NUMERICAL INVESTIGATION OF A CENTRIFUGAL
COMPRESSOR WITH VARIOUS DIFFUSER GEOMETRIES FOR
FUEL CELL APPLICATIONS

M. Schödel - M. Menze - J. R. Seume

Institute of Turbomachinery and Fluid Dynamics, Leibniz Universität Hannover
An der Universität 1, 30823 Garbsen, Germany
schoedel@tfd.uni-hannover.de

ABSTRACT

To meet the air supply requirements for automotive fuel cell applications, the centrifugal
compressor supplying the fuel cell needs a wide operating range. This paper describes an
experimentally validated numerical study of a centrifugal compressor with a variable-vaned
diffuser geometry, which accomplishes a wide operating range. The geometric variability is
achieved by adjusting the angle of the diffuser vanes. In real applications, this would require
gaps between the diffuser vanes and end walls. A numerical investigation of diffusers with
different vane angles with and without gaps considers the negative influence of these gaps on
the flow which results in a reduction of the maximum compressor efficiency by up to 2.75
percentage points. Furthermore, the compressor performance is compared numerically with
various fixed-vaned diffuser geometries, which are designed to shift the compressor map
towards lower mass flows and accordingly increase the surge margin by up to 44.9 percent.
These measures yield an increase in efficiency at high speeds and a reduction at low and medium
speeds.

KEYWORDS

FUEL CELL AIR SUPPLY, CENTRIFUGAL COMPRESSOR, RANGE EXTENSION

NOMENCLATURE

CI Confidence Interval
DS Design speed
FC Fuel cell
LE Leading edge
NS Near stall
OP Operating point
PV Pivoting vanes
SM Surge margin
TE Trailing edge
n Compressor speed
A Cross-sectional area
$p$ Static pressure
$c_p$ Pressure rise coefficient
$p_t$ Total pressure
$DP$ Design point
$th$ Throat
$h_{rel}$ Relative vane height
$\beta_D$ Diffuser vane angle
$i_D$ Diffuser incidence
$Pi$ Total Pressure ratio
$Ma$ Mach number
$\theta$ Circumferential angle
$m_{corr}$ Corr. mass flow rate
$2$ Impeller outlet plane

INTRODUCTION

Due to legal regulations and the global climate crisis, automobile manufacturers are increasingly
researching alternative drive concepts for future vehicles in order to meet CO$_2$ and other emission
limits. Fuel cell (FC) electric vehicles are a promising approach to achieve long driving ranges while
keeping pollutant emissions extremely low. By operating polymer electrolyte membrane FCs in
automotive applications at a high system pressure, the entire stack can be designed more compactly
and the FC operated more efficiently (Cunningham et al., 2001). In contrast to hydrogen (which is
provided for the anode side by high-pressure tanks), the ambient air for the cathode side must first be
compressed. A particular focus of current research is therefore the further development of the FC air
supply that determines the cathode pressure and has the highest parasitic power consumption of the entire FC system (Venturi et al., 2012). The main component of the air supply is either a displacement machine, or nowadays often an electrically-driven compressor that is usually designed to be centrifugal. Due to the high part-load efficiency of FCs, their electric air compressors are more often operated in the part-load range in contrast to turbocharger compressors for combustion engines. Thus, the aerodynamic design goals are high compressor efficiency and a wide operating range that covers the specific operating range of the FC in an automotive application and protects the compressor from surge.

The operating range of existing centrifugal compressors can be increased by equipping them with various devices. The choice of a suitable operating range extending device depends on the design and surge mechanism of a particular compressor. Kämmer and Rautenberg (1986) and van den Braembussche (2019) explained that compressor surge can result either from impeller or diffuser stall, independently of the diffuser being equipped with or without vanes. At low speeds, impeller stall is more common, whereas at high speeds diffuser stall is the dominant surge mechanism. The objective of this paper is to improve the stable operating range of an existing compressor and to adjust the operating range to a specific FC application.

