ABSTRACT
The present work proposes to use LES in a 3D transonic axial compressor configuration, the NASA Rotor 37. Three meshes are investigated: a 10 million, a 25 million and a 100 million grid points mesh. The convergence criteria, the choice of the numerical schemes and the determination of the time-step are investigated to propose a methodology for LES calculations. The time needed to compute such simulations and comparison with the experimental results are given. Finally, an analysis of the mesh dependency for LES is assessed.

NOMENCLATURE
LES, Large Eddy Simulation
RANS, Reynolds-Averaged Navier-Stokes
CFD, Computational Fluid Dynamics
SGS, Sub-Grid Scale
CPU, Central Processing Unit
CFL number, Courant-Friedrich-Levy number
DFT, Discrete Fourier Transform
Re, Reynolds number
$\Delta x^+, y^+, \Delta z^+$, mesh spacings in wall units
$f_c$, Cut-off frequency
$N_{ds}$, Number of data samples

INTRODUCTION
Numerical simulation is now a central tool for engineers, as it is well integrated in the design process of aerodynamic components. In fact, Reynolds-Averaged Navier-Stokes simulations give accurate information on the global performance (pressure ratio and efficiency for a turbomachine, lift coefficient for a wing . . . ), but fail to predict phenomena where turbulence plays a salient role. Indeed, turbulence models associated with RANS simulations are validated on particular test case, and are therefore not universal.

LES resolves turbulent structures by applying a space filter (Sagaut, 2009) to the Navier-Stokes equations, as opposed to RANS simulations, where a statistical or time filter is applied to obtain the equations that will be resolved. In fact, LES is based on the idea of resolving the large eddies whilst modeling the small ones. The former characterize most of the energetics structures, that is to say the large-eddies dominate the structure of the flow, whereas the latter is dissipative and universal (see Pope (2000)). Therefore, LES can be viewed as a high resolution simulation, which could help understand the physical phenomena conducted by turbulence.

Since CFD codes need to be validated, blind tests are frequently performed on open test cases, as for example the NASA Rotor 37 (see Dunham et al. (1998), Suder and Celestina (1996) and Suder (1996)). However discrepancies, concerning the physical reason of the pressure deficit downstream of the rotor below 40% span, remains between the RANS results and the experiments. Two explanations have been pointed out: on the one-hand, some believe that the pressure ratio deficit observed near
the hub in the wake region is due to injection and recirculation while on the other-hand, Hah and co-workers expect that this deficit is due to a corner flow separation (Denton, 1996). In this respect, the flow effects that occur in the Rotor 37, such as vortex shedding, tip-leakage vortex interaction with the shock and boundary layer-shock interaction, limit the predictability of RANS simulations and could account for these discrepancies. LES has been performed on the NASA Rotor 37 configuration by Hah (2009) showing that the results were closer to the experimental data than the RANS ones.

LES calculations have been performed on academic or limited turbomachine configurations such as: 1. small Reynolds numbers (see Carolus et al. (2006), Eastwood et al. (2009) and Guleren et al. (2010)) 2. configurations where an incompressible assumption was made (see Guleren et al. (2008) and Guleren et al. (2010)) 3. real-life configuration with relatively coarse mesh (see Hah et al. (2008), Black et al. (2005), Hah et al. (2009) and Hah (2009)) as compared to standard LES mesh requirement (Sagaut and Deck, 2009). These academic simulations demonstrate the ability of LES to better predict turbulent flows (see Ref Dufour et al. (2009)), but it appears that the assessment on industrial configurations needs to be pursued.

The aim of this study is to develop a methodology for LES computations on industrial turbomachinery cases regarding the mesh size, the numerical parameters and the convergence criteria. The comparison with experimental results will help to assess the benefit of LES calculations on such an industrial system. The open test case investigated is presented in Sect. 1, the methodology and the numerical procedure are then presented in Sect. 2. Finally, results on 100 million, 25 million and 10 million-grid-point meshes are analyzed to assess the benefit of LES computation.

**THE NASA ROTOR 37 TEST CASE**

The open test case under investigation is the well-known high-pressure-ratio axial compressor NASA Rotor 37. Overall characteristics of the nominal operating point is summarized in Fig. 1.

