An Hybrid Numerical Flux for Supersonic Flows with Application to Rocket Nozzles

Andrea Ferrero and Domenic D'Ambrosio

17TH INTERNATIONAL CONFERENCE OF NUMERICAL ANALYSIS AND APPLIED MATHEMATICS, 23-28 SEPTEMBER 2019, RHODES, GREECE

An Hybrid Numerical Flux for Supersonic Flows with Application to Rocket Nozzles

Andrea Ferrero[1,a)] and Domenic D'Ambrosio[1,b)]

[1]*Dipartimento di Ingegneria Meccanica e Aerospaziale, Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129 Torino, Italy*

[a)]Corresponding author: andrea_ferrero@polito.it
[b)]domenic.dambrosio@polito.it

Abstract. The numerical simulation of shock waves in supersonic flows is challenging because of several instabilities which can affect the solution. Among them, the carbuncle phenomenon can introduce unphysical perturbations in captured shock waves. In the present work, an hybrid numerical flux is proposed for the evaluation of the convective fluxes that avoids carbuncle and keeps high-accuracy on shocks and boundary layers. In particular, the proposed flux is a combination between an upwind approximate Riemann solver and the Local Lax-Friedrichs scheme. A simple strategy to mix the two fluxes is proposed and tested in the framework of a discontinuous Galerkin discretisation. The approach is investigated on the supersonic flow around a cylinder, on the supersonic flow on a flat plate and on the flow in a rocket nozzle.

INTRODUCTION

The numerical simulation of supersonic flows represents a fundamental tool for the design of aerospace propulsion systems (rocket nozzles, transonic compressors and turbines in air breathing engines) and re-entry space vehicles. The shock waves, which can be observed in supersonic flows, introduce several numerical problems that increase the difficulty of the simulation: order of accuracy reduction, convergence problems, instabilities. Among the different numerical instabilities which can be triggered by shock waves, the carbuncle phenomenon is one of the most investigated. The carbuncle consists in the develoment of unphysical distortions in the structure of the shock wave. This instabilitiy, which is observed in both 2D and 3D flows, can dramatically affect the numerical prediction of the bow shock wave in front of a re-entry vehicle or the shape of the Mach disk in an over-expanded rocket nozzle.

The carbuncle instability affects the existing numerical fluxes in different ways. A comparison of the behaviour of several numerical fluxes is reported in [1]. Some upwind fluxes like the ones proposed by Osher [2], Pandolfi [3], Roe [4] seem to be particularly prone to this instability. On the contrary, there are some other classical fluxes like the Rusanov or local Lax-Friedrichs flux [5], the van Leer flux vector splitting [6] or the AUSM+ [7] which are not significantly affected by this problem. Often, the ability to avoid carbuncle is associated to large numerical dissipation. For this reason, several efforts for the development of new carbuncle-free methods, which introduce low dissipation, have been done.

The mechanism behind the development of the carbuncle instability seems to be related to the lack of numerical dissipation in the direction tangential to the shock wave [1]. In order to fix this problem, rotated numerical fluxes have been proposed. In particular, Nishikawa and Kitamura [8] proposed a rotated flux which combines the Roe and the HLLC fluxes in order to add more dissipation in the required direction: their approach requires to solve two different Riemann problems along perpendicular directions and to combine the obtained fluxes. Recently, Guo and Tao [9] proposed a hybrid AUSM+-FVS flux which combines the accuracy of the AUSM+ in boundary layers with the robustness of FVS in shock regions.

An alternative strategy was proposed by Hu and Yuan [10]. They introduced an hybrid flux obtained by combining the HLLC and the FORCE scheme.

The present work describes a strategy which is similar in spirit to the approach of Hu and Liuan [10]. The proposed approach is different because it uses a simpler blending method. Furthermore, in the present work the strategy will be applied by mixing the Flux Difference Splitting (FDS) approach (defined according to [3] or [2]) and the local Lax-Friedrichs flux [5]. The obtained numerical flux will be used to approximate convective fluxes in the framework of a discontinuous Galerkin discretisation. However, the same numerical flux can be used in finite volume discretisations.

NUMERICAL FRAMEWORK

Space and Time Discretisation

The governing equations (Euler or Navier-Stokes equations) which describe the problems considered in this work are discretised by means of the method of lines. The spatial discretisation is performed by a second order accurate Discontinuous Galerkin (DG) method while time integration is performed with the first order explicit Euler scheme. Since steady problems are considered here, the steady solution is obtained by marching in time. The domain can be discretised by unstructured grids with both triangular and quadrilateral elements.
The DG scheme used here requires to introduce a modal hierarchical basis inside each element. In particular a basis obtained by the orthonormalisation of a set of monomials is chosen, following the approach of [11]. Diffusive fluxes are computed by means of a recovery based scheme [12], while shock capturing is performed by means of a filtering approach [13]. The use of DG schemes allows to easily implement adaptive meshing strategies which can be particularly useful in the presence of shock waves or separated boundary layers [14, 15]. Furthermore, the flexibility of the scheme allows to use empirical basis functions trained on an high-fidelity database [16].

