INVISCID-VISCOUS INTERACTION METHOD FOR 3D INVERSE DESIGN OF CENTRIFUGAL IMPELLERS

M. Zangeneh
Department of Mechanical Engineering
University College of London
London, United Kingdom

ABSTRACT
A 3D inverse design method for the design of the blade geometry of centrifugal compressor impellers is presented. In this method the blade shape is computed for a specified circulation distribution, normal (or tangential) thickness distribution and meridional geometry. As the blade shapes are computed by using an inviscid slip (or flow tangency) condition, the viscous effects are introduced indirectly by using a viscous/inviscid procedure. The 3D Navier-Stokes solver developed by Dawes is used as the viscous method. Two different approaches are described for incorporating the viscous effects into the inviscid design method. One method is based on the introduction of an aerodynamic blockage distribution throughout the meridional geometry. While in the other approach a vorticity term directly related to the entropy gradients in the machine is introduced.

The method is applied to redesign the blade geometry of Eckardt's 30° backswep impeller as well as a generic high pressure ratio (transonic) impeller. The results indicate that the entropy gradient approach can fairly accurately represent the viscous effects in the machine.

NOMENCLATURE

- $B$ : number of blades.
- $B_r$ : blockage factor.
- $C_p$ : specific heat at constant pressure.
- $e$ : unit vector.
- $\phi$ : blade wrap angle ($\theta$ value at the blade).
- $h$ : static enthalpy.
- $I$ : Rotary stagnation enthalpy or rothalpy.
- $r$ : radius.
- $(r,\theta,z) c$ cylindrical-polar co-ordinate system.
- $s$ : entropy.
- $S(\alpha)$ : sawtooth function.
- $T$ : Static temperature.
- $v$ : periodic velocity.
- $V$ : velocity.
- $W$ : relative velocity.
- $\alpha$ : angular co-ordinate of blade surfaces.
- $\delta(\alpha)$ : periodic delta function.
- $\rho$ : density.
- $\Phi$ : potential function.

INVISCID-VISCOUS INTERACTION METHOD FOR 3D INVERSE DESIGN OF CENTRIFUGAL IMPELLERS

M. Zangeneh
Department of Mechanical Engineering
University College of London
London, United Kingdom

ABSTRACT
A 3D inverse design method for the design of the blade geometry of centrifugal compressor impellers is presented. In this method the blade shape is computed for a specified circulation distribution, normal (or tangential) thickness distribution and meridional geometry. As the blade shapes are computed by using an inviscid slip (or flow tangency) condition, the viscous effects are introduced indirectly by using a viscous/inviscid procedure. The 3D Navier-Stokes solver developed by Dawes is used as the viscous method. Two different approaches are described for incorporating the viscous effects into the inviscid design method. One method is based on the introduction of an aerodynamic blockage distribution throughout the meridional geometry. While in the other approach a vorticity term directly related to the entropy gradients in the machine is introduced.

The method is applied to redesign the blade geometry of Eckardt's 30° backswep impeller as well as a generic high pressure ratio (transonic) impeller. The results indicate that the entropy gradient approach can fairly accurately represent the viscous effects in the machine.

NOMENCLATURE

- $B$ : number of blades.
- $B_r$ : blockage factor.
- $C_p$ : specific heat at constant pressure.
- $e$ : unit vector.
- $\phi$ : blade wrap angle ($\theta$ value at the blade).
- $h$ : static enthalpy.
- $I$ : Rotary stagnation enthalpy or rothalpy.
- $r$ : radius.
- $(r,\theta,z) c$ cylindrical-polar co-ordinate system.
- $s$ : entropy.
- $S(\alpha)$ : sawtooth function.
- $T$ : Static temperature.
- $v$ : periodic velocity.
- $V$ : velocity.
- $W$ : relative velocity.
- $\alpha$ : angular co-ordinate of blade surfaces.
- $\delta(\alpha)$ : periodic delta function.
- $\rho$ : density.
- $\Phi$ : potential function.

