Numerical predictions of low Reynolds number compressible aerodynamics

T. Désert a,b, T. Jardin b,*, H. Bézard a, J.M. Moschetta b

a ONERA, Toulouse, France
b ISAE-SUPAERO, Université de Toulouse, France

A B S T R A C T

Interest in low Reynolds number compressible flows is emerging due to prospective applications like flight on Mars and in the stratosphere. However, very little knowledge is available, both regarding the flow physics underlying this unique regime and the accuracy of numerical methods for its prediction. In this paper, low and high fidelity numerical approaches are compared with experimental measurements on both airfoils and rotors in the low Reynolds number compressible flow regime. It is shown that low fidelity approaches are suited to aerodynamic optimization despite high viscous and compressible effects. In addition, high fidelity approaches help reveal unique flow features of this regime.

Keywords:
Low Reynolds number
Compressible
Low pressure
Airfoils
Rotors
Mars

1. Introduction

The exploration of the planet Mars with Rovers started more than a decade ago, allowing a more detailed description of the planet surface than what was possible so far with Mars orbiters. However, a major drawback of these Rovers is their relatively low speed (partly due to harsh terrain) which, until now, only allowed them to explore a few tens of kilometers - to be compared with the 21,000 km planet circumference. One way to facilitate surface exploration would be to use autonomous flying vehicles which could act as scouts for the Rovers. Unfortunately, the atmosphere of Mars is far from flight-friendly. On the one hand, its density is low which (1) results in low Reynolds number flows that promote both flow separation and high viscous drag and (2) requires to be compensated for by higher speed of the lifting surface to provide a sufficient lift force. On the other hand, its composition and temperature are such that the speed of sound is relatively low, which is conducive to supersonic flow regimes. As a consequence, a typical flying vehicle designed for Mars exploration would operate in the low Reynolds number, compressible flow regime, for which very little knowledge is available.

Table 1 shows properties of the atmosphere of Mars at the ground level and compares it with properties of the atmosphere on Earth at sea level and 30 km above sea level (stratosphere). Note that the purpose here is to show orders of magnitude rather than precise time-averaged values of highly fluctuating properties. It is shown that the properties at 30 km above sea level are quite similar to those on Mars, at ground level, suggesting that vehicles flying in the stratosphere would operate under similar conditions to those described above, i.e. in the low Reynolds number, compressible flow regime, albeit with higher gravity force. The development of stratospheric vehicles has also recently gained interest for applications like Earth observation, telecommunications and navigation.

Overall, low Reynolds number, compressible flow regimes are very poorly documented because of the relatively new applications to which they are related. Apart from Mars and stratospheric flight, the development of high speed trains in low pressure tube (e.g. Hyperloop) could greatly benefit from deeper knowledge of these regimes, as well as liquid atomization where micro-sized droplets are formed and travel at high speed. Yet, a few authors brought to the fore unique features in low Reynolds number, compressible flows. For example, [1] numerically and experimentally investigated the flow past a triangular airfoil at Reynolds numbers 3000 and 10000 and Mach numbers 0.15 and 0.5. They showed that compressibility tends to elongate the wake, causing the transition from linear to non-linear lift and the subsequent vortex-induced lift to be delayed to higher angles of attack. [2] demonstrated similar wake elongation for the flow past a two-dimensional circular cylinder, below and above the critical Reynolds number of the first, Hopf bifurcation. Above the critical Reynolds number, this results in a larger vortex-shedding wavelength, i.e. a lower shedding fre-
quency. Wake elongation was found to be associated with higher drag and, in the stable state, with delayed separation.

In addition to little knowledge on the flow physics, it is not clear how conventional, numerical tools used to predict aerodynamic performance are suited to low Reynolds number, compressible flows. Yet, the design of small size unmanned vehicles flying in low pressure environment relies on the accuracy of such numerical tools.

In this paper, this issue is addressed by comparing predictions from low and high fidelity numerical tools with experimental results obtained under extreme conditions with typical Reynolds and Mach numbers in the range [100 – 10000] and [0.1 – 0.9] respectively. These numerical tools are applied to the flow past airfoils and rotors operating in such regimes. It is shown that while high fidelity approaches provide reasonable estimates of aerodynamic forces for all operating conditions, low fidelity approaches are limited to low angle of attack/pitch angle cases, which still make them suitable to aerodynamic optimization. In addition, unique features of low Reynolds number compressible flows are revealed, including, for example, wake elongation and subsequent damping of lift fluctuations, and displacement of shock foot far from the airfoil surface. Finally, experimental data for rotors (which are virtually unavailable in the literature) are reported and deeper insight into the resulting, enhanced aerodynamic forces (due to leading edge vortex stability) is provided by means of numerical approaches.

2. Numerical approaches

In this section, the theory and numerical procedures underlying three different models with increasing complexity are briefly described: (i) two-dimensional potential approaches, (ii) three-dimensional vortex lattice methods (VLM) and (iii) numerical simulations of the two and three-dimensional unsteady Navier-Stokes equations (NS).

