50h 3AF International Conference
on Applied Aerodynamics
29-30 March – 01 April 2015, Toulouse - France

FP49-2015-chauchat

A contribution to the way of the all-Mach number in aerodynamics

Nicolas Chauchat [1], Eric Schall [2], Nicolas Lantos [3], Gilles Leroy [4]

[1] IPRA-SIAME – UFR Sciences et techniques de Pau – Avenue de l'Université, 64013 Pau, France, nicolas.chauchat@univ-pau.fr
[2] Corresponding author, IPRA-SIAME – UFR Sciences et techniques de Pau – Avenue de l'Université, 64013 Pau, France, eric.schall@univ-pau.fr
[3] ONERA – The French Aerospace Lab – 29 avenue de la Division Leclerc, 92332 Châtillon, France, nicolas.lantos@onera.fr
[4] SAFRAN – TURBOMECA – Avenue Joseph Szydlowski, 64510 Bordes, France, gilles.leroy@turbomeca.fr

ABSTRACT
Aerodynamic in industrial applications is used on a very wide range of Mach number, from very low to very high hypersonic flows. Induced physical phenomena deeply change and the associated numerical methods used to simulate these applications also change. The building of a Computational Fluid Dynamics (CFD) code, which can deal, reliably and accurately, with all kind of Mach numbers is still today a great challenge. Important effort has been done to widen as much as possible the range of applications but this is, to our knowledge, currently done only by the use of a toolbox of dedicated methods but not an All-Mach algorithm. As a first step in this bold goal, this current paper aims to benchmark three CFD codes from various origins and distinct numerical schemes: Concha [1], *elsA* [2], ANSYS-FLUENT [3] on a wide range of steady and unsteady test cases from very low Mach number 0.0001 to high 3. This benchmark allows to better define the limit of the domain of validity of each code and the respective benefits of associated numerical methods. This also allows to have further insights on necessary features of an all-Mach algorithm.

1. INTRODUCTION AND CONTEXT

Industrial aerodynamic applications obviously deeply rely on scientific computing and on the numerical tools used to solve the Fluid dynamics equations. The wide range of Mach number of every day industrial simulations is a challenge for numerical schemes that have to capture complex radically distinct physical phenomena. The constant evolution of numerical schemes originally dedicated to a given range of Mach number widens their relevance domain although their use outside this domain of validity is always uncertain. Despite these grey zones, the toolbox composed of the conglomerate of dedicated schemes achieves to cover now an impressive range of applications, but requires a sound knowledge of the software.
The present study is issued from the three-way collaboration between the University of Pau and Pays de l'Adour (UPPA), ONERA and SAFRAN-TURBOMECA. Its goal is to describe for three CFD softwares from various origins: Concha (an academic code), *elsA* (developed by an aerospace research centre) and ANSYS-FLUENT (FLUENT) (a commercial code), these grey zones via the run of a benchmark on five inviscid steady or unsteady test cases from very low Mach number 0.0001 to high 3. This benchmark is obviously far from exhaustive but already gives insights on scheme limitations.
The paper is organized as follows. After a presentation in section 2 of the three chosen codes and their respective numerical methods, we present, in section 3 the five test cases and the obtained numerical results. Finally in section 4, we draw conclusions and future developments.

2. SOFTWARE PRESENTATIONS

The three chosen software are issued from various origins and use distinct numerical methods to resolve fluid dynamics problems.

Concha software [1] is an academic library developed at Pau University (UPPA). This code

implements, among other finite elements solvers, the Discontinuous Galerkin Finite Element Method (DGFEM) for hyperbolic problem from fluid dynamics (see for example [4]). In this study, we will only consider the DGFEM with the HLLC flux scheme [5] and Tu limiter [6] and we present results approximated with piecewise constant (P0) and linear (P1) functions. These methods are from now on noted DG0 and respectively DG1. The time marching algorithm used is the implicit two-step Backward Differentiation Formulas (BDF2) coupled with a Newton iteration process.

