Non-Intrusive Flow distortion measurements within a Turbofan Intake (NIFTI)- De-risking the test campaigns with CFD analysis

W. Zhang
Imperial College London
w.zhang15@imperial.ac.uk
London, UK

B. Tiedemann
Imperial College London
benedikt.tiedemann20@imperial.ac.uk
London, UK

S. Hegen
German-Dutch Wind Tunnel
sinus.hegen@dnw.aero
DNW, Marknesse, Netherlands

M. Vahdati
Imperial College London
m.vahdati@imperial.ac.uk
London, UK

Pavlos Zachos
Cranfield University
p.zachos@cranfield.ac.uk
Cranfield, UK

F. Zhao
Imperial College London
fanzhou.zhao11@imperial.ac.uk
London, UK

ABSTRACT
Short and slim aero-engine intake designs for very high bypass ratio configurations cause notable levels of steady and unsteady distortions at the fan face, especially under crosswind or AOA operation during aircraft take-off. Such distortions can adversely affect the engine’s performance, operability, structural integrity and safety margin with potentially catastrophic consequences for the entire propulsion system. There is a significant lack of open published measured data for fans and compressors operating under inlet distortions, which is mainly due to the difficulties and cost involved in setting up such experimental campaigns. The Non-Intrusive Flow distortion measurements within a Turbofan Intake (NIFTI) project aims to address this gap by demonstrating a non-intrusive technique for measuring velocity fields across a plane located upstream of a large diameter fan of a high bypass ratio aero-engine which has never been achieved. This method will provide synchronous datasets across the measurement plane with at least one order of magnitude higher spatial resolution than current methods. The experimental activities will be supported by numerical campaigns and advanced data processing of measured data, which will be used to analyse the highly unsteady nature of the flow distortions. The outcome of this project will ultimately unlock the complex aerodynamics of closely coupled fan-intake systems and aid the validation of CFD codes, as well as establish novel design guidelines for future, stall-tolerant aero-engines. This paper reports on the initial finding from CFD analysis of NIFTI project, focusing on the investigation of the interaction between the distortion caused by the AOA (upstream) and pylon (downstream) on a low-speed fan blade operation and addressing the possible aerodynamic instabilities which can arise for such a system. The aim is to use CFD to predict aerodynamic and in-future aeroelastic stability of the coupled fan-intake configuration prior to the test campaigns in order to computationally determine the operating envelope of the rotating fan experimental facility.

INTRODUCTION
Performance and environmental targets drive the design of next generation aircraft architectures towards closer integration between the propulsion system and the airframe. Current research suggests that these architectures are likely to feature aero-engine concepts with very high bypass ratios and larger fan diameter, where a short and slim intake design will be necessary to compensate for the additional aerodynamic drag and weight penalties of the increased diameter fan [Suder et al., 2013; Hughes, 2011]. Although such aircraft configurations may meet the future targets, the capability of the short inlet duct to attenuate the inlet distortion via internal flow diffusion is decreased which makes the occurrence of inlet distortion more severe. Such distortions can adversely affect the engine’s performance, operability, structural integrity and safety margin with potentially catastrophic consequences for the entire propulsion system. Today these problems are typically discovered and addressed late in development programmes and can consequently have serious negative impacts on the aircraft development timescales, cost and certification.

This work is licensed under Attribution-NonCommercial-NoDerivatives 4.0 International (CC-BY-NC-ND). See: https://creativecommons.org/licenses/by-nc-nd/4.0/legalcode
With the increase of computational capability and improvements in numerical models, Computational Fluid Dynamics (CFD) is becoming increasingly powerful and favoured by scientific researchers and industrial engineers. Time-accurate, high-fidelity CFD simulations of the fan behaviour at extreme operational conditions (such as in-stall with inlet distortion) has become possible [Lee et al., 2019]. Moreover, CFD provides an economical alternative strategy to subscale rig test for the modelling of (an aero-engine size) fan aerodynamic and aeroelastic performance with inlet distortion. The above indicates that validated CFD codes are going to play an important role in future designs, which highlights the need for reliable, accurate and detailed validation data from measurements. However, a literature survey of publications shows that there is a significant lack of open published measured data for fans operating under inlet distortions, which is mainly due to the difficulties and cost involved in setting up such experimental campaigns. A look at available literate shows that many researchers still use NASA Rotor 67 [Strazisar et al., 1989] and NASA Rotor 35 [Reid, 1978] for CFD validation purposes. These measurements were carried out with outdated techniques and contain large uncertainties in measurement (e.g., tip clearance, size hub leakage flow and complex undefined geometries at the hub and casing). Moreover, these experimental campaigns were carried out by using fixed distortion screens upstream of the fan. It has been shown in [Lee et al., 2019; Carnevale, 2017] that due to the flow acceleration upstream of the fan face, the fan loading has a suppression effect on the intake separation. Moreover, the current measured data shown in literature, such as [Tambe et al., 2020], is proprietary and not open. The above indicates that fan/distortion interaction plays an important role on the fan performance and stability, which cannot be modelled using simple distortion screens.

