

Aerodynamic Technologies for High-efficiency and High Specific-flow-rate Fan and Centrifugal Compressor

Hisato TANAKA* Mineyasu OANA*

ABSTRACT

This paper describes the development of aerodynamic technologies for a high-efficiency and high specific-flow-rate fan and centrifugal compressor during the preceding research for the HF120 turbofan engine in order to achieve low specific fuel consumption and high thrust-weight ratio.

For the fan, a swept fan blade design was selected to decrease the pressure loss from shock waves. Computational Fluid Dynamics analysis provided an estimate of the decreased pressure loss from shock waves at swept fan leading edge and flow passage, and the efficiency enhancement throughout the span. Small-scale rig testing indicated that the newly designed fan improves efficiency by 1.5% or more and inlet specific flow rate by 2% in comparison with the fan of the HF118 turbofan engine, in three typical operating conditions: takeoff, climb and cruise.

For the centrifugal compressor with increased aerodynamic loading from higher exit specific flow rate, a new blade configuration was designed to raise aerodynamic loading at the forward area of the blade where the boundary layer is thinner, followed by a confirmation of high efficiency via Computational Fluid Dynamics analysis. Small-scale rig testing indicated enhancement of exit specific flow rate by 7% and efficiency by 0.8% in cruise condition in comparison with the centrifugal compressor of the HF118 turbofan engine. The aerodynamic technologies described in this paper have been incorporated into the fan and centrifugal compressor of the HF120 turbofan engine.

1. Introduction

In 1986, Honda initiated fundamental research and development of a small-sized aircraft jet engine. In 2003, Honda completed stand-alone development of the HF118 turbofan engine (HF118)⁽¹⁾. In 2006, Honda and General Electric Company (GE) of the US initiated joint development of the HF120 turbofan engine (HF120), successor to the HF118. In 2013, the United States Federal Aviation Administration awarded Type Certificate to the HF120.

Figure 1 shows HF120 engine features. The HF120 aims for lower specific fuel consumption and a higher thrust-weight ratio than those of the HF118.

To achieve low specific fuel consumption with low fuel flow rate and high thrust, enhancement of efficiency of aerodynamic parts is one of the key elements. Also, enhancement of specific flow rate by decreasing the frontal projected area of aerodynamic parts and increasing the air flow rate is important to achieving high thrust-weight ratio with less engine weight and high thrust.

Most HF120 thrust is proportional to the flow rate of the fan located at the front region of engine. Most of the HF120's total pressure ratio is produced by the centrifugal

compressor. Thus, the fan and the centrifugal compressor have crucial roles among the aerodynamic parts of this engine.

This paper describes the aerodynamic technology of the design concept, the performance prediction method with computational fluid dynamics (CFD), and performance

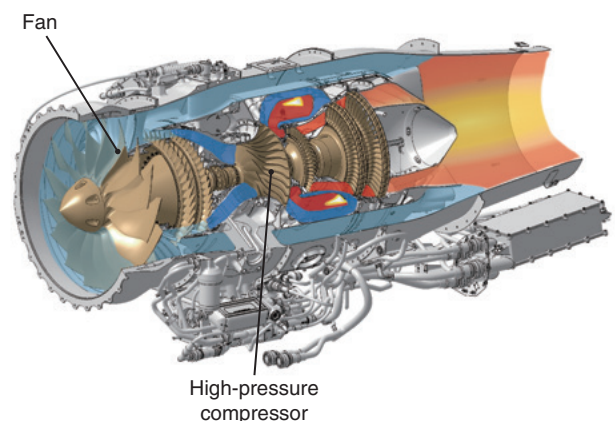


Fig. 1 HF120 turbo fan engine features

* Aircraft Engine R&D Center

substantiation testing by small-scale rig, enabling a high-efficiency and high specific-flow-rate fan and centrifugal compressor.

Honda is responsible for aerodynamic design of the fan and centrifugal compressor in the HF120 engine, and the technology described in this paper is incorporated into engine design.

2. Development Target

The HF120 concept was to enhance specific fuel consumption and thrust-weight ratio over those of the HF118. Numerical targets to preceding development of the fan and centrifugal compressor were to enhance efficiency by 1.5% and 0.5%, respectively, and 2% and 6%, respectively, for specific flow rate.

For specific flow rate calculation, frontal projected area is defined using outer diameter, which greatly affects engine weight. For the fan, inlet specific flow rate is defined using the inlet area. For the centrifugal compressor, exit specific flow rate is defined using the exit area. The HF118 engine's outer diameter at the axial location of the centrifugal compressor is larger than the engine inlet's outer diameter, so reducing the centrifugal compressor's exit diameter is crucial to enhancing thrust-weight ratio (Fig. 2).

