INSTITUT DE FRANCE
Académie des sciences

Comptes Rendus

Mécanique

Jean Paul Bonnet

Experimental and Computational Fluid Dynamics: decades of *turbulent* EFD/CFD complementarity

https://doi.org/10.5802/crmeca.113

Part of Special Issue: More than a half century of Computational Fluid Dynamics
Guest editor: Mohammed El Ganaoui (Professeur des Universités, Spécialiste : Mécanique des Fluides et Transferts de Chaleur et de Masse, Université de Lorraine)

CENTRE MERSENNE

Les Comptes Rendus. Mécanique *sont membres du*
Centre Mersenne pour l'édition scientifique ouverte
www.centre-mersenne.org
e-ISSN : 1873-7234

Comptes Rendus
Mécanique
Published online: 3 October 2022
`https://doi.org/10.5802/crmeca.113`

More than a half century of Computational Fluid Dynamics / *Plus d'un demi-siècle de mécanique des fluides numérique*

Experimental and Computational Fluid Dynamics: decades of *turbulent* EFD/CFD complementarity

Mécanique des fluides expérimentale et mécanique des fluides numérique : des décennies de complémentarités en écoulement turbulent

Jean Paul Bonnet[a]

[a] Institut Pprime, UPR3346 CNRS—Université de Poitiers—ISAE/ENSMA, Site du Futuroscope Bat. H2 TSA 51124, 86073 POITIERS cedex 9, France
E-mail: jean-paul.bonnet@univ-poitiers.fr

Abstract. An overview of the parallel evolution of computations and experiments in fluid dynamics in the context of turbulent flows is given, viewed from an experimental side. It is evident that experiments can no longer be seen as only validations for computations and that forecasting for "numerical wind tunnels" is not yet valid. The growing evidence of organized motions in turbulent flows pushes in parallel both experiments and numerical approaches to develop new measurement technologies and numerical methods. Many tools and concepts are shared both by experiments and computations. These approaches appear to be quite complementary and both communities will gain from the mutual fertilization.

Résumé. Une mise en perspective de l'évolution parallèle des calculs et des expériences en dynamique des fluides en régime turbulent est présentée depuis un point de vue d'expérimentateur. Il apparait que les expériences ne peuvent pas être considérées comme de simples éléments de validation des calculs et que la prédiction de l'avenir des « souffleries numériques » n'est actuellement pas pertinente. La mise en évidence de l'importance du caractère organisé des écoulements turbulents a conduit au développement quasi parallèle des technologies des approches expérimentales et des méthodes numériques. De nombreux outils et concepts sont partagés par les expérimentateurs et les numériciens. Ces approches apparaissent très complémentaires et les deux communautés devraient profiter de leurs complémentarités pour accroitre les échanges dans une fertilisation mutuelle.

Keywords. Computational fluid dynamics, Experiments, Turbulence, DNS, LES, HWA, PIV.

Mots-clés. Mécanique des fluides numérique, Expériences, Turbulence, DNS, LES, HWA, PIV.

ISSN (electronic) : 1873-7234 `https://comptes-rendus.academie-sciences.fr/mecanique/`

1. Introduction

In the following, we will present some salient features of the evolution of Experimental and Computational Fluid Dynamics (EFD and CFD, respectively) in the context of turbulent regimes. A brief historical perspective will be given and the main mutual influences and limitations will be presented, based on several decades of personal experimental practice, without trying to be exhaustive and with somewhat limited and arbitrarily selected references.

In the eighties, the future in Fluid Mechanics was clearly the “Virtual Wind Tunnel”, formalized for example by Bryson and Levit [1]. Later, Matsumo [2] introduced again the concept of “Numerical Wind Tunnel”. With some hindsight in time we will discuss the validity of these conjectures.

Presently, we concentrate on the discussion of turbulent flows. Indeed, the turbulent regime is encountered in most natural and industrial situations and remains one of the main modelling goals from both the fundamental and the applications point of view, using experimental as well as computational approaches (and more and more combined). It is well known that the turbulent flows always evidence 3D and unsteady characters with a wide range of physical scales inherent to turbulent regimes, with random and sometimes more deterministic behaviours. The turbulent character corresponds to significant fluctuations of the velocity, pressure, temperature and density that drives energy from the mean flow down to dissipation and impacts many engineering characteristics such as mixing, combustion, noise etc. Most of these effects have strong environmental impacts. The organization in space and time of any turbulent flow forces the consideration of both large-scale (energetic, organized) as well as small-scale (dissipative, random) eddies. This implies several limitations for experimental methods as well as computational approaches. These limitations are indeed different in nature for EFD and CFD. We will bear in mind some of those constraints, and will evidence that EFD and CFD should be considered today as complementary methods.

In some sense, the specific physics of turbulent flows establishes a link between experiments and computations, sharing the complexity of the phenomena and the specific knowledge of the associated physics and theories, but also carries their complementary advantages and drawbacks. These specificities of turbulence remain the driving forces and the scientific and technical challenges of both measurement techniques and numerical simulations.

In addition to the turbulent characteristics, some areas of Fluid Mechanics associate some extra complex physics, such as compressibility for high speed flows, plasmas, non-Newtonian behaviour, multiphase flows etc. When industrial applications also (heat exchangers, turbomachinery) or environmental flows are considered, heat transfers and buoyancy are of paramount importance and the turbulent temperature and density fields have to be considered. For such research areas, sometimes under rapid development, common EFD and CFD approaches are required. Let us recall, as an illustration, the well-known example of the compressibility effects on the expansion rate of supersonic mixing layers. This effect has been historically evidenced from experiments (Smits and Dussauge [3]) and, later on, much better handled by theory and computation (Gatski and Bonnet [4]).

2. A parallel evolution

2.1. *General considerations*

There is a somewhat quasi-parallel evolution—and co-fertilization—of EFD and CFD. In short, historically both methods start by exploring the mean quantities based on the observation or modelization of Reynolds (or Favre, if compressibility matters) averaged fluctuations. Unless direct visualizations were available for a while (from Da Vinci schematic views to *turbulent boundary layers* (TBL) by Falco [5] or mixing layers by Brown and Roshko [6]), quantitative measurements of fluctuating fields were essentially based on Hot Wire Anemometer (Comte-Bellot [7], Bruun [8], Tropea *et al.* for a review [9]). These experimental data have then been extensively used for the validation of CFD codes based on Reynolds Averaged Navier Stokes equations (RANS) (Hanjalic and Launder [10]). One illustrative example of the use of EFD results for CFD validations is found in the proceedings of the Stanford Conference on Complex Turbulent Flows held in 1981 [11].

