LARGE EDDY SIMULATION OF A TRANSIENT SPOILER DEPLOYMENT IN A NACA0012 PROFILE IN SUBSONIC AND TRANSONIC SPEEDS

Linn R. V.*, Awruch A. M.*
***Federal University of Rio Grande do Sul (UFRGS) – Porto Alegre, Brazil**

Keywords: *Turbulence, Moving Meshes, LES, FEM, Mesh Adaptation*

Abstract

Spoilers can be used in airfoils to control the lift and drag forces at different speed ranges. In aircrafts, they can be used at low subsonic speed for landing or also to produces controlled stall conditions at transonic fighting speeds. Spoilers are also used as active systems of load control for wind turbines. The immersed flow conditions, the spoiler and airfoil geometry and the relative movement are important variables to be considered in such situation. The lift and drag forces acting along the deployment can achieve a higher pick value than the full deployed condition or a distinct aerodynamic phenomenon can take place, highlighting the need of investigating the transient motion.

Particularly, turbulence, compressibility and the motion are the main difficulties of such simulations. For the consideration of turbulence and compressible effects, Large Eddy Simulation (LES) combined with the Characteristic-Based Split scheme (CBS) is used in this work. A compressible dynamic Smagorinsky model is employed for the compressible LES model and the CBS scheme is employed in a Finite Element Method (FEM) context for space and time discretization using unstructured meshes. The relative movement of the spoiler is performed using Radial Basis Function (RBF) interpolation techniques. An accurate mesh adaptation scheme is frequently applied along the simulation including mesh refinement, mesh coarsening, edge or face swapping and a vertex smoothing technique in order to control the interpolation error and the mesh size. A feature-based error estimation is employed considering multi-scale and multi-metric intersection in order to capture turbulence effects. The combined use of mesh adaptation allows the control of the mesh quality along the simulation and the capture of local transient effects such as shock waves and eddies.

The transient spoiler deployment in a NACA0012 profile is studied in this work for a Reynolds number of 350000. The subsonic case is compared with available experimental incompressible data and other numerical investigations. The transonic case is also here investigated, showing distinct aerodynamic effects such as buffeting and the presence of shocklets. The transient evaluation of the lift and drag coefficients are presented for the two cases and are important for the correct understanding and quantification of the developed aerodynamics effects.

1 Introduction: Adaptative Simulation of Turbulent Compressible Flows with Moving Boundaries

Relevant industrial and academic problems frequently involve compressible flows around moving bodies with complex geometry. For some of these problems, the movement is known in advance while for other cases motion is directly dependent on the flow dynamics. Numerical simulation of such problems addresses additional computational difficulties once the mesh must me modified along the simulation. Examples can be found in missile deployment problems [1], parachuting dynamics [2], moving components in wings or airfoils [3], rotating helicopter blades [4] and aircraft manoeuvring [5]. In the present work, the fast transient spoiler deployment in a NACA0012 profile is investigated for subsonic and transonic speeds.

The simulation of high-speed turbulent compressible flows has several difficulties. For external aerodynamic problems, the accurate capturing of the complex shock waves, vortical structures and some other flow features is a challenging task which is hard to be accomplished without making use of a mesh adaptation methodology. The adequate mesh to be used in the simulation depends on the desired resolution, the adopted numerical algorithm, the flow characteristics and the domain geometry, being hard to be determined *a priori*, even for an expert in CFD [6]. The large mesh resolution required to correctly evaluate the smallest scales in flows at high Reynolds number is practically prohibitive. Such constraints are alleviated by making use of turbulence modelling strategies such as Large Eddy Simulation (LES) combined with mesh adaptation. Large Eddy Simulation directly calculates the large and energetic vortical structures in turbulent flows, while modelling is necessary to represent the smaller-scales eddies. However, the mesh resolution required for LES simulation still remains too costly due to the wide range of excited length and time scales found in such engineering problems involving turbulent flows with large Reynolds number [7]. As a result, LES simulation of turbulent flows are many times limited to simplified or reduced geometries and are mainly limited to research works due to extremely large computational requirements [8, 9]. Further than the numerical approach for the governing equations, a transient simulation of high-speed compressible flows with large moving boundaries involves special considerations of the changing topology domain and mesh adaptation becomes necessary to handle excessive mesh deformation and correctly capture transient flow features such as moving shocking waves.

In this work, a dynamic compressible Smagorinsky LES model [10] is combined with the Characteristic-Based split Scheme (CBS) [11] in order to simulate turbulent compressible flows. The CBS scheme is a unified approach for Computational Fluid Dynamics (CFD) with capability to cover a wide range of flow speeds with good stability and accuracy compared with other numerical schemes of the same order [12]. The CBS scheme is employed in a Finite Element Method (FEM) context for space and time discretization using unstructured meshes with an adaptation methodology [13, 14], allowing the representation of complex geometries with accuracy.

Mesh adaptation is performed frequently along the simulation, keeping only one simulation for the whole problem. The adaptation is performed by local mesh modification operations, including mesh refinement, coarsening, edge and face swapping and vertex smoothing [10, 13]. The algorithm adaptation performs the following sequence of local mesh modifications:

1. Perform vertex smoothing;
2. Refine edges with error above a given error threshold value;
3. Edge/face swapping;
4. Perform vertex smoothing;
5. Coarse edges with error below a given error threshold value;
6. Edge/face swapping;
7. Perform vertex smoothing;

Several difficulties arise when evaluating the solution error on turbulent compressible flows. The first one is the large range of scales on the flow. Strong shock waves and small perturbations on the flow field are required to be simultaneously captured. For evaluating all the scales of the flow field, a multi-scale evaluation of the error in the $\mathbf{L}^p$ norm is here employed [15, 16, 17]. Furthermore, one variable should be selected for evaluating the error. For steady-state compressible problems, the mass density or the local Mach field are usually employed [6], but this chose is not suitable for turbulent compressible flows. A featured-based estimation of error is here employed [14, 15] combining metric intersection of all conservative variables of the flow in order to cover turbulent effects along mesh adaptation.

