INFLUENCE OF DIFFUSER DIAMETER RATIO ON THE PERFORMANCE OF A RETURN CHANNEL WITHIN A CENTRIFUGAL COMPRESSOR STAGE

Jan Bisping  
Institute of Jet Propulsion and Turbomachinery, RWTH Aachen University  
bisping@ist.rwth-aachen.de  
Aachen, Germany

Tim Rossbach  
Institute of Jet Propulsion and Turbomachinery, RWTH Aachen University  
rossbach@ist.rwth-aachen.de  
Aachen, Germany

Daniel Grates  
Institute of Jet Propulsion and Turbomachinery, RWTH Aachen University  
Aachen, Germany

Andre Hildebrandt  
MAN Diesel & Turbo SE  
Oberhausen, Germany

Peter Jeschke  
Institute of Jet Propulsion and Turbomachinery, RWTH Aachen University  
Aachen, Germany

ABSTRACT

This paper describes the influence of the diffuser diameter ratio on the flow and the local loss mechanisms in the return channel of a single-shaft, high flowrate, multistage centrifugal compressor. The analysis, based on detailed measurements and numerical data of two diffuser diameter ratios, enables a deeper insight into the changed flow phenomena.

The diffuser diameter ratio varied between $d_4/d_2 = 1.75$ and $d_4/d_2 = 1.55$. The same design of the return system was used for both diffuser diameter ratios. Comprehensive experimental data was measured for a diffuser diameter ratio of $d_4/d_2 = 1.75$ on a single-stage centrifugal compressor test rig. As such, well-validated, numerical simulations complete the basis of this investigation. When the diffuser diameter ratio is reduced, the analysis reveals an operating point-dependent loss of up to 0.6% points stage efficiency, a decrease in the average circumferential flow angle at the stage exit and an increased variation in the flow angle. The flow inside the impeller and the first part of the vaneless diffuser is not influenced by the diffuser diameter ratio. The changes can therefore be linked to a shift in total pressure loss and static pressure recovery from the vaneless diffuser to the return system, so that the aerodynamic load on the return channel vanes increases. In addition, the flow conditions at the leading edge of the return channel vanes are influenced by the change in the diffuser diameter ratio, which leads to higher, negative incidence in the hub region. A decreased solidity of the return channel vanes results in less flow turning and intensified secondary flow, which influences the circumferential flow angle at the stage exit.

INTRODUCTION

When the diffuser diameter ratio is reduced, the overall stage efficiency depends on a shift in total pressure loss and static pressure recovery from the diffuser to the return system. The aim of reducing the diffuser diameter ratio is to decrease the manufacturing costs without diminishing the efficiency or operating range of the stage. This process will be done in two steps. First, the diffuser diameter ratio is reduced from $d_4/d_2 = 1.75$ to $d_4/d_2 = 1.55$ and its influence analyzed. The second step will be then to design a novel and improved return channel which is based on the knowledge acquired. This paper covers the results of the first phase.

The investigations are based on detailed measurements on a full stage centrifugal compressor test rig, previously described by Rossbach et. al. [1] and Rube et. al. [2], with a diffuser configuration of $d_4/d_2 = 1.75$ and numerical simulations of both diffuser configurations, $d_4/d_2 = 1.55$ and $d_4/d_2 = 1.75$. The numerical setup is validated against the measurement data. Without measurement data, Franz et. al. [3] used a simplified numerical domain to investigate the loss mechanisms in the return channel with a diffuser diameter ratio of $d_4/d_2 = 1.75$.

Linder’s [4] earlier examinations showed the influence of the diffuser diameter ratio on the overall performance. A
In order to overcome efficiency losses generated by a reduced diffuser diameter ratio, this paper investigates the influenced flow phenomena. The insights gained will be used to design a novel return channel for a reduced diffuser diameter ratio.