Hussain and Reynolds [12] introduced, as early as in 1970, the triple decomposition of the fluctuating fields, anticipating the evidence for the importance of the organized character of turbulent flows (Cantwell [13]). This aspect has been formalized through many experiments, in particular with the pioneering paper “Coherent structures: reality and myth” by Hussain [14]. The evidence of organized motions in turbulent flows were associated with the development of lasers and cameras, in the eighties. Particularly the development of Particle Image Velocimetry (PIV) accompanied this description of turbulent flows, providing both statistical quantities and spatial organization (Adrian [15]). A wide community of experimentalists devoted their research to the development of this diagnostic (Stanislas *et al.* [16]).

In parallel, CFD was devoted to RANS closure models based on statistical descriptions as mentioned before. Theoreticians and CFD researchers rapidly started exploring order in chaos (Bergé [17]), and they recognized that non-universal large scales were important in flow dynamics. These concepts were concomitant with the evolution of CFD towards the Large Eddy Simulations (Lesieur *et al.* [18]). It should be mentioned that the need to consider the organized motions in turbulent flows has also been evoked early by Ha Minh as “semi-deterministic” [19]. Other methods involving large-scale flow organizations such as unsteady RANS approaches (URANS), and hybrid methods (Detached Eddy Simulations and their developments, Lattice-Boltzman etc.) are still currently under development as pointed by Sagaut [20] and need several levels of physical analysis of turbulent phenomena. The direct numerical resolution of the Navier-Stokes equations, without any model, opens unique and complete views of all the descriptions of turbulent flows in terms of statistics, large-scale organization etc. (Moin and Mahesh [21]). However, direct numerical simulation (DNS) are generally restricted to “basic” flows with simple geometries. Contrarily, the other CFD methods cited ahead are designed for complex flows, typically of industrial interest, and can cover most of the EFD test cases.

During these evolutions, the two communities were faced with the question of the definition of the so-called “Coherent”, often “Large Scale”, sometimes “Energetic” structures, the Fourier transform being not adapted to non-homogeneous flows. Identification processes are needed to extract those organized motions inside the phase random turbulent signals. Stochastic methods such as Proper Orthogonal Decomposition, and Conditional Samplings, are data post-processing methods shared between EFD and CFD (Bonnet and Glauser [22]). Wavelet transform is also a powerful mathematical tool used for data processing for EFD and CFD, typically for data processing and coherent vortex simulations (Farge [23], Farge *et al.* [24]). Later, for acoustic applications, the concept of wave packets has been introduced by Jordan and Colonius [25], here also based on both experimental and numerical results.

These quasi-parallel evolutions are linked, and promote progress in theoretical physics, applied mathematics, data processing and storage capabilities, which are shared by both approaches. The rapid innovations in laser and cameras technologies for EFD, computers for CFD and data storage capabilities for both appoaches are indeed directly motivated by the demand of the scientific as well as the industrial communities.

2.2. *Some contribution of EFD to CFD*

In the following, we will discuss some inputs that experiments can provide to computations.

As mentioned earlier, for a long time, experiments have provided only the mean values and averaged fluctuations (rms). These data are obviously of interest for all types of CFD for initial conditions and for the validation of the results. With the introduction of time-resolved apparatus such as Hot Wire Anemometry (HWA), the one point spectra and two (or more) time and space correlations are available to develop turbulence modelling and validate time-resolved CFD. Some CFD require implementing new quantities (such as higher-order moments for instance) that will often correspond to a complete reconsideration of the entire experimental work and data processing.

As stated earlier, for a long time, flow visualizations have also been used for global investigations to better understand the flow physics; more recently, quantitative visualizations available from PIV in particular have become essential data for validation and improvements in computation. The development of rakes of hot wires (Citrinity and George [26], Tutkun *et al.* [27]) associating a good spatial resolution with an excellent time resolution, approaches in some sense the DNS results. Presently, time-resolved particle image velocimetry PIV (TRPIV) and Tomographic PIV provide information not only on the flow organization in space and its statistics, but also on the dynamical behaviour of the flow organization for all the CFD approaches.

The experiments also provide real initial, boundary and external conditions such as wall proximity in wind tunnels. These elements can be essential for CFD. Compared to most computations, EFD measurements are able to collect long time series, and can then capture rare events in the time series. This can be very useful for several industrial applications.

Indeed, EFD still suffers from limitations due to the limited extend of the diagnostic capabilities. Most measurements are limited to single points or slices of the flows, and, although providing excellent statistics, are often limited in terms of time resolution and dynamic range. For pressure measurements, the non-local character of the information adds some complexity to the data analysis. Also, EFD suffers from limitations in terms of simultaneity of the collection of different quantities. As an example, some signals are continuously sampled (e.g. Hot Wire or Pressure measurements) when the PIV is often acquired at a low sampling rate, (allowing to consider the samples as statistically independent when the turbulent characteristics are concerned), or can be triggered by some events. Consequently, the entire flow description in space and time is still unachievable from experiments with an adequate resolution. A possible solution, which will be detailed later, could be to combine EFD and CFD methods in a joint approach.

To conclude this part, it is important to mention that EFD measurements, despite their limitations, provide results that are difficult to compute: flows with complex geometries and complex physics (for example with solid particles, plasma or rarefied gas effects…).

2.3. *CFD contribution to the development of EFD*

CFD can be essential for the design of experiments, such as defining wind tunnel characteristics, model implementations in the test sections, design of test models (with sensors and actuators in case of control), model holding devices, wall effects and corrections (particularly in transonic regimes), model deformation compensations etc. These inputs are particularly important when experiments are performed in large-scale facilities. More details on these CFD contributions to experiments can be found in Chanetz *et al.* [28].

