



Open Archive Toulouse Archive Ouverte (OATAO)

OATAO is an open access repository that collects the work of Toulouse researchers and makes it freely available over the web where possible.

This is an author deposited version published in: <http://oatao.univ-toulouse.fr/>
Eprints ID:

To link to this article: DOI: 10.1098/rsta.2013.0323

URL: <http://dx.doi.org/10.1098/rsta.2013.0323>

To cite this document: Gourdain, Nicolas and Sicot, Frédéric and Duchaine, Florent and Gicquel, Laurent *Large Eddy Simulation of flows in industrial compressors: a path from 2015 to 2035*. (2014) Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 372 (2022). ISSN 1364-503X

Any correspondence concerning this service should be sent to the repository administrator:
staff-oatao@inp-toulouse.fr

Subject Areas:

CFD, turbomachinery

Keywords:

CFD, Large Eddy Simulation, industrial, compressor

Author for correspondence:

Nicolas Gourdain

e-mail: nicolas.gourdain@isae.fr

Large Eddy Simulation of flows in industrial compressors: a path from 2015 to 2035

N. Gourdain^{1,2}, F. Sicot¹, F. Duchaine¹ and L. Gicquel¹

¹CERFACS, 42 avenue Coriolis 31057 Toulouse Cedex 1, France

²ISAE, Aerodynamics, Energetic and Propulsion Dpt., 10 avenue Édouard-Belin BP 54032 - 31055 Toulouse CEDEX 4, France

A better understanding of turbulent unsteady flows is a necessary step towards a breakthrough in the design of modern compressors. Due to high Reynolds numbers and very complex geometry, the flow that develops in such industrial machines is extremely hard to predict. At this time, the most popular method to simulate these flows is still based on a Reynolds Averaged Navier-Stokes (RANS) approach. However there is some evidence that this formalism is not accurate for these components, especially when a description of time-dependent turbulent flows is desired. With the increase in computing power, Large Eddy Simulation (LES) emerges as a promising technique to improve both knowledge of complex physics and reliability of flow solver predictions. The objective of the paper is thus to give an overview of the current status of LES for industrial compressor flows as well as to propose future research axes regarding the use of LES for compressor design. While the use of wall-resolved LES for industrial multistage compressors at realistic Reynolds number should not be ready before 2035, some possibilities exist to reduce the cost of LES, such as wall-modelling and the adaptation of the phase lag condition. This paper also points out the necessity to combine LES to techniques able to tackle complex geometries. Indeed LES alone, *i.e.* without prior knowledge of such flows for grid construction or the prohibitive yet ideal use of fully homogeneous meshes to predict compressor flows, is quite limited today.

1. Introduction

The aeronautic industry faces a formidable challenge by targeting in 2050 a reduction of CO_2 and NO_x emissions by 75% and 90% per kilometre and per passenger, respectively (compared to the 2000 level). Ultra high Bypass Ratio ($BR > 20$) turbofan and ultra-high Overall Pressure Ratio ($OPR > 70$) core engines have the potential for such significant reductions in fuel burn. However, after many years of optimization, the design of ever more efficient gas turbines now requires understanding of the complex unsteady flow appearing within each individual component and in an integrated fashion. Among all components, the compressor remains a critical part of a gas turbine, especially regarding its efficiency and stability. Due to the adverse pressure gradients, compressors are naturally subject to unstable flows such as surge and rotating stall [21], which can potentially lead to mechanical failure. These aerodynamic instabilities impose serious constraints on the design (the so-called “surge margin”) and thus on performance. In this context, maximizing the efficiency of compressors remains particularly complex and the improvement of the compressor robustness and performance thus requires good understanding of the flow physics.

Complementary to experimental investigations, the numerical simulation of flows, commonly referred to as Computational Fluid Dynamics (CFD), is a very promising way to predict and study flows in industrial compressors [67]. However, an accurate prediction of these flows is still a challenge for CFD. The complexity of the geometry and the nature of the flow (wall bounded, high Reynolds number) impose the use of turbulence modelling to represent the cascade of energy (the so-called Reynolds Averaged Navier Stokes -RANS- approach). However, as reported in the literature [77, 83], there is some evidence that such turbulence models are not accurate for complex flows as encountered in turbomachinery.

With increasing computing power, Large Eddy Simulation (LES) appears as an alternative to classical RANS methods. LES has already been successfully applied to a wide range of turbomachinery problems, such as off-design operating conditions [45, 62], secondary flows [8, 88], heat transfer [4, 27, 41, 89] and aero-acoustics [43]. However, due to the intrinsic computing cost of LES, most works only deal with simplified configurations (such as an isolated blade or vane). A recurring difficulty for CFD of industrial configurations is also the need to deal with complex geometries. At this time, only a few works propose recommendations about numerical methods and meshing techniques to apply LES for industrial turbomachinery [17, 81, 82].

This paper proposes a state-of-the-art and perspectives of what LES can do for the design of compressors. The added understanding of the related flows as well as the new point of view it provides are also underlined. The paper includes a prospective exploration of what LES will be in the next decades, considering both the evolution of computing power and numerical methods. First, the paper proposes a short review of current challenges related to compressor flows. Then the paper is organized to address three questions: 1) how computing means will grow in the next decades and what will be the implication for CFD, 2) what is the status of LES in 2015 for the simulation of compressor flows and 3) which should be the path to fully deploy LES and take maximum advantage from it as a design tool for compression systems?

2. Review of some challenges

Two concepts have been associated with the design of compressors for 60 years [32]: efficiency and stability. Further improvements of the compressor design requires dealing with unsteady flows that take place in these components.

A classification of the unsteady flow phenomena in turbomachinery is shown, Fig. 1, as proposed by Hodson [50]. Periodic flows can be addressed with a standard (U)RANS approach: steady RANS only predicts the time-averaged effects of unsteady flows, while unsteady RANS can also describe time-dependent flows. In specific cases, transient phenomenon can also be investigated thanks to unsteady RANS. Actually the main limitation with (U)RANS is that all

turbulent scales are modelled and so turbulence (chaotic) is not directly represented. Since the turbulent flow in compressors is highly anisotropic and characterized by complex physics, such as separation and laminar-to-turbulent transition, turbulence modelling can be inaccurate. A description of large turbulent scales can only be achieved by means of LES (while the smallest scales are modelled).

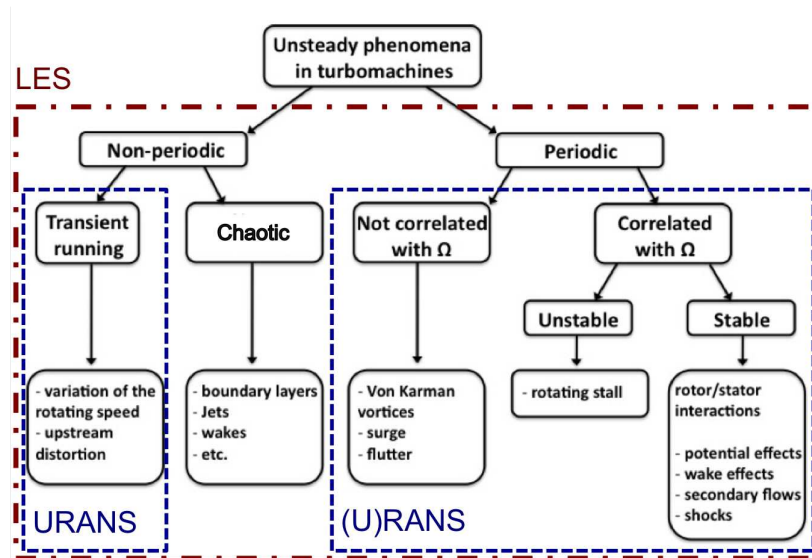


Figure 1. Overview of unsteady flow phenomena in turbomachinery [50]. RANS only address steady flows (but it can take into account the mean effect of periodic flows). URANS can describe time-dependent flows such as periodic and transient flows. LES can virtually address all unsteady flow phenomena, including those that are chaotic such as turbulence.

The overall performance of an industrial compressor is impacted by the matching conditions between the stages and the time-averaged unsteady flow effects. For example, rotor tip clearance and end-wall flows are responsible for local off-design operating conditions that affect both efficiency [26] and aerodynamic stability [20, 51]. Relative motions between fixed and rotating parts induce periodic unsteady flows (the so-called rotor-stator interactions, wakes and potential effects), which also impact performance and aerodynamic stability [1, 2]. The mean performance is also influenced by laminar-to-turbulent transition on blade walls [61] by several efficiency points. At moderate Reynolds number ($Re < 10^6$), there is a lack of evidence that boundary layers become turbulent close to the leading edge [39], which is unfortunately a classical hypothesis in (U)RANS simulations. An accurate estimation of the boundary layer state requires thus to be able to reproduce transition mechanisms (by means of modelling [63] or turbulent flow simulation), including the influence of very small details of the geometry such as surface roughness [7].