**METHODOLOGY**

**Experimental Setup**

The experimental investigations were conducted on a closed loop, centrifugal compressor test rig at RWTH Aachen University. The test rig features a single-stage of a single-shaft, multistage compressor consisting of a shrouded impeller, a vaneless diffuser, a U-bend and a vaned return channel, as illustrated in figure 1. The analyzed stage is designed for high flow rates with a design flow coefficient of $\varphi = 0.15$. Air is used as the working fluid and the inlet conditions of the stage were regulated to be a pressure of 1 bar and a constant total temperature of 295 K.

![Figure 1 Cross Section of the Test Rig](image)

The shrouded 3d-impeller consists of 15 blades and has an outer diameter of $d_2 = 0.4$ m, which leads to a design circumferential Mach number of $M_{\text{circ}} = 0.87$. The rated point is equivalent to a total pressure ratio of $\pi_t = 1.57$. The current parallel walled, vaneless diffuser has a diffuser diameter ratio (DDR) of $d_d/d_2 = 1.75$ and will be reduced to a value of $d_d/d_2 = 1.55$ in a future test campaign. The 14 return channel vanes are cylindrical with an inlet metal angle of approximately 26° and an outlet angle of 94° measured from tangential. The shortened diffuser ratio features downscaled vanes with a slightly reduced inlet angle. The solidity of the 14 downscaled return channel vane decreases because of their position on a lower radius. Beyond that, no further design variations are made. After passing through the diffuser, the flow enters the return system. The fluid is redirected by the U-bend to face radial inwards and enters the vaned return channel. The flow leaves the stage through a final L-bend.

The diffuser is instrumented with pressure tabs at different diffuser radii on the hub and shroud side. In the return system, many measurements are conducted including wall pressure and pressure distribution of the vanes. The diffuser and the return system exhibit access for five-hole probes.

In both diffuser configurations, the measurement plane 33 is positioned at a fixed radius. Whereas, planes 39, 5 and 6 are located at the same relative meridional length. Flow profiles at the planes shown were gathered using five-hole-probes.

The measurement uncertainties are specified using propagation of uncertainties proposed by Grabé’s alternative error model [9]. The process and the resulting measurement errors can be found in Rosbach et al. [1]. Rube et al. [2] used the described procedure to calculate a measurement uncertainty of 0.377% for the total-to-total stage efficiency. In this paper, the errors are given in each figure.

**Numerical Method**

To get a better insight into the flow field inside the centrifugal compressor stage, numerical simulations were carried out by using the flow solver FineTurbo of Numeca. Two numerical domains are examined. The first domain contains the complete stage with inlet, impeller and all impeller cavities. The second domain features only the vaneless diffuser and return system. The numerical domain of the full stage (FS) for both DDRs and also for the reduced return channel domain (RCD) are visible in figure 2.

![Figure 2 Numerical Domain](image)

Neglecting the impeller, by using measured diffuser flow profiles as inlet conditions for the return channel domain, avoids the uncertainty of modeling the impeller flow. These flow profiles are available for selected operating points only.

The computational grid is structured with a refinement near all the walls, which leads to $y^+$ values of approximately 1, enabling a low Reynolds treatment of all walls. The grid for a whole stage simulation consists of 30 million cells, whereas the grid without the impeller domain reduces to 7 million cells.
Further refinement did not perceptibly change the numerical results. A full grid connectivity is achieved for the rotor cavities. Sensitivity studies have shown that it is crucial to include all rotor cavities in the numerical domain to get a good match between the numerical and experimental results, in particular in the diffuser. The large impeller fillets at hub and shroud were considered in the numerical domain. A minimum skewness angle of 21° was reached in the impeller domain, which was also the overall minimum cell angle. The grid topology and the number of grid points used for both diffuser configurations \(d_4/d_2 = 1.55\) and \(d_4/d_2 = 1.75\) are the same, except for a reduction of cells in a stream-wise direction in the diffuser for a diameter ratio of \(d_4/d_2 = 1.55\).

All simulations were run at steady state, assuming periodic flow in all passages. A mixing plane (MP) approach was used at the rotor stator interface at a position 80% along the diffuser meridional length, as shown in figure 2. This ensures that a major part of the mixing is captured in the diffuser. The interaction between the impeller and the return channel vanes is weak for both diffuser diameter ratios investigated on account of the large distance between them. The simulations were conducted using a \(k-\varepsilon\) turbulence model. The turbulence model was chosen for stability reasons and is in agreement with the experimental results.