The investigations of this paper are part of the research project “Charging of Fuel Cell Systems through Interdisciplinary Developed Electrically-Driven Air Compressors” (German short form: ARIEL). As described by Menze et al. (2019), the objective of the project ARIEL is to develop an electrically-driven compressor for the cathode air supply of a FC, which can be used in the automotive sector. The investigated system consists of a centrifugal compressor with a vaned diffuser, an electric motor with power electronics and a turbine that is used to recover energy from the FC exhaust air and thus to increase the air supply efficiency. Within ARIEL, a wide range of topics is studied, such as: the bearing concept, the manufacturing concept, the power electronics and electric motor design, as well as the aerodynamics of the air supply system. The project ARIEL is based on an existing prototype that is to be optimized for feeding a FC, whose operating line is shown later. In the following section, an overview of measures to extend the operating range for both, impeller and diffuser will be presented. However, it is worth mentioning that the numerical investigations of this paper focus on improving the diffuser flow.

OPERATING RANGE EXTENDING DEVICES FOR CENTRIFUGAL COMPRESSORS

Casing treatments are most often used to suppress impeller stall. While circumferential grooves and axial slots are more commonly installed in axial or mixed flow compressors, centrifugal compressors mainly use recirculation channel casing treatments with one port upstream and one port downstream the impeller leading edge (LE). Sivagnanasundaram et al. (2010) and Christou et al. (2016) described that recirculation channel casing treatments stabilize the impeller flow by reducing the blockage and thus the incidence in the shroud region upstream of the impeller. Furthermore, they weaken the impeller tip gap vortex through the extraction of low-impulse fluid near the downstream port. However, due to the recirculation of already compressed fluid and mixing losses, recirculation channel casing treatments have a negative influence on the efficiency of the compressor (Sivagnanasundaram et al., 2010). Herbst and Eilts (2015) and Fischer et al. (2018) investigated a variable inlet geometry which was implemented by a cone geometry or an orifice plate. In both studies a shift of the surge line was achieved by improving the impeller inflow. In addition, variable inlet guide vanes can be used to influence the compressor operating range and shift the surge line towards lower mass flow rates by reducing the impeller incidence (Herbst and Eilts, 2015).

According to Fischer et al. (2018), a variability of the diffuser width is an effective measure for unvaned diffusers to extend the compressor operating range. They combined the above-mentioned variable orifice plate upstream of the impeller with a variation of the diffuser width and state that a reduction of the diffuser width has a positive effect on the surge margin (SM) of a low-trim compressor, whereas it has only a small effect on the SM of a high-trim compressor.
To improve the operating range of a centrifugal compressor with a vaned diffuser, Senoo et al. (1983) and Eynon and Withfield (1998) recommended the use of a low-solidity diffuser. The solidity is the ratio of the vane chord length to the spacing between the vanes. It is usually in the order of 0.7 for low-solidity diffusers. The absence of a throat allows a more stable flow near stall (NS), higher choke mass flows and thus a wider operating range. Ebrahimi et al. (2017) or Wöhr et al. (2017) researched the concept of a variable adjustment of the diffuser vane angle in bigger machines, like heavy-duty truck engine turbochargers. They reported that the SM and even the choke limit can be improved significantly, which in turn increases the stable operating range of the compressor. This measure is usually implemented by pivoting the vanes, although other actuation mechanisms are also possible. However, for a pivoting vanes (PV) concept there must be gaps between the vanes and the diffuser end walls that cause aerodynamic losses and thus reduce efficiency. Eynon and Withfield (1998) highlighted with their study the considerable influence of the diffuser LE angle on the stable operating range and the efficiency of a compressor. They emphasized that in a compressor with variable diffuser vanes, attention must also be paid to the matching of the diffuser outlet angle with the volute design. Guandal and Govardhan (2019) investigated leaned diffuser vanes with a non-uniform LE angle over the vane height. They recognized a shift of the operating lines towards lower mass flow rates due to the reduced incidence and the decrease of the throat area towards the shroud. Casing treatments can also be used in the diffuser or at the impeller exit to enhance the operating range. On the one hand, Bareiß et al. (2015) investigated circumferential grooves above the impeller trailing edge (TE). They found that the grooves weaken the impeller tip gap vortex and thus improve the diffuser inflow and the compressor SM at design speed (DS). On the other hand, Galloway et al. (2018) proposed diffuser recirculation techniques similar to the impeller recirculation channel casing treatment to achieve a wider compressor operating range. Other authors, such as Herrmann et al. (2020) investigated an end wall contouring at the diffuser hub and achieved a slight improvement in SM without an efficiency penalty.

