Effects of Low Reynolds Number on Performance in a Centrifugal Compressor*

Jaehyo KIM,1) Heungsu JEON,1) Yohan JUNG,2) Jun Young PARK,3) and Minsuk CHOI1)†

1)Department of Mechanical Engineering, Myongji University, Yongin 17058, Republic of Korea
2)SFR System Design Division, Korea Atomic Energy Research Institute (KAERI), Daejeon 34057, Republic of Korea
3)Department of Extreme Energy Systems, Korea Institute of Machinery and Materials (KIMM), Daejeon 34103, Republic of Korea

The centrifugal compressor is an essential part of the auxiliary power unit (APU) for airplanes and helicopters. As aircrafts are sometimes required to operate at high altitudes, the aerodynamic condition of the gas turbine and APU lies at low Reynolds numbers. This study presents numerical simulations to investigate the effects of low Reynolds numbers on the flow and performance of a centrifugal compressor, including the impeller, diffuser and volute. For the reference Reynolds number (8.4×10⁴), the numerical results agreed well with the experimental measurement in total pressure ratio. It was found that the performance of the overall compressor decreases slowly as the Reynolds number decreases, but drops significantly below the threshold of 200,000. The flow inside the impeller didn’t separate in spite of thick boundary layers, even at the lowest Reynolds number (8.4×10⁶). In this study, it was found that the diffuser is the most susceptible part for separation caused by low Reynolds numbers. The large separation in the diffuser vanes degrades the flow, and decreases the efficiency of the diffuser and volute successively.

Key Words: Low Reynolds Number, Centrifugal Compressor, Loss, Separation

Nomenclature

LE: leading-edge
p: static pressure
p₀: inlet total pressure
PS: pressure surface
SS: suction surface
TE: trailing-edge
Vm: meridional velocity magnitude
η: efficiency
θ: circumferential angle

1. Introduction

Recently, Weinberg and Wyzykowski1) tested the PW545 jet engine above 18.3 km, while the engine was originally designed to operate at 13.7 km. They reported that overall engine efficiency and performance significantly decreased due to the low air density at high altitude. According to U.S. standard atmosphere,2) air properties such as density and viscosity change continuously and considerably with altitude. In particular, at the altitude of interest, 20 km, the density and viscosity of the air are respectively 7% and 77% of their sea-level values, and the corresponding Reynolds number decreases to 10% of the sea-level value due to the low air density. Castner et al.3) conducted numerical simulations for a gas turbine with changing Reynolds numbers and compared their results with the experimental data of Weinberg and Wyzykowski.1) There was good agreement between the test data and the calculated performance within Reynolds numbers from 30,000 to 295,000. In their numerical study, they considered Reynolds numbers only to describe air property changes at high altitudes.

Lower Reynolds numbers increase the boundary layer thickness on blade surfaces, thereby making turbomachinery susceptible for flow separation. For axial turbines, Fielding4) found that the total pressure loss in a turbine generally varies according to the −0.2 power of the Reynolds number. He also reported the critical Reynolds number of 100,000, below which the total pressure loss increases significantly. Van Treuren et al.5) ran wind turbine tests at different Reynolds numbers from 25,000 to 50,000 and found that large separation is generated near the trailing-edge of a turbine cascade and causes large total pressure loss. Matsunuma and Tsutsui6) found, in their experiments, that a decreasing Reynolds number causes high turbulence intensity in the wake and large flow fluctuations between stators and rotors. In the paper that followed, Matsunuma7) found that total pressure loss varies according to the −0.35 power of the Reynolds number, but the tip leakage loss is not affected by the Reynolds number. Regarding axial compressor cascade, a long time ago, Wassel8) carried out experiments to analyze the relationship between the flow and Reynolds number and found the critical Reynolds number of 200,000. Schreiber et al.9) measured transition phenomena on the blade surface in a compressor cascade by changing the Reynolds number. They observed a laminar separation bubble with reattachment on the suction surface at a relatively low Reynolds number. Choi et al.10,11) and Kim et al.12) conducted extensive numerical simulations to investigate the effects of low Reynolds numbers on the performance of three different axial compressors: a subsonic single rotor, a transonic single rotor, and a transonic one-stage...
axial compressor. Based on their numerical results, they confirmed that axial compressors have critical Reynolds numbers around 250,000, below which the total pressure loss in the axial compressors increases rapidly and significantly. In particular, Choi et al.10,11) divided the total pressure loss into each loss category using Denton’s loss model.13) They found that the wake loss only increases significantly due to large separation on the blade suction surface, but other losses generated by tip leakage flow, shock and boundary layers on the end-wall decrease or remain unchanged as the Reynolds number drops.