RESULTS AND DISCUSSION

Global Stage Performance

The performance curve for a circumferential Mach number of \(M_{u_2} = 0.87\) is measured for a diffuser diameter ratio of \(d_4/d_2 = 1.75\). The polytropic total-to-total stage efficiency \(\eta_{p,tt}\) and the work input factor \(\psi_{h,tt}\) are plotted in figure 3. Both quantities and the flow rate are normalized by a reference value. The numerical performance curves match the experimental data well. The performance curves for the diffuser with \(d_4/d_2 = 1.55\) are also calculated based on the validated numerical setup. In order to analyze the overall performance, the FS setup is represented by lines in figure 3. In addition, the RCD is displayed using symbols only.

The work input factor \(\psi_{h,tt}\) reveals that the CFD results for a diffuser diameter ratio of \(d_4/d_2 = 1.75\) lie within the measurement uncertainties. In combination with the flow profiles at plane 33, presented later, the figure reveals that the reduction in the diffuser diameter ratio has no detectable influence on the work input, the flow field of the impeller or the first part of the diffuser.

A reasonable agreement between CFD and experiment can also be identified for the stage efficiency \(\eta_{p,tt}\). The trend of the efficiency reveals a match with the efficiency peak almost at the same flow rate. Nevertheless, an offset of up to 3% points is visible. When only the RCD is considered, the offset of the efficiency diminishes to 1.5%, points which reveals a better estimation of the reduced computational domain.

![Figure 3 Stage Performance at \(M_{u_2} = 0.87\)](image)

Considering the full stage setup, the influence of the diffuser diameter ratio on the stage efficiency can be detected near the surge and choke limits. Under choke conditions, the shorter diffuser loses more than 10% points in efficiency, which reduces rapidly to 0.3% points at a flow rate of \(\varphi_{n} = 1.15\). At the design point, both diffuser configurations provide the same efficiency. The efficiency drops again by up to 0.3% points when the flow rate drops further to the surge limit at a flow rate of \(\varphi_{n} = 0.9\). However, the RCD predicts the same trend, but with an intensified loss generation which leads to an efficiency loss of 0.6% points at the measured operating points near choke and surge for the reduced diffuser. The operating point near surge of the FS lead to a different prediction for the change in efficiency than the RCD. The reason for that is a flow separation in the diffuser, shown later in figure 6, which is predicted differently by the two numerical setups.

Before proceeding to make a detailed analysis, the performance will be discussed with respect to compressor components. The differences in the efficiency trend can be explained when the diffuser and the return system are examined separately. Figure 4 shows the total pressure loss coefficient \(\omega\) and the pressure recovery \(c_p\) for the diffuser and the return system. The loss coefficients \(\omega_{33-39}\) and \(\omega_{39-\text{out}}\) are referenced on the dynamic pressure at plane 33, so that a direct comparison is possible.

When the DDR is reduced, the total pressure losses \(\omega_{33-39}\) in the diffuser decrease due to the shortened meridional length. In addition, the fluid has a higher kinetic energy at the exit of the diffuser and losses behind the exit of the diffuser are not taken into account. Further analysis of the diffuser shows that the principle flow structure in the diffuser does not change when the meridional length is reduced. The loss in the rear part of the diffuser is mainly driven by friction. Thus, the accumulated losses reduce because of shortened flow paths. The meridional velocity depends on the radial position in the diffuser. On higher radii the velocity decreases,
which also reduces the losses caused by friction. The reduction in the accumulated loss is therefore not linear: the flow path is reduced to 57.3% of its length and the losses only decrease to 60-62% of their total amount compared to a DDR of 1.75.

As such, a shortened diffuser leads to fewer total pressure losses, but also leads to a reduced pressure recovery of the diffuser $c_{p,33–39}$ 5 because of the reduced diffusion. This means that the flow enters the return system at a higher velocity.

