FLOW PHENOMENA IN A VANED DIFFUSER
OF A CENTRIFUGAL STAGE

P. Dalbert
Sulzer-Escher Wyss Limited
Zurich, Switzerland

G. Gyarmathy
Swiss Federal Institute of Tech.
Zurich, Switzerland

A. Sebestyen
Sulzer-Escher Wyss Limited
Zurich, Switzerland

Abstract: The 3D viscous flow code BTB3D of Dawes has been applied to a standard geometry of a vaned diffuser of an industrial centrifugal compressor stage. The objective of the work was to check the ability to calculate the performance of cascade diffusers and to investigate possibilities of representing the results for a better understanding of the flow phenomena. The computations are compared with the measurements at different operating points. The theoretical results are represented in numerous 2D and 3D illustrations for the description and comparison of the design and off-design flow fields within the investigated diffuser. They show that the knowledge of the wall pressures and wall streamlines does not fully reveal the extremely complex 3D flow within such a diffuser. Moreover, the technique of flow visualization by using moving pictures on a video film was applied, aimed at enhancing the physical understanding of the flow effects in vaned diffusers.

Nomenclature

b2: diffuser width
c: flow velocity
D2: impeller tip diameter
i: incidence = "flow angle" minus "vane angle"
Mu2: tip speed Mach number = u2/\(\sqrt{\gamma R T_0^*}\)
p0: inlet stagnation pressure
R: gas constant
r*: reduced radius = r/r2
r2: impeller tip radius = D2/2
s: length
s(max): diffuser blade length
T0*: inlet stagnation temperature
u2: blade tip speed
\(\gamma\): ratio of specific heats
\(\Pi\): compressor total pressure ratio

Subscripts

2: impeller outlet
c: cross component
h: hub
i: incidence
m: mean
s: shroud

Introduction

Vaned diffusers in radial stages of industrial turbocompressors produce about 40 to 50% of the enthalpy rise of the stage. In spite of a reduction in range, the vaned diffuser is used frequently by many manufacturers. Due to its higher pressure recovery capability it leads to a higher stage efficiency and has the advantage of a comparatively limited radial extent. A well documented and successful design procedure for vaned diffusers has probably never been developed. Most of the published data on diffusers is restricted to comparisons of performance characteristics and to the development of empirical correlations for loss and pressure recovery, as was collected and reviewed e.g. by Japikse (1984). This leads to the fact that the design of a new radial diffuser is a time consuming mixture of applications of proven geometries, experience and correlations, where the latter are very often out of their tested range, instead of a clear and fast scientific procedure. Likewise, the many measured isobars of static pressure in diffuser channels as shown by Yoshinaga et al. (1980) can only provide limited information on the flow pattern. Many of the performance and flow field measurements in diffusers were carried out in 'vortex rigs', as shown by Dutton et al. (1986) and Brownell et al. (1987) or by Davis and Flack (1990), which can only provide some of the information necessary to understand the diffuser flow of a radial stage. The very significant variation of the flow field at the impeller exit at different operating points is missing in these cases. Seno et al. (1986) or Starke and Hergt show among other things wall flow patterns in vaned diffusers of a radial compressor stage respectively a centrifugal pump. But even wall flow patterns cannot give sufficient information for a thorough assessment of the complex diffuser flow field. This explains the difficulties encountered, if the assessment of the performance of a radial diffuser in a centrifugal stage is based only on measurements of the three-dimensional flow field, and provides the reason why the use of 3D viscous codes are of utmost interest. The past decade has brought forth numerous computational codes for the calculation of the fully turbulent flow in arbitrary shaped 3D flow boundaries. Many of these codes have proven their practical usefulness in the calculation of the flow in turbocompressor components. The present work was primarily aimed at testing the capability of theoretical performance calculations in the case of a radial vaned diffuser. Additionally, the authors' activities have been focused on developments in the field of presentation methods for the computed flow data. Efficient presentation of the computed results is an absolute requirement for the interpretation and a better comprehension of the highly complex flow phenomena in this as well as other investigated parts of turbomachines.
In a scientific cooperation with the Swiss Federal Institute of Technology numerical flow calculations were carried out within a vaned diffuser of an industrial centrifugal stage used typically in multi-stage compressors. The well known computational codes by J. Denton (1986) and W. N. Dawes (1988) for fully three-dimensional, turbulent flow were applied. The results of Dawes' BTOB3D code are used for the final presentation.

The theoretical results are compared with extensive experimental data at three different operating points. Moreover, the results of flow computations have been graphically visualized in order to inspect and compare design and off-design flow fields within the investigated diffuser.