The coupling with the flow solver is performed by adapting the mesh after 20 iterations of the finite element solver. All the conservative variables of the flow are intersected in order to evaluate the error and a minimum allowed edge size is directly imposed for the whole simulation [17].

Methodologies to handle with moving boundaries are essentially divided into three groups. The first one uses a monolithic mesh where the computational domain is fitted to the moving body [18, 5]. This means that a mesh deformation technique is necessary. Usually, an Arbitrary Lagrangian-Eulerian (ALE) formulation is used to incorporating the mesh deformation motion into the governing equations. The second class is the Chimera or overset grids methods [19], where each moving body uses an individual mesh and the simulation is performed using interpolation between different superimposed meshes. The last methodology is the immersed boundary technique [20], where the boundary of the body is not contained in the topological boundary of the mesh. For such case, the motion of the body is considered by repositioning the immersed boundary with a fixed mesh.

The use of a monolithic mesh with an ALE framework is usually restricted to small displacement because excessive distortions may considerably reduce precision and stability. In this work, an hybrid technique is employed combining a monolithic approach with overset meshes. The mesh is monolithically moved with the moving body using Radial Basis Function Interpolation (RBFI) [21]. Instead of an ALE description, the solution is transferred from the original mesh to the new mesh.

2 Transient Spoiler Deployment in a NACA-0012 Airfoil

A fast spoiler deployment in an airfoil is here investigated. The NACA-0012 airfol profile is considered with unitary chord $c = 1$, with the spoiler located at the upper boundary between $c = 0.7$ and $c = 0.8$, as indicated in Fig. 1. The angle β with a direction parallel to the airfoil at $c = 0.7$ is defined as the inclination of the spoiler.

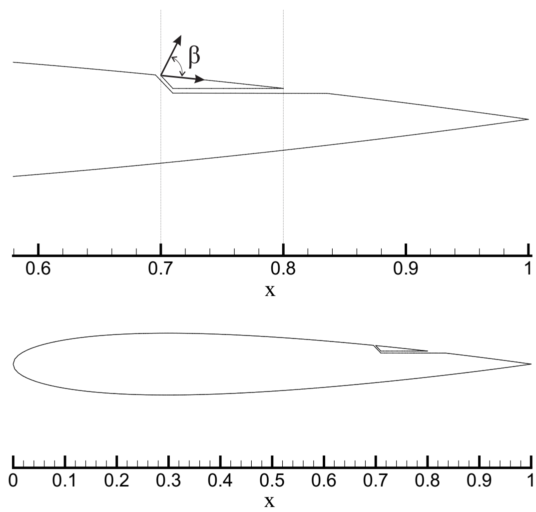


Fig. 1 : Spoiler located at the NACA-0012 airfoil.

The free-stream Reynolds number is $Re_\infty = 3.5 \times 10^5$ and the angle of attack of the airfoil is null. The simulation is performed up to the non-dimensional time $T = 20$, which is divided in the following steps: the initial step, up to time $t_1 = 5.5$, where the flow is developed around the airfoil maintaining the spoiler at the initial position $\beta(t_1) = 0$ (closed). After, the deployment of the spoiler starts, rotating around the left-most point of the spoiler up to the final position $\beta(t_2) = 90^0$ at time $t_2 = 10$ with the following variation [22]:

$$\beta(t) = \beta(t_2)\frac{1}{2}\left(1 - \cos\left(\left(\frac{t - t_1}{t_2 - t_1}\right)\pi\right)\right) \quad (1)$$

Finally, the simulation continues with the spoiler fixed at the fully deployed position. The two-dimensional simulation is performed using a free-stream mass density $\rho_\infty = 1$ and a free-stream velocity $u_\infty = 1$. Two cases of Mach numbers are analysed: 0.4 (subsonic) and 0.8 (transonic), with the subsonic case being compared with the incompressible experimental results [22] and three-dimensional incompressible simulation [3].

Mesh adaptation is performed with a target interpolation error $\varepsilon_{\mathbf{L}^2} = 0.05c$ and with a minimum allowed edge size $h_{min} = 0.001c$.

Figs. 2 and 3 show three different instants for the flow and the adapted mesh along the deployment of the spoiler and Figs. 4 and 5 show a detailed view of the flow and mesh at the position of full deployment of the spoiler. The transient variation of the aerodynamic coefficients and mesh size are shown in Fig. 6.

At the initial stage ($\beta = 0$), there is no separation at the boundary layer for the subsonic case, while buffeting take place for the transonic case. At the intermediary stage ($\beta = 45^0$), the flow separates for both cases, generating vortices behind the spoiler. This zone, where the vortices are generated, is a low pressure zone, and the lift coefficient is suddenly increased for small values of the spoiler inclination angle. This increase of the lift coefficient is soon suppressed when the vortices are moved down and the pressure increases at the frontal area of the spoiler, leading to a high decrease of the lift coefficient. When the spoiler is almost fully deployed ($\beta \geq 70^0$), the flow continues to separate, interacting with the shear layer formed at the trailing edge, leading to a vortex shedding phenomenon.

The present case for $Mach = 0.40$ is reasonable close to the experimental incompressible results presented in [22] and numerical incompressible solution investigated in [3] for the lift coefficient. The drag coefficient exhibits a discrepancy with respect to incompressible flow results presented in [3] at the initial stage of the deployment, while results agree better at the final stage. Finally, it should be observed that present results were obtained with a compressible flow.