<table>
<thead>
<tr>
<th>Number of Rotor Blades</th>
<th>36</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip radius at leading edge</td>
<td>252 mm</td>
</tr>
<tr>
<td>Aspect ratio</td>
<td>1.19</td>
</tr>
<tr>
<td>Hub-tip radius ratio</td>
<td>0.70</td>
</tr>
<tr>
<td>Tip solidity</td>
<td>1.288</td>
</tr>
<tr>
<td>Tip clearance height</td>
<td>0.356 mm</td>
</tr>
<tr>
<td>Rotation speed</td>
<td>17188.7 r/min</td>
</tr>
<tr>
<td>Tip speed</td>
<td>454 m/s</td>
</tr>
<tr>
<td>Total pressure ratio</td>
<td>2.106</td>
</tr>
<tr>
<td>Mass flow (corrected)</td>
<td>20.19 kg/s</td>
</tr>
<tr>
<td>Choked mass flow</td>
<td>20.93 kg/s</td>
</tr>
</tbody>
</table>

This test case is frequently used as a validation case for CFD codes as it is well described experimentally (see Fig. 1):

1. radial distributions of the total pressure ratio, temperature ratio, of the absolute angle and the isentropic efficiency are available at two stations upstream and downstream of the rotor: station 1 ($z=-4.19$ cm) and station 4 ($z=10.67$ cm),

2. pitchwise distributions of the relative mach number are available at three stations: station 2 (20% $x/c$), station 3 ($z=4.57$ cm) and station 4 ($z=10.19$ cm), at 30%, 50%, 70%, 90%, 95% of
the spanwise direction, allowing to compare the position of the calculated shock to the experimental one,

3. performance parameters are computed using the radial distributions, the pressure ratio being energy averaged and the temperature ratio being mass averaged across the annulus, allowing to compute the adiabatic efficiency.

Despite its use as a validation case, RANS simulations often fail to predict the global and local parameters of the NASA Rotor 37, probably because of the physical phenomena that develop in this compressor (shock, shock/tip leakage vortex interaction). In fact, a noteworthy discrepancy is the total pressure ratio deficit near the hub at design mass flow. This pressure deficit is believed to be due to injection/recirculation or to a corner flow separation (see Denton (1996)) but no agreement has been found yet. Since corner flow separation stems from an unsteady behavior of the flow field, this configuration seems relevant for assessing the benefit of LES on a complex geometry. Furthermore, this compressor is isolated, i.e. the computational domain is small enough to allow a high density of mesh cells for given CPU resources.

TOOLS AND NUMERICAL PROCEDURE

**elsA CFD code Presentation**

The results presented in this study are performed with the elsA CFD code developed by ONERA and CERFACS. elsA solves the Favre-averaged compressible Navier-Stokes 3D equations with a cell-centered finite-volume formulation on multi-block structured meshes. A backward Euler integration with implicit LU schemes and SSOR correction is used for the integration of the discrete equations. Temporal discretization uses a standard second order accurate dual time-stepping algorithm with Newton’s sub-iterations. [More information about this flow solver can be found in Ref. (Cambier and Veuillot, 2008) and validation for turbomachinery applications is assessed by Castillon et al. (2002). Furthermore, the ability of the elsA CFD code to predict turbulent flows with LES is detailed in Leonard et al. (2010).]

**Numerical Models**

The numerical model used in the present study is outlined in the following. For the RANS simulations, a second-order Roe scheme is used for the discretization of the convective fluxes (using a Van Albada limiter) and a $k - \omega$ Menter (Menter (1993)) was employed to estimate the eddy viscosity. For LES, the convective fluxes are discretized using a third order Roe scheme since decentered scheme are better for predicting discontinuities like shocks. The SGS model is the Wall-Adapting Local Eddy-Viscosity (WALE) model, which is build to have the right behavior near walls as opposed to Smagorinsky SGS. All details of this SGS model and its validation can be found in Nicoud and Ducros (1999).

**Computational Grid**

The 3D computational domain corresponds to a single blade channel. The geometry is meshed using an 04H topology: a 2 million-grid-point mesh for RANS computation, a 10 million-grid-point mesh for LES computation, a 25 and a 100 million-grid-point meshes for computing LES. For all cases, the size of the first mesh cell is set to $3 \mu m$ to ensure $y^+ \sim 1$. All quality details of these meshes are summarized in Table 1.

