Influence of the Inlet Distortion on Fan Stall Margin at Different Rotational Speed

Wenqiang Zhang  
Department of Mechanical Engineering  
Imperial College London  
w.zhang15@imperial.ac.uk  
London, SW7 2AZ, UK

Mehdi Vahdati  
Department of Mechanical Engineering  
Imperial College London  
m.vahdati@imperial.ac.uk  
London, SW7 2AZ, UK

Abstract

The performance and aerodynamic stability of the fan operating within a circumferentially non-uniform inlet flow is a key concern for the design and operability of turbofan engine manufactures. The aim of this paper is to develop a reliable and accurate numerical strategy that can be used to study the effects of inlet distortions on the aerodynamic stability of fan blades. As an initial step towards achieving this goal, three-dimensional Unsteady Reynolds Average Navier-Stokes (URANS) simulation were carried out to predict the influence of total pressure distortion on the loss of stall margin of a fan blade. The numerical modelling of this research is carried out by using the Computational Fluid Dynamics (CFD) & aeroelasticity Code AU3D, which is written at Imperial College and developed over many years with support from Rolls-Royce. NASA rotor 67 was used for this study, for which a significant amount of measured steady and unsteady data is available. In the first instant the steady and unsteady results were compared against the measured data to validate the CFD code used.

In the second part of this work, unsteady simulations were carried to study the effects of fan corrected speed on the stall margin of the blade. The result showed that for the same level and pattern of inlet distortion, the stall margin of the blade decreases as the corrected speed decrease.

INTRODUCTION

Inlet distortion refers to the non-uniform flow upstream of an aero engine. The performance of the fans and compressors can be detrimentally affected by inlet distortion in terms of efficiency and pressure rise. Inlet distortions can also reduce the aerodynamic and aeroelastic stability limits of an engine, which is the main driving force of this research.

The trends of engine design have concentrated on cutting down specific fuel consumption (SFC) by improving propulsive efficiency and thermal efficiency, which leads to larger bypass ratio (BPR) and lower fan pressure ratio (FPR). [1] However, the benefit of lower fuel consumption from increased fan diameter can be offset by the increased weight of the low pressure (LP) system and the nacelle drag. The target to increase the fan diameter without incurring weight and drag penalty, results in the design of intakes with reduced axial length [2], and consequently, the capability of the inlet duct to attenuate the distortions via internal diffusion is reduced, which can make the fan more prone to instabilities.

A large amount of experiments have been performed to explore the relationship between the inlet distortions and the compressor performance [3]. However, to the best knowledge of the authors, there are no recent public domain experimental data which considers the effects of distortion on the stall margin of the blades. Such experimental testing can be quite costly and time consuming, and alternative methods are required during the early design stages of an engine to determine the loss in the stability of a fan blade due to inlet distortions. CFD provides an alternative route which can be more economical, but to use such an approach, a validated numerical strategy is required. As a result of lack of (public domain) measured data for distorted flows near stall, the CFD results are compared against the unsteady results near peak efficiency. However, it has been shown in the past [4-6] that the CFD code used in this work is quite accurate for predicting the flows near stall. Although, such a strategy is not ideal, it is the only possible approach that can be taken with the amount of experimental data available.

The paper is organized in the following manner;
1. Validate the simulation strategy by comparing the CFD results with the experimental data for steady and unsteady flows.
2. Find a reliable strategy to perform the simulation for stall margin prediction with inlet distortion
3. Study the effect of rotational speed (blade loading) on the stall margin of the compressor with inlet distortion.

The Test Case

NASA rotor 67 [7] is chosen as the test case in this paper. It is an undampered low-aspect-ratio design rotor. The rotor design pressure ratio is 1.63 at a corrected mass flow of 33.25kg/sec. The design rotational speed is 16043 rpm, which yields a tip speed of 429m/sec and inlet tip relative Mach number of 1.38. The mean hub/tip ratio of this blade is 0.43. The rotor row has 22 blades. The stator row S67A used in the distortion experiment [8] has 34 blades. Initially, tests were carried out at different rotational speed in a clean flow. Total pressure, total temperature and flow angles of the flow field in the compressor were measured with probes at fixed circumferential and axial position, shown as Fig. 1.

