Numerical simulation of 3D transonic inviscid flow over a swept wing

Furmánek, Petr
Department of Technical Mathematics, Faculty of Mechanical Engineering, Czech Technical University

Dobeš, Jiří
Department of Technical Mathematics, Faculty of Mechanical Engineering, Czech Technical University

Fürst, Jiří
Department of Technical Mathematics, Faculty of Mechanical Engineering, Czech Technical University

Fořt, Jaroslav
Department of Technical Mathematics, Faculty of Mechanical Engineering, Czech Technical University

他

出版情報：COE Lecture Note. 14, pp.31-39, 2009-02-12. Faculty of Mathematics, Kyushu University
バージョン：
権利関係：

Proceedings of the Czech-Japanese Seminar in Applied Mathematics 2008
Takachiho / University of Miyazaki, Miyazaki, Japan, September 1-7, 2008
pp. 31-39

NUMERICAL SIMULATION OF 3D TRANSONIC INVISCID FLOW OVER A SWEPT WING

PETR FURMÁNEK[1], JIŘÍ DOBEŠ[1], JIŘÍ FÜRST[1], JAROSLAV FOŘT[1],
MILAN KLADRUBSKÝ[2], KAREL KOZEL[1]

Abstract. The aim of this work is to investigate some of the phenomena occuring in the numerical solution of transonic flows in outer aerodynamics. A comparison of various computational schemes based on the Finite Volume Method (FVM) is presented, particularly their ability to capture the important characteristics of an inviscid transonic flow around a wing, the well known test case - the Onera M6 wing. All the results are compared both in-between and with the experimental data. For the numerical computation at first two rather classical schemes based on Lax-Wendroff approach were used (cell-centered MacCormack and cell-vertex Ron-Ho-Ni) with added artificial dissipation along with two different structured finite volume meshes (C and H type). Simultaneously, the computation was carried out with the use of an unstructured grid using two modern schemes the first of which was based on Roe-Riemann solver and the second on an implicit scheme using the combination of HLLC numerical flux and WLSQR scheme.

Key words. FVM, transonic flow, HLLC numerical flux, Lax-Wendroff scheme, Roe-Riemann solver, TVD.

AMS subject classifications. 76M12, 76H05, 76N99

1. Introduction. This paper compares four different FVM schemes based on various approaches. The first two are derived from classical Lax-Wendroff scheme - a modification of predictor-corrector MacCormack scheme in TVD (Total Variation Diminishing [14]) cell-centered form and Ron-Ho-Ni scheme in cell-vertex form.

2. Mathematical Model. Inviscid compressible flow in three dimensions is described by the system of Euler equations, which can be written in the following conservative vector form:

$$W_t + F_x + G_y + H_z = 0, \tag{2.1}$$

where

$$\begin{aligned} W &= \left(\rho, \rho u, \rho v, \rho w, e\right)^T, \\ F &= \left(\rho u, \rho u^2 + p, \rho uv, \rho uw, (e+p)u\right)^T, \\ G &= \left(\rho v, \rho uv, \rho v^2 + p, \rho vw, (e+p)v\right)^T, \\ H &= \left(\rho w, \rho uw, \rho vw, \rho w^2 + p, (e+p)w\right)^T. \end{aligned} \tag{2.2}$$

W is a vector of conservation variables with components: ρ - density, $\mathbf{w} = (u, v, w)$ - velocity vector, e - total energy per unit volume and p - static pressure. F, G, H are inviscid fluxes. System (2.1) is enclosed by the Equation of State:

$$p = (\gamma - 1)\Big[e - \frac{1}{2}\rho(u^2 + v^2 + w^2)\Big], \quad \gamma = \frac{c_p}{c_v}. \tag{2.3}$$

---

[1]Department of Technical Mathematics, Faculty of Mechanical Engineering, Czech Technical University, Prague.
[2]Aeronautical Test and Research Institute, Prague.

where c_p and c_v are specific heat capacities under constant pressure (at constant volume).

