NUMERICAL INVESTIGATION OF INTERNAL FLOW FIELD FOR MODIFIED DESIGN OF ECKARDT BACKSWEPT IMPELLER

JongSik Oh
Korea Institute of Machinery & Materials
P.O.Box 101, Yusong, Taejon, REPUBLIC OF KOREA

ABSTRACT
The three dimensional blade shape of Eckardt's backswept impeller was modified, expecting better aerodynamic performance of the internal flow field. Blade angle distributions and parts of the meridional contours were changed, while impeller diameter, blade number, blade thickness and blade inlet and exit angles remained unchanged. The casing contour in the vaneless diffuser is additionally changed in a smooth manner to obtain 15% pinched flow channel at the exit. With the help of the three dimensional compressible Navier-Stokes analysis method, some improvements in the aerodynamic characteristics of the internal flow field were found; a more uniform flow field in the circumferential direction at impeller discharge was established, and a more favorable rise of static pressure near the casing in the impeller passage was made. A more effective increase of static pressure in the vaneless diffuser was also found. But a less uniform flow field in the spanwise direction at impeller exit resulted. The same levels of total-to-total pressure ratio and isentropic efficiency of the compressor were obtained, while an increase of choking flow rate was obtained due to increased throat area.

NOMENCLATURE
\( p \) static pressure
\( P_{i} \) inlet total pressure
\( W \) relative velocity
\( U_{t} \) blade tip speed
\( C_{m} \) meridional absolute velocity
L.E. leading-edge
T.E. trailing-edge
P.S. pressure side
S.S. suction side
\( PROT \) rotary total pressure (\( = p + 0.5p(W^{2} - U^{2}) \))
\( CMU_{2} \) \( = C_{m}/U_{t} \)

1 Senior Researcher, Ph.D.
Dept. of Thermal-Fluid & Environment Research

INTRODUCTION
Two kinds of centrifugal impellers whose internal flow fields were experimentally investigated by Eckardt(1980) have been known and helpful to aerodynamic designers and/or analysts of centrifugal compressors. Detailed test data of typical impellers with radial and backswept bladed discharge are available, from which the complicated internal flow field can be more understood. Especially three dimensional flow separation near the shroud, three dimensional non-uniformity, flow structure of the jet/wake at impeller exit plane, generation of secondary flow, and flow losses with slip distributions were investigated with the application of a laser velocimeter. Those experimental results have been utilized as a fundamental base for the design of centrifugal impellers, and also for validation of any developed methods for numerical flow analysis.

The backswept impeller is, however, a simple alternative version of its former impeller with radial ending blades, with the same shroud contour, the same exit width, and the same number of full blades. The backward curvature starts at a radius ratio of 0.8 and ends at a blade exit angle of 60 degree from tangential. It is easily found that the backswept impeller seems not to have been designed with optimal design consideration of conventional backswept impellers. Therefore, it becomes natural to find out the original design concepts, and to try to modify the design for better aerodynamic performance.

In the present study, the expected improvements in performance are restricted to the internal flow field of the impeller passage, because the main concern is how to design the blade shape. There are two prescribed conditions before re-design; (a) overall dimensions (e.g., inlet and exit diameters, exit width, axial length, and blade thickness) are unchanged. (b) inlet and exit blade angles are unchanged. The re-design is made, for the Eckardt backswept impeller, with minor changes of meridional
contours and considerable changes in the blade angles, with the help of a systematic design procedure. For convenience, the re-designed impeller is hereinafter called "Modified Impeller", and the Eckardt backswept impeller is hereinafter called "Original Impeller".

**DESIGN AND ANALYSIS METHODS**

In the design of three dimensional blade shapes, there are three major design parameters; shroud/hub meridional contours, blade angle distributions and blade thickness distributions. In the initial step, they can be assumed separately to be assembled for the construction of three dimensional impeller blade shape. But its acceptance should be repeatedly judged by the distributions of blade loadings and static pressure from any numerical analysis methods. For saving design time, approximate methods of calculating surface velocities, such as methods by Stanitz (1951), M. C. Farland (1984) and Howard, et al. (1994), are usually applied. In the final step, quasi-three dimensional compressible inviscid flow analysis (Hirsch, 1978, and Oh, 1992) and/or three dimensional compressible Euler flow analysis (Denton, 1983) and/or three dimensional compressible viscous turbulent flow analysis (Dawes, 1988) are applied to complete the aerodynamic design.

