AERODYNAMIC DESIGN AND ITS VALIDATION OF AN OPEN-TYPE ENGINE RUN-UP FACILITY

Masafumi Kawai*, Kiyoyuki Nagai*, Shigeru Aso**
***IHI Corporation, Japan, **Kyushu University, Japan**

Keywords: *Aircraft, Run-up Facility, Aerodynamic Characteristics, CFD, Wind Tunnel*

Abstract

The aerodynamic characteristics of an open-type engine run-up facility (RUF) have been studied by wind tunnel experiment and computational fluid dynamics to provide a new design procedure. Then the validation test has been performed for the actual RUF constructed based on the new procedure. All the validations and commercial operations went successfully.

1 Introduction

Aircraft are required to be tested prior to flight when its engines are maintained in an airport. The test is usually done in an exhaust gas diffusing device, called a blast fence. Since the blast fence has poor noise suppression performance, when noise problems take place in an airport, a noise suppression facility, or run-up facility (RUF) is used. RUFs are roughly categorized into two kinds: closed and open [1] (intermediate types [2][3] also exist). The former has better noise suppression ability, but it is difficult to establish good aerodynamic characteristics. The latter, with a typical one shown in Fig. 1, surrounds the aircraft by walls of suitable height, area and material. With this type it is rather easy to directly take the atmosphere into the engine. However, since engines are designed for open-air conditions, aerodynamic problems may still arise in the open-type RUF.

Among the existing RUFs, there are several implementations of closed type (hush house). On the other hand, there are only a few examples of large scale open-type RUFs which surround the entire commercial airliner.[1] There were some cases for closed-type RUFs in which it had been difficult to establish good enough aerodynamic performance. Hence the development of the procedure for aerodynamic design is needed

Fig. 1. Open-type RUF

Is this paper, the aerodynamic characteristics of an open-type RUF, as shown in Fig. 1, have been studied by wind tunnel experiment and computational fluid dynamics (CFD) to provide a new design procedure. The target of the aerodynamic performance was set as “operability nearly equal to open air.” Then the validation tests were performed for the actual RUF constructed based on the new design procedure.

2 Conditions Required for the RUF

2.1 Acoustical Design

The main object of a RUF is to reduce the noise generated in engine operation to the required level; the aerodynamic performance must be studied after the noise suppression performance

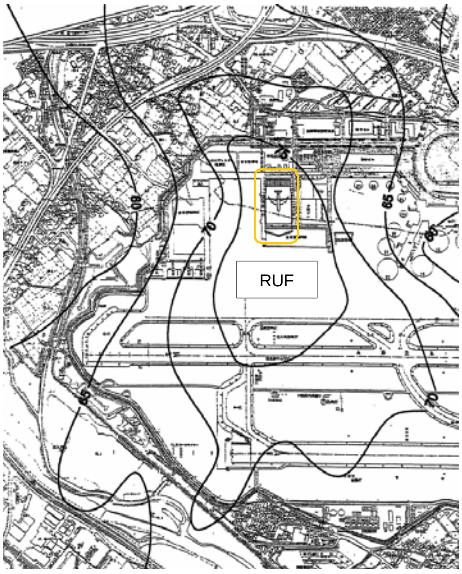


Fig. 2. Example of Noise Contour

has been satisfied. Prior to the aerodynamic design, acoustical design using noise propagation simulation under various conditions (different types of aircraft/engine, engine operation modes, and weather) was performed to find out the necessary RUF height to satisfy the noise suppression requirement. Before the simulation, validation was carried out for the existing blast fence configuration. The result of the simulation is expressed by a noise contour as the example in Fig. 2 shows. From the results, the height of the RUF has been determined to be 20 meters for the side walls and the back wall. As for the front part, considering the aerodynamic performance, the height should be as low as possible. With a trade-off with the propagation of the engine noise to the front direction, the height of 5 meters was chosen.

2.2 Engine Intake Flow Requirement

Jet engines are designed with the air entering directly from the front. If the flow has a certain angle to the engine axis, total pressure at the inlet plane loses uniformity and the engine performance deteriorates. In extreme cases, an engine stall may take place. The aerodynamic performance of RUF can be said that how it can supply the intake flow with enough rate and good quality under various conditions. Here the condition of the intake flow required from engine has been determined.