3. Numerical Methods. A classic computational fluid dynamics (CFD) validation case - the Onera M6 wing was chosen for the numerical testing. Initial conditions were the same as in the AGARD AR 138 report [7] i.e. initial Mach number $M_\infty = 0.8395$ and angle of attack [16] $\alpha = 3.06°$. System (2.1) was solved using following FVM schemes:

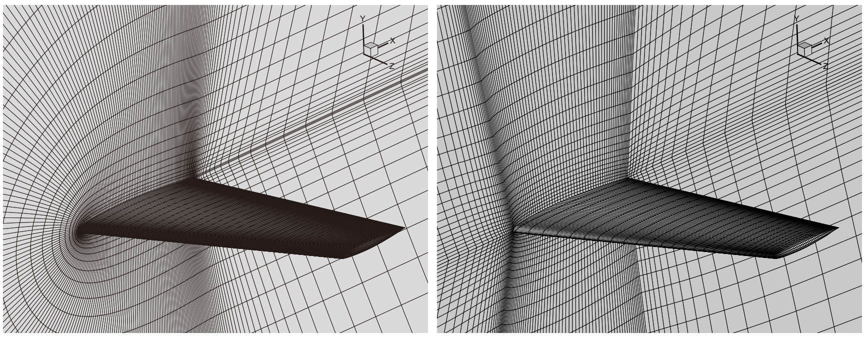
Method 1 As the first method the cell-centered 3D MacCormack predictor-corrector scheme in so called Modified Causon's form was chosen [10]. It's Influence in each spatial dimension is given by a switch dependent on the value of gradient of the vector W. Although this scheme does not posses the TVD property, it is able to deliver even better results than TVD variant of MacCormack scheme. For discretization of the computational area following FVM meshes were used: C-mesh (4.1-a) with 493000 cells and H-mesh (4.1-b)with 687456 cells.

Method 2 In this case the cell-vertex Ron-Ho-Ni scheme described in [2], [5] was used. It is one-step explicit Lax-Wendroff scheme with 3^{rd} order Jameson's artificial dissipation. The same computational mesh as for Method 1 was used.

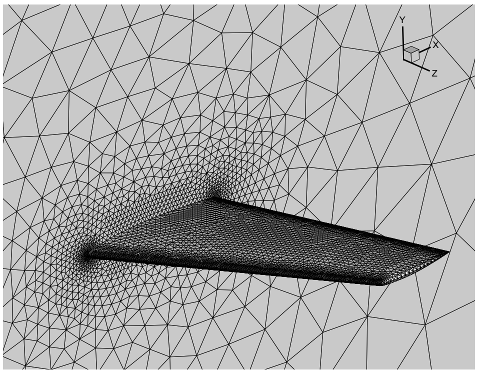
Method 3 Considering the first modern method, the computational area was discretized by an unstuctured mesh with quadrilateral computational cells. The problem was solved by FVM in a cell-centered formulation. Than the Roe-Riemann solver [5] was used to solve the Riemann problem on each side of each finite volume. Spatial accuracy of the method was increased by linear reconstruction using the least square method [6]. For the time discretization the linearized backward Euler method was used. Final system of equations was solved by GMRES method with ILU(0) preconditioning [6].

Method 4 Last used method is an extension of the high-order FVM weighted least-square (WLSQR) scheme mentioned in [4] into three dimensions. The high order WLSQR reconstruction is combined with the HLLC [15] flux and the resulting semi-discrete system of equations is solved by the linearized backward Euler method. Resulting sparse system of linear equations is then solved with GMRES method [6] with modified ILU(0) preconditioner.

The boundary conditions were considered in a standard way for transonic compressible flow. At the inflow boundary the variables (u, v, p) were given by initial values $(u, v, p) = (U_\infty, V_\infty, P_\infty)$, at the outflow boundary the outlet pressure was given by $p = p_\infty$ and on the wall the slip condition was prescribed as $(u, v)_n = 0$.

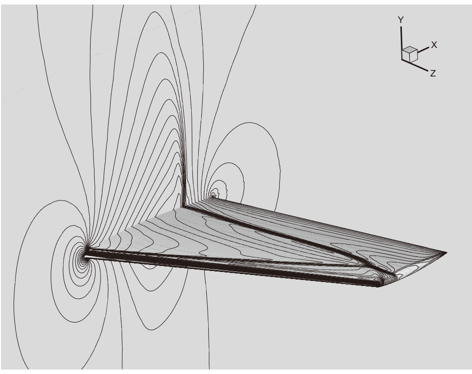


a) Structured C-mesh. b) Structured H-mesh.

