Numerical and experimental investigation of a radially reduced diffuser design concept for a centrifugal compressor performance at design point.

https://doi.org/10.1016/j.ast.2022.107590

Published in:
Aerospace Science and Technology

Document Version:
Publisher's PDF, also known as Version of record

Queen's University Belfast - Research Portal:
Link to publication record in Queen's University Belfast Research Portal

Publisher rights
Copyright 2022 the authors. Published by Elsevier Masson SAS.
This is an open access article published under a Creative Commons Attribution License (https://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution and reproduction in any medium, provided the author and source are cited.

General rights
Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

Take down policy
The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact openaccess@qub.ac.uk.
Numerical and experimental investigation of a radially reduced diffuser design concept for a centrifugal compressor performance at design point

Laura McLaughlin a,∗, Stephen Spence a, Daniel Rusch b, Lee Galloway c, Marco Geron c, Kwok Kai So b, Magnus Fischer b

a Trinity College Dublin, Dublin, Ireland
b ABB Switzerland AG, Turbocharging, Baden, Switzerland
c Queen’s University Belfast, Belfast, United Kingdom

A R T I C L E   I N F O

Article history:
Received 21 September 2021
Received in revised form 28 January 2022
Accepted 22 April 2022
Available online 28 April 2022
Communicated by Kivanc Ekici

Keywords:
Aerodynamics
CFD
Compressor
Optimization
Experimental
Visualization

A B S T R A C T

Large scale, high pressure ratio (PR) centrifugal compressors are commonly made up of a radial impeller with a vaned diffuser. In research to date, the majority of research and design has focused on extending the operating range or improving the compressor efficiency. However, a cost and weight saving can be achieved by reducing the compressor dimensions. This is a study of a vaned diffuser design concept aiming to reduce the overall radial dimensions of the compressor without any sacrifice of compressor performance at design point (DP).

The study parameterized a diverging endwall diffuser concept and optimized it to achieve a similar performance within a reduced radial outlet dimension. A metamodel assisted Multi Objective Genetic Algorithm (MOGA) method has been used. A numerical approach has been used to investigate how the flow physics within the diffuser passage changed with the new geometry and detailed experimental measurements have been used to validate the numerical approach. The result of this study is a summary of the impact of vane geometry defining parameters within a diverging endwall concept which offers design guidance for application of the concept. The measured performance showed that the same efficiency performance was achieved within a 15% radially reduced diffuser passage using the diverging endwall concept.

© 2022 The Author(s). Published by Elsevier Masson SAS. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/).

1. Introduction

Minimizing the size of a centrifugal compressor is often not the main priority for high PR centrifugal compressor design. Conventional applications of high PR centrifugal compressors are not usually limited for space, e.g. marine applications, and so the typical design process has focused on improving compressor efficiency or increasing the operating map width (MW). However, future applications in advanced aerospace and marine propulsion systems will bring greater emphasis on weight and cost. By achieving a modest percentage reduction in the overall dimensions of a high PR centrifugal compressor there are significant size, weight, and cost reduction benefits. The result is a high PR centrifugal compressor more applicable to aerospace applications. Fig. 1 shows a meridional sketch of the centrifugal compressor considered in this study.

In an attempt to reduce the radial dimensions of the centrifugal compressor, this study has investigated the radial diffuser design. Vaneless diffusers are often considered when the aim is to improve the operating MW since there are no issues with incidence at the diffuser vane leading edge (LE). On the other hand, vane diffusers are able to achieve the same amount of diffusion in a shorter passage by turning the flow more. Hence, the study focuses on a vane diffuser. Van den Braembussche [1] noted that, although many different radial diffuser geometries exist, they can be categorised by two types, curved vane diffusers and straight divergent channels. A sudden area expansion exists at the outlet of the straight divergent channel type diffuser which is undesirable for volute performance [2]. There is also a special case, the pipe diffuser, which falls outside of either category. For high PR centrifugal compressors, a pipe diffuser is known to be able to achieve a higher efficiency and lower frictional losses [3]. The pipe diffuser has LE cusps which generate counter rotating vortices (CRV). These
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>flow angle (from tangential)</td>
</tr>
<tr>
<td>$\alpha_i$</td>
<td>vane inlet angle (from tangential)</td>
</tr>
<tr>
<td>$C_{T,\text{max}}$</td>
<td>percentage position of chord for vane max thickness.</td>
</tr>
<tr>
<td>$\eta$</td>
<td>isentropic efficiency</td>
</tr>
<tr>
<td>$O.F$</td>
<td>Objective Function</td>
</tr>
<tr>
<td>$p, p_1$</td>
<td>static/total pressure</td>
</tr>
<tr>
<td>$\tilde{p}$</td>
<td>non-dimensionalised pressure</td>
</tr>
<tr>
<td>$r_1, \ldots, r_5$</td>
<td>radial value at planes 2, ..., 5 (Fig. 1)</td>
</tr>
<tr>
<td>$r_{LE}$</td>
<td>radius of circular leading edge</td>
</tr>
<tr>
<td>$r_{TE}$</td>
<td>radius of circular trailing edge</td>
</tr>
<tr>
<td>$RoC$</td>
<td>rate of curvature of suction side</td>
</tr>
<tr>
<td>$T, T_1$</td>
<td>static, total temperature</td>
</tr>
<tr>
<td>$\ell_{\text{max}}$</td>
<td>vane maximum thickness</td>
</tr>
<tr>
<td>$U_2$</td>
<td>blade speed at impeller outlet</td>
</tr>
<tr>
<td>$\nu_r$</td>
<td>radial velocity component</td>
</tr>
<tr>
<td>$\nu_\theta$</td>
<td>tangential velocity component</td>
</tr>
<tr>
<td>$Y_P$</td>
<td>total pressure loss coefficient</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>ratio of specific heats</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
</tbody>
</table>

### Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>AF</td>
<td>Attached Flow</td>
</tr>
<tr>
<td>ANN</td>
<td>Artificial Neural Network</td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CRV</td>
<td>Counter Rotating Vortices</td>
</tr>
<tr>
<td>DP</td>
<td>Design Point</td>
</tr>
<tr>
<td>EXP</td>
<td>Experimental Results</td>
</tr>
<tr>
<td>ECL</td>
<td>Exit Cone Loss</td>
</tr>
<tr>
<td>FL</td>
<td>Friction Loss</td>
</tr>
<tr>
<td>IF</td>
<td>Interface</td>
</tr>
<tr>
<td>L</td>
<td>Constant speed line</td>
</tr>
<tr>
<td>LE</td>
<td>Leading Edge</td>
</tr>
<tr>
<td>LHS</td>
<td>Latin Hypercube Sampling</td>
</tr>
<tr>
<td>MOGA</td>
<td>Multi Objective Genetic Algorithm</td>
</tr>
<tr>
<td>MFP</td>
<td>Mass Flow Parameter</td>
</tr>
<tr>
<td>MVDL</td>
<td>Meridional Velocity Dump Loss</td>
</tr>
<tr>
<td>MW</td>
<td>Map Width</td>
</tr>
<tr>
<td>$N_{\text{gen}}$</td>
<td>maximum number of generations</td>
</tr>
<tr>
<td>$N_{\text{iter}}$</td>
<td>minimum number of iterations</td>
</tr>
<tr>
<td>NS</td>
<td>Near Surge</td>
</tr>
<tr>
<td>OP</td>
<td>Operating Point</td>
</tr>
<tr>
<td>POP0</td>
<td>Initial population</td>
</tr>
<tr>
<td>POPfinal</td>
<td>Final population</td>
</tr>
<tr>
<td>PR</td>
<td>Pressure Ratio</td>
</tr>
<tr>
<td>PS</td>
<td>Pressure Surface</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>RMS</td>
<td>Root Mean Square</td>
</tr>
<tr>
<td>SS</td>
<td>Suction Surface</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>SVLS</td>
<td>Semi-Vaneless Space</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing Edge</td>
</tr>
<tr>
<td>TS</td>
<td>Total-to-static</td>
</tr>
<tr>
<td>TT</td>
<td>Total-to-total</td>
</tr>
<tr>
<td>Tr</td>
<td>Db Training Database</td>
</tr>
<tr>
<td>VL</td>
<td>Volute Loss</td>
</tr>
<tr>
<td>VLS</td>
<td>Vaneless Space</td>
</tr>
</tbody>
</table>

Fig. 1. Meridional Sketch of Compressor. (For interpretation of the colours in the figure(s), the reader is referred to the web version of this article.)

CRV help to prevent flow separation and reduce frictional losses by moving high energy fluid from the core flow to areas of low energy such as the boundary layers (BLs) [4,5]. The downfall is that the diffuser annulus requires many conical passages and this reduces the stable operating range, similar to a high solidity vane diffuser [6,7].

The study will focus on a curved vane diffuser. One example is an airfoil vane shape which is transformed from Cartesian coordinates into cylindrical coordinates and stacked in a constant line from hub to shroud. Gao [8] studied the diffuser vane shape and found efficiency improvements by defining the vane geometry in a novel way. The vane definition was created based on the idea of controlling the distribution of streamtube area through the diffuser passage. This streamtube idea is presented in Fig. 2 where there are several regions; vaneless space (VLS), semi-vaneless space (SVLS), channel, downstream SVLS, and downstream VLS. The diffuser inlet region (VLS and SVLS) is neglected in channel and cascade design approaches. Gao’s method set an ideal camber line as the suction surface (SS) to achieve a better distribution of streamtube area and therefore improve performance in the diffuser SVLS region. The thickness distribution was also defined, which ultimately defined the pressure surface (PS).

The flow field through a radial diffuser for a high PR centrifugal compressor is complex since the inlet flow is unsteady, non-uniform in pitchwise and spanwise directions, and has a high Mach number. A study by Krain [9] showed a variation of 36° in the time averaged flow angle from hub to shroud. The study also found a 10° variation in flow angle in time due to the unsteady effects from the rotating impeller. The flow near to the hub side of the passage had a negative incidence and impinged on the SS of
the LE, resulting in a region of low energy flow on the PS of the diffuser vane near to the hub. Marconcini et al. [10] showed good agreement with this and furthermore showed the development of a hub PS corner separation. The corner separation developed on the hub PS corner as a result of the spanwise variation in incidence and showed a pulsating characteristic of the separation as a result of the passing wakes from the impeller outflow.

Studies which consider the diffuser endwall geometry commonly focus on stabilizing the flow field. For example, to prevent flow recirculation at the impeller outlet and diffuser inlet region design recommendations are often to pinch or narrow the channel [1]. However, this reduces the through flow area and effectively limits the diffusion process. Recent investigations involve endwall profiling to control secondary flow features [11,12]. Clements and Artt [13] experimentally investigated wedge diffusers with different channel divergence angles and also a couple of diffusers with up to 4° endwall divergence (referred to as sidewall divergence in their paper). They concluded that the static pressure recovery depended on the area ratio and that the divergence angle did not have an impact. Rodgers et al. [14] investigated a spiral style diffuser with diverging hub and shroud endwalls. The study showed potential for maintaining performance, within a reduced radial dimension, by incorporating diverging endwalls. However, the CFD investigation was conducted a decade later than the experimental work and some information was lost or unavailable to complete the validation stage of the study.

As computational power has grown, along with increased capability of optimization processes, there has been an increase in the complexity of designs being considered. Numerical investigations can undertake Computational Fluid Dynamics (CFD) simulations in a reduced time meaning that it is now feasible to integrate an optimization study with many CFD simulations into a numerical design investigation. Additionally, the use of a metamodel now enables the optimizer to work through even more of the design space without the need for a CFD simulation of every design case investigated [15]. Of course, it is important to monitor the accuracy of the metamodel predictions. Elliott et al. [16] used a metamodel assisted MOGA optimization method to design a 3D turbine blade. The study required 1500 CFD simulations to initially train the metamodel. A further 3000 CFD simulations were conducted through the optimization study and the ANN covered much more of the design space.

The aim of this study was to achieve the necessary diffuser area ratio within a radially reduced diffuser passage by diverging the diffuser shroud endwall, Fig. 1. By achieving the necessary area ratio, the aim was to maintain the performance at DP within a radially reduced design. Section 2 of this paper outlines the numerical and experimental methods along with model validation. The optimization study is fully explained in Section 3 and Section 4 compares the performance of three designs. Section 5 presents detailed validation and explanation of the diffuser flow field. Next, the results of the three different diffuser geometries are compared and analyzed in Section 6 of the paper. Finally, conclusions are given on the potential advantages of using a diverging endwall diffuser concept to achieve a reduction in the radial dimensions of a vaned diffuser.

