Direct Numerical Simulation of Transitional Flow over a NACA66 Propeller Section

A.Ducoin\textsuperscript{1}, J. A. Astolfi\textsuperscript{2}, S. Sharma\textsuperscript{3}

\textsuperscript{1}LHEEA laboratory, Ecole Centrale de Nantes (CNRS UMR 6598), 44300 Nantes, France
\textsuperscript{2}French Naval Academy Research Institute (IRENav, EA3634), 29240 Brest Cedex 9, France
\textsuperscript{3}CORIA Laboratory-UMR 6614, CNRS-University & INSA of Rouen, St. Etienne du Rouvray, France

ABSTRACT
The objective of this paper is to investigate the behavior of laminar to turbulent transition on a NACA66TMB laminar propeller section. Direct Numerical Simulation are performed using the massively parallelized open source code NEK5000 that uses the high order spectral element method to solve the incompressible Navier Stokes equations. It is validated by comparing the obtained wall pressure coefficient with measurements performed in hydrodynamic tunnel at the French Naval Academy Research Institute (IRENav). The Reynolds number is Re=225,000, and the angle of attack is $\alpha$=4° which induces a laminar to turbulent transition close from the trailing edge of the hydrofoil.

The results suggest that a very sharp transition occurs upstream the Laminar Separation Bubble (LSB). This induces a high and localized level of wall pressure fluctuations with periodic behavior just upstream the LSB, followed by a rapid development of turbulence. This work will help to get a better understanding of cavitation inception on propeller sections, which typically occurs in the transitional region.

Keywords
Direct Numerical Simulation, laminar to turbulent transition, propeller section

1 INTRODUCTION
The laminar to turbulent transition occurring on propeller blades is known to be critical for the body performance and its structural integrity. For many marine applications operating at low angles of attack, laminar profiles can be chosen for the blade in order to reduce the friction at the body surface to get better performances. However, this lead to further development of transitional regimes, even at high Reynolds numbers where the boundary layer is, however, usually considered as fully turbulent. Hence, numerical and experimental investigations are also necessary to help the performance prediction and to increase the physical understanding of cavitation inception at the blade surface, which is known to incept inside the Laminar Separation Bubbles (LSB) in case of transitional flows.

Typically, this transition occurs at low to moderate Reynolds numbers and is triggered by a laminar separation and a reversed flow, due to an adverse pressure gradient. The development of the turbulent flow, which causes a momentum transfer in the wall normal direction, allows the flow to reattach, to form a so called laminar separation bubble (LSB) (Gaster 1969). Upstream of the LSB, the flow is usually highly unsteady and is governed by complex mechanisms identified as primary and secondary instabilities that lead to transition to turbulence.

Several experimental works were carried out on laminar sections in the past at the Naval Academy Research Institute (IRENav), France. Under relatively high Reynolds numbers (Re=300,000 to 1,000,000), highly transitional flows has been observed. It was shown that a very strong and localized transition to turbulence occurs inside the boundary layer, which induces intense pressure fluctuations (Ducoin et al. 2009). Moreover, it has been demonstrated that this type of transition induces important structural vibrations with low damping that can get close to resonance with the blade natural frequencies (Ducoin et al. 2012). These laminar profiles are indeed obtained by moving the maximum thickness to the center of the chord, which reduce the adverse pressure gradient and hence increase the critical Reynolds number Re$_\theta$ of transition, see Figure 1. It results in a delay of transition as compared conventional, non-laminar profiles.

From the numerical point of view, DNS have been performed and validated successfully on an aerodynamic Sd7003 section under different Reynolds numbers, using the massively parallelized open source code NEK5000 (Ducoin et al. 2016), which is used in the present study. The results have shown the ability of the current method to characterize the transitional region, and to extract the main instability mechanism that lead to turbulent flow.

The objective of the present work is then to characterize the specificity of transition on laminar section as compared to classic aerodynamic profiles, and to evaluate the effect on the overall blade performance. First, the mean flow is compared with the RANS calculation to check the validity of the current model. Then, the transitional flow is analyzed. The mechanism that leads to
transition is investigated together with the behavior of wall pressure fluctuations.

Figure 1. Critical Reynolds number Reθ along the chord obtained with Xfoil. Comparison of laminar (naca0012) and conventional (naca0012 and sd7003) profiles at Re=225,000.