3 Conclusions

The deployment of a spoiler in an airfoil was investigated, where large displacements and several flow features such as shock waves, boundary layer and viscous interactions were simultaneously present. Subsonic and transonic flow speeds were investigated and the subsonic case was compared with available incompressible results. The simulation was performed using an adaptative strategy to compressible flows with moving boundaries. The CBS scheme with the Finite Element Method and LES were used for the numerical evaluation of the turbulent flow. Radial Basis Function Interpolation (RBFI) was employed to handle moving boundaries and solution transfer. An anisotropic mesh adaptation was incorporated in order to control mesh distortion and correctly capture flow features.

References

[1] Zyluk A. Experimental validation of mathematical model describing external stores separation. *Journal of theoretical and applied mechanics*, Vol. 43, No. 4, pp 855-873, 2005.

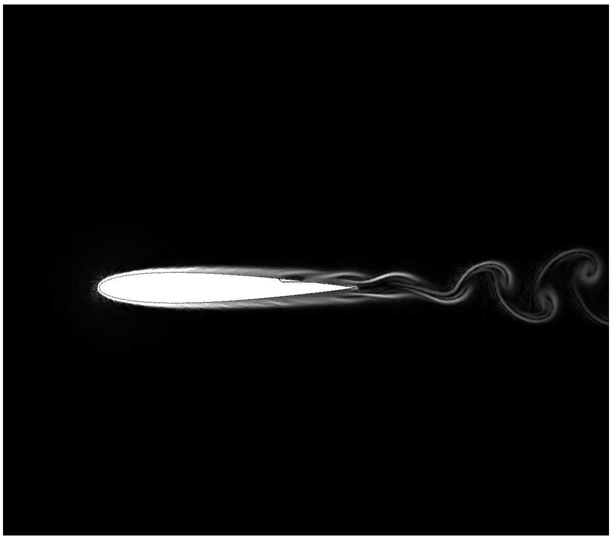
[2] Stein K, Benney R, Kalro V, Tezduyar TE, Leonard J, Accorsi M. Parachute fluidâĂŞstructure interactions: 3-D computation. *Computer Methods in Applied Mechanics and Engineering*, Vol. 190, No. 3-4, pp 373-386, 2000.

[3] Favre F, Antepara O and Lehmkuhl O and Borrell O and Oliva A. On the fast transient spoiler deployment in a naca0012 profile using les techniques combined with amr and imb methods. *Proceedings of the 11th World Congress on Computational Mechanics, WCCM XI*, July 20-25 Barcelona, Spain, pp 1-10, 2014.

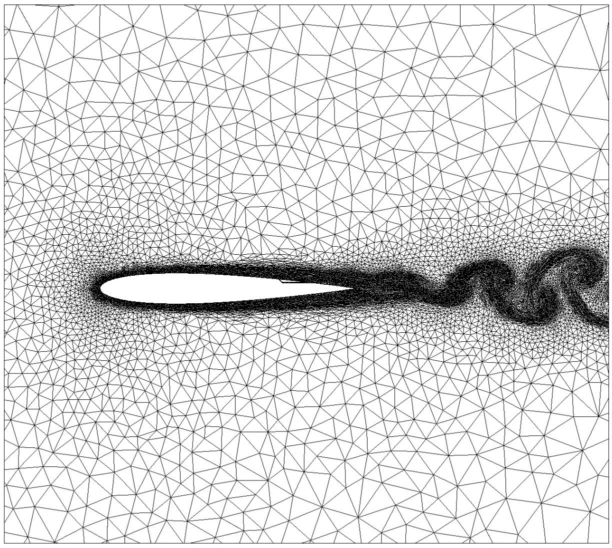
[4] Dindar M, Shephard M, Flaherty J, Jansen K. Adaptive CFD analysis for rotorcraft aerodynamics. *Computer Methods in Applied Mechanics and Engineering*, Vol. 189, No. 4, pp 1055-1076, 2000.

[5] Lesoinne M, Farhat C. Stability analysis of dynamic meshes for transient aeroelastic computations. *Proceedings of the 11th Computational Fluid Dynamics Conference, AIAA-33-3325*, Orlando, Florida, USA, pp 309-329, 1993.

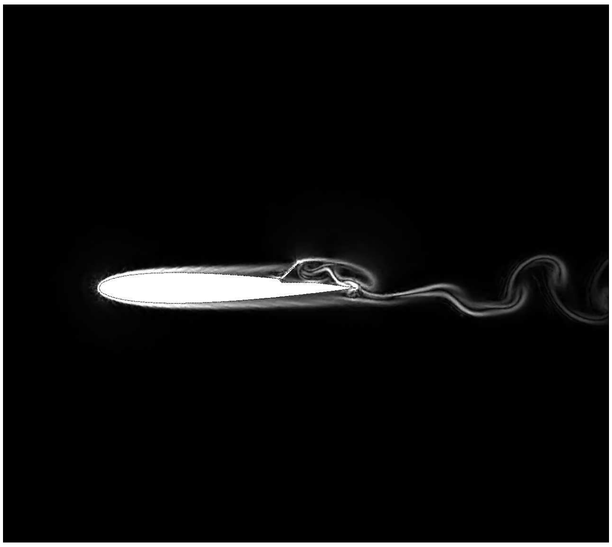
[6] Habashi WG, Dompierre J, Bourgault Y, Ait-Ali-Yahia D, Fortin M, Vallet MG. Anisotropic Mesh Adaptation: Towards User-Independent,



