Experimental Fluid Dynamics Applications in Radial Turbomachines: Inlet Recirculation in Centrifugal Compressor, Rotating Stall and Flow in Vaneless Diffuser, and Improvement in Accuracy of CFD for Predicting Flow Fields in a Radial Turbine Rotor

Hideaki Tamaki, Satoshi Ouchida, Masaru Unno
IHI Corporation, 1, Shin-nakahara-cho, Isogo-ku, Yokohama, Japan 235-8501

Abstract. Performance of a turbomachine strongly depends on its internal flow. CFD (Computational Fluid Dynamics) is now an essential tool obtaining information on such internal flow as well as its performance. However, when CFD is taken into a design process, the accuracy and feature of a CFD code must be understood. CFD requires EFD (Experimental Fluid Dynamics) for validating and ensuring its accuracy. Besides, EFD provides some empirical rules and models to complex phenomena that CFD needs high computational costs and long turnaround time and that are difficult to be integrated in a design work. Since CFD and EFD are complementary to each other, EFD with CFD or CFD with EFD serves as a powerful tool to understand flow phenomena in turbomachines. So, EFD is still important and frequently applied in industries to design and develop radial turbomachines. Examples of EFD conducted during the design of centrifugal compressors and radial turbines in the author’s affiliation are introduced here.

1. Introduction
In designing a fluid element of a turbomachine, it is usual that geometries of its flow channel are modified until desirable performance is obtained while the pattern of internal flow field, loss generations and overall performance such as pressure ratio and efficiency are being evaluated by CFD (Computational Fluid Dynamics). CFD provides all information about the flow field including physical quantities not measurable in experiments and is useful in a design work as above-mentioned. When CFD is taken into a design process, the accuracy and feature of a CFD code must be understood. The accuracy of the CFD code depends on specifications and shape of a turbomachine. Hence it is important to compare various measurement/experimental data with calculation results obtained at design stages, determine the discrepancies resulting from the differences in their specifications or shapes, and make a database of these information. This database enhances the design accuracy with CFD and shortens the development period of new turbomachines. Hence experiments i.e. EFD (Experimental Fluid Dynamics) are important for CFD and designing turbomachines.

A turbomachine must respond to various operating conditions depending on combinations of a rotor rotational speed and a flow rate. In general, the accuracy of performance prediction of a turbomachine with a CFD code deteriorates if any of operating conditions deviates from its design operating point.
Taking a compressor as an example, when the flow rate decreases on a constant rotor speed or the pressure increases under a constant flow rate, an unstable flow such as rotating stall or surge will occur. Speaking of such unstable phenomena, CFD is still in its research stage and hard to accurately predict the occurrence of these phenomena or cannot be integrated in the design work due to a high computational cost and long turnaround time. EFD can compensate such disadvantages of CFD as a design tool. EFD also can provide some empirical rules, criteria and models to complex phenomena CFD needs high computational costs to simulate.

Since CFD and EFD are complementary to each other, EFD with CFD or CFD with EFD serves as a powerful tool to understand flow phenomena in turbomachines. This paper introduces examples of the application of EFD for designing centrifugal compressors and radial turbines in the author’s affiliation. Since these activities were conducted during developing products, some are consistent, and others show a lack of consistency in experimental and calculation procedures. However, these would give something interesting to people engaging fluid mechanics in radial turbomachines. Following three topics are introduced here.

1. **Inlet recirculation in centrifugal compressor**
   - Oil flow, tuft flow techniques and PIV (Particle Image Velocimetry) were applied to a suction pipe of a turbocharger centrifugal compressor to find out the existence of reverse flow at the compressor inlet region called inlet recirculation prior to surge.

2. **Flow in vaneless diffuser**
   - 2-1. PIV was applied for better understanding flow structures of rotating stall in a vaneless diffuser. A tested compressor was designed to be accessible to measuring positions easily and to tend to initiate rotating stall in a vaneless diffuser.
   - 2-2. Flow fields in vaneless diffusers of two different automotive turbocharger compressors were compared by using PIV. Since both compressors had volutes applied to actual products, the flexibility of measurement was very limited compared to the previous case (2-1).

3. **Improvement in accuracy of CFD for predicting flow fields in a radial turbine rotor**
   - A measured flow velocity with LDV (Laser Doppler Velocimeter) downstream of turbine rotor was different from the one obtained by CFD which was used to design our radial turbines. Improvements in CFD accuracy were carried out by changing the modelling of rotor.