2.1. Two-dimensional potential flow methods

Aerodynamic performance of two-dimensional airfoils is first evaluated with a potential flow panel method combined with an integral boundary layer formulation. The latter treats both laminar and turbulent layers and empirically determines the transition point using an $e^N$ method. Preliminary linear stability analysis on triangular and cambered airfoils up to a Reynolds number of 6000 were performed and showed that transition was triggered for $N < 1$. Therefore, in what follows, the empirical value for $N$ is fixed sufficiently large to avoid transition. Moreover, compressibility is intrinsically accounted for in the compressible boundary layer formulation and applied as a Karman-Tsien correction in the potential method. Xfoil code [3] is used to efficiently solve the problem via a global Newton method. Further details on the numerical procedure can be found in [3]. The number of panels used to discretize an airfoil is such that the solution is converged with respect to spatial discretization and is on the order of 160 for all cases shown in this paper.

2.2. Three-dimensional vortex lattice methods

A vortex lattice approach is used to predict the aerodynamic performance of rotors with relatively low computational cost. Numerical simulations are performed using ONERA’s in-house code PUMA, which combines lifting line and free wake models [4]. The lifting line approach relies on two-dimensional airfoil polars (obtained from resolution of the Navier-Stokes equations, see next section) and applies 3D correction to account for blade sweep and unsteady phenomena (e.g. dynamic stall). The free wake approach relies on the theory developed by [5] which describes the unsteady evolution of a wake modeled by a potential discontinuity surface. The blade and wake are discretized using 50 non-uniformly distributed radial stations (square root distribution for increased resolution near the tip). The computation is advanced in time using forward Euler scheme with a time step that corresponds to a 10° rotation of the blades. Four rotations are needed for initial transients to sufficiently decay.

2.3. Numerical simulations of the Navier-Stokes equations

The two and three-dimensional unsteady Navier-Stokes equations are numerically solved using a finite volume method. Compressible and incompressible formulations are considered, typically for Mach numbers above and below 0.2 respectively. Resolution is achieved using ONERA’s in-house code elsA [6] and StarCCM+—commercial code [7]. Spatial and temporal discretization schemes are of second order in both codes, with explicit and implicit temporal marching for elsA and StarCCM+ respectively.

2.3.1. Airfoil simulations

For both two-dimensional and three-dimensional airfoil cases at Reynolds numbers on the order of $10^4$ and subsonic Mach numbers, it was shown that a typical spatial resolution of $\Delta x/c = 0.01$ allows for the Richardson extrapolated lift and drag [8] to be approximated within 1.5%. An example of grid convergence is shown in Fig. 1a where the lift coefficient is displayed as a function of the typical grid spacing in the wake of the airfoil, $\Delta x/c$. 2D and 3D results are obtained for a 10° angle of attack triangular airfoil [1] at Reynolds and Mach numbers 3000 and 0.15, respectively. Fig. 1b shows Q-criterion isosurfaces colored by spanwise vorticity, obtained on the 3D configuration for typical grid spacings of $c/50$ (2 million cells) and $c/160$ (70 million cells). It is shown that although the largest grid spacing $c/50$ captures three-dimensional spanwise instabilities in the airfoil wake (leading to hairpin vortices), it is not able to capture them near the leading edge where the flow exhibits a nearly two-dimensional pattern. As such, it is interesting to note that for this spatial resolution, lift obtained from 3D simulation converges towards that obtained from 2D simulation. Note that the 3D case shown in Fig. 1 corresponds to a 3.3 aspect ratio wing with symmetrical boundary conditions at the tips. Other 3D configurations with non-slip walls at the tips and with a 0.3 aspect ratio wing will also be addressed in section 5.1. Subscripts $0.3$ and $3.3$ are used to denote aspect ratio 0.3 and 3.3 respectively.

<table>
<thead>
<tr>
<th>Table 1</th>
<th>Properties of Earth and Mars atmospheres.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Earth Sea level</td>
</tr>
<tr>
<td>Gas</td>
<td>$N_2$ (78%) $O_2$ (21%)</td>
</tr>
<tr>
<td>Gravity (m.s$^{-2}$)</td>
<td>9.81</td>
</tr>
<tr>
<td>Density ($kg.m^{-2}$)</td>
<td>1.225</td>
</tr>
<tr>
<td>Temperature (K)</td>
<td>288</td>
</tr>
<tr>
<td>Speed of sound (m.s$^{-1}$)</td>
<td>340</td>
</tr>
</tbody>
</table>
Fig. 1. Lift coefficient as a function of grid spacing obtained for 2D and 3D flows past a triangular airfoil (a). Q-criterion isosurfaces colored by spanwise vorticity contours obtained for grid spacings $\Delta x/c = 0.02$ (top) and $0.00625$ (bottom) on the 3D, aspect ratio 3.3 configuration (b).

Fig. 2. Low pressure experimental facility (a), sketch of the test chamber (b) and of the rotor test bench embedded in the chamber (c).