INVESTIGATED GEOMETRY VARIATIONS

Previous investigations of centrifugal compressors with a vaned diffuser have shown that a PV concept has the greatest potential for a variable adjustment of the operating range. Therefore, in this study a PV diffuser is designed for the ARIEL compressor. For this purpose, the diffuser vanes are shortened in their chord length to allow a more flexible pivoting around their central axis without collisions with the volute or the impeller (see Fig. 1). A diffuser solidity of one results for the PV diffuser. The center of rotation for the PV concept is set at the mean radius between vane LE and TE. According to Aungier (2000), the cross-sectional area of the volute at a certain circumferential position \( \theta \) should be proportional to the outflow angle of the diffuser. To account for the decreased diffuser outflow angles when setting a negative pivot angle, a new volute with a linear cross-sectional area progression and a smaller outlet area is developed. Figure 2 shows the area progression for the PV and the baseline volute. Due to the linear area growth, an almost uniform diffuser load over the circumference should be achieved.
In the first part of the investigations, the PV concept is implemented without gaps between diffuser vanes and end walls. However, since a PV concept in a real machine requires gaps between the vanes and the end walls to allow the vanes to pivot freely, on both walls a gap of 2.8 % of the vane height is set in the second part of the investigations (without considering the geometry of an actuating mechanism). To demonstrate that significant improvements in the SM can also be achieved by a measure with fixed-geometry, in the last part of the numerical investigations, only the LE angle of the baseline diffuser is adjusted while maintaining the baseline diffuser solidity.

**NUMERICAL SETUP**

![Figure 3: Numerical setup (mixing plane setup)](image)

Within this paper, steady state simulations with the commercial software ANSYS CFX 19.2 are carried out. Figure 3 visualizes the numerical setup, consisting of the domains inlet pipe, impeller, backside cavity, diffuser, volute and outlet pipe. The inlet pipe and the backside cavity as well as the impeller that contains a main and a splitter blade are modeled as single passage domains with periodic boundary conditions. To consider the circumferential asymmetry of the volute, diffuser, volute and outlet pipe are modeled as full 360-degree domains. Mixing plane interfaces are used between inlet pipe and the rotating domain impeller as well as between impeller and diffuser. In contrast, the connection between impeller and backside cavity is implemented using a frozen rotor interface. While the inlet boundary condition is specified by the total pressure and total temperature, the mass flow is specified at the outlet. There is a second outlet with a mass flow boundary condition, which allows one percent of the main mass flow to leave the backside cavity. The described setup is referred to as mixing plane setup in the following.

ICEM CFD 19.2 is used for the meshing of the inlet pipe, the outlet pipe (both structured) and the volute (unstructured). Impeller and diffuser are meshed structured with ANSYS Turbogrid 19.2. Finally, ANSYS Workbench Meshing (19.2) is used for the unstructured meshing of the cavity backside. Maximum y+ values of less than five and average y+ values of less than two are set in order to keep the computational effort as low as possible while ensuring that the cells closest to the wall are within the viscous sublayer of the boundary layer flow. Furthermore, a grid convergence study was performed according to the American Society of Mechanical Engineers (2009) using three different grid sizes. Table 1 shows the number of cells of the selected mesh in each domain as well as the average y+ values at maximum speed and maximum mass flow rate. In total a simulation with 14,688,561 cells results. All simulations are performed using the Wilcox (1988) k-ω turbulence model, since the Menter (1993) k-ω SST turbulence model, which is also frequently used in the field of turbomachinery, often overestimates flow separations.