As a consequence, the trends in the return system are reversed, leading to a higher total pressure loss coefficient $\omega_{39–out}$ 2 in the case of a shorter diffuser and an increased pressure recovery $c_{p,39–out}$ 4. A reduction in the DDR leads to a transfer of pressure recovery and also production of total pressure losses to the return system. As a consequence, the operating range, where there is a recovery in static pressure from kinetic energy in the return system, shifts from a flow rate of $\phi_n = -0.95$ to $\sim 1.07$.

In total, the stage efficiency and pressure recovery depends on a trade-off between the reduced losses in the diffuser and the increased losses in the return system. In this particular case, this led to a reduction in overall stage performance near choke and surge. Since the losses not generated in the diffuser and the losses generated in the return system at the design point had the same order of magnitude, the stage efficiency was not affected at this operating point.

**Vaneless Diffuser**

Further analysis will focus on three operating points, the near-surge $\phi_n = 0.79$, the near-choke $\phi_n = 1.13$ and the design point $\phi_n = 1.01$ (marked light light blue in figure 3).

The distribution in the total pressure and the circumferential flow angle of both DDRs at plane 33 are displayed in figure 5. Since the lines of both DDRs are overlapping, the same line style was used. This plane marks the inlet section of the RCD and the experimental data for the flow angles, total pressure and a calculated total temperature are used as inlet conditions for the RCD setup.

In principle, the full stage simulations predict the same qualitative diffuser flow as observed in the experiments. There are no detectable differences between the flow distributions of the two DDRs. As already mentioned, the impeller flow is independent of the diffuser diameter ratio and the upstream influence of the U-bend does not affect the flow at plane 33.

![Figure 5 Flow profiles at plane 33](image)

Figure 6 shows the total pressure $p_t$ and the circumferential flow angle $\alpha$ for both diffuser diameter ratios near surge, where a difference between the full stage and the RCD can be detected. In addition to the remaining uncertainty modeling the impeller flow, the numerical predictions of the two domains are affected by a different extent of a reverse flow zone. The area of reversed flow on the hub side of the diffuser wall is larger in the full stage simulations than in the RCD or the measurements, displayed in figure 6 ①. The experimental data was obtained by using five-hole probes located at 4 different diffuser radii marked with black squares. By increasing the flow rate to design flow rate, the area of small flow angles moves from the hub side to the shroud side, see figure 7. At design point, the difference between the two numerical setups is slightly reduced. Accordingly, for further analysis, only the RCD results are displayed, because of the better matching between this setup and the measurement data.

![Figure 4 Analysis of the stator components](image)
For a detailed explanation of the described flow phenomena in the vaneless diffuser, see Ellis [10], Rebenik [11] and Senoo et. al. [12]. They describe the flow phenomena being based on an exchange of momentum and an induced vorticity, resulting from the meridional curvature of the impeller. The asymmetric flow in the diffuser will be analyzed in a separate publication.

In combination with figure 7 - which shows the total pressure, the Mach number and the circumferential flow angle at plane 39 - the influence of the diffuser diameter ratio on the diffuser can be explained. The Mach number at the end of the diffuser is higher at a diffuser diameter ratio of $d_4/d_2 = 1.55$ because of the reduced diffusion. The reduced meridional length in the vaneless diffuser leads to fewer total pressure losses due to smaller friction losses, so that the remaining total pressure is higher. The circumferential flow angle is only slightly affected, except for the operating point near surge. Near surge, the maximums of the total pressure and Mach number shift towards to hub. The reason for this behavior is the suction effect resulting from the convex hub of the U-bend. This reduces the size of the area of the reverse flow zone at the hub in radial and axial direction, as seen in figure 6 ②. Because of the reduced flow separation, the flow rate at the hub increases and the circumferential angle near the hub rises.

**Return channel**

In addition to the influence of the DDR on the stage efficiency previously mentioned, there is another major impact. The variation of the circumferential flow angle at the exit of the stage increases, see figure 8 (c). This is a direct result of the higher inlet velocity of the return system and the reduced length and increased load of the vanes. All in all, this triggers intensified secondary flows.

