NUMERICAL ANALYSIS OF INSTABILITIES FOR A LOW SPEED FAN BLADE

Kuen-Bae Lee  
Mechanical Engineering  
Department  
Imperial College London  
kuenbae.lee@imperial.ac.uk  
London, UK

Mark Wilson  
Rolls-Royce plc  
mark.wilson@rolls-royce.com  
Derby, UK

Mehdi Vahdati  
Mechanical Engineering  
Department  
Imperial College London  
m.vahdati@imperial.ac.uk  
London, UK

ABSTRACT

In recent years civil turbofan engine designs have moved toward high mass flow and low-pressure ratio fan designs to improve fuel consumption. As the fan (and hence intake) diameter increases, shorter intakes are required to reduce the overall weight and drag of the engine. The major emphasis of this work is the effects of intake on fan blade operation, and addresses the possible aero-mechanical and aerodynamic instabilities which can arise for such a fan-intake system. The work is carried out by using a validated three dimensional unsteady CFD model (AU3D), and for a modern low speed fan rig for which an extensive measured data is available.

In the first part, steady and unsteady flow simulations were conducted for this blade, at 80% speed, for which the measured constant speed characteristic includes a part with positive slope. The numerical study indicated that, for the points on the positive slope part of the characteristic, the flow contains a multi-cell, part-span rotating stall, which can only be modelled by using an unsteady whole assembly approach. Although this type of rotating stall does not result in surge or massive loss of power, it can cause excessive vibration and noise.

In the second part, flutter simulations were conducted for the test blade to compare the numerical results against measured data and to validate the CFD model used. The results were in a good agreement with measured data, indicating the suitability of this model for flutter simulations. Subsequent computations were performed to understand the effects of changing intake length on the flutter stability of this blade.

Finally, the effects of the inlet distortion on the stability of the fan blade was explored by considering crosswind. The results showed that the non-homogeneous flow field caused by the crosswind can improve the flutter stability, but can result in a loss in stall margin of the blade, as well as excessive vibration due to forced response.

INTRODUCTION

The aim of this paper is to analyse aero-mechanical and aerodynamic instabilities for a high mass flow and low-pressure ratio fan blade as the majority of engine manufacturers are moving toward such designs. These fan blades tend to be longer, more flexible and have higher loadings than conventional ones, and hence can be more prone to aero-mechanical/aerodynamic instabilities. Furthermore, as the fan (and hence the intake) diameter increases to improve specific fuel consumption, shorter intakes are required to reduce the weight and drag of the engine (Hughes, 2011; Peters et al., 2015). The fan and the intake will become closely coupled due to shorter intakes and hence it is necessary to analyse the effects of inlet distortions, such as crosswind, on the fan stability.

It is well known that stall flutter can occur at part speed operating conditions near the stall boundary. Extensive research has shown that two independent mechanisms can cause this type of flutter (Vahdati et al., 2015; Vahdati and Cumpsty, 2015): flow driven and acoustic driven flutter. For the flow driven flutter, shock induced boundary layer separation (and hence the loading of the blade) is the key aerodynamic driver. Flow separation and consequent radial migration of the flow toward the tip triggers the loss of aerodynamic damping of the blade. On the other hand, acoustic driven flutter is caused by upstream propagating pressure waves (created by blade vibration) and their consequent reflection from the intake duct. As mentioned earlier, the new engine designs are moving towards more loaded fan blades with shorter intake ducts, and hence a detailed analysis of the flutter stability of a low-speed/highly loaded fan with shorter intake is imperative.
The occurrence of rotating stall is one of the main constraints on the design and operation of fan blades. The stall studied in this paper is similar to the multi cell, part span rotating stall, which can occur on front stages of core compressors (Mailach et al., 2001; März et al., 2002; Young et al., 2013; Pardowitz et al., 2014; Dodds and Vahdati, 2015). Although this type of rotating stall does not result in surge or massive loss of power, it can cause excessive vibration and noise. Therefore, particular attention should be paid to the understanding of rotating stall and to the prediction of the stall pattern, as the number and speed of stall cells determine the unsteady frequency, and consequently the vibration and the noise levels of the fan blade.