2. **Inlet Recirculation in Centrifugal Compressor**

Inlet recirculation is a flow phenomenon that fluid discharges upstream from an impeller inlet and flows back to the impeller after merging with a main flow of fluid inside a suction pipe. It is likely observed near surge on a low speed-line of a centrifugal compressor with vaneless diffuser. The recirculation zone extended upstream from the rotor into the inlet duct/suction pipe of a centrifugal compressor has long been known. Beser et al. visualized the inlet recirculation by using lampblack traces and tufts [1]. And Amann pointed out that when the zone of recirculated air does not extend upstream to the compressor inlet thermocouples, the instrumentation indicates a fictitiously high work coefficient [2]. Andersen et al. measured the growth in inlet recirculation of the turbocharger impeller with thermocouples installed in various axial positions on the inlet pipe [3]. The generation of inlet recirculation is frequently encountered (whether it really exists or not) in results of steady analysis of a single pitch of an impeller blade using CFD as shown in figure 1. Harley et al. derived the relationship between the area of a reverse flow section due to inlet recirculation and operating conditions by applying a similar single pitch analysis to various impellers on turbocharger compressors and advocated the loss prediction model for inlet recirculation [4]. However, there are little studies about the inlet recirculation in centrifugal compressors. Hence, we applied several flow visualization techniques to an automotive turbocharger compressor and tried to clarify the whole picture of inlet recirculation experimentally. Besides, this study indicated the difference between the inlet recirculation and surge by pressure measurements with pressure transducers and by flow visualizations with PIV.
Figure 1. Inlet recirculation, (a) Meridional stream line of inlet recirculation [4] (b) Example of steady CFD result: circumferentially averaged streamline (upper left), axial flow velocity of tip-clearance flow (lower left), streamline through the full blade tip clearance (right) [5].

The main parameters of the automotive turbocharger compressor investigated at this section are listed in Table 1. The outer radius of the impeller is \( r_2 = 25.5 \text{mm} \) and it has six full blades and six splitter blades respectively.

Table 1. Dimensions of tested compressor.

|       | \( r_{1s}/r_2 \) | \( r_{1h}/r_{1s} \) | \( b_2/r_2 \) | \( \beta_{b1s} \) | \( \beta_{b2} \) |
|-------|------------------|---------------------|---------------|----------------|----------------|
| Impeller | 0.77             | 0.29                | 0.13          | 61 deg.        | 43 deg.        |
| Diffuser | 1.15             | 1.86                | 0.75          |                |                |

r=radius, b=impeller or diffuser width, \( \beta_b \)=impeller blade angle
1=impeller inlet, 2=impeller outlet, 3=end of diffuser pinch, 4=diffuser outlet
h=hub, s=shroud

Figure 2. Compressor characteristics [5]

Figure 2 shows the tested compressor characteristics. \( m \) is a flow rate and \( m_d \) is a reference flow rate respectively. Mu is peripheral Mach number that is the ratio of impeller tip speed to sound speed based on the compressor inlet stagnation temperature. First, oil flow visualizations were applied to three different operational points shown in figure 2. The results are shown in figure 3. At the high flow rate (\( m/m_u = 0.81 \)), straight oil film patterns can be seen on the inner pipe wall. In contrast, when close
to the maximum efficiency point \((m/m_d=0.55)\) and to near surge point \((m/m_d=0.35)\), spiral oil film patterns can be observed. These were considered due to the inlet recirculation.

FIGURE 3. Results of oil flow visualization [5], impeller rotates clockwise.

Next, tuft flow visualization was applied to observe the flow upstream of impeller. Figure 4 shows sketches of tuft movements through an acrylic pipe. At higher flow rate, all the fluid flows towards the impeller inlet. As the flow rate is reduced, the tuft bowing and flowing towards the upstream of impeller at ③ can be confirmed. In surge condition, the tuft vibrates violently.

FIGURE 4. Tuft flow visualization, operational point ① to ④ (left), picture of tuft setting (upper right), picture and sketch of tuft movement (middle and lower right).

Then, measurements with PIV were conducted to the flow in the suction pipe. Because of an acrylic suction pipe, a low rotational speed \((\mu = 0.58)\) was selected as the test condition from the viewpoint of heat tolerance. Figure 5(a) shows locations of pressure transducers (Kulite XCE062) and figure 5(b) shows the measured (visualized) cross sections and regions. In this test, visualization of flows was conducted for two cross sections, one is the cross section in the pipe axial direction (between the position 3.12 times the impeller tip diameter upstream from the leading edge of the impeller and the trailing edge of the suction pipe) and the other is the cross section in a direction perpendicular to the pipe axis (at the position 1.05 times the tip diameter of the impeller upstream from the impeller leading edge).
Figure 5. Location of measurement

(a) Position of pressure transducer
(b) Measured plane with PIV [6]

Figure 6. Velocity vector distribution in central cross section along suction pipe axis [6].