The development of the flow conditions at the leading edge of the return channel vanes are illustrated in figures 8 (a) and (b), which show the flow profiles at plane 45 at the top of the U-bend and 5 at the leading edge of the vanes. The pressure gradient inside the U-bend can be approximated by equation 1 [3], where the radius $r$ is measured to the center of the U-bend and $\phi$ is the meridional angle in the U-bend.

$$\rho \left[ \frac{c_1^2}{r} + \frac{c_2^2 \sin(\phi)}{2r^2 \sin(\phi)} \right] = \frac{\partial p}{\partial r} \quad (1)$$

Higher inlet velocities increase the radial pressure gradient, which leads to an additional reduction in static pressure near the hub of the U-bend. This means that the meridional velocity near the hub increases. The circumferential velocity is not influenced by that pressure gradient and is increased or decreased uniformly by conservation of swirl at a given radius. This behavior explains the increased difference in the Mach number between the two DDRs in the hub region, see figure 8 (a) and (b).

In addition, a reduction in the DDR leads to a shift in the area of high total pressure to the hub region ①. This phenomena is based on a shift of mass flow to the hub region because of the high pressure gradient. The effect of the curvature of the convex hub side in the U-bend weakens when
The behavior of the circumferential flow angle inside the U-bend can be explained by looking at the meridional and circumferential velocity separately. The conservation of swirl leads to a more or less uniform increase in the circumferential velocity at plane 5, when the DDR is reduced. The meridional velocity rises above average in the hub region. Due to that, there are two regions of influence found at plane 5. On the one hand, near the hub, the increased meridional velocity results in an increased flow angle (2). On the other hand, near the shroud, the increase of the circumferential velocity is higher than the increase of the meridional velocity, which results in smaller flow angles (3). This behavior leads to a stronger variation in the circumferential flow angle and so to a stronger, negative incidence in the hub region of the vane leading edge.

The metal angle is displayed as black lines in figure 8 (b). It also shows that the reduction in the vane metal angle of the shortened diffuser does not favor a reduction in incidence. For further descriptions of the flow behavior inside a U-bend, see [13-16].

Before describing the flow field inside the return channel passage, the flow profile at the exit of the stage is discussed in figure 8 (c), which shows the circumferential flow angle. The flow of the diffuser with \( \Delta d_2 = 1.55 \) shows a decreased average circumferential flow angle. The metal angle at the trailing edge of the vane was kept constant. The higher blade loading and the decreased solidity lead to less flow turning at the stage exit, which reduces the average circumferential flow angle at plane 65. The second observation is that the under- and overturning resulting from secondary flow in the return channel is also increased, which is visible for the middle part of the channel height. The overturning near the hub is reduced because of the reduced average flow angle. When the flow rate is increased, the effect of the DDR on the variation in the flow angle diminishes. The effect is therefore strongest at the operating point near surge, but is also visible at the design point. The increased variation has a negative impact on a subsequent stage of a multistage compressor and so also reduces its efficiency.

The driving flow phenomena for these effects will only be discussed in detail for the design point. However, the influences described are valid for all operating conditions. A good insight into the flow behavior is provided by the normalized pressure distribution of the return channel vanes presented in figure 9 for a relative channel height of 10% and 90%. The measurement data reveals an agreement with the numerical prediction, taking into account a DDR of 1.75. Only the incidence behavior at the leading edge is underestimated. For a DDR of 1.55, the behavior of the return channel vane changes. In addition to the flow turning, the shortened return channel vanes also recover a considerable amount of static pressure, as can be seen in figure 9. This is in good agreement with the characteristic of pressure recovery mentioned previously, where the point of positive pressure recovery moves to higher flow rates when the DDR is reduced to 1.55 (see figure 4). Further, the static pressure at the exit of the vane is slightly lower when the DDR is reduced so that the decelerating of the flow ends almost at the same level of static pressure.
Figure 9 Pressure distribution return channel vane

As can be seen in figure 8 (b) at the design point, the incidence is negative for both channel heights. In the hub region the incidence is increased and in the shroud region the incidence is reduced. Because of the reduced solidity, for a DDR of 1.55 the pressure gradient between the suction and pressure side increases notably, which leads to an intensified blade-to-blade pressure gradient and thus to stronger secondary flows. The development of secondary flows in a return channel of DDR = 1.75 will be discussed in a separate publication.