HLLC-FORCE Hybrid Flux

Hu and Yuan [10] introduced a numerical flux obtained by a mix of the HLLC and FORCE fluxes. Consider the numerical approximation of the convective fluxes for a 2D problem across a generic interface of the mesh. The flux vector is defined as $\hat{\boldsymbol{F}} = \{\hat{F}_1, \hat{F}_2, \hat{F}_3, \hat{F}_4\}^T$, where the components refer to mass, momentum in the interface normal direction, momentum in the interface tangential direction and energy. The hybrid flux proposed in [10] is equal to the HLLC flux for the energy equation and for the momentum equation in the direction normal to the interface. In contrast, a linear combination of the two fluxes is performed for the density and momentum equation in the direction tangential to the interface. The coefficients of the combination are computed by comparing the interface normal and the shock normal which is approximated by the velocity vector jump across the interface [10].

Proposed FDS-LLF Hybrid Flux

In the present work an alternative hybrid flux is proposed by introducing a linear combination of the Flux Difference Splitting (FDS) [3] and the Local Lax-Friedrichs (LLF)[5] fluxes. The idea is to combine the accuracy of the FDS flux with the robustness of the LLF flux. While in the approach proposed in [10] the blending of the fluxes is performed only for two equations, here the same linear combination is performed for all the components of the flux vector:

$$\hat{\boldsymbol{F}} = \theta \boldsymbol{F}^{\boldsymbol{FDS}} + (1-\theta)\boldsymbol{F}^{\boldsymbol{LLF}} \tag{1}$$

$$\theta = \begin{cases} \frac{|\boldsymbol{\Delta u}\cdot\boldsymbol{n}|}{|\boldsymbol{\Delta u}|}, & |\boldsymbol{\Delta u}| > \epsilon \\ 1, & |\boldsymbol{\Delta u}| <= \epsilon \end{cases} \tag{2}$$

where ϵ is a small constant to avoid division by zero (here set to $\epsilon = 10^{-6}$), $\boldsymbol{n}$ is the interface normal and $\boldsymbol{\Delta u}$ is the velocity vector jump across the interface. The blending function, θ, is computed from the interface normal and the velocity jump across the interface. The idea is that the velocity jump direction identifies the normal to the shock wave: when the interface is aligned to the shock wave the FDS flux is used and when the interface is normal to the shock wave the LLF flux is adopted. In this way, more dissipation is added in the direction tangential to the shock and carbuncle is avoided. The algorithm is local and can be easily implemented in both Finite Volume and Discontinuous Galerkin Finite Element codes. Since the cost of the LLF flux is significantly lower with respect to the FDS flux, the proposed hybrid approach does not increase significantly the computational cost with respect to the original FDS flux. However, the robustness is strongly improved. Furthermore, the hybrid approach does not introduce any memory overload because it works on variables which are already available from the original numerical fluxes.

RESULTS

The proposed approach is tested on three different test cases and it is compared with the results provided by the FDS approach and the LLF approach.

Inviscid Supersonic Flow Around a Cylinder at Mach=4

The supersonic inviscid flow around a circular cylinder at Mach=4 is simulated with a second order accurate DG scheme. An ideal gas with specific heat ratio $\gamma = 1.4$ is assumed. The computational domain is discretised by means of a structured mesh with 160×80 elements. The Mach field reported in Figure 1 shows cleary that the FDS flux introduces a carbuncle instability close to the simmetry axis, while the LLF does not show the carbuncle, but it is very dissipative. The proposed hybrid flux avoids the carbuncle instability and maintains the accuracy.

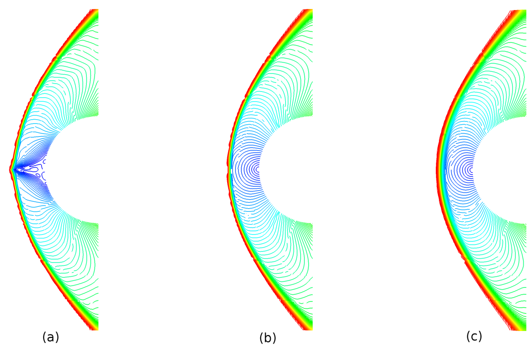

FIGURE 1. Mach field around a cylinder at $M_\infty = 4$: (a) FDS flux, (b) proposed hybrid flux, (c) LLF flux

Over-expanded rocket nozzle flow

The inviscid flow in a converging-diverging nozzle with an area expansion ratio $\epsilon_A = 80$ is studied. The domain is discretised with an unstructured mesh with both quadrilateral and triangular elements (22651 elements). The Nozzle Pressure Ratio (NPR, given by the ratio from the inlet total pressure and the ambient static pressure) is set to 121. The fluid is assumed to be an ideal gas with specific heat ratio $\gamma = 1.2$. In these conditions, the nozzle is strongly over-expanded and a shock wave system with a central Mach disk appears at the exit. Figure 2 shows the predicted Mach field: a comparison of the results obtained by the different fluxes in the Mach disk region leads to the same conclusions of the previous test case.