Since aerodynamic instabilities (stall and surge) may lead to mechanical failures, these instabilities should be avoided at all costs. Major improvements have been made to delay instabilities (partially thanks to CFD), but the simulation of such flows require very substantial computing resources. Rotating stall is a local instability characterized by one or more cells of stalled flow which rotate around the compressor annulus at a fraction of the rotor speed. However since the number of cells is not correlated to the number of blades or vanes, it requires a 360° simulation [16, 40, 84] (*i.e.* no assumption can be made to reduce the size of the domain, for example to an isolated rotor passage). Surge is a global mono-dimensional instability which induces very low frequencies $\mathcal{O}(1)$ Hz compared to the blade passing frequency $\mathcal{O}(10^3)$ Hz. In a deep surge regime, the flow direction can periodically reverse which induces strong mechanical constraints on the blade and difficulties for inlet/outlet boundary conditions in simulations.

Moreover, surge affects the whole compression system [18, 19] (the whole engine in case of gas turbine configurations or the experimental facility in case of research compressor). Indeed, to simulate surge one needs to run a CFD code able to deal with complex separated flows and periodic inversion of the flow direction, during a large time ($\mathcal{O}(100)$ rotations) in a large configuration. The capability to accurately predict the stability limit in a multistage compressor should help to reduce surge margin and design more efficient compressors. In that regard, improvements of the flow solvers reliability can be expected from the use of LES for stalled flows.

The design of flow control devices is a key-point to improve the performance of compressors and better manage surge margin [47]. A lot of solutions have been proposed in the literature such as active flow control [65, 85] and casing treatments [46, 78, 86], but the flow control mechanisms are not yet completely understood [11, 55]. The prediction of a control devices effectiveness is a challenge in itself due to the complexity of the flow that develops in a compressor equipped with a control solution. Moreover the shape and the size of these technological devices can impose severe constraints on the grid design. For example, riblets (thin slots manufactured on the blade wall) are known to be an effective solution to reduce losses [35, 68] but their size is only a few dozen micrometers, compared to a few dozen millimeters for the blade span.

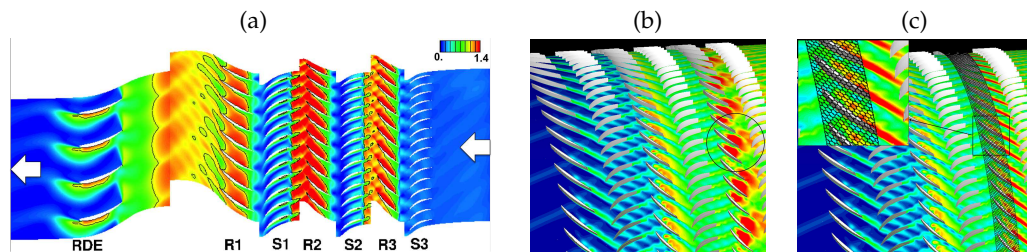


Figure 2. RANS-based simulation: (a) absolute Mach number during surge [19], (b) entropy flow field at near stall condition without control [42] and (c) entropy flow field at near stall conditions with a honeycomb style casing treatment [42].

Steady RANS simulations are already used to partially address these challenges, but the lack of validation of turbulence models in complex configurations limits the predictive capability of CFD. Even the use of unsteady RANS has a limited impact on the flow prediction [77]. LES appears to be a promising method to overcome the limits of turbulence models for turbomachinery flows [62, 81]. However, the capability to effectively use LES for complex geometries and high Reynolds flows relies on the capability of CFD codes to exploit the computing power delivered by supercomputers.

3. Impact of computing power

(a) Growth of computing power

Since the nineties and the advent of parallel computing, supercomputing relying on massively parallel architectures has allowed a sustained increase in computer power [79]. Based on these past evolutions, projections anticipate exascale computing to appear in the near future¹ which, for efficient use, requires code developers to anticipate developments and to follow this progress. Such architectures impose adaptation of current parallel CFD algorithms to take into account the hardware evolution, since the use of unsteady CFD simulations and more specifically LES [34, 72, 76] goes with the evolution of available computing power. In that context, LES has already led to major breakthroughs in other field of applications such as combustion [36, 64] and today LES

¹however, the date at which Exascale will be available for the scientific community is not obvious [13, 58], but it is reasonable to think such a computer will be available at the beginning of the 2020's.

also appears has an efficient turbulent flow modelling method for compressors to study isolated blade in static and rotating frames up to multistage configurations, Fig. 3.

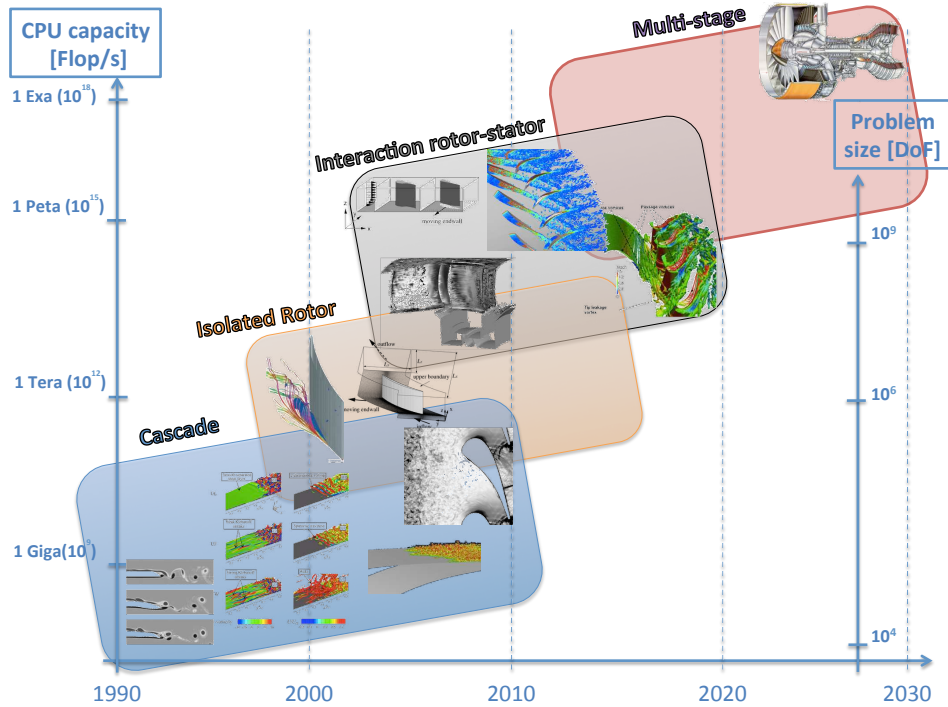


Figure 3. Typical evolution of the computing power growth over time (following a simple evolutionary model) and its potential impact on LES for compressor flow understanding and design.

Aside from the necessity to accurately validate such unsteady flow predictions against detailed temporal and spatial experimental data, the numerical developments required to follow the increase in computer power will most likely result from two distinct paths:

- (a) finer and finer grid resolutions for computational domains focused on specific fundamental flow features and dedicated to specific elements of a compressor. This path has evidently already been followed by the present research CFD community since directly taking advantage of existing CFD solutions with minimum code developments and modelling efforts. Limits are here essentially dictated by the existing CFD solutions and their potential to exploit current and future computing architectures. At some point and although modelling or numerics may not necessarily need to evolve in this context, the size of the problems greatly increasing, codes will require specific algorithmic adjustments for pre- and post-processing or memory management for example.
- (b) larger and larger computational domains as required to address more flow complexity and industrial problems. This path will inevitably rely on specific modelling strategies of such complex wall influenced flows. In the context of wall modelled or wall resolved LES [69, 70], fundamentals of turbulent closures need to be addressed and specific developments as well as validations will be required just like in RANS modelling. The fully unsteady formalism will however greatly complicate the work. Likewise, fundamentals of CFD including the impact of boundary conditions and scheme properties on flow predictions, they will need to be better understood in the specific context of LES. Finally critical steps already present in RANS codes will have

to be qualified such as the numerical interface treatment (and its impact on flow prediction at the rotor/stator interface) and the geometrical complexity of real machines. Ultimately, the size of the computational domain getting bigger, larger meshes will be handled necessitating at some point adjustment of the code algorithm to accommodate evolving computer architectures.