The change of the intake performance was studied by a basic experiment, with transverse flow taking place on a circular intake simulating the engine. Figure 3 shows the schematics of the experiment. The intake was modeled by placing a lip with the same shape as an actual engine at the intake of a circular duct. Intake flow is generated by a blower. The whole setup is placed in a wind tunnel, and the flow angle is given by setting the intake with an angle to the wind tunnel flow. Intake pressure loss was measured as the index of the effect of the flow angle to the intake. Velocity distribution was also measured by a split film sensor.

A typical result is shown in Fig. 4. The test showed that the duct intake pressure loss increases when the angle of the transverse flow exceeds 45 degrees, whereas no influence is found below 22.5 degrees. An existing study [4] shows that when the flow angle is gradually increased from zero, flow separates at approximately 40 degrees after keeping attached. The increase of pressure loss after 45 degrees in

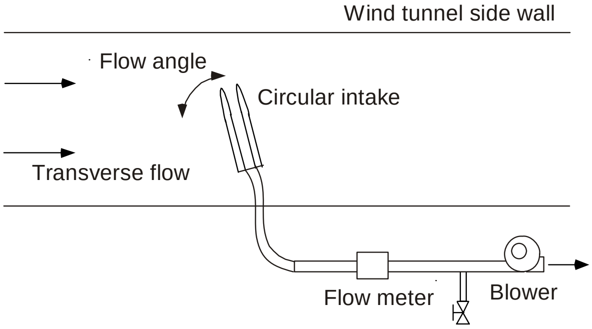


Fig. 3. Schematics of the Intake Performance Experiment

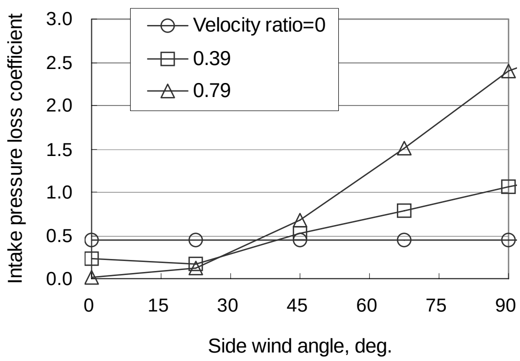


Fig. 4. Relationship between the Flow Angle and the Inlet Pressure Loss

current experiments show that clear separation occurred. Velocity measurements support this consideration. On the other hand, the operation manual of the target engine specifies the limit of engine stall to be 24 degrees, which is close to the range of no influence in our experiment. This angle is determined under steady and low turbulence flow, whereas the flow inside RUF is turbulent due to atmospheric wind condition. Under these considerations, and to set the requirement to the conservative side, the limit of the intake flow angle has been set as 24 degrees for instantaneous flow (each component of the velocity vector to be the sum of average and twice the RMS value of fluctuation).

Another criterion is given by the aircraft maintenance manual, which limits the wind direction and speed for engine operation in the open air (operation envelope). In RUF, however, it is not open and local variations of wind speed and direction are large. So we must specify the reference position to apply this criterion. In this study, the position 4*D* (*D*: engine intake diameter) in front of engine was chosen as the position which is well away from intake and can represent the flow into the engine.

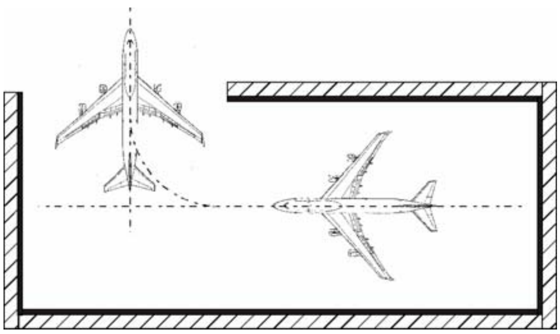
From the considerations so far, the requirements for the engine intake flow has been summarized as follows:

1) Flow angle at the engine intake must be smaller than 24 degrees for the fluctuating value.
2) Flow speed and direction at 4*D* from engine intake must satisfy the engine operation envelope.
3) No unsteady vortices. Continuous supply of enough intake flow.