1. INTRODUCTION
In recent years considerable progress has been made in our understanding of the complicated flow patterns inside centrifugal compressor impellers. The results of experimental work by Eckardt(1976), (1980) and later Krain(1988) indicate that the flow inside the impeller is dominated by complicated secondary flow patterns and gross boundary layer separation, resulting in a circumferential (the so-called jet-wake phenomenon) as well as spanwise exit flow non-uniformity. The results of these findings have been gradually incorporated into the impeller design procedures used by most manufacturers. For example the widespread use of backswep impellers in all state-of-the-art designs and the careful design of shroud geometry has resulted in appreciable improvements in impeller performance and exit flow distribution, in particular for low specific speed machines (Krain, 1988).

However, in the case of high specific speed machines there is little scope in optimising the shroud curvature and therefore without careful design of the vane geometry, gross boundary layer separation and secondary flow effects will result in the formation of the non-uniform exit flow pattern with its detorous effect on performance and stability of the machine.

In this paper, a 3D impeller vane design method is presented, which should provide an efficient means of optimising the impeller geometry. The method is based on Hawthorne's (1984) approach to the design problem in which the blades are represented by sheets of vorticity whose strength is determined by a specified distribution of circumferentially averaged swirl velocity, or $r \tilde{\theta}$ (directly related to the bound circulation $2\pi\tilde{V}_b$ and the blade loading ) defined as:

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
Cincinnati, Ohio May 24-27, 1993
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1993

The original application of the method was confined to the
design of 2D cascades of infinitely thin blades in incompressible
flow. The method has since been extended to the 3D design of
incompressible axial machines by Tan et al (1984) and applied to the
case of arbitrary meridional geometry in incompressible flow by
arbitrary geometry method was later extended to compressible flow
by Zangeneh (1990), using an approximate and an exact approach. In
the approximate approach the pitchwise variation in density is
neglected and as a result the algorithm is simple and efficient, while
in the exact approach the density is computed throughout the 3D flow
field. It was found, however, that the error in blade shapes computed
by the two methods is well within the manufacturing tolerances. The
method was applied to the design of a high (subsonic) speed radial-
inflow turbine and appreciable improvements in efficiency over
conventional impellers of identical size and design point were
obtained.Zangeneh (1990). Recently, a compressible version of the
method based on the finite volume approach was reported by Dang

The method described in Zangeneh(1991) assumes an
irrotational flow through the impeller. This assumption is not very
limiting in the case of most radial-inflow turbines due to the favourable
pressure gradients and high Reynolds numbers. In the case of centrifugal compressors, however, the flow is dominated by
viscous effects, which result in an entropy rise and a reduction in the
flow turning in the impeller. An impeller designed on purely
irrotational flow basis may have a lower than expected pressure rise
and slip factor.

In the case of almost all inverse design methods it is difficult to
introduce the viscous effects directly into the equations of motions,
as these methods rely on the inviscid slip (or tangency) condition at
the blade surface. One way to introduce the
viscous effects into the design method is to use a viscous/inviscid
interaction approach. Due to the 3D nature of the flow, it is
important to model the 3D viscous effect and so it is possible to use,
in order of decreasing accuracy and increasing computational
efficiency a 3D full Navier-Stokes solver, or an underrelaxed NS
technique or a 3D boundary layer method. For this application an
approach based on a 3D NS method is described. Once the viscous
method has been selected the problem is in what form the viscous
information can be fed back into the design method.

In this paper we shall outline two approaches for the introduction
of the viscous effects into the design method. One based on the
introduction of an aerodynamic blockage distribution and the other
based on the introduction of a vorticity term directly related to the
torque gradients in the impeller.

2. DESCRIPTION OF INVERSE DESIGN METHOD

In the theory which will be presented the following assumptions
will be made:

(a) The flow is steady and homenergetic (adiabatic and
uniform at inlet) but not homentropic.