The systematic three dimensional design method of an impeller blade used in the present study consists of the three dimensional generation of the geometric profiles and the approximate calculation of blade loadings. Shroud and hub contours and blade angle and normal thickness distributions are described by Bernstein-Bezier polynomials, like a method by Casey (1983). For the analysis of the original impeller, they are constructed through fitting procedures using simple input of the given geometry. Corresponding inviscid blade loadings are calculated by a single-pass computation through the use of a single streamtube model by defining streamtube slopes and curvatures, as described by Howard, et al. (1994).

In the final step of design/analysis, three dimensional compressible Navier-Stokes flow analysis is applied using a conventional time marching, finite volume method (Oh, 1994). The Reynolds-Averaged Navier-Stokes equations are cast into a conservation form where the laminar viscosity coefficient follows the Sutherland's law, and the turbulent viscosity coefficient is simulated by the Baldwin & Lomax model (1978). An explicit four-stage Runge-Kutta time integration scheme with the second and the fourth-order artificial dissipation terms (Jameson, 1981) is used, and one-level multigrid acceleration technique (Dawes, 1988) with local time stepping is employed for better convergence. The gradients at the center and on the boundary of a cell are calculated using the Green method (Martelli, 1987) and the area projection method (Turner, 1993), respectively. For simplicity, a sheared H-grid is used so that the streamwise and quasi-orthogonal surfaces are surfaces of revolution. Conventional characteristic boundary conditions are applied to both inlet and exit boundaries and zero-mass flux through boundaries is applied to solid wall. At the upstream boundary, the assumed aerodynamic blockage factor of 3% is consistently applied for establishing the velocity profiles. The capability of calculating leakage flow through tip clearance is also included. With the assumed values of thickness ratio, the impeller tip modeling is simply made. A series of thickness ratio of 0.0, 0.7 and 10 are given at the corresponding grid at tip, the next grid towards the hub, and the second next grid towards the hub, respectively. No wall functions were used in the method.

**MODIFIED IMPELLER**

The main features of design changes are as follows.

(a) Hub and shroud contours are changed for moderate distributions of relative velocity in the flow channel, as shown in Fig.1.

(b) The maxima in blade angle distributions are shifted forward in order to secure maximum loading near mid-passage, as shown in Fig.2.

(c) The casing contour in the vaneless diffuser is changed in a smooth manner for 15% pinched flow channel at exit, so as to avoid a large defect of flow momentum, as shown in Fig.1.

In Fig.3 and Fig.4, the approximate inviscid calculation of blade loadings for the original impeller shows a strong rear-loading pattern due to maximum blade angle in the rear part of the impeller. The blade loading parameter Q in Fig.4 is defined as a ratio of relative velocity between suction and pressure sides to their average value. That loading pattern usually promotes non-uniformity at impeller exit flow field through the rapid growth of boundary layers and secondary flows. But, for the modified impeller, a decent mid-loading distribution is generated as intended. The maximum value of Q is also found to be much lower (about 0.8 instead of about 1.4), which means that more favorable flows should be produced under the same diffusion level required.

With the same distributions of blade normal thickness, the front views of both impellers are presented in Fig.5, where a noticeable rake angle (about 70 degree from tangential) is found in the modified impeller. It is intentionally generated for partly suppressing non-uniform impeller exit flow field through moderating the development of secondary flows. The rake angle is defined as an angle between hub disk and blade span at the impeller exit plane. Due to the changes of blade angles, the throat area of the modified impeller is increased by a small amount of about 4%.
RESULTS OF THREE DIMENSIONAL COMPRESSIBLE VISCOUS FLOW ANALYSIS

For the numerical grids of computational domains for both impellers, a set of (19 x 91 x 19) grid system in blade-to-blade, streamwise, and spanwise directions is used. The streamwise grid numbers of leading-edge and trailing-edge are 20 and 70, respectively. A maximum coarse grid system is selected, for saving required computation time, based on the grid refinement study in Table 1. Since the grid generation in the spanwise direction is made with a simple stretching function, computational tip clearances at the impeller inlet and exit are 2.322 mm and 0.755 mm respectively. The actual experimental clearances are, however, reported as about 0.500 mm in both positions. The exit radius of computational vaneless diffuser is 300 mm, while measurement position with which compressor performance was estimated is 337.4 mm.

The closer match of the magnitude of the clearance and the exit plane position in the calculation is quite necessary for higher order investigation of the internal flow field. But it requires much more grids and computing time. In the present study, the main concern is to find the changes of the modified impeller performance in a designer's broad view for engineering purposes. Even with the disagreement, computation proceeded.