The measured results at the central cross section (rectangular plane which cut the cylinder along its diameter) along the suction pipe axis are shown in figure 6. Figure 6 nearly corresponds to the
absolute velocity vector which is projected to the central cross section. At operating points A and B, the suction pipe is filled with the fluid flowing to the impeller. In contrast, at operating points C and D, the reverse flow regions near the pipe wall appear and enlarge upstream and radial direction as the flow rate is reduced. The reverse flow regions do not extend to the inlet of suction pipe. The velocity at the middle section of the pipe increases due to the reduction in the effective area caused by the reverse flow near the pipe wall.

Figure 7 shows velocity distributions observing from the front side of the pipe. Figure 7 nearly corresponds to the absolute velocity vector which is projected to the suction pipe circular cross-section. In B, velocity vector components parallel to the circular cross-section are hardly observed. In C and D, swirling flow is generated near the pipe wall. This swirling flow results by the mixing between the incoming flow to the impeller and the reverse which has the same swirl direction with the impeller rotating direction. This means that the inlet recirculation induces the positive pre-swirl at the impeller inlet and reduces the work transfer from the impeller to the fluid through the impeller.

![Figure 7. Velocity vector distribution observed from front side of suction pipe [6].](image)

Surge accompanies the back flow from the impeller. In order to check that the reverse flow near the suction pipe wall observed prior to surge has nothing to do with the back flow induced by surge, the flow field in surge condition was visualized with PIV. Surge period was divided into four-time intervals equally and PIV measurements were conducted at each time. The period of surge was between 0.2 and 0.24 second. Results are shown in figure 8. Different from figure 6, there are periods that the axial velocity of the main flow in suction pipe takes zero.

Figure 9 shows FFT results of signals from the pressure transducers at operating points B, C, D in figure 6, and the surge condition in figure 8. In the case of operating points C and D, broadband spectra in frequency less than 1000Hz are observed at the pressure transducer i, located just upstream of the impeller inlet. However, they disappear at the pressure transducer v, located at the suction pipe inlet. While, high intensity of spectrum peak in frequency of 4 to 5 Hz and the broadband spectra in frequency less than 1000Hz are observed from the impeller inlet to suction pipe inlet at the surge condition. Figure 10 shows signals from the pressure transducer at the suction pipe inlet at the operation point C, D and surge. The low frequency pressure fluctuation can be seen clearly at surge.
These indicate that the inlet recirculation is different phenomenon from surge and the inlet recirculation itself is unsteady phenomena with the broadband spectra in frequency.

Figure 8. Static pressure variation during surge (above) and velocity distribution in central cross section along axis of pipe during surge (right) [6].

Figure. 9 FFT result at pressure transducer i, iii and v in figure 6(a).
3. Flow in Vaneless Diffuser

3.1. Flow Structures of Rotating Stall in a Vaneless Diffuser

CFD has difficulties to predict a minimum stable operational flow rate of a centrifugal compressor accurately and to simulate unsteady flow fields occurring below that minimum flow rate without a high computational cost. The application of EFD becomes indispensable to overcome these problems. This section picks up a measurement of rotating stall in a vaneless diffuser with PIV.

As the mass flow rate is reduced, flow instability develops in a centrifugal compressor. The location where instability occurs varies with its aerodynamic design [7]. The rotating stall in a vaneless diffuser is focused on in this section. The rotating stall in a vaneless diffuser is a phenomenon that is a circumferential periodic flow pattern rotating in the diffuser passage at a sub-synchronous impeller rotation speed [8]. Jansen demonstrated that a local reverse flow in a vaneless diffuser triggered the rotating stall [9]. Senoo et al. applied 3-D boundary layer analysis to a vaneless diffuser and proposed the critical flow angle at the inlet of vaneless diffuser [10~12]. Nishida and Kobayashi et al. modified Senoo’s critical angle by considering diffuser inlet shapes and widths [13, 14]. Both Senoo’s and Nishida’s idea are widely used as the criteria of stall of vaneless diffusers.