2 EXPERIMENTAL METHOD

Measurements are carried out in the cavitation tunnel at IRENav, France. The test section is 1 m long and has a h=0.192 m square section. The velocity can range between 0 and 15m/s and the pressure from 30mbar to 3bars. The hydrofoil is a NACA 66 which presents a camber type NACA a=0.8, a camber ratio of 2% and a relative thickness of 12% (Leroux et al. 2005). It is mounted horizontally in the tunnel test section (Figure 2). The chord is c=0.150m and the span is b=0.191m. It corresponds to a low aspect ratio b/c =1.3 and a confinement parameter h/c=1.28. Flow visualizations based on the cavitation inception on the suction side show that about 80 to 90% of the foil surface can be considered as a 2D flow (Leroux et al. 2005), depending on the angle of incidence.

Pressure measurements are carried out using seventeen piezo-resistive transducers (Keller AG 2 MI PAA100-075-010) of 10 bars maximum pressure. The pressure transducers are mounted into small cavities with a 0.5 mm diameter pinhole at the hydrofoil surface. The wall pressure spectrum measured by the transducer is attenuated from the theoretical cut-off frequency f_c=9152Hz. Experiments are led with a sample frequency of f=20kHz. The transducers locations are given in Figure 2. As shown, a main set of ten transducers is aligned along the chord on the suction side at mid-span, where the flow is considered as quasi-2D. It starts from the leading edge at reduced coordinate x/c=0.1 up to the trailing edge at coordinate x/c=0.90 with a step of 0.1c. Because the pressure transducers were designed for higher pressure signals i.e higher Reynolds numbers (typically Re>500,000) than the Reynolds number considered (Re=225,000). The ratio between, the noise and the pressure signal is then too high to extract correctly the effect of transition on wall pressure. Hence, it is necessary to remove this noise, as well as low frequencies induced by the pump or the natural frequency of the hydrodynamic tunnel. To do that, the signal is decomposed using the Empirical Mode Decomposition (EMD) (Huang et al. 1998), to extract the physical modes from the total pressure signal. This methods were used in the past for similar hydrodynamic studies (Benramdane et al. 2007) and (Ducoin et al. 2007)). This method consists on the decomposition of an unsteady signal x(t) into intrinsic oscillatory components called Intrinsic Mode Functions (IMFs) by means of an algorithm called sifting process. The basic principle is the extraction of intrinsic time scale components of the signal starting from finer temporal scales (high frequency modes) to coarser ones (low frequency modes). The total sum of the extracted IMFs matches the signal and therefore ensures signal complete reconstruction. (Huang et al. 1998) have introduced the EMD method for analyzing data from unsteady and nonlinear processes. The reconstructed signal can be written as:

\[
x(t) = \sum_{j=1}^{n} IMF_j(t) + r_n(t)
\]

where \( n \) is the number of modes, \( r_n(t) \) is the residue, and \( k \) is the low frequency cut-off.

3 COMPUTATIONAL METHODS

3.1 Direct Numerical Simulation

The dynamics of a three-dimensional incompressible flow of a Newtonian fluid are described by the Navier-Stokes equations

\[
\frac{\partial U}{\partial t} = - (U \cdot \nabla) U - \nabla P + Re^{-1} \Delta U
\]

where \( U = (u, v, w)^T \) is the velocity vector and \( P \) the pressure term. The velocity is non-dimensionalized by the upstream velocity \( U_\infty \) and the flow conditions are set according to the Reynolds number \( Re = U_\infty c/\nu \), where \( \nu \) is the kinematic viscosity of the water.

The Navier-Stokes equations are solved using the flow solver NEK5000 developed at Argonne National Laboratory by (Fisher et al. 2008). It is based on the spectral elements method (SEM), introduced by (Patera...
1984), which provides spectral accuracy in space while allowing for the geometrical flexibility of finite element methods. Spatial discretization is obtained by decomposing the physical domain into spectral elements within which the velocity is defined on Gauss-Lobatto-Legendre (GLL) nodes and the pressure field on Gauss-Legendre (GL) nodes. The solution to the Navier-Stokes equations is then approximated within each element as a sum of Lagrange interpolants defined by an orthogonal basis of Legendre polynomials up to degree N. The results presented in this paper have been obtained with a polynomial order of N=10. The convective terms are advanced in time using an extrapolation of order 3, whereas the viscous terms use a backward differentiation of order 3 as well, resulting in the time-advancement scheme labelled BDF3/EXT3. Nek5000 employs the MPI standard for parallelism (Deville et al. 2002). The computations are performed on the IBM Blue Gene/Q (Turing) at GENCI/IDRIS and uses up to 16000 CPUs for the higher order mesh.

For further details about the spectral elements method, the reader is referred to the books by (Deville et al. 2002) and (Karniadakis & Sherwin 2005).

3.2 Numerical setup
Because of the high Reynolds number considered, the DNS domain is reduced to the near wall region, and velocity boundary conditions are imposed at the domain boundaries from a transitional RANS calculation to reproduce the velocity gradient external to the hydrofoil boundary layer, see Figure 3. Hence, the total height of the DNS domain is 0.25c and 0.5c is set in the wake. The span has been reduced to 0.05c. The domain of the RANS calculation corresponds to the dimensions of the experimental test section of the hydrodynamic tunnel at IRENav.