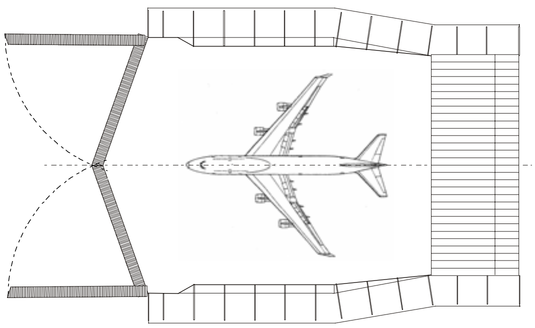
3 Aerodynamic Design

3.1 Base Plan Shape

To enable the handling of the aircraft, two configurations of the open-type RUF can be considered. One consists of fixed walls only and has an open section for aircraft movement. The

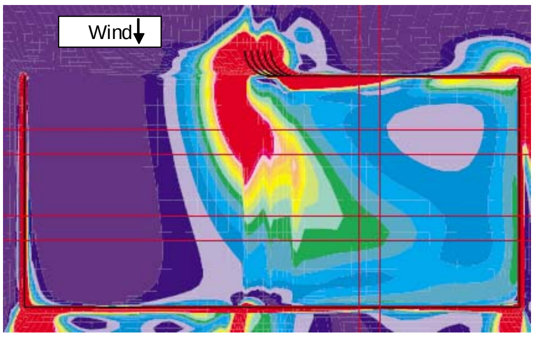


a) Fixed Wall Type

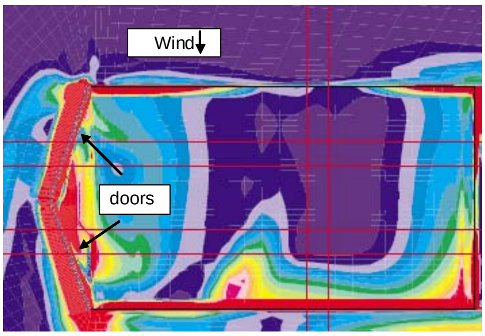


b) Door Type

Fig. 5. Schematics of the RUF Configurations



a) Fixed Wall Type



b) Door Type

Fig. 6. Vorticity Distribution inside the RUF under Side Wind

other has a movable door through which the aircraft goes in and out. Schematics of these configurations are shown in Fig. 5. Both have merits and demerits from cost and handling standpoints.

However, as for the aerodynamic characteristics, they have a clear difference. The fixed wall type has a wall edge at the opening on the side. Under side wind, large separation vortices may be generated from the edge and reach the engine intake. Door type can avoid this vortex generation by gradually changing the wall height. Under this estimation of the flow field, CFD was applied to the flow field inside these configurations. The CFD method will be explained later. The vorticity distribution under side wind shown in Fig. 6 is the result which most clearly shows the difference. In the fixed type, strong vortices generated at the end of the wall, which is necessary for the opening, reached the area of engines, which are shown by the intersecting points of the lines. On the contrary, the door type has no significant vortex generation. From this result the door type has been chosen. To enhance the aerodynamic performance and satisfy the noise suppression performance at the same time, the door is constructed of noise splitters.

As for the width of RUF, a minimum dimension was adopted considering the assumed aircraft and prescribed wing tip clearance.

3.2 Flow near the Intake

As the plan shape and height of the RUF have been determined, in order to determine the length of RUF, the relationship between the length and the aerodynamic characteristics, which is the flow near the intake, was studied through wind tunnel experiments.

The similarity of the experiment can be evaluated by the Reynolds number, and flow rate and momentum ratios of engine inlet/outlet flows to the wind. Reynolds numbers based on the width of RUF and the actual wind speed of 10 m/s are 5.3×10^7 for the actual RUF and 1.6×10^5 for the experiment. These are well within the turbulence region. However the flow through the slit of the door in the experiment will be only 400, so several schemes were taken to maintain similarity. Although the exhaust gas from the engine has a higher temperature than atmosphere, the Froude number based on