Some turbulent flow characteristics are not so simple to evidence via EFD and have been described via CFD. This is the case of very large Reynolds number (Re) effects on TBL (Marusic and Smits [29], Smits *et al.* [30]). Indeed, the usual Re ranges in academic wind tunnels are of

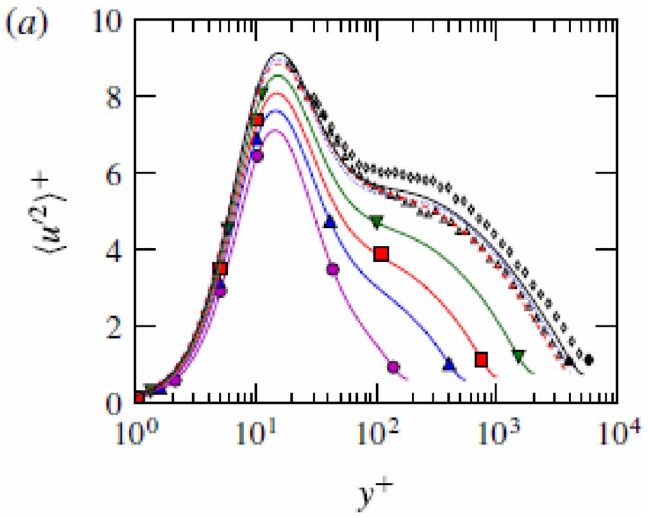


Figure 1. Evolution of the longitudinal velocity fluctuations with the Re. DNS from Lee and Moser [32] with permission. Data in viscous units. The bulk Re (based on the channel half-width) ranges between 3000 (magenta) and 143,000 (black).

the order of 10^6, ten times higher in large facilities such as the ONERA Modane ones, and can be compared with the 100×10^6 for airplane cruise conditions. Increasing the size of wind tunnels is not the only way to increase the Re. One can mention that high-pressure tunnels (among others the 100 bar, Schewe *et al.* [31], Princeton super pipe, for a review see Smits *et al.* [30]) allow for ultra high Re numbers; low-temperature wind tunnels can also reach very high values as is the case with the European Transonic Wind Tunnel (ETW) in Cologne. However, these facilities are very unique and of limited access for academic research. These Re number effects are somewhat difficult to quantify experimentally but some DNS (then model free) showed the modification of the distribution of the Reynolds stresses for very high Re. As shown in Figure 1 for example, the increase in the level of the velocity fluctuations in the outer part of the boundary layer is evidenced (Lee and Moser [32]). These effects are quite important where skin friction drag control is concerned for aeronautical applications (Agostini and Leschziner [33]). Then, when validated at Re available from experiments, CFD can extrapolate to higher ranges, sometimes with more representative values for industrial applications.

Other parameters are not so easy to measure, such as loads, skin friction, heat exchanges etc. By comparison, such parameters are easier to evaluate with CFD. The example of skin friction reduction given later is a good illustration. As another example, for some very complex flows with delicate experimental access, such as the Internal Combustion Engines, the association of EFD and CFD is essential, individually each approach being insufficient (Borée and Miles [34]).

2.4. *Common tools, concepts or limitations*

For turbulent flows, CFD and EFD share several constraints and tools. For example, the specific data processing which are necessary to extract and characterize the Coherent Structures (CS) evoked before are common to both approaches. In addition, the data restitution (3D, coloured images, time evolutions) is a common challenge. The huge storage requirements that are well known for highly resolved CFD such as DNS are now shared by most of PIV experiments.

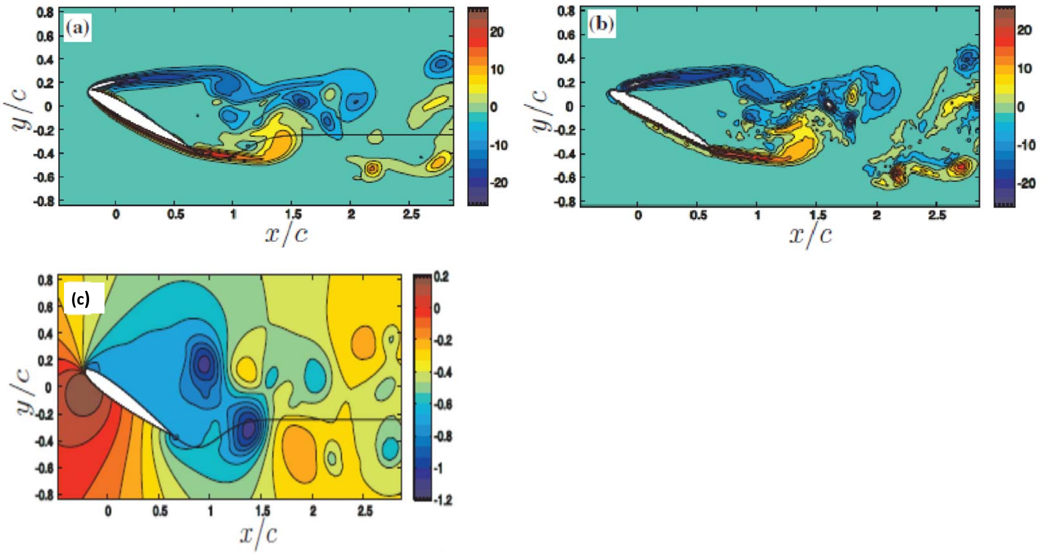


Figure 2. (a) Iso vorticity around an airfoil from Hybrid (DNS/PIV) computation, (b) PIV results, (c) pressure field issued from the hybrid method (after Suzuki *et al.* [39] with permission).

For many purposes, the description of turbulent flows with *low-order model* is essential for stability analysis, physical interpretations and flow control. Such requirements are indeed shared by both numerical and experimental methods. Introduced by Berkooz *et al.* [35], the proper orthogonal decomposition (POD) is a typical example of a low-order description. Many other methods are now in use for such a purpose (Rowley and Dawson [36]).

CFD can complement the missing data issued from experiments. Despite the progress in measurement methods, as shown earlier, the experimental results are still limited. Data assimilation techniques are developed allowing the recovery of mean flow fields that are solutions of RANS (Beneddine *et al.* [37]). Among these approaches, the DNS simulations that include PIV/PTV in such hybrid DNS simulation sequentially updates the Navier–Stokes equation by forcing time-resolved PIV/PTV data with a synchronized time-step (Suzuki *et al.* [38, 39]). This method assimilates the DNS solution into the PIV/PTV flow field and can simultaneously produce an unsteady hydrodynamic pressure field unavailable from experiments as shown on Figure 2.

2.5. *Both EFD and CFD still share open questions*

As mentioned earlier, the characterization of *initial* and *boundary conditions* is a key ingredient for EFD for the reproduction of experiments and for CFD for introducing correct initial and boundary conditions particularly when turbulent and transitional flows are concerned. For experiments, the very low level of external turbulence in good quality wind tunnels is difficult to measure, because it often approaches the background noise level of the sensors. In addition, for precise introduction of the turbulence level in CFD, the spectral behaviour *and* the spatial scales can be required, these characteristics being also sometimes available from experiments, but being quite difficult to measure at low fluctuation levels.