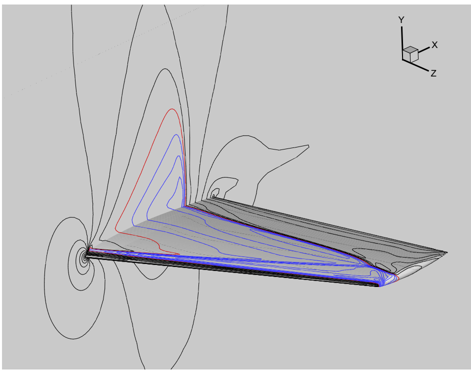


c) Unstructured mesh.

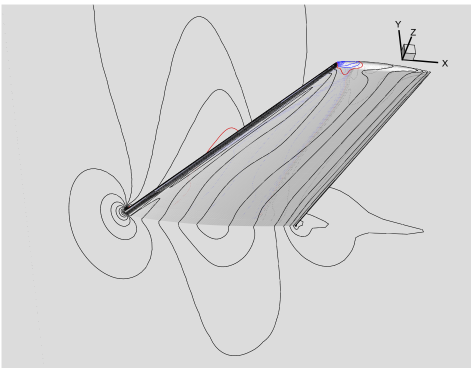
Fig. 4.1: Computational meshes.



a) c_p coefficient, C-mesh, top of the wing.

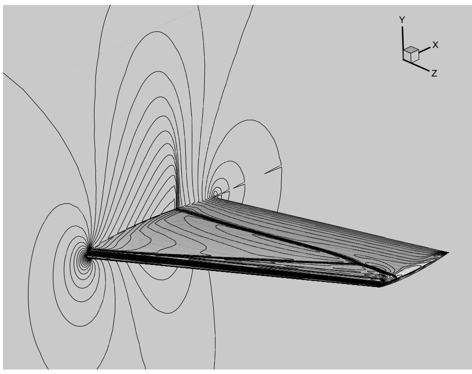


b) Mach number, C-mesh, top of the wing.

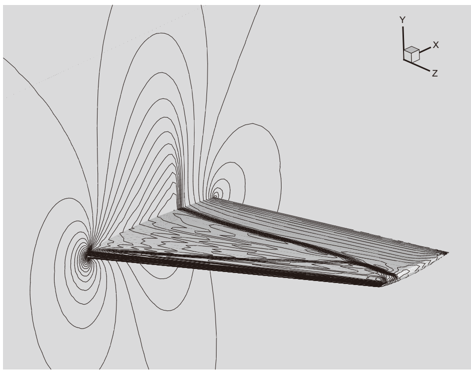


c) Mach number, C-mesh, bottom of the wing.

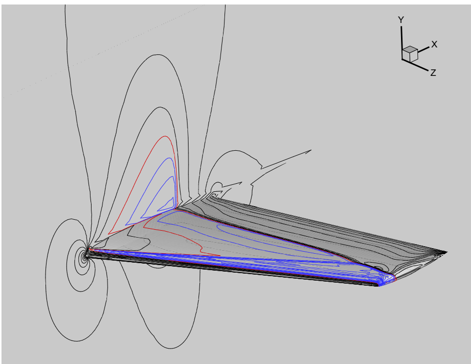
Fig. 4.2: Method 2 - isolines of the c_p coefficient and Mach number.



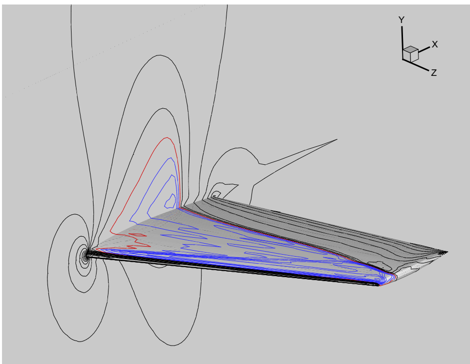
a) c_p coefficient, C-mesh, top of the wing.



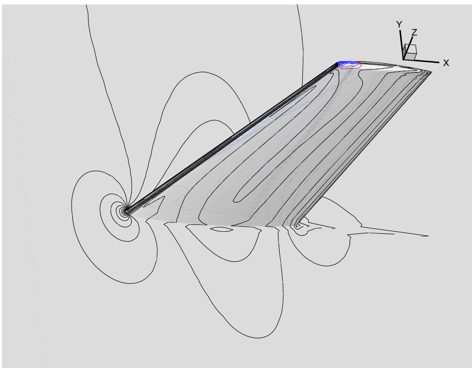
b) c_p coefficient, H-mesh, top of the wing.



