheme and extension of mentioned schemes for aeroelastic problems.

ACKNOWLEDGEMENT

This work was partially supported by the Research Plan VZ MSM 6840770010, VZ MSM 0001066902 and grant GACR 201/08/0012.

REFERENCES

[1] R. J. LeVeque, Numerical Methods for Conservation Laws, Basel, 1990, ISBN 3-7643-2464-3.

[2] Compendium of unsteady aerodynamic measurements. AGARD Advisory Report No.702, 1982.

[3] J. Donea: An arbitrary Lagrangian-Eulerian finite element method for transient fluid- structur interactions. Comput. Methods Apll. Mech. Eng., (1982), 33:689-723.

[4] J. Fürst: A weighted least square scheme for compressible fows. Submitted to “Flow, Turbulence and Combustion”, (2005).

[5] Kyu Hong Kim, Chongam Kim, and Oh-Hyun Rho: Methods for the accurate computations of hypersonic fows I. AUSMPW+ scheme. Journal of Computational Physics(2001),(174):38-80.

[6] Batten, P., Leschziner, M. A., Goldberg, U. C. Average-State Jacobians and Implicit Methods for Compressible Viscous and Turbulent Flows, Journal of computational physics 137, 1997

[7] Schmitt, V. and F. Charpin: Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers. Experimental Data Base for Computer Program Assessment. Report of the Fluid Dynamics Panel Working Group 04, AGARD AR 138, May 1979.

[8] J. Fürst: Numerical Solution of Transonic Flow Using Modern Schemes of Finite volume Method and Finite Differences, Dissertation thesis (in Czech), ČVUT, Praha, 2001.

[9] J. Fürst, K. Kozel: Application of Second Order TVD and ENO Schemes in Internal Aerodynamics. Journal of Scientific Computing, Vol. 17, No 1-4, December 2002, ISSN 0885-7474, pp. 263-272.

[10] Lesoinne, M., Farhat, C., Geometric conservation laws for flow problems with moving boundaries and deformable meshes, and their impact on aeroelastic computations, Computer Methods in Applied Mechanics and Engineering 134 (1996) 71-90.

[11] Causon, D. M., High resolution fnite volume schemes and computational aerodynamics. In Josef Ballmann and Rolf Jeltsch, editors, Nonlinear Hyperbolic Equations - Theory, Computation Methods and Applications, volume 24 of Notes on Numerical Fluid Mechanics, pages 63-74, Braunschweig, March 1989. Vieweg.

[12] Spalart, P. R. and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows", AIAA Paper 92-0439.(1992)

[13] Menter, F. R., Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA Journal, vol. 32, pp. 269-289.(1994)

[14] Friedrich O: Weighted essentially non-oscillatory schemes for the interpolation of mean values on unstructured grids, Journal of Camputational Physics 144 (1): 194-212 JUL 20 1998