There are many experimental studies on the rotating stall in vaneless diffusers via unsteady wall pressure measurements. Bianchini et al. equipped with 24 pressure probes along the diffuser circumference for the transient analysis of both the stall inception and the surge onset [15]. Studies with velocity measurement inside the diffuser passage using hot wire anemometry are reported by Fujisawa et al. [16]. They investigated rotating mechanism of diffuser stall via experimental and computational analyses in detail. There are a few experimental studies to visualize the rotating stall pattern with PIV. Dazin et al. measured the unsteady phenomena developing in a vaneless diffuser of a radial flow pump by using PIV coupled with unsteady pressure transducers [17]. Because of the limited area of optical window, an averaged flow field in a reference frame rotating with the instability was derived by synchronizing the PIV measurements with the rotation of instability obtained by unsteady pressure measurements.

The present study selected a turbocharger compressor for marine diesel engines as a realistic measurement target. Table 2 shows the main parameters of compressor investigated here. The outer radius of the impeller is \( r_2 = 65.4 \text{mm} \) and it has seven full blades and six splitter blades respectively. The vaneless diffuser width was not pinched to form a rotating stall easily.

**Table 2.** Dimensions of tested compressor.

| Impeller       | \( r_1 \)/\( r_2 \) | \( r_{1b} \)/\( r_{1s} \) | \( b_2 \)/\( r_2 \) | \( \beta_{b2} \) |
|---------------|-------------------|----------------|-----------------|-----------------|
|               | 0.70              | 0.29           | 0.16            | 30 deg.        |

| Diffuser      | \( r_3 \)/\( r_2 \) | \( r_e \)/\( r_2 \) | \( b_4 \)/\( b_2 \) |
|---------------|-------------------|-----------------|-----------------|
|               | -                 | 1.64            | 1.0             |
Figure 11 shows the tested compressor with positions of pressure transducers, a sight glass and gap sensor. A volute of the investigated turbocharger compressor was replaced with a collector to facilitate the installation of the sight glass for PIV and pressure transducers. Figure 12 shows the static pressures at different radial positions at different flow rates in the vaneless diffuser (A to D in figure 11) and pressure recovery between A and D.

![Diagram of Tested Compressor with Positions](image1)

**Figure 11.** Tested compressor with positions of pressure transducers, sight glass and gap sensor.

![Diagram of Static Pressure and Pressure Recovery in Vaneless Diffuser](image2)

**Figure 12.** Static pressure and pressure recovery in vaneless diffuser (a) Static pressure on vaneless diffuser (b) Pressure recovery in vaneless diffuser, $P_A$-ambient pressure, $P_D$ and $P_D$ are static pressures at A and D, $U_2$ and $\rho_{01}$ are impeller tip velocity and stagnation density at compressor inlet respectively.

The number of stall cells can be derived by the frequency of a pressure fluctuation, $f$ [Hz], and a stall propagation time, $\Delta t$ [sec], between two sensors located at the same radius position in different circumferential positions, $\varphi$ [rad].

$$\text{Number of stall cells} = 2\pi f \Delta t / \varphi$$

Figure 13 shows FFT results of unsteady pressures at operating points from (1) to (7) in figure 12. Figure 14 shows the time domain pressure signals from the sensors, A and E, located 60 degrees apart. Derived number of stall cells is superimposed in figure 13. Figure 15 shows the time variations of...
static pressures at different radial positions in the same circumferential position. A kind of frozen structure propagated circumferentially with a constant speed can be confirmed.

**Figure 13.** Frequency distribution of unsteady pressure at different operating points.

**Figure 14.** Time variation of static pressure at A and E at operating point of (6).

**Figure 15.** Time variation of static pressure at different radius positions at (6).

Figure 16 shows the system to lock the phase relation between the impeller blade and rotating stall. Although the spatial measurement area is limited by the size of the sight glass about 10 degrees in terms of circumferential coordinates, considering the rotating motion of flow field as a frozen structure makes it possible to compose the whole image virtually. Three methods to construct the whole flow field were applied here. First one was using a rotating stall pressure signal. PIV was conducted synchronizing with a particular phase of pressure signal of rotating stall. The data obtained here corresponds to a snapshot of the flow field along 10 degrees in terms of circumferential coordinates. Then PIV was conducted synchronizing the phase signal which was shifted by 10 degrees. The snapshot thus obtained indicates the flow field in the next 10 degrees from the previous snapshot. Repeating this process 35 times, 360-degree snapshot in a circumferential direction was constructed. This corresponds to the flow in the rotational frame of the stall cell.