(b) The blades have zero thickness, so that they can be
represented by a single sheet of vorticity. However, the blade
blockage effects are accounted for by using a mean stream surface
thickness parameter in the continuity equation of the mean flow.

(c) The working fluid is a perfect gas and the maximum
meridional Mach number (based on the meridional velocity) is less
than unity throughout the flow. The latter condition is a result of the
fact that when the mean swirl velocity is specified the governing
equation of the flow field remains elliptic as long as the Mach number
based on the mean meridional velocity is less than unity (Wu, 1955).
Therefore, this method (which uses central differencing to solve the
flow equations) can be used for the design of high pressure ratio
machines where the relative Mach number is greater than unity as
long as the meridional Mach number remains below unity and it has
already been used to design impellers with maximum relative Mach
numbers of up to 1.3-1.4. The current method, however, has no
problem in which the bound circulation is specified. It can be shown
(Zangeneh, 1991) that the bound circulation is given by

\[ \Omega_b = \nabla \times V = \nabla \times (V r \Theta_0 ) \delta \rho (\alpha) . \]

where \[ \alpha = \theta - f(x,z) = n \frac{2 \pi}{B} . \]
represents the blade surfaces, $\theta$ is the tangential co-ordinate of a cylindrical polar co-ordinate system and $r(z)$ is the angular co-ordinate of the point on the thin blade surface, or the so called wrap angle. $\delta\phi(z)$ is the periodic delta function, whose pitchwise mean is unity, so that the mean vorticity is given by

$$\bar{\Omega} = \nabla \times \bar{\bar{V}} = (\nabla \times \bar{V}_\theta \times \nabla \phi) .$$

(8)

When entropy gradients are specified the overall mean vorticity is given by

$$\bar{\Omega} = \bar{\Omega}_h + \bar{\Omega}_t$$

(9)

where $\bar{\Omega}_h$ accounts for the vorticity due to entropy gradients in the flow, which are generated by viscous effects. The method used to calculate $\bar{\Omega}_t$ is discussed in detail in section (4).

Now by considering the tangential component of mean vorticity we can see that

$$\bar{\Omega}_h = \frac{\partial \bar{V}_r}{\partial z} - \frac{\partial \bar{V}_z}{\partial r} = \bar{\phi}_h \left( \bar{\Omega}_h + \bar{\Omega}_t \right) .$$

(10)

Substituting equations (4) and (8) into (10) the following equation for the unknown stream function can be derived;

$$\left[ \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} \right] \bar{\phi}_h + \frac{1}{r} \frac{\partial^2 \bar{\phi}_h}{\partial \theta^2} = 0$$

(11)

where the first term on the RHS is zero outside blade row and the second term is zero when the aerodynamic blockage is specified. This elliptic equation is solved subject to boundary conditions at the end walls and upstream and downstream boundaries. At the end walls (hub & shroud), the no flow condition is satisfied by using the Dirichlet condition $\Psi = $ constant. The far upstream boundary condition is obtained from the specified far upstream velocity and a parallel efflux condition is used at far downstream.

The periodic flow field is determined by using the Clebsch formulation for the periodic velocity (Zangeneh, 1991), which is given as

$$v(r,\theta,z) = \nabla \phi \cdot (r,\theta,z) - S(\alpha) \nabla \bar{V}_\theta ,$$

(12)

where $S(\alpha)$ is the periodic sawtooth function and $\phi$ is the potential function of the periodic flow. In the absence of circumferential variations in density (approximate approach), the periodic component of the continuity equation can be written as

$$\nabla . v = - v . \nabla \ln \rho .$$

(13)

Taking the divergence of (12) and using the continuity equation (13) we obtain;

$$\nabla^2 \phi = S(\alpha) \nabla^2 \bar{V}_\theta + (\nabla \alpha \cdot \nabla \bar{V}_\theta) S(\alpha) - v \cdot \nabla \ln \rho .$$

(14)

where the first two terms on the RHS will be zero outside of the blade row. Using the periodicity of the periodic flow it is possible to express $\phi$ in terms of a Fourier series in the tangential direction. Furthermore, the RHS of equation (14) (ie $S(\alpha)$) and $S(\alpha)$ can also be expressed in terms of a Fourier series. As a result, the 3D Poisson equation in (14) is reduced to a 2D Helmholtz equation, which is then solved for each harmonic of the potential function. The use of a Fourier expansion in the tangential direction results in substantial savings in computational time and memory as the 3D flow field is solved without the need for a grid in the tangential direction, which would also require regridding after each blade iteration.