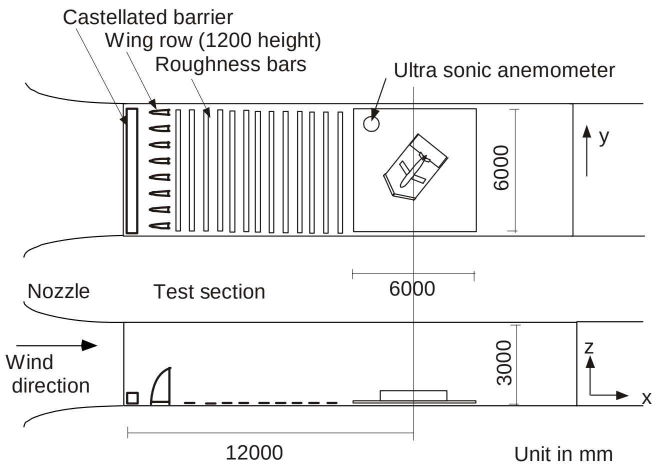


Fig. 7. Layout of the Wind Tunnel Experiment

Fig. 8. Turbulent Boundary Layer Generator and the Model in the Wind Tunnel

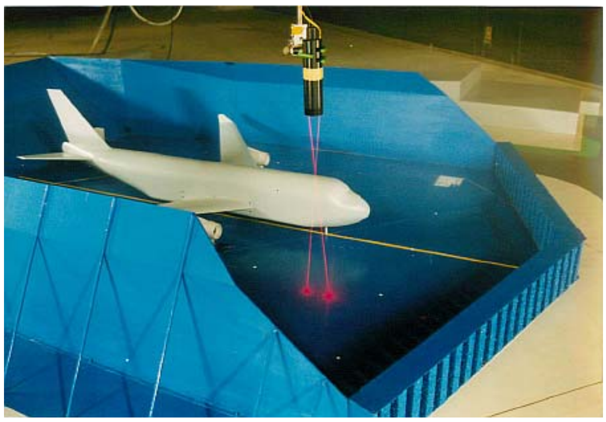


Fig. 9. Wind Tunnel Experiment Model and LDV Measurement

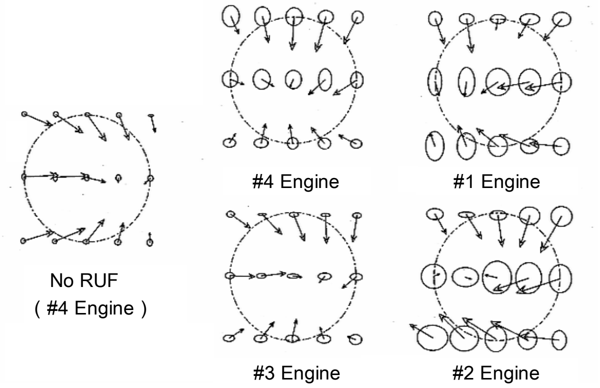


Fig. 10. Intake Flow for Each Engine under Skew Wind

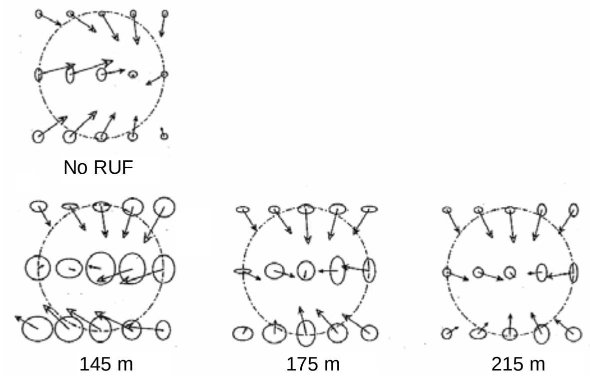


Fig. 11. Variation of #2 Engine Intake Flow According to RUF Length

exhaust gas inertia force and buoyancy force is very small, giving a rise angle of below 0.1 degree due to high exhaust speed. Hence no heating of the exhaust gas was made. The exhaust gas was ejected as a single jet with an equivalent momentum ratio to the wind coming out of a hole with the area equal to total of fan and core.

The experiment was performed in the IHI Atmospheric Wind Tunnel, with the dimensions of the test section as 24 m x 6 m x 3 m (length x width x height). Figure 7 shows the layout of the experiment. The length scale of the experiment is 1/50, and a 270 m x 270 m region around the RUF was taken, reproducing large buildings in the upstream position. Boeing 747 was the aircraft modeled. A turbulent boundary layer generator [5] was used. Figure 8 shows the boundary layer generator and the model inside the wind tunnel. The specifications of the CF6-80C2 jet engine were adopted. Two blowers, each controlling four (number of engines) individual flow lines, were used to generate the intake flow and exhaust flow, reproducing proper flow rate and momentum. The measurements were made mainly by LDV, as shown in Fig. 9.

From the experiment, it was found that the conditions giving the worst intake flow was the #2 engine under skew wind. Figure 10 shows the flow at the plane *D*/2 from the intake under skew wind. The case with no RUF is also shown for reference. Vectors are average flows and ellipses express the turbulence intensity and direction. The #2 engine has the most deviated vectors and high turbulence intensity. So flow visualization by smoke was made for the flow into the #2 engine. It was understood that the flow directly hitting the downstream wall after passing over the door approaches the #2 engine with a significant angle due to the influence of exhaust jet. This means the length between the hitting point, which is fixed relatively to the door, and the engine is important.