<table>
<thead>
<tr>
<th>Domain</th>
<th>Inlet Pipe</th>
<th>Impeller</th>
<th>Diffuser</th>
<th>Volute</th>
<th>Outlet Pipe</th>
<th>Backside Cavity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Cells</td>
<td>134,048</td>
<td>2,678,327</td>
<td>3,567,630</td>
<td>7,379,849</td>
<td>288,000</td>
<td>640,707</td>
</tr>
<tr>
<td>Average y+ Value</td>
<td>0.32</td>
<td>0.69</td>
<td>1.04</td>
<td>1.08</td>
<td>1.43</td>
<td>0.52</td>
</tr>
</tbody>
</table>
The convergence of the numerical simulations to RMS residuals of $10^{-4}$ and imbalances of 0.05 % at the maximum as well as the negative slope of the characteristic map and the pressure rise between impeller outlet and diffuser throat (also called the vaneless and semi-vaneless space) are chosen as stability criteria for a particular operating point. As stated by Van den Braembussche (2019), the semi-vaneless space is the critical part of the diffuser in terms of stability. The pressure rise coefficient in the vaneless and semi-vaneless space $c_{p,2-th}$ is defined by equation (1). The maximum value of $c_{p,2-th}$ for a stable operation was given by Elder and Gill (1985) among others and is typically in a range between 0.2 and 0.5. It depends on the diffuser inflow velocity and the specific compressor geometry. Within a certain value range of the wetted surface in the semi-vaneless space and the diffuser throat area, the maximum $c_{p,2-th}$ value is proportional to the wetted surface area in the semi-vaneless space and inversely proportional to the diffuser throat area (Elder and Gill, 1985).

$$c_{p,2-th} = \frac{p_{th} - p_2}{p_{t,2} - p_2}$$  \hfill (1)

In addition to CFD simulations with the mixing plane setup described above, simulations are also carried out with an identical setup, but using frozen rotor instead of mixing plane interfaces and a full 360 degree resolution of all domains. This setup is referred to as the frozen rotor setup. Due to the increased computational effort, it is only used for the experimental validation. In addition, the frozen rotor simulations are not useful for predicting the operating range of the compressor because the same impeller diffuser position is always considered.

**RESULTS**

**Experimental Validation**

Since experimental investigations of the compressor will be conducted later within the research project ARIEL, the CFD results of this paper are validated by experimental results of a similar compressor with the same aerodynamic design, provided by the Volkswagen AG. Figure 4 shows the comparison of the experimental and numerical speed lines of the mixing plane setup at DS and at 60 % of DS. It also includes some operating points simulated with the frozen rotor setup. All sizes in Fig. 4 are relative to the CFD best efficiency operating point at DS (mixing plane setup).

The CFD results with frozen rotor setup agree very well with the experimental results. However, the CFD simulations with mixing plane setup significantly overestimate the total pressure ratio, especially at DS. The deviations can be explained by the reduced interactions between diffuser and impeller as well as between the non-uniform volute pressure field and the impeller flow field due to the mixing plane. These effects result in further total pressure losses in impeller, diffuser and volute compared to the mixing plane setup.

Despite the deviations in the total pressure ratio, the deviations between numerical surge line of the mixing plane setup and experimental surge line are rather small. The experimental surge line is usually at lower mass flow rates than the numerical surge line of steady state simulations, since numerical instabilities near the surge line prevent the convergence of the simulations. For an exact
identification of the surge line, transient CFD investigations would be necessary, which, however, would exceed the scope of this study due to the high computational effort. The paper is based on the assumption that differences between the experimental surge line and the numerical surge line, determined with the mixing plane setup, remain the same when the diffuser geometry is changed. This assumption will be verified in future experimental studies with different diffuser geometry variations.

Baseline Compressor

The entire map and the polytropic efficiency of the baseline compressor are shown together with the FC operating line in Fig. 7 (a) and (b), among other configurations. One may observe that the FC operating line cannot be covered at low speeds and that the SM is small even at DS. That means it is necessary to shift the entire compressor map towards lower mass flow rates.