Through a repetitive assumption of static pressure at exit boundary plane, calculated mass flow rate approaches the design value of 4.54 kg/s. Selected inlet mass flow rate is 4.703 kg/s for the original impeller, and 4.702 kg/s for the modified impeller. The standard convergence is taken to occur when the maximum value of the change in meridional velocity per iteration is less than 10E-4 times the RMS velocity. However, there have been difficulties in approaching the design mass flow rate more exactly, because the sensitivity of solution to the variation of exit static pressure was found to be very high. It seems to be partly due to the coarse grid, and due to the inherent characteristics of a centrifugal impeller. In that case, the convergency is accepted when the continuity balance is for all the quasiorthogonals within 1.5%. This criteria can be more than sufficient for most of engineering purposes. The rotational speed in the computation is fixed at the design value of 14,000 rpm.

Fig.6 compares the circumferentially mass-averaged static pressure distributions along the meridional direction near the casing with the time-mean test counterpart. Good agreement is found between the present calculation and the test data for the original impeller, which partly validates the accuracy of the present calculation method. For the modified impeller, a considerable reduction of the static pressure recovery near half of impeller flow length is restored due to the mid-loading design, and also a gradual increase of static pressure in the vaneless diffuser is found due to the favorable flow field produced by the pinched wall.

Fig.7 shows the relative velocity distributions at half-pitch position at some selected sections of the meridional flow channel (Eckardt, 1980), for which details are given in Table 2. Good agreement is also noted between the present calculation and the test data of the original impeller. For the modified impeller, a slight decrease of relative velocity, especially from mid-span to shroud, is found at Section III and Illa due to higher loadings, but it recovers to the level for the original impeller at Sections IV and V.

The relative velocity distributions on selected quasiorthogonal planes are shown in Fig.8, Fig.9 and Fig.10. Good agreement is found between the present calculation and the test data for the original impeller. The large amount of defect in the relative velocity near the casing and also near the suction surface at Section V for the original impeller represents typical jet/wake flow characteristics. But in the modified impeller, the amount of defect becomes remarkably small as expected, and finally at the exit of impeller more uniform distributions of relative velocity in the circumferential direction are obtained. In the spanwise direction, however, a gradient of relative velocity is generated over the most of the pitch at Section V for the modified impeller.

The enhanced uniformity in the circumferential direction at the exit plane of the modified impeller can be observed in the distributions of meridional velocity normalized with blade tip speed of Fig.11. The improved uniformity in the circumferential direction seems to be from the shifted changes of blade loadings and the generation of blade rake angle, which suppresses fluid migration from the secondary flows. The uniformity in the spanwise direction is, however, never improved, and moreover becomes worse. It seems to be from the prescribed conditions of geometry restriction, for example, the axial length of the impeller.

Fig.12 shows the distributions of calculated viscous blade loadings for both impellers which can be compared with the design intentions of Fig.3, based on the approximate inviscid calculations. The calculated viscous Mach number was obtained from the solution near each wall, not on the walls. The expected increase of blade loading near the middle of the impeller passage is clearly found near the hub. The distributions near the casing are not in the clearance space but very near blade tip. They are interpreted considering the development of secondary flow in the impeller passage like Fig.13 where the contours of meridional velocity normalized with blade tip speed are shown for selected quasiorthogonal planes. Because of
development of tip clearance flow, the low-velocity fluid area is formed near the casing and also near the suction surface.

For the original impeller, the center of low-momentum fluid near the casing moves from half-pitch to near the suction surface. This forms the typical jet/wake flow, and, therefore, a reversed pattern for which the relative Mach number near the suction surface falls below that near the pressure surface is produced near the exit in Fig. 12.

For the modified impeller, a larger low-momentum flow area is generated in the inducer, but its magnitude is maintained almost constant and then reduced near the exit. A reversed pattern is also found in Fig.12 near the casing, but the difference between the suction and pressure surface values is kept almost constant and then reduced near the exit. The exit shows that a more uniform flow field is generated in the circumferential direction at the impeller discharge. But the non-uniformity in the spanwise direction can be clearly found at the impeller exit, as seen before.

Fig. 14 shows the contours of rotary total pressure, at the exit plane of both impellers, which give a better understanding of secondary flow development. Low rotary total pressure region is concentrated near the suction and shroud walls for the original impeller. However, for the modified impeller, it is rather scattered near the shroud, and even more, the minimum region is moved towards the pressure side. The shift of secondary flow pattern seems to be due to the change of blade angle distributions and the generation of rake angle. The blade-to-blade curvatures are known to be strongly related with the creation of secondary flow driving the low rotary total pressure fluid towards suction side in conventional impellers. The non-uniformity in the spanwise direction seems to be strongly related with the meridional curvature, so that if it is allowed to change the axial length of the impeller as well as the meridional contours, an improved uniformity in the spanwise direction can be obtained.