Consequently the effect of the length of the RUF to engine intake flow was examined. Figure 11 shows the variation of the #2 engine intake flow according to the length of the RUF. As the length of the RUF increases, the average flow into the engine intake converges and the turbulence decreases. The relationship of the length of the RUF and the intake flow angle at the #2 engine for the worst flow conditions are summarized in Fig. 12. Considering criterion 1) given in previous section, the length of the RUF that has acceptable aerodynamic performance has been determined to be 185 meters.

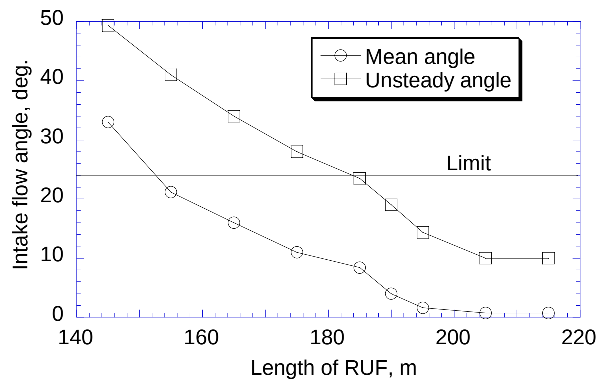


Fig. 12. Relationship between RUF Length and Intake Flow Angle

3.3 Flow Field inside the RUF

The plan shape and dimension of RUF have been determined so far. Finally, the flow field inside the RUF under various wind conditions was evaluated by CFD. The main conditions of the calculation are as follows:

1) Primary CFD method: CFD code Star-CD, steady, considering temperature and density change, finite-volume method, SIMPLE algorithm, standard k-ε turbulence model.

2) Calculation region and number of total grid points: 1.3 million points for 1200 m x 900 m x 510 m (streamwise x lateral x vertical) region, utilizing discontinuous grid to avoid excessive grid points while maintaining enough grid density around the aircraft.

3) Boundary conditions for the engine: Given velocity and extrapolated temperature for intake. Given velocity and temperature for separated core and fan, respectively.

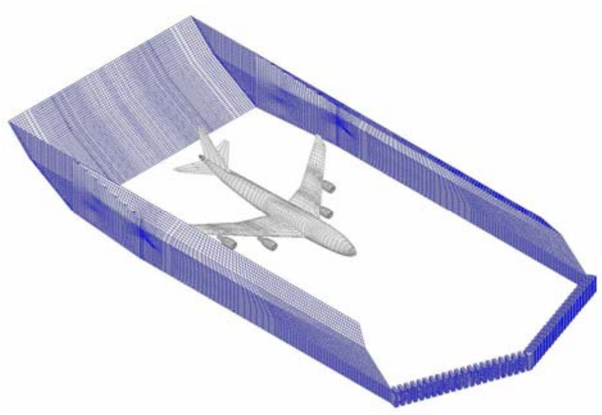


Fig. 13. Computational Grid for the Aircraft and RUF

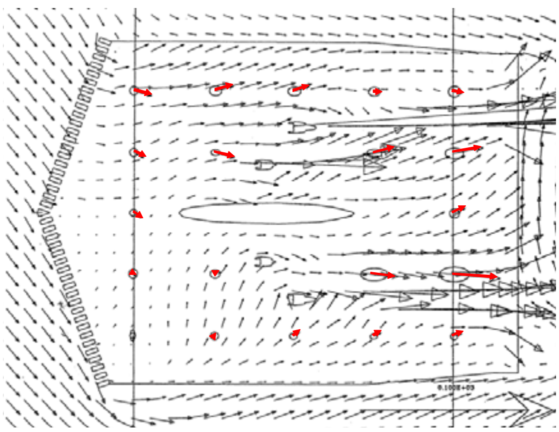


Fig. 14. Comparison of CFD Results to Wind Tunnel Experiments (red vectors)

Figure 13 shows the computational grid for the aircraft and RUF. Approximately one thousand iterations were necessary for converged solution. A comparison of the CFD results to the wind tunnel experiment is shown in Fig. 14. Both results show agreement, and the ability of CFD to evaluate the flow field inside the RUF has been verified. Even in the most severe cases, CFD results showed that the flow field inside the RUF is smooth and no influence from the front door takes place. Thus the adequacy of the final design of the RUF determined through noise propagation simulation, plan shape calculation, and wind tunnel experiment has been confirmed.