In the NIFTI project a non-intrusive technique based on a multi-camera S-PIV [Elsinga et al., 2006] will be performed. It will be used to measure the velocity fields upstream of the fan of an UHBR aero-engine in a 8 x 6 m² section industrial wind tunnel with air flow velocity representative of engine operating conditions. The project will deliver instantaneous velocity fields with further potential for fully time-resolved data in the order of 8-10 kHz. This will deliver a significant amount of measured data which can be used for CFD validation and development.

In the numerical part of the NIFTI project, CFD is used to predict aerodynamic and aeroelastic stability of the coupled intake-fan configuration ahead of the test campaigns in order to computationally determine the operating envelope of the rotating fan experimental facility. Whenever possible, available measured data is used to validate CFD results, but there is a limited amount of measured data in the form of total pressure and temperature which were obtained in 2009 for clean flow. Therefore, detailed analysis based on past experience and physical reasoning is used to demonstrate the validity of numerical simulations. Previous work [Lee et al., 2019; Zhang and Vahdati, 2019; Fidalgo, 2013; Pardo and Hall, 2021] has shown that inlet distortion can result in loss of stall margin. Therefore, for successful outcome of this test campaign it is essential to find safe operating points so that extensive measured data can be obtained. The objectives of this paper are:

- Validate the CFD code based on the limited amount of measured data available.
- Establish how the downstream distortion (e.g., pylon) influences the blade operating condition.
- Establish how the intake influences the blade operating condition.
- Establish the interactions between downstream distortion and intake distortions and their effects on fan operation.
- Determine the best way of mounting the geometry in the wind tunnel to avoid premature stall and large-scale vibration of fan blade.

In the first phase of this work, fan operation due to Angle of Attack (AOA) to the intake is considered and the interactions between upstream and downstream distortions is analysed. The unsteaday pressure obtained on the blade can also be used in conjunction with a Finite Element solver to predict aerodynamic forcing on the blade.

**TEST GEOMETRY AND DISCUSSION OF CURRENT AVAILABLE MEASURED DATA**

The starting point is an existing Turbine Powered Simulator (TPS) used at German-Dutch Wind Tunnels (DNW) which was used for wind tunnel experiments in order to analyse engine installation effects caused by the pylons. To demonstrate the novel non-intrusive measurement techniques NIFTI will deliver a wind tunnel model designed around the DNW 10.2” (260 mm) TPS fan unit [DNW PROPULSION, 2012]. The 10.2” TPS fan was specifically designed to represent turbofans with high bypass ratios and low Fan Pressure Ratio (FPR). A view test of Isolated TPS powered inlet rig in the DNW-NWB closed test section is shown in Figure 1(a). The fan stage (Figure 1(b)) was designed by DLR [Demmat, et al., 2009] and contains 19 rotors and 25 stators. The fan design pressure ratio is 1.37 and the hub/tip ratio is 0.42. The intake was designed by the NIFTI consortium using the Common Research Model(CRM) [Vassberg et al., 2008].

