Influence of the limited room space on inlet conditions and fan operation at the rotating test rig

A Sieradzki and B Łukasik
Łukasiewicz Research Network – Institute of Aviation, al. Krakowska 110/114, 02-256 Warsaw, Poland
adam.sieradzki@ilot.lukasiewicz.gov.pl

Abstract. The limited space of the laboratory room allows maintaining the selected ambient air parameters in the required ranges. It can be, however, a source of many problems in the case of rotating test rigs that blow a large amount of air. The aim of the paper is to examine the influence of limited space and its shape on the working conditions of axial fans test stand. The focus was particularly on the conditions prevailing at the inlet to the test rig and their potential impact on the tested fan operation. The article presents the results of CFD simulations for two rooms of different volumes and the results of experimental measurements carried out for the smaller one. Good qualitative agreement between the simulations and the experiment was obtained. The limited space around the test rig causes the unsteadiness of the axial velocity profile and introduces an unsteady swirl at the inlet. This leads, in turn, to local changes in the angle of attack of the fan blades and the resulting consequences related to the off-design working conditions. The paper proves the need to minimise the effects of the induced airflow in the test room and proposes some methods to accomplish this task.

1. Introduction

Experimental research on rotors and other air-breathing turbomachinery, which ingest large amounts of air, is an extremely difficult issue, especially if these are tested in a limited-space laboratory. During normal operation, these machines usually work in open space, often at a certain height, away from other objects which could interfere with airflow. Reconstruction of such conditions at the test rig is almost impossible. Usually, limited space in the test room results in the airflow pattern more or less similar to that generated by typical ceiling fans [1] or corresponding to a phenomenon called the vortex ring. For helicopter rotors, the vortex ring is a known flight state formed after achieving specific helicopter flight conditions (vertical descent at a specific descent rate) or if the rotor hovers over or inside a well-shaped ground object [2–3]. In both cases, it results in reduced rotor thrust, but the latter case is particularly interesting if the operation of any kind of rotor in a confined space is considered. The development and demise of the vortex ring state of the rotor is a well-researched topic using both experimental [4–6] and numerical [7-8] methods. The wall-proximity effect was also indicated by the NASA Ames Research Center during the tests of the full-size UH-60 helicopter rotor in the largest wind tunnel in the world. The hover performance of the rotor changed with a change of its orientation relative to the surrounding walls of the wind tunnel, probably due to different ground-wall effects intensity [9]. Generally speaking, the above observations relate to the case in which the rotor is located close to the centre of the surrounding geometry, so the influence of the environment does not significantly depend on the azimuthal direction. The matter seems to be much more complicated if the surrounding geometry is
more complex or the rotor is placed not in a center point, but e.g. close to one wall or the corner. A similar situation for a helicopter rotor is presented in [10] and could lead to a non-axisymmetric loading of the rotor due to a partial vortex ring and pulling the helicopter toward the side wall.

Observations of the operation of an unducted rotor in a closed room or near side walls are much better described in the literature than those of a ducted fan operating at a test rig. It can be, however, presumed that in certain configurations this type of fan may also be exposed to distorted inflow resulting from induced airflow velocities in the room. On the other hand, the research carried out on the boundary layer ingesting fans have shown that nonuniform inlet conditions can result in a noticeable decrease in fan performance [11-12] and stall margin [13], and may have a negative impact on aeromechanics and fatigue life [14-15]. Therefore, the influence of the phenomena related to the induced air movement near the test stand should be carefully considered when the experiment is prepared. The best way to eliminate such effects is to build a test rig using existing or specially designed wind tunnels [16-17], but this approach significantly increases the cost of the entire experiment.

The aim of the paper is to examine the influence of limited space and its shape on the working conditions of the axial fans test rig. The stand was built as part of a project related to the development of distortion-tolerant fan design methods. The distortion, in the potential application of such a fan, would be caused by the ingestion of a boundary layer from the fuselage of an aircraft. On the test stand, it is introduced by the distortion gauze with non-uniform porosity, located upstream of the rotor. However, this paper focuses only on the pre-tests of the developed test rig to provide information on the issues related to the test rig operating in a limited space area.