The CFD Solver

The CFD solver used in this study is AU3D which has been developed at Imperial College VUTC over the past 25 years with support from Rolls-Royce. It is an implicit, time-accurate 3D compressible Reynolds-averaged Navier-Stokes (RANS) solver. This flow solver is based on a cell-vertex finite volume methodology and mixed-element unstructured grids. The mesh is characterized by using an edge-based data structure and the grid is presented to the solver as a set of node pairs connected by edges. The central differencing scheme is applied with a mixture of second and fourth-order matrix artificial dissipation for stabilization. In addition, a pressure switch, which guarantees that the scheme is total variation diminishing (TVD) and reverts to a first-order Roe scheme in the vicinity of discontinuities, is used for numerical robustness. The resulting semi-discrete system of equations is advanced in time using a point-implicit scheme with Jacobi iteration and dual time stepping. Solution acceleration techniques, such as residual smoothing and local time stepping, are employed for steady-state flow calculations.

The current computations use the one-equation Spalart-Allmaras turbulence model. The parameters in Spalart-Allmaras have been adjusted on previous fans to get good agreement near the stability limit; the parameters 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 [9-11]. For unsteady computations, an outer Newton iteration procedure is used where the time steps are dictated by the physical restraints and remain fixed through the solution domain. Within a Newton iteration, the solution is advanced to convergence using previously mentioned acceleration techniques. Further details can be found in Sayma et al [12].

Computational Domain and Boundary Conditions

The computational model was constructed by using the actual values of intake length and axial gap (between the rotor and stator) from the test rig. The steady simulation domain for the single rotor case includes a whole-annulus intake duct, a single passage rotor, an exit duct and a nozzle. The steady simulation domain for the whole stage case includes a whole-annulus intake duct, a single passage rotor, a single passage stator, an exit duct and a nozzle. The plenum is not included in the computational domain. The length of the exit duct is 1.5 times that of the rotor diameter. The rotor is modeled with a tip clearance of 1.0 mm. The nozzle downstream of the fan is choked, which impedes the disturbance from propagating upstream, i.e. the flow becomes independent of exit condition. The operating point of the fan is only determined by the nozzle area; by closing or opening the nozzle, the corrected mass flow of the fan is reduced or increased [13]. It is found that this form of imposing outlet conditions is ideal for computations with inlet distortions as it would allow radial and circumferential variations of exit flow quantities. At the inlet, total pressure, total temperature and flow angles are specified. In the unsteady computations, a distortion plane with a low total pressure sector of 120 degree is placed upstream of the inlet duct (blue region in Fig. 2). Total temperature is uniform and the circumferential flow angle is zero at the distortion plane.

For the steady computations, a single passage model is used and the boundaries between stationary and rotating blades are treated as mixing planes. For the computations with the inlet distortion, a whole annulus domain is used and the boundaries between stationary and rotating blades are treated as sliding planes. As shown in Fig. 2, the domain for unsteady computations is composed of a whole annulus inlet duct, a whole annulus rotor row, a whole annulus stator row (for the stage computation), exit duct and a whole annulus nozzle. The steady solutions are used as an initial solution for the unsteady computations to improve the convergence.

Fig. 1 Flow path schematic for NASA 67 fan stage
Fig. 2 Isometric view of the computational domain of NASA rotor 67 with inlet distortion

Steady computations

In this section, the steady CFD results (for a clean intake) are compared against measured data. Before preceding with detailed comparisons, a grid size study was performed to obtain an optimum mesh (in terms of computational cost and accuracy) for the rotor blade.

Fig. 3 Characteristic of the single rotor with different grids

Three grids with different densities were used to simulate the flow at 100% corrected speed. Fig 3 shows the comparison of the CFD results against the experimental data (EXP); although the agreement of the CFD and the experimental data for a mesh with 1.5 million points is better, the grid with 1 million points was chosen to seek a balance between the simulation performance and the computational cost.

Fig. 4 Rotor characteristics in clean flow

Fig. 4 shows the comparison of computed and measured total pressure and isentropic efficiency characteristic maps for the rotor alone case. As can be seen, there is a satisfactory agreement between CFD and the experimental data for the steady characteristics. The mass flow and total pressure ratio of the near stall points are computed accurately, indicating that AU3D (CFD code used in this study) is capable of predicting an accurate stall boundary for undistorted flows. The choke mass flow in simulation is slightly lower than that in the test data, which is possibly due to the fact that the blade used in the simulation is slightly more closed than the one used in the experiment due to the untwist cause by centrifugal force or aerodynamic loading. Also, the efficiency at part speed is higher in the experimental data than the CFD results, which is in common with other published CFD results for this test case [8].