2. Methodology

This study was carried out on a baseline compressor geometry that was provided by an industrial collaborator of the project. The compressor consisted of an impeller with nine main blades and nine splitter blades, a diffuser with seventeen vanes, and a single scroll volute. It was a high PR, transonic compressor stage with impeller diameter around 360 mm. This study maintained the same impeller geometry, but changed the meridional contour of the diffuser geometry as well as the diffuser vane shape, shown in Fig. 1. The diffuser passage was 15% shorter in the radial direction and had a diverging shroud endwall. From preliminary CFD investigations, an effective diverging endwall angle has been identified which has been applied in the present study.

Throughout the study a series of spanwise-circumferential reference planes were considered for both the CFD and experimental (EXP) work, labelled 1-5 in Fig. 1. The 5 stations have been marked on figures throughout the paper with the red square and station number inside. Stations 4 and 5, at the diffuser trailing edge (TE) and diffuser outlet respectively, had a smaller radial position for the new diffuser designs with diverging endwall compared to the parallel endwall baseline case.

This section of the paper describes the configuration of the numerical models, the experimental set-up and the initial validation of the baseline model compared to the experimental results.

2.1. Numerical investigation set-up

The commercial CFD code Ansys CFX v19.2 was used to conduct numerical simulations for this investigation. In order to accommodate a large number of numerical simulations within an optimization study, a balance had to be reached between the fidelity of the model and the need to control the computational cost for the anticipated large number of simulations. Pre-existing experimental data for the baseline compressor was available from the industrial collaborator to enable initial validation of the model and ensure that the model fidelity was adequate to accurately capture the compressor performance.

Three models have been used in this study. The first model was steady state Reynolds’ Averaged Navier Stokes (RANS) and considered a single passage of the rotating impeller domain and a single passage of the stationary diffuser domain without the volute domain. Downstream of the diffuser outlet, a pinch was added to the endwall geometry to aid with simulation convergence, which is illustrated Fig. 3. Upstream of the impeller domain there was a stationary inlet channel, with the same pitch as the impeller domain, which matched the inlet passage of the experimental rig. The fluid domain through each passage was meshed with a hexahedral grid. The inlet passage mesh was created using ICEM and both the impeller and diffuser fluid domains were meshed in ANSYS TurboGrid. The turbulence model selected was the SST model following the recommendation of a study by Gibson et al. [17] on a similar type of compressor. A systematic grid convergence study was conducted following the method outlined by Celik et al. [18] using a refinement factor of 1.25 between each increment of grid refinement. Following the grid convergence study, the selected grids had a combined element count of 2.43 million, shown in Table 1. The numerical uncertainty for TS efficiency us-
ing the selected grid was 1.84% for RANS simulations. The choice of grid was also influenced by the achieved $y^+$ value since literature recommended that a $y^+ < 3$ is appropriate in a turbomachinery application [19].

To model the change of reference frame, a multiple reference frame interface and a mixing plane interface were used between the inlet and impeller domains and the impeller and the diffuser domains, respectively (labelled IF1 and IF2 on Fig. 1). The mixing plane interface was located at 107.5% $r_2$. The location of IF2 was deliberately positioned in a location that allowed the diffuser geometry to be changed while the impeller mesh could remain consistent through the study. Although the mixing plane interface averages the circumferential distribution, a study by Everitt et al. [20] showed that the circumferential non-uniformity was not so important when considering diffuser performance, except for near surge (NS).

Endwalls and blade/vane surfaces were modelled as no-slip adiabatic walls. The total temperature and total pressure were defined at the inlet boundary, while the outlet boundary condition used either the mass flow rate or the static pressure depending on whether the operating point (OP) was close to surge or choke on the operating map. The static pressure at outlet or the mass flow through the passage was aligned with the experimental results and the CFD model was not used to predict the surge flow rate.

The simulation convergence criteria were set as:

i. RMS residuals $< 1E-4$.
ii. Imbalances $< 0.015$, and
iii. Total-to-total (TT) efficiency fluctuations $< 0.01%$.

The second model was similar to the first RANS model except: the full annular diffuser was modelled, the endwall pinch was removed at the diffuser outlet, the volute component was modelled, and a general connection with no frame change interface was used between the diffuser and the volute.

Previous studies had [9] concluded that steady computations could achieve good agreement on performance with experimental results. However, some studies concluded that the mixing plane can produce some inaccuracies at lower mass flow rate OPs [20–22]. Trebinjac et al. [21] found that steady computations did not capture the performance well due to a shock interacting with the interface in the impeller-diffuser gap. The impeller-diffuser radial gap was much smaller in Trebinjac’s study meaning that there was a larger interaction between the interface and the shock compared to the present study. Other studies [20,23] have shown that steady computations are capable of capturing the performance accurately across most of the operating map, with the exception of lower mass flow rate OPs. Since the DP for this particular compressor was close to the surge line a third CFD model was used to investigate the flow field through the diffuser passage. The third CFD model was a transient set up, Unsteady Reynolds’ Averaged Navier Stokes (URANS).

The computational domain used the same grids as the first steady RANS model, but with two diffuser passages modelled to achieve a pitch ratio between the rotor and stator domains closer to unity. Ansys CFX offered a Transient Blade Row method and a Time Transformation interface was selected for both IF1 and IF2 in Fig. 1; this type of interface was used to overcome the non-unity pitch ratio [24]. Following a study of the time step for the unsteady simulation, one hundred time steps per period were set.

A previous study conducted by Galloway et al. [25] which considered a similar compressor and impeller rotational speed found that eighty time steps per period was sufficient. Converged RANS simulations from the first steady model were used as initial results for the URANS model and convergence was considered to have been achieved when pressure values at selected points through the impeller and diffuser domains were fluctuating periodically.