In Georgiadis et al. (2010), the recommended quality details for meshing an LES configuration are:

$$50 < \Delta x^+ < 150$$
Table 1: **Mesh quality details.**

<table>
<thead>
<tr>
<th></th>
<th>$2M_{RANS}$</th>
<th>$10M_{LES}$</th>
<th>$25M_{LES}$</th>
<th>$100M_{LES}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>mean $\Delta x^+$</td>
<td>490</td>
<td>302</td>
<td>169</td>
<td></td>
</tr>
<tr>
<td>mean $y^+$</td>
<td>1.8</td>
<td>1.2</td>
<td>1.37</td>
<td>1.43</td>
</tr>
<tr>
<td>mean $\Delta z^+$</td>
<td>55</td>
<td>62</td>
<td>37</td>
<td></td>
</tr>
<tr>
<td>points in the axial direction</td>
<td>209</td>
<td>229</td>
<td>289</td>
<td>485</td>
</tr>
<tr>
<td>points in the radial direction</td>
<td>105</td>
<td>457</td>
<td>649</td>
<td>1161</td>
</tr>
<tr>
<td>points in the azimuthal direction</td>
<td>57</td>
<td>89</td>
<td>105</td>
<td>109</td>
</tr>
<tr>
<td>spanwise points in tip</td>
<td>29</td>
<td>57</td>
<td>81</td>
<td>117</td>
</tr>
</tbody>
</table>

$y^+ < 1$

$15 < \Delta z^+ < 40$

One can notice that the results for the 100 million-grid-point mesh are in the upper limit of the recommended quality details. This explain roughly why a LES is so costly from the computational resources point of view.

**Simulation Procedure**

*Choice of the Time-Step*

The methodology used in this study is to consider a CFL number close to one with a characteristic length of the size of the boundary layer thickness close to the leading edge, and a characteristic velocity assumed to the speed of sound. The idea is that we want the smallest structures, correctly computed with the LES approach (i.e. no low pass filtering is done on these structures), to be of the size of the boundary layer. This means that the time-step does not act as a low pass filter, this function being done by the mesh. One definition of the CFL number is:

$$\text{CFL} = \left( u + a \right) \frac{\Delta T}{\Delta X} \Rightarrow \Delta T = \text{CFL} \frac{\Delta X}{u + a}$$

where $u$ denotes the axial velocity, $a$ the local speed of sound, $\Delta T$ the time-step and $\Delta X$ a characteristic length of the cell. As mentioned before, the authors consider the characteristic length to be of the size of the boundary layer (extracted from a RANS computation near the leading edge) and the speed of sound for the axial velocity:

$$\Delta X \sim \delta \sim 2.10^{-4} \text{ m}, \text{ and } (u + a) \sim 2.680 \text{ m.s}^{-1}. $$

Finally,

$$\Delta T \sim 3.10^{-7} \text{ s}$$

*Statistics on the Flow-Field*

The period considered to perform the statistics is the through-flow time (let us recall that the through-flow time is the convective time that a particle needs to travel all the numerical domain). One through-flow time is computed in 4000 iterations and 100 samples are used to average the flow-field.

*Convergence Assessment*

One of the main issue when conducting unsteady flow calculations is to determine when the computation is statistically converged, so that statistics on the flow field are relevant. For a steady simulation, the iterative convergence criteria (see Casey and Wintergerste (2000) for instance) is a
well-established criteria to stop the computation but is hard to extend to unsteady flow computations. In fact, the difficulty with this type of simulation is that data can be periodic or fully unsteady, yielding to the impossibility to apply the former criteria. A way to solve the problem is to consider the time-average of the data. Although this approach works well with URANS simulations, this is not true with LES computations, since the spectrum of turbulent scales is large. Therefore, the frequency content of a LES signal might be of prior interest.