One concern is the relatively small size of the simulator unit (10.2”) which is a fraction of an engine scale fan. This can result in presence of laminar and or transitional boundary layers and their interaction with shockwaves which can potentially cause large amplitude vibration [di Mare et al., 2009]. To avoid such scenario, transition strips will be used close to the leading edge (LE) of fan blades in this test campaign.
A schematic representation of the instrumented nacelle is given in Figure 2a. In the original measurement campaign (without inlet distortion) fan performance map was obtained with three exhaust geometries and measured characteristics are plotted in the fan map of Figure 2b. Moreover, at each operating point, total pressure and temperature were measured downstream of the stator vanes (axial position 3 in Figure 2a) by placing rakes around the circumference, and 5 probes in the radial direction were used at each circumferential position. Typical measured data is shown in Figure 3b. It is seen from this plot that there is a significant variation in fan performance at different circumferential position, for example the fan pressure ratio is around 1.5 for the rake 1 whereas it is 1.3 for rake 5. A schematic plot of fan performance at different circumferential positions using an orbit method [Fidalgo, 2013; Zhang and Vahdati, 2019] is shown in Figure 2b. The cause of this so-called ‘orbit’ behaviour of fan performance is the pylons which are downstream of the stator vanes. There are two pylons which are located at the Top Dead Centre (TDC) and Bottom Dead Centre (BDS) respectively. As can be seen from Figure 4(a), the upper pylon at TDC is much thicker and is axially closer to the stator vanes than the lower pylon. Consequently, it has more impact on the pressure profiles as can be seen from Figure 3b. It can be seen from the orbit shape (dashed line) in Figure 2(b), that although the steady operation of the whole assembly is away from stall boundary, the existence of the pylons causes some of the blades around the circumference to operate close to the stall boundary. It was shown in [Zhang and Vahdati, 2019] that the inlet distortion can create similar orbit patterns and cause some blades to operate above the stall line. The current test data indicates that the existence of pylon does not cause the fan to stall with the three exhaust geometries (see Figure 2b, the three working lines are below the stall line) which was used in the test campaign. However, in presence of crosswind or AOA there will be an addition distortion pattern upstream of the fan that will interact with the downstream distortion. It was shown in [Zhang and Vahdati, 2019; Zhang and Vahdati, 2020] that in presence of inlet distortion, three main parameters that determine if the fan will stall or not are 1) Operating point (exhaust geometry), 2) Distortion strength and circumferential extent and 3) Blade timing. As the blade is already designed condition 3 will not be addressed in the CFD work. One of the main objectives of the preliminary CFD work is to recommend an exhaust geometry so that the test campaign with inlet distortion stays away from stall boundary when subjected to maximum levels of inlet distortions. The other objective is to find the best mounting orientation of the intake to minimise the stalling effects that the interactions between upstream and downstream distortion will create.

1 The throttle area downstream of the compressor
FLOW SOLVER

This study is based on the in-house hybrid RANS/LES flow solver HADES. It uses a node-centred finite volume scheme with an edge-based data structure. The numerical fluxes are solved by the Jameson-Schmidt-Turkel scheme [Jameson et al, 1981] with a Roe’s matrix [Roe, 1981], leading to second-order accuracy in space except for the vicinity of a shock. The second-order implicit time integration scheme is applied for temporal discretization. For simulation of turbulent flows, the standard Spalart-Allmaras turbulence model [Spalart, and Allmaras, 1994] and its variants [He et al., 2020] are available for the RANS branch, and the standard DDES method [Spalart et al., 2006] and its upgrades [He et al., 2021] are available for the hybrid RANS/LES branch. Validations of HADES can be found in previous works [He et al., 2020; He et al., 2021]. The computations carried out in this paper use the RANS/URANS mode of the solver by using the SA turbulence model. In future studies DDES will be used in the intake to model inherently unsteady nature flow in the intake. The performance of the DDES in this code is shown in [He et al., 2020; He et al., 2021].

GRIDS USED

The computation domain contains 5 component geometries: intake, rotor, stator, pylon and nozzle. Initially a grid independence study was performed for each of the 5 domains and optimum grids based on accuracy, computational efficiency and past experience [Zhang and Vahdati, 2017] was chosen for each domain. The rotor and stator domains are meshed by using AUTOGRID [NUMECA, 2020] with 1.0 million and 0.5 million points respectively. The number of mesh points in the spanwise direction is 70 for the rotor and 50 for the stator. To model tip leakage flow accurately, 11 layers are placed inside the tip gap. The average value of first layer $Y^+$ is around 20. The standard wall function based on the law of the wall [Bradshaw and Huang, 1995] is used to calculate the shear stress at these points. The intake and pylon grids which contained 4.5 and 1.1 million points respectively are generated using GAMBIT. The nozzle domain is meshed by a hexahedral grid with 0.5 million points.
STEADY RESULTS