2.2. Experimental investigation set-up

The experimental set up was similar to that described in the study by Galloway et al. [26] since the testing was also conducted on the same SuMa full-scale turbocharger compressor test facility at ABB Switzerland AG (Baden, Switzerland). The closed loop compressor test-rig consisted of an electrically driven impeller, a vaned diffuser and a volute. The inlet total temperature was controlled at a constant value, within approx. $\pm 1$ K variation, by a water-cooled heat exchanger. The inlet air density could be reduced by using sub-ambient inlet pressure. Doing so enabled high PRs to be tested without requiring excessive power input.

Overall compressor performance was determined from measurements of pressure and temperature in the upstream and downstream ducts, while a V-cone was used for volumetric flow rate measurement. The maximum measurement uncertainties were 1% for volume flow rate, 0.2% for total PR and 0.5% for efficiency. The value given for efficiency uncertainty was the maximum uncertainty across the entire operating map. The static pressure was measured at Station 5 to determine the total-to-static (TS) efficiency of the impeller and diffuser elements without the volute. The pressure is known to be non-uniform around the circumference of a typical centrifugal compressor due to the influence of the volute tongue. Therefore, 11 equally spaced pressure taps were instrumented around the circumference at Station 5 on the hub side and averaged to determine static pressure at diffuser exit. Finally, more detailed static pressure measurements were instrumented through a single diffuser passage on the hub endwall to capture static pressure rise along the diffuser vane pressure side (PS), labelled P1-5 in Fig. 4. The selected passage was opposite the volute tongue position and it was expected to be least impacted by the volute tongue geometry.

A flow visualization technique was implemented in the diffuser passage adjacent to the passage instrumented P1-5. This was planned in an attempt to capture the flow separation expected in the corner between the hub and the vane PS side. Three holes (F1-3 in Fig. 4) were used to leak in water at the peak efficiency point. The mass flow rate of the water added into the airstream was less than 1% of the mainstream mass flow rate in one diffuser passage. Water was leaked into F1, F2, and F3 for three minutes each, respectively. As the water evaporated the lime content left a white mark on the surface of the metal. Zones of greater accumulation of white lime were representative of a region of recirculating flow where the flow had a longer period of residence.

2.3. Initial validation

The predicted and measured TS efficiency characteristics (Eqn. (1)) are compared in Fig. 5. Time averaged URANS results for two OPs on the highest speedline have also been included. The TS efficiency has been presented because it was the most direct comparison since static pressure measurements were taken at Station 5 in the experimental work and the numerical work did not include the volute geometry. The mass flow parameter (MFP) values have been normalized to the choke value for the design speed line (L4). The TS efficiency values have also been normalized to the experimental peak efficiency point on L4. The MFP values have been normalized to the experimental maximum MFP on L4.
The discrepancies in the TS efficiency at the DP and the MFP at choke have been calculated and presented in Table 2. With discrepancies of around 1%, which were close to the levels of uncertainty in the experimental measurements, the capability of both CFD models to give accurate predictions of overall compressor performance was considered to be well validated.

\[
\eta_{TS} = \frac{(p_5/p_{11})^{(Y-1)/Y} - 1}{(T_{TS}/T_{11}) - 1}
\]  \hspace{1cm} (1)

Discrepancies are slightly larger at off-design points where there exists more secondary flow features, which may be unsteady, and the RANS CFD method struggles to capture these. This is especially the case near surge, where the unsteady interactions between the moving impeller blades and the stationary diffuser vanes would impact the development of separations in the diffuser passage, which are often over-predicted in extent by a steady RANS simulation.

3. Optimization

An objective of this study was to achieve a 15% radial reduction for the diffuser passage. Some preliminary numerical investigations were conducted and the results were used to select an effective divergence angle on the shroud side of the passage beginning close to the vane LE. The value of divergence angle was similar to optimum angles considered by Runstadler [27]. A vane optimization study was used to obtain the best performance using the chosen endwall divergence angle.

In order to reduce the dimensionality of the optimization, some geometry parameters were fixed to reduce the number of variables impacting the flow field, as shown in Fig. 6. The fixed parameters included: the new radial position of the diffuser outlet (Station 5); the shroud endwall divergence angle (\(\alpha_d\)); the radial position of the start point of the divergence (\(r_d\)); the radial position of the diffuser vane LE and TE (Stations 3 and 4); the minimum thickness of the vane LE and TE (\(t_{LE\text{min}}\) and \(t_{TE\text{min}}\)); and the diffuser throat area (\(A_T\)). In Fig. 6, it can be seen that Station 4 for the baseline geometry coincidently became Station 5 for the radially reduced geometry because of the chosen amount of reduction in diffuser size.

The radial position of the vane LE and the minimum thickness values were all fixed to avoid additional structural considerations in the study. Finally, the diffuser throat area remained fixed to maintain component matching with the impeller [28].

The addition of a diverging endwall altered the area schedule through the diffuser passage. A preliminary numerical investigation highlighted that this resulted in a corner separation between the vane PS and the hub surface at DP where the original flow field was near ideal for the baseline design.

3.1. Parameterization and constraints

Following the method presented by Gao [8], the vane geometry could be defined in a less conventional way, leading to a less constrained vane shape. The vane shape was parameterized using 6 variable parameters. Both the SS and PS were defined by Bezier curves. The Bezier control points for the SS were defined by a second order polynomial, controlled by one variable parameter, the rate of curvature parameter (\(k_{oc}\)). The \(k_{oc}\) was adjusted to control the loading distribution of the SS. The Bezier points for the PS were calculated from the SS points and a thickness distribution. The thickness distribution was based on a normal Gaussian distribution. The LE and TE of the vane were a circular shape with radii \(r_{LE}\) and \(r_{TE}\) respectively. These two parameters, along with the vane maximum thickness, \(t_{max}\), and the position of maximum thickness along the chord, \(C_{max}\), defined the thickness distribution. The variable parameters are listed in Table 3 and shown in Fig. 7. An iterative process that changed the vane outlet angle to achieve the constrained throat area was implemented and if the throat area was not achievable within the given range of vane outlet angles, the design was disregarded.
3.2. Objective functions