All of these issues will need to be adequately mastered by developers and end users making LES of compressor flows still for some time an expert tool, unless adequate interactions between turbulent modellers, code developers, experimentalists and industrialists prepare to this potential transition in method identified as a promising path to fully take advantage of the upcoming Exascale computing.

(b) When will LES be a design tool?

Addressing this question is a key issue, which does not admit a single answer, depending on the complexity of both the flow (Reynolds number) and geometry. As reported in [38, 41] for an isolated component (blade or vane), the cost ratio from RANS to wall-resolved LES is about 10,000 at a Reynolds number close to 10^6 . For a stage application (without periodic assumption), this cost ratio increases by a factor 10 to 100 (due to the number of blades and vanes in each row).

If computing means still grow following Moore's law (\approx power doubled each 1.5 years [58]), the increase in computing power needed by LES to become routine (as RANS was in the late 90's) represents a time period of about 30 years ($2^{20} = 1,048,576$). Indeed, LES is expected to become a standard design tool in the 2030's. First applications of LES to multistage compressors should be expected around 2020 in research laboratories.

However, such a prediction should be balanced. Looking at the past evolution, from 2000 to 2015, while the computing power has increased by 2^{10} (which corresponds to a factor 1,024), the size of simulated configurations has followed a different path and is closer to a factor 100, as shown in Table 1. As a consequence, a direct correlation between the size of the problem and the available computing power is not obvious.

4. Current research works

(a) How LES can help to improve CFD reliability?

CFD specialists agree to point out four sources of errors that limit the predictive capability of codes [25]: (i) numerical errors due to the grid/scheme adequacy; (ii) modeling errors (turbulence, transition, etc.); (iii) errors in boundary conditions; (iv) real geometry effects.

Among these sources of errors, LES is only able to address the category of modelling errors. Indeed, major breakthroughs in the predictive capability of flow solvers can only be achieved if all sources of errors are similarly reduced, otherwise one of these sources will play the role of "limiting factor". Previous works already pointed out the role of other sources of errors in the prediction of compressor flows, such as real geometry effects [81]. As a consequence, LES should be jointly used with a grid methodology well suited to the description of complex geometries, including geometric details (tip clearances, technological effects, cavities etc.). At least, if the real geometry can not be represented (uncertainties on the geometry, unknown details), an alternative is to quantify the sensitivity of the numerical solution to these uncertain/unknown parameters [67].

A brief (non-exhaustive) review of LES for compressor flows is proposed in Table 1. As reported in a review lecture by Dufour *et al.* [30], this field of research is very recent and there are only a few reported in the literature. Contrary to turbine configurations which benefit from high ease/cost ratio [83], LES in compressors suffer from a low ease/cost ratio, due to:

- high Reynolds numbers ($10^5 < Re < 10^7$ for jet engine applications), which impose to use a very large number of grid points to compute the near-wall flow [69, 70];
- complex geometry (e.g. tip clearances, flow control, wall roughness, interactions with upstream/downstream components, etc.);
- complex physics (laminar-to-turbulent transition, surge, rotating stall, etc.).

| Year | Method | Mesh | Grid points ($\times 10^6$) | Reynolds ($\times 10^6$) | Reference |
|------|------------------------------|------|----------------------------------|-------------------------------|----------------------------------|
| 2005 | LES (6th order, implicit) | STR | 9.8 (x1) | 0.45 | Teramoto <i>et al.</i> [80] |
| 2006 | LES (2nd order, implicit) | STR | 0.8 (x6) | 0.15 | Reese <i>et al.</i> [73] |
| 2006 | LES (2nd order, implicit) | STR | 25.0 (x1) | 0.40 | You <i>et al.</i> [87] |
| 2007 | LES (4th order, explicit) | STR | 3.0 (x1) | 1.00 | Boudet <i>et al.</i> [8] |
| 2009 | Hybrid (2nd order, explicit) | STR | 5.0 (x1) | 0.23 | Eastwood <i>et al.</i> [31] |
| 2009 | LES (2nd order, implicit) | STR | 13.8 (x1) | 1.20 | Hah [45] |
| 2010 | LES (2nd order, implicit) | STR | 26.0 (x1) | 6.00 | Guleren <i>et al.</i> [44] |
| 2011 | LES (3rd order, implicit) | STR | 100.0 (x1) | 1.20 | Gomar <i>et al.</i> [38] |
| 2012 | (I)LES (2nd order, implicit) | UNS | 1.5 (x10) | 0.50 | Hellstrom <i>et al.</i> [49] |
| 2012 | LES (2nd order, implicit) | UNS | 39.0 (x1) | 0.70 | McMullan <i>et al.</i> [62] |
| 2012 | LES (2nd order, implicit) | UNS | 36.0 (x2) | 0.18 | McMullan <i>et al.</i> [62] |
| 2012 | LES (2nd order, implicit) | UNS | 35.0 (x5) | 0.35 | McMullan <i>et al.</i> [62] |
| 2012 | LES (2nd order, implicit) | STR | 5.0 (x1) | 1.20 | Hah <i>et al.</i> [46] |
| 2013 | LES (2nd order, explicit) | STR | 7.0 (x2) | 0.70 | De Laborderie <i>et al.</i> [22] |
| 2013 | LES (3rd order, implicit) | STR | 122.0 (x7) | 0.70 | Gourdain [39] |
| 2013 | Hybrid (2nd order, implicit) | STR | 44.0 (x2) | 1.00 | Riera <i>et al.</i> [75] |

Table 1. Overview of works reported in the literature which deals with LES (or hybrid methods) for the simulation of compressor flows (the number of grid points is indicated for one rotor or stator passage and the coefficient in brackets indicates how many passages have been simulated). Note: STR stands for structured and UNS for unstructured grids.

An analysis of the data shown in Table 1 indicates that the most popular approach relies on the use of implicit time integration coupled with 2^{nd} or 3^{rd} order numerical schemes, including for aero-acoustics as reported by Reese *et al.* [73]. The most popular approach for the mesh generation relies on structured grids but these works usually consider academic configurations (isolated blade profile or simplified compressor geometry). The use of unstructured grids is preferred for complex geometries close to industrial configurations [49, 62].

Most works consider grids which are far from academic recommendations for wall-resolved LES: $x^+ < 100$, $y^+ < 1$ and $z^+ < 20$ [70]. Indeed, most studies deal with under-resolved LES, especially in the near wall region. This observation is particularly true at high Reynolds numbers (e.g. academic recommendations at $Re = 10^6$ lead to a grid with 10^8 points per blade passage). In order to reduce the constraints on the number of grid points at high-Reynolds number, some works propose to couple a RANS approach (close to walls) to LES in the rest of the domain [31, 75]. The use of wall laws adapted to LES [5] is not standard for compressors but it should be investigated (a difficulty is to take into account for rotation and curvature effects).

(b) Is an under-resolved LES potentially better than (U)RANS?

The typical trade-off game present for any industrial use of CFD (*i.e.* grid resolution versus modelling effort) naturally raises the question of the actual contribution of under-resolved LES compared to the (U)RANS approach. To address this specific question, a comparison of (U)RANS and LES has been performed [56] on a voluntary coarse grid ($y^+ < 5$), representing a vane at Reynolds $2 \cdot 10^6$. The numerical boundary layer profiles are compared to experimental

measurements on the suction side, close to the trailing edge, Fig. 4. While the velocity profiles in the boundary layer are correctly estimated by the (U)RANS simulations, LES fails to accurately estimate the gradients in the near wall region. This result indicates that LES is more sensitive to grid refinement than (U)RANS, which highlight the necessity to pay a particular attention to the grid with LES, especially close to walls. Focusing on the temporal evolution of the near wall flow dynamics provides however a different view. Figure 4(c) shows a comparison of the temporal evolution of the axial velocity within the vane wake. Compared to experimental measurements, LES provides a more accurate estimation of the Strouhal number than URANS: errors are 4% and 26%, respectively. Clearly, the debates about the trade-off between the grid resolution and modelling effort and their effective impact on the quality of the predictions are far from being easy to answer.