However, the experiments are able to produce long series of data, with limitations in the available flow characteristics needed to initialize CFD. It is possible to generate complete time

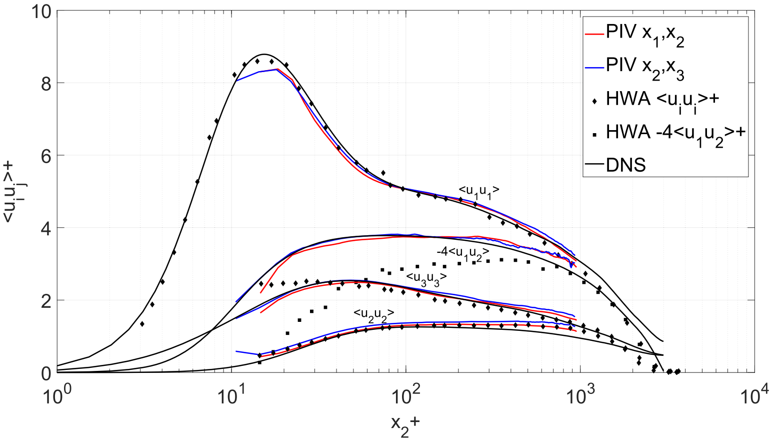


Figure 3. Comparisons among HWA, PIV measurements and DNS in the wall vicinity in a TBL at $\mathrm{Re}_\Theta = 7634$ (based on momentum thickness). x^+ is the wall distance expressed in viscous units (after Foucaut *et al.* [41] with permission).

series based on spectra obtained from experiments or based on some low-order models (Perret *et al.* [40]).

The studies on TBL require solving the spatial scales very close to the wall, typically of a few wall units. This is very demanding for measurements, the hot wires being more intrusive and their integration effects more present with parasitic heat exchanges with the wall. PIV measurements close to walls are also quite delicate due to light reflections. For CFD, the spatial resolution requires very fine meshes close to the wall, thus increasing the cost of the computation, and, for some methods, requiring specific modelling. Representative evidence of the essential complementarity of EFD and CFD is given in Figure 3 (Foucaut *et al.* [41]). Based on the DNS model free results, the different measurement limitations can be evaluated: better results for longitudinal velocity fluctuations from HWA (single wire operation), less at wall proximity for the other data from HWA (crossed wire operation) or PIV.

The *transition* studies are also of crucial interest for most turbulent research. For experiments, transition is influenced by many parameters that are often difficult to control and sometimes difficult or impossible to precisely reproduce (wall smoothness, external perturbations including acoustic ones etc.). In addition, the characterization of the transition fronts is quite delicate both for experiments and computations. These parameters are often easier to control when CFD is concerned but somewhat less easy to rely upon with the experimental conditions. Then, EFD and CFD are strongly complementary for transition studies, and the development of the local correlation-based transition models (LCTM) will benefit from the detailed knowledge of the spatio-temporal characteristics of the background turbulence (as an example, see Rubino *et al.* [42]).

In unsteady flow configurations, both EFD and CFD provide huge quantities of data that are too often used only for the analysis of averaged characteristics. When unsteady phenomena are concerned, the interactions between different scales, including the dissipative ones, are relevant and should be used for better physical understanding of the turbulent phenomena. The dynamics of the tip vortex (Jacquin *et al.* [43]) is a typical example in which the unsteadiness of the formation and the development of longitudinal eddies play an essential role in the aerodynamical phenomena.

Both EFD and CFD are subject to *uncertainties*. For experiments, as already mentioned, the HWA has limitations in spatial resolution, and frequency response (particularly for high-velocity flows). PIV uncertainties are related to the interrogation area, pixel resolution and, seeding bias dues to non-uniformity of seeding and particle lag effects. Both these experimental methods are also sensitive to data acquisition biases such as frequency sampling for time-resolved diagnostics or independence of samples for PIV. When high speed flows (from transonic to hypersonic) are concerned, both HWA and PIV suffer from limitations: shock wave in case of intrusive HWA and particle inertia (Knundsen Number) in case of shocked flows for PIV (Smits and Dussauge [3], Gatski and Bonnet [4]). A large panel of the different sources of uncertainties can be found in Tropea *et al.* [9]. Another source of scatter can be the wind tunnel conditions themselves. A typical example is the spreading rate of the supersonic mixing layer already evoked. Over decades, several groups have measured this quantity with a huge scatter, essentially due to wind tunnel effects (external turbulence level, presence of acoustic modes, wall proximity, parasitic shock waves etc.). Despite this scatter, a global trend has been achieved (and validated by DNS but is more delicate to compute with model-based CFD). Experiments, even with large uncertainties, are able, and sometimes the only way, to evidence important physical mechanisms (see Smits and Dussauge [3], Gatski and Bonnet [4]).

As far as CFD are concerned, great care has to be taken for meshing, mathematical formulations, etc; as an example, both space and time characteristics have to be selected via the Courant CFD number. Several bibliographical references are available for best practices (Guertz [44]). Here also shocked flows are particularly delicate to compute via CFD (Gatski and Bonnet [4]).

A concluding remark is that, for both EFD and CFD communities, it is essential to perform the studies while understanding the *specific physics* of the turbulent regimes. Due to the rapid evolution of the commercial simulation codes and PIV software, the user can potentially process data without any or with limited background in turbulence. Bonnet and Qin [45] evidence that a knowledge of the physics of turbulent phenomena is mandatory to adequately perform both experiments and computations.

3. Flow control: an illustration of complementarity between EFD and CFD

3.1. *General considerations*

In order to illustrate the previous considerations, we present some illustrations of common and complementary contributions in the case of control of turbulent flows.

Flow control is a challenging and multi-disciplinary area of innovation for both experiments and computations and for many years now (among others, Gad-el-Hak *et al.* [46], Gad-el-Hak [47]). Indeed, specific challenges concern actuators, flow physics management and, particularly for turbulent regimes, optimization processes. Actuators require to consider different physical mechanisms such as micro flows, resonant cavities, burners, plasmas, piezo and micro electro mechanical systems (MEMS) etc. (Cattafesta and Sheplak [48]). The physical mechanisms can be as diverse as vortex generation, volume effects, thermal effects and may act on different scales of the flow, from large, energetic scales down to smaller, dissipative scales. This requires a detailed analysis of the turbulent characteristics of the flows under consideration (Bonnet and Qin [45]). These behaviours are sometimes analysed by stability theories and require advanced optimization methods (Noack *et al.* [49]). In this respect, low-order modelling is often used as well as Artificial Intelligence. For both communities, the data processing and optimization processes for flow control optimization are the same and the theory and numerical tools are entirely shared. For example, the optimization methods based on Low-Order Models, Genetic programming, machine learning (Brunton *et al.* [50]) are presently widely used both in experiments and computations.