Furthermore, decreasing the time step beyond $\Delta t U/c = 0.02$ does not yield significant changes in the results. Therefore, in what follows, NS results will be shown for $\Delta x/c \leq 0.01$ and $\Delta t U/c \leq 0.02$. Structured (hexahedral cells) and unstructured (hexahedral trimmed cells) grids are used with elsA and StarCCM+ respectively, resulting in a typical number of cells on the order of $10^5$ for 2D cases and $2 \times 10^9$ and $2 \times 10^7$ for 3D cases with 0.3 and 3.3 aspect ratio wing, respectively.

Simulations are run for 100 convective times to ensure that initial transients have sufficiently decayed. Afterwards, time averaged quantities are obtained by averaging instantaneous values over 20 convective times.

2.3.2. Rotor simulations

For three-dimensional rotor cases, similar spatial resolution to that used for airfoil cases is used ($\Delta x/c = 0.01$) resulting in a typical number of cells on the order of $7 \times 10^7$. Again, structured (hexahedral cells) and unstructured (polyhedral cells) grids are used with elsA and StarCCM+ respectively. On the other hand the time step is larger ($\Delta t U/c = 0.08$), which is made possible by the quasi-steady nature of the flow on the rotor considered in this paper (as will be shown in section 5.3). Also note that $U$ is the velocity at the blade tip such that the blade span operates at velocities below $U$, hence at lower non-dimensional time step.

Simulations are run for 5 rotor rotations to ensure that initial transients have sufficiently decayed. Time averaged quantities are then obtained by averaging instantaneous values over one rotor rotation.

3. Experiments

3.1. Low pressure wind tunnel

Experimental data by [1] are used to assess the validity of numerical approaches for airfoils operating in the low Reynolds number, compressible flow regime. Data are obtained using the Mars Wind Tunnel (MWT) at Tohoku University. The MWT consists of an in-draft wind tunnel housed inside a vacuum chamber that allows low-density compressible experiments. It has a 100 by 150 mm test section with typical turbulence intensity below 1%. The airfoil is triangular with 30 mm chord length and 100 mm span (hence aspect ratio 3.3) and a maximum thickness of 1.5 mm. A scheme of the airfoil is provided in section 5.1, Fig. 3a. Typical operating conditions lead to a Reynolds number on the order of $10^3 - 10^4$ with Mach numbers on the order of 0.1 - 1. More details on the experimental setup can be found in [1].

3.2. Low pressure chamber

Experimental measurements of the aerodynamic performance of rotors operating in low pressure environment are conducted in a low pressure chamber of 18 m$^3$ volume at ONERA Fauga-Mauzac. Fig. 2a. The test bench consists of a 0.457 m diameter rotor manufactured out of carbon and rotated by means of a Faulhaber 4490 H 024B brushless motor, Fig. 2b. The propulsion set is fixed to a mast on top of which a 10 Newton strain gauge is mounted for thrust measurements. A similar gauge is used at the rear of the motor for torque measurements. Thrust and torque measurements
are acquired at a frequency of 10 kHz during 10 seconds, which ensures statistical convergence. In addition, each measurement is performed four times and reported values are obtained by averaging these four measurements. Data accuracy is derived from the maximum deviation of each measurement to the average value of the four measurements and from discrepancies between measurements and calibration weights (under standard, atmospheric pressure conditions). It was estimated to be below 5\% of reported values.

Experiments are conducted at a pressure of 2000 Pa (minimum achievable pressure is 10 Pa). Both Air and CO\(_2\) (96\%) were tested and yielded similar results, in agreement with previous observations by [9]. Therefore, Air was used for all cases shown in this paper.

4. Data reduction

Results are analyzed in terms of global and local aerodynamic performance as well as flow quantities.

Global aerodynamic performance of airfoils is assessed using lift 
\[ C_L = 2L/\rho SU^2 \] and drag 
\[ C_D = 2D/\rho SU^2 \] coefficients, where \( \rho \), \( U \), \( L \) and \( D \) are the fluid density, freestream velocity, lift and drag forces respectively. \( S \) is the surface area, which is equal to the wing chord \( c \) for 2D cases and to the wing chord times wing span \( c \times b \) for 3D cases. Global aerodynamic performance of rotors is assessed using the thrust 
\[ C_T = T/\rho \pi R^2 U^2 \] and torque 
\[ C_Q = Q/\rho \pi R^3 U^2 \] coefficients, where \( R \) and \( U \) are the rotor radius and blade tip velocity respectively.

Local aerodynamic performance of airfoils is assessed using the pressure coefficient 
\[ C_p = 2(p - p_{\infty})/\rho U^2 \] where \( p \) and \( p_{\infty} \) are the dynamic static pressure on and far upstream the airfoil, respectively. Local aerodynamic performance of rotors is assessed using the sectional thrust coefficient 
\[ C_{T_s} = sT/\rho SU^2 \] and sectional pressure thrust and \( S \) is the area of the blade spanwise section (or blade element).

Finally, flow features are displayed using non-dimensional vorticity \( \omega^* = \omega c/\pi U \), Q-criterion and pressure. Their corresponding oscillating frequencies \( f \) are non-dimensionalized using the wing chord and freestream velocities \( St = f c/\pi U \).