The results of the flow visualization were animated on a video film to enhance the physical understanding of the flow effects in cascade diffusers.

Test stand and reference operating points

The measurements have been carried out in a single stage centrifugal test rig at the Swiss Federal Institute of Technology in the frame of a research program sponsored by the Swiss government. The test stand used for the measurements was described in detail by Gyarmathy et al. (1991). It is a closed loop single stage centrifugal compressor test rig with an impeller diameter of $D_2 = 280$ mm. Figure 1 shows the cross section of the stage consisting of an axial inlet, the impeller with 30° back leaned blading, a vaned diffuser and a large toroidal collector which guarantees a nearly constant pressure distribution over the circumference.

The width $b_2$ of the vaned diffuser is 16.8 mm. There are 24 blades with a constant thickness of $d_2 = 0.021$. The circular arc leading edge is positioned at a radius ratio $r^* = 1.15$, the blunt trailing edge at $r^* = 1.54$. Within the diffuser 81 static pressures taps are located in the front wall ('shroud') to determine the pressure distribution within one of the diffuser channels. Another set of static pressure taps is distributed at mid span in two blades to measure the pressure rise on both sides of the blades. During the research program a very wide range of impeller tip speeds with different diffuser angles as well as with vaneless diffusers has been measured with this stage, see Hunziker (1990). In order to provide reference data for the calculations three operating points have been selected from the performance map. For these measurements the diffuser inlet blade angle was set to $25°$ resulting in a blade outlet angle of about $39°$ with respect to the circumferential direction. Figure 2 shows the three selected points $/A/$, $/B/$ and $/C/$ on the constant speed line corresponding to the tip speed Mach number $\mu_2 = 0.75$.

Computations

The calculations were carried out with Dawes' BTOB3D Navier-Stokes solver (Dawes, 1988). The computational mesh, shown in Figure 3, consists of $19 \times 79 \times 19$ gridlines in the pitchwise (I), streamwise (J) and spanwise (K) direction resulting in 28,519 nodes.

The relatively coarse grid led to acceptable calculating times considering the fact that up to 5000 time steps had to be used to achieve an reasonably converged solution. The necessary boundary conditions at diffuser inlet for the calculations in terms of the circumferentially averaged values across the span of flow angle and stagnation pressure are taken from measurements with a vaneless diffuser at the operating points $/A/$, $/B/$ and $/C/$. Figure 4, where detailed flow measurements at the impeller exit were carried out. For this purpose three operating points seemed to be sufficiently near to the operating points with the vaned diffuser of Figure 2. The static pressure at diffuser outlet has to be specified at the exit boundary of the computational mesh at $r^* = 1.935$. The values relative to the mean value of the inlet stagnation pressures were for the three operating points $/A/$, $/B/$ and $/C/$: $p/p_0 = 0.95$, 0.973 and 0.934. This resulted at $r^* = 1.786$ in absolute static pressure values at hub of $p = 1.517$, 1.513 and 1.384 bar, while the three reference points from the vaned diffuser measurements are $p = 1.515$, 1.505 and 1.338 bar.
The specification of the boundary conditions at the inlet to the diffuser on the basis of available measurements was a major problem at the beginning of these calculations. The first streamwise gridline at J = 1 is located at \( r^* = 1.015 \). The cylinder probe used for the measurement of the spanwise distribution of stagnation pressure and flow angle was however mounted at \( r^* = 1.050 \). This means that the boundary conditions at \( r^* = 1.015 \) had to be determined such as to reproduce as good as possible the measured distributions at \( r^* = 1.050 \). This turned out to be only possible by dropping the "naive" assumption of flow angle conservation between the inlet boundary and the measurement plane, especially in zones near to the walls. In these areas small regions of backflow had to be allowed for case /A/ and (near the hub only) for /B/. As an example, Figure 5 compares the stagnation pressure and flow angle specified at 1.015 and obtained at 1.05 to those measured at 1.05 in the test stand for the case /B/.

Fig. 5 Inlet boundary conditions for case /B/

The specification of the boundary conditions at the inlet to the diffuser on the basis of available measurements was a major problem at the beginning of these calculations. The first streamwise gridline at J = 1 is located at \( r^* = 1.015 \). The cylinder probe used for the measurement of the spanwise distribution of stagnation pressure and flow angle was however mounted at \( r^* = 1.050 \). This means that the boundary conditions at \( r^* = 1.015 \) had to be determined such as to reproduce as good as possible the measured distributions at \( r^* = 1.050 \). This turned out to be only possible by dropping the "naive" assumption of flow angle conservation between the inlet boundary and the measurement plane, especially in zones near to the walls. In these areas small regions of backflow had to be allowed for case /A/ and (near the hub only) for /B/. As an example, Figure 5 compares the stagnation pressure and flow angle specified at 1.015 and obtained at 1.05 to those measured at 1.05 in the test stand for the case /B/.