Second was using the blade position. When an impeller blade pass through the gap sensor, a square wave pulse of B in figure 16 is generated, and the blade position can be detected. Since the rotational speed of impeller is known, it is possible to construct 360-degree flow field by the same way as first one. This provides the flow field synchronized with the rotational speed of impeller, the flow in the rotational frame of the impeller.

Third was using both signals from the pressure transducer and gap sensor, phase locking measurement for both rotating stall motion and impeller rotation. The pulses from the pressure sensor of A and the gap sensor B in figure 16 are superimposed as a signal C. And the signal C was used as the trigger of measurement. The first measurement was conducted when the trigger exceeded the threshold. Then the next measurement was conducted synchronizing the phase signal of the pressure sensor of A which was shifted by 10 degrees after the trigger exceeded the threshold. Increasing the phase shift of the pressure sensor of A up to 350 degrees made it possible to construct the flow field which synchronized both the motions of rotating stall and rotation of the impeller.
Figure 16. System to lock phase relation between blade and rotating stall.

Figure 17. Velocity and flow angle distribution at 50% span at (6) (left: velocity, right: flow angle).

Figure 17(a) shows the velocity and flow angle distribution constructed by using the rotating stall signal. The rotating stall structures, high velocity areas with high flow angle, low velocity areas with low flow angle called stall cells, can be observed. Figure 17(b) shows the velocity and flow angle distribution constructed by using the impeller blade position. Jet and wake structure can be observed. Figure 18 shows the flow field constructed by using both of the rotating stall signal and impeller blade position. The flow field with the rotating stall and jet-wake structure can be visualized at the same time.
Figure 18. Velocity and flow angle distribution at 50% span at (6), phase lock to rotating stall and impeller blade (flow angle is measured from circumferential direction).

Figure 19. Span-wise flow angle distribution and 0 degree-flow angle surface (flow angle is measured from circumferential direction).

Figure 20. Spanwise averaged velocity vector and flow angle (flow angle is measured from circumferential direction) [18].

Figure 19 shows span-wise flow angle distribution and 0-degree flow angle surface which corresponds to the boundary of reverse flow and forward flow. Figure 20 shows the spanwise averaged velocity vector and flow angle. These figures indicate the following flow features of rotating stall in the vaneless diffuser. The blockage caused by the stall cell which occupies the whole span around the vaneless diffuser exit leads to a reduction in the flow angle at vaneless diffuser inlet on one side of the stall cell and an increase in that on the other of the stall cell. The reduction in the flow angle induces
the flow separation from the inlet to the midst of vaneless diffuser along hub, while the increase in the flow angle promotes the stall recovery. As the result, the stall cell rotates in the vaneless diffuser.

3.2. Flow Fields in Vaneless Diffusers of Two Different Automotive Turbocharger Compressors

In developing a product, there is a strong demand for assessing flow in a real machine, not in a machine which is designed for an experiment. This section introduces measurements of flow in vaneless diffusers of automotive turbocharger compressors.

A size of an automotive turbocharger compressor is usually small. Inserting probes into a compressor passage would cause flow blockage, which will affect the flow structure in the vicinity of probes [19]. The smaller a compressor is, the more severe blockage effect is. Hence, PIV was selected as the method of measurement.

Two different turbocharger compressors, Comp.A and Comp.B, were tested. Figure 21 includes pictures of tested compressor impellers, compressor main dimensions and their characteristics. The flow rates, $Q$, of both compressors’ characteristics in figure 21 are divided by each choke flow rate, $Q_c$, at a maximum speed. Focusing on the compressor characteristics on the four speed lines, the slope of the compressor characteristics of Comp.A near surge is positive but that of Comp.B is negative. Hence the operation of Comp.A is more unstable than that of Comp.B. In fact, the surge limit of Comp.A is located on the right side of that of Comp. B. As a part of activities of checking the reason why this difference occurred, the flows in both vaneless diffusers were measured with PIV and compared.

![Figure 21](image)

**Figure 21.** Pictures of impellers, main dimensions and compressor characteristics [20].

A schematic view of Comp.A is shown in figure 22. The external appearance of Comp.B is almost similar to that of Comp.A.