In the first step pseudo-steady results were obtained which are used as the initial condition for unsteady computations. The domain used for steady state computations is shown in Figure 5(a). The domain consists of 360° models of the intake and the pylon domain but only single passage for the rotor, stator and nozzle. The interface boundary between different domains is modelled using mixing planes and the computations are performed in a steady mode. Although this type of modelling cannot model the transfer of circumferential non-uniformities between different blade rows, it can provide the closest approximation which can be used as an initial condition for the unsteady computations as the flow quantities such as mass flow will be matched between different domains. Moreover, such computations are low-cost (as they are steady) and can provide first-order approximation of the distortion impact. This approach was also used for computations with AOA and will be used for crosswind computations in future. The inlet conditions are specified at far field by specifying ambient pressure and temperature, as well as aircraft inflow Mach number and AOA. The flow through the fan is controlled by placing a variable-area nozzle downstream of the pylon [Vahdati et al., 2005]. This nozzle allows the computation to be conducted at any point on the constant speed characteristic by simply modifying the nozzle area ratio. This nozzle is choked and hence the solution is independent of the conditions specified downstream of it.

![Figure 5 (a) computational domain for steady computations (SP denotes single passage and FA denotes full annulus) (b) compressor map of steady computations](image)

Steady computations were performed at three corrected speeds and the corresponding constant speed characteristics were obtained by adjusting the throat area of the downstream nozzle. Figure 5(b) shows the corresponding steady results. The computed working line (WL) is also plotted against the measured data. Despite the flow being assumed circumferentially symmetric across the mixing plane, it can be seen that the results agree qualitatively with the measured data at higher speeds. This will be discussed in the next section.

INITIAL UNSTEADY COMPUTATIONS

Four sets of unsteady computations were performed at a rotational speed of 28900 rpm and a nozzle area ratio which correspond to a mass flow of 8.42 and pressure ratio of 1.37 shown in Figure 5(b). The difference in the computations was in the presence and location of upstream and downstream distortions. The computed cases correspond to:

1) Case A0-P - Zero AOA with pylon; which explores the effects to downstream distortion on the fan performance
2) Case A30 - 30° AOA without pylon; which explores the effects to upstream distortion on the fan performance
3) Case A30-P(90) - 30° AOA with pylon; which explores the effects of the combined upstream AOA at 90 degree (Figure 3a) and downstream pylon on the fan performance
4) Case A30-P(180) - 30° AOA with pylon; which explores the effects of the combined upstream (AOA) at 180 degree (Figure 3a) and downstream pylon on the fan performance

The inlet Mach number in both cases was 0.3, hence the ‘steady’ operating condition of the fan blade remains the same for all cases.

The aim of this approach is to isolate each component of distortion and, initially, look at their effects separately. Moreover, there is limited measured data for Case A0-P which can be used for the validation of CFD code. The computational domain for the unsteady computations of Case A0-P, Case A30-P(90) and Case A30-P(180) is shown in Figure 6. It includes the full annulus fan assembly with stator vanes and pylons, a symmetric intake upstream of the fan, and an external volume, which contains the rig. In future work an external boundary similar to that of the wind tunnel will be used. The same domain was used for Case A30 but the pylon geometry was removed.

---

² In the absence of the pylon and AOA, steady and unsteady computations obtain similar results in terms of mass flow, total pressure ratio and circumferentially averaged radial profiles, not shown here.)
Figure 6 Computational domain for unsteady computations

Figure 7 shows the numbering convention that will be used for the blade sectors and circumferential angles in the rest of this paper. Each blade sector represents a circumferential control volume at the fixed position in the absolute frame of reference.

Figure 7 The blade sector number and circumferential angle in the simulation

**Case A0-P**

The results for **Case A0-P** are discussed next. Figure 8 shows instantaneous variations of Mach number at 80% blade height of the rotor. It can be seen that the upper pylon is much larger and axially closer to the stator vanes, and hence will have a larger impact on the fan operation. Figure 8(a) shows the contour of the relative Mach number at 80% rotor span, which extends from 0.5 axial chord length upstream the LE to 0.5 axial chord length downstream of the TE. The impact of upper pylon (0 degree in Figure 7) can be noticed by the flow on the blades shown in the circled area and comparing the wake thickness. The blades which have rotated past the upper pylon have thicker wakes indicating that they have higher loading. It is also seen that it takes about 2-3 blade passing time for the blade passage to be affected by the pylon. Cousins [Cousins, 1997] showed that the rotor response time for an axial fan blade subjected to inlet distortion can be calculated as the time for a flow particle to travel the distance from the LE of the aerofoil to the throat of the blade passage (blade time constant), and a rule of thumb suggested that it takes 4 to 5 times of the blade time constant for the stability limit of the compression system to be affected. Similar analysis regarding the blade time constant of the downstream distortion will be shown in future work. It is seen from Figure 8b that the flow remains almost symmetric for the blades which are immediately in front of the lower pylon (180° in Figure 7), which indicates that this pylon has a relatively small impact on the fan operation.
Figure 8 Contour of the relative Mach number at 80% rotor span, zero incidence, upstream of the upper pylon (a) and lower pylon (b);