To evaluate the reasons for the compressor stall, the flow at the NS operating point at DS (OP3) and at 30 % of DS (OP1) is analyzed. Since at DS no major flow separations in the impeller occur at the NS operating point (OP3; not shown here), it can be concluded that at this speed the impeller is not critical for the stability. Looking at the relative Mach number at 30 % of DS (OP1) within the impeller in a meridional and a blade-to-blade view at 80 % of the blade height in Fig. 5, a region with low Mach numbers can be observed near the shroud. This region originates from the impeller tip gap vortex as well as from the high flow incidence in the impeller shroud region. The flow separation near the impeller shroud can contribute to compressor stall at low speeds.

However, there are strong flow separations in the diffuser at both, high and low speeds. Figure 6 shows the Mach number at 20 % and 80 % of the diffuser height for OP3. The diffuser flow-field at OP 1 is similar at a lower overall velocity level (not shown here). The flow separations in the rear part of the diffuser vanes result from the volute design of the baseline compressor that has very large cross sections and a non-uniform widening of the area (see Fig. 2). More critical for the stability is the positive incidence observed at 80 % of the vane height in the diffuser inflow, which causes a flow...
separation near the vane LE. The pressure recovery coefficient in the vaneless and semi-vaneless space of the diffuser $c_{p,2-th}$ is between 0.46 and 0.49 for the different speeds at the NS operating points. Overall, the analysis of the baseline compressor shows a clear optimization potential for the diffuser. This leads to the hypothesis that an improvement of the flow conditions in the diffuser by operating range extending devices can help to increase the centrifugal compressor SM.

**Pivoting Diffuser Vanes**

It is necessary to explain the influence of replacing only the volute (leaving the baseline diffuser unchanged) before evaluating the influence of the PV diffuser together with a redesign of the volute. Redesigning the volute results in a more uniform flow over the circumference and reduces the flow separations in the rear part of the diffuser vanes as well as in the volute. This increases the polytropic efficiency and the total pressure ratio over the entire operating range of the compressor. The maximum efficiency at the different speed lines increases by 0.80 to 0.98 percentage points. However, the operating range remains nearly unaffected by the replacement of the volute.

Figure 7 (a) and (b) show how a hypothetical PV diffuser with no gaps together with a volute redesign influences the compressor map and the polytropic efficiency. By setting a negative pivot angle, the speed lines can be shifted significantly in the direction of lower mass flow rates. This results in an increase of the SM by up to 53.8 % at 60 % of DS. Here, the difference in the SM between baseline and PV compressor is calculated according to equation (2).

$$\Delta SM = \frac{\Pi_{t,NS} \cdot \dot{m}_{corr,Baseline,NS}}{\Pi_{t,Baseline,NS} \cdot \dot{m}_{corr,NS}} - 1 \quad (2)$$

$$\Delta \eta_{pol,max} = \eta_{pol,max} - \eta_{pol,max,Baseline} \quad (3)$$

![Figure 7: Compressor map (a) and polytropic efficiency (b) of the baseline compressor and several pivoting vane configurations](image)

The improvement of the SM indicates that even at low speeds, it is not the impeller but rather the diffuser that is critical to stability. However, it should be mentioned that, especially with large negative pivot angles, the choke mass flow rate decreases through the reduction of the diffuser throat area, which reduces the map width. Positive pivot angles shift the surge line and thus the compressor map towards higher mass flow rates. In total, the FC operating line can be better covered with PV. Only at part-load in the range of very small mass flow rates there are still FC operating points which can’t be reached with the PV compressor map. For small FC mass flows, a bypass control is required accordingly. This means that a part of the compressor mass flow is bypassed from the compressor outlet to the turbine inlet or even blown off into the atmosphere, which reduces the FC system.
efficiency at part-load. As a result, the compressor is able to supply the FC with the required mass flow and pressure, while the compressor can still operate in the stable operating range.

