Multi-fidelity simulation for a transonic compressor with inflow distortions

Jiahuan Cui, Yunfei Ma, Bryn Ubald, Paul Tucker

Abstract
The low and high fidelity methods are used to study a transonic fan case. This transonic fan case features a distortion generator upstream of the rotor. The low fidelity methods (immersed boundary method and immersed boundary method with smeared geometry) show promising results against hi-fi simulation (unsteady RANS) and experiments.

The flow in the tip region, especially the interaction between the distorted inflow and tip leakage flow, are investigated in detail. It is found that the tip leakage flow is the dominant flow structure near the casing. It is strong enough to sustain even at the exit of the stator. Apart from analysing the tip leakage flow structures, the stalling mechanism is also explored. As mass flow decreases, more rotor blades are contaminated by the distorted flow. A rotating cell starts to develop shortly after all the rotor blade passages are contaminated. The rotating cell is found to be swept in circumferential direction at the 65% speed of the rotor.

Keywords
Unsteady flow — Stall mechanism — Inlet distortion

INTRODUCTION
Flow distortions in a modern aero engine system are often induced due to the integration of different components. Inflow distortions can reduce the performance and the stall margin of a compression system [1]. Nacelle stall and boundary layer ingestion are two common scenarios where inflow distortions are interacting with compressors/fan. In these two scenarios, inlet distortion mainly affects the tip region of the compressors/fan.

Flow in the tip region of the rotor is highly complex and unsteady [2, 3]. Due to the clearance between the rotating blade tip and the stationary casing, several vortices are formed. As demonstrated by You and Moin [4] and also illustrated in figure 1, tip vortex system consist of 1) tip-leakage vortex, 2) induced vortex, and 3) tip-separation vortex. The tip-leakage vortex is the dominant one.

For a transonic fan/compressor, the tip leakage vortex is strongly affected by the shock. The shock intersects the trajectory of the tip leakage vortex. As the flow coefficient is reduced, the tip leakage vortex strengthens and moves upstream [5]. When the tip leakage vortex cannot overcome the adverse pressure gradient, the breakdown of the tip leakage occurs. As a result, the blockage in the tip region increases, which further reduces the velocity. As less mass flow travels through the region near the casing, the blade loading decreases. If the mass flow decreases further, the tip leakage vortex starts to travel tangentially towards the adjacent blade leading edge[6]. This is also termed leading edge tip clearance flow spillage. At this condition, the rotating stall can occur.

The stalling process becomes more complicated in the presence of a distorted inflow. Such highly turbulent and complex flow is very challenging to be predicted accurately, even for the state of the art of computational fluid dynamics[7]. In the current study, a transonic fan (the Darmstadt case) will be used to investigate the interaction between the inflow distortion and the downstream rotating blades. The Darmstadt case is featured by a distortion generator (DG) located two axial chord upstream of the compressor. The DG is attached to the case and it is designed to generate a large separation to represent a stalled engine nacelle [8]. The current paper will investigate the general flow features in the tip region.
affected by the inlet distortion. A range of different levels of fidelity modelling strategies will be used to study different aspects of the flow. In addition, the stalling mechanism under the influence of inlet distortion will be explored.

1. CASE SETUP

The Darmstadt transonic compressor case is used in the current study. The case itself has been studied by many other researchers [8, 9, 10]. It consists of 16 rotor blades and 29 stator blades. The spool speed is 20,000 rpm. This gives a total pressure ratio around 1.5 at the design condition. To investigate the inlet distortion on the performance and stability margin of the downstream compressor, a distortion generator is attached to the casing with a distance of two axial chord length upstream of the rotor leading edge. The height of the distortion generator is designed to be 10% of rotor span. Such a distorted inflow is large enough to influence the rotor tip region.

The computational domain used in the current study is shown in figure 2. Depending on the fidelity level of the simulation, the blades and distortion generator are modelled by different methods.

High fidelity simulation. A range of unsteady RANS simulations are carried out to study the detailed flow features at different total pressure ratio along the performance line, including the stall point. Figure 3 shows the surface mesh of the blades and the distortion generator. In total, around 65 million cells are used to resolve the whole domain, with 23 million cells for the rotor blades and 34 million for the stator. On average, there are around 1.5 million cells for each blade passage.

Low fidelity simulation. At the design stage, the ability to account for the effects of the fan on the flow upstream is extremely beneficial, especially for a short-intake design at the high angle of attack. The low fidelity method can accurately capture the potential field that created by the fan with a cost only a fraction of the hi-fi simulations. The current study performed several low fidelity simulations with the distortion generator resolved by the immersed boundary method (IBM) and the blades by immersed boundary method with smeared geometry (IBMSG). Details of these methods are giving in the Numerical Methods section.