Since 1997, the *elsA* software is a multi-purpose CFD solver developed by ONERA, the French aerospace research centre. This software package capitalizes over time the innovative results of research and is simultaneously used as a basis for research, a tool for investigation and understanding of flow physics, and an industrial software for applied CFD and multiphysics. The *elsA* multi-application CFD simulation platform relies on the resolution of the compressible 3-D Navier-Stokes equations and covers a very wide range of aerospace applications: turbomachinery, aircraft, helicopters, tilt-rotors, counter-rotating open rotors (CROR), unmanned aerial vehicles (UAV), missiles, launchers … (see [2] and <http://elsa.onera.fr> for a exhaustive review of accomplishments both from research and industry).
elsA software resolves here Euler equations with the Finite Volume Method (FVM) and a Riemann solver. For spatial discretization, two upwind schemes are used to compute the flux at the interface of each cell: the well known Roe scheme [7] and the AUSM+(P) MiLES [8] scheme more dedicated to low Mach computations (shorten as AUSM). In all cases, the AUSM parameters were the free stream flow condition. We also use a second order limiter called “minmod” (see Roe [7]) but no additional low speed preconditioner. All computations presented here are simulated with the version 3.3-p2 of the code.

The last chosen code is ANSYS® FLUENT, release R15.0.7. FLUENT is a commercial CFD software developed by Ansys Inc. It allows to model flow, turbulence, heat transfer, and reactions for industrial applications via a large physical modelling capabilities. This software is widely used in industrial community as a multipurpose tool (more information is available on the company web site). The Euler equations are here also solved with a FVM solver with two distinct numerical schemes: the first one, dedicated to incompressible flow simulations, is the so-called Pressure-Based Method (PBM) with the associated SIMPLE [9] and PISO [10] algorithms. The second one is (like *elsA)* a Density-Based Method (DBM) designed for compressible flow simulations. The scheme used is called Roe Flux-Difference Splitting [7] (Roe-FDS) and is, roughly speaking, the same Roe scheme as the one implemented in the *elsA* software (although additional in-house tunings are present).
For each case the CFD is beginning with the same a priori mesh for all codes. In the case where the accuracy of computations is not satisfactory compared to the others, thinner meshes are then tested until the desired degree of accuracy is reached.
Noticeable aspects of the three softwares are summarized in Tab. 1 entitled "Softwares overview". Two-dimensional meshes used to run the computation are composed of Triangle (Tri) or of Quadrilateral (Quad).

Table 1. Softwares overview.

	Modelling	Type method	Type cells	Numerical method
FLUENT	Incompressible	FVM	Tri or Quad	Pressure-based
	Compressible	FVM	Tri or Quad	Density-based
elsA	Compressible	FVM	Quad	Riemann solver (Roe, AUSM …)
Concha	Compressible	DGFEM	Tri or Quad	Riemann solver (HLLC …)

3. THE BENCHMARK: CHOSEN TEST CASES

We have benchmarked the three previously presented codes on five 2D test cases.
These cases cover a wide range of Mach number and they all highlight difficulties inherent to their velocity magnitude: shock waves (straight and oblique), expansion waves, rarefaction and vorticity zones ... As much as possible, we have chosen cases that admit an analytical solution.
Test cases are both steady and unsteady state inviscid simulations from Mach 10^{-4} to Mach 3 and all codes resolve the Euler equations. Main characteristics of these cases are summarised in Tab. 2. Second order numerical schemes are used for space and time discretizations, unless stated otherwise.

We use UNAMALLA software [11] to generate high quality quadrangular structured meshes (case 1 and 5).

Table 2. Summary of test cases.

Test case	Mach	Compressibility	Isentropic	State	Analytical solution
Fraenkel test case	0.0001	Incompressible	YES	Steady	YES
Low-speed nozzle	0.036	Incompressible	YES	Steady	YES
Shock tube	-	Compressible	NO	Unsteady	YES
Isentropic vortex	2.39	Compressible	YES	Unsteady	YES
Cylinder M3	3	Strongly compressible	NO	Steady	NO

Table 3. Discretisation time method of test cases

Test case	FLUENT	*elsA*	Concha
Fraenkel test case	Steady / PBM and DBM-explicit	Implicit	Implicit
Low-speed nozzle	Steady / PBM and DBM-explicit	Implicit	Implicit
Shock tube	Transient explicit	Explicit	Implicit
Isentropic vortex	Transient explicit	Explicit	Implicit
Cylinder M3	Steady / DBM-explicit	Implicit	Implicit

3.1. The Fraenkel test case (FTC)
M 0.0001 - 2D - incompressible - steady

In [12], Fraenkel obtains an analytical solution for an inviscid shear flow around a circular cylinder. The obtained flow contains recirculation zones near the front and rear of the cylinder as presented in Fig. 1. This test case belongs to the very incompressible domain and details about numerical parameters are given on Tab. 4.