All these innovative aspects are challenging for the experimental community. The demonstrators are developed at laboratory scale, then in general with very small dimensions of actuators and high frequencies. For example, a typical turbulent jet diameter in a research wind tunnel is of the order of a few cm while, for fluidic control methods, the tubing devices are of the order of mm, and frequencies of the order of kilohertz. These constraints can limit the availability of demonstrators. Comparatively, real-scale applications of propulsive jets in aeronautics are of the order of meters in diameter and with frequencies around hundreds of hertz. The, somewhat counter-intuitive, although more expensive to perform, real-scale experiments can be potentially easier to realize, due to the large dimensions and lower frequencies requirements for actuators.

On the other hand, for the computational community, the major difficulty is to obtain both an excellent resolution of the flow itself (that can be particularly delicate for TBL) and, at the same time, to simulate the actuator device with enough details such as, depending on the method, tubing, driving cavities, and the associated physics.

The optimization needs, in all cases, to adjust the activation parameters via parametric studies.

Some parameters can be easily (and with a minimum cost) adjusted in the case of EFD. As an example, it is easy to vary the angle of incidence of a wing in a Wind Tunnel, to vary the flow rate of pneumatic actuators, to change its frequencies and duty cycles, etc. Long experiments allow very detailed analysis and make it possible to consider the effect of rare events. However, as we will see later, the measurement of some effects of the control can be delicate, such as skin friction, loads etc. In addition, many control strategies being located at the wall, global measurements are difficult to perform simultaneously with the proximity of the actuators. By comparison, it can be difficult (time consuming and expensive) to modify the space distribution of actuators and sensors, particularly when flow controls on models are concerned.

For computations, it can be easy to check the influence of several parameters such as complex time laws for the actuation, actuator locations, and, of paramount importance for optimization, the sensor location. As an example, plasma actuators are developed by several experiments (Moreau [51]) but so many parameters are involved for optimization (location of the actuator, orientation, size, relative placement of the embedded and exposed electrodes, materials, applied voltage, frequency…) that CFD is required. However, such a flow is not so easy to compute due to the electro-hydrodynamical forces that require the introduction of experimentally determined forcing terms in the closure models, such as the density of charges, the Debye number etc. (Benard *et al.* [52], Brauner *et al.* [53]). Then a dual approach of this kind of configuration is essential.

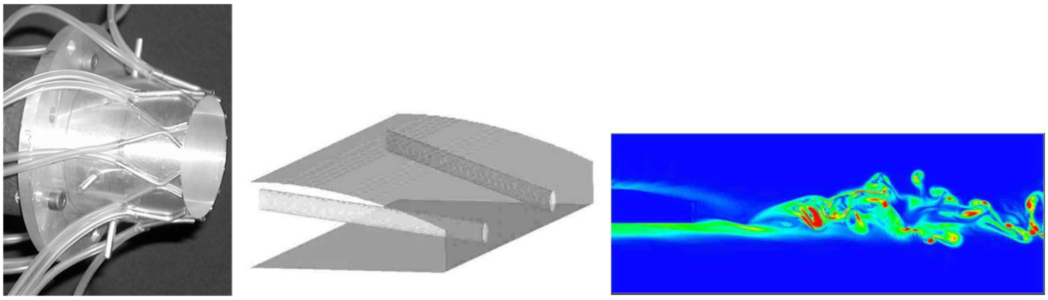
Indeed, optimization requires cost functions that are themselves easy to choose from CFD. However, some parameters are difficult to vary. For example, the modification of the geometries can require heavy work for meshing. As a difference to EFD, computations provide the entire flow field description, allowing a better analysis of the flow control physics.

Without looking for completeness, some examples in which EFD and CFD show their limitations and complementarities, are presented next.

Historically the major developments of flow control concepts originated from EFD. We will demonstrate that CFD is essential for optimization and validation including scaling validation.

3.2. *Examples of complementarities for flow control*

We first consider an example of *jet noise control* developed in the context of take-off noise reduction for airplanes. We focus here on a fluidic method, an active method derived from the well-known "Chevron" passive geometry. The demonstrator (Figure 4a) was built from a set of converging micro jets lying on the lip of the main jet (Laurendeau *et al.* [54]). The far field noise was easy to measure and, after some optimization, shows typically 1dB of noise reduction with this device. However, from the experiment, it was very difficult to analyse in detail the physics of

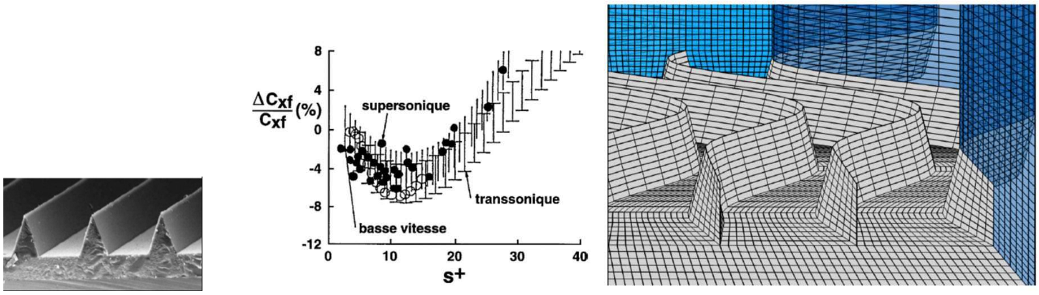


a) b) c)

Figure 4. Noise reduction by fluidic chevrons. (a) View of the experimental demonstrator (main jet diameter is 5 cm), (b) CFD geometry, (c) typical color map of the computed enstrophy (IBM-DNS), convergence plane of the control jets (after Laurendeau *et al.* [51]).