The changes in the secondary flow for a reduced DDR can be explained by the three dimensional visualization of the flow at the stage exit in figure 10. This figure features the rear part of the centrifugal compressor stage, which includes the trailing edges of two return channel vanes, plane 6 and plane 65. The surface of one vane and the hub of the flow channel are colored in grey. On plane 6 and 65, contours of streamwise vorticity are shown. Part of the hub shows the contour of static pressure. The black streamlines represent surface streamlines. The colored streamlines are volume streamlines. The static pressure is shown at the hub (1). At the trailing edge at plane 6, the reduction in the DDR to 1.55 leads to a higher pressure gradient from the suction to pressure side. In the case of a DDR of 1.75, the pressure gradient between the suction and pressure side is lower and the gradient is mainly dominated by the hub-to-shroud pressure gradient which is imposed by the L-turn. The flow of the DDR = 1.55 at the trailing edge is therefore more influenced by the blade-to-blade pressure gradient than by the hub-to-shroud pressure gradient, which influences the secondary flow. In plane 6, the variations between the differently developed secondary flows are visible. One major difference is that the hub passage vortex intensifies (2). This results from the larger pressure gradient between the suction and pressure side. The shroud-side passage vortex is weak for both diffuser configurations. For both diffuser configurations, the shroud-sided boundary layer is accelerated towards the suction side (7). However, the shroud-sided passage vortex is not fully developed for both cases. For the shorter diffuser the acceleration is stronger. The second difference is the presence of a horseshoe vortex (3) in case of DDR = 1.55, which is formed at the leading edge at the hub and passes along the suction side of the vane. As a result of the decreased meridional length through the passage and an increased incidence at the leading edge of the vane, a stronger horseshoe vortex develops. For a DDR of 1.75, there is a corner separation located at position (4). This separation is visualized by streamlines, which indicate the area of reversed flow. The separation diminishes considerably, reducing the DDR to 1.55. The stronger passage vortex transports more fluid into that region and the flow is reenergized. The positive and negative stream-wise vortices located at the trailing edge are regions where the fluid flows around the trailing edge and follows the pressure gradient, which is imposed by the L-turn. The changes in the secondary flow structure at plane 6 also lead to a different flow pattern at plane 65. The intensified hub-side passage vortex (2) is still detectable at plane 65. The flow path of this vortex can be traced by the light blue streamline.

Figure 10 Secondary flow structure at design point
For a DDR of 1.75 the hub-side passage vortex is connected to the area of negative vorticity values located at the trailing edge [6]. This vorticity is weakened, when the DDR is reduced. The dark blue streamline shows that the location moves slightly to a lower radius. The shroud-side boundary layer is weak for both diffuser configurations and feeds an area of positive vorticity [5]. In the case of \( d_4/d_2 = 1.55 \), the area of positive vorticity increases. This is because it is supplied not only by the positive part of the vorticity located at the trailing edge, but also by the remaining horseshoe vortex. The path of the horseshoe vortex is marked by a yellow streamline. The pink streamline shows that the second part of the vorticity has its origin at the trailing edge in the shroud region of plane 6.

The white line at plane 65 visualizes a flow separation at the junction of the L-turn and the axial part of the stage exit. The separation recedes, when the DDR is lowered. This can be explained by the fact that the hub-side passage vortex collects an enlarged part of the low momentum fluid of the secondary flow inside the return channel passage. The hub-side passage vortex creates an area of increased blockage in the middle of channel, so that areas of higher velocities are relocated towards the shroud. For that reason the relocation of parts of the low energy fluid leads to smaller area of reversed flow.

The stronger secondary flow is the result of an increased undesirability in the region from 30 to 60% of the channel height, as seen in figure 8 (c). The overturning near the hub is decreased. This is also the case for the operating points near surge and choke. Near Surge, those phenomena are even more marked than at the design point. All in all, the skewed flow profiles at plane 65 can be linked to intensified secondary flows.