Although axial turbomachinery is mainly used in many applications in the aerospace industry, a centrifugal compressor is also one of the important parts in the APU for airplanes and helicopters. In addition, the turbocharged spark-ignited gasoline engine has been considered as the only option for the subsonic propulsion system of high-altitude unmanned air vehicles (UAVs).14) Studies that analyze the Reynolds number effects on the performance of centrifugal compressors have been ongoing for a long time. When an existing compressor that is originally designed for air is applied for use with different gases in the chemical and petroleum industries, or a scaled-down version of it is required for small applications, the Reynolds number can change. Wiesner15) and Casey16) analyzed the effects of Reynolds number on the performance of centrifugal compressors. They applied the same scaling factor to all geometries of the compressor overall and also that of each compressor so that any scaling effect on a centrifugal compressor can change. Wiesner15) and Casey16) suggested a prediction model for Reynolds number effects on the performance of centrifugal compressors. In particular, Casey’s model is based on the similarity of the Reynolds number effects on performance. Zheng et al.18) analyzed the effects of Reynolds numbers on the performance of a transonic centrifugal compressor for a turbocharger experimentally and numerically. They found that a significant interaction between the boundary layers and the tip leakage flow is a main cause of the increased loss caused by a low Reynolds number. Tiainen et al.19) conducted a numerical study of the Reynolds number effect on a centrifugal compressor with an impeller only, where the size of two different impellers was reduced to change the Reynolds number. They applied the same scaling factor to all geometries of each compressor so that any scaling effect on the performance was eliminated. Based on their numerical results, it was found that the loss related to the tip clearance flow and the boundary layer on the impeller blades increases as the Reynolds number decreases.

Although the Reynolds number effect has been investigated experimentally and numerically in many previous studies, most of them focused on only the flow and loss in the impeller. However, a centrifugal compressor is usually composed of three parts; namely, the impeller, diffuser and volute, and the other two parts are also susceptible to boundary layer separation. The objective of this study is to analyze the effects of low Reynolds numbers on the performance of the centrifugal compressor overall and also that of each part, and to confirm the critical Reynolds number of 200,000 suggested in previous studies.

2. Test Facility

The centrifugal compressor used for numerical simulations (Fig. 1) was designed and tested at the Korea Institute of Machinery and Materials (KIMM). The design speed of the compressor is 50,000 rpm and the corresponding Reynolds number is about 0.8 million based on the impeller inlet tip speed and impeller outlet diameter. The design mass flow rate is 0.3 kg/s and the design total pressure ratio is 2.45, as summarized in Table 1. This compressor consists of 15 impeller blades and 17 diffuser vanes. The impeller is an unshrouded type with a constant tip clearance of 0.3 mm from the leading-edge to the trailing-edge. The tip clearance height corresponds to 6% of the impeller blade exit height. The diffuser vanes have a constant height of 5 mm, which is equal to the impeller exit height.

3. Computational Method

3.1. Numerical method and Reynolds number

A commercial flow solver, Ansys-CFX, was utilized to calculate the flow in the centrifugal compressor. The convection terms in the compressible Reynolds-averaged Navier-Stokes (RANS) equations were discretized applying a high-resolution scheme to obtain second-order spatial accuracy, and the diffusion terms were calculated from the shape
function based on the finite element method. A fully implicit time stepping method with first- and second-order accuracies was applied to obtain steady and unsteady solutions, respectively. The turbulent viscosity was obtained using the k-ω shear stress transport (SST) model, and the laminar viscosity was calculated using Sutherland’s law. A transition model was not included in the simulation in order to reduce the computational cost and improve numerical stability. The total pressure, total temperature and flow angle were fixed at the inlet, and the mass flow rate was specified at the outlet of the volute. The interface between the impeller and diffuser was treated as the mixing plane for steady simulations and the sliding plane for unsteady simulations. Since steady simulations used a single passage in the impeller, a periodic condition is required to enable flow variables to be continuous across the boundary.

