A Calculation Procedure for Three-Dimensional Turbulent Flow in a Centrifugal Impeller with any Blade Geometry

XI GUANG, WANG SHANGJIN, MIAO YONGMIAO
Dept. of Power Machinery Engineering
Xi’an Jiaotong University, China

Abstract

A method for calculating 3—D turbulent flow in a centrifugal impeller is developed by solving the incompressible, steady, time averaged N—S equation in general curvilinear coordinates. The K—e two—equation turbulence model is utilized to describe Reynolds stresses. A calculation scheme is proposed which divides a centrifugal compressor impeller into three calculation zones, i.e. the inlet zone, the channel zone, and the exit—vaneless diffuser zone. A simple and time—saving method for generating 3—D body—fitted coordinate system of a centrifugal impeller is established by means of combining an algebraic transformation with the solution of 2—D elliptic partial differential equations. This method is applied to calculate the turbulent flow in an industrial centrifugal compressor impeller. The tendency of the " jet—wake" formation and growth in the impeller can be clearly seen, and the secondary flow pattern calculated is similar to Eckardt’s measurements. The calculation results at the impeller exit are also in good agreement with the experimental results performed by the authors.

Nomenclature

$\overline{a}, \overline{b}$ = direction vectors of $x_1, y_1$ coordinates
$b_l$ = inlet width of the vaneless diffuser
$b_2, D_2, R_2$ = impeller exit width, diameter, radius
$\overline{C}_m$ = dimensionless meridional velocity ($C_m / U_2$)
$\varepsilon^k$ = permutation tensor
$p^*$ = reduced pressure ($p^* = p - \frac{\rho}{2} R^2 \Omega^2$)
$p_s$ = static pressure
$p.s., s.s.$ = pressure side, suction side
$R$ = radial distance to axis of impeller
$U'_i U'_k$ = fluctuating contravariant and covariant velocity components
$U_2$ = impeller tip speed
$U_i$ = physical velocity components of $i$ coordinate direction
$\Omega_i$ = covariant components of angular velocity of the rotor
$\Omega$ = angular velocity of the rotor
$(x,y,z)$ = cartesian coordinate system fixed to impeller
$(r,\phi,z)$ = cylindrical coordinate system fixed to impeller
$\Gamma_{m,s}$ = diffusion coefficients and source terms
$\Psi_{m}(x,r)$ = mean flow surface equation

Introduction

Flows in impellers of centrifugal compressors are among the most complex three—dimensional flow phenomena. They are strongly influenced by the passage curvature, viscous, rotation forces and many other factors. Presently, the analysis methods based on quasi—three—dimensional models must be supplemented by an extensive experimental data. Recently researchers are trying to develop analysis methods to predict the real flow field in centrifugal impellers. Moore(1981), Rahmatalla et al.(1984), Dawes(1987) and Hah et al.(1988) have published papers to calculate the 3—D turbulent flows in centrifugal impellers. They used the time—averaged Navier—Stokes equations and various turbulence models. These full 3—D turbulence analysis methods should reveal more detailed flow phenomena, such as jet—wake structure and secondary flows, than the presently used analysis methods, and can provide a more comprehensive reference for engineering designs.

The aim of this paper is to develop a simple and time—saving analysis method for 3—D turbulent flows in centrifugal impellers with any blade geometry, which can be easily used in engineering design process. For this purpose, a calculation scheme is proposed which divides the flow region in a centrifugal compressor impeller into three calculation zones, i.e., the inlet zone, the channel zone and the exit—vaneless diffuser.
zone (Fig. 1). A simple and time-saving method for generating a 3-D body-fitted coordinate system of a centrifugal impeller blade passage is established by means of combining an algebraic transformation with numerical solutions of 2-D elliptic partial differential equations. The K-\varepsilon two-equation turbulence model is used to estimate Reynolds stresses. The numerical scheme employed in this paper uses the contravariant components on a staggered grid (thus avoiding the "pressure checkerboard" problem) as the dependent variables of the governing equation.

An existing three-dimensional centrifugal compressor impeller that was designed by "all-over-controlled vortex method" (Miao et al., 1981) was chosen for assessing the validity of the present calculation procedure. Flows in the impeller were investigated in detail; the numerical results were compared with the experimental data.

Coordinate systems

The channel zone coordinate system \((s, \xi, \eta)\). A 3-D body-fitted coordinate system \((s, \xi, \eta)\) has been used in the channel zone calculation. The centre streamline of the \(S_0\) mean flow surface is chosen as the calculation line; the planes normal to the \(s\) line are chosen as the calculation surfaces. The \(s\) coordinate direction is chosen tangent to the calculation line \(s_c\) and \(\eta\) lines are body-fitted coordinate lines on the calculation planes (which are also the constant \(s\) coordinate surfaces). For deriving the relationships between \((s, \xi, \eta)\) and \((x, y, z)\) coordinates, a \((s, x_1, y_1)\) coordinate system was introduced, which is similar to the "semi-path" coordinate system (Fraser et al., 1982). The definition of \(s\) coordinate in the \((s, x_1, y_1)\) coordinate system was introduced, which is similar to the "semi-path" coordinate system (Fraser et al., 1982). The definition of \(s\) coordinate in the \((s, x_1, y_1)\) coordinate system was introduced, which is similar to the "semi-path" coordinate system (Fraser et al., 1982). The definition of \(s\) coordinate in the \((s, x_1, y_1)\) coordinate system was introduced, which is similar to the "semi-path" coordinate system (Fraser et al., 1982).