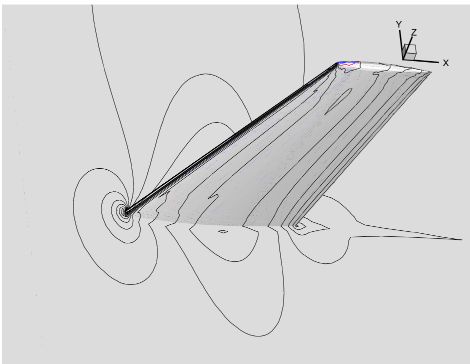
c) Mach number, C- mesh, top of the wing.



d) Mach number, H-mesh, top of the wing.

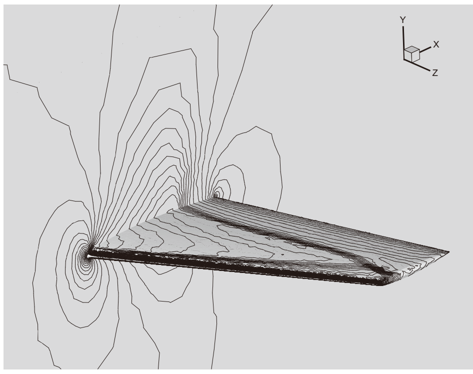


e) Mach number, C-mesh, bottom of the wing.

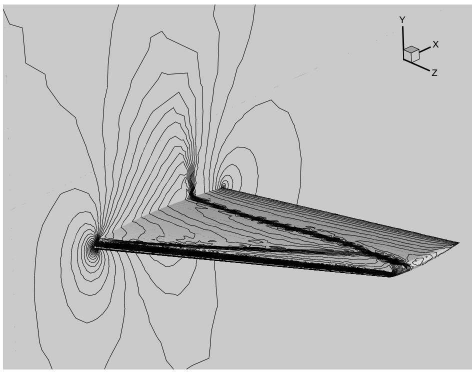


f) Mach number, H-mesh, bottom of the wing.

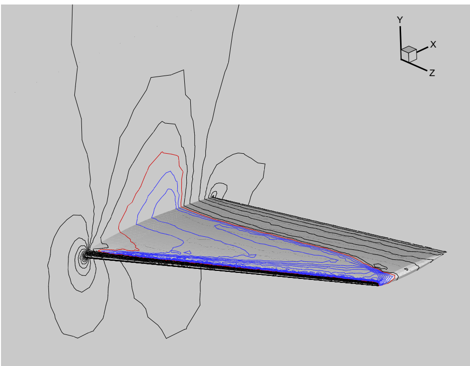
Fig. 4.3: Method 1. Isolines of c_p coefficient and Mach number.



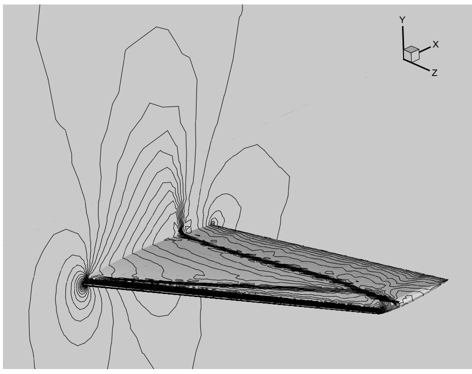
a) c_p coefficient, Method 3, top of the wing.



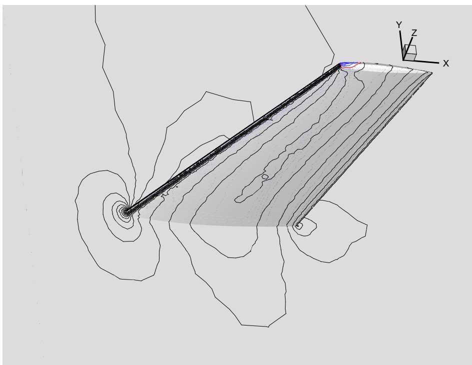
b) c_p coefficient, Method 4, top of the wing.