This paper is organized as follows. Firstly, the numerical model and the test case used are introduced. Following that, the numerical steady and unsteady results are compared with the measured data in terms of the pressure ratio, mass flow and radial profiles. Unsteady computations are performed near the stall boundary to investigate the flow and to provide insight into the rotating stall features of the fan. The unsteady results are compared with the experimental data in terms of the number and the speed of rotating cells. Full annulus flutter analyses are performed over a speed range and a detailed analysis is conducted in terms of the flow features near stall boundary. The analysis includes the effect of the intake reflections on the flutter stability. Finally, the effect of crosswind on the stability of the fan is explored.

METHODOLOGY

Numerical Model

The computations are based on a three-dimensional, time accurate, viscous, finite-volume compressible flow solver (Sayma et al., 2000). The unsteady flow cases are computed as Reynolds-averaged Navier–Stokes, with the basic assumption that the frequencies of interest are sufficiently far away from the frequencies of turbulent flow structures. The flow variables are represented on the nodes of a generic unstructured grid and numerical fluxes are computed along the edges of the grid. The numerical fluxes are evaluated using Roe’s flux vector difference splitting to provide matrix artificial dissipation in a Jameson–Schmidt–Turkel (JST) scheme.

The overall solution method is implicit, with second-order accuracy in space and time. For steady-state flow computations, the solution is advanced in pseudo-time using local time stepping, while dual time stepping is used for unsteady computations to preserve stability at high Courant numbers. For steady-state flow calculations, solution acceleration techniques, such as residual smoothing and local time stepping are employed.

The current computations use the one-equation Spalart-Allmaras turbulence model (Spalart and Allmaras, 1992). It is well known that standard Spalart-Allmaras model overpredicts the separation zone which leads to blockage of passages, limits the static pressure rise, a considerable total pressure loss, and premature stall (Liu et al., 2011; Li et al., 2014). The situation becomes worse for blades which have flat characteristics such as the low-speed fan. In order to suppress unnecessarily larger separation zone, the production term is modified based on the pressure gradient and velocity helicity (Lee et al., 2017). The parameters in Spalart-Allmara are held constant in all the present work. The resulting CFD code has been used over the past 20 years for flows at off design conditions with a good degree of success (Choi et al., 2011; Choi et al., 2013; Dodds and Vahdati, 2015; Vahdati and Cumpsty, 2015; Lee et al., 2016).

Test Case

A rig wide-chord low speed fan blade was used as the benchmark geometry for this study. There are 18 blades and the hub-tip ratio at inlet to the rotor is about 0.3. The test rig was designed for the study of instabilities (aero-mechanical and aerodynamic) and CFD validation. There is an extensive good quality measured data available for this rig.

The domain used for the computations is shown in Fig. 1. It includes the fan, with outlet guide vanes (OGV) for the bypass flow and engine-section stators (ESS) for the core and a symmetric intake (with L/D=0.35) upstream of the fan. The flow through the fan is controlled by placing two choked variable-area nozzles downstream of the fan. The nozzle downstream of the ESS controls the bypass/core flow ratio and is fixed at each speed. The nozzle downstream of the OGVs allows the computation to be conducted at any point on the constant speed characteristic by simply modifying the nozzle area. As the flow is choked in the nozzle, the solution will be independent of the conditions specified at the nozzle exit. The operating conditions considered correspond to sea level static and zero inlet swirl.

Figure 1 Domain used for computations

The grids used for the blading are semi-structured (Shardella et al., 2000), with hexahedral elements around the aerofoil, in the boundary layer region, and prismatic elements in the passage. The radial grid distribution which consists of 49 radial mesh layers is refined toward hub and casing to allow representation of the end-wall boundary layers. The fan is modelled with a typical rig tip clearance, using 6 grid cells in the tip gap. The total number of grid points used in this study contains about 1.3×10^7 nodes, with typically 0.6×10^6 nodes per passage.
RESULTS AND DISCUSSION

Flow solution

In this section, a sample of steady and unsteady aerodynamic results are presented and are compared against the measured data. For a detailed analysis of flow for this blade, the reader is referred to (Lee et al., 2017). Fig. 2 shows the comparison of the measured and computed characteristic map at the reference speed, 80%. AU3D computations were performed by using a steady single passage model, as well as an unsteady whole assembly one and the results obtained by both models are shown in Fig. 2.