The steady incompressible time-averaged N-S equations, continuity equations and K-\varepsilon equations are solved for the current problem. Their generalized tensor forms are as follows:

\[ \nabla \cdot (\rho U) = 0 \quad (4) \]
\[ \rho \frac{D U}{D t} = -\nabla P + \nabla \cdot \tau \quad (5) \]

Governing Equations

The exit-vaneless diffuser zone coordinate system \((s, \phi, \theta)\). For the convenience of applying the periodic condition in the vaneless diffuser, cylindrical surfaces are chosen as calculation surfaces. A plane curve is chosen as the \(s\) calculation line. The components of \((s, \phi, \theta)\) and \((r, \phi, z)\) coordinates are related as follows:

\[ s = s^0(r) \]
\[ \phi = \phi^0(r, z) \]
\[ \theta = \theta^0(r, z) \]

where \(s^0(r), \phi^0(r, z)\) are circumferential boundaries of the calculation zone, and \(\epsilon_r(t), \epsilon_z(t)\) are axial boundaries. Using the above relations, it is not necessary to solve any partial differential equations to generate the \((s, \phi, \theta)\) coordinate system.

Laplace equations on a given calculation plane. Fig. 3 is the distribution of constant \(s\) and \(\eta\) coordinate lines on a typical calculation plane.

Using the divergence theorem to derive the finite difference equations, the above governing equations are generalized as

\[ \nabla \cdot (\hat{U} \Phi - \Gamma \nabla \Phi) = S \quad (\Phi = \frac{1}{2} U^T K U) \]

The finite difference equations have been derived by integrating the above equations over their control volumes (Xi, 1990). In the present cal-
calculation, the partially-parabolic assumption is used, i.e. the diffusion terms are neglected in bulk flow direction. The component equations needed for calculation in the \((s, \xi, \eta)\) and \((s, \phi, z)\) coordinate systems are obtained by substituting their metric tensors and Christoffel functions into the generalized tensor form equations.

**Calculation Details**

The present numerical solution scheme is similar to SIMPLE method (Patankar, 1980), but the contravariant components are chosen as the solution variables to avoid the checkerboard pattern of the pressure field in general non-orthogonal coordinates.

Before the viscous calculations of the channel zone and the exit-vaneless diffuser zone, the inviscid calculation was performed with a quasi-three-dimensional method. The applied shroud-to-hub solutions were obtained with a streamline curvature method (Wang et al., 1976), only one \(S_{\text{sh}}\) mean flow surface was calculated. Flows on the \(S_1\) revolution surfaces are calculated by the singularity method (Seno et al., 1971). The inviscid calculation results are taken as the final calculation results of the inlet zone.

For the channel zone viscous calculations, the inlet boundary of the channel zone (also the outlet boundary of the inlet zone) is located at a place where the calculation plane of the \((s, \xi, \eta)\) coordinate system can wholly intersect with the solid walls of two neighbour blades. The inlet boundary velocity profile is given by the inviscid calculations. An empirical relation similar to (Habib et al., 1982) was used to estimate the inlet turbulence kinetic energy and turbulence kinetic energy dissipation rate. In our calculation, \(K_{s1} = 0.004U_1^3\), \(\kappa = 3/2^\nu/2\) / (0.03D), where, \(D\) is the inlet hydraulic diameter of the channel. Velocity no-slip condition and wall function treatment (Reggio et al., 1987) are used at all wall boundaries. At the outlet boundary, only static pressure needs to be fixed. Considering numerous studies (Swanson, 1982; Prince et al., 1984) show that the measurements of static pressure in the impeller are close to the inviscid calculations especially near the optimum efficiency points, we use the inviscid calculation results to estimate the outlet static pressure for simplicity of engineering applications.

In the exit-vaneless diffuser zone, the inlet boundary is chosen as a cylindrical surface at \(R / R_2 = 0.8\), where it is overlapped with the outlet boundary of the channel zone. The inlet boundary conditions are given by the finished viscous calculation results of the channel zone. The outlet boundary is located as a cylindrical surface at \(R / R_2 = 1.15\), where the static pressure can be easily obtained from experiments. In the vaneless diffuser, the periodic condition is used at the circumferential boundaries, no-slip condition and wall function treatment are also used at the walls of diffuser. Two \((12 \times 15 \times 15)\) grids are used for the channel zone and the exit-vaneless diffuser zone viscous calculations. When the maximum relative error of flow rate at a calculation surface has been reduced to 0.005, the calculation is considered to be convergent. The calculation CPU time is about 60 hours on a Vax-II microcomputer.