Figure 23 shows pictures of test stands. Figure 23(a) is the test facility used in “Section 3.1.” and figure 23(b) is that for the present tests. It is easy to imagine difficulties to install measurement instruments to the present compressors, i.e. automotive turbochargers. An impeller rotation signal shown in figure 24(a) was used as a reference signal to determine the position of impeller and measurements similar to the second phase lock method in Section 3.1 were conducted. Figure 24(b) shows examples of measured flow angle distributions at every 10 degrees from the reference position. These were obtained by shifting the measurement instance by time corresponding to 10 degrees of rotation time of the impeller. After 360-degree data in circumferential direction was obtained, the data were averaged circumferentially. This averaged data corresponds to the time averaged data per one revolution of the impeller.
Figure 22. Schematic view of Comp.A [21].

(a) Test facility of Section 3.1
(b) Present test facility

Figure 23. Comparison of size of test facility.

(a) Impeller rotation signal
(b) Flow angles from reference position

Figure 24. Impeller rotation signal and examples of measured flow angles.

Figure 25 shows the spanwise velocity and flow angle distributions at different radial positions at \(Q/Q_c=0.37\). The spanwise velocity and flow angle distribution of Comp.A are more nonuniform than those of Comp.B. The flow angle near shroud of Comp.A increases towards the vaneless diffuser outlet and reaches 80 deg. at 1.5 times of impeller radius. The higher the flow angle in vaneless diffuser is, the more unstable the flow in vaneless diffuser is. Hence, the high flow angle near shroud
of Comp.A due to the high spanwise flow nonuniformity can be considered as one of reasons why the operation at low flow rate of Comp.A is more unstable than that of Comp.B.

Figure 25. Spanwise velocity and flow angle distributions at Q/Qc=0.37 [20].

4. Flow in Radial Turbine
Credibility of radial turbine performance derived from an experiment is not always high due to the unclearness of mechanical losses and turbine exit swirl velocity. CFD sometimes plays a role of checking the validity of an experimental result. Hence the high reliability of CFD is essential. In this section, one of the activities to improve the accuracy of CFD for a flow in radial turbine by using experimental results is introduced.

Figure 26. Spanwise velocity and flow angle distribution downstream of turbine rotor.
Figure 26 shows the spanwise velocity and flow angle distributions downstream of a turbine rotor obtained with 3-hole-yaw meter, LDV and CFD. CFD results are widely different from LDV. The CFD model used here was one vane pitch calculation using periodic boundaries and mixing plane interface between a stator and rotor without considering a rotor back-face cavity and scallop. Figure 27 shows a picture of test rig with LDV and a schematic view of tested radial turbine and measured plane. The measurement plane is 95mm axially downstream from the hub side of the rotor leading edge.

(a) Picture of test rig                    (b) Schematic view of measured radial turbine

**Figure 27.** Test rig and measured plane with LDV.

Figure 28 shows the comparison of axial velocity distributions obtained with LDV and CFD. The low velocity region is observed in the LDV. However, the low velocity region in CFD disappears gradually and vanishes at the measured plane (at 95mm axially downstream from the hub side of the rotor leading edge). CFD failed to reproduce the low velocity region downstream of the rotor.

(a) LDV

(b) CFD

**Figure 28.** Comparison of axial velocity distributions obtained with CFD and LDV at 95mm axially downstream from hub side of the rotor leading edge.

In order to clarify where the discrepancy between LDV and CFD started, the velocity in the rotor at about 25% of blade chord length from the rotor leading edge was measured. Figure 29 shows the absolute velocities and flow angle distributions obtained with LDV. A low velocity region caused by the tip leakage vortex and a high circumferential velocity region due to the leakage jet can be found. Figure 29 also illustrates the flow near the rotor tip by the sketch using the data obtained with LDV. Figure 30 shows the comparison of the circumferential velocities obtained with LDV and CFD. A high velocity region appears in CFD but does not in LDV.

A new CFD model with a back-face cavity and scallop was created as the original CFD model did not include these. Figure 31 shows the circumferential velocity obtained by the model with the back-
face cavity and scallop. The high velocity region disappears and the low velocity region at hub-suction corner observed with LDV can be simulated.

**Figure 29.** Velocity and flow angle distribution obtained with LDV.

**Figure 30.** Circumferential velocity obtained with LDV and CFD.

**Figure 31.** Circumferential velocity obtained with LDV and CFD with back-face cavity and scallop.
Figure 32 shows the streamlines from the back-face cavity and tip clearance obtained with the new CFD model. The contours in figure 32 are entropy distributions at 25% chord and at the rotor exit. The leakage flow driven by the pressure difference between the pressure and suction surface near rotor blade leading edge at hub and scarping flow between the stationary wall called the heat shield and the rotor back-face are induced in the back-face cavity [22]. Besides, the fluid vicinity of the rotor back-face tends to flow radially outward due to the centrifugal force by the rotor rotation [23]. The leakage flow and scarping flow induce vortices which flow out of the cavity and into the rotor passage near the suction surface of rotor blade leading edge at hub. Figure 33 shows the schematic of flow in the back-face cavity. The flow from back-face cavity introduce the high loss region in the rotor. Figure 34 shows the axial velocity distributions at the measured plane. The CFD with back-face cavity and scallop still could not reproduce the low velocity region.