5. Results

5.1. Evaluation of numerical methods on triangular airfoil

The lift and drag coefficients and the lift-to-drag ratio obtained on a triangular airfoil using both potential flow (Xfoil) and Navier-Stokes (elsA and StarCCM+) solvers are first compared with experimental and numerical results from [1]. The potential flow solution is two-dimensional while solutions to the Navier-Stokes equations are presented for 2D and 3D cases. 3D cases with aspect ratios 0.3 and 3.3 are considered. These aspect ratios are chosen to be consistent with numerical simulations (aspect ratio 0.3) and experiments (aspect ratio 3.3) from [1]. An additional case taking into account wind tunnel walls is also simulated. Fig. 3 illustrates the airfoil profile and the aspect ratio 3.3, 3D case with wind tunnel test section. Note that this particular airfoil is here considered precisely because both numerical and experimental data in compressible, low Reynolds number flow conditions are readily available in the literature for comparison [1], [1] selected this triangular airfoil as a potential candidate for the design of propellers for Martian aircrafts due to its simple geometry, with sharp edges and flat surfaces. These characteristics (i.e. sharp leading edges and flat surfaces) were previously found to promote aerodynamic performance in the low Reynolds number compressible flow regime [10].

Fig. 4a compares two-dimensional numerical results with experimental results for Reynolds number 3000 and Mach number 0.5. Reasonable agreement between numerical and experimental data are observed at low angles of attack, in the linear regime where the flow is mostly attached. As \( \alpha \) increases and flow separation becomes significant, beyond \( \alpha \approx 5^\circ \), the lift curve obtained from 2D Navier-Stokes computations progressively diverge from the experimental curve and eventually reaches large discrepancies as the flow fully separates from the airfoil, beyond \( \alpha = 11^\circ \). The drag curve exhibits a rather similar trend, yet with better approximation of experimental data in the range \( \alpha \in [5^\circ - 11^\circ] \). On the other hand, lift and drag obtained from the potential flow solver remains consistent with experimental data up to large angles of attack.

Figs. 4b and 4c compare the lift obtained from 3D Navier-Stokes computations with that obtained from experiments. Fig. 4b focuses on simulations with spanwise extent 0.3 chord (referred to as 3DO3 NS). Recall that symmetrical boundary conditions are set at the wing tips which makes the configuration nominally two-dimensional but allows three-dimensional (spanwise) instabilities to develop and alter the overall flow structure. Again, it is shown that both lift and drag obtained from 3D Navier-Stokes computations are in reasonable agreement with experimental data for low angles of attack. In addition, there is no observable differences between 2D and 3D Navier-Stokes computations for \( \alpha < 11^\circ \). This suggests that three-dimensional effects are weak in this range of angle of attack and is consistent with the fact that the flow is not yet fully separated. Beyond \( \alpha = 11^\circ \), massive separation occurs and promotes three-dimensional spanwise instabilities. Hence, both lift and drag coefficients are reduced with respect to those obtained from 2D Navier-Stokes computations. Yet, despite this reduction, 3D Navier-Stokes computations with spanwise extent equal to 0.3 chord cannot recover experimental results.

Fig. 4c focuses on simulations with spanwise extent 3.3 chords (referred to as 3D03 NS). Here, both symmetrical and wall boundary conditions at the wing tips are considered. Comparing with previous results, it is shown that for symmetrical boundary conditions, the spanwise extent of the computational domain has no significant influence on aerodynamic performance, at least for spanwise extents above 0.3 chord. In other words, a spanwise extent of 0.3 chord is sufficient to capture three-dimensional instabilities and their influence on aerodynamic loads at high angles of attack. This is consistent with results from [1]. Conversely, taking into account wall boundary conditions has a significant impact on both lift and drag predictions and results in a much more accurate estimation for all three angles of attack tested.
Overall, these results suggest that 3D effects arising from the wind tunnel test section tend to limit flow separation. Therefore, 2D Navier-Stokes computations and 3D Navier-Stokes computations with symmetrical boundary conditions predict early separation, which leads to lift coefficients rapidly diverging from experimental values. On the contrary, because 2D potential flow solvers fail to predict massive flow separation (at least without special empirical treatment), the potential flow solution converges towards a partially attached flow even at high angles of attack. That is, the solution is somehow similar to that arising from 3D effects induced by wall boundary conditions. Therefore, despite the seemingly accurate prediction of aerodynamic loads over the full range of angles of attack tested, potential flow theory should not here be viewed as an accurate approach for high angles of attack aerodynamics.

Furthermore, because 3D Navier-Stokes computations with wall boundary conditions predict experimental data with reasonable accuracy, it can be inferred that 3D Navier-Stokes computations with symmetrical boundary conditions are suitable to the prediction of aerodynamic loads on a nominally two-dimensional configuration at large angles of attack. Building on that, the similarity between 2D and 3D numerical results at low angles of attack suggest that all 2D methods are suited to the prediction of aerodynamic loads when the flow is mostly attached, i.e. when three-dimensional effects are limited and where the efficiency of the airfoil is maximum. A direct outcome is that potential flow theory, as well as 2D Navier-Stokes computations, can be used with reasonable accu-
racy for airfoil shape optimization in the context of low Reynolds number compressible flows.