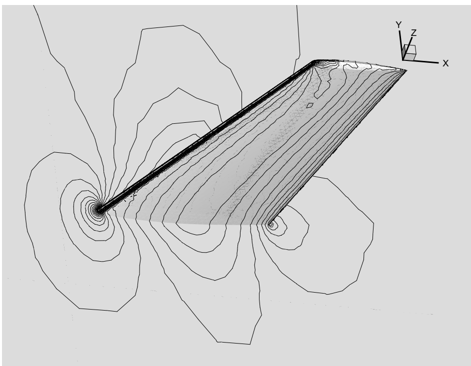
c) Mach number, Method 3, top of the wing.



d) Mach number, Method 4, top of the wing.

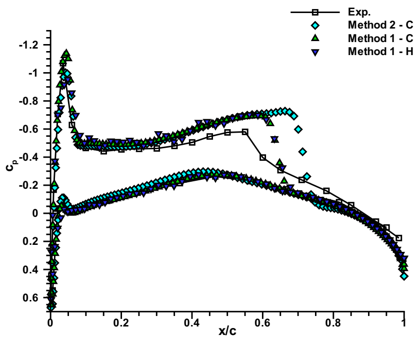


e) Mach number, Method 3, bottom of the wing.

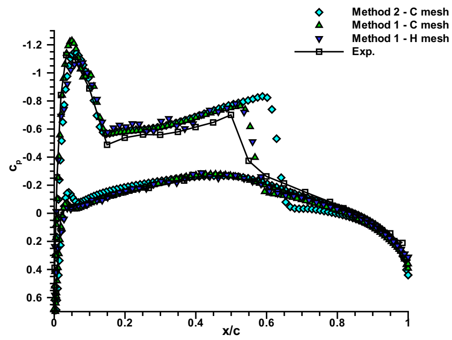


f) Mach number, Method 4, bottom of the wing.

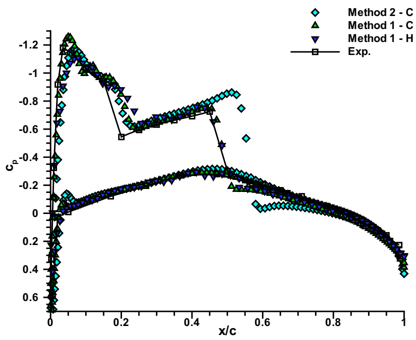
Fig. 4.4: Methods 3 and 4, unstructured mesh. Isolines of c_p coefficient and Mach number alongside the wing.



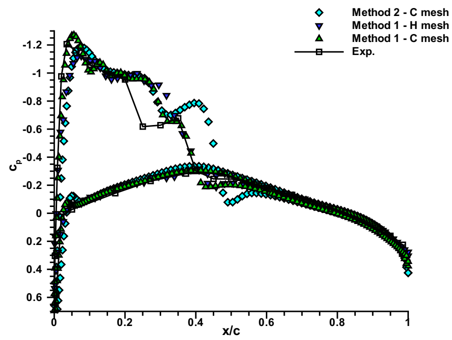
a)



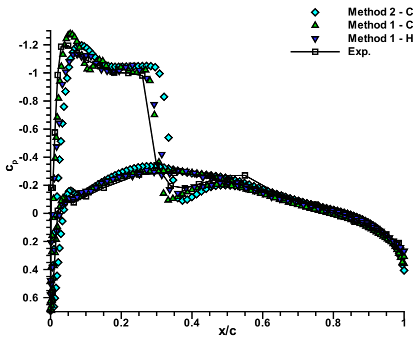
b)



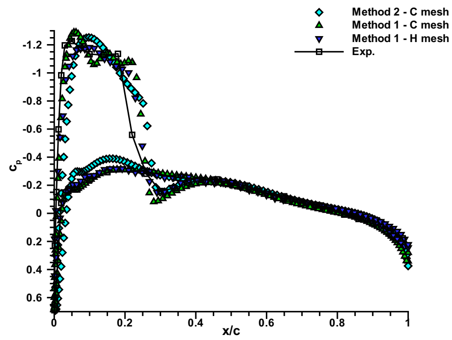
c)



d)



e)



f)

Fig. 4.5: Methods 1 and 2, comparison with the experimental results. Behaviour of the c_p in the cut (x - spatial coordinate, c - chord): a) 20 %, b) 44 %, c) 65 %, d) 80 %, e) 90 %, f) 95 % of the wing spread.