the interaction of the micro jet and the main jet, particularly due to the 3D unsteady character occurring in this part of the flow. This information is however essential for the interpretation and optimization of the noise reduction mechanisms. In addition, in the demonstrator, the scale of the micro jets was very small, less than 1 mm in diameter, making the detailed characterization of the jet's interaction hardly possible, as evoked earlier. On the other hand, it was very challenging to perform a complete CFD (here CAA for Computational Aero Acoustics, Bailly and Bogey [55]), associating a fine description of the hydrodynamic phenomena at the lip of the micro-jets, the local and global effects acting on the far field noise, the final input of the control. An Immersed Boundary Method-based DNS allowed to describe in detail the complex flow in the vicinity of the jet exit. This CFD was then limited to two converging micro jets penetrating into a jet flow, this location corresponding a priori to important noise sources (Figure 4b). These relatively limited computations did not allow to provide an entire description of the flow and associated sound effects, but were informative enough to provide a better understanding of the interaction mechanisms responsible for the noise reduction. Indeed, the DNS time series are not sufficiently long for the computation of turbulence statistics to be compared with the available statistics provided by the experiments, but the enstrophy visualizations shown in Figure 4c do give a sense of the control effect on the vorticity: compared to the uncontrolled case, the interaction between the control jets and the shear layer is found to lead to the generation of small scales. This effect remains localized in the jet convergence region $z = 0$ shown in the figure, with an increase in the enstrophy level compared to side regions where the vortex dynamics are found to be similar to the uncontrolled case. Such results allow for a better understanding of the impact of the control on the scales interaction and were not accessible from the experiments.

A second example corresponds to a *passive drag reduction* method based on longitudinal grooves at the wall, named as "riblet". Numerous experimental studies in TBL at different regimes were devoted in the 90s to this method that mimics the skin of some fishes. The height of the riblets, in usual wind tunnel conditions, are of the order of ten viscous wall units (h^+), see Figure 5a. Indeed, this corresponds in Wind Tunnel to micron scales and is difficult to machine accurately. In addition, the gains in terms of viscous drag are relatively small (a few %) and very difficult to measure. Large scatter in the results is shown on Figure 5b. However significant gains were obtained (Coustols and Cousteix [56]) with evidence of an optimal spacing (s^+) lying between the ribs roughly corresponding to 12 wall units. From the experiments, the details of the mechanisms of the skin friction reduction were not so clear. More detailed flow descriptions were obtained from CFD, showing how the turbulence-generated motions originating very close



a) b) c)

Figure 5. Skin friction drag reduction by riblets. (a) Physical realization, (b) experimental evidence of the evolution of the longitudinal skin friction coefficient (C_{xf}) in terms of the distance between longitudinal riblets expressed in viscous unit (s^+), (Coustols and Cousteix [53]), (c) CFD geometry for 3D riblets after Bannier [54] (with permission).

to the wall are affected by the geometry. However, some more recent results show that these interpretations can be of limited extent due to the Re effects mentioned earlier. Indeed, when very high Re number values are considered (as is the case for aeronautical applications) large-scale boundary-layer sized events become more and more energetic and encompass the wall proximity effects. This effect, evidenced by DNS, can explain some limits in the riblet efficiency. In order to investigate other riblet representations, potentially more efficient and somewhat closer to those present on shark skins, 3D configurations can be a possible evolution. It is obvious that machining such devices, with the associated parametric studies at micron scale is even more difficult and, as previously stated, not easy to test in wind tunnels. On the contrary, CFD approaches are well designed for testing such possible 3D configurations. This was performed by Bannier [57]. The large eddy simulation (LES) computations are able to analyse in details the different contributions of different phenomena on the skin friction (Bannier *et al.* [58]). Longitudinal and transverse drag decomposition can be made and the authors show that the increase of pressure drag on the ribs overtakes significantly the gain in terms of friction drag. Such new detailed results and physical interpretations cannot be obtained presently from experiments.

A third and last example is the flow control of airfoils. Based on the observation of bird flight, an advanced conceptual hybrid design has been proposed by a EFD demonstrator. It is based on electro-morphing of the airfoil camber via Shape-Memory Alloy and associated to trailing edge vibration thanks to piezoactuators (Jodin *et al.* [59]). This kind of active flow control is quite complex to introduce experimentally in models and the domain of variation of parameters (amplitudes of displacements, frequency and laws of control, combinations of camber and flapping etc.) is indeed limited in the experiments. In addition, the unsteady character of the controlled flows is not easy to measure. The association with CFD appears to be essential for better understanding of the separation and wake behaviour. In addition, as we discussed earlier, this is a unique way to preform, after validation with the available experiments, parametric studies. Then, associated with EFD, detailed studies with Time-Resolved PIV, led Simiriotis *et al.* [60] to develop a CFD model that allowed to explore and optimize the feasibility and potential performance of the proposed conceptual hybrid design. A more complex example is the physics of the buffet phenomena arising on the wings at transonic speeds. Another comparable hybrid design has been investigated for controlling the buffet instabilities with some success via CFD and EFD by To *et al.* [61]. However, in this regime, experimental results are very difficult to obtain because transonic studies require large-scale facilities (Dandois *et al.* [62]).

4. Conclusion

To conclude, it is obvious that EFD can no longer be seen uniquely as a CFD validation tool. In particular for turbulent flows, a growing recognition of the importance of organized motions opens new research areas. These unsteady and spatially organized motions embedded in the turbulent random field promote parallel developments of experimental as well as numerical approaches. Among them the PIV and LES generic popular methods appear as relevant. Associated mathematical and data processing tools are used by both communities. A quasi-parallel evolution of both methods benefits—and promotes—the evolution of hardware and software. Many methods, concepts and theories are shared by the two communities, including particularly the fundamental knowledge of turbulent phenomena. Quasi-real time EFD/CFD comparisons are now possible.

Flow control is a domain in which both approaches are closely linked and their complementarity essential.

Despite the tremendous development of CFD, the last 50 years have shown that one cannot yet consider the concept of a numerical/virtual wind tunnel as a reliable alternative to wind tunnel testing. From the author's point of view, experimental fluid dynamics will not be made obsolete in the foreseeable future.

To progress, both CFD and EFD communities should make the most of sharing methods and tools, maybe more importantly, of sharing their culture and background, but surely of running researches which closely combine the two complementary approaches.

Conflicts of interest

The author has no conflict of interest to declare.

References

[1] S. Bryson, C. Levit, "The virtual wind tunnel: An environment for the exploration of three-dimensional unsteady flows", Tech. Report RNR-92-013, 1991.

[2] Y. Matsumo, "Numerical wind tunnel: History and evolution of supercomputing", *Fujitsu Sci. Tech. J.* **53** (2017), no. 3, p. 15-23.