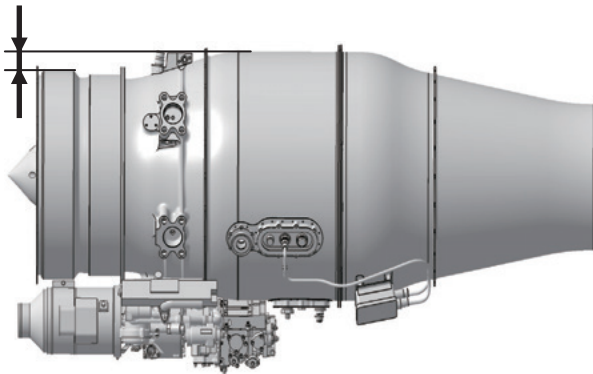


Fig. 2 Engine side view of HF118

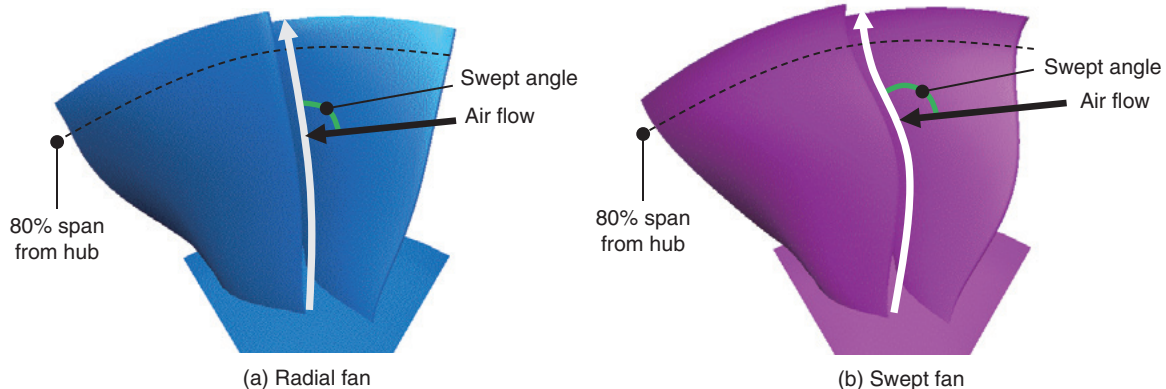


Fig. 3 Front view of fan rotor

3. Aerodynamic Technology – Fan

3.1. Fan Design Concept

Conventional fan rotor design placed emphasis on controlling centrifugal stress of rotating blades by designing the blade sections at various radial locations and stacking them in the radial direction by matching the position of their center of gravity. Figure 3(a) is a front view of the HF118 radial fan, designed using this conventional method.

The relative velocity of inlet air to the fan rotor is supersonic. The shock waves generated at the fan rotor can cause pressure loss. Shock waves whose angles against air flow are 90 degrees are the strongest and create the maximum pressure loss and large loss of efficiency. The swept angle, defined as the angle between inlet air and the leading edge of the fan rotor, of the radial fan rotor is almost 90 degrees from hub to tip, and there is a large pressure loss on the leading edge of the fan rotor.

To decrease this large pressure loss on the leading edge of the fan rotor, a swept fan with various swept angles apart from 90 degrees from hub to tip was designed. Figure 3(b) shows the configuration of this swept fan. Unlike the radial fan, the positions of the center of gravity of each blade section are not always matched, but combining with studies of blade shape and thickness at each section enabled the resulting centrifugal stress to be within permissible limits.

3.2. Fan Performance Prediction

Performance prediction was produced with our in-house CFD analysis tool on the swept fan rotor⁽²⁾.

Figure 4 shows analysis results of the radial fan and the swept fan at the cruise condition that is one of the typical operating conditions. These are relative Mach number contours on a cylindrical surface at 80% span from the hub, which are shown by the dashed lines in Fig. 3. The contour line density of the swept fan at the shock wave points in the green circle near the leading edge and the blue circle in the flow passage is lower than that of the radial fan. This result indicates that the shock waves in these areas are weakened and the pressure loss is reduced.