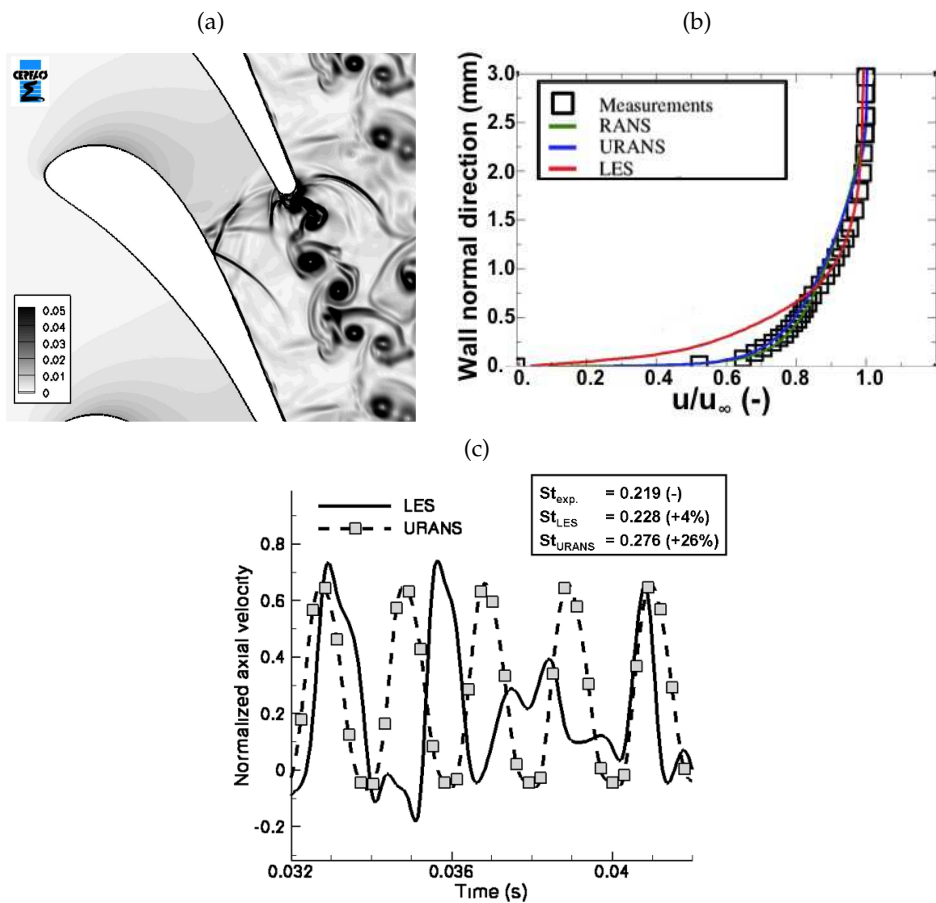


Figure 4. Influence of under-resolved grid ($y^+ < 5$) on RANS and LES predictions [56]: (a) instantaneous density gradient flow field (LES), (b) velocity profiles in the suction side boundary layer and (c) estimations of the Strouhal numbers measured in the wake, behind the vane trailing edge.

(c) Boundary conditions

Non-reflective boundary conditions [57] and/or sponge layers are usually considered with LES to avoid spurious acoustic reflections on inlet and outlet conditions. Moreover the use of a Navier-Stokes Characteristic Boundary Condition (NSCBC) naturally ensured radial equilibrium at the

outlet [54]. However an accurate description of the flow at the inlet plane is also mandatory to achieve a correct accuracy. In that regard, the lack of experimental data (such as turbulent intensity and spectrum) is often reported in the literature [41, 62, 75] as a major source of uncertainty to validate LES results.

Another difficulty is related to the definition of boundary conditions at walls. The standard of compressor simulations is to use an adiabatic wall assumption. Nevertheless, the increase in compression leads to more and more important heating of the flow which reaches temperatures higher than wall temperature. The adiabatic assumption is thus no longer pertinent, especially at the hub and casing. In that regards, heat transfer plays an important role in total temperature and pressure distribution and consequently on the performance prediction [12].

(d) Industrial applications

There is evidence that LES can help achieve a better understanding of the flow physics in industrial compressors. Figure 5(a) shows a description of the wall-resolved turbulent flow in a stage of an axial compressor [39]: some complex flow phenomena have been highlighted such as the laminar-to-turbulent transition process in the stator, due to incoming wakes. Figure 5(b) and Fig. 5(c) show some investigations of the shock/boundary layer interaction in a transonic compressor, which is out of reach with a classical (U)RANS approach. However, most works presented in Table 1 deal with either isolated blade profile or periodic stage configuration (by means of a blade/vane rescaling if necessary).

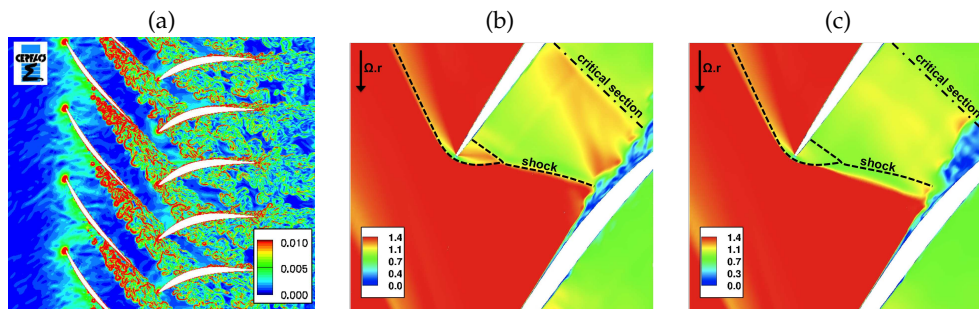


Figure 5. Example of LES performed in industrial compressor configurations: (a) entropy flow field in a compressor stage at $Re = 700,000$ [39], (b) and (c) instantaneous Mach number flow fields at two different instants in a transonic compressor, showing the shock/boundary layer interaction [38].

At this time and as far as the authors know, two challenges thus remain for compressor stage applications:

- achieve a well-resolved LES without any periodicity assumption of a multistage configuration at realistic Reynolds number;
- adapt the phase-lagged condition to LES in order to reduce a stage configuration to a periodic sector (see section 5(a)).

(e) Summary

The use of LES to simulate compressor flows is a recent research field (most contributions reported in the literature are under 10 years). A literature review shows that the current numerical methods are a compromise between cost and accuracy. The most popular method relies on the use of second/third order schemes with implicit time integration. In 2015, the "world" of LES for compressor flows can be divided in two categories: on one side, the academic community

considers properly resolved LES, validated for idealized configurations (usually with structured grids) and, on the other side, the industrial community deals with under-resolved LES of flows in complex geometries (usually with unstructured grids). Major improvements could be expected by combining the experience of these two worlds [62]. A better integration of experimental campaigns with CFD is also a prerequisite to ensure effective validation of LES methods (*e.g.* inlet turbulence).

5. Future research works

The objective of this section is to provide an overview of future work associated with the development of LES for compressor flows. As already mentioned, the use of LES for industrial compressor flows is still a challenge, mainly due to a high computational cost. Indeed, a reduction of this cost will help to popularize the use of LES for such applications. At the same time, bottlenecks such as boundary conditions and real geometry effects should also be investigated.

(a) Methods to reduce the cost of LES: wall-modelling and phase-lag boundary condition

As reported in paragraph 3(b), the cost ratio from RANS to wall-resolved LES is as much as 1,000,000 for a typical Reynolds number around 10^6 . This cost ratio can be decreased by a factor 1,000 by considering both wall-modelling (grid point savings in the boundary layer and larger time step) and phase-lag boundary conditions (to reduce a compressor to one blade or vane passage per row). Such a gain corresponds to a time period of 15 years (2^{10}) regarding computing power. In this case, a LES of a compressor stage at realistic Reynolds number ($Re \approx 10^6$) would already be accessible in 2015.

The way in which wall modelling can help to save grid points is obvious: in a boundary layer, the number of points required by LES scales as $N \sim Re^{1.8}$, which is similar to DNS requirements [70]. If the internal part of the boundary layer can be modelled, this number of points now scales as $N \sim Re^{0.4}$. At high-Reynolds number ($Re 10^6$), more than 90% of the grid points are found in the internal part of the boundary layer. Moreover, while a $y^+ \approx 1$ condition is required for wall-resolved LES, $y^+ < 30$ is sufficient for wall-modelled LES, leading to an increase of the time step. Applications to external aerodynamics have been successful [5, 6, 52]. However, the rotation and curvature effects encountered in compressors are responsible for a modification of the near-wall flow that should be taken into account in wall-modelling, as suggested in [14, 29]. At least a proper validation of wall modelling in compressors can be performed against wall-resolved LES, already available [39, 62].