![Figure 32. Streamlines from the back-face cavity and tip clearance.](image)

After these calculations, the attention was paid to the streamlines downstream of the rotor. The streamlines downstream of the rotor cross the periodic boundary three times from the blade trailing edge to the measured plane (“A” in figure 32). It was concerned that some circumferential flow information would be lost gradually every time the flow crossing the periodic boundary. In order to
avoid the loss of the information, meshing was re-examined. Grid lines at the periodic boundary were kept orthogonal as much as possible and the span-wise grid density was increased. Figure 35 shows the axial velocity distributions obtained by the previous meshes and the new meshes. After modification of meshing the low velocity region could be reproduced.

**Figure 34.** Axial velocity at measured plane.

**Figure 35.** Axial velocity at measure plane obtained with previous mesh and new mesh. Both models have a back-face cavity and scallop.

5. **Summary**

Three examples of application of EFD to centrifugal compressors and a radial turbine were introduced in this paper.

(1) The existence of the inlet recirculation was confirmed by three different experimental methods. There are various measurement techniques from a comparative ease and low-cost method like oil flow visualizations to a relatively complex and high cost method like PIV (other techniques are shown in [5]). It is important to select the most effective and efficient method in accordance to an experimental purpose.

(2) Two different experiments concerning flow in the vaneless diffuser were introduced. One was for understanding the flow structure and it was allowed to design and to utilize the test facility which was suitable for the measurement. The other was for measuring the velocities in the automotive turbochargers. In the case of measuring flow properties in a product, locations of measurement and applicable sensors are restricted. Furthermore, complex manufacturing is sometimes required to realize measurements. Due to the difficulties to measure several kinds of flow properties at the same time, it is necessary to focus on measurement items and select the method suitable to them.
(3) As an example of mutually complementary relationship between CFD and EFD, the improvement of accuracy of the CFD modelling by using the measured flow pattern with LDV was introduced.

This paper mainly introduced EFD for understanding flow in the radial turbomachines. In a product development, EFD is essential in the process raising TRL (Technology Readiness Level) to evolve a new technology concept to a proven technology and ensure the effectiveness of application of that concept for the product [24]. And there is a need for sophisticated experimental techniques that can capture the slight changes in efficiency and characteristics produced by new technology concepts because the efficiency of many turbomachines has been improved and saturated.

References

[1] Benser W. A., Moses J. J., 1945, An Investigation of Backflow Phenomenon in Centrifugal Compressors, National Advisory Committee for Aeronautics Report No. 806.

[2] Amman C. A, Discussion in Stahler, A. F., 1965, The Slip Factor of a Radial Bladed Centrifugal Compressor, Journal of Engineering for Power Vol. 87 No. 2, pp 181-192.

[3] Andersen, J., Lindstoem F., Westin F., 2008, Surge Definitions for Radial Compressor in Automotive Turbochargers, SAE Technical Paper 2008-01-0296, pp 1-14.

[4] Harley P., Spence S., Early J., 2014, Inlet Recirculation in Automotive Turbocharger Centrifugal Compressors, 11th International Conference on Turbochargers and Turbocharging, pp 89 -100.

[5] Tamaki H., Unno M., Tanaka, R., Yamaguchi, S., Ishizu, Y., 2016, Enhancement of Centrifugal Compressor Operating Range by Control of Inlet Recirculation with Inlet Fins., ASME. J. Turbomach., Vol.138 No.10, 101010.

[6] Tamaki H., Oouchida S., Unno M., Tanaka R., Yamaguchi S., 2016, Enhancement of Centrifugal Compressor Operating Range by Use of Inlet Fins., IHI ENGINEERING REVIEW Vol.49 No.2, pp 45 - 55.

[7] Cumpsty N. A., 1989, Compressor Aerodynamics, Longman Scientific & Technical, Essex, England.

[8] Braembussche R. V. D., 2019. Design and Analysis of Centrifugal Compressors, ASME PRESS, New York, USA.