Figure 5 shows span-wise efficiency distribution in the cruise condition. Efficiency of the swept fan at 80% span point increased by 2% due to the weakened shock wave described in Fig. 4. There is no large difference between the efficiency of the radial fan and the swept fan at 90% and greater span area because of leakage flow at the tip gap. Efficiency is enhanced as the design intention in a broad area (0% to 80% span).

Calculated efficiency by integrating efficiency distribution and multiplying correlation factor between CFD analysis and testing shows that the fan efficiency is enhanced by around 1.5%, which means the target efficiency is achievable.

Additionally, specific flow rate is expected to be enhanced by 2%, making the target achievable, since not only pressure loss is reduced by a weakened shock wave at the swept fan, but also the development of the boundary layer behind the shock wave is decreased.

3.3. Fan Performance Substantiation

After fan performance prediction with CFD analysis, aerodynamic performance tests are done using a small-scale prototype to reduce engineering time on manufacturing as well as time up to substantiation testing.

Special consideration was made on processing accuracy of the prototype, because the accuracy of the small-scale prototype is tighter than that of the full-scale model by scale ratio. The geometry accuracy of the prototype fan blades was verified to satisfy the aerodynamic requirement through evaluation with a 3D coordinate measuring machine.

Measurement accuracy was secured by the combination of pressure and temperature measurement at multiple points

on both circumferential and span-wise direction at fan exit and other measurement methods.

The small-scale swept fan prototype is shown in Fig. 6, and the small-scale rig test facility is shown in Fig. 7.

Figure 8 shows a small-scale rig test efficiency map of the radial fan and the swept fan in three typical operating conditions: takeoff, climb, and cruise. This map indicates that the peak efficiency of the swept fan is increased by 1.5% or more at every operating condition, which means the target efficiency is achieved in wide operating conditions, by incorporating the developed aerodynamic technology.