Fig. 5 Circumferentially averaged, radial distributions downstream of the rotor. PE and NS, 100% speed

Fig. 5 shows the comparison of the CFD and the measurements results of the radial profiles for the total pressure ratio, total temperature ratio and swirl angle downstream of the rotor at near peak efficiency (PE) point and near stall (NS)
point at 100% speed. Generally, the radial profiles match well with the experimental results at both operating points. The computed flow angle is around 3 degrees higher than that in the experiment, which is in common with other CFD results for this test case [14-16].

The results shown in Fig. 4 were for a rotor alone. Fig. 6 shows the computed and measured total pressure and isentropic efficiency characteristic maps for the stage study. The overall simulated pressure ratio and efficiency are quite close to the experimental data and the predicted stall points at 80% and 90% corrected speed match the measured values. The slight under-prediction of stall boundary at 70% is probably due to the fact that the same blade geometry is used at all speeds, which would result in a more open blade at 70% speed.

Validation of the Unsteady Case with Inlet Distortion

Experimental tests with inlet distortion were carried out by imposing a 120° low total pressure sector as shown in Fig. 2. The total pressure in the distorted section was 10.5% lower than the total pressure in the clean sector. The experiment was carried out at the corrected mass flow of 32kg/s and a pressure ratio of 1.46 at 90% corrected speed [8]. Measurements were performed upstream of the rotor, downstream of the rotor and downstream of the stator (denoted by 1, 2 and 3 in Fig. 1).

In CFD simulations, the accuracy of unsteady calculations is dependent on the size of the times-step used in the analysis and thus a time step convergence study is required before proceeding with detailed analysis. The convergence is monitored by tracking lift on the rotor, and the simulation is deemed as converged when the rotor lift becomes periodic.

Figure 7 Time-averaged absolute circumferential total pressure at middle span upstream of the rotor

Figure 8 Time-averaged circumferential measurement results downstream of the rotor at mid-span

Fig 7 and Fig 8 show the comparison of circumferential variations of total pressure upstream and downstream of the rotor for different values of time step. It is seen from this plot that there is only a minor difference between 400 and 2200 time-steps/cycle. Similar results were obtained for other flow variables and at other stations (not shown here), and thus it was deemed that 400 time steps/cycle is sufficient for the unsteady analysis for this flow condition. Based on previous experience, it is known that for a given frequency, AU3D requires about 200 time-step per cycle for accurate modeling of that frequency. The above observation suggests that, for the PE operating point, correct modelling of the 1st two harmonics of distortion would be sufficient to produce a temporal converged solution.

A DFT (Discrete Fourier Transform) of the total pressure at the mid-span of the rotor is carried out to generate the harmonic spectrum due to the inlet distortion (both at leading edge and trailing edge of the blade). The circumferential wave
number and amplitude of the harmonics are shown in Fig. 9 and Fig. 10. The normalized amplitude is calculated as

\[ A_{n,s} = \frac{\mathcal{P}_{n,s}}{\mathcal{P}_{0,s}} \]  

(1)

where \( \mathcal{P}_{0,s} \) is the mass-averaged total pressure at measurement station \( s \) with inlet distortion, \( \mathcal{P}_{n,s} \) is the amplitude of the \( n \)th harmonic of the distortion. The harmonics with the normalized amplitude less than 0.3 are not presented in the plots. It is seen from Fig. 9 that the amplitude of the distortion upstream of the fan decreases with increasing circumferential wave number. Moreover, Fig. 10 shows that, as the distortion passes through the rotor, there is a significant decrease in the intensity for all the harmonics, and the harmonics higher than 6 are almost eliminated. Moreover, the loss in amplitude of the distortion as it passes through the fan passage increases as the circumferential wave number increases. Therefore, the amplitudes of the higher harmonics of the distortion are significantly reduced and thus their contribution to the flow field can be ignored. Fig. 11 shows the comparison of the harmonic contents of the distortion for PE and NS operating points downstream of the rotor. It is evident from this plot that, for the NS point the rotor is not capable of attenuating the higher harmonics (as efficiently as PE operating point) and therefore for this operating point finer temporal resolution is required for accurate modelling of the flow. The effects of temporal resolution on the computed stall boundary will be discussed in a later section.