For FLUENT, we were not able to obtain a solution with the pressure-based method although it is supposed to be dedicated to incompressible flows. Our solution provided by the density-based algorithm for compressible flows is neither satisfactory as the simulation does not converge: residual stabilizes after losing only three orders of magnitude.

elsA has been tested with two different schemes (Roe and AUSM) and three different grids (e1, e2 and e3) (see Tab. 4 and 5). The history of residuals indicates the loss of more than eight order of magnitudes for both schemes. Nevertheless, only the AUSM scheme manages to capture the two-recirculation zones as shown in Fig. 2 and 3.

Concerning Concha and its DGFEM solver, we test the first and the second order method on two meshes (c1 and c2). It is worth noticing that, as expected from the work of Guillard [13], the first order method, DG0, applied on triangular mesh (c1), achieves to capture the recirculation regions (see Fig. 4) while the use of a quadrangular mesh (c2) does not. Guillard demonstrates that for a FVM upwind scheme in the resolution of the Euler equations for a low Mach number regime, the lack of convergence toward the solutions of the incompressible system disappears with the use of a triangular mesh. This lack of convergence also disappears with the use of DG1, whatever the type of cell used and the recirculation zones are, in any case, well captured (see Fig. 5).

Table 4. Methods and solution software (FTC).

	Methods	Satisfactory solution
FLUENT	PBM SIMPLE	NO
	DBM Roe-FDS	NO
elsA	Roe + minmod	NO
	AUSM + minmod	YES
Concha	HLLC + Tu	YES

Table 5. Meshes details (FTC).

	Type of elements	Number of elements
FLUENT	Quad	108 960
elsA mesh e1	Quad	10 192
elsA mesh e2	Quad	40 000
elsA mesh e3	Quad	6 400
Concha mesh c1	Tri DG0	4 608
Concha mesh c2	Quad DG1	6 400

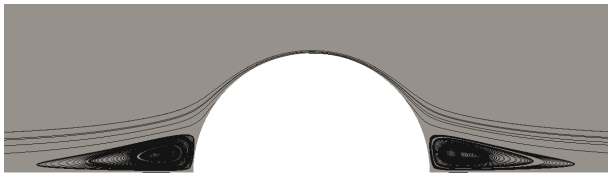


Figure 1. Streamlines (Fraenkel solution on mesh e2: 40 000-Quad)

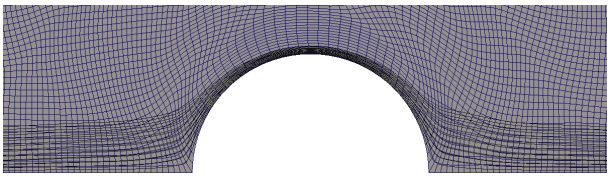


Figure 2. Streamlines with elsA (Roe - mesh e2: 40 000-Quad).

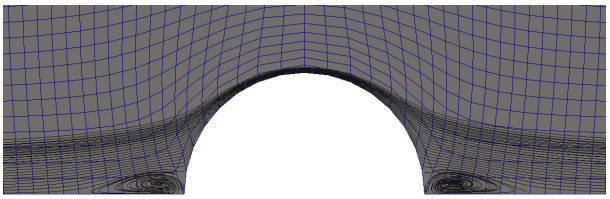


Figure 3. Streamlines with elsA (AUSM - mesh e3: 6 400-Quad).

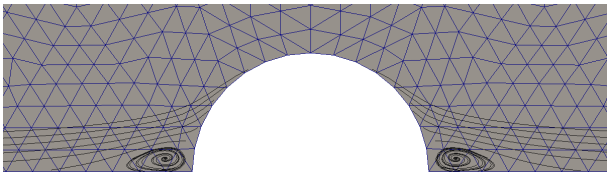


Figure 4. Streamlines with Concha (HLLC – DG0 - mesh c1: 4 608-Tri).

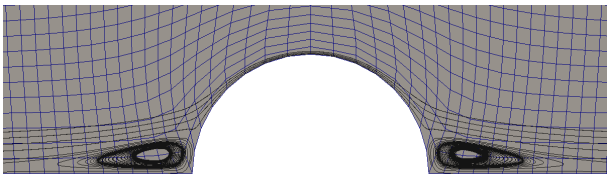


Figure 5. Streamlines with Concha (HLLC – DG1 - mesh c2: 6 400-Quad).

3.2. Internal low speed Nozzle (ILSN)
M 0.036 - 2D - incompressible – steady

In this test case, we simulate a subsonic steady state internal flow. The geometry of nozzle is known and analytical solutions are well-known in the subsonic regime [14]. All computations have been done on a 2D structured mesh of 1000 quadrangular cells. Fig. 6 shows the Mach number along the centred axial direction of the nozzle (y=0). We observe a good agreement between every numerical simulations and the analytical one. It is worth noticing that FLUENT gives almost identical results with its incompressible and its compressible schemes.