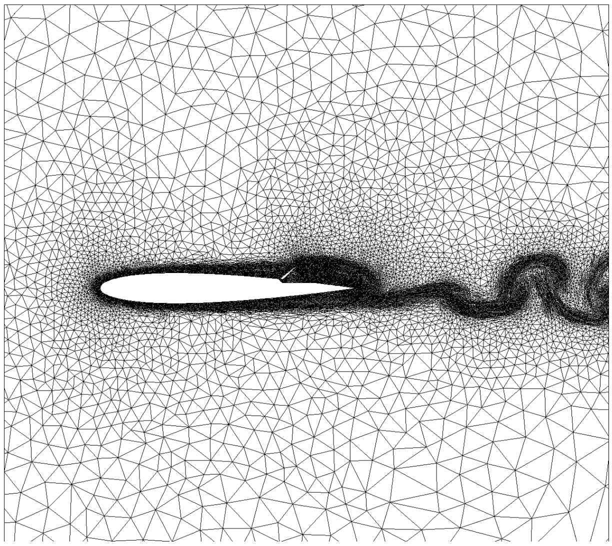
(a) $|\nabla\rho|$ for $\beta = 0$



(b) Mesh

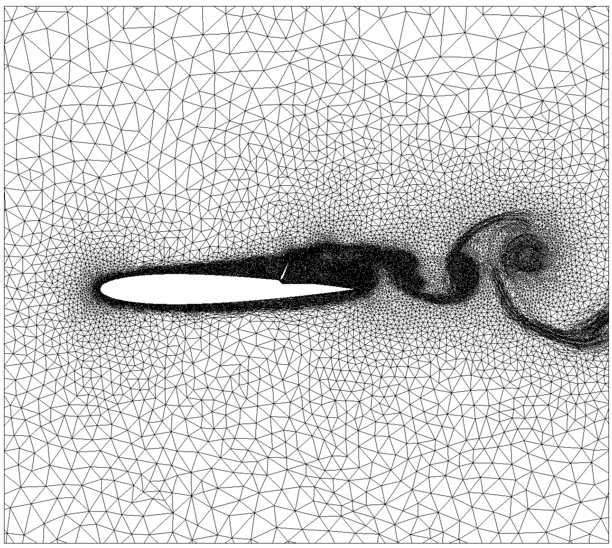


(c) $|\nabla\rho|$ for $\beta = 45^0$



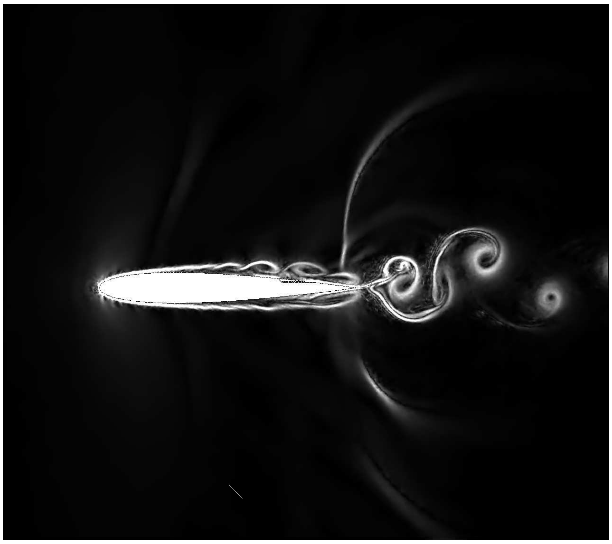
(d) Mesh

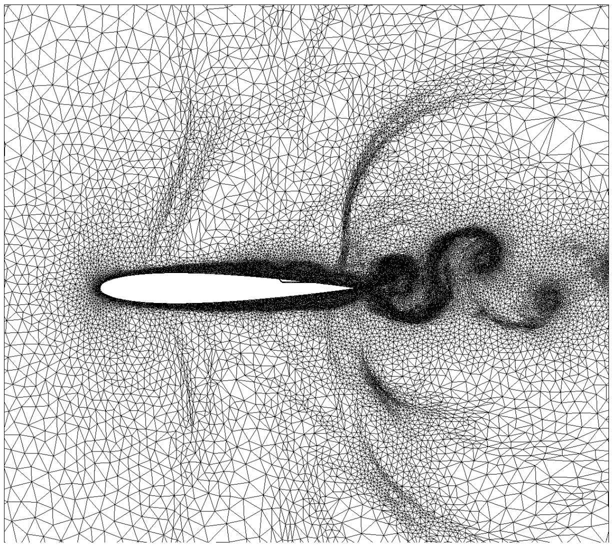
(e) $|\nabla\rho|$ for $\beta = 70^0$



(f) Mesh

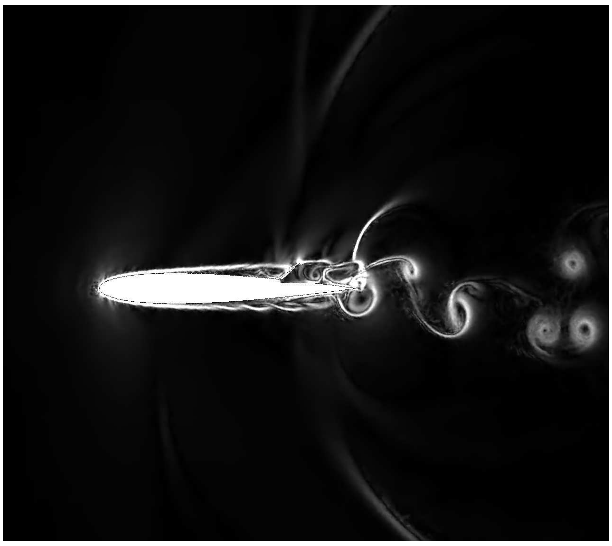
Fig. 2 : Magnitude of the specific mass gradient and adapted meshes for the subsonic case with $Mach = 0.4$