Figure 9 Amplitude of the harmonics of the total pressure at the mid-span upstream of the rotor, PE

Figure 10 Amplitude of the harmonics of the total pressure at the mid-span downstream of the rotor, PE

Figure 11 Amplitude of the harmonics of the total pressure at the mid-span downstream of the rotor, PE and NS

Figure 12 Time-averaged absolute circumferential swirl angle at mid-span at rotor upstream, PE and NS

In the next phase of this study the flow field at peak efficiency point (PE, corrected mass flow =31.81kg/s) was compared against measured data. Although measurements were not performed at the near stall point (NS, corrected mass flow = 29.3kg/s), CFD results at this condition were still included in the comparison plots to evaluate the change in the flow feature as the mass flow is reduced. Fig. 12 to 17 compares the numerical time-averaged parameters for peak efficiency and near stall point against the experimental data for the peak efficiency, upstream and downstream of the rotor. As shown in the figures, the overall performance of the simulation matches well with the test data. Fig. 12 and 13 show the circumferential profile of the flow swirl angle and radial profile of the total pressure upstream of the rotor (station 1). The profile of the total pressure distortion essentially remains the same as it passes through the inlet duct. However, despite the swirl angle being zero at the inlet, circumferential flow is induced by the static pressure distortion. This phenomenon verifies that swirl distortion can be generated by pressure distortion even when there is no vortex in the incoming flow.
phase of the swirl angle are well reproduced with the largest difference being about 1 degree. It is also seen from Fig. 12 that, the swirl angle at the fan inlet differs between peak efficiency and near stall point. The main difference is at 240º, which correspond to circumferential position that the rotor moves from the distorted sector to the clean sector. The difference in flow angle between the two flow coefficients is due to the fact that at the near stall point, the axial flow is reduced while the circumferential velocity does not change. The result in Fig.12 also shows that more circumferential redistribution occurs at PE than NS upstream of the fan. The two different radial profiles in Fig. 13 correspond to low total pressure and the high total pressure sector. The simulation plots match well with the experimental data except for the hub region of the low total pressure sector. As can be seen there is a high pressure area near the hub in the experimental data which is over-predicted by the CFD simulations. The high pressure region is formed as a result of the redistribution flow from high pressure sector to low pressure sector. It is seen from Fig. 13 that, at the lower flow coefficient (NS), the high-pressure region expands radially, which is due to an increase in the redistribution flow near the hub.

Figure 13 Time-averaged radial profile of total pressure upstream of the rotor at circumferential angle $\theta=322.5^\circ$ and $\theta=175.5^\circ$

Downstream of the rotor (station 2), the time-averaged total pressure, total temperature and the circumferential swirl angle around the annulus are compared with measured data, as shown in Fig. 14. The CFD results in the clean flow simulation match the experimental data while a slight deviation of total pressure and swirl angle can be observed in the distorted sector (between $\theta=120^\circ$ and $\theta=240^\circ$). As can be seen from Fig 14, a total temperature distortion has been formed downstream of the rotor. On comparing, the PE and NS operating points, it is seen that in the circumferential position between $255^\circ$-$275^\circ$ the pressure rise is lower for the NS operating point than PE point, which indicates the initiation of stall in this region which is held stable by the rest of the circumferential part. The above observation can also be seen in the swirl angle plot of Fig. 14 which shows less turning for the NS point in the region between $255^\circ$-$275^\circ$.
Figure 15 Time-averaged radial profile of total pressure downstream of the rotor at circumferential angle $\theta = 283^\circ$

Figure 16 Time-averaged radial profile of total pressure downstream of the rotor at circumferential angle $\theta = 73^\circ$

Fig. 17 shows the comparison of flow variables downstream of the stator vane. It is seen from this plot that, there is a good correlation between CFD and measured data at this axial station. It is also evident from this plot that there is a considerable difference between the NS and PE operating points at this station. The most noticeable difference is in the flow angle plot which clearly shows that the stator vane is unable to remove the swirl from the flow at NS operating conditions. The largest swirl angle is at 150º which corresponds to the distorted region, which is shown by the black circle in Fig.18 as well.

