Proceedings of ASME Turbo Expo 2008: Power for Land, Sea and Air
GT2008
June 9-13, 2008, Berlin, Germany

GT2008-51262

DESIGN, VALIDATION AND APPLICATION OF A RADIAL CASCADE FOR CENTRIFUGAL COMPRESSORS

Alexander Simpson
GE Global Research
Munich, Germany

Christian Aalburg
GE Global Research
Munich, Germany

Michael Schmitz
GE Global Research
Munich, Germany

Robbert Pannekeet
GE Global Research
Munich, Germany

Florian Larisch
Technical University Munich
Munich, Germany

Vittorio Michelassi
GE Oil & Gas
Florence, Italy

ABSTRACT

A novel sector test rig has been used to evaluate a new airfoil concept for multistage radial compressors. The test rig is supported by a blow-down facility where the operation conditions are adjusted by controlling mass flow, pressure and temperature. At inlet to the sector test rig itself a set of adjustable inlet guide vanes provide the test vanes with the correct inlet three-dimensional flow-field.

The rig is equipped with instrumentation to allow a detailed description of the inlet and outlet conditions, as well as the blade pressure loading. This rig, using rapid prototyped vanes, allows design candidates to be screened quickly and is ideal for conducting an experimental investigation of a design space using a Design-of-Experiments approach.

In this paper the rationale for the sector approach is described, the design of the test rig with 3D-CFD methods is outlined and a detailed validation of the rig is presented. For the vane in question detailed investigations of different operation points close to stall are reported, blade pressures and inlet and exit flow profiles are given. Where applicable, measurement data from the sector rig was compared to 3D-CFD calculations of the full annulus multistage configuration, to 3D-CFD calculations of the sector rig itself and to the test results from a 1.5-stage rotating test rig. The measurement data are compared to the CFD predictions and served as a calibration basis for the design tools.

NOMENCLATURE

Symbols

c	Chord length at mid-span [m]
D	Rotor outer diameter [m]
m	Mass-flow [kg/s]
μ	Molecular viscosity [kg/ms]
ϕ	Flow coefficient [-]
P	Pressure [N/m^2]
R	Gas constant [J/(kg K)]
Re	Reynolds number [$\rho v c/\mu$]
ρ	Density [kg/m^3]
T	Temperature [°K]
Theta	Angular coordinate [°]
U	Rotor speed at outer diameter D [m/s]
v	Velocity [m/s]

Subscripts

s	Static quantity
0	Stagnation quantity

INTRODUCTION

In the design of industrial multistage centrifugal compressors it is sometimes desirable to redesign the stator parts without altering the impeller. Testing these designs in a full rotating test is both an expensive and time-consuming process requiring the commitment of significant resources. It would thus be advantageous to have an inexpensive and flexible test rig that could be used to screen different return channel vane designs prior to a final validation test in a rotating rig.

In the field of axial turbomachinery linear cascades have been used to facilitate high fidelity measurements in a simplified environment thus providing valuable information on blade performance. These test rigs are relatively inexpensive and simple to construct/operate requiring only a wind tunnel as supporting infrastructure. Whilst the cascade approach fails to capture some of the important flow phenomena, such as unsteadiness from the upstream impeller, adaptations have been made to counter these shortcomings [1].

The need for an inexpensive test rig for the screening of vane designs and the observation of the usefulness of axial turbomachinery cascades lead to the development of a "sector test rig" for the testing of radial compressor stator vanes. This rig, via the use of a novel inlet design and a set of preswirl vanes, provides the correct flow angle profile to the return channel vane in a test rig that can be attached to a suitable blow down test facility.

The development and commissioning of the sector test rig has been introduced in a previous paper [2] in which the design objectives were summarised as follows:

1. The vanes and flow channel should not be scaled geometrically. This is to facilitate a one-to-one comparison to the expected operating conditions. Further, a reduction of the dimensions of the flow channel would potentially impact on the practicality of detailed measurements.
2. The rig should be constructed such that the vanes can be readily interchanged.
3. It should be possible to utilise the test section with a range of blow-down test facilities.
4. The operating condition should be variable to allow experiments at different flow coefficients to be conducted.
5. The test rig should be designed to allow a range of high-resolution measurement techniques to be applied at a variety of locations.
6. The turbulence conditions at the inlet to the test section should be adjustable in order to facilitate investigations of the influence of turbulence. This is in recognition of the controversy that surrounds the effect of turbulence in the inflow, [3].

In this paper the sector test rig, the test facility and the commissioning will be briefly summarized. The discussion will then consider a range of results from the rig in greater detail. These will include experiments at low flow coefficients and the impact of surface roughness on performance.

SUPPORTING INFRASTRUCTURE

One of the blow-down facilities available at GE Global Research Munich, described in more detail in [2], has been used to provide a carefully controlled flow for the sector test rig. Mass-flow, pressure and temperature are controlled and are variable on the ranges indicated in table 1.

Flow Parameter	Range
Mass-flow	0-900 g/s
Pressure	0-9 Bar
Temperature	0-120 °C

Table 1: Flow parameters for the blow-down test facility used to supply the sector test rig.

Mass-flow measurements were conducted with vortex type mass-flow controllers attached to the blow-down facility. Accuracies, varying with mass-flow, were of the order of ±0.5% for the flows typically set for the sector test rig.