The phaselag conditions developed by Erdos *et al.* [33] allow to solve for only one blade passage per row, reducing drastically the mesh size. By periodically storing the flow field (*e.g.* every time step) at azimuthal boundaries and blade row interface, one can interpolate it at the required lagged time to set the opposite boundary. As the mesh required for LES is finer and the time step smaller (compared to classical unsteady RANS simulations), the direct store method would be very memory demanding. He [48] maps the flow variables on a Fourier basis leading to the so-called "Shape Correction" method. When only retaining a subset of all possible harmonics of blade passing frequency (BPF), the memory requirement drops compared to the classical direct store method. However only a discrete and narrow part of the spectrum is captured: Figure 6 shows the LES of the flow over a circular cylinder. The downstream block is rotating and two configurations are investigated: (i) a periodic sector with azimuthal periodic condition and a sliding nonmatch at the interface and (ii) a 4:3 configuration where azimuthal and interface boundaries are solved using the shape correction phaselag method by retaining 32 harmonics of the BPF. The power spectral density of the speed fluctuations in the downstream block is plotted in Fig. 6(c) in the periodic case. The spectrum is large band and contains frequencies that are not integer multiple of the BPF, mainly due to the vortex shedding behind the cylinder

and resolved free turbulence. The 32-harmonics shape correction method results in a serious loss of information. Indeed, one can observe drastic changes of spectrum in Fig. 6(c).

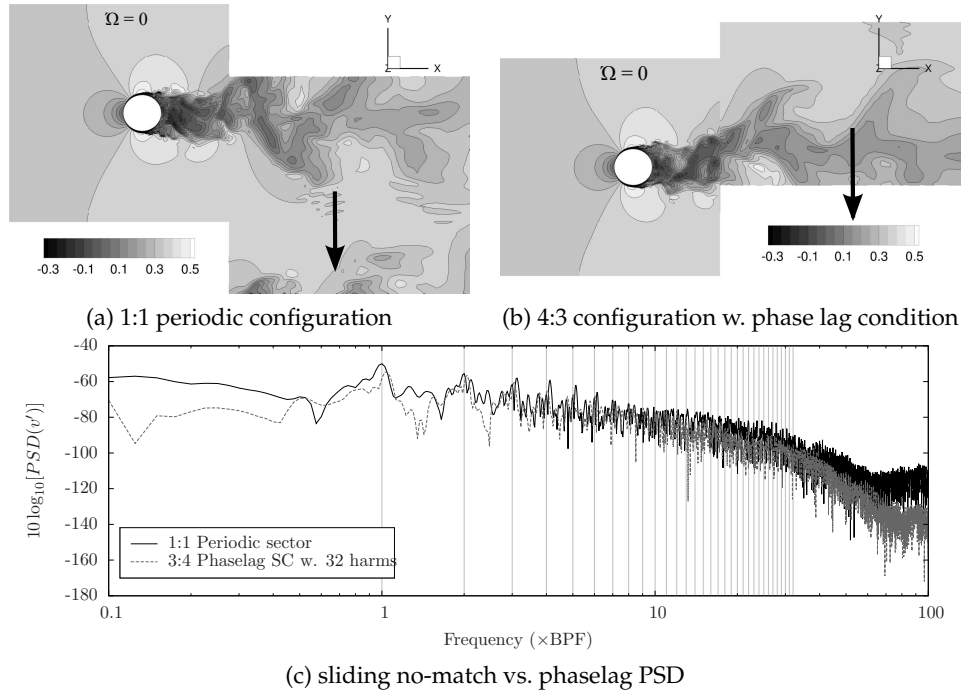


Figure 6. Power Spectral Density of velocity fluctuation of a LES simulation over a cylinder. The gray vertical lines denote the first 32 harmonics of the BPF.

Another alternative is developed by Giles [37] who proposes to incline the computational plane in time so that the flow is periodic in the azimuthal direction. This method does not seem to be as employed as the Shape Correction mainly due to the implementation hurdles and stability issues. Furthermore the specific post-processing required would be crippling for LES.

Instead of using an *a priori* basis such as Fourier or even wavelet, a data-driven basis would be better suited for LES. The Proper Orthogonal Decomposition (POD), also known as Principal Component Analysis or Karhunen-Loève Decomposition [9, 53], has been used in the analysis of turbulent flows for decades [3]. The main advantage of POD is that it takes advantage of both spatial and time structures of the flow. This methodology allows a dynamic building of surrogate models for phaselag conditions.

(b) Towards real geometry

As already pointed out, most works dealing with LES in compressors are currently limited to academic or simplified configurations. Using LES alone will limit the increase of CFD accuracy, due to real geometry effects [67]. The use of LES with unstructured meshes allows one to take into account more complex geometry and fluid flow interactions in compressor applications [62]. This approach is also well-suited to High-Performance Computing requirements (load balancing, reduction of MPI size messages, etc.). In that context, numerical methods based on high-order reconstructions, such as Discontinuous Galerkin [23, 74] and Spectral Volumes methods [15], represent a very interesting way to run CFD codes in highly parallel computing environment.

An example of complex physics accessible from such a simulation is the fully unsteady interaction between rotor, stator, turbulence and mixing in the upstream cavity of a blade. Figure 7 presents a LES performed on a high pressure turbine stage. The geometry contains a periodic

sector with 2 stator vanes and 3 rotor blades as well as upstream and downstream cavities around the rotor's hub. The instantaneous temperature field at the first off wall cells, Fig. 7(a)), underlines hot and cold regions at the same relative position for different rotor blades (slices #1 and #2). Figure 7(b) shows in more detail the flow topologies in the corresponding slices. Vortices are clearly visible in the cavity, bringing hot gases from the vein to the cavity.

Penetration of hot gases is illustrated on the slice #1 while on slice #2 the cold flow exists on a large part of the cavity passage and then mixes with hot air coming from the main passage. These patterns are a complex consequence of many flow features: interaction of the stator wakes with a blockage effect induced by the presence of the rotors, important deviation of the flow caused by rotors that helps hot air to tangentially penetrate in the cavity and finally the geometry of the cavity and flow conditions that induce large recirculation zones. Resolved turbulent eddies participate in the mixing process between hot and cold streams adding randomness in the pattern caused by the previous mentioned interactions.

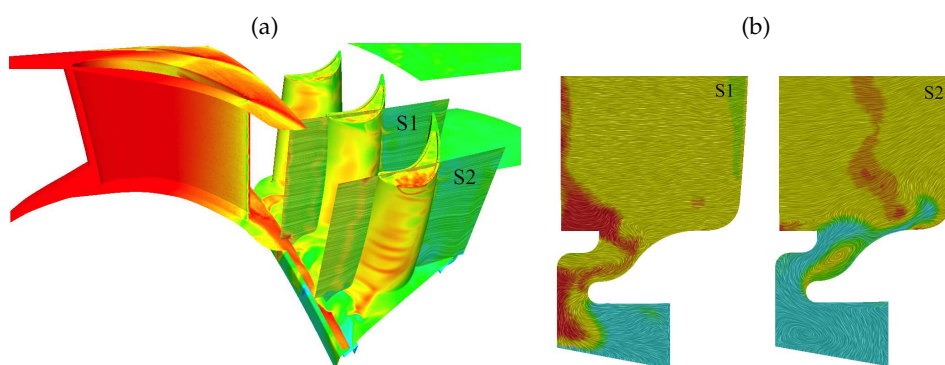


Figure 7. LES in a high-pressure turbine stage: a) periodic domain representation of the rotors and stators with cavities and b) zooms on cavities slices on hot and cold regions.

Following the same philosophy, but more prospective, is the use of Lattice-Boltzmann method (LBM). The advantage of LBM is its capability to deal with very complex geometries since the grid generation is no longer managed by the user. LBM relies on a kinetic description which also makes the modelled physics simpler than with Navier-Stokes codes. Indeed, a LBM solver requires less lines of code, which is an asset when it must be adapted to new computing paradigms (for example to GPU [10]). Recent works have also shown that LBM is compatible with a LES framework [59]. Since LBM does not require any additional artificial dissipation which make the method well suited to the simulation of turbulence in general and aero-acoustics problems in particular [60].

However, what makes LBM not yet ready for LES in compressor flows is its current limitation in terms of Mach number ($M < 0.4$). While not an intrinsic limitation of the method, time is needed to extend the application range to transonic flows. Indeed this framework is expected to progressively grow in the next few years, before occupying a significant place for compressor applications at high Mach and Reynolds number.

(c) Multi-physics applications

Another way to improve the reliability of flow solvers is to associate many kind of physics. A compressor is not affected only by the flow aerodynamics, but it evolves in a complex environment including aero-thermics and fluid/structure coupling.

A compressor under high stage loading conditions can experience flutter, a destructive fluid/structure interaction that must be avoided. Figure 8 shows that the compressor performance map is all surrounded by regions where flutter can occur. If classical (U)RANS methods can

perform well in choked flow (region II) and supersonic regimes along the operating line, LES could bring deep insights into the different stall flutters. Forced response can also be better estimated as wakes and tip leakage flows are better captured with LES than RANS (see for instance the flow structure of Fig. 5(a)). To the authors' knowledge, no aeroelastic simulations have been performed on representative compressor configuration with LES.

