ParCFD'2020
32^{nd} International Conference on Parallel Computational Fluid Dynamics
May-11-13 2020, Nice FRANCE

APPLICATIONS OF ADAPTIVE MESH REFINEMENT ON HIGH ORDER SOLVER FOR SUPERSONIC FLOWS

Gregorio G. SPINELLI AND Bayram CELIK

Istanbul Technical University,
Department of Astronautical Engineering
34469 Istanbul, TURKEY
e-mail:spinelli15@itu.edu.tr; celikbay@itu.edu.tr

Key words: finite volume method, central essential non-oscillatory scheme, inviscid flows, supersonic flows, forward facing step flow, type VI shock-shock interaction

Abstract. Our in-house code is developed as a high order finite volume solver with Adaptive Mesh Refinement (AMR) capabilities. Spatial accuracy is achieved via Central Essentially Non-Oscillatory (CENO) scheme, while temporal accuracy is given by Runge-Kutta scheme. The CENO scheme works with a fix central stencil, which is generated by using ghost cells at boundaries. In order to reduce the memory requirement of reconstruction stencils at refinement interfaces, we coded the refinement as a block based rather than cell based. To test the adaptive mesh refinement capabilities of the code, we solved two test cases, forward facing step flow, and a type VI shock-shock interaction. For both cases the solutions show that the refinement is carried along regions with shocks and/or discontinuities.

1 INTRODUCTION

One of the scope of CFD Vision 2030 [1] is to address the reliability of modern Computational Fluid Dynamics codes. The missing element of the puzzle is to decrease all form of errors. Nowadays high order methods are good candidates for this aim.

The main advantage of high order methods is the reduction of the number of elements present in the domain. In literature several scheme are available, such as Essentially Non-Oscillatory scheme (ENO) [2], Weighted ENO (WENO) [2], Central ENO (CENO) [3], etc. Among all, the CENO scheme has the advantage to work always on the same central stencil, reducing the time needed for spatial interpolation within cells.

To further boost the potential of high order scheme one can couple it with adaptive mesh refinement [3]. By starting with a coarse mesh the solver automatically adapts the mesh where discontinuities or shock waves are present. This technique gives the same accuracy of the solution obtained with the finest resolution all over the domain. Furthermore the total execution time is decreased.

The aim of this paper is to solve the supersonic backward facing step flow and a type VI shock-shock interaction with a high order code by adapting the mesh to the flow solution.

2 NUMERICAL METHOD

Our in-house code is a high order finite volume solver. The CENO scheme [3] computes the spatial reconstruction within each cell. As of today, the highest order of accuracy in space available in the code is 5^{th} order. For unsteady solution, the classic Runge-Kutta [4] is used as temporal interpolation scheme. The user can chose among 1^{st} and 4^{th} order approximation in time.

Our solver uses the Advection Upstream Splitting Method (AUSM) [5] family scheme as flux splitting method. Within the code the users can opt for 6 different AUSM schemes [5, 6, 7, 8, 9, 10].

The CENO scheme uses a Smoothness Indicator (S) to flag cells which are not fully resolved by the high order interpolation.

$$S = \frac{\alpha}{\max(1-\alpha, \epsilon)} \frac{N_{\text{SOS}} - N_{\text{DOF}}}{N_{\text{DOF}} - 1}, \tag{1}$$

where $N_{\text{SOS}}, N_{\text{DOF}}$ are the number of unknown and the size of the stencil for the reconstruction, ϵ is the machine accuracy, and α is evaluated as in [11]. This indicator recognizes discontinuities and shock waves.

By coupling S with a h-type refinement through the following formula:

$$R = e^{-\frac{\max(0,S)}{U_S S_C}}, \tag{2}$$

the code is adapting the mesh in regions where cells are flagged as under-resolved. In Eq. 2 R is the refinement parameter, U_S is a scaling coefficient, and S_C is the cut-off value [11]. It is clear that R assumes values between $(0, 1]$. A value close to 1 flags the region as refinable and viceversa.

Considering the work load generated by the high order methods, one is keen to adopt parallelization paradigm. Our code takes advantage of shared memory parallelism via openMP [12].

3 RESULTS

The first problem consists of a supersonic flow at Mach 3 confined into a channel with a step. The initial conditions and boundary conditions are the same of Woodward [13].

We obtained the solution of this problem with the AUSM+up flux splitting scheme [6], a 5^{th} order CENO spatial scheme and a 4^{th} order Runge Kutta temporal scheme, where the maximum refinement level is set as 5. The initial mesh consists of 3 blocks and about 900 cells, where the coarse resolution is 0.01 along both directions. When the simulation time reaches 4 seconds, the mesh consists of 393 blocks, and the finest resolution is $3.25 \cdot 10^{-4}$. The block distribution is shown in Fig. 1 along with the density contour. We observe that the refinement is carried along regions which are crossed by shocks and/or discontinuities.

The second problem analyzed is a type VI shock-shock interaction at Mach 9. The problem is set as in Schwing [14].