The inlet included several components to ensure the sector test rig received a well-conditioned inlet flow. These included a diffuser, a flow straightening section and a metal gauze. The gauze was included to set the turbulence intensity at the inlet of the test section. An exhaust duct connected the exit of the sector test rig to the throttle system.

ROLE IN OVERALL STAGE DEVELOPMENT STRATEGY

In an investigation by the same authors that has been reported in a separate paper, [4], an attempt has been made to reduce the outer radius (i.e. arrive at a reduced radius design) of a stage for a multistage centrifugal compressor. The radius reduction is measured in terms of a "diffusion ratio" which is defined as the ratio of the outer radius of the vaneless diffuser to the inner radius. The "baseline design" had a diffusion ratio of 1.45 (redesigns were attempted with diffusion ratios of 1.30 and 1.19). Schematics of multistage radial compressor stator parts illustrating different diffusion ratios are illustrated in Fig. 1.

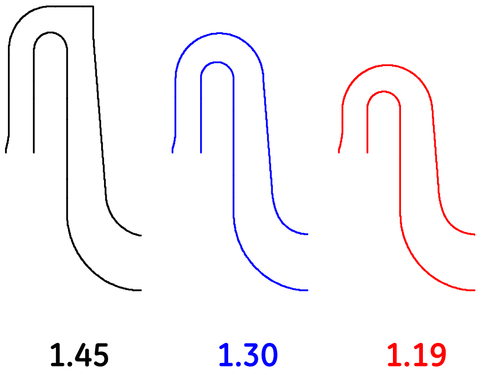


Figure 1: Three schematics of the flow path of a centrifugal compressor stage. The number under each schematic corresponds to diffusion ratio.

The overall design objective was to achieve the radius reductions without penalizing efficiency or operating range. This was achieved. In terms of the reduced radius effort the role of the sector test rig is to allow elements of the off-design performance of the stator vanes to be understood such that the operating range of the reduced radius design could be extended to lower flow coefficients. Thus, the experiments reported in this paper will focus on operating points at lower mass flows than that corresponding to design.

THE SECTOR TEST RIG

This section will present a description of the test rig and instrumentation discussed in this paper and includes some insights into the design process. The interested reader is pointed to [2] for more details. Figure 2 shows a schematic of the flow path of a centrifugal compressor stage highlighting key components. From this point on this stage will be referred to as the "design configuration".

The sector test rig was designed as a 90-degree sector of the return channel of the design configuration. A schematic (front and axial view) is shown in Fig. 3. Photographs (front and approximately axial view) are shown in Fig. 4. A carefully designed inlet and set of guide vanes effectively mimicked the impeller, diffuser and bend. From this point on these guide vanes will be referred to as "preswirl vanes" and the section around it will be referred to as the "preswirl section". The inlet and the preswirl vanes were designed to produce flow profiles at the exit of the preswirl section that matched numerical predictions at the entrance to the return channel of the design configuration. It should be noted that both the preswirl and the deswirl vanes are three-dimensional airfoils.

In a typical full annulus test a "pseudo stage", [4], is used to mimic the effect of an upstream stage (apart from unsteadiness). The pseudo-stage consists of a set of pre-swirl vanes (preswirler) followed by a scaled version of a return channel (deswirler), typically corresponding to the return channel of the test stage. The inlet of the sector test rig presented a significant design issue due to the bend downstream of the diffuser. The radial pressure distribution attributable to the bend leads to a substantial divergence of flow angle, of the order of 40°, between the hub and the shroud. This renders a preswirl configuration in the form of a traditional pseudo stage (configuration (a) in Fig. 5) impractical as it yields streamlines across the bend that, due to their divergent nature (they would have to intersect numerous vanes), are impossible to follow using sidewalls. In order to address this issue, a number of alternative configurations were studied, all with the preswirl vanes located downstream of a potential bend. The configurations studied are illustrated in Fig. 5. It was concluded that only the configuration with the reverse inlet and a half bend (Fig. 5 configuration (f)) generated the desired spanwise flow profiles at the inlet to the deswirl section. For more details please consult [2].

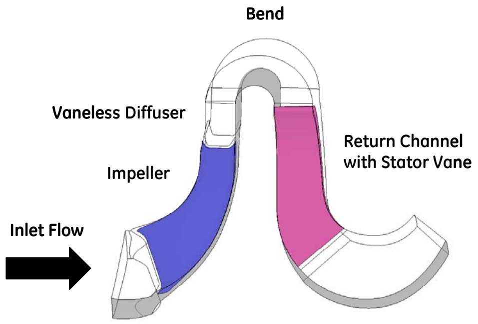


Figure 2: Schematic of the flow path of a centrifugal compressor stage.

