30. Setkání kateder mechaniky tekutin a termomechaniky

HYDRO TERMO

22.-24.6. 2011

Špindlerův Mlýn

Jednotlivý příspěvek ze sborníku

TECHNICKÁ UNIVERZITA V LIBERCI

evropský sociální fond v ČR
EVROPSKÁ UNIE
MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY
OP Vzdělávání pro konkurenceschopnost
INVESTICE DO ROZVOJE VZDĚLÁVÁNÍ

CE-SE SCHEME AND BOUNDARY CONDITIONS

Martin KOSÍK[1], Jiří FÜRST[2]

[1] Ing. Martin Kosík, Department of Technical Mathematics, Faculty of Mechanical Engineering, CTU in Prague, Karlovo nám. 13, Prague, martin.kosik@fs.cvut.cz
[2] Doc., Ing., Jiří Fürst, Ph.D., Department of Technical Mathematics, Faculty of Mechanical Engineering, CTU in Prague, Karlovo nám. 13, Prague, jiri.furst@fs.cvut.cz

Abstract: *CE-SE scheme is new numerical methodology for calculation of conservation laws. The concept and methodology of this method are significantly different from those in the well-established traditional numerical methods. It enforces of both local and global flux conservation in time and space. The unknowns are values and gradient in each mesh points SE. Thus, it is possible to build a method of the higher order of accuracy, which uses only unknowns from neighboring points. Space and time are unified in the current method. This article describes the CE-SE scheme for 2D compressible flows. Further, it describes two types of boundary conditions which are used for computation of inviscid flows in GAMM channel. The results obtained with these two types of boundary conditions are compared with reference solution obtained with second order finite volume scheme on very fine mesh.*

1. Introduction

The CE-SE (Conservation Element – Solution Element) was developed by Dr. Chang of NASA Glenn Research Center. It enforces of both local and global flux conservation in time and space. The unknowns are values and gradient in each mesh points SE. Thus, it is possible to build a method of the higher order of accuracy, which uses only unknowns from neighboring points. Space and time are unified in the current method.

2. Description of CE-SE for 2D

We consider a flow of inviscid compressible fluid described the system of the Euler equations:

$$W_t + \frac{\partial F}{\partial W} W_x + \frac{\partial G}{\partial W} W_y = 0 \qquad (1)$$

$$p = (\kappa - 1)[e - 0.5\rho(u^2 + v^2)] \qquad (2)$$

where is ρ the density; u and v are the velocity components along the x and y axis; e is the total energy per unit volume; p is the pressure; κ=1,4 is the Poisson's constant. Vectors W, F, G are:

$$W = [\rho, \rho u, \rho v, e]^T \qquad (3)$$

$$F = [\rho u, \rho u^2 + p, \rho u v, (e+p) u]^T \qquad (4)$$

$$G = [\rho v, \rho u v, \rho v^2 + p, (e+p) v]^T \qquad (5)$$

From Eq.1 we can get:

$$W_t = -\frac{\partial F}{\partial W} W_x - \frac{\partial G}{\partial W} W_y \qquad (6)$$

Fig.1 shows one CE elements ABCD and the SE element AEFC. In order to update the solution in SE element [i,j] we need four CE elements sharing the node [i,j].