Figure 9 (a) Total pressure contour downstream of the stator together with numerical rake positions. (b) Radial profile of the total pressure ratio at different rake circumferential positions.

Figure 9 shows the time-averaged variations of total pressure downstream of the stator for *CasA0-P*. In this plot the upper pylon is at 0 degree and the fan is rotating clockwise. Figure 9(b) shows the variations of total pressure rise (normalised with the static pressure in the tank) at the rake positions shown in Figure 9(a). It is seen from this plot that the presence of the upper pylon produces a significant variation in downstream total pressure which is in line with the measured data. The total pressure pattern becomes asymmetric when the blade approaches and moves away from the pylon (see Rake 1 and Rake 6). There is a large drop in the measured total pressure on rake 1 from 80% upwards which is matched well by CFD. Figure 9(a) shows that this drop is due to the low momentum flow at this region (separation near the tip, which can be clearly identified in Fig10(a)) and stall is expected to initiate in this region. Moreover, most of the circumferential variation appears at Rakes 1 and 6, which is due to increased loading by the potential effects of the upper pylon. It is also apparent from this plot that the fan blades recover to normal operation at circumferential positions away from pylon. The lower pylon has much smaller influence as can be seen by the small differences between Rakes 2-5. In the rest of the paper, pylon refers to the large pylon at circumferential position 0 degree.

**Case A30**

The result for *Case A30* is discussed next. Figure 10 shows the variations of total pressure at the rotor inlet and downstream of the stator for *Case A30*. In Figure 10(a), time-averaged variations of total pressure are shown. In this plot the AOA distortion is at 90 degree (see Figure 9(a)) and the fan is rotating clockwise.
Figure 10 (a) Total pressure contour at the rotor inlet. (b) Total pressure contour downstream of the stator. (c) Circumferential flow angle and total pressure at the rotor inlet.

The circumferential flow angle and the total pressure at the 80% span are plotted in Figure 10(c). It can be seen that the AOA distortion increases the swirl angle in the regions between 270–360 and 0–90 (which corresponds to the upper half of the annulus). Consequently, the blade loading in this region increases due to a higher incidence. The region between 90–270 degree has negative swirl angle. This region corresponds to the lower half of the annulus and the blade loading within this region is lower due to the reduced incidence. This can be seen from Figure 10(b). The total pressure has a drop between 0 and 180 degree (right half of the annulus) at the rotor inlet which is due to AOA-induced flow separation (blue region in Figure 10(a)). On comparing the total pressure contour in Figure 9(a) and Figure 10(b) it is apparent that the effects of AOA are less pronounced than the pylon on the fan unsteady operation.

Comparison of Case A0-P and Case A30

In this section the flow physics of Case A0-P and Case A30 are compared.

Figure 11 The control volume used for orbit calculation, mid-passage

In order to examine the blade performance at different instants of time as it rotates around the annulus, an orbit method is used [Fidalgo, 2013]. The original orbit method requires recording and post-processing a series of flow solutions of the whole domain at fixed time intervals. Significant amount of interpolations and calculations are needed to account for the cross-passage flow migration due to distortion. In the analysis shown in this work, however, it is considered that the circumferential flow migration is not significant and a simplified orbit method (SOM) is used. In this method, an instantaneous solution of the whole annulus rotor domain (containing all 19 blades) is taken and the whole annulus domain is split into 19 sectors by using the mid-passage surfaces (see Figure 11). Each sector is a closed control volume which extends from the rotor LE to the rotor TE. The corrected mass flow at the inflow surface of each sector and the total pressure ratio between the exit surface and inflow surface are computed and plotted on the compressor map. One important advantage of the original orbit method is that it takes advantage of the time-averaged streamlines to split the sectors. In this approach the inlet and exit mass flow will match for every sector. For the SOM used here, the time-averaged streamlines can be different from the mid-passage as circumferential flow migration can be affected by the upstream flow incidence and the downstream pylon. It is assumed that this method will inevitably introduce errors on the estimation of the working condition of the blade. Therefore, the SOM is used in this work as a fast, low-fidelity post-processing method to estimate the performance of the blade around the annulus. To measure the uncertainty of the SOM, the relative mass flow difference is computed as:

\[
Error_{mass} = \frac{|Mass_{inlet} - Mass_{exit}|}{Mass_{inlet}}
\]