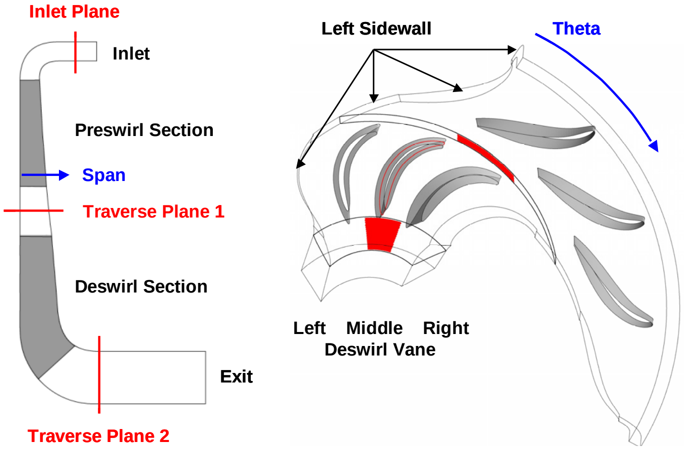


Figure 3: Schematic of the sector test rig (axial and front view). Traverse plane locations are marked in red. Coordinate directions are marked in blue.

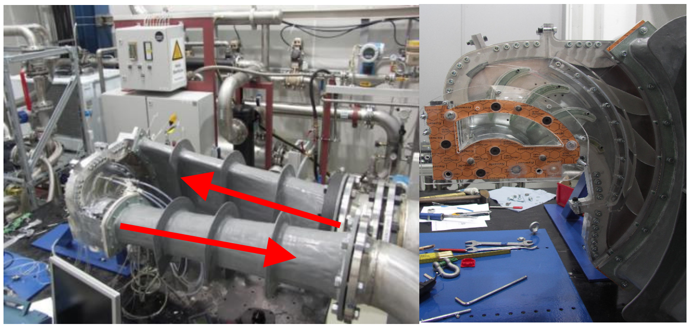
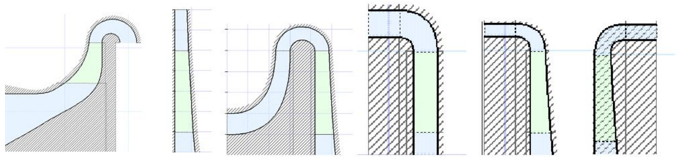


Figure 4: Photographs of the sector test rig (axial and front view). Red arrows indicate flow direction.



(a) (b) (c) (d) (e) (f)

Figure 5: Pre-swirl configurations investigated in this study: (a) traditional pseudo stage, (b) radial inlet, (c) full bend left, (d) half bend left with constant width, (e) half bend left with constant area, (f) half bend right with changing width and area.

A numerical DOE employing 3D-CFD predictions was utilized to determine the optimal geometry for the vanes for three separate flow coefficients. The endwalls of the preswirl section are slightly divergent from inlet to the deswirl section. This is in order to decelerate the flow in the radial direction and thus achieve the required turning. The sidewalls of the preswirl section of the sector test rig consisted of two “half-vanes” of the preswirl vanes corresponding to the 80% operating point.

The inlet is instrumented with a combination of thermocouples and RTD’s (upstream of the gauze, accuracy better than ±0.5°C), nine circumferentially distributed static pressure taps on the shroud and nine kiel probes at mid-span (upstream of the preswirl vanes). In all cases pressure measurements were acquired using a combination of 5psid differential pressure transducers (accuracy of ±0.05%) and 2Bar absolute pressure transducers (accuracy of ±0.3%). A radial port at the same location facilitates hotwire measurements. A traverse slot downstream of the preswirl vanes, shown in Fig. 3 (marked in red), facilitated the determination of flow profiles at this location. Traversing was performed on a 12x30 measurement grid covering a little over one pitch.

The design configuration has 16 vanes in the return channel. The return channel portion of the sector test rig, referred to as the “deswirl section”, has three whole vanes and two “half” vanes making up the side-walls (i.e. defining the limit of the section). These three whole vanes will be referred to as “deswirl vanes” from this point on and are identical in geometry to those from the design configuration. The hub and shroud walls of the sector test rig were designed such that the preswirl and deswirl vanes were both readily interchangeable.

In the region between the preswirl vanes and the deswirl vanes the sidewalls were shaped to follow the predicted streamlines between the two vanes. This was done in order to achieve optimum periodicity among the flow channels. It is important to note that while the preswirl and deswirl vanes in the flow path are exchangeable (as well as the endwalls in the deswirl section), the sidewalls of the present sector test rig are not adjustable. In order to achieve optimum periodicity, different operating points would clearly merit the implementation of adjustable sidewalls (relatively straight forward). A careful CFD based optimization produced a reasonable compromise of sidewall design that did not justify the added complexity of a design with variable sidewalls.

The value of Reynolds number (Re) at the leading edge of the deswirl vane was 1.2×10^6 (at the controlled operating conditions corresponding to 80% mass-flow) and tests were conducted with an exit stagnation pressure of 1.4Bar. The deswirl vanes were equipped with static pressure taps for the measurement of static pressure profiles. The middle deswirl vane, see Fig. 3, was equipped with static pressure taps at 20%, 50% and 80% span on the suction surface and at 50% span on the pressure surface. The left and right deswirl vanes (labeled in Fig. 3) were equipped with taps at 50% span on both the suction and pressure surface. A traverse slot downstream of the deswirl vanes facilitated traverse measurements. A 12x30 measurement grid was used at this location and covered approximately two pitches.

NUMERICAL METHODS

Two separate CFD codes were used throughout the investigations discussed in this paper:

1. Ansys-CFX 11, a commercial CFD package,
2. and an structured grid non-linear Navier-Stokes solver (developed in-house for turbomachinery applications).