Figure 2 Pressure ratio versus mass flow at 80% speed

It is seen from this plot that the computed steady result matches the measured characteristic for almost all the points on the negative slope part of the characteristic (at normalised mass-flows between 0.95 and 1.2). However, the steady computations fail to predict the positive slope part of the characteristic (at normalised mass-flows between 0.83 and 0.95) shown in the measured data. In previous studies (Mailach et al., 2001; März et al., 2002; Young et al., 2013; Pardowitz et al., 2014), it was shown that localized asymmetric unsteadiness around a fan blade, such as rotating stall, start to form and propagate on the negative part of the characteristic. The steady results have been obtained by using a single passage model with mixing planes, which does not allow asymmetric features in the solution, and hence under predicts the stall boundary, as it fails to capture the correct physics. This assumption was tested by performing an unsteady whole assembly computation, the results of which is shown (by the green line) in Fig. 2. The results in this plot show that at higher mass flows, steady and unsteady solutions are very similar, and therefore a single passage steady computation is sufficient for predicting the aerodynamic performance of the fan. The steady and unsteady results start to deviate as the mass flow is decreased, and the unsteady simulations can predict the whole characteristic, which shows a close agreement with the measurement.

Fig. 3 compares the steady radial profiles of measured and computed pressure ratio downstream of the rotor for the reference speed, 80%, at two mass flows, 0.95 and 1.07. It is seen from this plot that the steady predictions show good agreement with the measured data for both points, confirming that the code is capable of predicting correct mean flow.

The results of unsteady whole assembly computations conducted at 80% speed are presented next.
is formed (Fig. 4(b)). The amplitude of axial velocity in Fig. 5 clearly shows the growth of the stall cells compared to that for the mass flow of 0.9. Further reduction of the mass flow to 0.7, results in the formation of a stable pattern of three uniform stall cells, which occupy the top 40% of the blade and propagate in the circumferential direction along the annulus (Fig. 4(c)). It is also evident from Fig. 4(c) that the existence of the stall cells would result in a highly three-dimensional flow on the blades. The pattern of three uniform stall cells can also be observed in Fig. 5.

In the experiment, the unsteady pressure in the stationary frame of reference was measured by upstream Kulites probes on the casing and the measurements in a rotating frame of reference were taken by using strain gauges mounted on the rotor. Numerical sensors were placed on the casing to measure the unsteady pressure in the stationary and on the blade to measure them in a rotating frame.

![Figure 6 Comparison of frequency spectra between (a) CFD and (b) experiment at 80% speed at mass flow of 0.85](image)

The computations at the flow coefficient of 0.85 (shown in Fig. 2) is used to validate the numerical results with the experimental data. The time histories of the unsteady static pressure for both CFD and the experimental data at the same circumferential position are analysed using a Fourier transform and are compared in Fig. 6. First of all, it is important to recognize that the measured data also show unsteadiness at this flow coefficient, and hence the comments made earlier regarding inability of single passage steady modelling for computing the flows near stall is accurate. It is seen from Fig. 6 that, the computed result shows relatively good agreement with the measured data, exhibiting a similar trend for harmonics in the low frequency region. As can be seen, the dominant frequencies are far below the blade passing frequency (BPF). A similar pattern of frequency spectra was also found in previous studies for low-speed compressors (Mailach et al., 2001; März et al., 2002; Young et al., 2013; Pardowitz et al., 2014). Mailach et al. observed a frequency band with a large amplitude below the BPF in a low-speed compressor and reported this phenomenon as rotating instability. Young et al. also observed very similar frequency distribution which showed a dominant frequency band far below the BPF in a low-speed compressor. They described this flow feature as pre-stall disturbance. It is known that this type of rotating stall could become a source of the tip clearance noise and blade vibration (Mailach et al., 2001; März et al., 2002; Young et al., 2013; Pardowitz et al., 2014).