The Fast Fourier Transform is an efficient mathematical method to compute the Discrete Fourier Transform (DFT). If \( x(t = n\Delta T/N) \) is a discrete signal where \( n \in [0; N] \), the \( k^{th} \) Fourier coefficient \( X_k \) is defined as:

\[
X_k = \sum_{n=0}^{N-1} x(n) e^{-i\frac{2\pi kn}{N}}
\]

The convergence criterion of Ahmed and Barber (2005) relies on the DFT to evaluate the convergence of the "physical amplitudes". In fact, when computing the Fourier transform of a signal, a cutoff-frequency corresponding to the spectral resolution fits in with the error contained in the signal (since the signal is discrete, Fourier transform cannot be resolved for scales smaller than \( f_c = 1/N_{ds} \Delta T \), where \( N_{ds} \) denotes the number of data samples). The criterion of Ahmed and Barber (2005) states that a computation is converged if the amplitude of the physical frequencies of the problem (frequencies that appear during the simulation and that are higher than \( f_c \)) is very large compared to the dominant frequency in the cutoff region and if these amplitudes are stable between consecutive time intervals. When this criteria is satisfied, one can perform statistics on the flow-field. To assess this convergence criterion, 15 probes were placed in the computational domain at locations of interest. The spatial mean of the mass flow rate at outlet has also been extracted to assess this criterion on a conservative and spatially averaged variable. It should be noted that the probes are placed in the relative frame.

![Graph](image1)

(a) Zoom near the cutoff-frequency, \( f_c \sim 120 \) Hz, of three consecutive through-flows. Transitory part.

![Graph](image2)

(b) Comparison of two consecutive through-flows after 140,000 iterations.

Figure 2: Assessment of the convergence criterion.

The results presented in Fig. 2 are Fourier transform of the outlet mass flow signal. In fact, the criterion has also been assessed on probes signal but these have a relatively chaotic frequency content, preventing the assessment of the criterion on such signals.

Application of the convergence criterion for this industrial configuration raised several issues. As can be seen in Fig. 2(a), in the cutoff region \( (f_c \sim 120 \) Hz), the low (unphysical) frequencies damp and the larger (physical) ones become predominant between consecutive through-flows. This behavior is seen in all the present LES calculations and occurs in the early time of the calculation: only three through-flows are needed to have the maximum amplitude in the cutoff region at 20%. 

5
However, after 35 through-flows iterations, no convergence on the amplitude is found between two consecutive through-flow times (see Fig. 2(b)), despite the fact that the maximum amplitude of the frequencies cutoff region are smaller than 5%. This behavior is due to the frequency content of LES. In fact, turbulence is a random process, explaining why no stabilization on the amplitude is seen.

Statistics, on the overall and local parameters, have been performed after 40,000 iterations on the present LES calculations. The variance $\sigma$ computed is: $\sigma \sim 1.0E-5$ implying that the computation is clearly converged in mean.

The criterion of Ahmed and Barber (2005) seems to be unadapted to the physics of a LES calculation. In fact, LES computations contain a large range of frequencies, unlike URANS calculations, preventing any convergence in amplitude of the Fourier transform, whereas the convergence in mean is reached. Indeed, LES Fourier transform can not see their amplitude stabilize since no deterministic period is a natural phenomenon. Therefore, for LES simulations, a restriction of the criterion of Ahmed and Barber (2005) might be that the physical amplitudes dominate the spectrum.

RESULTS

Comparison of the Resources Needed

<table>
<thead>
<tr>
<th></th>
<th>Number of Nehalem processors</th>
<th>Iteration</th>
<th>CPU time (Hour)</th>
</tr>
</thead>
<tbody>
<tr>
<td>RANS$_{2M}$</td>
<td>1</td>
<td>4,000</td>
<td>12</td>
</tr>
<tr>
<td>LES$_{10M}$</td>
<td>32</td>
<td>80,000</td>
<td>6,144</td>
</tr>
<tr>
<td>LES$_{100M}$</td>
<td>512</td>
<td>80,000</td>
<td>$\sim$ 66,000</td>
</tr>
</tbody>
</table>

Table 2: Resources needed for the computations

Table 2 summarizes the time needed to simulate an industrial configuration with a LES approach. The time needed to compute the 100 million grid point mesh is really expensive compared to RANS simulations (by a factor 5500).