Figure 17 Time-averaged circumferential measurement results downstream of the stator at mid-span

Figure 18 Time-averaged contour of circumferential velocity downstream of the stator

Fig. 19 Stall line of the single rotor with inlet distortion

In the next step, orbit method [17] was applied to analyze the difference in the rotor flow as it rotates around the circumference. The corrected mass flow and total pressure ratio of the 22 sectors (corresponding to each rotor) together with the steady (undistorted) characteristic is plotted in Figure 19. The 22 sectors are connected by a line to form an “orbit” (shown by square) around the operating point of the fan. It is seen from this plot that as the rotor rotates around the circumference, it experiences considerable change in flow conditions. The influence of the distortion are: (1) move the blade to a higher/lower speed line, as the part A and C in Fig.19 ;(2) move the blade along the constant speed characteristic, as the part B and D. The change in corrected mass flow is due to the existence of the distortion, which reduces the axial velocity in the distorted sector and can be explained with the classical parallel compressor theory [18]. As mentioned earlier, the total pressure distortion induces a swirl distortion up stream of the
rotor. The swirl angle changes the incidence of the flow and hence the effective rotational speed of the rotor. Consequently, the operating point is moved to a higher or lower rotational speed. The operating point is pushed along the constant speed line. On comparing the PE and NS operating points on Fig. 19, it is evident that the orbit shape is considerably different at the different flow conditions. Currently it is difficult to explain the shape of the orbit at the near stall point. As the orbit method uses streamtraces to split the time-averaged rotor domain, continues and accurate streamlines are difficult to obtain when the flow is near to stall due to the existence of separation and vortex.

### Influence of the Temporal Resolution

It was shown in a previous section that for flows away from the stall boundary, 400 time step/cycle was sufficient to obtain a time converged solution. It will be shown in this section that, the temporal resolution has a greater influence as the mass flow drops and the stall boundary is approached. The sensitivity of the stall margin prediction to time-step size is investigated next. Defining the mass flow range loss as

\[
\psi_{\text{loss}} = \frac{\text{Mass}_{\text{distortion, stall}} - \text{Mass}_{\text{clean, stall}}}{\text{Mass}_{\text{clean, choke}} - \text{Mass}_{\text{clean, stall}}} \times 100\% \quad (2)
\]

Fig. 20 shows the mass flow range loss with the inlet distortion for different values of time steps/rev at 90% corrected aero speed. It is seen that the time-step size has a great influence on the predicted stall boundary of the blade and an increase in the time-step size will result in a decrease in the stall margin of the blade. It has been shown by [19] that for spike stall, the higher order harmonics in the flow field trigger the stalling of the blade, which cannot be modelled with larger time-step values. For a compressor operating away from the stall line, the flow is relatively stable and no obvious separation happens. Besides, the rotor is capable of attenuating the high order harmonics, and therefore their accurate numerical modelling is not essential. Consequently, there is a slight difference between the overall performance for different temporal resolutions. The above proposition is supported by the fact that the discrepancy between the simulation results of 400 and 2,200 time steps/rev for the peak efficiency point is not substantial (see Fig. 7-8). Fig. 21 shows the time history of the mass flow/pressure ratio for different temporal resolutions. The initial solution is identical for all these computations. It can be seen that when 550 time steps/rev is used the rotor is stable but for 1,100 time steps/rev and higher the rotor becomes unstable and moves to a secondary stall characteristic. Thus the temporal resolution validation is essential before a stall margin prediction is performed.

![Fig. 20 The mass flow range loss with inlet distortion running with different time steps/rev](image)

Fig. 20 shows the mass flow range loss with the inlet distortion for different values of time steps/rev at 90% corrected aero speed. It is seen that the time-step size has a great influence on the predicted stall boundary of the blade and an increase in the time-step size will result in a decrease in the stall margin of the blade. It has been shown by [19] that for spike stall, the higher order harmonics in the flow field trigger the stalling of the blade, which cannot be modelled with larger time-step values. For a compressor operating away from the stall line, the flow is relatively stable and no obvious separation happens. Besides, the rotor is capable of attenuating the high order harmonics, and therefore their accurate numerical modelling is not essential. Consequently, there is a slight difference between the overall performance for different temporal resolutions. The above proposition is supported by the fact that the discrepancy between the simulation results of 400 and 2,200 time steps/rev for the peak efficiency point is not substantial (see Fig. 7-8). Fig. 21 shows the time history of the mass flow/pressure ratio for different temporal resolutions. The initial solution is identical for all these computations. It can be seen that when 550 time steps/rev is used the rotor is stable but for 1,100 time steps/rev and higher the rotor becomes unstable and moves to a secondary stall characteristic. Thus the temporal resolution validation is essential before a stall margin prediction is performed.

![Fig. 21 Corrected mass flow history at the rotor outlet with different time step size](image)

![Fig. 22 Normalized axial velocity of the rotor outlet, 2,200 (left) and 550 (right) time steps/rev](image)
**Influence of the Rotational Speed**

Fig. 23 shows the predicted stall line of the single rotor case with imposed inlet distortion DC120. The total pressure loss in the distorted area is fixed at 10.5% for all the speeds. The distorted simulations were performed until the slope of the characteristic became positive. The last stable points of three rotational speeds were connected by a parabolic curve. It is found that the stall margin loss increases as the rotational speed drops.