3.4 Estimation of the Operation Envelope

The maximum wind speed satisfying the criteria defined in section 2.2 for all wind directions was determined from the intake flow results of wind tunnel experiment. This limit gives the operation envelope of the current RUF. Figure 15 shows the envelope together with the limit given by the manuals of the B-747 and the B-767 aircraft. Considering the wind environment of the assumed airport, the operation frequency has been calculated as +0.3 % for the B-747 and -2.1 % for the B-767 compared to ideal (open air) condition. These figures can be judged to satisfy the target which was specified at the start of this study.

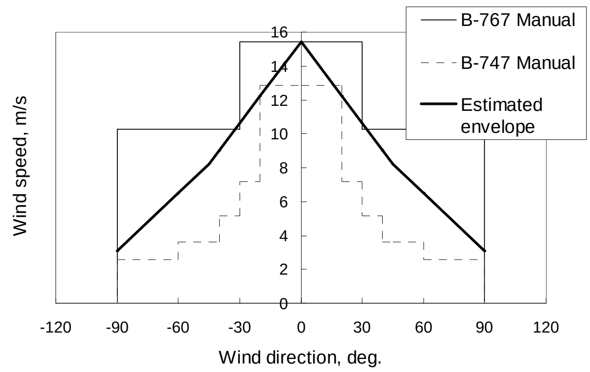


Fig. 15. Estimated Operation Envelope

4 Validation on the Actual RUF

4.1 Measurement of the Flow inside the RUF

Based on the new aerodynamic design procedure as developed in the previous chapters, an actual RUF, shown in Fig. 16, has been constructed in Osaka International Airport. To validate the effectiveness of the present design procedure, the following measurements and survey have been performed.

First, the flow inside the RUF under various wind conditions was measured. The data from the base anemovane placed at the tip of the door at a height of ten meters, displayed on the information board on the wall of the RUF for monitoring from the cockpit, were video-recorded and taken to be the atmospheric condition. Nine anemovanes at a height of 1.5 meters and with a sampling period of one second were used in the measurements without the aircraft. Ten of them were used for the measurements with an Airbus A300 aircraft (not operating). The layout in the latter case is shown in Fig. 17.

A data mining technique was developed to acquire data series corresponding to a one-minute steady wind state, as the data was compared to both wind tunnel experiment and CFD results, which both assumed steady flow. The detailed procedure is as follows:

1) Extract a two-minute steady series from the reference (base anemovane) data. The series was regarded to be steady when it satisfied the following conditions: (a) Same wind direction continuously for more than two minutes, with 'same' defined as a variation in wind direction within 22.5 degrees (one sixteenth of the whole wind direction). (b) Mean wind speed in the series is larger than five knots (2.6 m/s), since there is no constraint for engine operation below this speed. (c) The series does not overlap with a series already extracted.

2) This two-minute series means that the wind at the base anemovane was steady in that period. However due to the distance from the base anemovane to the engine position, the steadiness requires time to propagate through the whole RUF. This time was calculated as one minute. Therefore the first half of the two-minute series is cut and the latter half is taken, with the average wind speed and wind direction being extracted to be the steady measurement data.

Fig. 16. RUF Constructed Based on the New Procedure

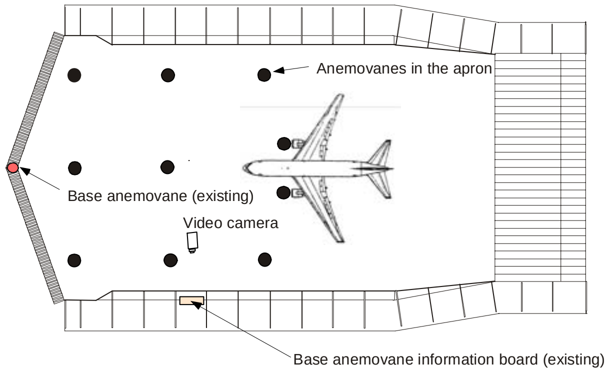


Fig. 17. Layout of the Flow Measurement in RUF

Figure 18 is an example of the reference wind raw data and extraction process. Using this data mining, from 26 hours of data from 15 measurements, 280 sets of effective data were obtained.

Figure 19 shows a representative case, where the wind condition coincided with that of a CFD case. The measured vectors agree well with the CFD results, which shows again that the accuracy of CFD was acceptable. In both configurations with and without aircraft in the RUF, the flow field inside the RUF was smooth as predicted in CFD. Flow speed and direction in front of the engines were below the limits, and no specific vortices were found. Some data were obtained for the wind from the back of the RUF and still no specific vortices were found, which means there is no potential aerodynamic problem in the RUF.