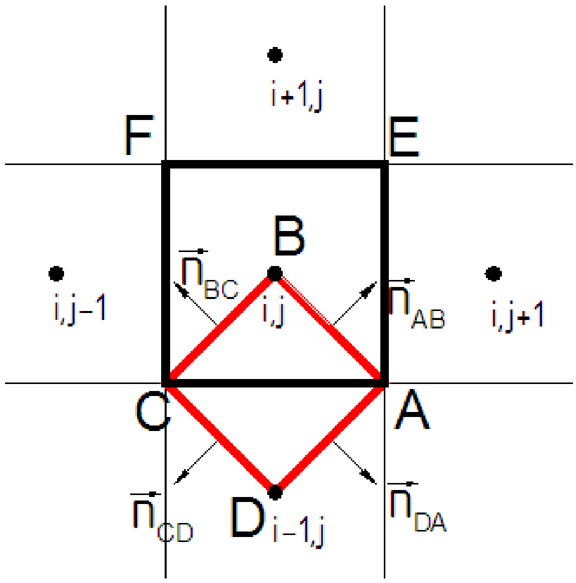

Fig. 1: Position of CE and SE elements in 2D

For the SE element we get from Taylor series expansion: $(x,y,t) \in SE \Rightarrow W(x,y,t) = W^n_{i,j} + W^n_{x_{i,j}}(x - x_{i,j}) + W^n_{y_{i,j}}(y - y_{i,j}) + W^n_{t_{i,j}}(t - t^n)$. For individual CE element we get from equation of conservation: $\iint_{A^n B^n C^n D^n} W^n - \iint_{A^{n+1} B^{n+1} C^{n+1} D^{n+1}} W^{n+1}$

$$- \iint_{A^n B^n A^{n+1} B^{n+1}} \vec{n}(F,G) - \iint_{B^n C^n B^{n+1} C^{n+1}} \vec{n}(F,G)$$

$$- \iint_{C^n D^n C^{n+1} D^{n+1}} \vec{n}(F,G) - \iint_{D^n A^n D^{n+1} A^{n+1}} \vec{n}(F,G) = 0 \,.$$

For each CE we get a new value of W in time-step n+1, and then the new value of W in corresponding SE element is:

$$W^{n+1}_{i,j} = \frac{\sum_{i=1}^{4} |CE_i| . W_i}{\sum_{i=1}^{4} |CE_i|} \qquad (7)$$

Values of derivatives W^{n+1}_x and W^{n+1}_y in SE element are computed using a central differences, see[1].

3. Computational results

We demonstrate CE-SE method for two-dimensional flow in GAMM channel. Geometry of channel is shown in Fig.2, where Γ_{uw} and Γ_{dw} are up-wall and down-wall, Γ_i is inlet of flow to channel and Γ_o is outlet of channel. We used the rectangular mesh with 150x50 cells.

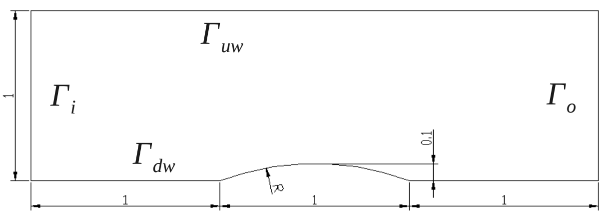

Fig. 2: Geometry of GAMM channel

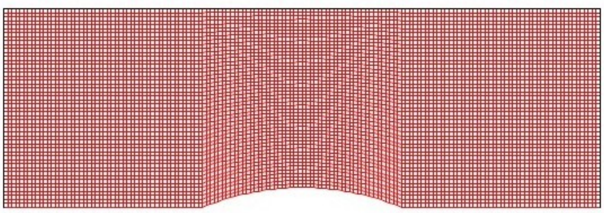

Fig. 3: Mesh of GAMM channel 150x50

We look for solution for inviscid, compressible flows. This flow is described with Euler equations (Eq.1). We considere that the flow is subsonic. In our numerical scheme there aren't any grid points on the border. Instead, we create ghost points (such as G1, G2, G3; see Fig.4, 5, 6) around the border which are mirror of points I1, I2, I3 with respect the border.