During the design phase two approaches were used to discretize the geometry. In the initial phase the vane shapes were developed by using fully periodic boundary conditions for the modeling of the flow channels. The mesh was generated using an in-house mesh generator (structured grid, 1.3 million cells).

In the second phase the full sector test rig was modeled (four flow channels, no slip condition at sidewalls). This was for design of the sidewalls and verification of the flow characteristics. This study was carried out using Ansys-CFX 11 with a structured grid consisting of 5.8 million cells. The grid resolution at the wall surfaces, in terms of the dimensionless wall distance, was of the order of y+=1. Wall integration was used to compute wall shear stresses and turbulence effects were approximated using the k-ω model.

The boundary conditions utilized in both cases were fixed total pressure, temperature and turbulence properties at the inlet to the transition duct and fixed mass-flow at the exit of the test rig.

RESULTS AT 80% DESIGN FLOW COEFFICIENT

This section will consider measurements and numerical predictions corresponding to the key sections of interest throughout the test rig for the 80% design flow coefficient case. In all of the plots the hub lies at 0% span and the shroud lies at 100% span. All numerical predictions regarding the sector test rig itself reported in the remainder of this paper have been performed using Ansys CFX-11 using the geometry shown in Fig. 3. This allows a one-to-one comparison between the measurements from the sector test rig and the numerical predictions.

Inlet

The inlet pressure profiles, as a function of circumferential angle theta, are shown in Fig. 6. These profiles are taken using the kiel and static pressure taps available at inlet to the test section as discussed in the section titled “The Sector Test Rig”. The stagnation pressure measurements indicate a uniform inlet. It should be noted that the measurement plane at inlet is close to the leading edge of the preswirl vanes. The agreement between the static pressure measurements and the CFD prediction at this location is generally satisfactory with the influence of the potential fields due to the downstream preswirl vanes clearly evident. The slant of the profiles is believed to be attributable to the asymmetry of the test rig caused by the influence of the sidewalls. The turbulence intensity and integral length scale were measured at inlet and found to be 3.4% and 1.2mm respectively.

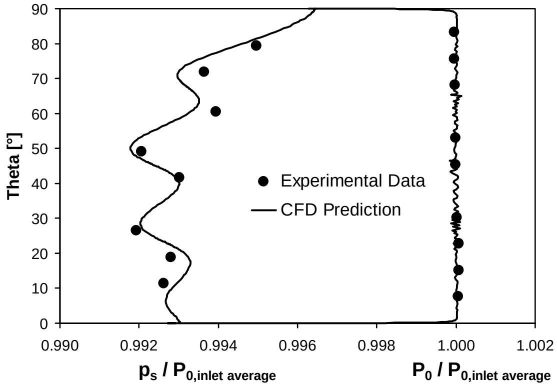


Figure 6: Circumferential inlet profiles of stagnation and static pressure from the sector test rig at 80% design flow coefficient (inlet plane). Stagnation pressure measurements are taken at mid-span, static pressure measurements at the shroud. Numerical predictions were performed using Ansys-CFX11.

Downstream of the Preswirl Vanes

Having considered the inlet to the test rig it is now appropriate to consider flow profiles at inlet to and exit from the deswirl vanes. It should be noted that all measurements and predictions concerning the sector test rig presented in this paper, with the exception of Fig. 5, are limited to the region around the middle preswirl or deswirl vane (1 pitch), where good symmetry has been established (discussed later).

Figure 7(a) presents contour plots whilst Fig. 7(b) illustrates area-averaged pressure profiles from traverse measurements and numerical predictions downstream of the preswirl vanes. As such, these profiles represent the inlet condition to the deswirl vanes. The contours appear qualitatively similar whilst the averaged profiles are generally in reasonable agreement. The stagnation pressure is in good agreement across the majority of the span with a deviation visible at the hub. There is a slight offset in terms of static pressure that is attributable to a slight deviation in yaw angles between the predictions and the measurements. This will be shown later.

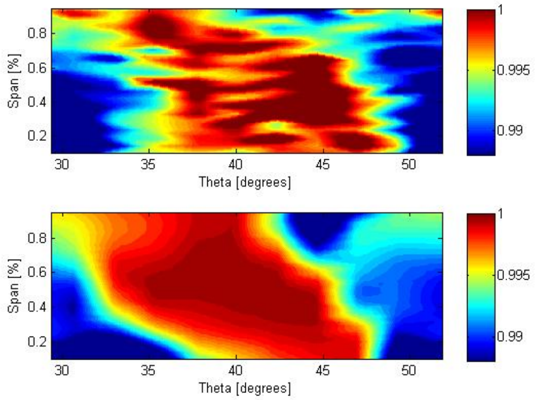


Figure 7(a): Sector test rig contour plots of measured (top) and predicted (bottom) stagnation pressure downstream of the middle preswirl vane (traverse plane 1) at 80% design flow coefficient. Numerical predictions were performed using Ansys-CFX11.