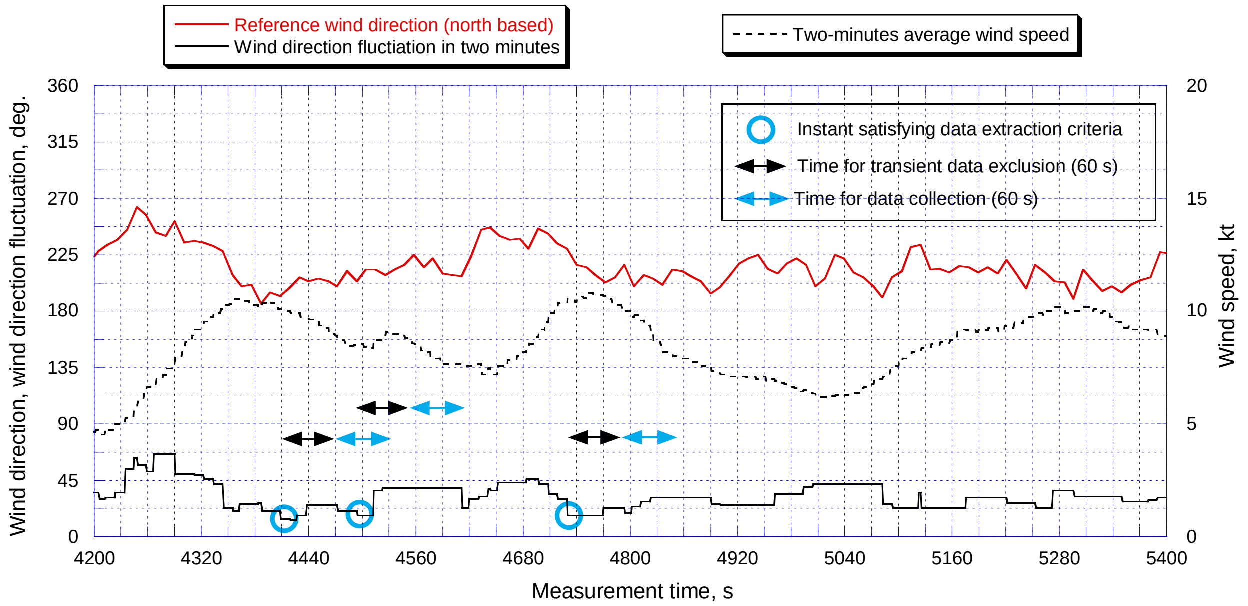


Fig. 18. Example of the Reference Wind Raw Data and Extraction Process

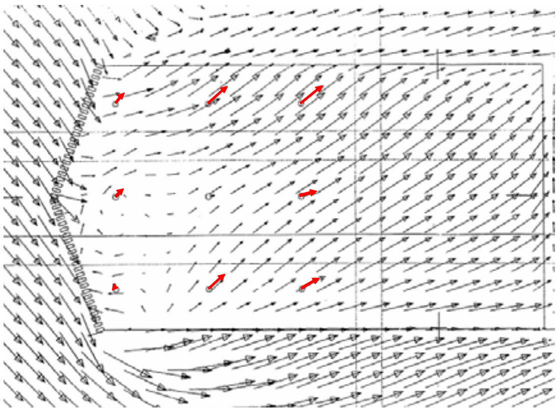


Fig. 19. Comparison of Measured Flow (red vectors) with CFD Result

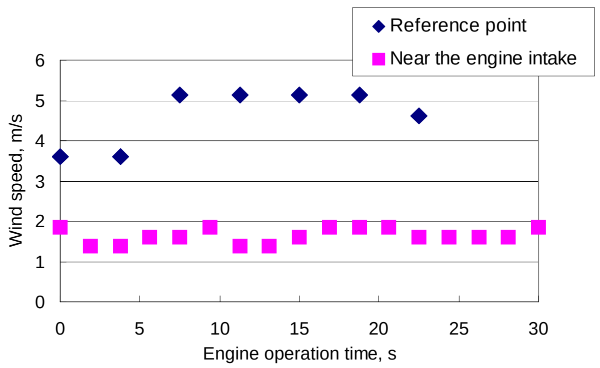


Fig. 20. Wind Speeds at the Reference Point and near the Engine Intake

4.3 Engine Operation

As the aerodynamic characteristics of the RUF have been confirmed to be fair from the flow measurements, engine operations were performed with noise measurements carried out simultaneously. Flow measurement was done again to understand the detailed flow field in the RUF. Figure 1 shown before is a scene from preparation. All the engine operations up to takeoff power (eight times for four-engined aircraft, and thirteen times for twin-engined) went successfully. From the flow measurement it was found that when the reference wind direction has significant angle to the engine axis, the flow in front of the engine has lower speed and the angle is reduced. Typical data is shown in Fig. 20. This tendency had also been found in the experiment, which shows the preferable aerodynamic characteristics of the current RUF.

4.4 Survey under Commercial Operation

After a series of tests, the commercial operation of the RUF has been started. To further understand the aerodynamic characteristics of the RUF, a customer survey was made, from which the direction and speed of the wind and the engine operation conditions have been accumulated. Figure 21 is the summary. In nine

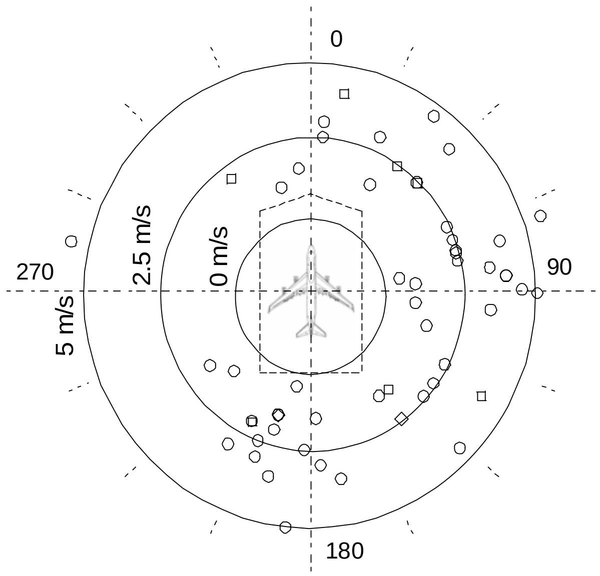


Fig. 21. Wind Data of Commercial Operations

months of operation, 85 engines tests have been completed. No aerodynamic trouble was reported, so all the plots represent the wind conditions for successful engine operations. In addition to the operations corresponding to the estimated envelope shown in Fig. 15, a considerable number of test cases with reverse flow winds can be seen, which was not considered in the design procedure. This shows the aerodynamic robustness of this RUF.