Considering the compressor efficiency, one can find that negative pivot angles reduce the maximum efficiency by up to 4.58 percentage points at 60 % of DS (PV -6°). Due to the volute redesign and by setting a negative pivot angle, the maximum efficiency can even be increased at maximum speed. Table 2 gives an overview of the changes in maximum efficiency and SM due to PV at different speeds.

Table 2: Changes in the surge margin ($\Delta SM$) and maximum polytropic efficiency ($\Delta \eta_{pol,max}$) of several pivoting vane configurations compared to the baseline compressor according to equation (2) and (3)

<table>
<thead>
<tr>
<th>Compressor Speed</th>
<th>$0.3 \cdot n_{DS}$</th>
<th>$0.6 \cdot n_{DS}$</th>
<th>$1.0 \cdot n_{DS}$</th>
<th>$1.2 \cdot n_{DS}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>PV +2°</td>
<td>$\Delta SM$ in %</td>
<td>-5.4</td>
<td>-0.3</td>
<td>-8.9</td>
</tr>
<tr>
<td></td>
<td>$\Delta \eta_{pol,max}$ in %</td>
<td>+1.30</td>
<td>+0.51</td>
<td>-0.14</td>
</tr>
<tr>
<td>PV -2°</td>
<td>$\Delta SM$ in %</td>
<td>+12.6</td>
<td>+21.2</td>
<td>+11.2</td>
</tr>
<tr>
<td></td>
<td>$\Delta \eta_{pol,max}$ in %</td>
<td>-1.01</td>
<td>-0.73</td>
<td>-0.25</td>
</tr>
<tr>
<td>PV -6°</td>
<td>$\Delta SM$ in %</td>
<td>+38.6</td>
<td>+53.8</td>
<td>+17.2</td>
</tr>
<tr>
<td></td>
<td>$\Delta \eta_{pol,max}$ in %</td>
<td>-4.03</td>
<td>-4.58</td>
<td>-2.15</td>
</tr>
</tbody>
</table>

Regarding the NS operating point at 60 % of DS (OP2), there is a positive diffuser incidence between 67 % and 90 % of the vane height for the baseline case, as depicted in Fig. 8. By setting a negative pivot angle, the incidence of the diffuser can be significantly reduced over the entire vane height. Consequently, the tendency of the diffuser towards instability is reduced. However, as shown in Fig. 7 (b), the efficiency decreases due to the strong negative incidence over a large part of the vane height.

Gaps between End Walls and Pivoting Diffuser Vanes

As explained above, the concept of PV can only be implemented in real machines by considering gaps between diffuser vanes and end walls. In Figure 9 (a) and (b) one can observe how a gap that is 2.8 % of the channel height (on both walls) influences the compressor map and efficiency. Especially at higher speeds, the gaps reduce the total pressure ratio of the compressor. Furthermore, the peak efficiency decreases by introducing the gaps in the diffuser. At DS there is a resulting loss in peak efficiency of 2.23 percentage points for PV 0° and a loss of 2.75 percentage points for PV -4° compared to the PV configurations without gaps. The SM remains almost constant with the introduction of the gaps at low speeds, but increases slightly at higher speeds. Due to the gaps, 5.6 % of the diffuser channel height is unvaned. In comparison to the fully vaned diffuser, the operating behavior of the diffuser with gaps thus also develops more in the direction of the operating behavior of an unvaned diffuser with a lower total pressure ratio and a lower efficiency.

The circumferentially averaged diffuser incidence which is not shown in this paper is nearly unaffected by the introduction of a diffuser gap. However, it should be mentioned that the effect of the diffuser potential field is not taken into account by the circumferential averaging. If, for example,
the vanes are pivoted or a gap is added to the diffuser, the blockage and thus the diffuser potential field is weakened. This results in a diffuser inflow pressure field that is more uniform over the circumference. Thus, the effective incidence at the LE of the vane can be reduced, while the circumferential average of the incidence is nearly constant. In addition, the diffuser potential field also has an upstream effect on the impeller flow despite the mixing plane. A region of loss near the shroud at the impeller outlet can be reduced by weakening the diffuser potential field either by using PV −4° or by introducing a gap at PV 0°. Finally, it must be noted that a comparison between the PV 0° compressor and the baseline compressor shows almost no change in SM. From these results, it can be concluded that the reduction of vane solidity at PV 0° has only a minor effect on the operating range.