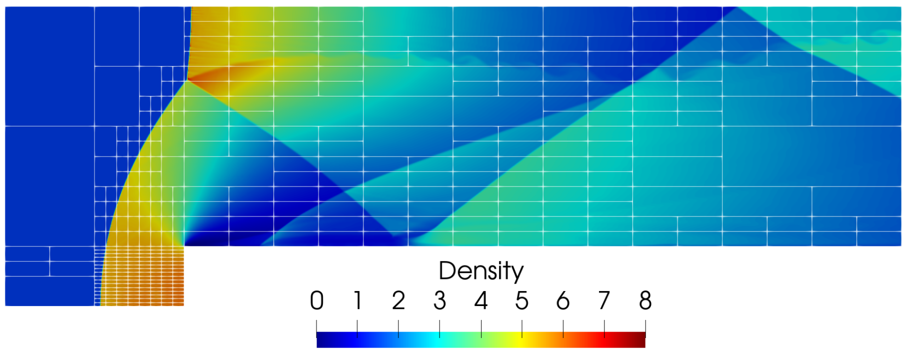

Figure 1: Blocks and density contour of the forward facing step flow of $M = 3$ at $t = 4(s)$

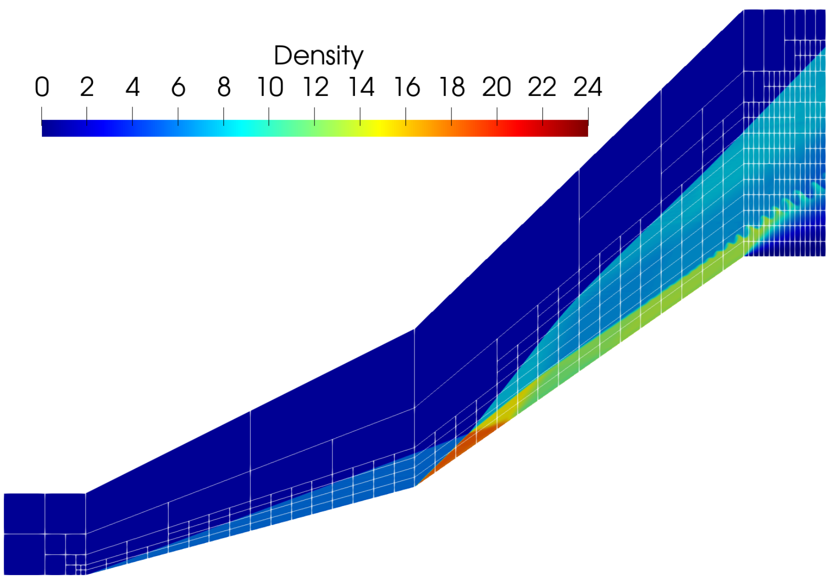

Figure 2: Blocks and density contour of the type VI shock-shock interaction of $M = 9$.

This solution is obtained with the same solver configuration of the previous case, but as flux splitting method we used AUSMPWM [10] and the maximum number of refinement is set as 4. The initial domain is formed by 3 blocks. When the simulation is over, solutions consists of 391 blocks, which are shown in Fig. 2. For this case, the coarse resolution is 0.0333 while the finest resolution is 0.002. As previously observed, the refinement follows the shocks and discontinuities.

4 CONCLUSIONS

Our in-house finite volume solver is used to compute two inviscid supersonic flows: forward facing step flow and type VI shock-shock interaction. We successfully coupled adaptive mesh refinement with central non oscillatory spatial scheme and AUSM flux splitting methods. The developed code is capable of refining the domain where shocks or discontinuities are present.

An interesting future work would be a comparison between WENO, ENO and CENO schemes for supersonic applications. Furthermore, our in-house code can be extended to solve viscous flows.

References

[1] NASA. *CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences*. Tech. rep. 2014.

[2] Chi-Wang Shu. "High order ENO and WENO schemes for computational fluid dynamics". In: *High-order methods for computational physics*. Springer, 1999, pp. 439–582.

[3] Lucian Ivan. "Development of high-order CENO finite-volume schemes with block-based adaptive mesh refinement". In: *University of Toronto* (2010).

[4] John Charles Butcher. "A history of Runge-Kutta methods". In: *Applied numerical mathematics* 20.3 (1996), pp. 247–260.

[5] Meng-Sing Liou. "A sequel to ausm: Ausm+". In: *Journal of computational Physics* 129.2 (1996), pp. 364–382.

[6] Meng-Sing Liou. "A sequel to AUSM, Part II: AUSM+-up for all speeds". In: *Journal of computational physics* 214.1 (2006), pp. 137–170.

[7] Keiichi Kitamura and Eiji Shima. "Towards shock-stable and accurate hypersonic heating computations: A new pressure flux for AUSM-family schemes". In: *Journal of Computational Physics* 245 (2013), pp. 62–83.

[8] Kyu Kim, Chongnam Kim, and OH Rho. "Accurate computations of hypersonic flows using AUSMPW+ scheme and shock-aligned grid technique". In: *29th AIAA, Fluid Dynamics Conference*. 1998, p. 2442.

[9] Feng Qu et al. "An improvement on the AUSMPWM scheme for hypersonic heating predictions". In: *International Journal of Heat and Mass Transfer* 108 (2017), pp. 2492–2501.

[10] Feng Qu et al. "A parameter-free upwind scheme for all speeds' simulations". In: *Science China Technological Sciences* 58.3 (2015), pp. 434–442.

[11] Lucian Ivan and Clinton PT Groth. "High-order solution-adaptive central essentially non-oscillatory (CENO) method for viscous flows". In: *Journal of Computational Physics* 257 (2014), pp. 830–862.

[12] *https://www.openmp.org/*. last reached on 21.01.2020.

[13] Paul Woodward and Phillip Colella. "The numerical simulation of two-dimensional fluid flow with strong shocks". In: *Journal of computational physics* 54.1 (1984), pp. 115–173.

[14] Alan Michael Schwing. "Parallel Adaptive Mesh Renement for High-Order Finite-Volume Schemes in Computational Fluid Dynamics". PhD thesis. Faculty of the Graduate School of the University of Minnesota, Aug. 2015.