Fig. 5 Inlet boundary conditions for case /B/
Comparisons

Figure 6 represents measured (left side) and calculated (right side) static pressure distributions on the diffuser shroud. Due to the much higher number of grid points in the calculation the resolution of the isobars is better on the computed plots than on the measured ones which allows, of course, a much more detailed analysis of the calculated pressure distributions.

At the operating point near mild surge /A/ (top) the measured and computed isobars within the vanned diffuser channel (i.e. downstream of the throat) agree quite well although the measured value of 1.5 bar is reached by the computation farther downstream in the vaneless diffuser exit.

In the "semi-vaneless" triangle directly upstream of the throat the better resolution of the calculation becomes obvious. The steep pressure gradient around the suction side corner, reaching down to a computed minimum of 1.13 bar and followed by steep pressure increase, can only be identified in the computed plot. Such steep pressure gradients, which extend in fact over the whole span of the diffuser, usually signify a high risk of separation.

Another interesting point is the alignment of the isobars in the triangle. The measured isobars are more or less cylindrically aligned with respect to the axis of rotation of the impeller and therefore oblique with respect to the flow, whereas the computed isobars are perpendicular to it. It is not clear if the reason is the much lower minimum pressure at mid span at high massflows. This effect must be approximately the same downstream gradient which moves the points of equal pressure further downstream on the blade suction side compared with the experiment, or if the pressure gradient itself is not reproduced correctly. This phenomenon is less strongly marked at mid span but it indicates, nevertheless, some discrepancies between calculated and actual flow patterns.

At operating point /B/ the calculated isobars show similar tendencies as discussed above. However, generally there is a difference of about 60 mbar between the theoretical and experimental pressure levels at the inlet. This can be explained as follows: The massflow is determined by the inlet stagnation pressure and temperature, the inlet flow angle and the outlet static pressure. Since the inlet angle corresponds well with the measured value (see Figure 5) the lower inlet static pressure a higher massflow of the calculated operating point. An additional uncertainty arises from the fact that the flow angle at diffuser inlet has to be preselected. However, the calculated volume flow is approximately the same as in the measurements, whereas the massflow is generally too high, indicating that the computed losses are underestimated.

The operating point near choke /C/ shows the best agreement between experiment and theory. Also the inlet pressure level is very similar in both cases. Both show a strong low pressure bubble on the pressure side (upper side) at the leading edge of the blade as a consequence of the high incidence at mid span at high massflows. This effect must be and is even stronger at mid span, as the calculations show. This feature of the static pressure field which as in case /A/ is by no means constant across the span, is a surprising observation since the incidence at the shroud is still negative (see below in Figure 9). The deep pressure "hole" in the throat on the pressure side of the blade leads to a steep pressure gradient along the blade surface and across the throat. The first should lead almost necessarily to a separation on the pressure surface whilst the second causes a secondary flow in regions of low energy from the suction toward the pressure surface.

The calculated static pressure rise along the vane at mid span is shown in comparison with the measured values in Figure 7. The peaks of low and high pressures around the leading edge indicate the positions of the highest velocity and the stagnation point. In general, the computed and measured curves show a good agreement and the same tendencies and similar pressure gradients. In some cases the pressure levels at the first measurement points are about 50 to 100 mbar higher than the calculated values which has to be reduced to the too high values of the input static pressure at the exit and some possible errors in the initial values of the flow angle.

It is interesting to note that in case /C/ the flow on the suction side is accelerated into the throat at about 45° of the relative blade length, followed by a rather rapid diffusion resulting in a steep pressure gradient up to the pressure level of the pressure side. But, since there is no pressure rise in the "semi-vaneless" triangle, and the pressure rise within the channel is only marginally increased from case /A/ to /C/, the overall pressure rise of the diffuser in case /C/ is much less than in case /A/ and /B/.

Flow visualization

The computed flow data have been postprocessed on a Silicon Graphics Workstation in order to show the flow patterns within the diffuser by plotting three-dimensional streamlines. Wall streamlines can often be observed in machines, e.g. when multistage centrifugal compressors in the test bed have to be reopened after mechanical tests. They usually were in operation for several hours at maximum massflow with the stages operating near choke. In Figure 8 the calculated streamlines near choke /C/ and corresponding oil traces show that such a stage show a very good qualitative agreement.