1.1. Fan test rig

The test rig was designed to operate in the vertical position (vertical axis of rotation). It is about 5 m high and it consists of a bell mouth inlet, followed by a 1.5 m inlet duct where the desired velocity profile can be obtained with distortion gauzes. The single-stage fan is located right beneath the inlet duct. It is a low-pressure ratio fan with a diameter of 0.58 m. The outlet section is less than 1 m long and is closed by a throttling cone which allows the adjustment of the mass flow of the air flowing through the fan. The tests are carried out without forced external flow. The design speed of the fan is 2500 rpm and the possible mass flow is up to 11 kg/s. The test rig is presented in Figure 1.

Figure 1. Rotating test rig (on the left: without a damping plenum, on the right: with a damping plenum).
1.2. Considered test rig locations
Two different relatively high-volume rooms were taken into consideration to finally choose the place for the test rig – the wind tunnel mockups preparation workshop and the test section of the 5-meter wind tunnel. The volume of the workshop is much smaller than that of the wind tunnel test section room, i.e. 1200 m$^3$ vs 3300 m$^3$, but it was decided to check whether the mockups preparation workshop could be acceptable in terms of flow uniformity due to its high availability (wind tunnel is much busier and could be used only in short test periods).

2. Numerical model
The CFD simulations were performed for two different test rooms, with the assumption of maximum expected air mass flow rate value generated by the test rig. The simplified geometric models of both rooms are shown in Figure 2. The simulations were carried out for several different room configurations – with closed or open doors and with additional flow defectors (only in the case of the wind tunnel test section room). The fan itself was simulated under the pressure jump boundary condition. This approach simplified the mesh greatly and allowed performing the simulations with a much lower computational time. The pressure jump value was adjusted to obtain the desired mass flow rate value with the throttle cone moved fully downstream. The opening of the rooms was carried out by replacing the wall type boundary condition with the pressure outlet boundary condition.

The simulations were based on Unsteady Reynolds Averaged Navier-Stokes (URANS) equations and used one of the most widely recognised industrial standard solvers - ANSYS Fluent. A pressure-based solver with the incompressible flow and k-ω SST turbulence model [18] was used. For spatial discretisation, second-order schemes were used for pressure, momentum, and two turbulence equations. Time integration was performed with the first-order implicit method and time step size of 0.005 seconds. Hybrid meshes of a fully structural hexahedral grid away from walls and a tetrahedral mesh with prism boundary layers near walls were generated using the ANSYS ICEM software. The advantage of such a hybrid mesh with hexa-core is that most of volume is meshed using high-quality perfectly orthogonal hexahedral elements (structural mesh). Preparing this kind of grid is not very time-consuming either because sophisticated geometry shapes are reproduced using unstructured tetrahedral elements generated automatically. The simulations were performed with the mesh of 10.6 million elements for the mockup preparation workshop and 12.1 million elements for the wind tunnel test section room.

Figure 2. Room geometries used in the CFD simulations - mockups preparation workshop (1,200 m$^3$) and wind tunnel test section room (3,300 m$^3$).
3. Experimental measurements
The test stand was equipped with many probes and sensors to measure pressure, flow direction and velocity in the channel, temperature, and other parameters. For this work, some additional measurements were also taken outside the test rig, but only in the smaller mockups preparation workshop. Installing the test rig in the wind tunnel test section room is considered in the future.

![Figure 3. Location of the velocity measurement points and adopted coordinate system in the wind tunnel mockups preparation workshop.](image)

The flow velocity in the room was measured with the KIMO SH100 windmill probe, at the points marked in Figure 3. The measurement was to confirm the occurrence of the specific flow structures identified by the CFD calculations and the possibility of influencing the flow field through various room geometry and conditions modifications. At the selected points, the mentioned flow structures had one dominant direction, which means that the windmill probe should give reasonable results. The average y-velocity values were measured for every point (averaging time – 2 minutes). The uncertainty of the air velocity measurement was 0.1 m/s.

The second measurement was done inside the inlet duct by two five-hole pressure probes placed on opposite azimuths. The probes were mounted on the traverse system capable of changing automatically the position of the probes to measure the swirl angle of the flow at any point of the cross-section located about one channel diameter upstream of the rotor (averaging time for every measurement – 3 seconds). For the purpose of this work, however, only radial sweeps were used. The uncertainty of the swirl angle measurement was 0.1°.