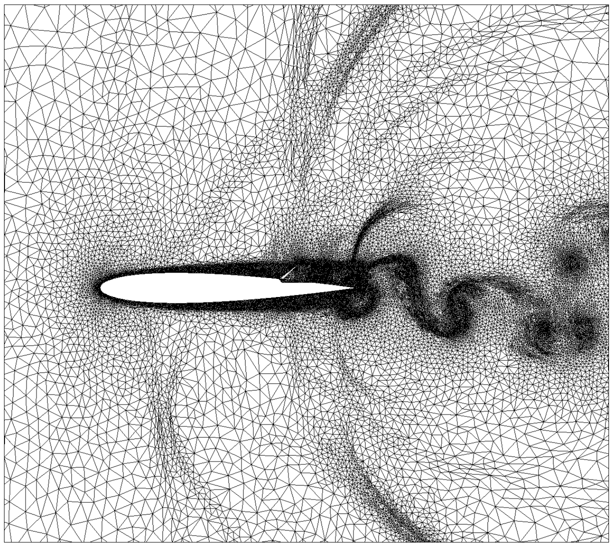




(a) $|\nabla\rho|$ for $\beta = 0$

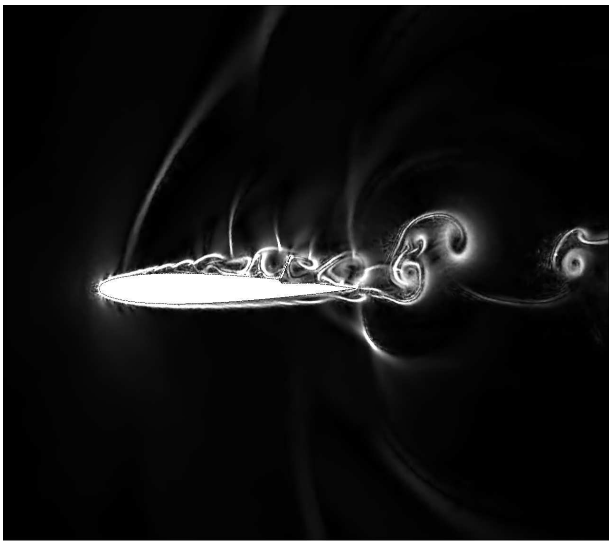
(b) Mesh

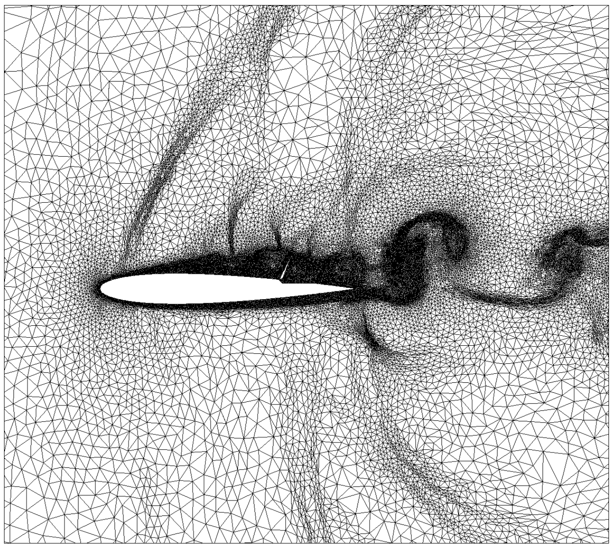




(c) $|\nabla\rho|$ for $\beta = 45^0$

(d) Mesh