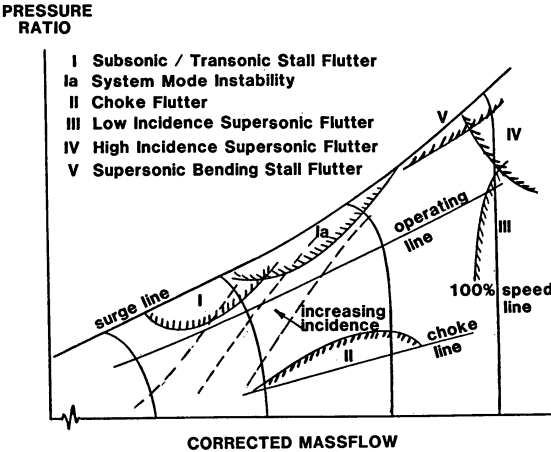


Figure 8. Flutter regions on the compressor map from Platzer and Carta [71].

As the pressure ratio of compressors increases (e.g. ultra-high overall pressure ratio core engines), the use of adiabatic walls become more and more inaccurate. As a consequence, an isothermal boundary condition appears to be more accurate and more realistic than the adiabatic one [12]. The emerging question is: what is the temperature to impose at the wall? Moreover, in multi-stage compressors with cavities, the temperature increases along the different passages and a single fixed temperature for all walls will probably lead to erroneous results. The ultimate solution would be more realistic simulations such as a conjugate heat transfer approaches to better match the physics. Wall temperatures are unknowns of the problem that are resolved by the coupled simulations. This implies coupling a flow solver with a conduction solver [28] as well as the radiative transfer equation when compression is important, leading to high static temperatures (according to the Stefan-Boltzmann law, radiative heat transfer scales as T^4).

(d) Post-processing

Post-processing is an easy task in the case of purely RANS simulation, as in most industrial applications the main interest is to extract global parameters from the database (such as pressure ratio, efficiency, etc.).

The data deluge promised by higher and higher fidelity simulations will become an issue and post-processing such simulations will evolve rapidly towards the concept of a numerical wind tunnel (i.e. the results are reliable but since not all the data can be stored, we can not access to the information everywhere in the simulated configuration).

In the case of a full unsteady solution, the storage capacity and I/O overhead can become the limiting factor. For instance, in [39] the grid was made of about 10^9 cells: the storage of all the information over one full revolution of the compressor generates a 100Tb database. CFD already enters the "Big data" age. Actually one way to process such database comes from internet companies such as Google or Facebook which developed automated efficient, scalable and resilient tools to easily deal with massive data flows: the Mapreduce framework [24] became popular and many open source implementations such as Hadoop are available. Not all processing

fit into Mapreduce but pioneering works from Nichols [66] opened this path. Although these big data technologies work, they still require the simulation to periodically dump the solution on disks which is time and space consuming. Such simulations would benefit from co-processing: that is the runtime in-memory processing of the flow. One can extract cuts and probes, update statistics on the fly and only write the processed data. Although it is very efficient as no memory duplication occurs and I/O are minimized, it requires *a priori* definition of the processing, going back to the numerical wind tunnel issue: scientists or engineers should ask questions they want to address before running the simulation.

6. Conclusion

A roadmap for the deployment of LES for the design and improvement of compressors has been proposed. The use of LES is justified to address complex physics problems such as state of boundary layers (laminar to turbulent transition), heat transfer, off-design operating conditions and flow control. The following conclusions can be drawn:

- in 2015, wall-resolved LES is not realistic for industrial compressors design, due to computational cost at high Reynolds number;
- the growth in computing power should make LES ready for multistage compressors around 2030;
- approaches exist to reduce the cost of LES: wall modelling and adaptation of phase lag to LES should help to reduce this cost by a factor 1,000 (corresponding to 15 years of computational growth);
- the use of LES alone is not sufficient to increase the flow solver reliability. Methods are needed to tackle more complex geometries. The best candidates at this time are the Navier-Stokes-based unstructured methods. In the future, the LBM framework could represent a valuable candidate;
- the difficulties in the deployment of LES for the compressor design are not just technical: it requires the capability of many worlds to work together (academic and industry, experiments and numerics). The validation of LES for industrial applications requires to share open industrial test cases and results, with a capability to reproduce the results of other research groups. At this time, this requirement is not fulfilled.

Like most recent fields of investigation, there is still a large place on this topic for new ideas. Some ways for research and some milestones have been proposed in this paper, it would be very interesting to check in 2035 which of these predictions were true.

Acknowledgements

This work has benefited from the help of a lot of people: PhD students A. Gomar, T. Leonard, G. Mouret, colleagues from Université Sherbrooke (S. Moreau, G. Wang, J. de Laborderie) and University of Naples (M. Manna). Experimental data were provided by Von Karman Institute, Laboratoire de Mécanique des Fluides et Acoustique (Lyon), Institut Jean le Rond d'Alembert (Paris). Thanks for the great cooperative piece of work. The authors are also grateful to SAFRAN (with a special mention to L. Pons, M. Dumas and G. Leroy for their constant help and support). This work has benefited from CERFACS internal and GENCI-TGCC computing facilities (under the project gen6074). ONERA also provided help and support for the CFD code *elsA*. These supports are greatly acknowledged. Last, but not least, the authors would like to thank P. Tucker for organizing this issue.

References

- 1 Arnaud, D., Ottavy, X. & Vouillarmet, A. 2004 Experimental investigation of the rotor-stator interactions within a high-speed, multi-stage, axial compressor. part 1 - experimental facilities

- and results. In *Asme turbo expo: Power for land, sea, and air*, vol. 5 of paper GT2004-53764, pp. 903–914. (doi:10.1115/GT2004-53764)
- 2 Arnaud, D., Ottavy, X. & Vouillarmet, A. 2004 Experimental investigation of the rotor-stator interactions within a high-speed, multi-stage, axial compressor. part 2 - modal analysis of the interactions. In *Asme turbo expo: Power for land, sea, and air*, vol. 5 of paper GT2004-53778, pp. 915–924. (doi:10.1115/GT2004-53778)
 - 3 Berkooz, G., Holmes, P. & Lumley, J. L. 1993 The proper orthogonal decomposition in the analysis of turbulent flows. *Annual Review of Fluid Mechanics*, **25**(1), 539–575. (doi:10.1146/annurev.fl.25.010193.002543)
 - 4 Bhaskaran, R. & Lele, S. K. 2010 Large eddy simulation of free stream turbulence effects on heat transfer to a high-pressure turbine cascade. *J. of Turbulence*, **11**(6). (doi:10.1080/14685241003705756)
 - 5 Bocquet, S., Jouhaud, J. & Sagaut, P. 2012 A compressible wall model for large-eddy simulation with application to prediction of aerothermal quantities. *Physics of Fluids*, **24**(6), 065103. (doi:10.1063/1.4729614)
 - 6 Bodart, J. & Larsson, J. 2011 Wall-modeled large eddy simulation in complex geometries with application to high-lift devices. In *Annual research briefs, center for turbulence research*. Stanford University.
 - 7 Bons, J. P. 2010 A review of surface roughness effects in gas turbines. *J. Turbomach.*, **132**(2), 021004. (doi:10.1115/1.3066315)
 - 8 Boudet, J., Caro, J., Shao, L. & Leveque, E. 2007 Numerical studies towards practical Large-Eddy Simulation. *J. Thermal Science*, **16**(4), 328–336. (doi:10.1007/s11630-007-0328-3)
 - 9 Braconnier, T., Ferrier, M., Jouhaud, J.-C., Montagnac, M. & Sagaut, P. 2011 Towards an adaptive pod/svd surrogate model for aeronautic design. *Computers & Fluids*, **40**(1), 195 – 209. (doi:http://dx.doi.org/10.1016/j.compfluid.2010.09.002)
 - 10 Brandvik, T. & Pullan, G. 2010 An accelerated 3d navier-stokes solver for flows in turbomachines. *J. Turbomach.*, **133**(2).
 - 11 Brignole, G., Kau, H. P. & Wilke, I. 2005 Numerical evaluation of important parameters ruling the effectiveness of casing treatments in transonic compressor. In *17th isabe*, paper 1095.
 - 12 Bruna, D. & Turner, M. G. 2013 Isothermal boundary condition at casing applied to the rotor 37 transonic axial flow compressor. *J. Turbomach.*, **135**(3), 034501. (doi:10.1115/1.4007569)
 - 13 Cappello, F., Geist, A., Gropp, B., Kale, L., Kramer, B. & Snir, M. 2009 Towards exascale resilience. *Int. J. of High Performance Computing Applications*, **23**(4), 374–388. (doi:10.1177/1094342009347767)
 - 14 Cazalbou, J.-B., Chassaing, P., Dufour, G. & Carbonneau, X. 2005 Two-equation modeling of turbulent rotating flows. *Physics of Fluids*, **17**(5).
 - 15 Chikhaoui, O., Gressier, J. & Grondin, G. 2010 Assessment of the spectral volume method on inviscid and viscous flows. In *6th international conference on cfd*.
 - 16 Choi, M., Smith, N. H. S. & Vahdati, M. 2012 Validation of numerical simulation for rotating stall in a transonic fan. *J. Turbomach.*, **135**(2), 021004. (doi:10.1115/1.4006641)
 - 17 Collado Morata, E., Gourdain, N., Duchaine, F. & Gicquel, L. 2012 Effect of free-stream turbulence on high pressure turbine blade heat transfer using structured and unstructured les. *Int. J. of Heat and Mass Transfer*, **55**(21-22), 5754–5768.
 - 18 Courtiade, N. & Ottavy, X. 2013 Experimental study of surge precursors in a high-speed multistage compressor. *J. Turbomach.*, **135**(6), 061018. (doi:10.1115/1.4023462)
 - 19 Crevel, F., Gourdain, N. & Ottavy, X. submitted Numerical simulation of aerodynamic instabilities in a multistage high-speed high-pressure compressor on its test-rig : part 2 - surge. *J. Turbomach.*
 - 20 Crook, A. J., Greitzer, E. M., Tan, C. S. & Adamczyk, J. J. 1993 Numerical simulation of compressor endwall and casing treatment flow phenomena. *J. Turbomach.*, **115**(3), 501–512. (doi:10.1115/1.2929280)
 - 21 Day, I. J. 2006 The fundamentals of stall and surge. part 1: Stall. part 2: Surge. In *Advances in axial compressors aerodynamics*, VKI Lecture Series.