4. Results and discussion

4.1. Mockups preparation workshop (CFD, experiment)
The results of the simulation for the preparation workshop with closed doors are presented in Figure 4. The figure presents a snapshot in time of the unsteady CFD simulations and focuses on flow velocity visualisation at two planes parallel to the floor surface. The first plane is just above the lip of the bell mouth inlet and the second one is where the constant test rig diameter section begins (right beneath the bell mouth). There is a noticeable flow nonuniformity at the inlet of the test rig (second visualisation plane) which is also highly unsteady. The unsteadiness of the flow in this case is due to the air stream coming from one of the corners of the room and the unstable wake of the column near the corner. Opening the door to the wind tunnel room significantly reduced this phenomenon (Figure 5) so a reduction of the flow velocity near the test rig was observed. Notably, however, the swirl of the flow is still visible at the inlet of the test rig, even with the door open.

For both cases, the y-velocity measurements were made based on the previously defined map of measurement points (Figure 3). The results are shown in the point chart (Figure 6). The values presented have been related to the maximum average speed registered for any point, for the case with closed doors. The experimental measurements were averaged over 120 seconds, whereas the CFD results were prepared using 60 seconds averaging time. In this particular case, the highest values of velocity were
measured at point 4 (both in the CFD and in the experiment) so all other velocity values were related to the value from the same point. As a result, dimensionless velocity was obtained and allowed for a qualitative evaluation of the presented results. The basic reason for such a comparison is that at the stage of preliminary tests of the stand the exact mass flow rate of the rig was unknown (but for sure smaller than in the CFD simulations) so the quantitative comparison of the velocities induced in the room did not make sense.

![Figure 4. Wind tunnel mockups preparation workshop (closed) – contours of axial velocity (Vz) and velocity magnitude calculated in the xy plane (Vxy).]

![Figure 5. Wind tunnel mockups preparation workshop (open) – contours of axial velocity (Vz) and velocity magnitude calculated in the xy plane (Vxy).]

The prepared map of the measurement points was developed in such a way to confirm the occurrence of flow patterns identified in the CFD simulation. It can be concluded from Figure 6 that the opening of the wind tunnel door and the entrance gate significantly reduces the flow velocity near the test bench. In the case of the closed room, all points were measured. The character of the changes between points is similar to the computational one, except for points 1 and 2. The experiment shows that the wake of the column is much less noticeable. This may result from a more complicated and dissipated flow in the room, caused by the presence of other elements not included in the CFD model. In the case of the room with open doors and entrance, only three points with the highest velocities (along the identified air stream, points 4, 7, 8) were measured again but for different ambient conditions outside the building. The measurements taken during a calm day showed a decrease of the flow velocity at the points analysed, similar to the results of the CFD simulation. On the other day, a strong wind caused a draft in the room and as a result similar flow velocities as in a closed room. Therefore, opening the room to the outside can theoretically reduce flow velocities but in real life has introduced variable conditions around the test bench so it is not recommended.
Figure 6. Results of the CFD simulations and experimental measurements (non-dimensional averaged flow velocities).

For the closed-room case, the flow swirl angle in the channel upstream of the fan was measured by two five-hole pressure probes placed on opposite azimuths. The experimental data are presented in Figure 7. The probes were traversed to perform measurements at different radial positions. The measurements were carried out twice so two passes of each probe were obtained. Such an approach enabled us to evaluate the steadiness of the flow swirl angle at a given point. It turned out that the values measured during the first pass were significantly different from the second one. This observation confirms the thesis that the limited space and complicated shape of the room surrounding the test rig can result in unsteady flow at the inlet. The measured flow swirl angle values are also similar to those predicted by the CFD simulation. Table 1 presents the range of swirl angle values calculated from the axial (Vz) and tangential (Vt) velocity changes obtained numerically (with assumed baseline axial velocity Vz=35 m/s).