![Fig. 23 Stall lines of the single rotor with inlet distortion](image)

A possible explanation for this finding is that the slope of the characteristic is related to the stability of the compressor when subjected to an inlet distortion. It is considered [20] that the “steep” characteristic is more robust than the “flat” characteristic. A compressor with a steep characteristic can achieve considerable pressure rise without a significant change in the mass flow rate. Moreover, steeper characteristics can induce more cross flow upstream of the fan so the axial velocity and swirl angle variations are smaller. As can be seen from Fig 23, the shape of the characteristic gradually becomes flatter as the rotational speed decreases, which reduce the ability to repel the impact of disturbances. To achieve the static pressure of the clean case, the mass flow of the spoiled sector has to decrease in order to produce a large enough pressure rise and thus the sector is forced to operate closer to the stall line. For a relatively flat characteristic, the compressor might fail to provide sufficient static pressure rise even when the mass flow is pushed to the stall line, which may trigger stall.

The Loss in stall pressure ratio is defined as the loss of total pressure ratio for the same corrected rotational speed, and it is calculated as:

\[
\Delta PR_s = 1 - \frac{PR_{st}}{PR_{st, s}} \left( \frac{N}{N_{ref}} \right)^{\text{Constant}}
\]

Table 1 summarizes the loss in mass flow range and stall pressure ratio. It can be seen that the stall margin decrease as the rotational speed decrease.

**Table 1 Loss in mass flow range and stall pressure ratio, choked nozzle**

<table>
<thead>
<tr>
<th>Rotational speed</th>
<th>( \Psi_{loss} )</th>
<th>( \Delta PR_s )</th>
</tr>
</thead>
<tbody>
<tr>
<td>100%</td>
<td>31.9%</td>
<td>1.07%</td>
</tr>
<tr>
<td>90%</td>
<td>37.7%</td>
<td>0.88%</td>
</tr>
<tr>
<td>80%</td>
<td>38.24%</td>
<td>1.40%</td>
</tr>
</tbody>
</table>

Another point that is worth mentioning is that, in the current simulations, the unsteady computations with distortion commences by placing a distortion plane upstream of the steady clean solution at the same nozzle angle, which implies a sudden appearance of the distortion to the engine (and represents a very abrupt aircraft manoeuvre). However, it is found that, if the computations start from a converged distorted case (which is away from stall) and the nozzle is gradually closed, the compressor will stall at a lower mass flow which denotes less loss in the stall margin.

**Conclusion and Future Work**

Steady simulation and unsteady simulation with inlet distortion of DC120 were carried out for the NASA rotor 67 compressor blade. The simulation results have been compared with experimental data, and overall, a good agreement was found between the two results, which verifies the suitability of AU3D (CFD code used) for this test case. A time-step convergence study indicated that 400 time-step/rev was sufficient for the flow at peak efficiency but needed to be increased to 2200 time-step/rev for the modelling of stall boundary. That observation suggest that for the flow away from the stall accurate modelling of the 1st two harmonics would be sufficient to produce a ‘temporal converged’ solution but for prediction of stall boundary, accurate modeling of harmonics up to 11 (in this case) is needed. Stall margin computations with inlet distortion were performed to predict the influence of the distortion on the stall margin of this blade. For an inlet distortion with fixed strength, the stall margin loss increases as the rotational speed decreases.

**NOMENCLATURE**

- BPR: Bypass Ratio
- DC: Distortion Coefficient
- DFT: Discrete Fourier Transform
- EXP: Experimental data
- FPR: Fan Pressure Ratio
- LP: Low Pressure
- NS: Near stall
PE  Peak efficiency
ΔPRs  Loss in Stall Pressure Ratio
PRd,s  Distorted total pressure ratio, near stall
PRu,s  Clean flow total pressure ratio, near stall
Pt  Total Pressure
PVD  Pressure Variation Diminishing
SFC  Specific Fuel Consumption
Tt  Rotor inlet total temperature
RANS  Reynolds-Averaged Navier-Stokes
URANS  Unsteady Reynolds-Averaged Navier-Stokes

Acknowledgments
The first author would like to thank CSC (China Scholarship Council) for funding this work.

Reference