- 22 de Laborderie, J., Moreau, S. & Berry, A. 2013 Compressor stage broadband noise prediction using a large-eddy simulation and comparisons with a cascade response model. In *19th aiaa/ceas aeroacoustics conference*, 2013-2042. (doi:10.2514/6.2013-2042)
- 23 de Wiart, C. C., Hillewaert, K., Duponcheel, M. & Winckelmans, G. 2014 Assessment of a discontinuous galerkin method for the simulation of vortical flows at high reynolds number. *Int. J. Numer. Meth. Fluids*, **74**(7), 469–493.
- 24 Dean, J. & Ghemawat, S. 2004 Mapreduce: Simplified data processing on large clusters. In *6th symposium on operating system design and implementation*. San Francisco (CA), USA.
- 25 Denton, J. D. 2010 Some limitations of turbomachinery CFD. In *Asme turbo expo*, GT2010-22540.
- 26 Domercq, O. & Escuret, J.-F. 2007 Tip clearance effect on high-pressure compressor stage matching. *J. Power and Energy*, **221**(6), 759–767. (doi:10.1243/09576509JPE468)
- 27 Duchaine, F., Corpron, A., Moureau, V., Nicoud, F. & Poinso, T. 2009 Development and assessment of a coupled strategy for conjugate heat transfer with large eddy simulation. application to a cooled turbine blade. *Int. J. of Heat and Fluid Flow*, **30**(6), 1129–1141.
- 28 Duchaine, F., Maheu, N., Moureau, V., Balarac, G. & Moreau, S. 2013 Large eddy simulation and conjugate heat transfer around a low-mach turbine blade. *J. Turbomach.*, **136**(5), 051015. (doi:10.1115/1.4025165)
- 29 Dufour, G., Cazalbou, J., Carbonneau, X. & Chassaing, P. 2008 Assessing rotation/curvature corrections to eddy-viscosity models in the calculations of centrifugal-compressor flows. *J. Fluids Eng.*, **1**(9).
- 30 Dufour, G., Gourdain, N., Duchaine, F., Vermorel, O., Gicquel, L., Bousuge, J.-F. & Poinso, T. 2009 Large Eddy Simulation applications. In *Numerical investigations in turbomachinery: the state-of-the-art*, VKI Lecture Series.
- 31 Eastwood, S., Tucker, P., Xia, H. & Klostermeier, C. 2009 Developing large eddy simulation for turbomachinery applications. *Phil. Trans. R. Soc. A*, **367**(1899), 2999–3013. (doi:10.1098/rsta.2008.0281)
- 32 Emmons, H. W., Pearson, C. E. & Grant, H. P. 1955 Compressor surge and stall propagation. *Trans. ASME*.
- 33 Erdos, J. I., Alzner, E. & McNally, W. 1977 Numerical solution of periodic transonic flow through a fan stage. *AIAA J.*, **15**(11), 1559–1568. (doi:10.2514/3.60823)
- 34 Ferziger, J. H. 1977 Large eddy simulations of turbulent flows. *AIAA J.*, **15**(9), 1261–1267. (doi:10.2514/3.60782)
- 35 García-Mayoral, R. & Jiménez, J. 2011 Drag reduction by riblets. *Phil. Trans. R. Soc. A*, **369**(1940). (doi:10.1098/rsta.2010.0359)
- 36 Gicquel, L., Staffelbach, G. & Poinso, T. 2012 Large eddy simulations of gaseous flames in gas turbine combustion chambers. *Prog. Energy Comb. Sci.*, **38**(782-817).
- 37 Giles, M. B. 1988 Calculation of unsteady wake/rotor interaction. *Journal of Propulsion and Power*, **4**(4), 356–362. (doi:10.2514/3.23074)
- 38 Gomar, A., Gourdain, N. & Dufour, G. 2011 High-fidelity simulation of the turbulent flow in a transonic axial compressor. In *9th european turbomachinery conference*, paper 236.
- 39 Gourdain, N. 2013 Validation of large-eddy simulation for the prediction of compressible flow in an axial compressor stage. In *Asme turbo expo*, GT2013-94550.
- 40 Gourdain, N., Burguburu, S., Leboeuf, F. & Michon, G.-J. 2010 Simulation of rotating stall in a whole stage of an axial compressor. *J. Computer and Fluids*, **39**(9), 1644–1655.
- 41 Gourdain, N., Gicquel, L. Y. M. & Collado, E. 2012 Comparison of RANS and LES for prediction of wall heat transfer in a highly loaded turbine guide vane. *J. Propulsion and Power*, **28**(2), 423–433.
- 42 Gourdain, N., Wlassow, F. & Ottavy, X. 2012 Effect of tip clearance dimensions and control of unsteady flows in a multi-stage high-pressure compressor. *J. Turbomach.*, **134**(5), 051005. (doi:10.1115/1.4003815)
- 43 Greschner, B. & Thiele, F. 2011 Wall modeled LES simulation of rotor-stator-cascade broadband noise. In *17th aiaa/ceas aeroacoustics conference*, paper 2873.
- 44 Guleren, K. M., Afgan, I. & Turan, A. 2010 Predictions of turbulent flow for the impeller of a nasa low-speed centrifugal compressor. *J. Turbomach.*, **132**(2), 021005. (doi:10.1115/1.3140824)