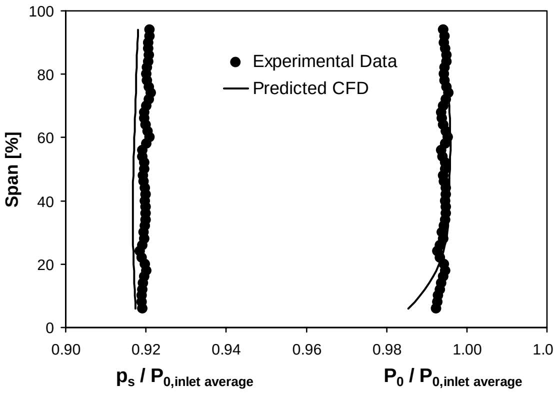


Figure 7(b): Sector test rig circumferentially area averaged profiles of stagnation pressure and static pressure downstream of the middle preswirl vane (traverse plane 1). Numerical predictions were performed using Ansys-CFX11.

Figure 8(a) illustrates the measured and predicted (pre-measurement) contours of yaw angle. It is noted that there is reasonable agreement between the two plots. The corresponding circumferentially area averaged spanwise profiles of yaw angle are shown in Fig. 8(b). In this case the agreement is good across the majority of the span (within the ±0.5° measurement error), however, a significant deviation is seen in the outer 30% of the span (towards the shroud). This deviation will be discussed in greater detail later.

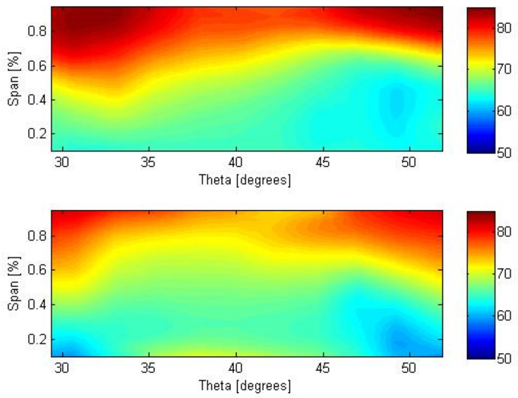


Figure 8(a): Sector test rig contour plots of measured (top) and predicted (bottom) yaw angle downstream of the middle preswirl vane (traverse plane 1) at 80% design flow coefficient. Numerical predictions performed using Ansys-CFX11.

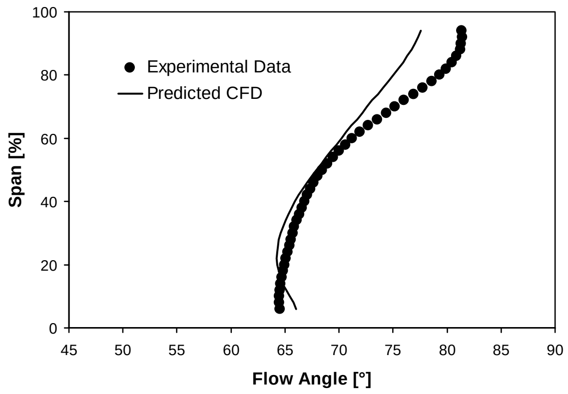


Figure 8(b): Sector test rig circumferentially area averaged profiles of yaw angle downstream of the middle preswirl vane (traverse plane 1). Numerical predictions using Ansys-CFX11.

Next, static pressure distributions from the deswirl vanes of the sector test rig and a corresponding stator vane in a full annulus rotating machine are considered. Figure 9(a) illustrates the measured static pressure distribution of the deswirl vanes at mid-chord and the periodicity of the two flow channels around the middle vane. Note that only the pressure surface of the left vane and the suction surface of the right vane are plotted. There is clearly reasonable agreement between the static pressure profiles, indicating good symmetry within the two flow channels (left and right of the deswirl vane) at the center of the deswirl section.

Figure 9(b) illustrates the numerical predictions and the measurements for the static pressure profiles of the middle vane at three spanwise positions. It should be noted that measurements are only available on the pressure side at 50% span. There is generally good agreement between the two sets of profiles.

Figure 9(c) presents two distinct sets of measurement/numerical prediction comparisons. First, the static pressure profiles from the measurements and the numerical predictions (using ANSYS-CFX 11) are overlaid. Second, the measurements from full annulus rotating rig test and CFD predictions from the in-house design code are overlaid.

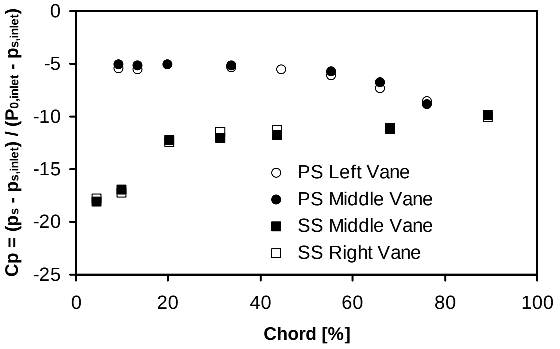


Figure 9(a): Sector test rig static pressure profiles at mid-span for deswirl vanes at 80% design flow coefficient. PS = Pressure Surface, SS = Suction Surface.

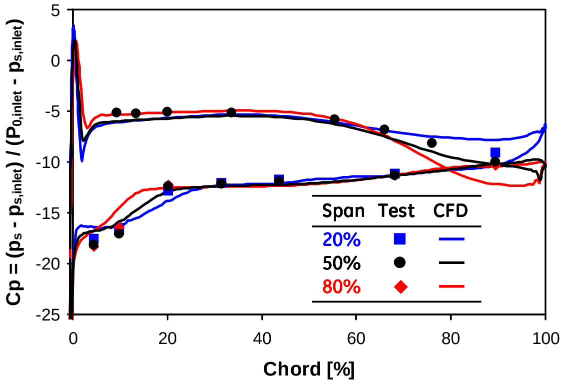


Figure 9(b): Sector test rig measurements and predictions of static pressure distributions of the middle deswirl vane at 20%, 50% and 80% span. Numerical predictions performed using Ansys-CFX11.