5.2. Reynolds and Mach number effects on cambered airfoil

In light of the above conclusions, two-dimensional Navier-Stokes simulations of the flow past a cambered airfoil are performed to highlight Reynolds and Mach number effects on aerodynamic performance in the low Reynolds number compressible flow regime. The airfoil is a circular cambered plate and has 6.35% camber and 1% thickness with sharp leading edge and blunt trailing edge and was previously found to exhibit relatively good performance at low Reynolds number [11]. Computations are performed for a range of angles of attack \( \alpha \in \{0°−15°\} \) (with a step of 1°), for Reynolds numbers 100, 1000, 3000 and 10000 and Mach numbers 0.1, 0.5, 0.7, 0.8 and 0.9.

5.2.1. Mach number effects

Fig. 5 shows the lift and drag coefficients and the lift-to-drag ratio obtained at Re=3000 for Mach numbers 0.1, 0.5, 0.7 and 0.8. First, it is observed that lift increases with Mach number for a broad range of angles of attack tested, i.e. \( \alpha \in \{2°−10°\} \). Yet, the trend is opposite below 2° and above 13°. This point will be discussed in the next paragraph. Second, it is shown that drag increases with Mach number in a more significant amount than lift, leading to a decrease in lift-to-drag ratio. Finally, it can be observed that the angle of attack corresponding to maximum lift-to-drag ratio decreases with Mach number.

To provide insight into the mechanisms responsible for these trends, Fig. 6 displays spanwise vorticity contours obtained for Mach numbers 0.1, 0.5 and 0.8 at Reynolds number 3000. For \( \alpha = 2° \), the flow separates near the trailing edge and the upper and lower surface, opposite sign vorticity layers interact at the trailing edge and roll up into opposite sign vortical structures, leading to a von Kármán vortex street. Alternatively, one can understand the emission of trailing edge vortices as the unsteady response of bound circulation to the emission of vortices induced by separation on the upper surface. It can be seen that the width of the vortex street slightly increases with the Mach number, indicating that the separation point moves upstream. The corresponding Strouhal number decreases from 2.17 to 2.07 and 1.79 at Mach 0.1, 0.5 and 0.8 respectively (see appendix for further analysis on unsteady response of forces and moments to vortex shedding). Similar observations can be made at \( \alpha = 5° \) and 8°. At this point, the effect of increasing the Mach number at a given \( \alpha \) can thus be viewed as similar to increasing \( \alpha \) at a given Mach number. This tends to increase lift (in the range of \( \alpha \) considered), which supports previous observations on lift trend.

For Mach=0.1 and \( \alpha = 10° \) the upper vorticity layer interacts with the upper surface of the airfoil and rolls up into a clockwise rotating vortex before reaching the trailing edge. This ‘early’ interaction leads to an upward deflected wake. This phenomenon is more clearly visible at \( \alpha = 12° \). For Mach=0.5, such a wake pattern is not observed at \( \alpha = 10° \) but appears at higher \( \alpha \), see for example \( \alpha = 12° \) and 15°. Conversely, the flow at Mach=0.8 still exhibits a non-deflected wake up to \( \alpha = 15° \). For the latter, it can be seen that the upper vorticity layer does not roll up into a vortex prior to interacting with the trailing edge, although the separation point appears very close to the leading edge. A direct outcome is that increasing the Mach number reduces the fluctuating loads on the airfoil. For instance, the relative standard deviation of the \( \alpha = 15° \) lift coefficient is equal to 0.104, 0.098, 0.039 and 0.019 for Mach numbers 0.1, 0.5, 0.7 and 0.8 respectively (see appendix). The absence of ‘early’ vorticity layer roll up can be correlated with the absence of rapid increase in the lift versus \( \alpha \) curve, which explains why lift increases as the Mach number decreases for the highest values of \( \alpha \) tested.

The time-averaged distribution of pressure coefficients \( C_p \) on Fig. 7 further highlights the vorticity roll up mechanism and its influence on aerodynamic loads at \( \alpha = 12° \). It is shown that the \( C_p \) distribution on the upper surface of the airfoil is rather flat at \( M = 0.8 \), where no roll up is observed. On the other hand, vorticity roll up at \( M = 0.1 \) induces a bump in \( C_p \) distribution, near \( x/c = 0.6 \). This bump is followed by a drop at the rear of the airfoil where vortices are shed and advected into the wake. This specific bump-drop pattern resembles that induced by a laminar separation bubble but is here the time-averaged footprint of the unsteady formation and shedding of clockwise rotating vortices. Fig. 7 also shows the \( C_p \) distribution at \( M = 0.9 \). It is interesting to note that the distribution is similar to that obtained at \( M = 0.8 \) although the flow is transonic (see Fig. 8). That is, it is shown that the presence of shocks do not significantly affect the pressure distribution in this low Reynolds number compressible flow regime. Here, enhanced viscous effects (i.e. smooth velocity gradients associated with thick boundary/shear layers) tend to move the shock foot away from the airfoil, which is a unique feature of low Reynolds number compressible flow regimes. In other words, extensively investigated phenomenon such as shock wave boundary layer interaction may not apply here, resulting in completely different responses of aerodynamic performance to transonic/supersonic regimes.