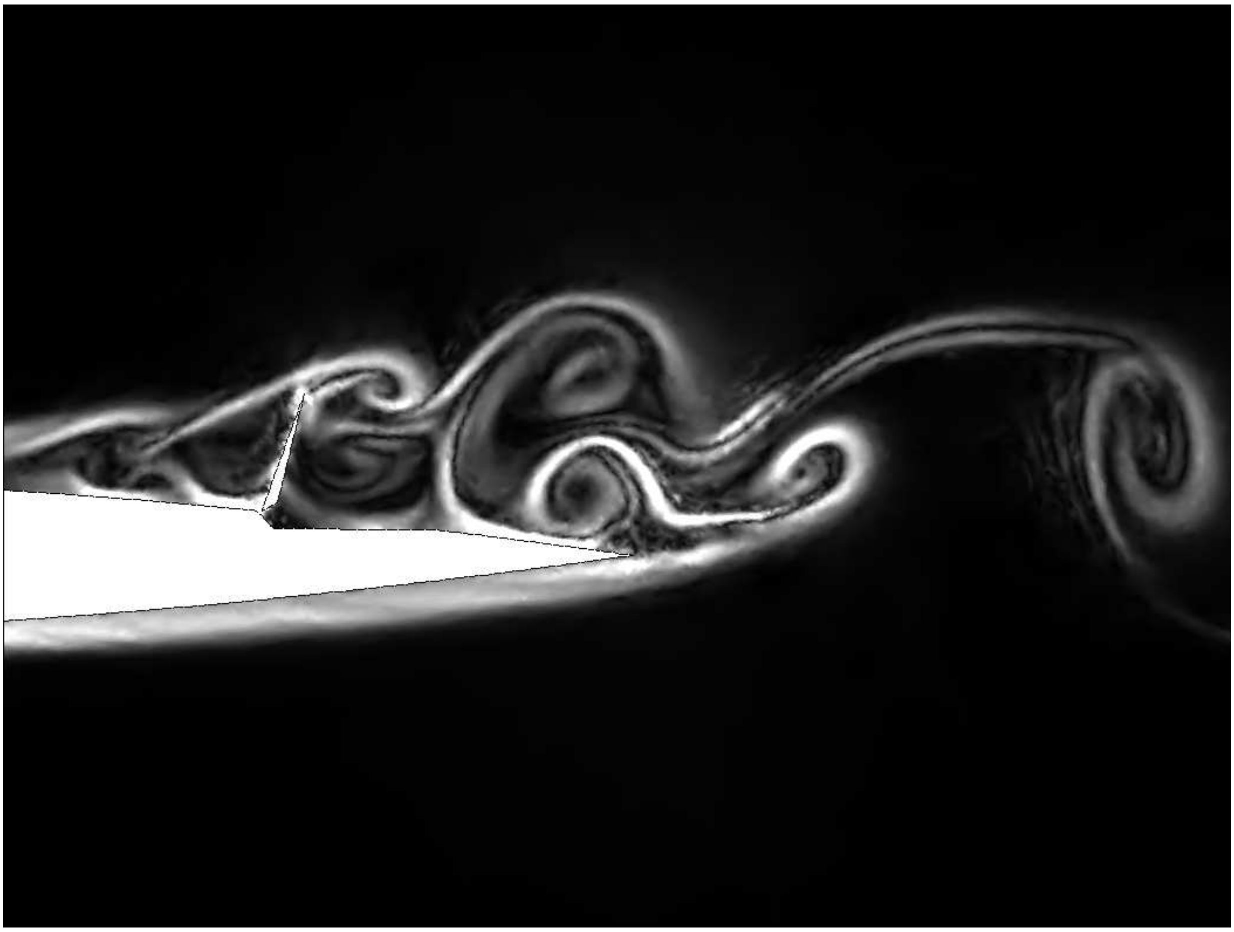




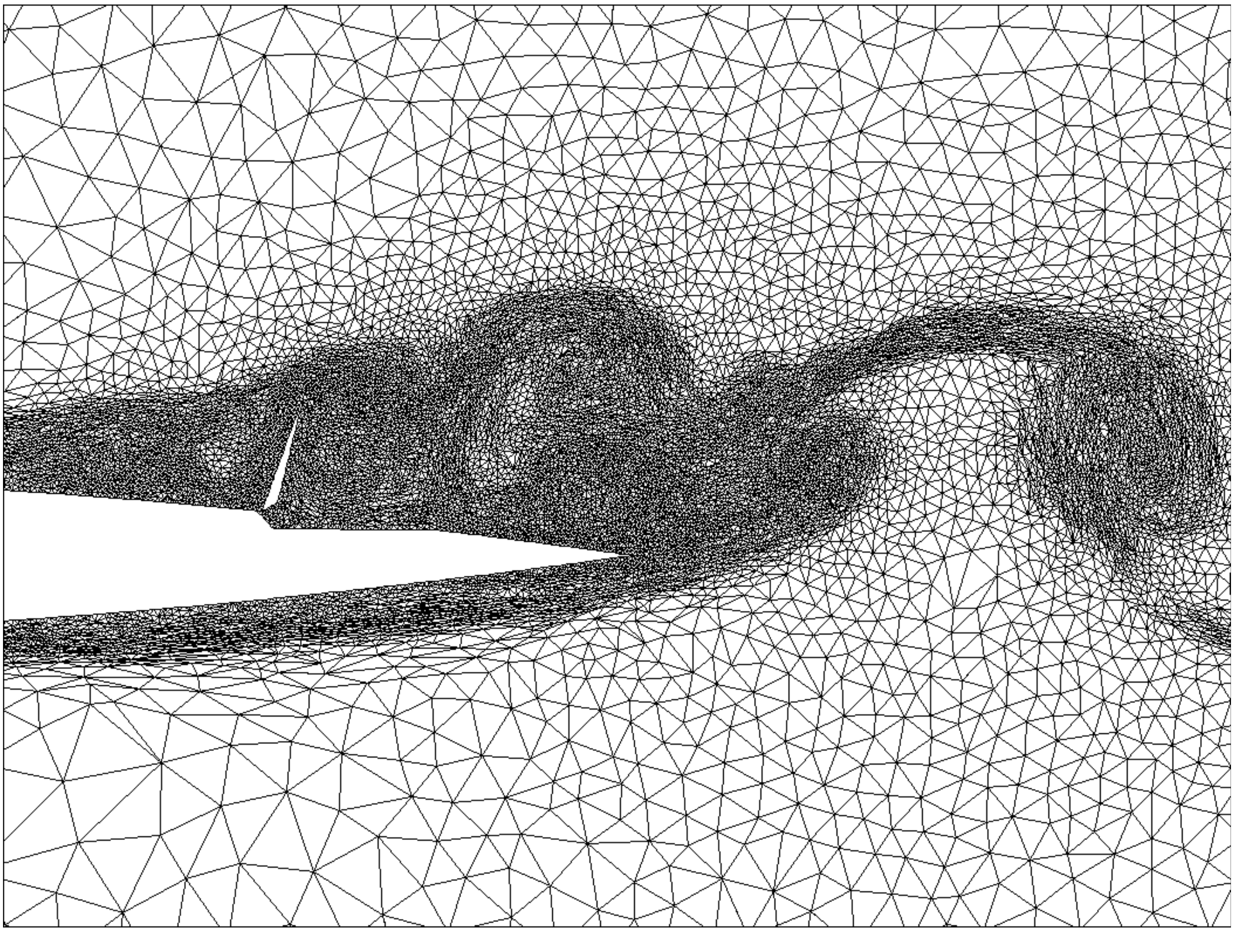
(e) $|\nabla\rho|$ for $\beta = 70^0$

(f) Mesh

Fig. 3 : Magnitude of the specific mass gradient and adapted meshes for the transonic case with $Mach = 0.8$