As stated above, Castner et al.3) changed Reynolds numbers only to consider the variation of air properties depending on high altitudes. For the current centrifugal compressor, the Reynolds number at the design point is 8.4 × 10^5. For this study, this is the value that is considered as the reference Reynolds number and referred to as “Ref. Re.” Since the Reynolds number at an altitude of 20 km decreases to be 10% of its sea-level value, the simulations were conducted using five different Reynolds numbers between 8.4 × 10^4 and 8.4 × 10^5. For convenience, the lowest value is referred to as “Low Re.” For considering changes in the air density and viscosity, the Reynolds numbers in the simulations were changed by kinematic viscosity.

### 3.2. Computational mesh and grid-dependency test

The computational domain for steady simulations consists of a single impeller passage and all diffuser passages including the volute. Using a commercial mesh generator, Ansys ICEM-CFD, a hexahedral mesh was generated over the regions of the main flow channel in the impeller, diffuser and volute, but a tetrahedral and prism meshes were generated around the volute tongue as shown in Fig. 2. For the grid-dependency test, three different grids with different resolutions were generated, as listed in Table 2. To resolve the tip clearance flow in the impeller, each grid had 10, 16 and 25 nodes in the spanwise direction inside the tip gap.

Before proceeding with the study, a grid-dependency test for the given centrifugal compressor was carried out using steady simulations. Figure 3 shows the total pressure ratios at the design point depending on different grid resolutions. Grids 2 and 3 have similar values for the total pressure ratio at the design point for the reference Reynolds number, but the total pressure ratio of Grid 1 is smaller than other two cases. No further improvement was expected by increasing node numbers, so Grid 2 was selected for simulating steady flows in the compressor and analyzing the effects of low Reynolds numbers on performance. The unsteady simulations were performed using the same grid resolution as Grid 2, where the computational domain with a whole impeller has 8,913,589 nodes.

### 4. Computational Results

#### 4.1. Steady flow

In the performance test, the rotational speed of the centrifugal compressor was fixed at 80% of the design speed due to some mechanical problems, so validation was conducted at a reduced speed. However, numerical simulations were conducted at both 80% and 100% of the design speed, as shown in Fig. 4. For the 80% speed, the predicted total pressure ratio matches well with the experimental data. It is clear that the total pressure ratio decreases as the Reynolds number decreases, and it seems that the reduction in total pressure rise is similar both at the design speed and at the reduced speed. At 0.22 kg/s with maximum efficiency for the 80% speed, the total pressure ratio decreases by 5.48% as the Reynolds number decreases. For the design speed, total pressure ratio at the design mass flow rate of 0.3 kg/s decreases by 5.45%. Another important thing in this figure is the change in the choking mass flow rate. The choking mass flow rate decreases as the Reynolds number decreases. This is because large separation in the diffuser causes significant blockage in the vane passages. Actually, the dependency of the internal flow in the centrifugal compressor on the Reynolds number is quite sim-

---

Table 2. Three different grids for the dependency test.

| Grid No. | Impeller | Diffuser | Volute | Total |
|----------|----------|----------|--------|-------|
| 1        | 229,125  | 543,150  | 310,270 | 1,082,545 |
| 2        | 480,604  | 1,150,900| 553,629 | 2,185,133 |
| 3        | 749,104  | 1,814,336| 769,073 | 3,332,513 |

Fig. 2. Computational domain and mesh (geometry, computational mesh, impeller mesh and diffuser mesh).

Fig. 3. Total pressure ratio at the design point.
ilar regardless of the rotating speed. Accordingly, numerical results at the design speed are mainly presented for different Reynolds numbers from now on. In this study, predicting a numerical stall point was not attempted.