5.2.2. Reynolds number effects

Fig. 9 shows the lift and drag coefficients and the lift-to-drag ratio obtained at Mach=0.5 for Reynolds numbers 100, 1000, 3000 and 10000. The trends are in line with existing literature (under
incompressible conditions, e.g. [12]) demonstrating a decrease and an increase in lift and drag coefficients with Reynolds number, respectively. The increase in drag coefficient is particularly severe as the Reynolds number is decreased from 1000 to 100, where viscous drag becomes dominant. As a consequence, the lift-to-drag coefficient significantly increases with Reynolds number, being an order of magnitude greater at Re=10000 than at Re=100. In addition, it can be seen that the maximum lift-to-drag ratio is obtained at larger angles of attack as the Reynolds number is decreased. The dependence of aerodynamic performance on Reynolds number can here again be correlated with the vorticity flow fields depicted in Fig. 10.

Fig. 10 displays spanwise vorticity contours obtained for Reynolds numbers 100, 1000 and 10000 at Mach number 0.5. For
Fig. 9. Lift and drag coefficients and lift-to-drag ratios obtained from 2D Navier-Stokes computations for a cambered airfoil at Mach number 0.5 and Reynolds numbers 100, 1000, 3000 and 10000.

\( \alpha = 2^\circ \), the flow at \( \text{Re}=100 \) is characterized by two thick, opposite sign vorticity layers on the upper and lower surfaces of the airfoil, exhibiting a steady pattern. The flow at \( \text{Re}=1000 \) is rather similar, yet with thinner and stronger shear layers. As \( \text{Re} \) is increased to 10000, the upper and lower shear layers interact near the trailing edge, leading to an unsteady wake characterized by opposite sign vortical structures. These flow patterns are qualitatively similar at \( \alpha = 5^\circ \).

As \( \alpha \) is increased, the flow at \( \text{Re}=100 \) remains roughly unchanged. Conversely, it becomes unsteady at \( \text{Re}=1000 \), with the shear layers interacting near the trailing edge.

At \( \text{Re}=10000 \) and \( \alpha = 5^\circ \), the upper shear layer rolls up into a clockwise rotating vortex near mid-chord. The latter is advected towards the trailing edge where it eventually interacts with the lower shear layer. At \( \alpha = 10^\circ \), the roll up of the upper shear layer is more severe, which leads to a strong interaction with the upper surface and the subsequent deflected wake pattern (previously described). While the interaction between the clockwise rotating vortex and the upper surface of the airfoil at \( \text{Re}=10000 \) becomes stronger as \( \alpha \) is further increased, i.e. as the separation point moves upstream, the upper shear layer at \( \text{Re}=1000 \) is immune to roll up before interacting with the lower shear layer at the trailing edge.

Fig. 10. Instantaneous spanwise vorticity flow fields obtained from 2D Navier-Stokes computations for a cambered airfoil at Mach number 0.5 and Reynolds numbers 100, 1000 and 10000. Snapshots are shown for six angles of attack \( \alpha = 2^\circ, 5^\circ, 8^\circ, 10^\circ, 12^\circ \) and \( 15^\circ \).
Fig. 11. Comparison of thrust and torque coefficients and thrust-to-torque coefficients ratio obtained from numerical approaches and experiments on a two-bladed rotor.

Fig. 12. Surface pressure contours and Q-criterion isosurfaces obtained from 3D Navier-Stokes computations for a two-bladed rotor with pitch angle 15°, 19°, 25°, and 30°.

Therefore, while the wake at Re=1000 exhibits a relatively simple pattern characterized by alternate vortices up to α = 15°, the flow at Re=10000 transits to a chaotic state (see also fluctuating forces and moments in the appendix).

The roll up of the upper shear layer at Re=10000 explains the sudden increase in lift observed at α = 8° on Fig. 9. Conversely, the robust shear layers at Re=100 and 1000 explain the smooth increase of the lift coefficient with α. That is, at the lowest Reynolds numbers, there is no drastic changes in the flow as α is increased, leading to a relatively simple relation between lift and the airfoil projected area c × sinα.