Overall performances of the compressor with both impellers are shown in Fig. 15, where total-to-total pressure ratios obtained from the present three dimensional compressible viscous flow analysis are plotted with the experimental data (Eckardt, 1980). The calculated pressure ratios are obtained through mass-averaged total pressure at the exit plane. Both impellers give almost the same total-to-total pressure ratios, as expected, because the driving parameters such as diameter, blade exit angle, exit width and blade number remain unchanged. Generally the numerical performance is predicted higher than the experimental performance. Comparison of the compressor efficiency between numerical and experimental data is not possible, because the numerical data cannot include external losses. Both impellers give almost the same isentropic efficiency (95.5% in the original impeller, and 95.3% in the modified impeller) at the design point. An increase of choking flow rate is found in the modified impeller, as expected from the design calculations. The predicted surge (or stall) flow rates cannot be estimated realistically because there always exists a numerical breakdown in the region of nearly constant pressure for the performance characteristics due to the inherent problems of the exit boundary conditions.

CONCLUSIONS

The design of Eckardt backswept impeller was modified, expecting better aerodynamic performance, with focus on the internal flow field. Parts of the meridional contours and the blade angle distributions were changed. Through the three dimensional compressible Navier-Stokes analysis, the following performance features were found:

- A more uniform flow field in the circumferential direction at the impeller discharge is established due to the changes in blade angle distribution and also the effect of blade rake angle.
- A less uniform flow field in the spanwise direction at impeller discharge is established due to the restricted changes of the meridional contours.
- A favorable increase of static pressure in the vaneless diffuser is found due to the 15% pinched casing wall.
- The same levels of total-to-total pressure ratio and isentropic efficiency of the compressor are obtained, and an increase of choking flow rate is found due to the increased throat area.

REFERENCES


Pumps and Compressors, ASME, pp.77-96.


Table 1 Grid refinement study

<table>
<thead>
<tr>
<th>Grid</th>
<th>17x9x17</th>
<th>19x9x19</th>
<th>21x9x21</th>
<th>23x9x23</th>
<th>25x9x25</th>
</tr>
</thead>
<tbody>
<tr>
<td>E_p (%)</td>
<td>2.81</td>
<td>2.70</td>
<td>2.63</td>
<td>2.60</td>
<td>2.61</td>
</tr>
</tbody>
</table>

\[ E_p = \frac{P_{CAL} - P_{EXP}}{P_{EXP}} \times 100 \] : maximum error between test data and calculated data of static pressure near casing (Fig.6)

Table 2 Section data of measurement positions

<table>
<thead>
<tr>
<th>Section No.</th>
<th>Normalized Meridional Distance</th>
<th>Grid No.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Eckardt</td>
<td>Present</td>
</tr>
<tr>
<td>I</td>
<td>0.08</td>
<td>0.081</td>
</tr>
<tr>
<td>II</td>
<td>0.43</td>
<td>0.443</td>
</tr>
<tr>
<td>III</td>
<td>0.59</td>
<td>0.586</td>
</tr>
<tr>
<td>IIIa</td>
<td>0.73</td>
<td>0.720</td>
</tr>
<tr>
<td>IV</td>
<td>0.87</td>
<td>0.870</td>
</tr>
<tr>
<td>V</td>
<td>1.01</td>
<td>1.000</td>
</tr>
</tbody>
</table>

Fig.14 Rotary Total Pressure Contours at SECTION(V)

Fig.15 Compressor Performance Curves
Fig. 1 Meridional Contours of Impeller with Vaneless Diffuser Passage

Fig. 2 Blade Angle Distributions of Impeller

Fig. 3 Approximate Mach Number Distributions of Impellers

Fig. 4 Approximate Blade-to-Blade Loading Distributions of Impellers

Fig. 5 Front View of Impellers
Fig. 6  Mass-Averaged Static Pressure Distributions near Casing

Fig. 7  Relative Velocity Distributions at Half-Pitch Position
Fig. 8 Relative Velocity Distributions on Quasi-Orthogonal Planes of SECTION (IIIA)

Fig. 9 Relative Velocity Distributions on Quasi-Orthogonal Planes of SECTION (IV)

Fig. 10 Relative Velocity Distributions on Quasi-Orthogonal Planes of SECTION (V)
Fig. 11 Meridional Velocity Distributions at SECTION(V)

(a) Original Impeller - Test (Eckardt, 1980)
(b) Original Impeller - Calculation
(c) Modified Impeller - Calculation

Fig. 12 Calculated Blade Loading Distributions of Impellers
Fig. 13 Meridional Velocity Contours on Quasi-Orthogonal Planes