The adiabatic efficiency was calculated from the inlet to the impeller exit, diffuser exit and volute outlet at the design point, and is shown in Fig. 5(a). At the design mass flow rate, the adiabatic efficiency inside the impeller decreases by 3.15% points when the Reynolds number decreases to one-tenth of the reference value. From the inlet to the diffuser exit, the lowest Reynolds number causes efficiency to decrease by 6.15% points. Totally, the loss of efficiency from the inlet to the volute exit reaches 7.44% points due to the lowest Reynolds number. It is worth noting that the efficiency loss is relatively small at lower mass flow rates than the design point as compared to higher mass flow rates. Using the efficiency relationship in Eq. (1), the overall efficiency can be divided into the efficiency of each component and the results are shown in Fig. 5(b).

$$\eta_{\text{compressor}} = \eta_{\text{impeller}} \cdot \eta_{\text{diffuser}} \cdot \eta_{\text{volute}}$$

At the design point, the efficiency loss at the lowest Reynolds number is 3.15% points, 3.69% points and 1.86% points in the impeller, diffuser and volute, respectively. The amount of loss in the diffuser is higher than in the other parts and the volute efficiency is not significantly affected by the Reynolds number change, implying that lower Reynolds numbers have the greatest effect on the vaned diffuser in a centrifugal compressor.

The flow pattern inside the impeller could be easily understood in the rotating frame, showing the relative total pressure distribution at the impeller exit (Fig. 6). The high relative total pressure region appears near the pressure-side of the blade and the low relative total pressure region near the suction-side of the blade. This is a typical jet-wake flow pattern in the discharge flow of an impeller in a centrifugal compressor. The high-loss flow including the tip leakage flow accumulates on the blade suction surface along the passage and it is finally recognized as a wake at the impeller exit. As the Reynolds number decreases, the low total pressure region expands in the circumferential direction and causes the loss of efficiency. The high total pressure region on the pressure surface also shrinks as the Reynolds number decreases. However, there was no separation observed on the impeller blade regardless of the decreasing Reynolds number. This might be caused by the absence of a transition model.

Figure 7 shows the tip leakage flow colored by vorticity magnitude for two different Reynolds numbers. Tip leakage flow at the front part interacts with the main flow and the shear layer on the shroud. It then rolls up strongly and forms a vortex with high vorticity. The tip leakage vortex turns down to the hub at the mid-chord position and moves along the impeller passage as vorticity magnitude decreases. As in-
dicated by “A” in Fig. 7, the tip leakage vortex passes through the wake region thereafter. The tip leakage flow at the rear part moves in the circumferential direction without rolling up. The tip leakage flow pattern looks similar and independent of Reynolds numbers, but the tip leakage flow at the front part (B) is stronger at the reference Reynolds number, while the tip leakage flow at the rear part (C) is stronger at the low Reynolds number. This is caused by the change in blade load near the tip depending on the Reynolds numbers. As shown in Fig. 8, the static pressure difference between pressure and suction surfaces decreases at the front part, but increases at the rear part at a low Reynolds number. Here, it is worth noting that the circumferential position of the front tip leakage flow (A) moves towards the pressure surface on the impeller exit plane at low Reynolds numbers, which causes expansion of the low relative total pressure region shown in Fig. 6.

The circumferential distribution of the meridional velocity magnitude at the mid-span of the impeller exit is extracted from the numerical results and shown in Fig. 9 in order to investigate the flow pattern change as the Reynolds number decreases. The meridional velocity is large near the pressure-side of the blade and small near the suction-side of the blade. As the Reynolds number decreases, the velocity increases in the jet region near the pressure-side and decreases in the wake region near the suction-side. The boundary layer becomes thick on the blade, hub and shroud surfaces when the Reynolds number is low, and the thicker boundary layers work to block the flow passage. Consequently, more of the flow passes through the middle of the blade passage. The change in impeller exit flow at the low Reynolds number affects the incidence angle of the diffuser vanes, and successively has a large effect on the flows inside the diffuser and volute.