Table 6. Methods and solution software (ILSN).

	Methods	Satisfactory solution
FLUENT	PBM SIMPLE	YES
	DBM Roe-FDS	YES
elsA	Roe + minmod	YES
	AUSM + minmod	YES
Concha	HLLC + Tu	YES

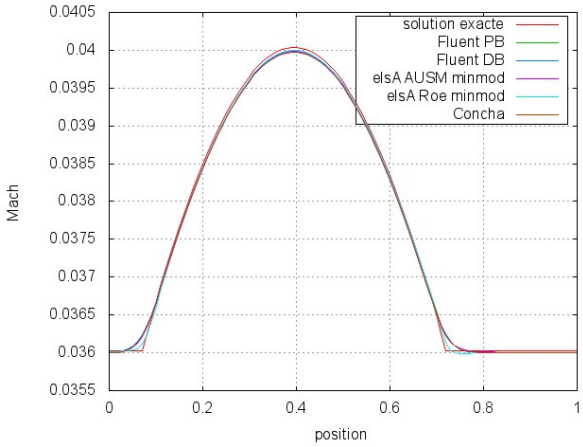


Figure 6. Mach number along the centred axial direction (y=0) of the Nozzle with Concha, elsA and FLUENT.

3.3. Shock-tube (ST)
2D->1D - compressible – unsteady

Among all the Sod shock tube problems, our unsteady test case is described in Tab. 7. As initial conditions, $(\rho, p, u)^L$ and $(\rho, p, u)^R$ are separated by a discontinuity in the middle of the computational domain (x=0). This test case is an 1D quasi 2D simulation *i.e.* although it should theoretically be computed in a one dimensional domain, 2D simulations are led on a thin tube. All simulations are computed on a 2D structured mesh of 1580 quadrangular cells. In all this section, solutions presented here are 1D extraction along the longitudinal direction at y = 0.
Solutions plotted in Fig. 7 demonstrate the good agreement of the computed densities with the analytical solution. DG1 method associated with HLLC + Tu scheme seems to be a little more diffusive than FVM–Roe + minmod around position x=2.

Table 7. Shock-tube, initial conditions.

Left condition	Right condition
ρ = 1.18 kg.m^{-3}	ρ = 0.15 kg.m^{-3}
P=101 325 Pa	P=10 132.5 Pa
V=0 m.s^{-1}	

Table 8. Methods and solution software (ST).

	Methods	Satisfactory solution
FLUENT	DBM Roe-FDS	YES
elsA	AUSM + minmod	YES
	Roe + minmod	YES
Concha	HLLC + Tu	YES

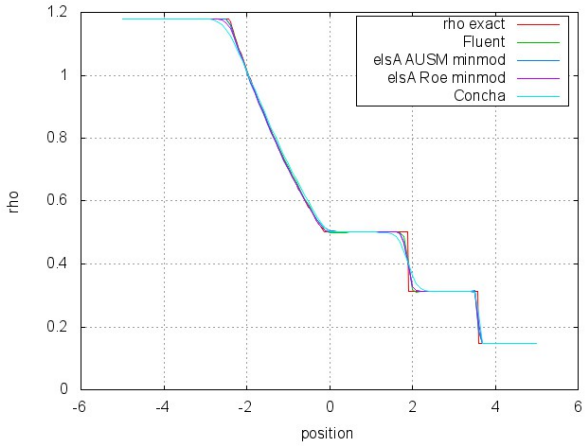


Figure 7. Solution with Concha, elsA and FLUENT, at time t =0.007s.

3.4. Isentropic supersonic vortex (ISV)
M 2.39 - 2D - compressible – unsteady

This two-dimensional case is an unsteady vortex advection (see [15]). The analytical solution of the compressible Euler equations is given by the following Eqs (1)-(5):

$$\mathrm{P}=P_{\mathrm{inf}}\left(\frac{T}{T_{\mathrm{inf}}}\right)^{\frac{\gamma}{\gamma-1}} \quad (1)$$

$$V_x = u_{inf} - \frac{5(y-y_0)}{2\pi} e^{\frac{1}{2}(1-(x-x_0)^2-(y-y_0)^2)} \quad (2)$$

$$V_y = v_{inf} + \frac{5(x-x_0)}{2\pi} e^{\frac{1}{2}(1-(x-x_0)^2-(y-y_0)^2)} \quad (3)$$

$$T = T_{inf}\left(1 - \frac{25(\gamma-1)e^{(1-(x-x_0)^2-(y-y_0)^2)}}{8\gamma\pi^2}\right) \quad (4)$$

$$P_{inf} = 1\ Pa$$
$$\rho_{inf} = 1\ kg.m^{-3} \quad (5)$$
$$u_{inf} = v_{inf} = 2\ m.s^{-1}$$