The objective functions (OF) for the optimisation were carefully selected based on experience. First, \( OF_1 \) (Eqn. (2)) was created to maximize the TS efficiency of the impeller and diffuser, (Eqn. (1)) and therefore the static pressure rise through the diffuser passage. Second, \( OF_2 \) (Eqn. (3)) was created to maximize the TT efficiency of the entire compressor stage, (Eqn. (4)) and therefore avoid a design with high losses through the diffuser and volute components. The optimization study used the first CFD model, Section 2.1, which did not include the volute component. A simplified low order volute loss (VL) model (Eqn. (5)), taken from ref. [1], was used to reduce the computational cost while allowing the volute performance to be considered through the optimization process. \( OF_2 \) helped to avoid a final diffuser design with an undesirable outflow, resulting in a detrimental volute performance. Both objective functions were equally weighted. The optimization process considered only the DP at the design speed line, DP on Fig. 5.

\[
OF_1 = 2 - \eta_{TS} \\
OF_2 = 2 - \eta_{TT} \\
\eta_{TT} = \frac{\left(\frac{\rho_{t5} - VL}{\rho_{t1}}\right)^{(\gamma-1)/\gamma} - 1}{\left(\frac{T_{t5}}{T_{t1}}\right) - 1} \\
VL = c_1 + \left(0.5 \cdot \rho_{t5} \cdot v_{t5}^2\right) + \left(c_2 \cdot \rho_{t5} \cdot v_{t5}^2\right)
\]

The VL model consisted of three components; exit cone loss (ECL), meridional velocity dump loss (MVDL), and friction loss (FL). The two constants, \( c_1 \) and \( c_2 \), were obtained by calibrating the model to a RANS simulation involving the volute geometry, the second model in Section 2.1. The exit cone loss was considered a constant loss across all designs considered, \( c_1 \). The equation for ECL is provided in literature and values from the CFD models were used to calculate the value for \( c_1 \) [1]. To calculate \( c_2 \), the calculated isentropic total pressure at volute outlet was compared to the CFD value of total pressure at volute outlet. The other two components varied with the change in diffuser outlet velocity and angle. The MVDL was proportional to the square of the radial velocity component at station 5 and the FL was proportional to the square of the tangential velocity component at station 5.

3.3. Framework

The framework for this optimization study was created in MATLAB by Elliott et al. [16], where it was applied to a mixed flow turbine. The optimization method applied was a metamodel-assisted MOGA. The metamodel used was an Artificial Neural Network (ANN). The optimization framework is illustrated in Fig. 8.

To train the ANN, a training database (Tr Db) was created using the Latin Hypercube Sampling (LHS) approach. The method was selected to address the non-linearity of the relationship between geometrical parameters and output performance parameters in turbomachinery design applications. The LHS method selected the values for the 6 variable input parameters and RANS simulations were conducted for each case to obtain the values for OF1 and OF2. The Tr Db began with 100 designs and reduced to 88 due to convergence issues or inability to meet the applied constraint of matching the baseline diffuser throat area.

A population size of 100 was selected and remained the population size as the MOGA created new generations through mutation and crossover. To begin the MOGA, an initial population was created (POP0). Similar to the Tr Db, LHS was used to assign values for the six input variable parameters. However, POP0 differs from the Tr Db as the trained ANN was used to predict the OF values. The MOGA was run for a maximum number of generations (Ngen) or until convergence of the results. After that point, three elite designs were selected from the final population (POPfinal) and simulated using the CFD model. The simulation results were added to Tr Db and the ANN was retrained. Additionally, the simulation results were compared to the ANN predictions and adjustments were made to Ngen if necessary. If the ANN predictions were within a specified tolerance of the simulation results, then Ngen was increased. Likewise, if the discrepancy was outside of the specified tolerance, then Ngen was decreased. This approach prevented the optimizer from going in a certain direction based on a poorly trained ANN but also reduced the computational cost when the ANN was sufficiently accurate. At this point, the first iteration of Loop 1 had been completed and POPfinal from the completed iteration was set as POP0 for the next iteration. Loop 1 ran for at least Niterm iterations and until the simulation results did not improve for 10 iterations. Loop 2 was added to ensure that the optimizer did not find a local optimum and ensured that the design space was not reduced prematurely due to a poorly trained ANN on the first few iterations. When Loop 1 was complete, a new POP0 was created by LHS and fed into the MOGA to begin Loop 1 again.
3.4. Optimization results

The optimizer ran through Loop 2 three times, meaning that the optimizer was re-seeded twice. The first two ran for 15 and 18 iterations of Loop 1, respectively. The final Loop 2 ran for 10 iterations and no performance improvement was found. At that point, the optimizer was considered to have finished and a total of 286 simulations had been completed. The TS and TT efficiencies, normalized to the baseline efficiencies, for all of the simulations conducted in the optimization study are shown in Fig. 9. The Pareto front has been highlighted with a dotted red line. The variation of TT efficiency was much smaller than the variation of TS efficiency. The TT efficiency also relied on the volute loss model, which was a simple empirical loss correlation with associated uncertainty. For these reasons, the best TS efficiency point along the Pareto front was considered as the optimized design (the point at the left of the red dotted Pareto front).

Overall, the diverging endwall was successful in allowing the size of the diffuser to be significantly reduced in the radial direction. The TT and TS efficiencies achieved by the optimized design were 99.7% and 98.7% of the baseline case, respectively. This indicated a higher kinetic energy at the diffuser outlet and volute inlet for the optimized case, but no significant increase in compressor stage losses due to the reduced radial size of the diffuser.

4. Overall performance comparison

Throughout the rest of this paper, three different cases are compared: Case A was the baseline case; Case B was the radially reduced diffuser passage with an altered baseline vane shape; and Case C was the radially reduced diffuser passage with the optimized vane shape. The altered vane shape in Case B was derived from the baseline airfoil provided by the industrial collaborator, but in order to maintain the throat area with the reduction in diffuser outlet radius, the curvature of the vane camber line was varied. Both radially reduced diffuser passages, Cases B and C, had shroud endwall divergence. All three cases were tested with the same volute geometry in the experimental work.

Cases B and C are shown in Fig. 10, and compared by superimposing the two vanes. Some of the differences are obvious. For example, Case B had a forward thickened airfoil shape and Case C was close to a constant thickness from LE to TE. The figure also clearly shows a different TE for both vanes; Case C had a more tangential TE angle. What is less obvious in the figure are the variations to the LE; these variations were small but important for diffuser performance. Case C had a thicker LE than Case B due to its circular shape; the LE of Case B was less rounded as a result of the airfoil definition used. Finally, the LE angle of Case B was 1.37° more tangential. In order to minimise the number of dimensions controlling the vane shape (Section 3.1), the LE of the vane in the optimization study was fixed as circular and therefore was unable to achieve a vane shape identical to Case B. The performance metrics at the DP, obtained using the RANS model of the impeller and diffuser coupled with the empirical VL model, were normalized to the value of baseline Case A and are summarized in Table 4.