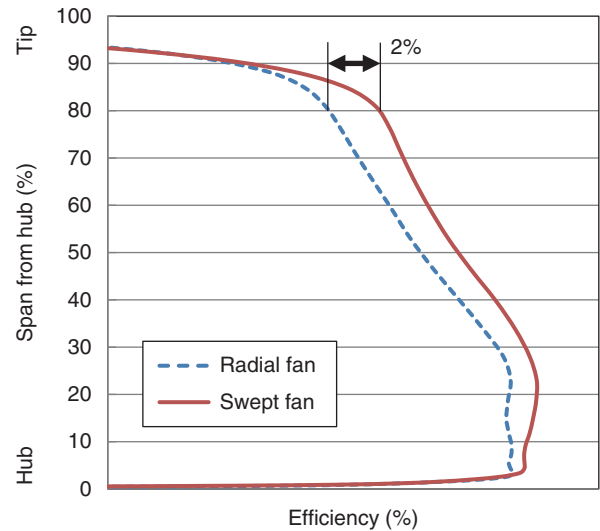


Fig. 5 Span-wise efficiency distribution of fan rotor exit

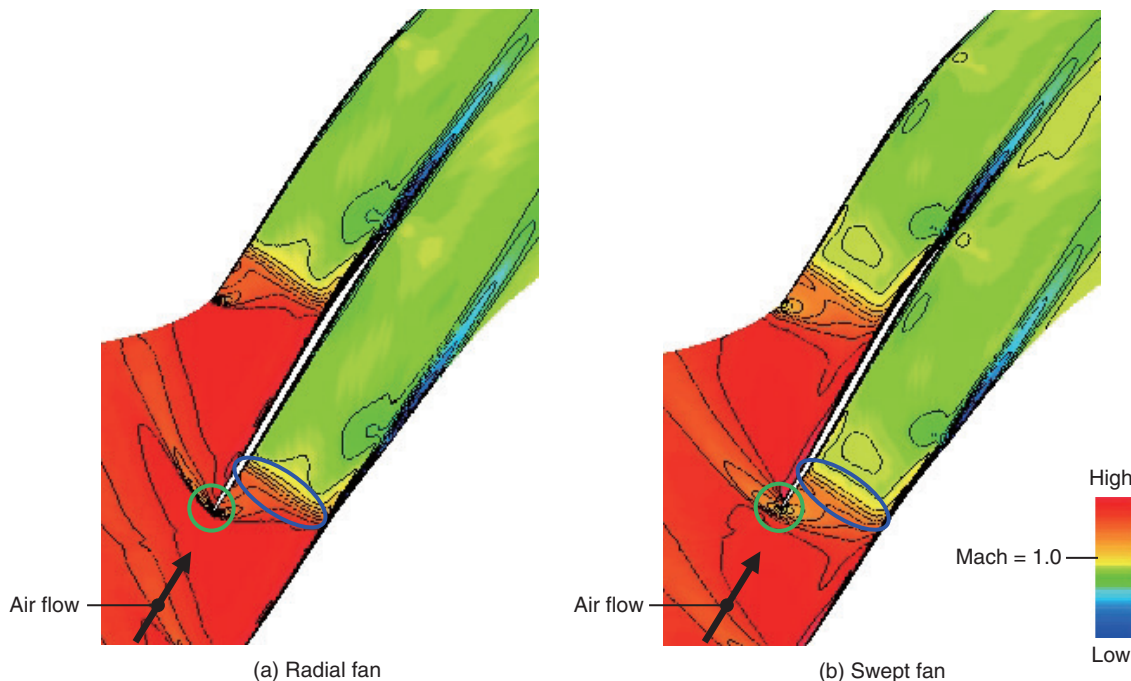


Fig. 4 Mach contours of fan rotor at 80% span from hub

Inlet specific flow rate is also increased by 2%, thus satisfying flow rate target. Small-scale rig tests in various engine conditions were also conducted and all results satisfied the requirements in each condition.

With these results, confidence in design and analysis were substantiated.



Fig. 6 Swept fan of small-scale rig

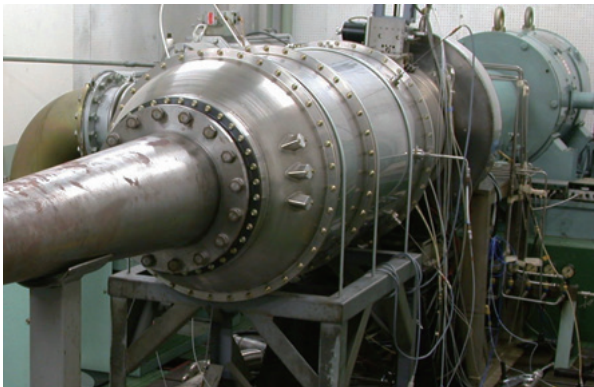


Fig. 7 Small-scale rig of fan

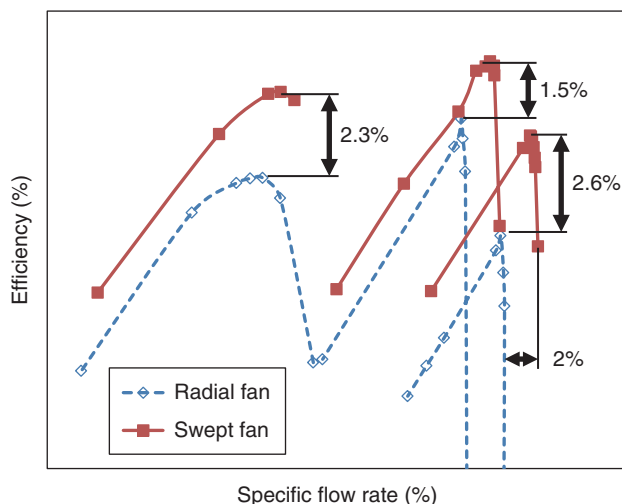


Fig. 8 Fan efficiency of small-scale rig

4. Aerodynamic Technology - Centrifugal Compressor

4.1. Centrifugal Compressor Design Concept

The high specific-flow-rate centrifugal compressor, as researched prior to the HF120, was designed with a smaller outer diameter than the conventional Honda centrifugal compressor of the HF118⁽³⁾, as shown in the schematic view in Fig. 9. Therefore, the specific flow rate of the compressor is increased and aerodynamic load on the rotor should be significantly increased. However, such increase of aerodynamic load can cause drastic pressure loss induced by flow separation, and result in lower efficiency.

Therefore, the number of blades was increased to prevent load from exceeding the threshold, and the configuration of the blade was changed to increase aerodynamic load on the front portion of the rotor where the boundary layer is thinner to decrease pressure loss and increase efficiency.

Figure 10 shows a schematic of aerodynamic load distribution in airflow direction at the tip of the rotor. The goal of this design was to increase aerodynamic load on the high specific flow compressor in the 10% to 60% area in the flow direction from the front end of the centrifugal impeller.