It describes the isentropic advection of a vortex with the free stream velocity (u_{inf}, v_{inf}). Initially at (x0, y0) = (-10, -10) in a bounded domain Ω = $[-20, 20]^2$, the vortex is supposed to reach (10, 10) at time t=10s. Fig. 9 represents an extraction along the diagonal of the computational domain (i.e. y=x) of the density solutions and the analytical solution. All results presented are almost superimposed. Note that FVM-AUSM and FVM-Roe needed 2.4 times more unknowns than DGFEM to get a solution with the same accuracy.

Table 9. Methods and solution software (ISV).

	Methods	Satisfactory solution
FLUENT	PBM SIMPLE	NO
	DBM Roe-FDS	YES
elsA	AUSM + minmod	YES
	Roe + minmod	YES
Concha	HLLC + Tu	YES

Table 10. Meshes details (ISV).

	Type of elements	Number of elements
FLUENT	Quad	63 504
elsA	Quad	63 504
Concha	Quad DG1	6 400

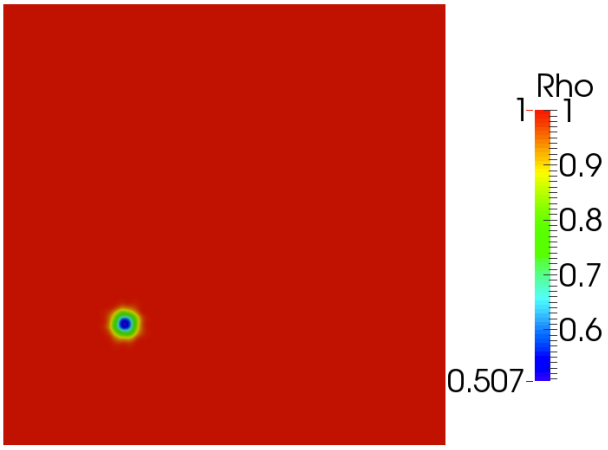


Figure 8. Initial field of the ISV.

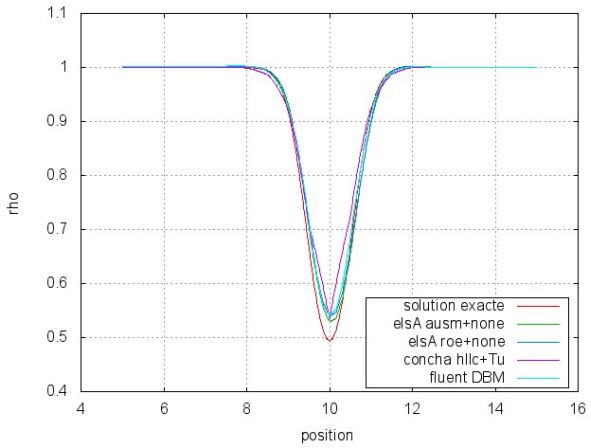


Figure 9. Profile of the density along the diagonal of the domain at time t =10s.

3.5. Cylinder Mach3 (CM3)

M 3 - 2D - compressible – steady

This case [16] is the only one which does not admit any analytical solution because of the complexity of the physical phenomena at stake: bow shock in front of the body and shocks with both recirculation and rarefaction zone in the rear part. Simulations options and meshes details are given respectively in Tab. 11 and 12. Fig. 10-13 show the iso-values of the Mach number for the three codes. While Fig. 10 represents the FLUENT solution, that of *elsA* and Concha are respectively represented in Fig 11, 12 and 13. We note a good agreement between all solutions and there is no carbunkle phenonenon. The streamlines comparisons seen in Fig.14-15 show a relative agreement between all the results as they all capture the re-circulations in the rear part of the cylinder. It is worth recalling that the same mesh is used for both FVMs and DGFEM. Despite this disadvantage in terms of number of degrees of freedom, FVMs solutions are really satisfactory.

Table 11. Methods and solution software (CM3).

	Methods	Satisfactory solution
FLUENT	DBM Roe-FDS	YES
elsA	AUSM + minmod	YES
	Roe + minmod	YES
Concha	HLLC + Tu	YES

Table 12. Meshes details (CM3).