The experimental TT efficiency measurements in Fig. 11 showed that the radially reduced diffusers, Cases B and C, maintained the performance of the baseline design, Case A, across all speed lines (L1-L4). The TT efficiency, Eqn. (4) (using the measured total pressure value at volute outlet instead of using the VL model), improved slightly across the operating map.

The graph illustrates that the efficiency performance has been maintained at DP. Hence, the overall goal of reducing the diffuser radial dimensions, while maintaining performance at DP, was achieved. Furthermore, Fig. 11 shows that the efficiency was maintained or improved across the entire operating map and the operating MW was extended for the design speed line. Both of these positive outcomes exceeded the original aims of this study and highlighted potential for future investigation.

5. Flow field validation

The experimental campaign considered Cases A, B, and C which enabled further validation of the numerical investigation. Perfor-
mance metrics lined up well between experimental and RANS results across the entire operating map for all three cases; Case A was presented earlier in Section 2.3, which showed that the RANS CFD model was capable of capturing the overall performance accurately.

A strong corner separation had been identified in the CFD predictions for the radially reduced Cases B and C in the corner of the hub endwall and the vane PS (this flow feature is considered in more detail in Section 5). This section presents results for Case B, which had a diverging shroud endwall but used a transformed version of the baseline airfoil.

Fig. 12 (left) shows detailed pressure measurements for selected OPs along the top speedline, L4, which are considered here. The five static pressure taps along the vane PS, shown in Fig. 4, were used in an attempt to identify a region of recirculating flow. In Fig. 13, the hub skin friction lines have been plotted for the same OPs. By focusing on the ‘Choke’ OP, it can be seen in Fig. 12 (right) that the RANS CFD model captured the initial decrease in static pressure followed by a sudden rise in static pressure, indicative of supersonic acceleration followed by a shock. The RANS results also showed a region of almost constant pressure downstream of the shock, which indicated a region of recirculating flow. This region of constant pressure was not captured by the experimental results since there were only 5 discrete pressure taps, however, the 5 points were enough to show good confirmation of the supersonic acceleration followed by a separation. Fig. 13 shows the flow separating from the vane PS between P2-P3, and a subsequent region of recirculating flow for the choke OP. The mid-map OP also showed good agreement between RANS results and experimental results. At this OP, there was no region of recirculating flow captured by the pressure rise graph, which was in agreement with Fig. 13. For the lower mass flow OPs, DP and NS1, the discrepancy between experimental measurements and RANS results was larger. The hub skin friction lines, Fig. 13, show the existence of a corner separation, which can be seen as a reduction in the pressure rise gradient downstream of P3 on Fig. 12.

In this study, at the lower mass flow rate OPs, DP and NS1, the pressure at P1 was higher for the RANS predictions than the experimental measurements, suggesting that the real flow field may not have been accurately captured due to the mixing plane in the VLS between the impeller and diffuser. For this reason, URANS simulations were used to further investigate the diffuser flow field at the DP. Further experimental results are presented here to verify that the URANS CFD model correctly predicted the localized details of the flow field.

The peak efficiency OP for Case B, NS1, was selected for flow visualization, which has been explained in Section 2.2. This was conducted for Cases B and C but Case B is presented here. Fig. 14 shows the experimental flow visualisation on the hub surface. Red arrows have been superimposed on Fig. 14(a) to highlight the flow separation and the region of recirculating flow. Red arrows have also been added to Fig. 14(b) to show the flow spilling into the adjacent passage at the TE. Fig. 15 looks onto the hub surface to show predicted skin friction lines on the URANS numerical model. Red arrows corresponding to the arrows on Fig. 14 have been added to Fig. 15 to identify the flow features for this discussion.

First, looking at the hub surface and vane PS in Fig. 14(a), the flow has clearly moved from the hub to the shroud side of the passage on the vane PS following flow separation. A region of recirculating flow has been captured downstream of F2 and extended over F3. Additionally, the flow can be seen to have reattached briefly, downstream of F3. The same flow pattern can be seen in Fig. 15. The region of attached flow (AF) has been labelled on both Fig. 14(a) and Fig. 15. At the TE of the vane, both experimental and RANS results show the flow wrapped around the TE from PS to SS, as can be seen in Fig. 14(b) and Fig. 15. Finally, both experimental and RANS results showed highly tangential flow at diffuser outlet on the hub surface. The same was seen on the shroud surface and a comparable wake feature was captured (not presented here).
6. Flow field analysis and discussion

The experimental performance for the three different cases has been compared in Section 4, Fig. 11. The URANS model was validated against experimental work for analyzing the flow field in Section 5. This section uses time-averaged URANS CFD results to investigate; (i) the impact of a radially reduced diffuser passage with diverging endwall on compressor performance, and (ii) any improvements found with the optimized vane shape.

6.1. Impact of radial reduction

Since Case B was generated from the baseline airfoil vane shape, it is considered here in detail to understand the impact on performance at DP of radially reducing the passage. The overall TT efficiency was maintained. A corner separation was identified in Section 5, indicating higher losses through the diffuser passage. Therefore, it was expected that the volute losses were reduced so that the diffuser losses were compensated to achieve the same overall TT efficiency.

The URANS numerical study was used to compare the total pressure loss through the diffuser and the volute loss model, Eqn. (5), was used to compare the total pressure loss through the volute for the three cases. To calculate the volute loss, the radial and circumferential velocity components at diffuser outlet were extracted from the URANS model. Fig. 16 shows the flow angle at diffuser outlet. From this plot, it is evident that there existed some recirculating flow on the hub endwall for Case B. Using a similar approach to that of Stuart et al. [26], the diffuser passage was divided into two sections; recirculating flow and mainstream flow. The mass flow rate through the mainstream flow section was equal to the net mass flow rate across Station 5. Considering the mainstream flow portion for Case B and the entire outlet area at Station 5 for Case A, the mass flow averaged, time averaged, and flow angles were calculated. For the radially reduced Case B, the Mach number was 12.1% higher than Case A and the flow angle for Case B was 80.7% of Case A. In summary, the diffuser outflow for Case B had a higher Mach number and a more tangential flow angle.