4.2. Centrifugal Compressor Performance Prediction

A CFD analysis tool was utilized for centrifugal compressor performance prediction to verify the feasibility of the targeted 6% specific flow increment. Figure 11 shows span-wise efficiency distribution of both conventional and high specific-flow-rate centrifugal compressor. The high specific-flow-rate centrifugal compressor shows enhanced efficiency at the 90% span point (near the tip) of

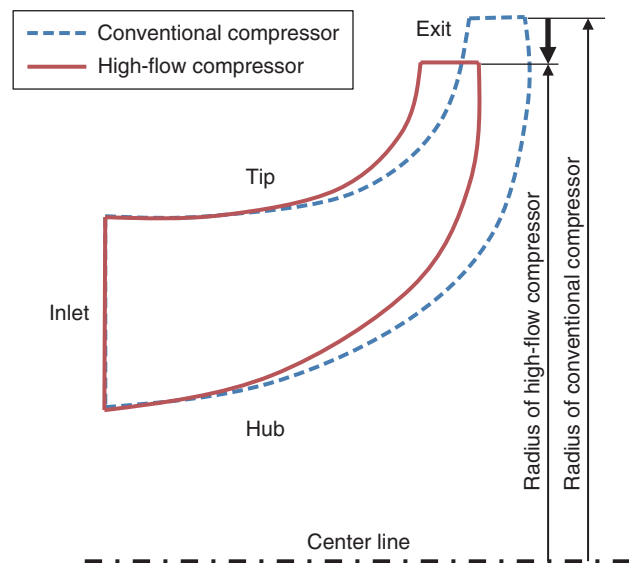


Fig. 9 General side-view of conventional and high-flow centrifugal impeller

1.5%, and retained at least equivalent or more efficiency at other points, although the aerodynamic load is increased.

Calculated efficiency by integrating efficiency distribution to span-wise direction, and multiplying by the correlation factor between CFD analysis and testing showed that the centrifugal compressor efficiency was increased by 0.5%, indicating that the target efficiency is achievable.

4.3. Centrifugal Compressor Performance Substantiation

Following completion of the centrifugal compressor performance prediction by CFD analysis, aerodynamic performance tests were performed using a small-scale prototype to reduce engineering time of manufacturing and duration for substantiation testing in the same manner as the fan. Special consideration was also given to processing accuracy on prototype manufacturing and blade geometry was verified with a 3D coordinate measuring machine.

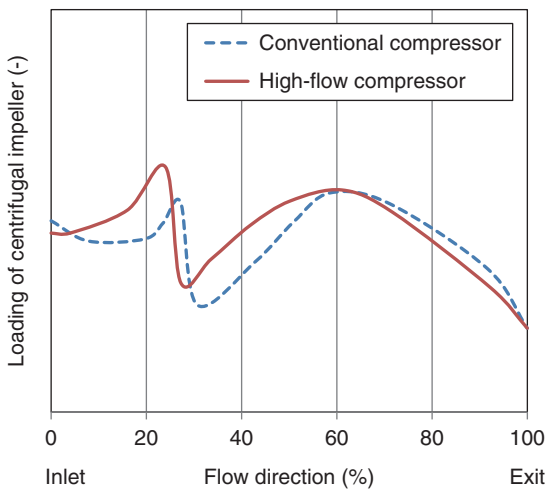


Fig. 10 Loading of centrifugal impeller

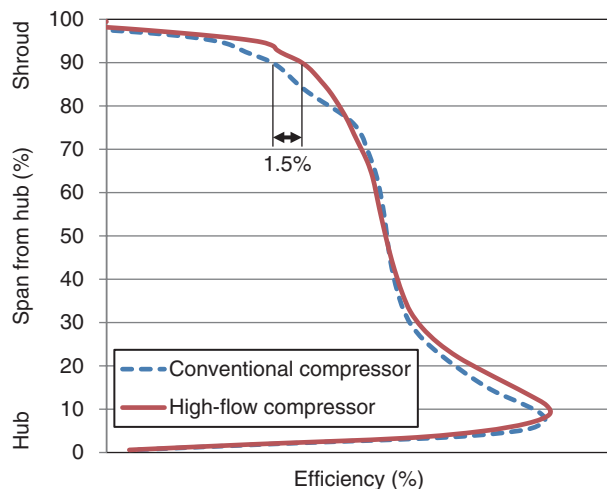


Fig. 11 Span-wise efficiency of centrifugal impeller exit

During the rig test operation, pressure and temperature measurement above and downstream of the centrifugal compressor where the air flow is stable was conducted to provide accuracy of test results.

The small-scale centrifugal impeller prototype is shown in Fig. 12, and the small-scale rig test facility is shown in Fig. 13.