	Type of elements	Number of elements
FLUENT	Quad	1 680
elsA	Quad	1 680
Concha	Quad	1 680

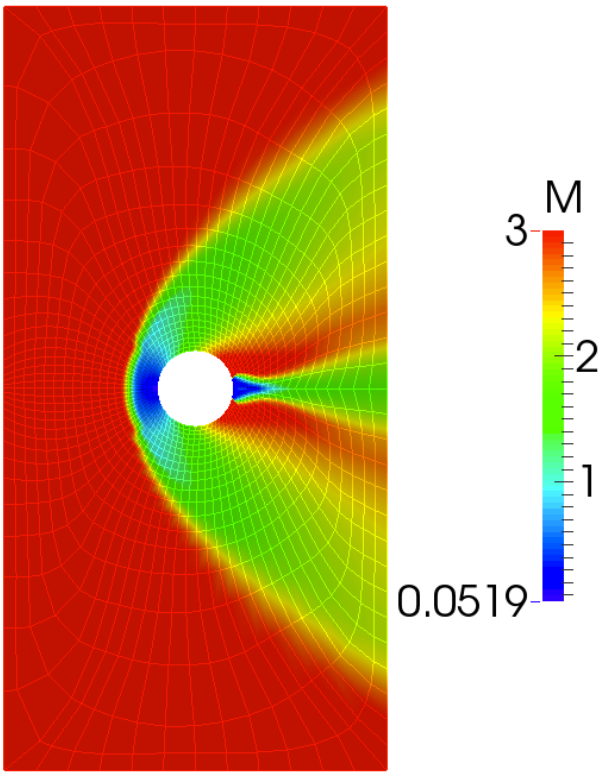


Figure 10. Mach number (FLUENT Roe-FDS).

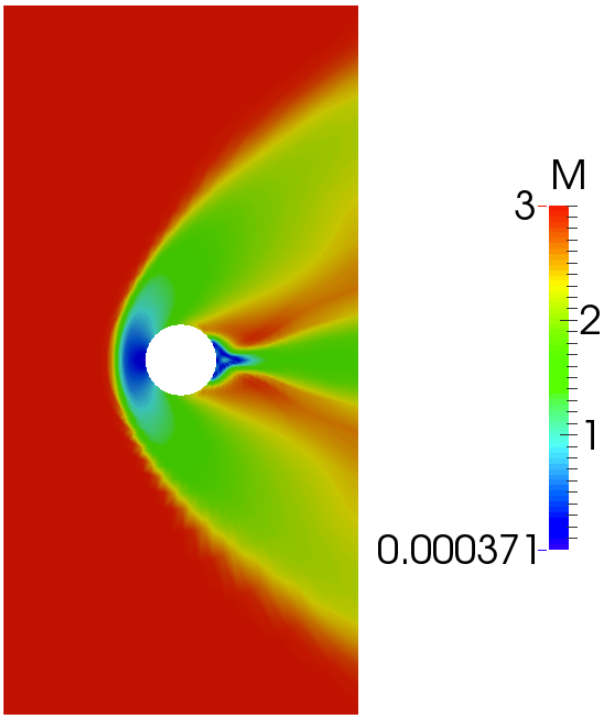


Figure 11. Mach number (elsA AUSM+minmod).

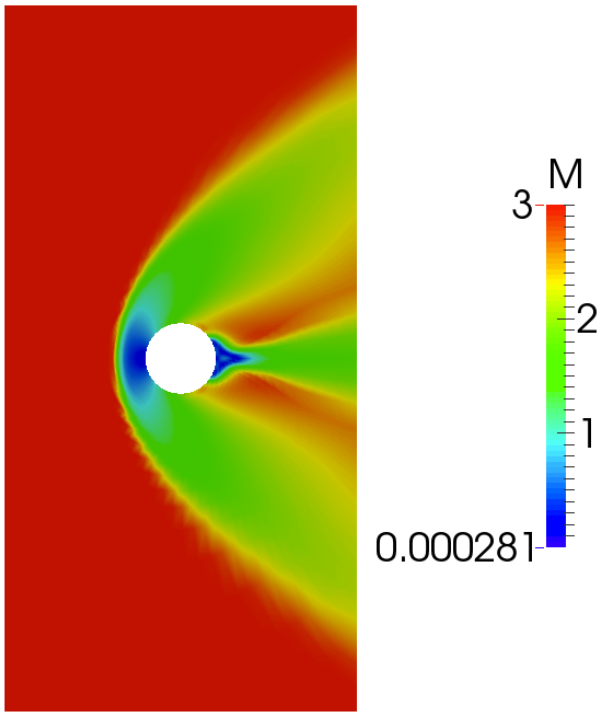


Figure 12. Mach number (elsA Roe+minmod).