Validation

The performance maps (pressure ratio and efficiency) obtained by the LES and RANS calculations are given in Fig. 3. One can see that the RANS calculations are better predicting the pressure ratio whilst LES simulations seems to be in better agreement with adiabatic efficiency experimental data. Overall, the results are not as good as the one obtained by Hah (2009). Let us recall that these calculations were made on a 14 million-grid-point mesh and that the design and the stall calculations fit perfectly the experimental data. The choked mass flow rate of Hah (2009) is 20, 91 kg.s$^{-1}$, choke mass flow rate is (since this is the only operating point computed with the LES approach) 20, 95 kg.s$^{-1}$ for the 10, 25 and 100 million grid-point mesh (the experimental choke mass flow rate is 20, 93 kg.s$^{-1}$).

Local flow characteristics

Fig. 4 presents computational results at choke (which is the operating point computed with the LES approach), therefore the comparison is done between computational results calculated at choke and experimental results acquired at design point. Unfortunately, local experimental results are not available for choke operating point. However, following the comparison performed by Hah (2009), the nominal experimental data are plotted on the graph to assess the present results. Regarding the absolut angle, since the operating point is not strictly similar, the result are different and might be an explanation of the discrepancies observed. However, concerning the radial distribution of the pressure ratio, the shape of the LES curve is significantly influenced by mesh density. Furthermore,
the pressure deficit, which is of paramount interest, is not found in any of the LES calculations. Only the RANS simulation reveals a small pressure deficit. Hah (2009) shows that a corner stall develops at design and choked mass flow rate. In Figure 5, friction lines for the 10 million-grid-point mesh LES is shown, revealing no corner stall but a separation at $\sim 30\% \, x/c$. Even if the results of the present study are at choked mass flow rate, they are in contradiction with the ones given by Hah (2009). Furthermore, Shabbir et al. (1997) studied the effect of a hub leakage flow on the local distribution of
the pressure ratio for the NASA Rotor 37 and the NASA Rotor 35, and their results indicated that the pressure deficit near the hub can be induced by a leakage flow. Moreover, their results highlight the fact that the ratio of hub leakage mass flow over the total mass flow can be set to fit the experimental data quite perfectly. In fact, this parameter seems to be a major one and should be set properly, without trying to match the experimental data.

Figure 5: Friction line on the blade of the NASA Rotor 37 at choke, LES 10 million-grid mesh-point.

The radial distributions of efficiency of the LES calculations have a good shape compared to the experimental data, even though the operating point is slightly different. This tendency is visible for all the LES calculations. Since the computations are all at chock mass flow rate, it seems obvious that the absolute angle is not in agreement with experimental data. One of the major observation that can be done on the radial distributions is that the phenomenology observed in the three LES calculations is completely different than the RANS calculations (that is obvious, since the operating point is different) and it seems to have a grid density dependency for these parameters.

Mesh dependency
The comparison shown in Fig. 6 highlights the mesh dependency that is inherent to a LES simulation. The results presented are at choke mass flow. Indeed, one can see that in the NASA Rotor 37, there is an oblique shock (1), near the leading edge of the rotor, which interacts with the boundary layer inducing a separation (2). This separation provokes a secondary shock by reducing the passage section. The flow field near the trailing edge develops vortex shedding. The vortices are better resolved with the 100 million grid point mesh. Therefore, the structures captured with a finer mesh are smaller and gives accurate information on the flow field phenomenology.

One can notice that the shocks are thinner with the 100 million grid point mesh than with the 10 million grid point mesh. In fact, this results is emphasized when regarding the secondary shock. A major discrepancy is the interaction of the secondary shock with the pressure side, which triggers a turbulent transition for the 100 million grid point mesh (see (4) in Fig. 6(b)) but not with the 10 million grid point mesh (see Fig. 6(a)).

CONCLUSIONS
The methodology presented in this study helps choosing the time-step, assessing the quality of the different mesh and stopping the calculation at the adequate time. The convergence assessment shows
that the criterion of Ahmed and Barber (2005) should be restricted to the evacuation of unphysical frequencies.

LES helps investigating complex flow fields. However, at that point of the study, the current LES method does not show any clear improvements with respect to RANS based simulations. For the particular test case NASA Rotor 37, the results seem to indicate that no corner stall is represented.

Moreover, this observation is true for the finner mesh indicating that this phenomena is not mesh dependent.

ACKNOWLEDGMENT

The authors want to thank the Centre Informatique National de l’Enseignement Supérieur (CINES, Montpellier, FRANCE, project name fac6074) for the computational resources made available and CSG for the intern resources made available at CERFACS.

References