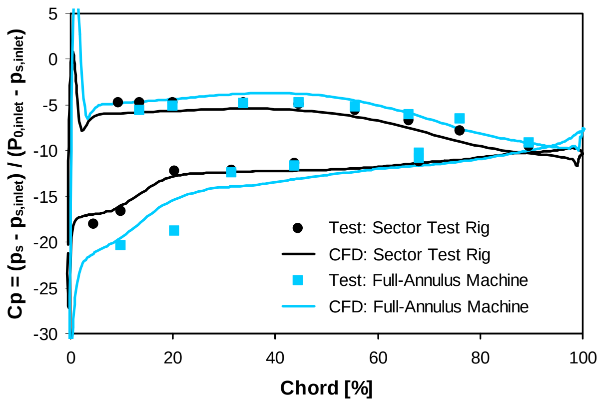


Figure 9(c): Measurements and predictions of static pressure of the middle deswirl vane in the sector test rig and of a stator vane of the full annulus rotating machine at 50% span, respectively. Full-annulus numerical predictions were performed using the in-house CFD code. Sector test rig numerical predictions was performed using Ansys-CFX11.

The deviation between the two sets of results (i.e. from the sector test rig and the full annulus rig) is caused by a difference in the boundary conditions between the two. The sector test rig was designed using pre-test (full annulus) numerical predictions of impeller exit velocity and yaw angle profile. It was found later on the basis of a rotating rig test that there was a reasonable disagreement between the assumed profiles and the measured profiles. More information regarding this subject can be found in [4].

Downstream of the Deswirl Vanes

At 80% of the design mass flow the return channel as operated in the steady environment of the sector test rig experiences a highly separated flow along the return channel vane, making it a particularly challenging environment to produce good agreement between numerical predictions and measurements. Improving this agreement is the objective of an ongoing study, therefore, only preliminary results are presented here. Early results from this study confirm that the return channel/deswirl vane is highly sensitive to the inlet condition at this mass flow and thus the deviation seen between the measurements and the numerical predictions downstream of the preswirl vanes (see previous section) is critical.

Figure 10(a), (b) and (c) illustrate the predicted and measured circumferentially mass averaged spanwise profiles of pressure, yaw angle and axial Mach number downstream of the deswirl vanes in the sector test rig respectively. In the case of the pressure (non-dimensionalized by the inlet stagnation pressure), the measurements and numerical profiles are in fair agreement with the hub section resolved quite well. There is however a slight offset between the measurement and numerical data towards the shroud.

Figure 10(b) shows the yaw angle as a function of span with measurements plotted alongside CFD predictions of the entire test rig and predictions of the rotating stage calculation. Considering first the measurements and numerical predictions of the test rig itself it is apparent that whilst the general "wave-like" trend in the profile is captured the CFD predicted profile appears exaggerated when contrasted with the measurements. The rotating test prediction is included to show the impact of the inlet boundary conditions on the flowfield downstream of the vane. As one might expect, [5], the flow conditions at the exit of the deswirl vanes are highly sensitive to the conditions at the inlet to the deswirl vanes. This emphasizes that accurate prediction of the flow condition at the leading edge of the present highly separated return channel vane is essential to obtaining good predictions at the deswirl exit.

For completeness Fig. 10(c) shows measurements and predictions of axial Mach number as a function of span. The agreement is again fair considering the challenge associated with predicting highly separated flows and the sensitivity of the predicted flow at the stage exit to the predicted inlet conditions at the deswirl section.

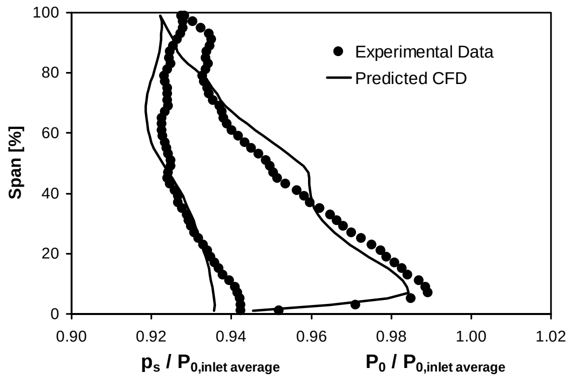


Figure 10(a): Mass averaged pressure profiles downstream of the middle deswirl vane (traverse plane 2) from measurements and numerical predictions. Numerical predictions performed using Ansys-CFX11.