Table 1. Swirl angle due to inlet flow disturbances (based on the CFD data).

| Swirl angle = \( \tan^{-1} \left( \frac{\Delta V_t}{V_z + \Delta V_z} \right) \) | values in | \( \Delta V_z \) [m/s] |
|---------------------------------|----------|---------------------|
| degrees                        |          | -1.0   | -0.5  | 0.0   | 0.5   | 1.0   |
| \( \Delta V_t \) [m/s]         | 0.0      | 0.00   | 0.00  | 0.00  | 0.00  | 0.00  |
| 1.0                            | 1.68     | 1.66   | 1.64  | 1.61  | 1.59  |
| 2.0                            | 3.37     | 3.32   | 3.27  | 3.22  | 3.18  |
| 3.0                            | 5.04     | 4.97   | 4.90  | 4.83  | 4.76  |
| 5.0                            | 8.37     | 8.25   | 8.13  | 8.02  | 7.91  |

Table 2. Rotor incidence angle due to inlet flow disturbances (83% blade span, based on the CFD data).

| Incidence angle = \( \tan^{-1} \left( \frac{\Delta V_t - \Omega \cdot R}{V_z + \Delta V_z} \right) \) | values in | \( \Delta V_z \) [m/s] |
|---------------------------------|----------|---------------------|
| degrees                        |          | -1.0   | -0.5  | 0.0   | 0.5   | 1.0   |
| \( \Delta V_t \) [m/s]         | -5.0     | -1.33  | -1.70  | -2.06  | -2.42  | -2.78  |
| -2.5                            | -0.28    | -0.64  | -1.00  | -1.35  | -1.70  |
| 0.0                             | 0.70     | 0.35   | 0.00  | -0.35  | -0.69  |
| 2.5                             | 1.63     | 1.28   | 0.94  | 0.60  | 0.26  |
| 5.0                             | 2.50     | 2.16   | 1.83  | 1.49  | 1.16  |

Table 1.

Figure 7. Swirl angle measured in the duct (2 probe passes) – without a damping plenum.

Figure 8. Swirl angle measured in the duct (5 probe passes) – with a damping plenum.
4.2. Influence on fan operation

Irregularity of axial velocity at the inlet and presence of variable swirl of flow can significantly affect operating conditions of the fan. Both factors affect the angle of attack of blades (non-zero incidence angle) and so their load, in a similar way as in the case of BLI fans [11,12]. To estimate the range of the changes in the incidence angle of the fan blades, simple calculations were carried out. The analysis was performed for a radius R of 0.24 m (83% of the blade span). The baseline case was assumed to have an axial velocity (Vz) of 35 m/s and a zero tangential velocity (Vt). The results are summarised in Table 2. The negative component of the tangential velocity means the flow swirl towards the fan rotation (co-swirl), whereas the positive one in the opposite direction (counter-swirl). The counter-swirl increases the angle of attack of the blade and its load as opposed to the co-swirl. As presented in Table 2, assuming the axial (ΔVz) and tangential (ΔVt) velocity change ranges obtained from the CFD simulations, in the worst scenario, the incidence angle of the blade can be over 2.5 degrees (positive or negative). Assuming that the maximum angle of attack of the blade is about 15 degrees, it can be concluded that the incidence changes could be relatively large. They can cause variations of the blade load up to about ±15% of the maximum load obtainable under given operating conditions. As a result, the fan can operate in conditions deviating from the design point. This fact may affect its efficiency and other measured parameters, resulting in difficulties in the interpretation of the results of a given experimental test. Variable blade load can result in excessive fatigue wear on selected components such as motor bearings. For these reasons, any undesired flow distortions at the inlet to the test rig should be minimised.

The quality of the inlet stream could be improved, for example, with the flow straightener (honeycomb) in the inlet duct. Honeycombs are used very often in wind tunnel flow preparation sections, mainly to reduce the level of turbulence [19-20]. The swirl component of the flow can also be efficiently reduced with a properly designed flow straightener. Honeycombs can, however, provide a more uniform and axial flow but at the cost of noticeable pressure loss. If a honeycomb is planned, this loss should be taken into account during a test rig design process.