Figure 10(a) shows the meridional velocity distribution along the span at the impeller exit, which is circumferentially mass-averaged. It is clear that the boundary layer on the hub surface grows larger as the Reynolds number decreases. The velocity defect region above 80% span, where there is a dominant effect caused by the tip leakage flow, remains similar even at the lowest Reynolds number. As the boundary layer...
increases with decreasing Reynolds number, the velocity in the core flow region between the spans of 10% and 80% increases due to the increased blockage near the hub and shroud. As shown in Fig. 10(b), the spanwise distribution of isentropic efficiency is significantly affected by the change in Reynolds number. Although the difference in efficiency depending on the Reynolds numbers is relatively small between the spans of 5% and 60%, efficiency is reduced considerably near the hub and casing. On the hub, the friction loss inside the boundary layer increases significantly. Above a 60% span, the larger viscosity increases the loss as the tip leakage flow mixes into the shroud boundary layer, consequently reducing efficiency. This result agrees well with the results of previous studies.18,19)

Figure 11 shows the absolute total pressure distribution at the mid-span of the diffuser for the design mass flow rates. At the reference Reynolds number, relatively thin separation on the diffuser suction surface is observed. The size of separation changes slightly in the circumferential direction due to static pressure variation inside the volute. Although the flow behaves well and separations are still thin at 30% of the reference Reynolds number, much larger separation occurs on the suction surface of the diffuser vanes at the lowest Reynolds number. This large separation on the vanes could degrade performance of the centrifugal compressor and block the flow passage inside the diffuser, thereby decreasing the choking mass flow rate.

The size of the separations on the diffuser vanes is closely related to the flow angle upstream of them, and the flow angle distribution along the span is shown in Fig. 12, which is mass-averaged in the circumferential direction. For the reference Reynolds number, the diffuser inlet flow has a proper incidence angle at spans from 5% to 80%, but a negative flow angle is observed above a span of 90% due to the tip clearance flow in the impeller, even at the design point. As the Reynolds number decreases, the flow angle increases significantly in the core flow region. However, it decreases near the hub and casing, which is caused by concentration of the flow in the core flow region inside the impeller. This change in flow angle makes the separations on the vane suction surfaces worse and causes large loss in the diffuser.

Figure 13 shows the limiting streamlines on the diffuser vane surfaces in order to analyze the structure of separations. At the reference Reynolds number, the boundary layer on the pressure surface separates only at the trailing-edge near the end-wall with small flow angles, while large separation appears on the suction surfaces. The separation covers the upper part near the shroud after the mid-chord position. As the Reynolds number decreases, the flow angle increases in the core flow region and decreases near the end-wall so that the separations on the vane surfaces become large. On the pressure surface, separation at the trailing-edge increases in size with a negative flow angle near the shroud. The flow on the suction surface separates completely after the 25% chord position and its size becomes even larger.

Figure 14 shows entropy distribution in the streamwise direction, where the entropy has been calculated using mass-averaged static temperature and pressure. For the reference Reynolds number, the entropy is negligible in the inlet duct because the boundary layer is too thin. But it increases significantly inside the impeller due to the friction on the blades, the tip leakage flow and the wake mixing. In the diffuser, the increase in entropy remains small, implying that the diffuser efficiently converts fluid velocity to pressure without much loss. At a low Reynolds number, the boundary layer on the wall becomes thick so that the entropy increases along
the entire flow passage. In particular, entropy continues to increase in the diffuser vanes because of the large separation on the vane surfaces.

The volute plays an important role in gathering the flow from the diffuser vanes and recovering the static pressure at the cost of decreasing velocity magnitude, but it has a bit complex geometry. It is better to define the positions of the planes before analyzing the flow inside the volute. In the circular part, the angle increases in the clockwise direction and the starting point is defined as shown in Fig. 15. In the straight part, the position is defined by the x-coordinate. Figure 16 shows the entropy distribution on the planes indicated in the previous figure in order to investigate the flow inside the volute. Since the flow patterns in the volute are quite similar to one another independent of the Reynolds number and mass flow rate, only the flow at the design point is presented. A high entropy region appears on the wall and at the center of the volute: the former is due to friction in the boundary layer and the latter is due to mixing of the non-uniform flow from the upstream diffuser vanes.