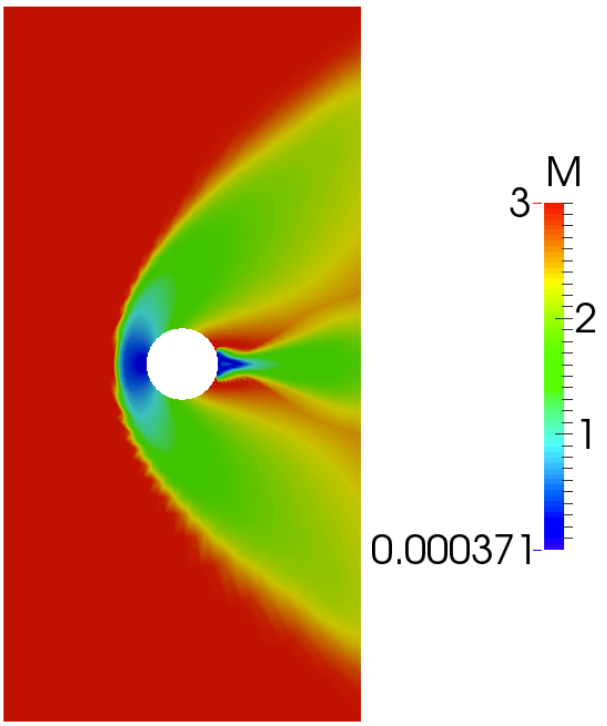


Figure 13. Mach number (Concha DG1).

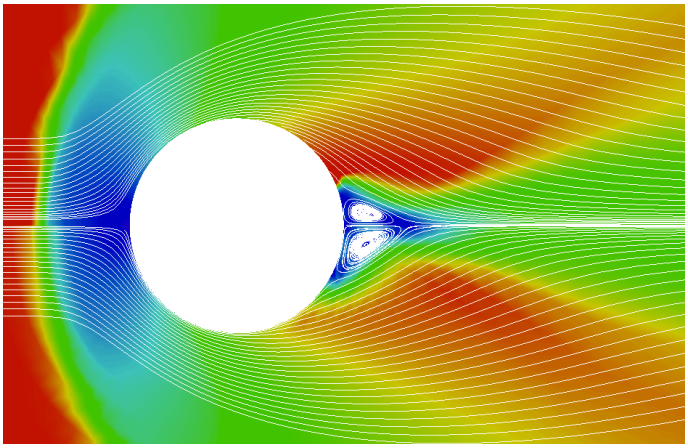


Figure 14. Streamline over the density with Concha (HLLC + Tu) in the upper part and elsA (AUSM + minmod) in the lower part..

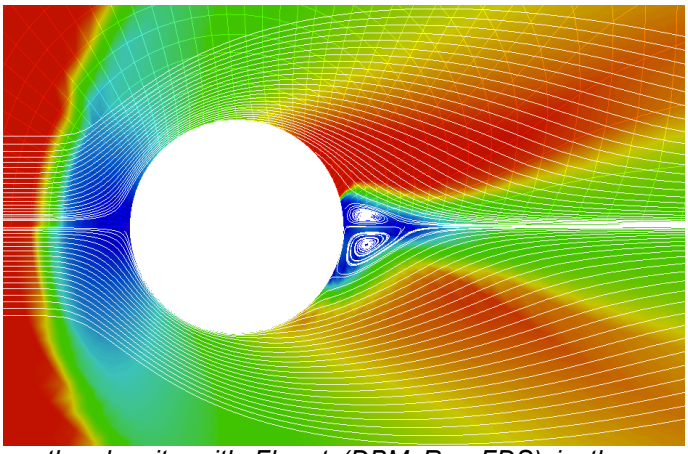


Figure 15. Streamline over the density with Fluent (DBM Roe-FDS) in the upper part and elsA (Roe + minmod) in the lower part.

4. CONCLUSION AND PERSPECTIVES

In this study we benchmark three CFD codes from various origins: Concha (UPPA), *elsA* (ONERA) and FLUENT (ANSYS-FLUENT) on five inviscid steady or unsteady test-cases from very low Mach number 0.0001 to high 3. All codes resolve the Euler equations. Concerning FLUENT and despite our effort, neither the incompressible Pressure-based solver nor the compressible Density-Based Roe-FDS scheme are able to give satisfactory results in all cases. The *elsA* FVM-Roe scheme leads to the same conclusion as it does not achieve to capture the two recirculation zones for the Fraenkel test case (very low Mach number).
Both the *elsA* FVM-AUSM scheme (for structured mesh) and Concha DGFEM-HLLC scheme (for every cell type) provide for every case, reliable solutions in good agreement with the analytical solution (when available).