The velocity profiles extracted from the RANS calculation are shown in Figure 4. In this calculation, the upstream velocity is 1.5m/s, leading to Re=225,000. Due to confinement of the test section, the chordwise velocity reach almost 2m/s at the leading edge of the hydrofoil, whereas it tends to the upstream velocity $U_\infty=1.5$m/s in the wake. $\nabla U \cdot x = 0$ is set at the outlet, whereas a no slip condition is set on the wing surface. A periodic boundary conditions is imposed on the vertical side planes of the domain.

Figure 3. Computational domain of the NACA66 hydrofoil.
of 188,480,000 cells.

As the DNS code is semi-implicit, it requires the local CFL number to be strictly $CFL = \frac{\Delta u}{\Delta x} < 0.75$. The computations are carried out with $CFL \approx 0.5$.

### 3.3 Isotropic von Kármán turbulence model

In order to add the perturbation at the inlet of the computational domain, an isotropic von Kármán turbulence model (von Kármán 1937) is implemented into the NEK5000 code.

This model is based on the energy spectra of the perturbations. The energy spectrum is set numerically by using random number generating algorithms (Park & Miller 1988). Hence the random angles corresponding to each random number is generated resulting in a randomized field. The von Kármán isotropic turbulence model considers the linear velocity and the angular velocity components of the disturbances as spatially varying stochastic processes and specifies power spectral density to each of them.

### 3.4 Convergence study

A convergence study is lead according to the element order from O(6) to O(10). First, the flow in the transition region is analyzed. Figure 7 show the velocity field in the unsteady region of the LSB. The lower order mesh (O(6)) clearly show that the mesh is too coarse to correctly capture the LSB vortex shedding. Numerical instabilities disturb the unsteady region of the LSB, which results in the formation of small structures, and hence further numerical instabilities are forming downstream. Starting from O(8), this numerical effect seems suppressed and the shedding process is correctly captured.

![Figure 6. Turbulent structures at the inlet of the computational domain, contours of w](image)

![Figure 7. Convergence of the LSB shedding. Velocity contours downstream the laminar separation bubble for 3 different element order.](image)
frequency of 125Hz is observed against 87Hz and 85Hz for O(8) and O(10), respectively. The drag coefficients ($C_D$) is converged for the lower order mesh, whereas the lift coefficient ($C_L$) converges for O(8).

<table>
<thead>
<tr>
<th>Element order</th>
<th>$C_L$</th>
<th>$C_D$</th>
<th>$f_{shed}$ (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>O(6)</td>
<td>0.8198±0.055</td>
<td>0.0213±0.0049</td>
<td>125</td>
</tr>
<tr>
<td>O(8)</td>
<td>0.8272±0.061</td>
<td>0.0219±0.0034</td>
<td>87</td>
</tr>
<tr>
<td>O(10)</td>
<td>0.8274±0.062</td>
<td>0.0215±0.0057</td>
<td>85</td>
</tr>
</tbody>
</table>

Table 1. Convergence of the lift and drag coefficients and of the LSB vortex shedding frequency as function of the element order.

4 RESULTS AND DISCUSSION

4.1 Comparison with RANS calculation

To check the validity of the RANS velocity boundary conditions inside the DNS domain, velocity profiles are shown in Figure 8. The boundary layer thicknesses are in good agreement; however the DNS calculation shows higher external velocities compared to the RANS calculation due to confinement of the computational domain. It is observer to be about 8% higher. This numerical error is corrected in the results in section 4.2, i.e for the pressure coefficient calculations and the shedding frequencies to compare with experimental values. As expected, the velocities are the same at the upper boundary as RANS profiles are imposed in the DNS calculation.

4.2 Pressure fluctuations: validation with experiments

First, a complete map of the wall pressure coefficients along the chord, during t=0.7s is shown in the 3D plot in Figure 9. Starting $x/c=0.75$, periodic fluctuations are observed that growth very quickly and reach its maximum at $x/c=0.8$, after what it decreases and becomes more random.

The wall pressure coefficient obtained with the converged DNS at order O(10) is then compared with the measurements performed in the hydrodynamic tunnel. For each signal, the low and high frequencies components are removed to compare with calculations. Because only 3 pressure measurements can be analyzed (each $x/c=0.1$), we have choose to advance the DNS signal by 0.05, assuming that the higher external velocities observed in Figure 8 could advance the transition.

Figure 8. Comparison of RANS and DNS boundary layer velocity profiles upstream the transition region.