The Helmholtz equation is solved subject to the flow tangency condition at the endwalls and zero potential condition at the far upstream and far downstream boundaries.

2.2 Calculation of Density

For a homenergetic irreversible flow, the energy equation in relative frame can be reduced to a constant entropy condition, Denton (1986). This form of energy equation is widely used in 3D Navier-Stokes calculations, e.g. Dawes (1988). Therefore the energy equation is given by;

$$1 = h + \frac{1}{2} (W, W) - \frac{1}{2} \omega^2 \bar{r}^2 = \text{constant}$$

(15)

where $\omega$ is the rotational speed and $W$ is the relative velocity. By using the perfect gas relations it is possible to show that:

$$\rho \frac{\partial^2}{\partial t^2} = \frac{1}{2} \frac{\omega^2 - \omega^2 \bar{r}^2}{C_p T_i}$$

(16)

where $s$ is the entropy and subscript $i$ refers to a reference value. In the case of the approximate approach, the latest values of mean velocity, calculated from equation (4), are used in equation (16) to obtain a new estimate for the mean density. The exponential of the entropy term in equation (16) is set to unity when the aerodynamic blockage distribution is specified.

2.3 Calculation of Blade Shape

Once the flow field is determined, then it is possible to compute the blade shape by using the blade boundary condition that the blade must be aligned to the velocity vector there. This condition can be expressed as:

$$W_{BH} \cdot \nabla \phi = 0 .$$

(17)

where $V\phi$ is a vector normal to the blade surface and $W_{BH}$ is the relative velocity at the blade surface ($W_{BH} = W^+ + W^-/2$, where $W^+$ and $W^-$ are the velocities on the upper and lower surface of the blades). Equation (17) is a first order hyperbolic partial differential equation, which has to be integrated along the meridional projections of streamlines on the blade surface in order to find the blade shape. The integration, as in the case of other initial value problems, can not be completed without some initial condition on $f$. This initial value will be called the stacking condition of the blade. In this method the stacking condition is implemented by giving as input, the values of blade wrap angle $f$ along a quasi-orthogonal, for example at the leading edge. The effect of different stacking conditions on the flow field is discussed in detail in Zangeneh (1992).

2.4 Extension to Non-Free Vortex Blade Design

In some design conditions, for example for very high pressure ratio compressors, it may be necessary to specify a spanwise non-uniform loading distribution in order to optimise the performance. This type of loading distribution generates shed vorticity at the trailing edge of the blade so that the flow downstream of the blade row becomes rotational. This rotational flow is a Beltrami flow as the shed vortex lines lie along the streamlines and of course can not support static pressure changes across them (Tan, 1981).

The enthalpy change across the blade is given by (Zangeneh, 1991):

$$h^+ - h^- = \frac{2\pi}{B} (W_{BH} \cdot \nabla \bar{V}_\theta)$$

so that the Beltrami flow downstream of the blade satisfies the following condition:

$$W_{BH} \cdot \nabla \bar{V}_\theta = 0$$

(18)

where $W_{BH}$ is the mean of the velocities on the upper and lower part of the trailing vortex sheet. Equation (18) can be solved to find the $\bar{V}_\theta$ distribution downstream of the blade row.