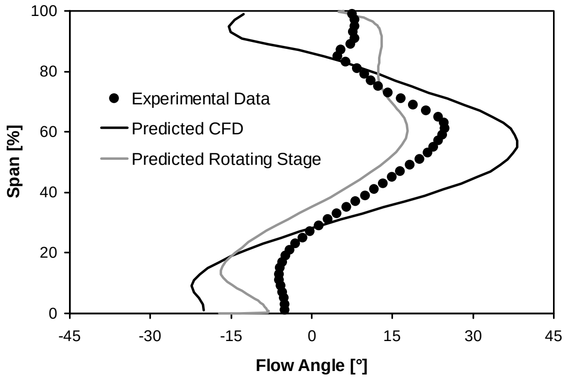


Figure 10(b): Mass averaged flow angle profiles downstream of the middle deswirl vanes (traverse plane 2) from measurements and numerical predictions. Numerical predictions of the sector test rig (labeled predicted CFD) were performed using Ansys-CFX11. The rotating stage numerical predictions were conducted using the in-house design code.

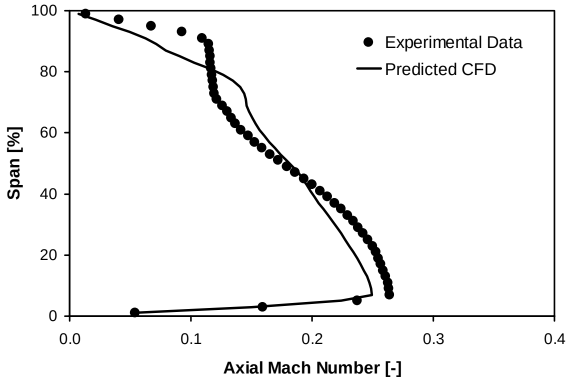


Figure 10(c): Mass averaged flow angle profiles downstream of the middle deswirl vanes (traverse plane 2) from measurements and numerical predictions. Numerical predictions performed using Ansys-CFX11.

RESULTS AT 70% DESIGN FLOW COEFFICIENT

This section will consider measurements and numerical predictions downstream of the preswirl vanes and downstream of the deswirl vanes for the configuration corresponding to 70% design flow coefficient. This was achieved by adjusting the conditions at the inlet to the test rig and by exchanging the preswirl vanes in the sector test rig with a new set of vanes that was designed to deliver yaw angle and velocity distributions corresponding to the 70% operating point in the rotating test configuration. In all of the plots the hub lies at 0% span and the shroud lies at 100% span.

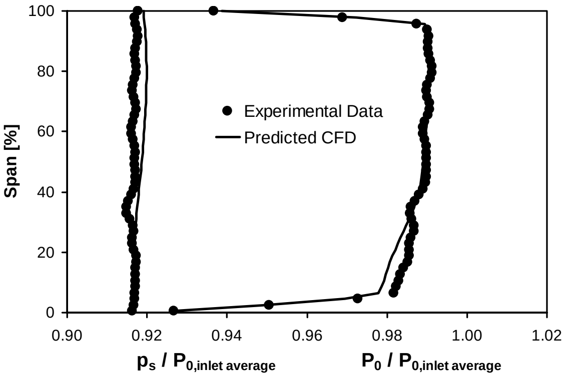


Figure 11(a): Spanwise mass averaged profiles of pressure downstream of the preswirl vanes (traverse plane 1) at 70% of the design flow coefficient. Numerical predictions performed using Ansys-CFX11.

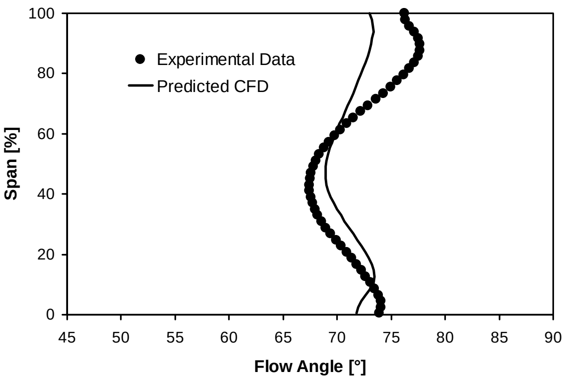


Figure 11(b): Spanwise mass averaged profiles of flow angle downstream of the middle preswirl vane (traverse plane 1) at 70% of the design flow coefficient. Numerical predictions performed using Ansys-CFX11.

Figure 11 illustrates measured and predicted circumferentially mass averaged spanwise profiles of total and static pressure (Fig. 11(a)) as well as yaw angle (Fig. 11(b)) downstream of the preswirl vanes. The agreement between measurements and predictions is found to be good for both total and static pressure. The measured yaw angles agree qualitatively well with the desired (=predicted) distribution. Deviations between measurements and predictions are within measurement uncertainties across a large part of the span, however, similar to the 80% configuration, significant under prediction by CFD can be observed towards the shroud.

Figure 12 plots the measured and predicted static pressure distribution at mid-span as a function of chord for the deswirl vane at 70% of the design flow coefficient. There is qualitatively good agreement between the measurements and the numerical predictions. The slight offset between the two sets of data, which is somewhat larger than the 80% case, is believed to be attributable to the difference in the inlet yaw angle.

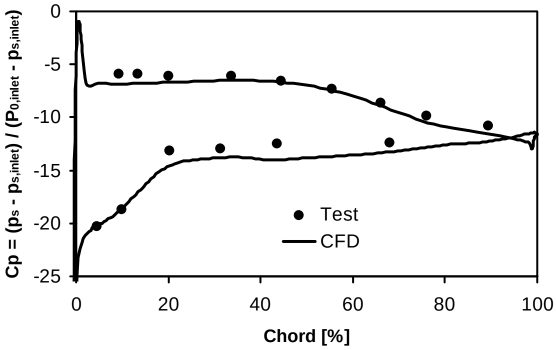


Figure 12: Measurements and predictions of static pressure profiles at mid-span for the sector test rig deswirl vanes at 70% design flow coefficient. Numerical predictions performed using Ansys-CFX11.