Fig. 17 shows the total pressure loss through the diffuser and volute passages from the URANS numerical study and the VL model, non-dimensionalised by Eqn. (6). The hub and vane PS corner separation (investigated in more detail later in this section) along with the recirculating flow at the diffuser exit on the hub endwall of the passage were two sources of loss identified in Case B. This was seen in Fig. 17 as increased total pressure loss through the diffuser. As was expected, the modelled VL was reduced for Case B.

\[
\Delta \bar{p} = \frac{\Delta p}{\rho_1 \cdot U_2^2} \quad (6)
\]

To understand the reason for a reduced volute loss prediction, the components of the volute loss model have been considered. In Fig. 18, the normalized volute loss has been plotted for a range of flow angles at diffuser outlet along with the three different components. From this, it was clear that the calibrated volute loss model was mostly influenced by the MVDL. The MVDL is proportional to the radial velocity component. A higher radial velocity component at volute inlet caused more swirl in the volute flow, in the r-z plane (Fig. 1). This increased the length of the streamline through the volute and therefore increased the losses through the volute as part of the swirl was dissipated by internal shear. Hence, as the flow became more tangential (lower flow angle) at diffuser outlet for Case B, the loss predicted by the VL model reduced, Fig. 17. Since the same volute was used for all 3 cases in the experimental set up, there was potential to match a new volute design to Case B and improve the overall stage performance.

Fig. 19 shows the total pressure loss coefficient, calculated using Eqn. (7), plotted on constant radius planes for DP through both diffuser passages. Where the total pressure loss coefficient is calculated based on values at the impeller TE (Station 2) and the downstream local total pressure value, \( p_1 \). Fig. 20 shows the hub skin friction lines for Cases A and B superimposed on the hub pressure contour. For Case A there was some accumulation of BL flow in the hub PS corner, Fig. 19, and a small corner separation was detected in the skin friction lines, Fig. 20. For Case B, the total pressure loss through the passage was significantly increased, Fig. 19. It is evident in Fig. 20 that the corner separation occurred earlier in the passage for Case B and a large recirculating flow region downstream of the corner separation was present.

\[
\gamma_p = \frac{p_{r,2} - p_1}{p_{r,2} - p_2} \quad (7)
\]
The hub skin friction lines in Fig. 20 show positive incidence at the hub endwall for both A and B. The development of a hub PS corner separation was described by Krain [9] to stem from negative incidence at the hub side of the passage in that specific case. Fig. 21 shows the flow angle through the meridional cross section of the diffuser. In the present study, the impeller-diffuser VLS was larger than in Krain’s study and allowed more flow mixing and BL development upstream of the diffuser vane LE. Within the VLS between Stations 2 and 3, the hub BL flow became more tangential at the diffuser LE, resulting in positive incidence and flow spilling to adjacent passages, Fig. 20. The result was still a region of low energy flow on the PS of the diffuser vane passage near to the hub.

A corner separation is caused by the two adverse pressure gradients which the flow passing through a vaned diffuser passage is subject to, Fig. 22. The first adverse pressure gradient is obviously experienced as the flow travels from low pressure at the diffuser inlet to a higher pressure at outlet. Secondly, the hub flow is highly tangential near to the hub as it enters the diffuser passage and travels from SS to PS. In doing so, the flow travels along an adverse pressure gradient from the low pressure SS to the high pressure PS. These adverse pressure gradients encourage BL growth on the vane and endwall surfaces and typically lead to corner separation, which is illustrated at the corner of the hub endwall and vane PS in Fig. 22.
In Fig. 21, the larger corner separation in Case B was more apparent. The recirculating flow identified in Fig. 16 for Case B was also seen in the meridional plot. Through the diffuser, the main source of loss was the mixing of the BL flow with mainstream flow as well as the recirculating flow downstream of the corner separation mixing with the mainstream flow, Fig. 19. Hence, the corner separation in the diffuser passage, on the vane PS and hub corner, was found to be a dominant secondary flow feature within the diffuser passage sub-component.

The change in flow field for the radially reduced case can be explained by a change in the radial pressure gradient through the diffuser. Fig. 23 shows the pressure rise through the diffuser passages. This was the rise in pressure from the inlet total pressure condition. Colour has been used to separate the pressure rise through the different sections of the diffuser, as labelled in Fig. 2. There was a brief reduction in pressure downstream of the vane LE for Case B, followed by a steep increase in pressure. The steep increase in pressure is a result of the pressure reduction upstream and also the more aggressive increase in area through the passage.

Fig. 24 shows the percentage area of reversed flow at constant radial planes through the diffuser passage. The high percentage of reversed flow in the passage of Case B, Fig. 19, the corner separation and the consequent recirculating flow which can be seen in the pressure rise in Fig. 19, and the region of low flow angle, labelled (2) in Fig. 21.

Fig. 20 shows that the entire pitch of the hub flow of Case B fed in to the recirculating flow instead of travelling downstream. The hub side of the passage was blocked and the streamlines show the flow recirculating in the downstream SVLS of the diffuser passage. The hub side recirculating flow with a negative flow angle, labelled (3) in Fig. 21, explains why the blockage did not return to 0% in Fig. 24 downstream of the corner separation.

In summary, a radial reduction and endwall divergence created a stronger streamwise adverse pressure gradient which first resulted in a corner separation and consequently created a region of recirculating flow on the hub endwall. Both features were additional sources of loss for Case B. The diffuser outflow for Case B had a higher Mach number and a lower flow angle (more tangential flow). Although a lower Mach number at volute inlet is desirable to minimize losses through the volute, Gao [8] noted that there is a balance to be achieved between flow angle and Mach number at volute inlet. In this case, the experimental results implied that the volute losses were lower for Case B as a result of a more tangential flow at inlet.

6.2. Change in performance with optimized vane

Overall, in Section 4, it was seen that Cases B and C achieved similar efficiencies in the experimental campaign. URANS results predicted a reduction in non-dimensionalised total pressure loss from 0.255 for Case B to 0.199 for Case C, a reduction of 22%. The volute loss model predicted the same non-dimensionalised total pressure loss for Cases B and C.