CONCLUSIONS

This paper describes a detailed investigation of flow behavior in a return channel when subjected to different diffuser diameter ratios and reveals the impact of this geometric change on the stage performance and the development of secondary flows. The numerical setup of the return system matches the experimental data well, but has the shortcoming that this setup needed detailed measurement data as an inlet condition. The analysis of the stator components revealed a shift in total pressure loss and static pressure recovery from the vaneless diffuser to the return system, which led to a decrease in stage efficiency of up to 0.6% points in a major part of the operating range.

The flow inside the impeller and the first part of the diffuser up to plane 33 was not influenced by a change in the DDR. The reduced total pressure loss in the diffuser was mainly driven by the reduced meridional length, where losses can take place. In addition, the fluid has a higher kinetic energy at the exit of the diffuser and losses behind the diffuser exit increase therefore the total pressure losses in the return system. Subsequently, the Mach number of the fluid entering the return system was increased, which led to a higher, negative incidence in the hub region at the leading edge of the return channel vanes. The shortened vanes experienced a higher load because of a reduced solidity and increased Mach number. The blade-to-blade pressure gradient therefore increased, resulting in intensified secondary flow. The strength and size of the hub-side passage vortex increased. The remaining horseshoe vortex at the trailing edge of the vane for a DDR of 1.55 enlarged the area of positive vorticity. Finally, this led to a reduced average circumferential flow angle at the exit of the stage. In addition, the variation in the flow angle increased, which was considered to be a negative impact on the performance of a possible second stage in a multistage centrifugal compressor.

A novel return channel will therefore be designed as a result of the findings brought about by this paper. To this end, a numerical optimization will be carried out to compensate for the negative impacts on the reduction of the compressor size.

NOMENCLATURE

\( \alpha \) Absolute circumferential flow angle, measured from tangential [\(^\circ\)]
\( d_2 \) Diameter at impeller exit [m]
\( d_4 \) Diameter at diffuser exit [m]
\( d_4/d_2 \) Diffuser diameter ratio [-]
\( \eta_{\text{p,tt}} \) Polytropic total-to-total stage efficiency [-]
\( \phi = \frac{4v_\text{lin}}{\pi d_2 u_2} \) Flow rate [-]
\( \psi_{h,tt} = \frac{\Delta h_{\text{ht}}}{u_2^2} \) Work input factor [-]
\( M_{u2} \) Circumferential Mach number [-]
\( c_p = \frac{p_{\text{out}} - p_{\text{in}}}{p_{\text{lin}} - p_{\text{in}}} \) Static pressure recovery [-]
\( \omega = \frac{p_{\text{lin}} - p_{\text{out}}}{p_{\text{lin}} - p_{\text{in}}} \) Total pressure loss coefficient [-]
\( p_t \) Total pressure [Pa]
\( Ma \) Mach number [-]
\( h \) Normalized channel height, zero at hub [-]
\( \pi_{\text{tt}} \) Total pressure ratio [-]
\( y^+ \) Dimensionless wall distance [-]
\( s \) Relative chord length [-]
\( \text{RCD} \) Numerical return channel domain
\( \text{DDR} \) Diffuser diameter ratio
\( \text{FS} \) Full stage domain
\( c_m \) Meridional velocity [m/s]
\( c_u \) Circumferential velocity [m/s]
\( r \) Radius measured to the center of the U-Bend [m]
\( \phi \) Meridional angle in the U-bend [\(^\circ\)]
\( \Omega_{\text{sw}} = \nabla \times \vec{v}/|\vec{v}| \) Stream-wise vorticity [1/s]

Subscripts

\( n \) Normalized quantity

ACKNOWLEDGMENTS

The authors would like to gratefully acknowledge the German Federal Ministry of Economics and Technology (BMWi). Parts of the work were conducted in the framework...
of the research program "AG Turbo: Optimierter Radialverdichter". Simulations were performed with computing resources granted by RWTH Aachen University under project rwth0255. The authors would like to thank MAN Diesel & Turbo SE for their support and also for granting permission to publish results.

REFERENCES