Dodds and Vahdati (Dodds and Vahdati, 2015) examined the rotating stall pattern in a high-speed compressor by using a relationship between the stationary and rotating frames to determine the number and speed of stall cells.

\[
    \text{EO}_{\text{stat}} = \frac{f_{\text{stall}}}{N} = m\left(\frac{V_{\text{stall}}}{U}\right) \quad (1)
\]

\[
    \text{EO}_{\text{rot}} = \frac{f_{\text{rot}}}{N} = m\left(1 - \frac{V_{\text{stall}}}{U}\right) \quad (2)
\]

Here, \(\text{EO}_{\text{stat}}\) and \(\text{EO}_{\text{rot}}\) are engine orders, frequency normalized by shaft speed, measured in the stationary and rotating frames respectively. \(f_{\text{stall}}\) and \(f_{\text{rot}}\) are the frequencies measured in each frame respectively, \(N\) is the rotational speed of the shaft, \(U\) is the rotor speed, \(m\) is the number of stall cells and \(V_{\text{stall}}\) is the absolute velocity of stall cells. More details can be found in (Dodds and Vahdati, 2015).

<table>
<thead>
<tr>
<th>Table 1 Comparison of rotating stall characteristic</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
</tr>
<tr>
<td>-----</td>
</tr>
<tr>
<td>EO&lt;sub&gt;stat&lt;/sub&gt;</td>
</tr>
<tr>
<td>3.33</td>
</tr>
<tr>
<td>4.24</td>
</tr>
<tr>
<td>4.75</td>
</tr>
</tbody>
</table>

The dominant frequencies in both stationary and rotating frames can be obtained from the FFT of the unsteady data and the stall pattern can be calculated by using Eq. 1 and 2. Table 1 shows the comparison of the stall characteristic between the computed and measured data. In the experiment, 6 stall cells appeared at both Kulites and strain gauge and rotate with propagation speed of 82.1% of rotor speed. As can be seen in the table, there is a good agreement between the computation and experimental data regarding the number and speed of stall cells, indicating that the numerical approach used in this work can predict realistic flow features during rotating stall.

Flutter Computations

The starting point for the flutter analysis is the steady state solution at the flow condition studied. The single passage steady state mesh and solution are expanded to whole assembly. The unsteady solution commences by giving the blades an initial velocity in the chosen modes. Aerodynamic damping of the blade is obtained by tracking the time history of the blade motion (Vahdati et al., 2001). The mechanical (or
structural) damping of the blade is not known and therefore is not considered in the work reported here and hence the result are pessimistic in terms of flutter boundary.

Figure 7 Characteristic map with stability boundaries for the model fan used in this study

Fig. 7 shows the comparison of the calculated and measured flutter boundaries. There are two measured flutter boundaries in this figure which have been obtained by using two different intakes with the same fan set: the long intake is shown red and the short intake blue. The length of the long intake is about 3.5 times that of the short intake. It is seen from Fig. 7 that there is a significant difference between the measured flutter boundaries of the two intakes. The short intake shows a sharp drop in stability (flutter bite) at around 80% speed, whereas the longer intake does not have a flutter bite. The AU3D computations shown in Fig. 7 is for the short intake with hard walls and show a good agreement with the measured data. The AU3D computations were performed by including all nodal diameters, but all the modes apart from 2ND were stable. Therefore, the AU3D flutter boundary of Fig. 7 is for a 2ND mode which is in agreement with measured data. The calculations show instability due to flutter at somewhat higher mass flow rates than those measured. The lower measured mass flow for instability is attributed to the presence of some mechanical damping, mistuning and acoustic liner in the test rig, which are wholly omitted in the present computation.

Figure 8 1F/2ND aero-damping at HWK line

Fig. 8 shows the 1F/2ND aero-damping for an artificially created high operating line, labelled as HWK (dashed line in Fig. 7) plotted against the rotation speed of the rotor for both intakes. It can be clearly seen from this plot that for the short intake, flutter occurs between 79% and 82% with the bottom of the flutter bite (least stable speed) at 80%. For the long intake, there is no flutter on this operating line and the values of aero-damping are higher for the long intake at all speeds.

Crosswind Computations

The effects of crosswind are critical for an airplane on the ground in particular before take-off, as in such conditions intake separation and the consequent non-homogeneous flow field at the fan face become the primary concerns. Moreover, because of shorter intakes, the fan and the intake will become closely coupled and hence the effects of inlet distortions, such as crosswind, will become more important on the fan stability. However, before proceeding with stability computations, it is very important to establish the limitations of AU3D (which is a URANS code) in dealing with flow separation and attachment in the intake duct. To achieve this purpose, AU3D results are compared against measured data. The test case used...
for the validation of inlet separation is shown in Fig. 10. The domain used in the validation study contained only the intake. The flow conditions correspond to cruise flight at a 16,600 ft altitude and the flight Mach number is 0.25. The test case used here is not a crosswind simulation, but it can verify the capability/limitation of the CFD code in terms of the separation angle and the distortion level.