Fig. 25 shows the total pressure loss coefficient through Case C. The loss through the diffuser passage had reduced compared to Case B in Fig. 19, but there was still a corner separation present in Case C. Fig. 26 shows the hub skin friction lines superimposed on a pressure contour for Case C. Again, the recirculating flow on the vane PS was apparent but noticeably smaller than what was identified for Case B.

The area expansion through the diffuser passage was more aggressive for both Cases B and C compared to Case A. However, the optimized Case C has eliminated the area reduction and associated pressure drop immediately downstream of the vane LE which was apparent in Case B, Fig. 23. The area reduction for Case B was a re-
sult of the forward thickened airfoil shape and the new shape for Case C resulted in a smoother increase in pressure from LE to TE. Although the pressure rise through Case C did not clearly identify an aerodynamic blockage similar to Case B, a region of higher total pressure loss was noted in Fig. 25 downstream of the corner separation. The blockage due to the recirculating flow following the corner separation was identified in Fig. 24, 2, and was significantly smaller than for Case B. The streamlines in Fig. 26 show that not all of the flow on the hub side of the passage was wrapped up in the recirculating flow following the corner separation and the blockage returned to almost 0% downstream of the corner separation in Fig. 24.

From the hub skin friction lines, Fig. 26, there was a second obvious difference in the flow through Case C compared to Case B, Fig. 20. For Case B the flow was recirculating at the hub side of the passage whereas for Case C the flow did not recirculate at diffuser outlet on the hub side. Fig. 27, comparable to Fig. 16, shows the flow angle at diffuser outlet for Case C. It is evident that the flow was not recirculating on the hub side of the passage but instead it was recirculating on the shroud side. The recirculating flow on the shroud side of the passage, 8, was also apparent in Fig. 21.

For Case C, Gao’s [8] method of defining the vane shape focused on using an ideal SS and the benefit can be seen here in Fig. 26. By looking at the SS of the vane, it was seen that the hub flow remained attached to the vane SS further downstream on Case C than Case B. For Case B, most of the hub flow mixed with the recirculating flow as it had separated early from the SS, Fig. 20. In comparison, for Case C the hub flow remained attached to the SS for longer and subsequently some of the hub flow crossing the passage was seen to attach to the vane PS, marked with a dashed line on Fig. 26, and flow out of the passage rather than interacting with the zone of recirculating flow. In Fig. 24, the aerodynamic blockage in Case C reduced to almost 0% following the corner separation. This was due to the flow reattaching to the hub and vane PS. Consequently, the pressure rise through the passage was larger for Case C compared to Case B, Fig. 23. Fig. 28 is a plot of the shroud skin friction lines on all three cases. For Cases A and B, the flow exited the diffuser passage on the shroud side. For Case C, the flow on the shroud side was recirculating. This recirculating flow was a result of the shroud BL flow separating due to the stronger adverse pressure gradient in the latter half of the diffuser passage. The region of shroud side recirculating flow was responsible for the increased blockage approaching the TE of Case C, which can be seen after the blockage due to the corner separation in Fig. 24.

7. Conclusion

Two new vane diffusers were designed for a centrifugal compressor that used endwall divergence to reduce the radial length of the diffuser passage. One of the new diverging endwall diffusers (Case B) used an adapted version of the airfoil from the baseline vane (Case A), while the other diverging endwall diffuser used a new vane profile from an intensive optimisation process (Case C) to maximise the benefit of the diverging endwall. By deploying divergence on the shroud endwall of the vane diffuser, a 15% radial reduction of the diffuser passage length has been successfully achieved without any penalty in stage efficiency. An increase in map width was achieved at some operating speeds even though this was not the objective of the study.

An extensive experimental investigation verified that the RANS CFD simulations captured the compressor performance well and also predicted the flow field features accurately for higher mass flow OPs. However, a third CFD model, URANS, was used to analyze the flow field since a lower mass flow rate point was the focus of the investigation. The URANS CFD results were used to conduct a detailed flow field analysis to interpret the overall design benefits. A higher total pressure loss was found through the diffuser passage for the two diffuser cases with a diverging endwall. Since overall efficiency was not reduced, this implied that total pressure loss through the volute was reduced, possibly due to a more tangential flow at diffuser outlet.

All three cases had a hub PS corner separation. Following the corner separation was a region of recirculating flow. This separation was very small for the baseline diffuser with parallel endwalls, but become a significant flow feature when the endwall divergence was added. The study showed that the more aggressive area expansion created a stronger adverse pressure gradient for the flow. The optimizer successfully found a new vane design with a smoother increase in area through the streamtube SVLS and channel. This resulted in a smaller region of low energy flow recirculat-
ing due to the corner separation. Since the aerodynamic blockage from the corner separation was smaller for the optimised vane in Case C, the mainstream flow through the channel was subject to a stronger adverse pressure gradient. This resulted in a recirculating flow region on the shroud side of the passage at diffuser outlet. Therefore, although Case C reduced the size of the corner separation flow feature, it did not yield any efficiency improvement over Case B with the diverging endwall that used a vane shape adapted from the baseline vane shape.

The results show that a diverging endwall diffuser was able to significantly reduce the size of the diffuser without any penalty in efficiency or stable operating range. A detailed analysis of those results showed additional secondary flow structures due to the more aggressive diffusion, but that these had been successfully controlled during the design process to avoid efficiency penalty. Since some additional loss was incurred in the diverging endwall diffusers, by implication the volute loss must have reduced by virtue of the more tangential inflow angle to avoid any deterioration in efficiency. This design approach is capable of delivering saving in size, weight, material and cost without any loss of performance.

Declaration of competing interest

The authors declare the following financial interests/personal relationships which may be considered as potential competing interests: Laura McLaughlin reports financial support, equipment, drugs, or supplies, and writing assistance were provided by ABB Switzerland.

Acknowledgements

The authors would like to thank ABB Switzerland AG for the significant contribution of funding, technical support, and experimental test work. The authors would also like to thank Gerhard Fitzky for operation of the compressor test facility and for gathering experimental results during a global pandemic, COVID-19. Finally, the authors would like to acknowledge the support of ANSYS Inc. for the use of their software.

References