Overall, the present results show that, in the low Reynolds number compressible flow regime, both Reynolds and Mach numbers have significant impact on the flow pattern and the resulting aerodynamic performance. In particular, changes in aerodynamic performance due to variations in Reynolds and Mach numbers are on the same order of magnitude. Therefore, for a given characteristic dimension (e.g. wing chord) and envirorning pressure, there exists an optimal operating speed (i.e. an optimal Re-Mach couple) that leads to optimal aerodynamic performance. This is fundamentally different to conventional aerodynamics where large Reynolds and Mach number effects are usually uncoupled.
5.3. Evaluation of numerical methods on two-bladed rotor

In this section, the comparison between numerical approaches and experimental data is extended to the flow past a two-bladed rotor operating under hovering conditions. Based on the previous results that 2D numerical approaches are suited to the prediction of aerodynamic loads in the low Reynolds number compressible flow regime, around maximum efficiency, 3D vortex lattice methods are here applied using polars from 2D Navier-Stokes computations and compared to 3D Navier-Stokes computations and experiments (Fig. 11). The blades consist of a 6.35% camber, 1% thick airfoil (see section 5.2) with constant chord and constant twist angle \( \beta \) along the span. It is similar to that tested in [13]. The rotation speed and ambient pressure are such that the Reynolds and Mach numbers at the blade tip are on the order of 6000 and 0.35 respectively.

Fig. 11 shows a relatively good agreement between NS and VLM approaches. These numerical methods, however, overestimate both experimental thrust and torque coefficients. Discrepancies in thrust and torque predictions somehow compensate each other, leading to a fair approximation of the thrust-to-torque ratio. Despite these quantitative discrepancies, the trends in thrust and torque versus pitch angle are qualitatively similar for numerical and experimental approaches. Overall, \( C_T \) increases to a maximum value at \( \alpha \approx 30^\circ \) and \( C_Q \) continuously increases with \( \alpha \). As a consequence, \( C_T/C_Q \) exhibits an optimal value, which is found to be around 10\(^\circ\) pitch angle.

A closer look at the NS and VLM curves reveals closer match between both approaches at low pitch than at high pitch angles. At high pitch angles, results obtained from Navier-Stokes simulations slightly overestimate VLM results. This ‘extra-lift’ obtained with NS simulations may arise from the development of a stable leading edge vortex on the upper surface of the rotor blades. This intrinsically three-dimensional mechanism cannot be predicted by VLM. Fig. 12 displays iso-surfaces of Q-criterion obtained from NS simulations for pitch angles 15\(^\circ\), 19\(^\circ\), 25\(^\circ\) and 30\(^\circ\). At higher pitch angles (\( \beta > 20^\circ \)), it is shown that the flow separates at the leading edge and rolls up into a conical leading edge vortex (LEV). On the contrary to the LEV that develops on a two-dimensional airfoil and that eventually sheds into the wake, the LEV here remains stably attached to the blade, inducing a sustained low pressure suction region on its upper surface, which in turn generates thrust. Stability of the LEV is a common feature in the aerodynamics of low aspect ratio flapping and revolving wings (which operate in a comparable range of Reynolds numbers) and is hypothesized to result from rotational accelerations, making it a fundamentally three-dimensional phenomenon [14,15]. Because of this stability, sectional aerodynamic forces on the inboard part of the blade are steady (in contrast to those on two-dimensional airfoils, see appendix). In the outboard region of the blade, i.e. near the tip, it can be observed that the LEV bursts into smaller scale structures as it merges with the tip vortex. This phenomenon is also in line with previous numerical and experimental observations at low Reynolds numbers (e.g. [16–18]).

Fig. 13 compares the sectional thrust distribution along the rotor blade obtained from NS computations with that obtained from VLM. While reasonable agreement between both approaches is observed up to 25\(^\circ\) pitch angle, non-negligible discrepancies are ob-
served at 30° pitch angle where the effect of the LEV on thrust is maximum. Discrepancies observed in the range 0.2 < r/R < 0.8 are partly compensated for by discrepancies at the tip where the dynamics of the vortical structures (merging between LEV and tip vortex) is also very complex and cannot be captured through VLM. As such, despite the relatively good agreement observed on Fig. 11, VLM should not be viewed as an accurate method for high pitch angles where separation occurs at the leading edge. Note however that special treatment of the two-dimensional polars used in VLM can be added to account for rotational effects on LEV stability [19]. Yet, because maximum efficiency is obtained at low pitch angles, present results suggest that VLM is suited to the aerodynamic optimization of rotors in the low Reynolds number compressible flow regime.

6. Conclusion

Flight on Mars and in the stratosphere, high speed trains in low pressure tubes and liquid atomization share unique atmospheric features in that they involve bodies moving in a compressible viscous flow, i.e. they operate in a low-to-moderate Reynolds number, compressible flow regime. Because of the very perspective nature of these applications, very little knowledge is available on the physics of low Reynolds number, compressible flows. Furthermore, it is not clear whether conventional numerical methods used to predict aerodynamic forces on moving bodies are suited to such flow regime.

In this paper, the accuracy of low and high fidelity numerical approaches in the prediction of aerodynamic forces at low
Reynolds numbers and moderate subsonic Mach numbers was assessed. Assessment was achieved by comparing numerical results with experimental data on airfoils from [1] and with present experiments on rotors. It was shown that high fidelity approaches, i.e. numerical resolution of the 3D Navier-Stokes (NS) equations, provide results in generally good agreement with experiments for the whole range of operating conditions tested. On the other hand, it was shown that numerical resolution of the 2D Navier-Stokes equations, potential flow approaches and Vortex Lattice Methods (VLM) provide reasonable estimation of aerodynamic forces when the angle of attack/pitch angle is low and the flow is attached (or weakly separated). For high angles of attack/pitch angles, these low fidelity approaches fail to predict massive flow separation and subsequent 3D effects, sometimes despite reasonable agreement with experimental data (which was shown to result from compensating errors). These results indicated that, in the low Reynolds number compressible regime, airfoil and rotor optimization can be achieved with reduced order models.