Figure 17 shows entropy distribution along the volute from the volute tongue to the outlet. The volute gathers the discharge flow from the diffuser vanes in the curved part so that severe mixing occurs throughout the volute. For the reference Reynolds number, entropy at the center along the volute decreases slowly and steadily as a result of the mixing process, while the mass-averaged entropy along the volute remains nearly constant. At a low Reynolds number, entropy increases significantly depending on the size of the separations on the vane suction surfaces. It is evident that the mixing process in the volute is more severe when there are larger separations on the diffuser vanes, but the trend in entropy change at the center of the volute is similar to that at the reference Reynolds number. The entropy average remains constant at the curved part due to the inflow from the diffuser, but slowly increases at the straight part due to the mixing.

As shown in Fig. 4 and Fig. 5(a), the reduction in performance caused by the Reynolds number increases as the mass flow rate increases, and the choking mass flow rate is significantly reduced as the Reynolds number decreases. The absolute total pressure distribution is displayed at the mid-span of the diffuser at the higher mass flow rate in Fig. 18. While the total pressure distribution is similar to Fig. 11(a) of the design point at the reference Reynolds number, the size of separations on the vane suction surfaces becomes much larger at a low Reynolds number. It is evident that these large separations could be caused by the increase in flow angle downstream of the impeller, where the flow angle at the higher mass flow rate is larger than that at the design mass flow rate. Moreover, more than half of the vane passages are blocked by large separations, which could limit the choking mass flow rate. Downstream of the diffuser, severely uneven flow enters the volute due to these separations. This causes violent mixing to occur, which degrades the flow and efficiency of the volute.
4.2. Unsteadiness for different Re

Unsteady flow simulations were conducted at the design point to analyze the effects of Reynolds numbers on unsteady flow inside the centrifugal compressor. A numerical pressure sensor was installed at the mid-span between the impeller trailing-edge and diffuser leading-edge. The unsteady simulations were continued for 26 impeller revolutions to obtain periodic solutions. During the last revolution, 15 large fluctuations repeatedly appeared as the impeller blade passed, as shown in Fig. 19. The peak-to-peak value of the pressure-time history decreases as the Reynolds number decreases, implying that the unsteadiness caused by the impeller-diffuser interaction is reduced in size.

The impeller-diffuser interaction significantly affects the flow in the diffuser, although it has little effect on the flow in the impeller at the design point. Figure 20 shows the instantaneous total pressure distribution at the mid-span of the diffuser. As observed in the steady results in Fig. 11, thin separations on the diffuser suction surface at the reference Reynolds number changes to become large separations at a low Reynolds number. At each Reynolds number, however, the size of the separations in the unsteady simulations is smaller than that in the steady simulations. It was observed that the impeller wake is chopped by the diffuser vanes and transported to downstream through the diffuser passages. The impeller wake undergoes both viscous mixing and inviscid strain inside the diffuser, where the inviscid strain is a reversible process. Therefore, the mixing loss of the impeller wake caused by the mixing plane in the steady simulations is much larger than the loss caused by the actual mixing inside the diffuser in the unsteady simulations. The more energetic flow in the unsteady simulations could suppress the separations on the diffuser vanes and increase compressor performance.

4.3. Performance variation depending on Re

Figure 21(a) shows the performance variation depending on the Reynolds number. Total pressure ratio and efficiency were obtained from the design point for 50,000 rpm and the peak efficiency point at 0.22 kg/s for 40,000 rpm. Here, the total pressure ratio is normalized by the value at the reference Reynolds number for two different speeds. Performance decreases slowly and steadily as the Reynolds number decreases, but the slope of performance reduction changes significantly around 200,000, which might be a critical Reynolds number and is the same as the value suggested in previous studies. The variation in the efficiency of each component in the compressor is plotted along the Reynolds number in Fig. 21(b). As the Reynolds number decreases above the critical number of 200,000, the efficiency of all components decreases slowly. Below the critical Reynolds number, the efficiency of the impeller, diffuser and volute drops significantly and simultaneously. By the way, the efficiency of the impeller was higher in the unsteady simulations than in the steady simulations by about 1% point. The difference in efficiency between steady and unsteady simulations mainly occurs inside the diffuser due to wake recovery, while efficiency in other components is similar. However, the performance predicted by unsteady simulations shows the same tendency as that for steady simulations.