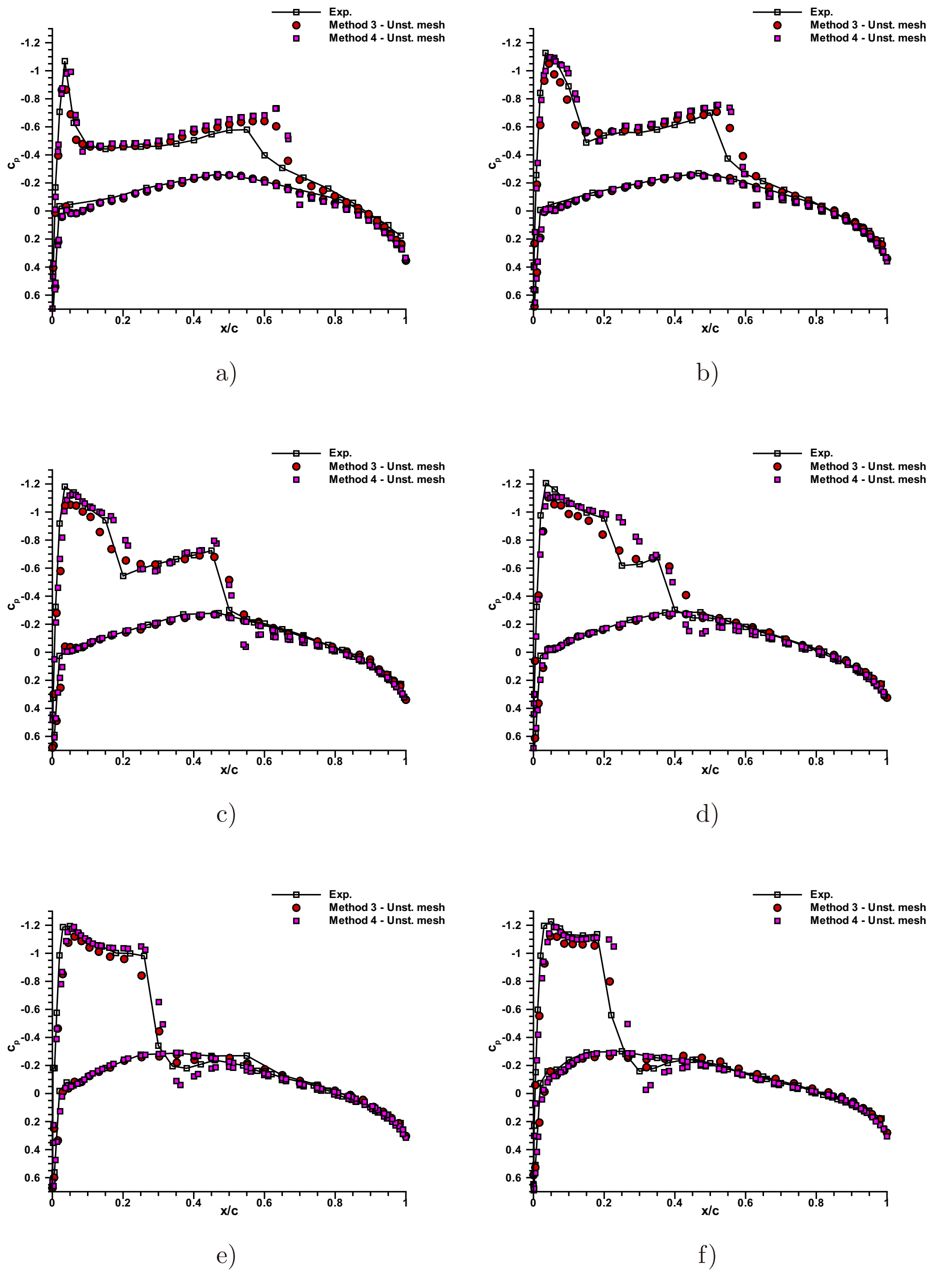


Fig. 4.6: Methods 3 and 4, comparison with the experimental results. Behaviour of the c_p in the cut (x - spatial coordinate, c - chord): a) 20 %, b) 44 %, c) 65 %, d) 80 %, e) 90 %, f) 95 % of the wing spread.