Figure 9: Compressor map (a) and polytropic efficiency (b) of several pivoting vane configurations with and without gaps between diffuser vanes and end walls

Variation of the Diffuser Leading Edge Angle

As a last step, the LE angle of the diffuser vanes is varied. Two configurations with a uniform reduction of the LE angle over the entire vane height and one configuration without a change at the hub, but with a 4° reduction at the shroud and a linear interpolation in between, are compared with the baseline compressor. The latter configuration should take into account the incidence profile of the baseline compressor (see Fig. 8) with positive incidence values, which are critical for stability, only in the upper part of the vanes. In contrast to the PV diffuser, the TE radius and angle of the vanes are not changed in the configurations of this section. Furthermore, the baseline volute is used. The resulting compressor map as well as the compressor efficiency are shown in Fig. 10 (a) and (b).

The uniform reduction of the diffuser LE angle shifts the speed lines towards lower mass flow rates and thus increases the SM by up to 21.7 % for a 2° reduction and by up to 44.9 % for a 4° reduction at 60 % of DS. Therefore, the changes in the LE angle of the diffuser allow a better coverage of the FC operating line. It should be noted that the increase in SM is much smaller at higher speeds. However, changing the LE angle again reduces the compressor peak efficiency by up to 1.43 percentage points for a 2° reduction at 30 % of DS and by up to 2.41 percentage points for a 4° reduction at 60 % of DS. At maximum speed, in contrast, the maximum efficiency increases compared to the baseline compressor.

Due to the non-uniform reduction of the LE angle, an increase of the SM by up to 28.4 % at 60 % of DS can be achieved. The benefit of this configuration is that the loss of efficiency compared to the baseline compressor is lower with a maximum loss of 0.80 percentage points at 30 % of DS. Furthermore, the shift of the choke line towards lower mass flow rates is less significant because the diffuser throat area at the hub is larger than for the configurations with constantly reduced LE angle. Therefore, the compressor map width is increased in comparison to the diffusers with constantly
reduced LE angle. An overview of the changes in maximum efficiency and SM due to LE angle variation at different speeds is given in Tab. 3.

Figure 10: Compressor map (a) and polytropic efficiency (b) of the baseline compressor and several configurations with a variation of the diffuser leading edge angle

Table 3: Changes in the surge margin ($\Delta SM$) and maximum polytropic efficiency ($\Delta \eta_{pol,max}$) of configurations with varied diffuser leading edge angle compared to the baseline compressor according to equation (2) and (3)

<table>
<thead>
<tr>
<th>Compressor Speed</th>
<th>$0.3 \cdot n_{DS}$</th>
<th>$0.6 \cdot n_{DS}$</th>
<th>$1.0 \cdot n_{DS}$</th>
<th>$1.2 \cdot n_{DS}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta \beta_{LE} = -2^\circ$</td>
<td>$\Delta SM$ in %</td>
<td>$+ 12.7$</td>
<td>$+ 21.7$</td>
<td>$+ 4.5$</td>
</tr>
<tr>
<td></td>
<td>$\Delta \eta_{pol,max}$ in %</td>
<td>$- 1.43$</td>
<td>$- 0.83$</td>
<td>$+ 0.19$</td>
</tr>
<tr>
<td>$\Delta \beta_{LE} = -4^\circ$</td>
<td>$\Delta SM$ in %</td>
<td>$+ 28.9$</td>
<td>$+ 44.9$</td>
<td>$+ 19.5$</td>
</tr>
<tr>
<td></td>
<td>$\Delta \eta_{pol,max}$ in %</td>
<td>$- 2.17$</td>
<td>$- 2.41$</td>
<td>$- 0.16$</td>
</tr>
<tr>
<td>$\Delta \beta_{LE} = 0 \ldots -4^\circ$</td>
<td>$\Delta SM$ in %</td>
<td>$+ 12.7$</td>
<td>$+ 28.4$</td>
<td>$+ 6.7$</td>
</tr>
<tr>
<td></td>
<td>$\Delta \eta_{pol,max}$ in %</td>
<td>$- 0.80$</td>
<td>$- 0.55$</td>
<td>$+ 0.24$</td>
</tr>
</tbody>
</table>