Based on the numerical results, a mechanism that causes loss to increase at low Reynolds numbers could be made for centrifugal compressors. It is well known that wall boundary layers become thicker as the Reynolds number decreases. Since the thick boundary layers work as blockage, the flow moves to the core flow region and the flow velocity increases at the center for all compressor components. In particular, the wake thickens in the impeller and diffuser at low
Reynolds numbers. Referring to the loss variation in axial compressors depending on Reynolds numbers,\textsuperscript{10,11} the loss caused by the boundary layers and wakes increases with decreasing Reynolds number, but there is no major increase without large separation. Above the critical Reynolds number for the centrifugal compressor, only thin separations are observed on the diffuser vane suction surfaces and there is no separation inside the impeller. Therefore, the loss increases steadily and slowly as the Reynolds number decreases above the critical value due to friction loss inside the boundary layers.

Below the critical Reynolds number, Zheng et al.\textsuperscript{18} reported that the interaction between the separated boundary layer and the tip leakage flow inside the impeller causes severe loss in a transonic centrifugal compressor. Even though there is no separation inside the impeller of the subsonic centrifugal compressor, as in this study and that of Tiainen et al.,\textsuperscript{19} considerable changes were observed in boundary layer thickness, wake and tip leakage flow as the Reynolds number decreased. These changes increase the loss in the impeller itself and affect the flow angle downstream of the impeller, consequently causing severe separation of the flow inside the diffuser. The large separation on the vane suction surfaces decreases diffuser efficiency further and limits the choking mass flow rate due to blockage. A highly non-uniform flow in the circumferential direction from the diffuser degrades the flow inside the volute successively and causes additional mixing loss.

5. Conclusion

Three-dimensional flow simulations were conducted to investigate the effect of Reynolds numbers on the performance of a centrifugal compressor composed of an impeller, diffuser and volute. The following conclusion were made based on the numerical results.

1. The performance at low Reynolds numbers dropped significantly in comparison to the reference Reynolds number. There was a 5.45% reduction in the total pressure ratio and 7.44% points reduction in adiabatic efficiency at the design point due to low Reynolds numbers. The loss of performance at low Reynolds numbers depends considerably on the mass flow rate. At high flow rates, performance drops more than at low flow rates.

2. The loss inside the impeller increases above the 60% span due to strong interaction of the tip leakage flow and thick shroud boundary layers as the Reynolds number decreases, although there was no separation observed. At low Reynolds numbers, the more flow passed through the middle of the impeller blade passage due to blockage of thick boundary layers, the higher the flow angle in the core flow region at the impeller exit.

3. The diffuser is the most susceptible part for separation at low Reynolds numbers. The large separation on the diffuser vane degrades the flow and performance below the critical Reynolds number. This is due to an increase in the flow angle upstream of the diffuser vanes. Moreover, the non-uniform flow from the diffuser causes large mixing loss in the volute and reduces compressor efficiency even further.

4. The critical Reynolds number in a centrifugal compressor was confirmed to be around 200,000, which is the same as other previous studies. Below the critical value, total pressure loss increases quickly and significantly, and this is closely related to the separation inside the diffuser.

Acknowledgments

This work was supported by 2016 Research Fund of Myongji University.