Figure 14 shows the small-scale rig test efficiency map only in cruise condition considering little efficiency variation at various operating conditions. Peak efficiency is increased by 0.8%, indicating that the target efficiency increase is achieved. Exit specific flow rate of the high specific-flow-rate centrifugal compressor, which is compensated by the pressure ratio corresponding to that of conventional centrifugal compressor, is increased by 7%, achieving target flow rate. Small-scale rig tests in various engine-operating conditions were conducted and all results satisfied the requirements in each condition.

With these results, confidence in design and analysis were substantiated.



Fig. 12 High-flow centrifugal impeller of small-scale rig

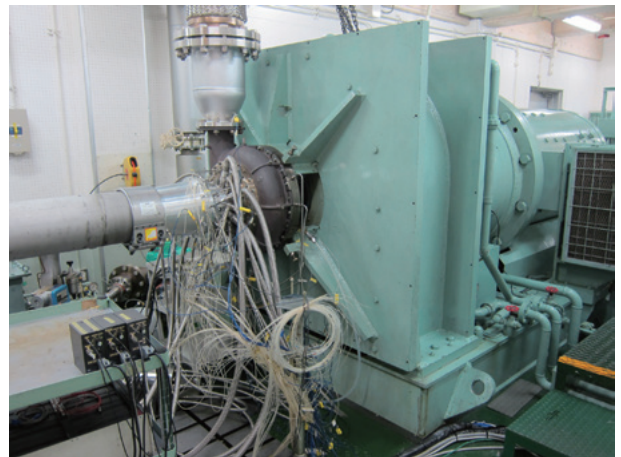


Fig. 13 Small-scale rig of centrifugal compressor

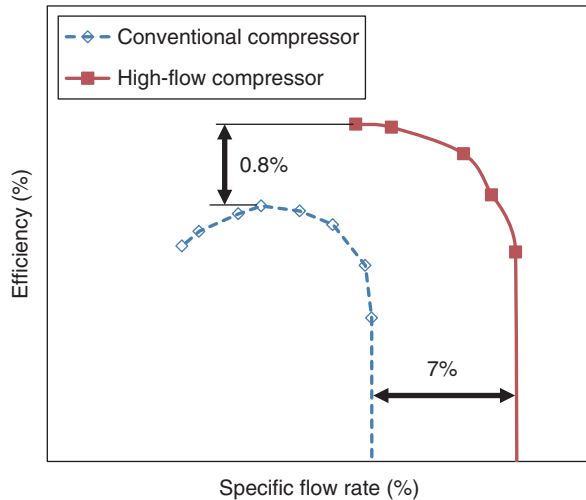


Fig. 14 Centrifugal compressor efficiency of small-scale rig

5. Conclusion

Prior to HF120 development, a study of aerodynamic technology that enables the design of a high-efficiency and high specific-flow-rate fan and centrifugal compressor was performed.

The fan design incorporated a swept fan configuration on fan rotor leading edge to weaken shock waves, and the centrifugal compressor design incorporated intentionally incremented aerodynamic load at the front portion of the blades where the boundary layer is thinner.

Performance prediction with CFD analysis verified reduced pressure loss from shock waves and enhanced efficiency of the entire span area of the fan, as well as enhanced efficiency of around at 90% span point on the centrifugal compressor.

Small-scale rig test substantiated enhanced fan efficiency by 1.5% or more and enhanced fan inlet specific flow rate by 2% in three typical operation conditions, plus enhanced centrifugal compressor efficiency by 0.8% and enhanced centrifugal compressor exit specific flow rate by 7% in cruise condition. Therefore, the test results of the fan and centrifugal compressor satisfied the numerical targets of the preceding research.

References

- (1) Sonoda, T., Noda, E.: Kogata Bijinesu Jyetto yo HF118 Tabofan Enjin no Kaihatsu, Journal of Gas Turbine Society of Japan, Vol. 34, No. 3, p. 165-171 (2006) (in Japanese)
- (2) Arima, T., Sonoda, T., Shirotori, M., Tamura, A., Kikuchi, K.: A Numerical Investigation of Transonic Axial Compressor Rotor Flow Using a Low

- Reynolds Number k - ϵ Turbulence Model, Journal of Turbomachinery, Vol. 121, No. 1, p. 44-58 (1999)
- (3) Oana, M., Kawamoto, O., Otani, H., Yamamoto, Y.: Approach to High-Performance Transonic Compressor Design, Journal of Propulsion and Power, Vol. 20, No. 1, p.164-170 (2004)

■ Author ■



Hisato TANAKA



Mineyasu OANA