In addition, the influence of Reynolds and Mach number effects on the flow past airfoils was analyzed and significant impact on flow separation and subsequent wake patterns was demonstrated. In particular, results indicated that compressibility tends to elongate the airfoil wake and prevent early roll-up of the upper shear layer, in line with previous observations by [1,2]. This resulted in a re-orientation of the incompressible deflected wake and to a smoother lift-to-angle-of-attack curve and a damping of lift fluctuations. Moreover, unique features associated with the displacement of the shock foot away from the airfoil surface were revealed for
transonic operating conditions. On the other hand, Reynolds number effects were found to be qualitatively similar to those reported in extensive works at low Mach numbers. Overall, it was demonstrated that in the low Reynolds number compressible flow regime, the impact of both Reynolds and Mach numbers on aerodynamic performance are on the same order of magnitude which suggests that for a given characteristic dimension and environing pressure, there exists an optimal operating speed that leads to optimal aerodynamic performance.

Finally, experimental data for rotors operating in the low Reynolds number compressible regime were provided and, along with numerical results, similar flow pattern to that observed on bio-inspired flapping and revolving wings (i.e. stability of the leading edge vortex and enhanced thrust production) were analyzed. These results provided insight into the physics of low Reynolds number compressible flows and into the accuracy of numerical approaches for their prediction, thereby helping future design of Mars and stratospheric flying vehicles.

### Declaration of Competing Interest

We declare that we have no competing interest.

### Acknowledgements

The authors gratefully acknowledge the Centre National d’Études Spatiales (CNES) for partly funding this work. They are also thankful to Rémy Chanton, Sylvain Belliot, Henri Dedieu and Valérie Allain from ISAE-SUPAERO for their technical support.

### Appendix A. Unsteady forces and moments on cambered airfoil

Unsteady responses of lift, drag and pitching moment obtained on a cambered airfoil at Reynolds number 3000 and Mach numbers 0.1, 0.5 and 0.8 are further analyzed for angles of attack 2°, 8° and 15°. Fig. 14 plots the instantaneous fluctuating parts of lift, drag and pitching moment coefficients ($C_l'$, $C_D'$ and $C_m'$ respectively) as a function of the convective time. Note that the pitching moment coefficient is defined as $C_m' = 2m/ρScU^2$, where $m$ is the pitching moment about a spanwise axis located a quarter chord downstream of the leading edge.

At $\alpha = 2°$, it is seen that amplitudes of force and moment fluctuations decrease as the Mach number is increased from 0.1 to 0.5 but then increase slightly as the Mach number is further increased to 0.8. As observed in section 5.2, this non-monotonic trend may result from two competing mechanisms. On the one hand, increasing the Mach number at a given Reynolds number tends to move the separation point upstream, hence a larger portion of the airfoil is under the influence of the unsteady, separated leading edge shear layer. On the other hand, it tends to stabilize the separated leading edge shear layer which seems immune to ‘early’ roll-up. At higher angles of attack $\alpha = 8°$ and 15°, the first mechanism is relatively weaker since separation already occurs close or at the leading edge even at low Mach numbers. Hence force and moment fluctuations monotonically decrease with Mach number as a result of the stabilization of the leading edge shear layer.

In addition, it is shown that the Strouhal number generally decreases with increasing Mach number. At $\alpha = 2°$, $St = 2.17, 2.07$ and 1.79 for Mach number 0.1, 0.5 and 0.8 respectively. At $\alpha = 8°$, $St = 1.78, 1.65$ and 1.33 for Mach number 0.1, 0.5 and 0.8 respectively. At $\alpha = 15°$, the Strouhal number first decreases from 0.75 to 0.59 as the Mach number is increased from 0.1 to 0.5, and then increases to 0.78 as the Mach number is further increased to 0.8. The precise reason for this non-monotonic trend is unclear at this point but it should be noticed that for such high angles of attack, strong non-linearities may trigger additional instabilities that may affect the whole shedding process. In particular, although not shown here for the sake of conciseness, the $\alpha = 15°$ case at Mach number 0.8 exhibits a low-frequency modulation of the aerodynamic loads, about 27 times lower that the fundamental vortex shedding frequency.

Instantaneous fluctuating parts of lift, drag and pitching moment coefficients at Mach number 0.5 and Reynolds numbers 100, 1000 and 10000 are displayed on Fig. 15. It can be seen that, for all three angles of attack $\alpha = 2°$, 8° and 15°, increasing the Reynolds number tends to increase both amplitudes and frequencies of fluctuations. In particular, and according to results shown in Fig. 10, force and moment signals are steady at $Re = 100$ and may exhibit chaotic fluctuations as the Reynolds number is increased to 10000.

### References