2. NUMERICAL METHODS

The numerical solver used by the current study is a Rolls-Royce in-house CFD (Computational Fluid Dynamics) code – HYDRA [11]. HYDRA is a second order finite volume RANS (Reynolds Average Naiver-Stoke) solver. The solver has been used to attack a range of challenging problems in aero engines [12, 13, 14]. Here, the IBMSG and IBM are developed within the HYDRA framework. A brief summary of these two methods are given below in this section.

2.1 IBM

IBM is used to avoid meshing the complicated DG geometry in the low fidelity simulation. In the currently study, the feedback force method [15] is used to calculate the force terms:

$$ f = \alpha \int_0^t (v - u)dt + \beta(v - u) $$

where, $v$ is the intended velocity on the boundary (zero in the current study) and $u$ is the actual local velocity on the boundary. The force term is added to the NS equations so that the actual local velocity is driven to the intended boundary velocity as the solution is converging. The constants, $\alpha$ and $\beta$, can affect the convergence speed.

2.2 IBMSG

The immersed boundary method with smeared geometry is used to develop a low-order compressor/fan model. This low-order model is useful in cases where calculations for fully resolved blades are extremely expensive. For example, at the
Multi-fidelity simulation for a transonic compressor with inflow distortions

1. Introduction

2. Methodology

3. RESULTS AND DISCUSSIONS

3.1 Validation.

To obtain the characteristic line from numerical simulations, the pressure at the outlet was increased step by step, from the peak efficiency (PE) condition to the near stall (NS) condition. Figure 4 shows the characteristic line at the 100% speed for experiments (solid line), low fidelity simulations (dots), and high fidelity simulations (squares). In general, both low and high fidelity simulations agree with the experimental results. Although the choking mass flow and the pressure ratio at near stall condition are slightly (1-2%) different between the numerical results and the experiments, these differences are marginal and do not compromise the conclusion of the current study.

Figure 5 shows the total pressure ratio in circumferential direction at three measurement stations ($S_1$, $S_2$, and $S_3$). These three stations are illustrated in figure 2. The effects of the distortion generator can be seen between $225^\circ$ – $325^\circ$, where the total

\[
\frac{\partial}{\partial t} \int_{\Omega} \lambda q d\Omega + \int_{A} \lambda F dS = \int_{\Omega} \lambda f_n d\Omega + \int_{\Omega} \lambda f_p d\Omega + \int_{\Omega} B d\Omega
\]  

(2)

where $q$ are the primitive variables. $F$ are the sum of inviscid and viscous terms. $f_n$ are the force terms normal to the blade camber line. $f_p$ are the force terms parallel to the blade camber line. $\lambda$ is the blockage factor. $B$ are the extra terms due to blockage. The detailed description and calculation for the each term can be found in [16].

Figure 6. Total pressure ratio in circumferential direction at $S_3$ for three spanwise locations.
pressure ratio is lower. The distortion generator creates a large separation, which dissipates energy and results a lower total pressure region downstream.

This lower pressure region is convected downstream. At the both rotor and stator exits (figure 5(b, c)), the lower pressure region is still visible. The distorted region is also extended in circumferential direction at the downstream. As the blade travels from right to left in the figure, at the exit side of the distorted region (the left side of the lower pressure region), the total pressure is increased slightly. This is due to the rotor incidence variation. More details are given in section 3.2.

In figure 6, the total pressure ratio is plotted at three different span locations: near hub, mid span and near case. The effects of the inflow distortion extends to the whole span at the exit of the stator. Both the URANS and IBMSG captures the distorted region, though the averaged total pressure in the IBMSG calculation is higher at the mid span and lower near the case.

### 3.2 General flow features.

In this and the following sections, results from URANS will be studied in more detail. Figure 7 shows the Q-criterion coloured by Mach number in the region between 85% - 100% span. To increase the visibility into the blade passage, only 15% of span is shown. Also, the annular cascade is transformed to a linear one. The y coordinate in figure 7 is the circumferential distance and the z coordinate is the radius. To calculate the circumferential distance, a fixed radius at 92.5% of span is used rather than the local one. This is to make sure that the blades at the lower and upper bound of the y coordinate are not leaned.

**Flow within the separated region immediately downstream of the DG.** The distortion generator creates a large separation upstream of the rotor. The separated flow constantly sheds vortices. This results in a highly unsteady region downstream of the DG. In this region, the velocity is lower than in the undistorted region. As a result, the rotor blades experience a variation of the incidence in the circumferential direction.

Since the static pressure in the distorted region is lower, the vortices formed at the circumferential end of the DG converge towards the centre (see figure 7). As a result, the circumferential velocity increases at the entry side of the distorted region and decreases at the exit side. Hence, the incidence is higher at the exit and lower at the entry side. This is the reason why the total pressure is higher at the exit in figure 5(b,c). Similar results can be found in section 3.1.