The circumferentially averaged incidence at the baseline NS operating point at 60 % of DS (OP2) in Fig. 11 allows the same conclusions. A uniform reduction of the diffuser LE angle reduces the incidence over the entire vane height. This means that the incidence decreases in both, in the part of the vane between 67 % and 90 % of the channel height with an originally positive incidence and in the part of the vane below with a negative incidence. The decrease in incidence in the lower part of the channel has a severely negative influence on the compressor efficiency. By twisting the diffuser vanes, the incidence decrease in the lower part of the diffuser can either be prevented or at least reduced, whereas in the part above 67 % vane height the incidence decrease is nevertheless noticeable. Thus, the efficiency deficit at low speeds is smaller for twisted diffuser vanes than for diffuser vanes with uniform LE angle reduction.

Figure 11: Circumferentially averaged diffuser incidence at the baseline near stall operating point at 60 % of design speed (OP2) for the baseline compressor and varied diffuser leading edge angles
CONCLUSIONS AND OUTLOOK

The performance of different operating-range extending devices in the diffuser of a centrifugal compressor for the air supply of a fuel cell is investigated with respect to surge margin and efficiency. The objective of the investigation is to improve the operating range of the compressor, while compromising the efficiency as little as possible in order to meet the requirements of the fuel cell.

First, it is shown that the diffuser is likely to be a critical component for the stability of the compressor over the entire speed range. A pivoting vane diffuser, which is combined with a redesign of the volute geometry and a reduction of the vane solidity, shows a very high potential for a flexibly adjustable operating range by reducing the diffuser incidence and the pressure rise coefficient in the semi-vaneless space. However, the necessary gaps between diffuser vanes and end walls result in high losses of efficiency. Whether these losses in maximum efficiency are an acceptable trade-off for a wider operating range must be considered for each specific application. The new design of the volute, by contrast, shows a potential for improving the efficiency of the compressor without affecting the operating range.

Mechanically simpler, the surge margin is also increased by choosing a fixed-diffuser geometry and just selecting a new leading edge angle. As a further improvement, a diffuser design with twisted vanes shows the best compromise between improving the surge margin and minimizing the loss of efficiency. If a diffuser design with twisted vanes is combined with a volute redesign, the loss in efficiency should be further minimized or even be prevented.

Overall both, the variable and the fixed-geometry configuration adjust the compressor operating range to match the fuel cell operating line. At very low mass flow rates, fuel cell operation is realized by a bypass. In order to be able to cover the operating line of the fuel cell in the part-load range of the compressor even better and possibly avoid the need for a bypass control, a redesign of the compressor impeller will be considered in further investigations.

The operating range extending devices proposed for the diffuser will be validated in subsequent experimental studies and presented in future publications. A compressor prototype with different static diffuser inserts will then be investigated experimentally.

ACKNOWLEDGEMENTS

The investigations presented in this paper are part of the research project ARIEL. The authors thank the Federal Ministry of Transport and Digital Infrastructure (BMVI) which financially supported the work within the framework of the National Innovation Programme (NIP) Hydrogen and Fuel Cell Technology as well as the NOW GmbH which coordinated the funding guideline. The authors would also like to acknowledge gratefully the entire project consortium consisting of the Volkswagen AG and all the participating institutes of the University of Braunschweig and the Ostfalia University of Applied Sciences.

REFERENCES

American Society of Mechanical Engineers, (2009). Standard for verification and validation in computational fluid dynamic and heat transfer. ASME, New York, USA


Van den Braembussche, R., (2019). Design and Analysis of Centrifugal Compressors. ASME Press, John Wiley & Sons Ltd., Chichester, West Sussex, New York, USA