We use standard boundary conditions at inflow and at outflow. At the inflow, we set the total pressure $p_o = 1$, total density $\rho_o = 1$ and vertical component of velocity $v_o = 0$. Pressure is extrapolated from the interior and the all other values are calculated from the isoentropic relations. We prescribe outlet pressure p = 0.843 which corresponds to isentropic Mach number M = 0.5. The all other variables are extrapolated from the interior. On the wall, we use two types of boundary conditions:

1) In the first case we use impermeability condition $\vec{u}.\vec{n} = 0$. It is realized by the so-called reflexion. Spatial derivatives at point G3 (see Fig.3), are set $W_x = 0$ and $W_y = 0$.
2) In the second case we use impermeability condition $\vec{u}.\vec{n} = 0$ which is realized by the so-called reflexion as in the previous case. But spatial derivatives at point G3 are obtained from those of point I3; see[2]. For the case of the wall in horizontal direction the solution in G3 is:

$$W = [w_1, w_2, w_3, w_4]$$

$$(w_l)_{G3} = (w_l)_{I3} \quad l = 1,2,4$$
$$(w_l)_{G3} = -(w_l)_{I3} \quad l = 3$$
$$(w_{lx})_{G3} = (w_{lx})_{I3} \quad l = 1,2,4$$
$$(w_{lx})_{G3} = -(w_{lx})_{I3} \quad l = 3$$
$$(w_{ly})_{G3} = -(w_{ly})_{I3} \quad l = 1,2,4$$
$$(w_{ly})_{G3} = (w_{ly})_{I3} \quad l = 3$$

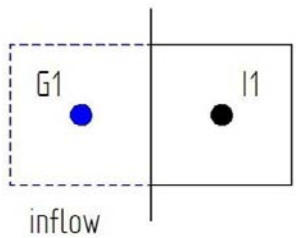

Fig. 4: Ghost points 'inflow'

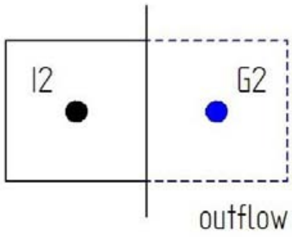

Fig. 5: Ghost points 'outflow'

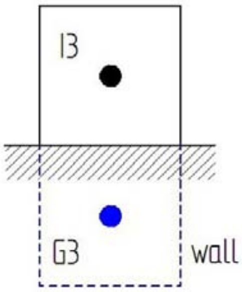

Fig. 6: Ghost points 'wall'

We compare two schemes: the CE-SE and the WLSQR (Weight Least Square Reconstruction). WLSQR is the method of higher accuracy developed by Doc. Jiří Fürst. This method is based on a finite volume method and improves the accuracy of scheme of Godunov type which often suffers from low accuracy. The improvement is achieved by a reconstruction procedure based on least square method combined with data dependent weights for avoiding interpolation across discontinuities, see [3].

The solution obtained with WLSQR scheme has been achieved on unstructured triangular mesh with 24 206 elements. The mesh was refined on down-wall of the GAMM chanel.