Flat Plate Boundary Layer at $M_\infty = 5$

Finally, the viscous flow on a flat plate is studied in order to evaluate the behaviour of the proposed method in the presence of boundary layers. The test case is characterised by a supersonic flow ($M_\infty = 5$) of air ($\gamma = 1.4$, $Pr = 0.72$) on a isothermal plate ($T_w = T_\infty = 72K$). The mesh contains 160×100 elements and a growing factor equal to 1.02 is adopted in the wall normal direction. The pressure distribution as a function of the wall distance is reported in Figure 3 for a station with a local Reynolds number $Re_x = 55625$: while the LLF flux introduces large numerical dissipation, the proposed hybrid flux gives results in line with the FDS flux and the reference data from [17].

REFERENCES

[1] M. Pandolfi and D. D'Ambrosio, Journal of Computational Physics **166**, 271–301 (2001).
[2] S. Osher and F. Solomon, Mathematics of computation **38**, 339–374 (1982).
[3] M. Pandolfi, AIAA journal **22**, 602–610 (1984).
[4] P. L. Roe, Journal of computational physics **43**, 357–372 (1981).
[5] V. V. Rusanov, USSR Computational Mathematics and Mathematical Physics **1**, 304–320 (1962).

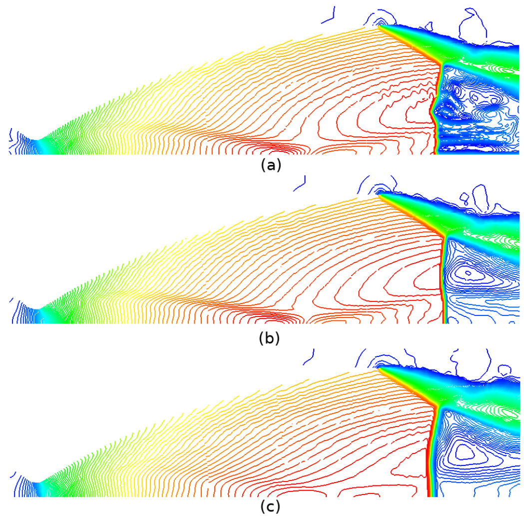

FIGURE 2. Mach field in a rocket nozzle with $NPR = 121$, $\epsilon_A = 80$, $\gamma = 1.2$: (a) FDS flux, (b) proposed hybrid flux, (c) LLF flux

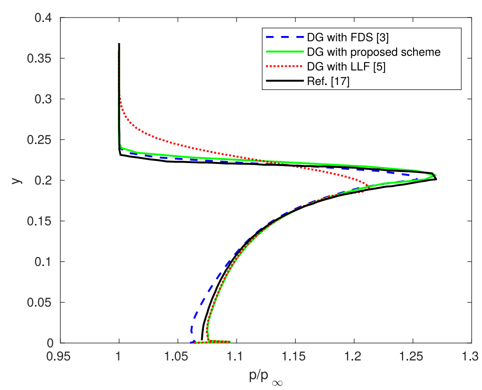

FIGURE 3. Pressure distribution in a boundary layer at $M_\infty = 5$, $Re_x = 55625$

[6] B. Van Leer, in *Upwind and High-Resolution Schemes* (Springer, 1997), pp. 80–89.

[7] M.-S. Liou, Journal of computational Physics **129**, 364–382 (1996).

[8] H. Nishikawa and K. Kitamura, Journal of Computational Physics **227**, 2560–2581 (2008).

[9] S. Guo and W.-Q. Tao, Numerical Heat Transfer, Part B: Fundamentals **73**, 33–47 (2018).

[10] L. J. Hu and L. Yuan, "A robust hybrid hllc-force scheme for curing numerical shock instability," in *Applied Mechanics and Materials*, Vol. 577 (Trans Tech Publ, 2014), pp. 749–753.

[11] F. Bassi, L. Botti, A. Colombo, D. A. Di Pietro, and P. Tesini, Journal of Computational Physics **231**, 45–65 (2012).

[12] A. Ferrero, F. Larocca, and G. Puppo, International Journal for Numerical Methods in Fluids **77**, 63–91 (2015).

[13] A. Ferrero and F. Larocca, Progress in Computational Fluid Dynamics, an International Journal **16**, 14–25 (2016).

[14] A. Ferrero and F. Larocca, "Adaptive cfd schemes for aerospace propulsion," in *Journal of Physics: Conference Series*, Vol. 841 (IOP Publishing, 2017) p. 012017.

[15] A. Ferrero, F. Larocca, and V. Bernaschek, Advances in aircraft and spacecraft science **4**, 555–571 (2017).

[16] A. Ferrero, A. Iollo, and F. Larocca, International Journal for Numerical Methods in Engineering **116**, 332–357 (2018).

[17] M. Pandolfi and D. D'Ambrosio, Computers & fluids **31**, 725–744 (2002).