- 45 Hah, C. 2009 Large Eddy Simulation of transonic flow field in NASA rotor 37. Tech. Rep. TM-2009-215627, NASA Glenn Research Center.
- 46 Hah, C., Mueller, M. & Schiffer, H. 2012 Study of convective flow effects in endwall casing treatments in transonic compressor rotors. In *Asme turbo expo*, GT2012-68411. Copenhagen (Denmark).
- 47 Hathaway, M. D. 2006 Passive endwall treatments for enhancing stability. In *Advances in axial compressor aerodynamics*, VKI Lecture Series.
- 48 He, L. 1990 An euler solution for unsteady flows around oscillating blades. *J. Turbomach.*, **112**(4). (doi:10.1115/1.2927714)
- 49 Hellstrom, F., Gutmark, E. & Fuchs, L. 2012 Large eddy simulation of the unsteady flow in a radial compressor operating near surge. *J. Turbomach.*, **134**(5), 051006. (doi:10.1115/1.4003816)
- 50 Hodson, H. 1998 Blade row interaction in low pressure turbines. In *Blade row interference in axial turbomachinery stages*, VKI Lecture Series.
- 51 Inoue, M., Kuroumaru, M., Yoshida, S., Minami, T., Yamada, K. & Furukawa, M. 2004 Effect of tip clearance on stall evolution process in a low-speed axial compressor stage. In *Asme turbo expo*, vol. 5 of *paper GT2004-53354*, pp. 385–394. (doi:10.1115/GT2004-53354)
- 52 Kawai, S. & Larsson, J. 2012 Wall-modeling in large eddy simulation: length scales, grid resolution and accuracy. *Physics of Fluids*, **24**.
- 53 Kosambi, D. D. 1943 Statistics in function space. *Journal of the Indian Mathematical Society*, **7**(76-88).
- 54 Koupper, C., Poinot, T., Gicquel, L. & Duchaine, F. in press Ability of characteristic boundary conditions to comply with radial equilibrium in turbomachinery simulations. *AIAA J*.
- 55 Legras, G., Gourdain, N. & Trebinjac, I. 2011 Extended methodology for analysing the flow mechanisms induced by casing treatment in compressor. In *9th european turbomachinery conference*, paper 242.
- 56 Leonard, T., Duchaine, F., Gourdain, N. & Gicquel, L. 2010 Steady / unsteady Reynolds Averaged Navier-Stokes and Large Eddy Simulations of a turbine blade at high subsonic outlet mach number. In *Asme turbo expo*, paper GT2010-22469.
- 57 Lodato, G., Domingo, P. & Vervisch, L. 2008 Three-dimensional boundary conditions for direct and large-eddy simulation of compressible viscous flows. *J. Computational Physics*, **227**(10).
- 58 Mack, C. 2011 Fifty years of moore's law. *IEEE Trans. on Semiconductor manufacturing*, **24**(2).
- 59 Malaspinas, O. & Sagaut, P. 2012 Consistent subgrid scale modelling for lattice boltzmann methods. *J. Fluid Mech.*, **700**, 514–542.
- 60 Marie, S., Ricot, D. & Sagaut, P. 2009 Comparison between lattice boltzmann method and navier-stokes high order schemes for computational aeroacoustics. *J. Computational Physics*, **228**(4), 1056–1070.
- 61 Mayle, R. E. 1991 The role of laminar-turbulent transition in gas turbine engines. *J. Turbomach.*, **113**(4), 509–536. (doi:10.1115/1.2929110)
- 62 McMullan, W. A. & Page, G. J. 2012 Towards large eddy simulation of gas turbine compressors. *Progress in Aerospace Sciences*, **52**, 30–47. (doi:10.1016/j.paerosci.2011.12.002)
- 63 Menter, F. R., Langtry, R. B., Likki, S. R. & Suzen, Y. B. 2006 A correlation-based transition model using local variables - part 1: Model formulation. *J. Turbomach.*, **128**(3).
- 64 Moin, P. 2002 Advances in Large Eddy Simulation methodology for complex flows. *Int. J. of Heat and Fluid Flow*, **23**(5).
- 65 Niazi, S. 2000 Numerical simulation of rotating stall and surge alleviation in axial compressor. Ph.D. thesis, Georgia Institute of Technology.
- 66 Nichols, J. 2013 Post-processing computational fluid dynamics data in mapreduce. In *2nd icme mapreduce workshop*. Stanford University (CA), USA.
- 67 Ottavy, X., Courtiade, N. & Gourdain, N. 2012 Experimental and computational methods for flow investigation in high-speed multistage compressor. *J. Propulsion and Power*, **28**(6), 1141–1155. (doi:10.2514/1.B34412)
- 68 Peet, Y. & Sagaut, P. 2009 Theoretical prediction of turbulent skin friction on geometrically complex surfaces. *Physics of Fluids*, **21**, 105105. (doi:10.1063/1.3241993)

- 69 Piomelli, U. 2008 Wall-layer models for Large-Eddy Simulations. *Progress in Aerospace Sciences*, **44**(6), 437–446. (doi:10.1016/j.paerosci.2008.06.001)
- 70 Piomelli, U. & Balaras, E. 2002 Wall-layer models for Large-Eddy Simulations. *Annu. Rev. Fluid Mech.*, **34**, 349–374. (doi:10.1146/annurev.fluid.34.082901.144919)
- 71 Platzer, M. F. ; Carta, F. O. 1987 Manual on aeroelasticity in axial-flow turbomachines. volume 1. unsteady turbomachinery aerodynamics. Tech. Rep. AGARD-AG-298-VOL-1, Advisory Group for Aerospace Research and Development.
- 72 Pope, S. B. 2000 *Turbulent flows*. Cambridge University Press.
- 73 Reese, H., Kato, C. & Carolus, T. 2006 Large eddy simulation of gust noise sources in a low pressure axial compressor. In *12th aiaa/ceas aeroacoustics conference*, 2006-2576. Cambridge (MA), USA. (doi:10.2514/6.2006-2576)
- 74 Ren, X. & Gu, C. 2013 Investigation of compressor tip clearance flow based on the discontinuous galerkin methods. In *Asme turbo expo*, GT2013-94435.
- 75 Riera, W., Castillon, L., Marty, J. & Leboeuf, F. 2013 Inlet condition effects on the tip clearance flow with zonal detached eddy simulation. *J. Turbomach.*, **136**(4). (doi:10.1115/1.4025216)
- 76 Sagaut, P. 2000 *Large eddy simulation for incompressible flows*. Scientific computation series. Springer-Verlag.
- 77 Schobeiri, M. T. & Abdelfattah, S. 2013 On the reliability of rans and urans numerical results for high-pressure turbine simulations: A benchmark experimental and numerical study on performance and interstage flow behavior of high-pressure turbines at design and off-design conditions using two different turbine designs. *J. Turbomach.*, **135**(6), 061012. (doi:10.1115/1.4024787)
- 78 Shabbir, A. & Adamczyk, J. J. 2005 Flow mechanism for stall margin improvement due to circumferential casing grooves on axial compressors. *J. Turbomach.*, **127**(4), 708–717. (doi:10.1115/1.2008970)
- 79 Sites, T. . S. Top500 list - november 2012.
- 80 Teramoto, S. 2005 Large-eddy simulation of a transonic compressor cascade with boundary layer transition. In *41st aiaa joint propulsion conference and exhibit*, 2005-3639. Tucson (AZ), USA. (doi:10.2514/6.2005-3639)
- 81 Tucker, P., Eastwood, S., Klostermeier, C., Jefferson-Loveday, R., Tyacke, J. & Liu, Y. 2012 Hybrid LES approach for practical turbomachinery flows - part 1: hierarchy and example simulations. *J. Turbomach.*, **134**(2), 021023. (doi:10.1115/1.4003061)
- 82 Tucker, P., Eastwood, S., Klostermeier, C., Xia, H., Ray, P., Tyacke, J. & Dawes, W. 2012 Hybrid LES approach for practical turbomachinery flows - part 2: further applications. *J. Turbomach.*, **134**(2), 021024. (doi:10.1115/1.4003062)
- 83 Tyacke, J., Tucker, P., Loveday, R., Vadlamani, N., Watson, R., Naqavi, I. & Yang, X. 2013 Large eddy simulation for turbines: Methodologies, cost and future outlooks. *J. Turbomach.*, **136**(6), 061009. (doi:10.1115/1.4025589)
- 84 Vahdati, M., Simpson, G. & Imregun, M. 2008 Unsteady flow and aeroelasticity behavior of aeroengine core compressors during rotating stall and surge. *J. Turbomach.*, **130**(3), 031017. (doi:10.1115/1.2777188)
- 85 Weigl, H. J., Bright, M. M., Strazisar, A. J., Paduano, J. D., Frechette, L. G., Epstein, A. H. & Greitzer, E. M. 1998 1997 best paper award – controls and diagnostics committee: Active stabilization of rotating stall and surge in a transonic single-stage axial compressor. *J. Turbomach.*, **120**(4), 625–636. (doi:10.1115/1.2841772)
- 86 Wilke, I. & Kau, H. P. 2004 A numerical investigation of the flow mechanisms in a high pressure compressor front stage with axial slots. *J. Turbomach.*, **126**(3), 339–349. (doi:10.1115/1.1731465)
- 87 You, D., Wang, M., Moin, P. & Mittal, R. 2006 Effects of tip-gap size on the tip-leakage flow in a turbomachinery cascade. *Physics of Fluids*, **18**, 105102. (doi:10.1063/1.2354544)
- 88 You, D., Wang, M., Moin, P. & Mittal, R. 2007 Large-Eddy Simulation analysis of mechanisms for viscous losses in a turbomachinery tip-clearance flow. *J. Fluid Mech.*, **586**, 177–204. (doi:10.1017/S0022112007006842)
- 89 Zhong, B. & Tucker, P. G. 2004 $k - l$ based hybrid LES/RANS approach and its application to heat transfer simulation. *Int. J. Numer. Meth. Fluids*, **46**(10), 983–1005. (doi:10.1002/fld.782)