These results should be reinforced on more test-cases dealing with: viscous fluids, more (low) transient fluids where the time marching algorithm plays an important role… The use of the same flux computation scheme (AUSM or HLLC), for both the FVM and the DGFEM should also provide some interesting information to distinguish the respective advantages of the numerical method (FVM or DGFEM) to capture all-Mach phenomena.

ACKNOWLEDGMENTS

The authors would like to thanks the Pau University (UPPA), the Conseil Régional d'Aquitaine and SAFRAN-Turbomeca for their financial support of the PhD thesis of Nicolas CHAUCHAT. Part of this research was also supported by Public Finance of the DGE (Direction Générale des Entreprises) via the three French aeronautical competitiveness clusters: Aerospace Valley, ASTech and PEGASE.
Authors also thank L. Cambier from ONERA. Altough he did not sign the paper, the authors would like to express their gratitude to Roland Becker for his fruitful advices on Discontinuous Galerkin methods.

REFERENCES

[1]. R. Becker, K. Gokpi, E. Schall, and D. Trujillo. Comparison of hierarchical and non-hierarchical error indicators for adaptive mesh refinement for the Euler equations. Proceedings of the Institution of Mechanical Engineers, Part G: *Journal of Aerospace Engineering*, 227(6): pp. 933–943, June 2013.

[2]. L. Cambier, S. Heib, and S. Plot. The ONERA *elsA* CFD software : input from research and feedback from industry. *Mechanics & Industry*, 14(03), pp.159–174, 2013.

[3]. Franklyn J. Kelecy. Coupling momentum and continuity increases CFD robustness, *ANSYS advantage*, volume II, issue 2, 2008.

[4]. F. Bassi, S. Rebay, and M. Savini, Discontinuous finite element Euler solutions on unstructured adaptive grids, in ‘‘Proceedings, Thirteen ICNMFD, Rome, July 6–10, 1992,’’ *Lecture Notes in Physics*, Vol. 414, pp. 245-249, 1993.

[5]. P. Batten, N. Clarke, C. Lambert, and D. M. Causon. On the choice of wave speeds for the HLLC Riemann solver, *SIAM J. Sci. Comput.* 18(6), pp. 15531570, 1997.

[6]. Shuangzhang Tu and Shahrouz Aliabadi. A slope limiting procedure in discontinuous Galerkin finite element method for gasdynamics applications. *International Journal of Numerical Analysis and Modeling*, 2(2), pp. 163–178, 2005.

[7]. P.L. Roe. Characteristic-based schemes for the Euler equations. *Ann. Rev. Fluid Mech*. 18 pp. 337365, 1986.

[8]. I. Mary and P. Sagaut. Large eddy simulation of flow around an airfoil near stall. *AIAA Journal*, 40(6), pp. 1139-1145, 2002.

[9]. S.V. Patankar and D.B. Spalding, "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", *International Journal of Heat and Mass Transfer*, Vol. 15, pp. 1787-1806, 1971.

[10]. R. I. Issa. Solution of the implicitly discretised fluid flow equations by operator-splitting, *J. Comput. Phys.*, Vol. 62, pp. 40-65, 1986.

[11]. UNAMALLA website. Available at: < http://lya.fciencias.unam.mx/unamalla/home_i.html> [accessed 12.2.2015].

[12]. L.E. Fraenkel. On corner eddies in plane inviscid shear flow. *Journal of Fluid Mechanics*, 11(3), pp. 400–406, 1961.

[13]. H. Guillard. On the behaviour of upwind schemes in the low Mach number limit. IV : P0 approximation on triangular and tetrahedral cells. *Computers & Fluids 38, pp.* 1969–1972, 2009.

[14]. S. Candel. Fluids mechanics, DUNOD, ISBN 2 10 005372 8

[15]. H. C. Yee, M. Vinokurand and M. J. Djomehri. Entropy splitting and numerical dissipation *J. Comput. Phys. 162(1), pp.33—8, 2000*

[16]. P. R. M. Lyra and K. Morgan, A review and comparative study of upwing biased schemes for compressible flow computation. III: Multidimensional extension on unstructured grids. *Arch. Comput. Methods Eng. 9* (3), pp. 207-256, 2002.