**Flow structures within the rotor blade passage.** Upstream of the rotor, the leading edge shocks create vortices on the casing. These vortices are relatively small compared to the tip leakage vortex. Similar to the findings in the literature, all the three main vortices – induced vortex, tip-leakage vortex and tip-separation vortex are captured in the simulation. However, in the distorted region, the tip leakage vortex breaks down immediately after the shock. After the rotor exits the distorted region, the flow recovers to the normal state. Around three/four blade passages are significantly affected by the distortion generator. However, as the mass flow...
Multi-fidelity simulation for a transonic compressor with inflow distortions

Figure 8. Flow at 95% span at PE point; (a) Mach number, (b) total pressure, (c) entropy, and (d) entropy generation rate.

Figure 9. Flow at 95% span at NS point; (a) entropy, and (b) local entropy generation rate.

decreases, it takes longer for the distorted flow to convect out of the rotor passage. More blade passages are expected to be affected.

Flow structures within the stator blade passage. The flow structure in the stator blade passage is relatively cleaner. The tip leakage vortex generated in the rotor convects downstream to the stator passage. They are not entirely dissipated even at the stator exit. In the blade passages that are affected by the distortion, the secondary flow becomes stronger. No tip leakage vortices can be observed in those blade passages, as the tip vortices have already been broken down in the rotor.

3.3 Flow at peak efficiency and near stall points.

After understanding the general flow features in the blade passage, the difference between the flow at the peak efficiency point and the near the stall point will be examined. Only the region near the casing is studied here.

Figures 8 and 9 show the flow at 95% of span for the peak efficiency point and near stall point, respectively. All the quantities are based in the absolute frame. As mentioned before, for both operating points, the distorted flow converges towards the middle. This leads to an increase of incidence at the exit side and a decrease at the entry side of the distorted region. As a result, higher entropy is observed at the exit side in the rotor blade passages (figure 8(c)). At the peak efficiency point, most entropy is generated in the separated region downstream of the distortion generator. Around four rotor blade passages are contaminated, where entropy generation rate is higher. Consequently, around five stator passages are contaminated. Other loss sources, such as shocks, wakes and secondary flows, can also identified in the figure 8(d).
Among them, wakes are the second dominant source of the entropy generation. Interestingly, the interaction of the rotor wakes and the stator suction surface is also a large source of entropy generation.

At the near stall point, only entropy and entropy generation rate are shown. Note that to highlight the entropy generation within the blade passages, the colouring levels are different from that in figure 8. At the near stall condition, the axial flow velocity is lower. The convection time for the distorted flow to travel through the rotor is longer. Hence, more rotor blade passages (around 9) are contaminated by the distorted flow. Other entropy generation sources also increase due to a higher pressure gradient in the rotor and stator. At this condition, all the boundary layers on the suction surface of the stator are separated, except for the three passages at the entry side of the distortion. This is, again, due to the lower incidence and lower pressure rise at the entry side. The separated boundary layer is also a large source of entropy generation at near stall condition.

3.4 Stalling process.
As mass flow decreases further, more blade passages will be contaminated. Eventually, the compressor would be fully stalled. Figure 10 shows the radial velocity signals at 95% of the span upstream of the rotor. At $t_0$, the mass flow rate is around 13.8 kg/s. At this mass flow rate, rotor passage between $-1$ radian to $3$ radian are contaminated, where the radial velocity is oscillating more than that at $-3$ radian to $-1$ radian. As the mass flow decreases, the contaminated region extends wider in the circumferential direction. At $t_1$, the mass flow rate drops to around 13.5. Almost all the rotor blade passages are contaminated. Shortly after all the blade passages are contaminated, a rotating cell starts to develop at $t_2$. It rotates along the circumferential direction at the 65% of the rotor rotating speed. As the rotating cell starts to develop, the mass flow rate drops faster.

4. CONCLUSIONS
The current paper demonstrated the applications of low and high fidelity simulations for a transonic fan. The low fidelity methods (IBM and IBMSG) showed promising results against hi-fi simulation (URANS) and experiments. These low order methods are useful in designing a coupled system where mesh resolved blades are computationally expensive.

For hi-fi simulations, the detailed flow features near the casing were analysed. It was found that the tip leakage flow was stable and strong enough to sustain even at the stator exit. Apart from the analysis of flow structures, the stalling mechanism was also investigated. When mass flow was decreased, more rotor blades are contaminated by the incoming distorted flow. As mass flow reduced to the near stall condition, separation can be observed over most of the stator blade suction surface. A rotating cell started to develop shortly after all the rotor blade passages were contaminated. The rotating cell was swept around at the 65% speed of the rotor.

ACKNOWLEDGMENTS
The authors would like to acknowledge the insightful discussion and input from Dr. Rob Watson, Dr. James Page, Dr. Yangwei Liu and Mr. Callum Mantell.

REFERENCES