4. Numerical Results. As can be seen from the comparison between the numerical and experimental results in the case of 3D flow over the Onera M6 wing (Figures 4.1 - 4.6), the schemes based on classical Lax-Wendroff approach still deliver satisfactory results, and can be used as a reliable numerical simulations.
Both the position and intensity of shockwaves are captured reasonably well, only the Method No.2 pushes the shockwave closer to the trailing edge (Fig.4.5) - more than the other methods. The observed differences are highly probable consequence of inviscid nature of the chosen model, which collide with viscous turbulent behaviour of the physical reality. Another constraint is explicit form of the classical schemes, which restrain their usabililty in three dimensional computation only to inviscid flows (both steady and unsteady). But considering the possibility of their conversion into an implicit version, both can be successfuly used also for the turbulent simulations. The modern schemes deliver a good results too, particularly the Method 4 (WLSQR scheme) shows a very good correspondence with the experimental data. Method 3 smoothes the shockwave more than the other methods, most probably because of insufficient density of the computational mesh.

5. Conclusion. Proposed FVM schemes for numerical solution of stationary 3D transonic inviscid flow show reasonably good accuracy and efficiency. Although the inviscid mathematical model have been chosen, the schemes were able to capture important flow characteristics as is the position and intensity of the schockwaves and proved themselves as a reliable numerical simulations of investigated cases. Some of the schemes however would need some further improvement (implicit form in the case of Modified Causon scheme, matrix-free GMRES in the case of WLSQR scheme).

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

REFERENCES

[1] Dobeš J., Fořt J., Fürst J., Kladrubský, M., Kozel K., and Louda P. *Numerical Solution of Transonic Flow over a Wing.* Research report (in Czech) V-1814/04, VZLÚ a.s., 2004.

[2] Dobeš, J., Fořt, J., Fürst J., Furmánek P., Kladrubský, M., Kozel, K., Louda, P. *Numerical Solution of Transonical Flow around a Profile and a Wing II.* Výzkumná zpráva V-1850/05, VZLÚ a.s. 2005

[3] Furmánek P., Fürst J., Kozel K. Numerical Solution of Inviscid Transonic Flow over Profile. Conference: TOPICAL PROBLEMS OF FLUID MECHANICS 2004, editors: K. Kozel, J. Prihoda, p. 53-54, ISBN 80-85918-86-2.

[4] Fürst J., Finite Volume WLSQR Scheme and its Applications to Transonic Flows. In *Programs and Algorythms of Numerical Mathematics 13*, Mathematical Institute, Academy of Sciences, Žitná 25, Prague, The Czech Republic, May 2006.

[5] Pelant J., Kyncl M., Kladrubský, M. *Project of CFD methods for the three-dimensional inviscid compressible flow around wings and cascades.* Zpráva VZLÚ, V-1817/04, 2004

[6] Saad, Y., Schulz, M. H., GMRES: a generalized minimalresidual algorithm for solving nonsymetric linear systems. *SIAM J. Sci. Stat. Comput.*, (7):856-869, 1986.

[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. Donea. An arbitrary Lagrangian-Eulerian finite element method for transient fluidstructure interactions. Comput. Methods Apll. Mech. Eng., 33:689?723, 1982.

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

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

[11] J. Fürst, K. Kozel: Application of Second Order TVD and ENO Schemes in Internal Aerody-

namics. Journal of Scientific Computing, Vol. 17, No 1-4, December 2002, ISSN 0885-7474, pp. 263-272.

[12] J. Fürst, K. Kozel: Second and Third Order Weighted ENO Scheme on Unstructured Meshes. Proceedings of "Finite Volumes for Complex Applications III", Ed. Herbin, D. Kröner, Hermes Penton Science, ISBN 1903996341, pp. 737-744.

[13] J. Fürst, M. Janda, K. Kozel: Finite Volume Solution of 2D and 3D Euler and Navier-Stokes Equations. Mathematical Fluid Mechanics, Ed. J. Neustupa, P. Penel, ISBN 3-7643-6593-5, 2001, Birkhäuser Verlag, Basel.

[14] Hirsch, C.:Numerical Computation of Internal and External Flows, Vol 2, Wiley.(1990)

[15] M. Feistauer, J. Felcman, I. Straškraba: Mathematical and Computational Methods for Compressible Flow. Numerical Mathematics and Scientific Computation. Oxford University Press, 2003. ISBN 0-19-850588-4.

[16] http://en.wikipedia.org/wiki/Angle_of_attack