During the further tests of the fan, another method was used to achieve a more stable and uniform inflow to the fan. A rod structure with a stretched-on-top thin partially-permeable fabric was built above the bell mouth inlet, as shown in the second picture of Figure 1. As a result, a cloth plenum attenuating the strongest disturbances was obtained, but at the same time it did not generate large pressure losses due to the low flow velocity in this region. The effectiveness of such a fabric was confirmed by measuring again the swirl angle of the flow in the duct. It turned out that the swirl angles and flow unsteadiness were significantly reduced. Previously, the swirl angles ranged from about -5 to 7 degrees and were significantly variable over time, as presented in Figure 7. On the other hand, Figure 8 shows the results of the measurements with this damping plenum. The range of measured swirl angle was reduced to (-0.7, 2.5) degrees. Using such a damping plenum above the bell mouth inlet reduced swirl variation by up to 75%. The decrease in the maximum achievable mass flow rate and hence the pressure loss, resulting from the installation of the fabric, was negligible. Unlike the straightener, the proposed solution should also mitigate axial flow velocity changes.

4.3. Wind tunnel test room (CFD)

The presentation of this case aimed to show the possibility of changing the flow pattern in the room to achieve better and more uniform conditions at the test rig inlet. The results of the baseline simulation (without flow deflectors) are presented in Figure 9, using the isosurface of velocity magnitude equalled to 2.3 m/s - this value gives the clearest view of the obtained flow pattern. It can be noticed that the stream of the air flowing out of the test rig spills over the floor, and then in the niche of the room changes its direction by 180 degrees and is directed towards the test rig at the level of its inlet. The described flow pattern develops in the part of the room with a lower ceiling, similar to the mockups preparation workshop, even with the door open (marked as the outlet in Figure 9). To reduce the flow velocity and improve its uniformity near the test rig, the two 1.6 m height baffles were mounted in the room to prevent the air from entering the niche in the side sections (near the corners). Such a room configuration and its impact on the flow are shown in Figure 10. The great improvement is noticeable if comparing
Figure 9 and Figure 10. The problematic airstream was eliminated by using baffles that deflect the airflow upwards before it enters the niche.

Another observation from the simulations is the reduction of the average value of the induced flow velocity in the wind tunnel test section room compared to the one in the mockup preparation workshop. However, this is not surprising if almost 3 times larger volume of the wind tunnel test section room is considered.

Figure 9. Wind tunnel test section room (without deflectors) – isosurface of velocity magnitude 2.3 m/s, the colours show the areas of high (red) and low (blue) static pressure, the dashed arrows show the observed airflow pattern.

Figure 10. Wind tunnel test section room (with deflectors) – isosurface of velocity magnitude 2.3 m/s, the colors show the areas of high (red) and low (blue) static pressure, the baffles completely solve the problem.

5. Conclusions
The limited space around the test rig may have a significant impact on its working conditions and results of the measurements. In general, the larger the room for tests is, the lower the probability of unwanted flow phenomena is. This is however not a rule of thumb because a lot depends on a room shape. CFD simulations are a very useful tool in identifying various types of flow structures that can negatively affect the quality of the inlet stream. On the other hand, an accurate reproduction of room geometry with all elements that may have an impact on induced velocity values is a very complicated and time-consuming task. Nevertheless, in the presented case, the CFD simulations based on simplified room models were able to reproduce the most important flow dependencies and inlet conditions on the test bench.

Usually, it is not possible to completely remove all aerodynamic interactions. Opening the room may theoretically reduce these disturbances, however, it is not advisable if it makes the experiment dependent on external weather conditions. It may be a better idea to properly shape the flow inside the room using, e.g. various types of baffles acting as flow deflectors. Their effectiveness was confirmed by the results of CFD simulations.
The problem that appeared in most of the performed simulations was the presence of the flow swirl at the inlet of the test rig. Flow swirl together with changes in axial velocity creates a non-zero incidence angle, which means that the fan does not work under design conditions. Therefore, the influence of the phenomena related to the induced air movement near the test stand should be minimised and taken into account during the preparation phase of an experiment and post-processing of results.

The most popular way to reduce the swirl component of the flow is to use a proper flow straightener (honeycomb) in front of a fan, at the expense of larger pressure losses. However, another method, i.e. mounting a damping plenum at the inlet section of the test rig inlet to reduce the impact of the external flow nonuniformity was proposed. Its high efficiency without a significant pressure loss penalty was confirmed experimentally.