**Results and Discussion**

A modern backswepth shrouded three-dimensional compressor impeller with high aerodynamic efficiency (Miao et al., 1981) is used to assess the 3-D viscous calculation procedure. The impeller has a 340mm tip diameter, 17mm exit width and 55 degrees blade exit angle referring to the tangential direction. The test stage consists of an impeller with a vaneless diffuser and a scroll. The vaneless diffuser has a constant axial width which is equal to the impeller exit width. The design stage pressure ratio is 1.62; its polytropic efficiency is 0.86. The design rotational speed is 16000rpm. In the experiment the speed is reduced to 7000rpm in order to be consistent with the incompressible assumption made by the present calculation. The operating conditions for tests and calculations are near the optimum efficiency point. The velocities and static pressure are measured with five-hole hemisphere and three-hole \("L"\) type conventional probes at various radii \(R / R_1 = 1.053, 1.21, 1.51\).

**Mainstream Velocity.** We use meridional velocities to describe the mainstream velocities in order to compare with the measurements (Eckardt, 1976). Fig.4(a1)(b),(c),(d)represent the continuous development of the calculated mainstream velocity from the inlet area to the impeller discharge. At plane I(Fig.4(a)), the velocity distribution is very regular; velocities near the suction side are clearly higher than that near the pressure side. Similar distributions appear in next plane II(Fig.4(b)), which is located in the middle part of the impeller channel. As the flow continues developing downstream, the trends of velocity distribution begin to change considerably. At plane III(Fig.4(c)), the velocity profile near the shroud side becomes flatter, a low velocity zone (normally called "wake") appears in the corner of shroud / suction side. As shown in Fig.4(d), the wake zone becomes clear at the shroud / suction side corner region on the discharge surface IV. From the above results, we observe that the calculated mainstream flow trends and the location of wake (low velocity zone) are qualitatively consistent with Eckardt’s measurements. However, it should be noticed that the wake zone and its defect value in our impeller are much less pronounced than that in the Eckardt’s impeller. These are probably due to the difference of our backswepth impeller with Eckardt’s radial impeller, and may also be related to the difference design methods used for the impellers.

**Secondary Flow.** Fig.5(a) shows the secondary flow distribution on calculation plane I. Four vortices are observed on the cross-section of the channel, the largest vortex is located at the corner region of the shroud / suction side. At plane II(Fig.5(b)), there still exist four vortices, but their locations are shifted, the size and strength of the pressure side vortex are increased considerably. The location of plane III(Fig.5(c)) is very close to the measurement plane IV in Eckardt’s tests. From Fig.5(c), we can see that the vortex near the pressure / hub side becomes very large, covering about 50 percent of the flow section, and that another noticeable vortex occurs at the corner region of shroud / suction side. These calculated secondary flow patterns are compatible with Eckardt’s measurements.

**Impeller exit circumferential average results.** Fig.7(a1),(b),(c) compare the calculated circumferentially mass—averaged velocities and static pressure distributions with the experimental data. As shown in Fig.7 (a1),(b), the calculated radial and circumferential velocities is in good agreement with the experimental data in most part of the width, a larger difference appears in the area adjacent to hub\((z / h_2 = 0.5-0.8)\). This is probably related to the defect of the used turbulence model. Fig.7(c) compares the
calculated static pressure (in the viscous calculation) with the experimental data, good agreement is noted.

Conclusions

The present method can successfully predict the real flows in a centrifugal impeller. The calculated secondary flow patterns are very similar to Eckardt's measurements. The calculated results in the impeller exit are also in good agreement with the test data. Compared to other analysis methods for 3D viscous flows in centrifugal impellers, the present method is simpler and time-saving.

The analysis for the modern high-efficiency backswept centrifugal impeller shows that, the jet/wake structure at the discharge is less pronounced than classical results, the velocity in the corner region of shroud/suction side does reduce continually from the inlet to discharge, i.e., the trend to form wake area does exist in the impeller.

References


Fig. 1 Divided zones for calculation of an impeller with vaneless diffuser
ABJK the inlet zone
BDHI the channel zone
CEFJ the exit vaneless diffuser zone

Fig. 2 Schematic cross section of the calculated and test impeller
I, II, III, IV are the location of the calculation surfaces

Fig. 3 Distribution of $\zeta$ and $\eta$ coordinate lines on a typical calculation plane

Fig. 4 Velocity distribution of the mainstream flow at various calculation surfaces
(a) at cal. surface I  (b) at cal. surface II
(c) at cal. surface III (d) at cal. surface IV
I, II, III, IV are defined in Fig. 2
Fig. 5 Secondary flow distribution at various calculation surfaces
(a) at cal. surface I
(b) at cal. surface II
(c) at cal. surface III
I, II, III are defined in Fig. 2.

Fig. 6 Basic secondary flow pattern in the radial part of the centrifugal impeller
(Eckardt's measurements [2])

Fig. 7 Distribution of the circumferential average velocities and pressure in the impeller exit ($R / R_2 = 1.053$)
(a) Radial velocity
(b) Circumferential velocity
(c) Static pressure