For the cases shown in this paper, the largest mass flow difference is around 4% while most of the errors are within 1%.
Figure 12 shows the performance of each sector for Case A0-P and Case A30 together with the fan constant speed characteristic at 28900 RPM. The overall performance of the fan for both cases is also shown in this plot. For a periodic unsteady solution, the instantaneous performance of each sector can be regarded as the operating condition of a rotor at different circumferential positions. Thus it can be seen that the existence of the pylon makes the operating condition of the fan blades highly unsteady. It can be seen that for the Case A0-P the loading of the blade increases as the blade rotates from sector 1 to sector 3, which is mainly due to the blockage of the downstream pylon. Sector 3 has the highest loading and operates past the stall line, hence it is expected to initiate stall. The corrected mass flow and pressure ratio decreases when the blade moves from sector 4 to sector 10. The working condition of sector 10 to sector 17 falls on the compressor steady map but have a larger mass flow and a lower total pressure ratio than the overall performance. This is due to the presence of pylon which unloads these blades as more flow diverts away from the pylon.

For the Case A30 (with 30° AOA from 90 degree), the orbit shape is less widespread than the Case A0-P, indicating that the 30° AOA has less impact on the fan operation than pylon. The blades can be classified into two groups: sectors 16-19 and sectors 1-4 which have a larger corrected mass flow and a higher pressure ratio, blade sectors 11-15 (Figure 7) which have a lower corrected mass flow and pressure ratio. The corrected mass flow is mainly influenced by the inlet total pressure and the pressure ratio is determined by the swirl angle at the rotor inlet [Fidalgo, 2013]. It was shown in Figure 10(c) that the incidence angle is higher between 270~360 and 0~90 degree (corresponding to sector 16-19 and sector 1-4) resulting in a higher pressure ratio than the average condition. Sector 6 to sector 15, with a reduced incidence, has a lower total pressure ratio than the overall working condition. As the sector from 180~360 has a higher total pressure (Figure 10(c)) and the sector from 0~180 has a lower total pressure, the mass flow drops from sector 1 to sector 12 but increases from sector 13 to sector 19. Sectors from 11 to 15 have negative swirl angle and lower total pressure. Sector 5 which correspond to the area of maximum distortion, is affected significantly by the inlet total pressure distortion caused by the AOA which reduces the mass flow.

Unsteady results at 30 AOA

In this section, 2 different cases are performed and analysed (see Figure 13):

1) Case A30-P(90) - 30° AOA at 90 degree angle with pylons
2) Case A30-P(180) - 30° AOA at 180 degree angle with pylons
The green blade in Figure 13(a) is at the circumferential angle of 0 degree (see Figure 7). The blade rotates clockwise and sweep the rakes from 1 to 6 sequentially. It can be seen that when the flow comes from 90°, a lip flow separation is generated within the intake which is significantly larger than when the flow comes from 180°, as indicated by the dashed circles. It was shown in [Lee et al., 2019] that the fan loading has a suppression effect on the intake separation due to the flow acceleration upstream of the fan face. In addition, it was identified that the suppression effects of the fan depend on the operating point of the fan and decrease as the fan approaches the stall boundary. It is clear from Figure 12 that for Case A30-P, the fan blades which are around the circumferential position 90° (sector 5) operate near stall and hence have less ability to suppress the inlet separation (at this position) than the blades which are around the circumferential position 180° (sector 10). Consequently, the level of inlet distortion is higher for the Case A30-P (90) than Case A30-P (180) which further reduces the range of fan operation.
influence of another distortion can almost disappear. This is advantageous for the measurements which are planned in NIFTI project.