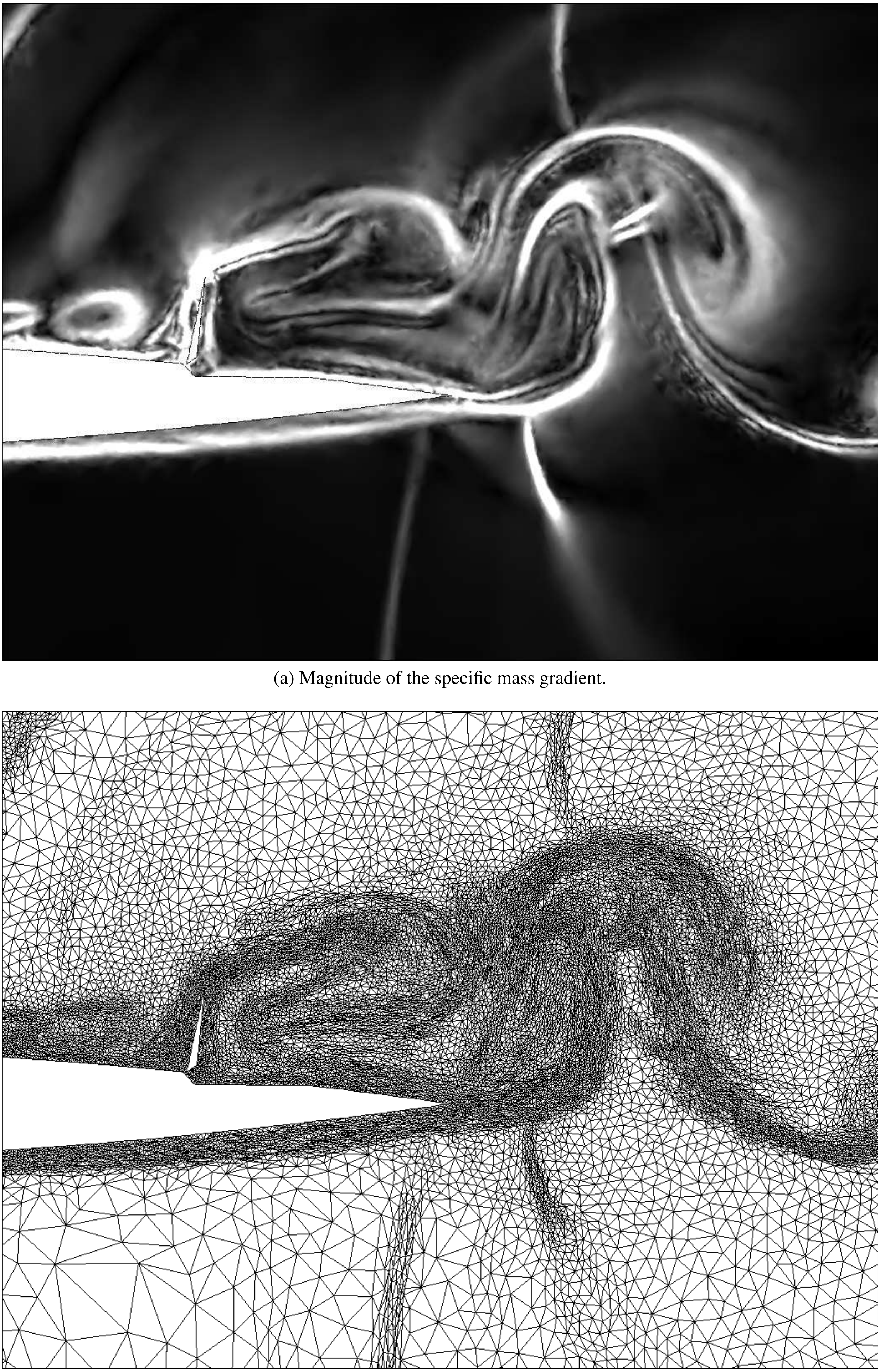


(a) Magnitude of the specific mass gradient.



(b) Mesh

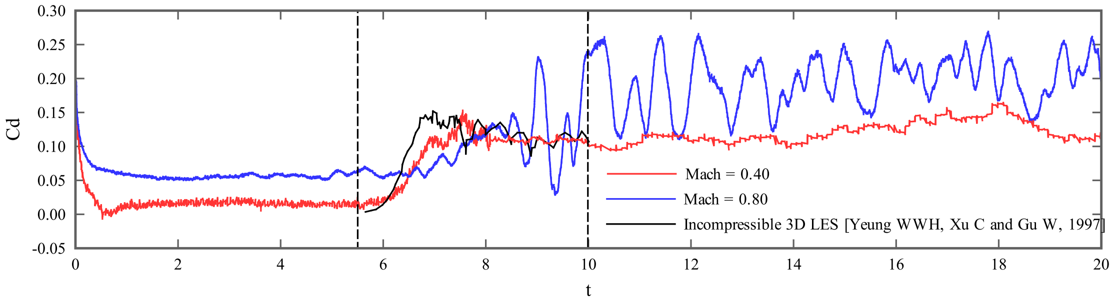
Fig. 4 : Mesh for the subsonic case with $Mach = 0.4$ and $\beta = 90^0$.



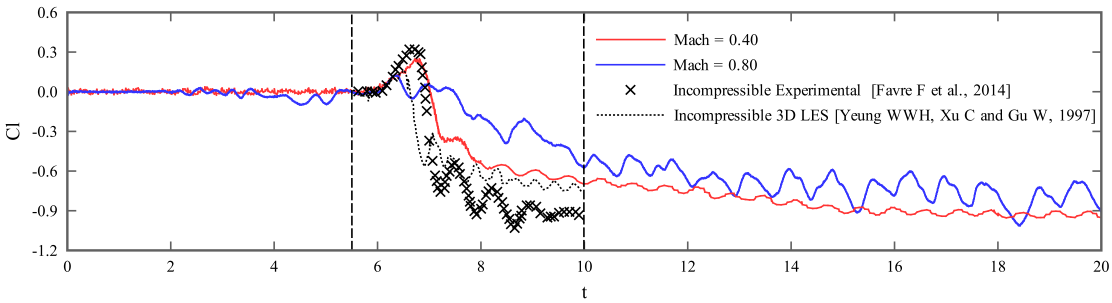
(a) Magnitude of the specific mass gradient.