References
[1] Bassiouney R and Korah N S 2011 Studying the features of air flow induced by a room ceiling-fan Energ. Buildings 43 1913–8
[2] Łusiak T and Dziubiński A 2007 Specific case of interference between a helicopter and surroundings: hovering flight over a well-shaped object Trans. Inst. Aviation 191 43–50
[3] Łusiak T, Dziubiński A and Szumanski K 2009 Interference between helicopter and its surroundings, experimental and numerical analysis TASK Quart. 13 379–92
[4] Green R B, Gillies E A and Brown R E 2005 The flow field around a rotor in axial descent J. Fluid Mech. 534 237–61
[5] Stryczewicz W and Surracze K 2014 Badania eksperymentalne stanu pierścienia wirowego na wimniku nośnym śmigłowca metodą anemometrii obrazowej (PIV) Trans. Inst. Aviation 235 17–27
[6] Surracze K, Ruchala P and Stryczewicz W 2015 Wind tunnel tests of the development and demise of vortex ring state of the rotor Adv. in Mech.: Theoret., Comp. and Interdiscip. Issues - Proc. of the 3rd Polish Cong. of Mech. (PCM) and 21st Int. Conf. on Comp. Meth. in Mech. (CMM) (Gdańsk: CRC Press) 8–11
[7] Stalewski W and Surracze K 2019 Investigations of the vortex ring state on a helicopter main rotor based on computational methodology using URANS solver MATEC Web Conf. 304
[8] Wang L, Xu G and Shi Y 2018 High-resolution simulation for rotorcraft aerodynamics in hovering and vertical descending flight using a hybrid method Chinese J. Aeronaut. 31 1053–65
[9] Shinoda P M, Yeo H and Norman T R 2004 Rotor performance of a UH-60 rotor system in the NASA AMES 80- by 120-foot wind tunnel J. Am. Helicopter. Soc. 49 401–13
[10] Dziubiński A 2016 CFD analysis of rotor wake influence on rooftop helipad operations safety Trans. Inst. Aviation 242 7–22
[11] Gunn E J and Hall C A 2014 Aerodynamics of boundary layer ingesting fans Proc. of the ASME Turbo Expo 2014: Turbine Tech. Conf. and Exp. (Düsseldorf, Germany: ASME)
[12] Sieradzki A, Kwiatkowski T, Turner M and Łukasik B 2020 Numerical modeling and design challenges of boundary layer ingesting fans Proc. of the ASME Turbo Expo 2020: Turbomach. Tech. Conf. and Exp. (ASME)
[13] Perovic D, Hall C A and Gunn E J 2019 Stall inception in a boundary layer ingesting fan J. Turbomach. 141
[14] Bakhle M, Reddy T S R, Herrick G, Shabbir A and Florea R 2012 Aeromechanics analysis of a boundary layer ingesting fan Proc. of 48th AIAA/ASME/SAE/ASEE Joint Prop. Conf. & Exhib. (Atlanta, GA: AIAA)
[15] Bakhle M A, Reddy T S and Coroneos R M 2014 Forced response analysis of a fan with boundary layer inlet distortion Proc. of 50th AIAA/ASME/SAE/ASEE Joint Prop. Conf. (Cleveland, OH: AIAA)
[16] Florea R V, Voytovych D, Tillman G, Stucky M, Shabbir A, Sharma O and Arend D J 2013 Aerodynamic analysis of a boundary-layer-ingesting distortion-tolerant fan Proc. of ASME Turbo Expo 2013: Turbine Tech. Conf. and Exp. (San Antonio, TX: ASME)
[17] Arend D J, Wolter J D, Hirt S M, Provenza A, Gazzaniga J A, Cousins W T, Hardin L W and Sharma O 2017 Experimental evaluation of an embedded boundary layer ingesting propulsor for highly efficient subsonic cruise aircraft Proc. of 53rd AIAA/SAE/ASEE Joint Prop. Conf. (Atlanta, GA: AIAA)

[18] Wilcox D C 2006 Turbulence modeling for CFD (La Cânada, Calif: DCW Industries)

[19] Mehta R D and Bradshaw P 1979 Design rules for small low speed wind tunnels Aeronaut. J. 83 443–53

[20] Scheiman J 1981 Considerations for the installation of honeycomb and screens to reduce wind-tunnel turbulence (Langley Research Center Hampton, VA: NASA)