THE EFFECT OF SURFACE FINISH

In order to investigate the effect of vane surface finish on the pressure loss across the return channel of the centrifugal compressor, two separate sets of vanes were prepared for use in the present test rig. The two sets of vanes (rapid prototyped from an epoxy resin) were identical apart from the quality of the surface finish. The first set of vanes had a relatively rough surface finish achieved using 320-grade sandpaper. The second set of vanes had a smoother finish that was achieved using wet 800-grade sandpaper and a polishing process. The pressure loss coefficient across the deswirl vanes is defined in terms of a loss coefficient which is defined as:

$$\xi = \frac{P_{0,\,\text{Deswirl Inlet}} - P_{0,\,\text{Deswirl Exit}}}{P_{0,\,\text{Deswirl Inlet}} - p_{s,\,\text{Deswirl Inlet}}} \qquad (1)$$

Figure 13 plots the loss coefficients (calculated from measurement data) across the vanes as a function of flow coefficient for the two different vane surface finishes.

Clearly, there is a marked and consistent difference between the loss coefficient measured for the rough and smooth vanes. In each case the smooth vane exhibits a lower loss coefficient. In order to check the repeatability of the loss coefficient measurement a second set of deswirl vanes with the rough surface finish was tested and the measurement data was found to be consistent with the first set. It is thus concluded that the surface finish has an important impact on the loss through the deswirl vanes. This is in line with observations reported in [6] where the impact of surface roughness on the performance of axial compressor stators was considered. This phenomenon is the subject of an ongoing investigation.

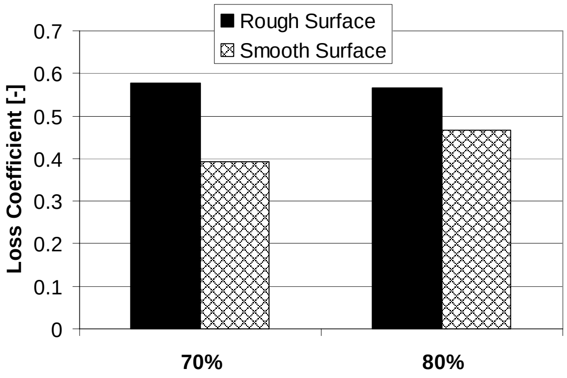


Figure 13: Measured loss coefficient across the deswirl section of the sector test rig for two different vane surface finishes each at 70% and 80% design flow coefficient.

CONCLUSIONS

The use of a sector test rig as a test vehicle for radial compressor stator vanes has been discussed. Measurements taken at different locations throughout the rig have been presented and compared to the results from numerical predictions and rotating test data. This has shown that:

- The preswirl section of the sector test rig produces the desired flow distribution for the stator vanes for both the 70% and 80% design flow coefficient configurations.
- The measurements in the rotating test rig and the sector test rig show reasonable agreement with differences attributable to a difference in the inlet boundary condition.
- The agreement between measurements and numerical predictions is generally quite good. Predicted flow details near the shroud and downstream of the highly separated deswirl vanes need further refinement/calibration.
- Surface roughness of the stator vane surface has an important effect on the measured loss coefficient across the deswirl section.

It is concluded that the sector test rig provides the designer with a useful tool for screening potential stator vane designs for radial compressors.

ACKNOWLEDGMENTS

The work reported in this paper was partially funded by the Bavarian Ministry of Economics under the contract number 0705/89375/1119/05/3910/06/3911/07. The contributions and comments of Kevin Kirtley, Carlos Haertel, Georg Toplack are gratefully acknowledged. The tests utilizing the rotating rig were performed within the test facilities of GE Oil & Gas, Italy. The authors would like to make special mention of Carlo Bacciottini and his team for their efforts. The authors would also like to thank GE Oil & Gas for their support and permission to publish the results.

REFERENCES

[1] Opoka, M.M., Hodson, H.P., 2007, "Transition on the T106 LP Turbine Blade in the Presence of Moving Upstream Wakes and Downstream Potential Fields" ASME GT2007-28077.

[2] Schmitz, M.B., Simpson, A., Aalburg, C., Michelassi, V., "Development of a Sector Test Rig for Radial Compressor Stators", Isromac12-2008-20254.

[3] Cumpsty, N.A., 1989, "*Compressor Aerodynamics*", Longman Scientific and Technical, ISBN 0-582-01364-X.

[4] Aalburg, C., Simpson, A., Schmitz, M., Michelassi, V., Bacciottini, C., "Design and Testing of Multistage Centrifugal Compressors with Small Diffusion Ratios", GT2008-51263.

[5] Japikse, D., 1996, "*Centrifugal Compressor Design and Performance*", Thomsom-Shore Inc.

[6] Gbadebo, S.A., Hynes, T.P., Cumpsty, N.A., 2004, "Influence of Surface Roughness on Three-Dimensional Separation in Axial Compressors", ASME Journal Of Turbomachinery, Vol. 126, pp. 455-463.