The upstream and downstream distortions are 180 degrees apart. The upstream distortion generated due to the pylon persists until about sector 8, as opposed to the quicker recovery from sector 2 to 5 in Case A30-P (90). The wider orbit pattern of Case A0-P (180) shows similar behaviour to the sector 4-7, which are driven by the distortion generated by AOA. Therefore, in a sense for this case the effects of the pylon and the AOA on the fan operation is somewhat decoupled and their contribution to fan unsteadiness can be analysed independently. Moreover, it is observed that the overall shape of the orbit for Case A30-P (180) is narrower than that of Case A30. This can be explained with the help of the decoupled analysis. Take sector 5 as an example: sector 5 in Case A30-P (180) experiences slightly higher inlet total pressure and higher incidence when AOA is at 180 degrees, which moves its operating point towards higher mass flow and higher pressure ratio compared to the mean (an effect more or less equivalent to the behaviour of sector 1 for Case A30), thus contributing to the contraction of the overall orbit envelope. Similar conclusions can be drawn for other sectors, which are not elaborated here.

The operating condition of the fan blades is much more widespread for Case A30-P (90). The overall shape of its orbit still resembles similarities with the case with the downstream pylon only (Case A0-P), however the addition of AOA magnifies the orbit circumference substantially. In this case, there is a strong interaction between the upstream and downstream distortions which is highly non-linear. To explain this phenomenon, again take sector 5 (worst condition) as an example: analysis of Figure 12 shows that both the isolated downstream distortion (Case A0-P) and the isolated upstream distortion (Case A30) pushes sector 5 towards stall from the mean, therefore when both distortions are present simultaneously in Case A30-P (90), sector 5 is driven further into stall which contributes to the increase in the overall orbit area. The wider orbit pattern of Case A30-P (90) can also be seen in Figure 15, where the flow separation at sector 2 generated due to the pylon persists until about sector 8, as opposed to the quicker recovery from sector 2 to 5 in Case A30-P (180).

These preliminary results indicate that the for the safest operation of the wind tunnel, the rig should be mounted so that the upstream and downstream distortions are 180 degrees apart. Further work will be carried out to investigate other configurations regarding the relative circumferential position of the large pylon and the AOA distortion (such as AOA at 0 and 270 degrees). The 180 degree installation has the added advantage that the distortions become almost decoupled which is advantageous for the measurements which are planned in NIFTI project.

---

3 “decouple” is used to denote that two distortions do not affect the fan blade simultaneously. When the fan blade is affected by one distortion, the influence of another distortion can almost disappear.

![Figure 15: Total pressure at the stator exit for the Case A30-P(90) and Case A30-P(180)](image)
CONCLUSIONS

In this paper the interaction between upstream and downstream distortions and their impact on fan operation was investigated using CFD. The analysis involved isolated downstream, upstream as well as combined distortion patterns. The available measured data for the case with downstream distortion (although limited) was used to validate the in-house CFD code HADES. The CFD results were in a relatively good agreement with measured data showing similar impact of pylon on fan unsteady operation in terms of total pressure rise. Initially, upstream and downstream distortion patterns were analysed separately, so that the impact of each component of distortion on fan operation can be studied. The results showed that the effects of small pylon can be ignored relative to the large pylon. Moreover, for the present set up, the large pylon has a bigger impact than 30° AOA in terms of fan unsteady operation. In the next phase, the effect of rig installation on the fan operation was studied by considering two cases with different circumferential position of inlet distortion (due to AOA) relative to the large pylon. The results indicate that the interaction between the upstream and the downstream distortions is significantly higher in the case for which the AOA distortion is at 90° relative to the pylon. This interaction results in a larger unsteady loading on the fan with some sectors operating past the stall line. For the case with AOA distortion at 180° angle relative to the large pylon, the upstream and downstream distortions become almost decoupled. It can be concluded that the safest operation of the wind tunnel can be achieved by mounting the rig so that the upstream (AOA or crosswind) and downstream (pylon) distortions are 180° apart. This form installation has the added advantage that the distortions become almost decoupled which is beneficial for the future measurements planned in NIFTI project. From an engine operation viewpoint, the results indicate that crosswind at 90° to the intake can potentially be more harmful than 30° AOA (take-off) when the amplitude and area of generated distortion are of similar scale.

ACKNOWLEDGMENTS

The authors would like to acknowledge the financial support from Clean sky EU under NIFTI project. This project has received funding from the Clean Sky 2 Joint Undertaking (JU) under grant agreement No 864911. They would also like to acknowledge DLR for allowing the use their design geometries of the fan stage. They would also like to take the other members of the consortium Bart van Rooijen from DNW and Dirk Michaelis and Bernhard Wienke from LaVision for valuable discussions.

REFERENCES