The inlet distortion level can be measured using distortion coefficient DC60 defined as:

\[
DC60 = \frac{(P_{0,60} - P_0)}{(P_0 - P)}, \quad (3)
\]

where \(P_0\) and \(P\) are the area averaged total and static pressures at the fan face and \(P_{0,60}\) is the lowest area averaged total pressure over a 60° circumferential section at the fan face. A negative value of DC60 implies that there is a deficit of total pressure due to a lip separation compared to the average at the fan face. Fig. 11 shows the comparison of the values of DC60 between the measurement and CFD against incidence angles. It is seen from the figure that AU3D predicts the onset of lip separation at an incidence very close to the measured data, demonstrating that the code is capable of providing correct predictions of the separation angle and the distortion level. It should be noted that, crosswind separations are typically driven by excessive diffusion levels and are notoriously difficult to predict using URANS methods, whereas high angles of attack separations are shock driven and URANS models are usually more accurate. Therefore, due to the different flow physics involved, getting good agreement between CFD and measurements at an angle of attack condition may not be sufficient to validate the prediction capability at crosswind and other test cases are required to validate the CFD. The shortcoming of the URANS model used here is acknowledged and to account for this, all results are presented as a function of distortion levels (DC60) and not the absolute values of crosswind. The other approach would have been to prescribe the inlet distortion but such a methodology would not have modelled the effects of the fan blade on the distortion levels and hence this type of model is deemed more accurate.

In the next part, unsteady computations under crosswind are considered to explore the effect of crosswind on the aerodynamic and aeroelastic stability of the fan blade. As mentioned earlier, all the computations in this work have been performed with a symmetric intake. The L/D in this computation is 0.35. Fig. 12 shows the computational domain for this study with the direction of crosswind and streamline near the intake lip. As shown in Fig. 12, the flow can separate on the lip under severe crosswind conditions and the level of distortion at the fan face is determined by the size of the separation. In this work, the effect of the ground vortex, as well as the headwind is not considered.

In order to investigate the effect of crosswind on the stability of the blade, unsteady whole annulus computations were performed for a range of crosswind speeds at 80% fan speed. Fig. 13 shows the computed characteristic for different

---

1 Sowden, J., RR Internal Rept. IPCR38228.
magnitudes of DC60 at mass flow of 1.01 with the whole characteristic for DC60 of 0. The fan pressure ratio shown here is calculated from fan leading edge to fan trailing edge. It is clearly shown from Fig. 13 that for DC60 of 0.292 the flow differs from the other solutions; showing a hysteresis (of mass flow/pressure ratio) as described in the previous section. Fig. 14 shows Mach contours for DC60 of 0 and 0.292 in the side view. The deviation of the mass flow at DC60 of 0.292 is due to the fact that the flow separates on the lip and hence the separation bubble narrows the inflow area at the fan face, resulting in the total pressure loss as shown in Fig. 13.

Fig. 15 shows the instantaneous variation of axial velocity at mid chord in the rotating frame of reference. As shown in the figure, the flow is symmetric at DC60 of 0 whereas a clear asymmetric pattern with stall cells is observed along the circumference for DC60 of 0.292, and the flow patterns look similar to that of clean flow at much lower flow coefficient (see Fig. 4(b)).

Fig. 16 shows the loss of stall margin against DC60. Here, the loss in stall margin is defined by

\[ \psi = \frac{M_D - M_C}{M_C} \]  (4)