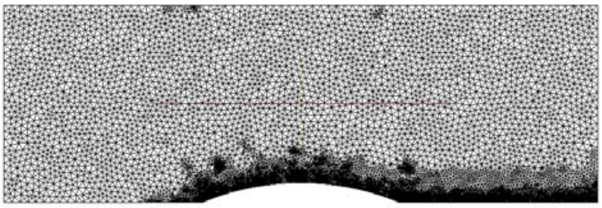

Fig. 7: Mesh for solution by WLSQR

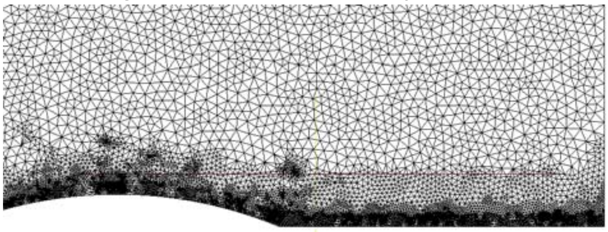

Fig. 8: Cut out of right down corner of figure 7

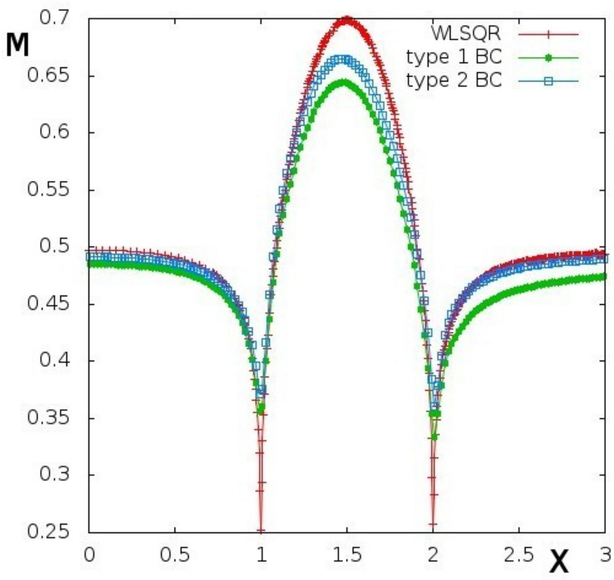

Fig. 9: Distribution of Mach number on the lower of GAMM channel

Fig.9 shows the comparison of two cases of the subsonic flow computed with CE-SE scheme with different boundary conditions (marked as 'type 1 BC' and 'type 2 BC') on the down-wall and the case computed with WLSQR scheme. Horizontal axis shows length of channel and the vertical axis shows Mach number on the lower wall of channel.

The distribution of Mach number has two minimas at x = 1 and x = 2, and one maximum at x = 1.5. The values of Mach number computed with first and second boundary conditions in these there points are:

	1^{st} type	2^{nd} type
x = 1	0.360679	0.376027
x = 2	0.333449	0.360996
x = 1,5	0.643948	0.665051

We can see that the second boundary conditions gives higher maximum Mach number. Moreover, the inlet Mach number is closer to 0.5 at the inflow. It seams that the second version of boundary conditions is more accurate.

It can be seen at the figure 9 that the reference solution obtained with WLSQR scheme gives maximum Mach number

0.698548 at x = 1.5 and two minimas M = 0.251735 at x = 1, M = 0.257745 at x = 2. Although it seems that the WLSQR scheme is much more accurate we have to emphasize that the results of CE-SE scheme were obtained with much coarser mesh and that the CE-SE scheme does not need any Riemann solver and is thereofore much simpler than the WLSQR scheme.

4. Conclusion

We have presented the CE-SE scheme for two-dimensional problems. A software for solution of inviscid, compressible flow in 2D channel has been developed. We have compared two types of boundary conditions with respect to reference fine mesh solution obtained with WLSQR scheme . Results were satisfying. In the future we would like to use this scheme for the solution transonic flows.

5. Acknowledgements

This work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS 10/244/OHK2/3T/12 and by the Grant Agency of Czech Republic, grant No. P101/10/1329.

6. Reference

[1] Qamar S., Warnecke G..: Application of space-time CE/SE method to shallow water magnetohydrodynamic equations, ELSEVIER, 2006

[2] Chang S.-C., Zhang Z.-C., John Yu S.T., Jorgenson P.C.E: A Unified Wall Boundary Treatment for Viscous and Inviscid Flows in the CE/SE Method

[3] Fürst J.: A Finite Volume Scheme with Weighted Least Square Reconstruction, Finite Volumes for Complex Applications IV, ISTE Ltd, 2005, ISBN 1 905209 48 7

[4] Dvořák R., Kozel K.: Mathematical modeling in aerodynamics, CTU in Prague, 1996, ISBN 80-01-01541-6

[5] Kosík M., Fürst J.: Description and application of the CE-SE scheme, Setkání kateder mechaniky tekutin a termomechaniky, 2010, ISBN 978-80-248-2244-0

[6] Kosík M., Fürst J.: CE-SE scheme and its applications, Conference Computational mechanics, 2010, ISBN 978-80-7043-919-7