(b) Mesh

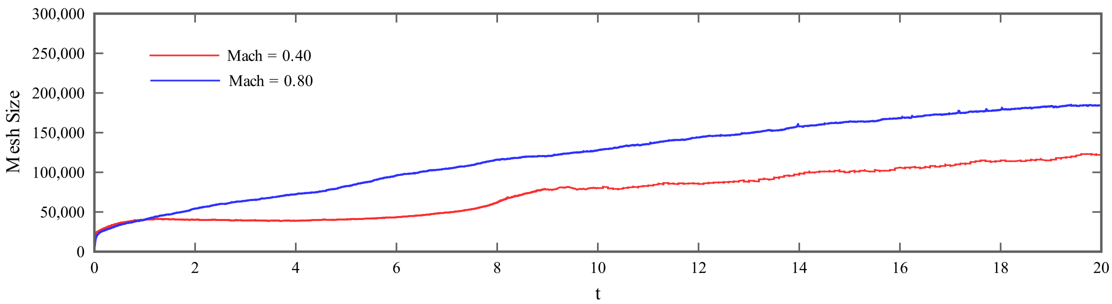
Fig. 5 : Mesh for the transonic case with $Mach = 0.8$ and $\beta = 90^0$.



(a) Transient drag coefficient



(b) Transient lift coefficient



(c) Mesh size

Fig. 6 : Transient solution for the spoiler deployment in the NACA-0012 Airfoil.

Mesh-Independent and Solver Independent CFD. Part I: General Principles. *International Journal for Numerical Methods in Fluids*, Vol. 32, pp 725-744, 2000.

[7] Sagaut P, Deck S. Large eddy simulation for aerodynamics: status and perspectives. *Philosophical Transactions of the Royal Society A*, Vol. 367, pp 2849-2860, 2009.

[8] Manoha E, Troff B, Sagaut P. Trailing-edge noise prediction using large-eddy simulation and acoustic analogy. *AIAA Journal*, Vol. 38, pp 575-587, 2000.

[9] Raverdy B, Mary I, Sagaut P, Liamis N. High resolution large-eddy simulation of the flow around a low-pressure turbine blade. *AIAA Journal*, Vol. 41, pp 390-397, 2003.

[10] Linn RV, Awruch AM. Large Eddy Simulation of Turbulent Compressible Flows Using the Characteristic Based Split Scheme and Mesh Adaptation. *Proc. of the VII European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2016)*, Hersonissos, Crete Island, Greece, pp 1-15, June 2016.

[11] Zienkiewicz OC, Codina R. A general algorithm for compressible and incompressible flow – part I: The split, characteristic-based scheme. *International Journal for Numerical Methods in Fluids*, Vol. 20, pp 869-885, 1995.

[12] Nithiarasu P, Codina R, Zienkiewicz OC. The Characteristic-Based Split (CBS) scheme– a unified approach to fluid dynamics. *International Journal for Numerical Methods in Engineering*, Vol. 66, pp 1514-1546, 2006.

[13] Linn RV, Awruch AM. Edge-Based Anisotropic Mesh Adaptation of Unstructured Meshes With Applications to Compressible Flows. *Engineering With Computers*, Vol. 33, No. 4, pp 1007-1025, 2017.

[14] Castro-Díaz MJ, Hecht F, Mohammadi B, Pironneau O. Anisotropic unstructured mesh adaptation for flow simulation. *International Journal for Numerical Methods in Fluids*, Vol. 25, pp 475-491, 1997.

[15] Frey PJ, Alauzet F. Anisotropic mesh adaptation for CFD computations. *Computer Methods in Applied Mechanics and Engineering*, Vol. 194, pp 5068-5082, 2005.

[16] Loseille A, Alauzet F. Optimal 3D highly anisotropic mesh adaptation based on the continuous mesh framework. *Proceedings of the 18th International Meshing Roundtable*, Springer-Verlag, Salt-Lake City, UT, USA, pp 575-594, 2009.

[17] Alauzet F, Frey PJ, George PL, Mohammadi B. 3D transient fixed point mesh adaptation for time-dependent problems: Application to CFD simulations. *Journal of Computational Physics*, Vol. 222, pp 592-623, 2007.

[18] Batina J. Unsteady Euler Airfoil Solutions Using Unstructured Dynamic Meshes. *AIAA Journal*, Vol. 28, No. 8, pp 1381-1388, 1990.

[19] Benek J, Buning P, Steger J. A 3D Chimera grid embedding technique. *Proceedings of the 7th AIAA Computational Fluid Dynamics Conference, AIAA-1523*, Cincinnati, OH, USA, 1985.

[20] Löhner RL, Baum JD. Adaptive h-refinement on 3D unstructured grids for transient problems. *International Journal for Numerical Methods in Fluids*, Vol. 14, pp 1407-1419, 1992.

[21] de Boer A, van der Schoot MS, Bijl H. Mesh deformation based on radial basis function interpolation. *Computers & Structures*, Vol. 85, pp 784-795, 2007.

[22] Yeung WWH, Xu C and Gu W. Reduction of transient adverse effects of spoilers. *Journal of Aircraft*, Vol. 43, No. 4, pp 478-484, 1997.

Contact Author Email Address

Renato Vaz Linn: renatolinn@ufrgs.br

Copyright Statement