[9] Jansen, W., 1964, Rotating Stall in a Radial Vaneless Diffuser, ASME J. Basic Eng, 86, pp 750-758.

[10] Senoo Y., Kinoshita Y., 1977, Influence of Inlet Flow Conditions and Geometries of Centrifugal Vaneless Diffuser on Critical Flow Angle for Reverse Flow, Transactions of the ASME Journal of Fluids Engineering, Vol.99 No.1, pp 98-103.

[11] Senoo Y., Kinoshita, Y. 1978, Limits in Rotating Stall and Stall in Vaneless Diffuser of Centrifugal Compressors. ASME Paper No.78-GT-19.

[12] Senoo Y., Kinoshita Y., Ishida M. 1977, Asymmetric Flow in Vaneless Diffuser of Centrifugal Blowers. Transactions of the ASME Journal of Fluids Engineering Vol.99 No.1, pp104-114.

[13] Nishida H., Kobayashi H., Takagi T., Fukushima Y., 1988, A Study on the Rotating Stall of Centrifugal Compressors (1st Report, Effect of Vaneless Diffuser Width on Rotating Stall), Transaction Japan Society of Mechanical Engineers (B Edition), Vol.54 No.449, pp 589-594 (in Japanese).

[14] Kobayashi H., Nishida H., Takagi T., Fukushima Y., 1990, A Study on the Rotating Stall of Centrifugal Compressors (2nd Report, Effect of Vaneless Diffuser Inlet Shape on Rotating Stall), Transaction, Japan Society of Mechanical Engineers (B Edition), Vol.56 No.529, pp 2646-2651 (in Japanese).

[15] Bianchini A., Biliotti D., Tommaso Rubino D., Ferrari L., Ferrara G. 2015, Experimental Analysis of the Pressure Field Inside a Vaneless Diffuser From Rotating Stall Inception to Surge ASME. J. Turbomach. Vol.137 No.11, 111007.

[16] Fujisawa N., Agari Y., Yamao Y., Ohta Y., 2021, Rotating Mechanism of Diffuser Stall in a
Centrifugal Compressor with Vaneless Diffuser, GT2021-59869, Proceedings of ASME Turbo Expo 2021 Turbomachinery Technical Conference and Exposition GT2021 June 7-11, 2021, Virtual, Online.

[17] Dazin A., Cavazzini G., Pavesi G., Dupont P., Coudert S., Ardizzon G., Caignaert G., Bois G., 2011, High-Speed Stereoscopic PIV Study of Rotating Instabilities in A Radial Vaneless Diffuser, Exp Fluids Fluids, Vol.51, pp 83-93.

[18] Ohuchida S., Kawakubo T., Tamaki H., 2013, Experimental Study of Rotating Stall in Vaneless Diffuser of a Centrifugal Compressor, GT2013-95468, Proceedings of ASME Turbo Expo 2013 Turbine Technical Conference and Exposition GT2013 June 3-7, 2013, San Antonio, Texas.

[19] Ma H., Li S., Wei W., 2014, Effects of Probe Support on the Flow Field of a Low-Speed Axial Compressor, J. of Thermal Science, Vol.21 No.2, pp 120-126.

[20] Kiriaki T., Tanaka R., Oouchida S., Yamagata A., 2015, Internal Flow Measurement of a Small Centrifugal Compressor by PIV., 43th conference of Gas Turbine Society of Japan, Yonago, Japan. B-15 (in Japanese).

[21] Tanaka R., Yamagata A., Maruyama S., Yamada K., 2015, Study on Internal Flow in Vaneless Diffuser of a Small-Sized Centrifugal Compressor using PIV., Turbomachinery Vol.43 No.4, pp 215-221 (in Japanese).

[22] He P., Sun Z., Guo B., Chen H., and Tan C., 2013, Aerothermal Investigation of Backface Clearance Flow in Deeply Scalloped Radial Turbines. ASME. J. Turbomach. Vol.135 No.2, 021002.

[23] Morita I., Tamaki H., Unno M., Kawakubo T., 2010, Numerical Analysis on Aerodynamic Effects of Impeller Back Cavity and Disk Shape of Radial Turbine, 9th International Conference on Turbochargers and Turbocharging May 19-20, 2010, London, UK, pp 307-316.

[24] Hughes C E., Zante D. E. V., Heidemann J. D., 2013, Aircraft Engine Technology for Green Aviation to Reduce Fuel Burn, NASA/TM-2013-217690 (AIAA-2011-3531).