[3] A. Smits, J. P. Dussauge, *Turbulent Shear Layers in Supersonic Flow*, Springer, New York, 2006.

[4] T. B. Gatski, J. P. Bonnet, *Compressibility, Turbulence and High Speed Flows*, Academic Press, Oxford, Amsterdam, 2013, ISBN 9780123970275.

[5] R. E. Falco, "Coherent motions in the outer region of turbulent boundary layers", *Phys. Fluids* **20** (1977), p. S124-S132.

[6] G. L. Brown, A. Roshko, "On density effects and large structure in turbulent mixing layers", *J. Fluid Mech.* **64** (1974), no. 1, p. 775-816.

[7] G. Comte, G. Bellot, "Hot-wire anemometry", *Annu. Rev. Fluid Mech.* **8** (1976), p. 209-231.

[8] H. H. Bruun, *Hot-wire Anemometry: Principles and Signal Analysis*, Oxford Unversity Press, Berlin, 1995.

[9] C. Tropea, A. L. Yarin, J. F. Foss (eds.), *Springer Handbook of Experimental Fluid Mechanics*, Springer, Berlin, 2007.

[10] K. Hanjalić, B. E. Launder, "Reassessment of modeling turbulence via Reynolds averaging: A review of second-moment transport strategy", *Phys. Fluids* **33** (2021), article no. 091302.

[11] S. J. Kline, *AFORS-HTTM-Stanford Conference on Complex Turbulent Flows: 1980-1981 Comparison of Computation and Experiments*, Department of Mechanical, Stanford University, Stanford, USA, 1982, ISBN 978-0960734832.

[12] A. K. M. F. Hussain, W. C. Reynolds, "The mechanics of an organized wave in turbulent shear flow", *J. Fluid Mech.* **41** (1970), no. 2, p. 241-258.

[13] B. J. Cantwell, "Organized motion in turbulent flow", *Annu. Rev. Fluid Mech.* **13** (1981), no. 1, p. 457-515.

[14] A. K. M. F. Hussain, "Coherent structures - Reality and myth", *Phys. Fluids* **26** (1983), p. 2816-2850.

[15] R. J. Adrian, "Twenty years of particle image velocimetry", *Exp. Fluids* **39** (2005), no. 2, p. 159-169.

[16] M. Stanislas, K. Okamoto, C. J. Kähler, J. Westerweel, "Main results of the second international PIV challenge", *Exp. Fluids* **39** (2005), no. 2, p. 170-191.

[17] P. Bergé, Y. Pomeau, C. Vidal, *L'ordre dans le chaos : vers une approche déterministe de la turbulence*, Hermann, Paris, 1984, ISBN 978-2-7056-5980-6.

[18] M. Lesieur, O. Metais, P. Comte, *Large-Eddy Simulations of Turbulence*, Cambridge University Press, 2005, ISBN 0521 78124 8.

[19] H. Ha Minh, "Modélisation et simulation des écoulements en situations industrielles. Exposé de synthèse", *La Houille Blanche* **73** (1987), no. 7/8, p. 555-562.

[20] P. Sagaut, *Large Eddy Simulation for Incompressible Flows: An Introduction*, Springer Verlag, Berlin, Heidelberg, 2006, ISBN 978-3-540-26344-9.

[21] P. Moin, K. Mahesh, "Direct numerical simulation: a tool in turbulence research", *Annu. Rev. Fluid Mech.* **30** (1998), no. 1, p. 539-578.

[22] J. P. Bonnet, M. N. Glauser (eds.), *Eddy Structure Identification in Free Turbulent Shear Flows*, Springer, Dordrecht, 1993, ISBN: 9780792324492.

[23] M. Farge, "Wavelet transforms and their applications to turbulence", *Annu. Rev. Fluid Mech.* **24** (1992), p. 395-457.

[24] M. Farge, G. Pellegrino, K. Schneider, "Coherent vortex extraction in 3D turbulent flows using orthogonal wavelets", *Phys. Rev. Lett.* **87**, no. 5, article no. 054501.

[25] P. Jordan, T. Colonius, "Wave packets and turbulent jet noise", *Annu. Rev. Fluid Mech.* **45** (2013), no. 1, p. 173-195.

[26] J. H. Citriniti, W. K. George, "Reconstruction of the global velocity field in the axisymmetric mixing layer utilizing the proper orthogonal decomposition", *J. Fluid Mech.* **418** (2000), p. 137-166.

[27] M. Tutkun, W. George, J. Delville, J. M. Foucault, S. Coudert, M. Stanislas, "Space-time correlations from a 143 hot-wire rake in a high reynolds number turbulent boundary layer", in *5th AIAA Theoretical Fluid Mechanics Conference Session: TFM-6: Turbulent Boundary Layers I, paper 2008-4239*, 2008.

[28] B. Chanetz, J. Délery, P. Gilliéron, P. Gnemmi, E. R. Gowree (eds.), *Experimental Aerodynamics: An Introductory Guide*, Springer, Cham, 2020, ISBN 3030355616.

[29] I. Marusic, B. J. McKeon, P. A. Monkewitz, H. M. Nagib, A. J. Smits, K. R. Sreenivasan, "Wall bounded turbulent flows at high Reynolds numbers: Recent advances and key issues", *Phys. Fluids* **22** (2010), article no. 065103.

[30] A. J. Smits, B. J. McKeon, I. Marusic, "High–Reynolds number wall turbulence", *Annu. Rev. Fluid Mech.* **43** (2011), p. 353-375.

[31] G. Schewe, N. P. van Hinsberg, M. Jacobs, "Investigation of the steady and unsteady forces acting on a pair of circular cylinders in crossflow up to ultra-high Reynolds numbers", *Exp. Fluids* **62** (2021), article no. 176.

[32] M. Lee, R. Moser, "Direct numerical simulations of trubulent channel flows up to Re_τ 5200", *J. Fluid Mech.* **774** (2015), p. 395-415.

[33] L. Agostini, M. Leschziner, "The connection between the spectrum of turbulent scales and the skin-friction statistics in channel flow at $Re_\tau \approx 1000$", *J. Fluid Mech.* **871** (2019), p. 22-51.

[34] J. Borée, P. C. Miles, "In-cylinder flow", in *Encyclopedia of Automotive Engineering*, John Wiley & Sons, 2014.

[35] G. Berkooz, P. J. Holmes, J. Lumley, "The proper orthogonal decomposition in the analysis of turbulent flows", *Annu. Rev. Fluid Mech.* **25** (2003), no. 1, p. 539-575.