where \( M_C \) is the stall mass flow in the absence of distortion (zero-crosswind) and \( M_D \) is the stall mass flow at a given crosswind. It clearly shows that at this operating line for the fan, there is a significant loss in stall margin above DC60 of 0.026. As shown in the figure, the distortion level of 0.292 is attributed to the intake separation due to a high incidence as shown in Fig. 14(b). It is clear from Fig. 16 that, although the intake is not separated for DC60 of 0.026 and 0.088, the blade can still stall prematurely due to non-homogeneous flow from the inlet. Fig. 17 shows swirl angle upstream of fan for DC60 of 0.012 and 0.088. It is clear from this plot that the tip of the fan blade can experience very high incidence angles for DC60 of 0.088, which is the reason for premature stall.
The effects of migration and asymmetry that is produced by mid-stagger is determined by the balance between the radial effects for flutter and mis-staggering is the fact that, for mistuning the mean flow remains unchanged and the same for all the blades, whereas for mis-staggering the mean flow will differ for different blades as the incidence onto each blade would be different. The results obtained in (Vahdati and Cumpsty, 2015) clearly showed that, for stall flutter, the onset of flutter is related to 3D separation and radial migration of flow on suction side of the blade. The result in (Salles and Vahdati, 2016) showed that, the introduction of mistuning distributes the vibration energy over many circumferential modes, and as most of the other circumferential modes are stable, they dissipate this energy resulting in an increase of aero-damping. The introduction of mis-stagger not only breaks the symmetry of the flow but also increases the radial migration on some blades. Therefore, as can be seen from Fig. 20 mis-stagger can have beneficial as well as detrimental effects on aero-damping of the blade; the overall effect of mid-stagger is determined by the balance between the radial migration and asymmetry that is produced by mid-stagger. The effects of crosswind is similar on flutter as different blades would experience different incidences onto them; low amplitudes of crosswind (low DC60) can move some of the blades towards stall boundary and hence reduce their aero-damping, without creating sufficient asymmetry in the flow to break the circumferential flutter mode. However, as the magnitude of the crosswind increases (high DC60), the asymmetry in the flow increases which allows transfer of energy between circumferential modes, and hence enhanced flutter stability.

CONCLUSIONS

The unsteady full annulus simulations were performed to map out the unsteady region of the characteristic and to validate the numerical model with respect to the experiment in terms of the stall characteristic. The results showed that the whole characteristic (including the positive slope region) can be obtained by the unsteady computations and the numerical characteristic showed a good agreement with the measured data. The time history of the (measured and computed) static pressure in the intake duct revealed that at lower flow coefficients, the flow becomes unsteady and asymmetric around the blade; which can result in vibration and noise issues for the engine.

Full annulus flutter simulations have been conducted over a range of speeds to understand stability behaviour of a low-speed fan. The numerical results are compared to the experimental data (when available) and are well-matched in terms of the pressure ratio, mass flow and the flutter stability boundary. Measured flutter boundaries were obtained for two intakes using the same fan set. The test result showed that the short intake has a flutter bite at 80% speed whereas there were no flutter bites for the long intake.

An airplane on the ground under crosswind is a critical case for the design of aero-engine. The unsteady full annulus simulations under crosswind were carried out at various crosswind speeds to analyse the interaction between the flow over an intake and the downstream fan. The results showed that for sufficiently high amplitudes of crosswind, the intake lip will separate, and will result in a loss of stall margin, as well as an increase in the vibration levels of the blade due to forced response. However, as the magnitude of crosswind speed increase and the level asymmetry grows the flutter margin of the blade increases.

NOMENCLATURE

- $L/D$ = ratio of the intake length to diameter
- $ND$ = nodal diameters
- $BPF$ = blade passing frequency
- $E_{0,rot}$ = engine order measured in rotating frame
- $E_{0,stat}$ = engine order measured in stationary frame
- $f_{rot}$ = frequency measured in rotating frame
- $f_{stat}$ = frequency measured in rotating frame
- $m$ = circumferential wavenumber
- $N$ = rotational speed of shaft (Hz)
- $V_{sat}$ = speed of propagation of stall cells
- $p'$ = static pressure (unsteady component)
- $P_{in}$ = inlet total pressure
- $DC60$ = distortion coefficient

![Figure 20 1F/2ND aero-damping against mis-stagger amplitude](image)
P_0 = area averaged total pressure
P = area averaged static pressure
P_{0,60} = area averaged total pressure for the 60° segment
\psi_y = loss of stall margin
M_C = stall mass flow in the absence of distortion
M_D = stall mass flow at a given crosswind

ACKNOWLEDGMENTS
The authors thank Rolls-Royce plc and CleanSky for both sponsoring this work and allowing its publication.

REFERENCES