5 Conclusion

An aerodynamic design procedure has been developed for a large scale open-type engine run-up facility (RUF). The procedure consists of the following steps:

1) Acoustical design by noise propagation simulation and determination of flow quality requirements

2) Aerodynamic design through wind tunnel experiments and CFD

3) Validation on an actual RUF including flow measurements, engine operations, and an operational data survey

The RUF constructed in Osaka, based on the new design procedure, is steadily operating to the present and no aerodynamic trouble has yet been reported. Thus the new aerodynamic design procedure can be said to be successfully developed and validated.

Acknowledgments

The design and validation works have been performed under the contracts from Osaka Civil Aviation Bureau, Ministry of Land, Infrastructure and Transport of Japan. The first two authors would like to express their acknowledgments. They also thank INC Engineering Co., Ltd. for the cooperation throughout the whole project.

References

[1] Barret D E. Acoustical Acceptance Testing of Portland (Oregon) International Airport Ground Run-Up Enclosure, *Transportation Research Record 1859*, Paper No. 03-2544, pp 38-44, 2003.

[2] Gooden J H M and Hoelmer W. Aerodynamic Development of a Ground Run-Up Enclosure for Propeller Transport Aircraft, *Journal of Aircraft*, Vol. 43, No. 3, pp 692-699, 2006.

[3] Schuller W M and van der Ploeg F D. Technical Note: Free Field Sound Reflection Measurements on Screens Used for Aircraft Ground Run-Up Noise, *Noise Control Engineering Journal*, Vol. 37, No. 1, pp 13-17, 1991.

[4] Hall C A and Hynes T P. Measurements of Intake Separation Hysteresis in a Model Fan and Nacelle Rig, AIAA-2002-3772, AIAA, 2002.

[5] Counihan J. An Improved Method of Simulating an Atmospheric Boundary Layer in a Wind Tunnel, *Atmospheric Environment*, Vol. 3, pp 197-214, 1969.

Copyright Statement