[36] C. W. Rowley, S. T. M. Dawson, "Model reduction for flow analysis and control", *Annu. Rev. Fluid Mech.* **49** (2017), no. 1, p. 387-417.

[37] S. Beneddine, R. Yegavian, D. Sipp, B. Leclaire, "Unsteady flow dynamics reconstruction from mean flow and point sensors: an experimental study", *J. Fluid Mech.* **824** (2017), p. 174-201.

[38] T. Suzuki, H. Ji, F. Yamamoto, "Unsteady PTV velocity field past an airfoil solved with DNS", *Exp. Fluids* **47** (2009), p. 957-976.

[39] T. Suzuki, L. Chatellier, Y. Jin Jeon, L. David, "Unsteady pressure estimation and compensation capabilities of the hybrid simulation combining PIV and DNS Mech. 41(02)", *Meas. Sci. Technol.* **29** (2018), no. 12, p. 241-258.

[40] L. Perret, J. Delville, R. Manceau, J. P. Bonnet, "Generation of turbulent inflow conditions for large Eddy simulation from steroscopic PIV measurements", *Int. J. Heat Fluid Flow* **27** (2006), p. 576-584.

[41] J. M. Foucaut, W. K. George, M. Stanislas, C. Cuvier, "Optimization of a SPIV experiment for derivative moments assessment in a turbulent boundary layer", *Exp. Fluids* **62** (2021), p. 244-259.

[42] G. Rubino, M. Visonneau, "Towards the modeling of laminar to turbulence transition for incompressible flows", in *22st Numerical Towing Tank Symposium (NuTTS 2019), October 2019*, Tomar, Portugal, 2019, hal-02873474.

[43] L. Jacquin, D. Fabre, D. Sipp, E. Coustols, "Unsteadiness, instability and turbulence in trailing vortices", *Mec. Ind.* **2** (2001), p. 421-434.

[44] B. J. Geurts, *Direct and Large-Eddy Simulation*, De Gruyter Series in Computational Science and Engineering, De Gruyter, Berlin, Boston, 2022.

[45] J. P. Bonnet, N. Qin, "Active flow control strategies and tools for turbulent flows", in *Advances in Effective Flow Separation Control for Aircraft Drag Reduction: Modeling, Simulations and Experimentations* (N. Qin, J. Periaux, G. Bugeda, eds.), Springer, Cham, 2020, p. 1-26.

[46] M. Gad-el Hak, A. Pollard, J. P. Bonnet (eds.), *Flow Control: Fundamentals and Practices*, Springer-Verlag, Berlin, 1998, ISBN 3540639367.

[47] M. Gad-el Hak, *Flow Control: Passive, Active and Reactive Flow Management*, Cambridge University Press, Cambridge, UK, 2000.

[48] L. N. Cattafesta III, M. Sheplak, “Actuators for active flow control”, *Annu. Rev. Fluid Mech.* **43** (2011), no. 1, p. 247-272.

[49] B. R. Noack, M. Morzynski, G. Tadmor (eds.), *Reduced-Order Modelling for Flow Control*, Springer, Vienna, 2011.

[50] S. L. Brunton, B. R. Noack, P. Koumoutsakos, “Machine learning for fluid mechanics”, *Annu. Rev. Fluid Mech.* **52** (2020), p. 477-508.

[51] E. Moreau, “Airflow control by non-thermal plasma actuators”, *J. Phys. D: Appl. Phys.* **40** (2007), no. 3, p. 605-636.

[52] N. Benard, S. Laizet, E. Moreau, “PIV-based dynamic model of EHD volume force produced by a surface dielectric barrier discharge. AIAA paper 2017-1579”, in *55th AIAA Aerospace Sciences*, AIAA, Grapeville, TX, 2017.

[53] T. Brauner, S. Laizet, N. Bénard, E. Moreau, “Modelling of dielectric barrier discharge plasma actuators for direct numerical simulations”, in *8th AIAA Flow Control Conference*, AIAA, Washington, 2016.

[54] E. Laurendeau, P. Jordan, J. P. Bonnet, J. Delville, P. Parnaudeau, E. Lamballais, “Subsonic jet noise reduction by fluidic control: The interaction region and the global effect”, *Phys. Fluids* **20** (2008), article no. 101519.

[55] C. Bailly, C. Bogey, “Contributions of computational aeroacoustics to jet noise research and prediction”, *Int. J. Comput. Fluid Dynam.* **18** (2004), no. 6, p. 481-491.

[56] E. Coustols, J. Cousteix, “Performances of riblets in the supersonic regime”, *AIAA J.* **32** (1994), no. 2, p. 431-433.

[57] A. Bannier, “Contrôle de la traînée de frottement d’une couche limite turbulente au moyen de revêtements rainurés de type riblets”, Thèse, Univ. Paris 6, 2016.

[58] A. Bannier, E. Garnier, P. Sagaut, “Friction drag reduction achievable by near-wall turbulence manipulation in spatially developing boundary-layer”, *Phys. Fluids* **28** (2016), article no. 035108.

[59] G. Jodin, J. Scheller, E. Duhayon, J.-F. Rouchon, M. Braza, “Implementation of a hybrid electro-active actuated morphing wing in wind tunnel”, in *Solid State Phenomena, Trans Tech Publications Ltd*, vol. 260, 2017, p. 85-91.

[60] N. Simiriotis, G. Jodin, A. Marouf, P. Elyakime, Y. Hoarau, J. C. R. Hunt, J. F. Rouchon, M. Braza, “Morphing of a supercritical wing by means of trailing edge deformation and vibration at high Reynolds numbers: Experimental and numerical investigation”, *J. Fluids Struct.* **91** (2019), p. 102676-102698.

[61] J. B. Tô, N. Bhardwaj, N. Simiriotis, A. Marouf, Y. Hoarau, J. R. C. Hunt, M. Braza, “Manipulation of a shock-wave/boundary-layer interaction in the transonic regime around super-critical morphing wing”, in *Symposium on Fluid-Structure-Sound Interactions and Control, FSSIC2019, 27-30 August 2019, Chania, Greece*, Chania, Greece, 2021.

[62] J. Dandois, P. Molton, A. Lepage, A. Geeraert, V. Brunet, J.-B. Dor, E. Coustols, “Buffet characterization and control for turbulent wings”, *The ONERA Journal Aerospace Lab* **6** (2013), p. 1-17.