References

1) Weinberg, M. and Wyzykowski, J.: Powering Unmanned Air Craft, \textit{Aerospace Engineering}, 21 (2000), pp. 23–26.
2) U. S. Standard Atmosphere, NASA-TM-X-7433b, 1976.
3) Castner, R., Chiappetta, S., Wyzykowski, J., and Adamczyk, J.: An Engine Research Program Focused on Low Pressure Turbine Aerodynamic Performance, GT-2002-3004, 2002.
4) Fielding, L.: \textit{Turbine Design}, ASME Press, New York, 2000, pp. 27–28.
5) Van Treuren, K. W., Simon, T., Von Koller, M., Byerley, A. R., Baughn, J. W., and Rivir, R.: Measurements in a Turbine Cascade Flow under Ultra Low Reynolds Number Conditions, *J. Turbomach.*, **124** (2002), pp. 100–106.
6) Matsunuma, T. and Tsutsui, Y.: Effects of Low Reynolds Number on Wake-Generated Unsteady Flow of Turbine Rotor, *Int. J. Rotating Machinery*, **2005** (2005), pp. 1–15.
7) Matsunuma, T.: Effects of Reynolds Number and Freestream Turbulence on Turbine Tip Clearance Flow, *J. Turbomach.*, **128** (2006), pp. 166–177.
8) Wassel, A. B.: Reynolds Number Effects in Axial Compressors, *J. Eng. Power*, **90** (1968), pp. 149–156.
9) Schreiber, H.-A., Steinert, W., and Küsters, B.: Effects of Reynolds Number and Free-stream Turbulence on Boundary Layer Transition in a Compressor Cascade, *J. Turbomach.*, **124** (2002), pp. 1–9.
10) Choi, M., Baek, J. H., Chang, H. T., Oh, S. H., and Ko, H. Y.: Effects of the Low Reynolds Number on the Loss Characteristics in an Axial Compressor, *J. Power Energy*, **222** (2008), pp. 209–218.
11) Choi, M., Baek, J. H., Park, J. Y., Oh, S. H., and Ko, H. Y.: Effects of Low Reynolds Number on Loss Characteristics in Transonic Axial Compressor, *Trans. Jpn. Soc. Aeronaut. Space Sci.*, **53** (2010), pp. 162–170.
12) Kim, T.-G., Jung, Y.-J., Jung, Y., and Choi, M.: Effects of Low Reynolds Numbers on Performance of a One-Stage Axial Compressor, *Trans. Jpn. Soc. Aeronaut. Space Sci.*, **58** (2015), pp. 280–288.
13) Denton, J. D.: Loss Mechanisms in Turbomachines, *J. Turbomach.*, **115** (1993), pp. 621–656.
14) Bents, D., Mockler, T., Maldonado, J., Harp Jr., J. L., King, J. F., and Schmitz, P. C.: Propulsion System for Very High Altitude Subsonic Unmanned Aircraft, NASA-TM-1998-206636, 1998.
15) Wiesner, F. J.: A New Appraisal of Reynolds Number Effects on Centrifugal Compressor Performance, *J. Eng. Power*, **101** (1979), pp. 149–156.
16) Casey, M. V.: The Effects of Reynolds Number on the Efficiency of Centrifugal Compressor Stages, *J. Eng. Gas. Turbines Power*, **107** (1985), pp. 541–548.
17) Casey, M. V. and Robinson, C. J.: A Unified Correction Method for Reynolds Number, Size, and Roughness Effects on the Performance of Compressors, *J. Power Energy*, **225** (2011), pp. 864–876.
18) Zheng, X. Q., Lin, Y., Gu, B. L., Zhuge, W. L., and Zhang, Y. J.: Effects of Reynolds Number of the Performance of a High Pressure-ratio Turbocharger Compressor, *Sci. China-Technol. Sci.*, **56** (2013), pp. 1361–1369.
19) Tiainen, J., Jaatinen-Väärä, A., Grönman, A., and Backman, J.: Numerical Study of the Reynolds Number Effect on the Centrifugal Compressor Performance and Losses, GT2016-56036, 2016.
20) Smith, L. H., Jr.: Wake Ingestion Propulsion Benefit, *J. Propul. Power*, **9** (1993), pp. 74–82.
21) Van Zante, D. E., Adamczyk, J. J., Strazisar, A. J., and Okishi, T. H.: Wake Recovery Performance Benefit in a High-Speed Axial Compressor, 97-GT-535, 1997.

Toshinori Watanabe
Associate Editor