This picture represents a good example for the 3D flow field of a radial diffuser and the difficulty to check the matching of the impeller and diffuser and by particle tracing. The oil traces show e.g. in the vaneless inner part of the diffuser very low flow angles with respect to the circumferential direction, corresponding to the measured low flow angles near the walls. Since the stage was operated at its...
Fig. 8 Calculated streamlines over the blade span and oil traces on a diffuser hub (near choke, case /C/)

maximum flow rate the mean flow angle is large with respect to the circumferential direction. Within the diffuser channel the trace of oil particles is almost identical to the calculated streamlines, starting at different positions in the lower part of the span and forming the boundary of a large wake in the channel. Since there is no separation in the diffuser channel at high massflow rates the observed oil particles must have been transported into the wake along the pressure side by a strong secondary flow within the boundary layers. The particles are then deposited along a line where the secondary flow movement meets the streamlines starting at the leading edge on the blades.

As noted before, the experimental examination of the extremely complicated flow fields of radial diffusers is a very time consuming task and is therefore very seldom carried out. An evaluation of the static pressure rise or pressure distribution on the diffuser walls and the vane surfaces provides important information about the pressure rise in this flow element, but provides very little reference to the velocity distributions. Also wall streamline patterns cannot reveal all the 3D effects in the stream channels.

However, deeper comprehension of the flow structure in a radial diffuser seems to be the only way to improve modern designs. Therefore a method was developed to visualize the three dimensional flow patterns at different operating points of the stage. The selected method in this case is the representation of the results of the 3D calculations discussed above by means of computational "particle tracing" and recording this sequence of pictures on a video tape. The individual sections of the video film produced show the flow patterns in the vaned and vaneless parts of the diffuser near the three operating points "surge", "design" and "choke".

To be able to summarize the impressions gained from this film we have to go back to two dimensional vector or static pressure diagrams and try to explain some of the flow phenomena. Figure 9 shows the velocity vectors, from top to bottom, near the hub, at mid span and near the shroud, each for the three operating points /A/, /B/ and /C/. All velocity vectors are scaled with the same reference value.

Fig. 9 Velocity vectors in the diffuser channel
At midspan there is very little qualitative difference between the three operating points. The incidences vary from about \( i_{A} = -5^\circ \) to \( i_{B} = 0 \) to \( +2^\circ \) and \( i_{C} = +10^\circ \), as shown in Figure 11. With the exception of a small region of low velocities in case \( A \) near the trailing edge on the suction side there is no flow separation noticeable on the blades. This fact should be recognized keeping in mind the pressure distributions of Figure 6 which indicated a possible flow separation mainly in case \( /A/ \). There are, of course, regions of low flow velocities at the suction side in case \( /A/ \) and at the pressure side in case \( /B/ \).

Near to the walls big differences can be recognized in the flow patterns from surge to choke, mainly created by the very different flow distributions leaving the impeller.

In case \( /A/ \), there is a very small layer of backflow on the hub side (about 2% of the span, Figure 11) and a relatively large zone of backflow near shroud (about 12%). The particle-tracing technique reveals a circumferentially oriented wave-like motion in the vaneless part of the diffuser. Near hub a small region of backflow in the "semi-vaneless" triangle and very low flow velocities in the channel can be identified. Near shroud there is an extended region of backflow turning into a cross-flow towards the impeller rotation. This can be visualized by using tracing particles which enter the diffuser at a level of about 70% of the span and are pushed by a strong secondary flow into this region of backflow. They leave the diffuser upstream instead of downstream as demonstrated in Figure 10. In this layer there is very little throughflow behind the throat with the exception of a small region near the pressure side.

In case \( /B/ \) the flow recovers very quickly from the low inlet flow angle existing near hub and shows a very smooth velocity distribution downstream of the throat as well as at mid span. Near shroud the large zone of negative incidence (about 40%, see Figure 11) leads to a backflow region on the suction side where a very high pressure gradient is found in Figure 7.