Figure 9. Time evolution of computed wall pressure coefficient fluctuations along the chord of the hydrofoil

The Figure 10 (a) show a low and random pressure level in the stable region of the LSB, where the high amplitudes are due to noise. Starting $x/c=0.8$, the pressure fluctuations becomes periodic due to LSB vortex shedding. The DNS signal at $x/c=0.75$ match correctly with the experimental signal at $x/c=0.8$, including the period of LSB shedding and the intermittency. Around $x/c=0.9$, the fluctuations becomes random with higher amplitudes, due to development of turbulent flow inside the boundary layer.

Figure 10 (a) $x/c=0.65$ (DNS), $x/c=0.7$ (exp)

(b) $x/c=0.75$ (DNS), $x/c=0.8$ (exp)
Figure 10. Comparison of computed and measured wall pressure coefficient.

The frequency spectra obtained from the pressure signals of Figure 10 (b) is represented in Figure 11. A peak around 85Hz is observed numerically, whereas it is around 80Hz experimentally. Hence, the vortex shedding frequency of LSB compares well between the DNS and the experiments.

Figure 11. Comparison of Experimental and numerical pressure spectra at x/c=0.8.

4.3 Flow Analysis

To analyze the behavior of wall pressure downstream the LSB, coherent structures colored with pressure coefficient contours are shown in Figure 12 for two periods of LSB vortex shedding. It is plotted just after the LSB starting x/c=0.65, up to the trailing edge. In Figure 11 (a), a 2D vortex forms due to development of Kelvin Helmholtz instability, together with a medium level of negative pressure (blue sky). When the vortex rolls up, it gets accelerated and decrease in size, and generates a high level of depression (dark blue). Following this, the vortex blows up into smaller structures and the pressure suddenly increases, shown by green/yellow color plots. Those small turbulent structures are first organized in packed and then mix together to form fully turbulent flow with uniform positive pressure (in red).

The intermittent behavior is also observed. The pressure intensity of the shed vortex seems proportional to the size of the 2D KH vortex. It is for example larger in Figure 12(a), leading to high depression inside the convected vortex in Figure 12 (c) and (d); whereas it is significantly smaller in Figure 12(c) (second KH vortex), leading to lower depression peak in Figure 12(e) and (f) when it sheds.

To observe the development of turbulence in the hydrofoil boundary layer, spectra of the upstream velocity \( V_X \) is shown in figure 13 for three locations. The monitors are taken outside of the boundary layer, compared to the one in section 4.2 that are taken near the wall i.e in the viscous sub layer. At x/c=0.7, only the LSB vortex shedding frequency around 90Hz is visible on the spectra. At x/c=0.85, this peak is still observed with higher amplitude, but the development of turbulence leads to the inception of higher frequency components. Finally, at x/c=1, the vortex shedding frequency is not visible, and the spectra shows a classical -5/3 slope resulting from the turbulent cascade.

Figure 12. Coherent structures (iso \( \lambda_2 \)) colored with wall pressure coefficient contours downstream the LSB.
5. CONCLUSION

In this paper, a first DNS is performed on a laminar NACA66 propeller section. Strong hypothesis has been made in order to be able to perform a full DNS at Re=225,000, which is the minimum Reynolds number, which allows to perform and process pressure measurements in hydrodynamic tunnel.

It is observed that the pressure gradient and the location of transition inception (i.e. laminar boundary layer separation) at the hydrofoil suction side is correctly reproduced by the DNS, whereas a higher external velocity to the boundary layer is observed due to confinement effect induced by the constrained domain.

A good agreement is observed on the wall pressure fluctuations between the present DNS and the experiments. The frequency of LSB vortex shedding is captured by the DNS, and so its time behavior including the intermittency downstream the LSB, and the transition to turbulence highlighted by random pressure fluctuations. The numerical results suggest that very strong level of fluctuations occurs in the establishment of the LSB vortex shedding, corresponding possibly to a bursting phenomenon. This pressure level seems proportional to the size of the shed vortex.

Future experimental and numerical works will aim at demonstrate the existence of this sharp transition as well as the high level of local depressurization observed before the transition to turbulence.

ACKNOWLEDGEMENTS

This work was performed using HPC resources of GENCI/IDRIS (Grant 2016-[100631]) at Orsay, France on the IBM Blue Gene/Q (Turing). Flow visualizations have been made using the open-source software VisIt (Childs 2012).

REFERENCES

M. Gaster (1969). The Structure and Behaviour of Laminar Separation Bubbles, HMS Q.


A. Ducoin, S. Benramdane, J. A. Astolfi, F. Deniset (2007), Etude expérimentale et numérique du champ de pression pariétale sur corps portant en
mouvement forcé, in: 18ème Congres Français de Mécanique.