Near choke, case \( /C/ \), where the flow is accelerated into the throat, the negative incidence near the shroud does not lead to separation. Figure 11 shows the calculated incidence distributions just upstream of the leading edge. In the first place it may be surprising that there are not more flow separations taking into account the extremely low flow angles produced by the impeller. Case \( /B/ \) shows a shear-flow gradient spanning almost from wall to wall. Traupel (1977) discusses the behavior of an inlet shear-flow in the diffuser. He represents the flow as shown in Figure 12 in a superposition of a basic flow with the mean velocity \( \bar{c} \) and the two velocities \( c_{s} \) and \( c_{h} \) at shroud and hub which create the two opposing components \( c_{i} \) near the walls. Due to the presence of the blades this leads to a secondary flow which is characterized by the components \( c_{i} \) and two cross components \( c_{c} \).

In low flow, wide throat diffusers, with a span to throat ratio of about 0.9 as used in the discussed diffuser, small \( c_{c} \) components will suffice to reduce the \( c_{i} \) to zero near the blade walls. Therefore, Traupel argues, in such diffusers the secondary flow pattern can be created without flow separation. This argument is strongly supported by the computational findings. However, in the investigated cases we observe that the components \( c_{i} \) near the side walls point to the same direction due to the low flow angles at both walls, as sketched in Figure 12, (II). At about 30% of the span from hub to shroud the flow angle reaches its maximum, leading to a component \( c_{c} \) opposing the components \( c_{i} \) near hub and shroud. This situation leads to a more complex secondary flow pattern in the diffuser channel. It consists of two opposing vortices which have, as in case (I), the effect of energizing the flow near the
blade surfaces and therefore help the flow to follow the channel geometry without separations. This may be the reason, after all, why this kind of diffusers are able to produce as much as 40 to 50% of the pressure rise of the stage. This occurs in spite of the often very unfavorable flow leaving the impeller mainly at off-design conditions. The secondary flow vortices can be observed very well by particle tracing.

Conclusions

The results of 3D viscous flow computations with Dawes' BTOB3D code are compared with measurements and the three-dimensional flow patterns are visualized. The main conclusions of this work are:

• Since the times of Yoshinaga (1980) not very much has changed in the fact that the design method of the diffuser section has not been well established and there are still many unknown parameters which hinder an improvement of this element of a centrifugal stage.
• The extensive published data on the pressure recovery of two-dimensional diffusers has never been consequently applied to the vaned radial diffuser design. This makes it very difficult to design a vaned diffuser for the optimum design point, let alone for off-design operating points.
• In spite of the fact that the calculations have been carried out at different massflow rates than the massflow of the measured operating points, the computed pressure distributions show a good qualitative agreement to the measurements in all three operating points. This agreement is a known fact and has been established also with results of potential flow analysis. This can lead to the assumption that viscous effects have very little influence on the pressure distributions at the vanes and the walls; a fact which was mentioned already by Starke and Hergt.
• This fact is emphasized by the observation that the pressure distributions do not noticeably reflect the complicated flow pattern in the diffuser. Foremost it has to be noted that the huge backflow regions in the operating point near surge and the backflow zone in the design point are not indicated by the pressure distributions.
• Also the secondary flow motion which stabilizes the flow in the diffuser cannot be attributed to the overall pressure distributions.
• The flow visualization and the pressure recovery analysis show that the most critical area in a radial vaned diffuser is the inlet triangle or the "semi-vaneless" part, where at all operating points the highest pressure gradients and the strongest backflow phenomena occur.
• Since the flow pattern in the diffuser is strongly influenced by the velocity and flow angle distribution at the impeller outlet an optimization of a diffuser can only be carried out by considering these 3D flow phenomena.
• Finally it seems questionable if in view of the complicated flow phenomena discussed above the calculation or the measurement of the pressure distribution alone can give any reliable information about the aerodynamic quality of the diffuser, especially with respect to off-design conditions.

Acknowledgments

The authors wish to thank Sulzer-Escher Wyss, Ltd., for the permission to publish this paper. We are also indebted to the research teams at the Institute of Energy Technology, especially to R. Hunziker for many discussions and suggestions concerning this paper as well as for his help in preparing the isobar plots. Moreover we are indebted to the Interdisciplinary Project Center for Supercomputing at the Swiss Federal Institute of Technology for their help in producing the video film. Thanks also to the Swiss Government (Kommission fuer die Foerderung der Wissenschaftlichen Forschung) for the financial support of this project.

References


Foerderung der Wissenschaftlichen Forschung) for the financial sup-

Gyarmathy, G., Hunziker, R., Ribi, B., Sprig, M., 1991, "On the change of impeller flow non-uniformities with flow rate in a cen-

Hunziker, R., Ribi, B., Sprig, M., Gyarmathy, G., "On the influ-


Starke, J., Hergt, P., "An experimental investigation of three dimen-