Now if we let $\alpha$ to represent the blade shape inside the blade region and the trailing vortex sheet downstream of the blade, we can represent the trailing vorticity by equation (5). Therefore, the flow downstream of the blade can be calculated from equations (11) and (14). Noting that in this case the term in bracket on the RHS of equation (11) and the first two terms on the RHS of (14) do not vanish downstream of the blade row. Furthermore, when solving
splitter blades is quite straightforward. For this purpose we let the distribution on the full blades be \(2f_rV_e^1\) and that on the splitter blades be \(2f_rV_e^2\) to represent the full and splitter blades respectively. If the circulation takes on the form of equation (17).

2.5 Design of Impellers with Splitter Blades

In some high pressure ratio applications splitter blades are used to reduce the blade loading. In conventional designs, it is common to use splitter blades with the same blade profile as the full blade located at mid-pitch. This choice of splitter blades is usually based on simplicity of design and manufacture rather than aerodynamic considerations. This inverse design method provides an ideal technique for optimising the splitter blade shapes as well as the full blades.

The extension of the method to the design of impellers with splitter blades is quite straightforward. For this purpose we let 
\[ \alpha_1 = \theta - f(r,z) \quad \text{and} \quad \alpha_2 = \theta - f_2(r,z) \]
to represent the full and splitter blades respectively. If the circulation distribution on the full blades is \(2\pi V_e V_\theta^1\) and that on the splitter blades is \(2\pi V_e V_\theta^2\), then the bound vorticity on the full and splitter blades are given by
\[ \Omega_1 = \left( \nabla \times \mathbf{V}_e \right) \delta_p(\alpha_1) \]
and
\[ \Omega_2 = \left( \nabla \times \mathbf{V}_e \right) \delta_p(\alpha_2) \]
so the overall bound vorticity is given by
\[ \Omega_b = \left( \nabla \times \mathbf{V}_e \right) \delta_p(\alpha_1) + \left( \nabla \times \mathbf{V}_e \right) \delta_p(\alpha_2) \]
and therefore the RHS of equations (11) and (14) are modified accordingly. The full and splitter blade shapes \(\alpha_1\) and \(\alpha_2\) are then calculated by applying the appropriate inviscid slip condition, equation (17).

2.6 Numerical Algorithm

The partial differential equations for the computation of the flow field, i.e. equations (11) and (14) and the blade shape, equation (17), are solved numerically by using a finite difference approximation. An algebraic transformation to a boundary fitted computational plane was used so that in the new curvilinear coordinate system the coordinate lines are coincident with the boundaries. The form of the differential equations in the computational plane are presented in Zangeneh (1991).

The governing equations of the flow field in the computational plane, equations corresponding to (11) and (14), are discretised by using a second order accurate finite difference formula. The boundary conditions are then applied and the resulting simultaneous algebraic equations are solved iteratively, using Brandt(1977) cycle C multigrid strategy to increase the rate of convergence. More details on the discretisation procedure and the form of the discretised equations can be found in Zangeneh (1991). The blade boundary condition, equation (18), is discretised by using the Crank-Nicholson scheme.

3. VISCOUS METHOD

For the viscous method the Navier-Stokes solver developed by Dawes (1988) was used. In this method the unsteady Reynolds’ averaged Navier-Stokes equations are discretised using a cell-centred finite volume formulation. The discretised equations are time marched using an implicit “pre-processed” algorithm. In this version of the program a simple algebraic turbulence model patterned after Baldwin and Lomax (1978) is used. A two level multi-grid is used to accelerate the rate of convergence to steady state.

At the inflow boundary, the total pressure, total temperature, the swirl and pitch angles are fixed and the derivative of static pressure in the streamwise direction is set to zero. At the outflow boundary, the static pressure is held constant and other variables are extrapolated from the interior. At the solid surfaces zero normal fluxes of mass, momentum and energy are imposed. In satisfying the wall boundary conditions, the casing wall is assumed to be stationary (as in unshrouded impellers) while the hub (including the region upstream and downstream of the blade row) rotates with the blade. A wall function is used in order to compute the wall shear stresses when the mesh is too coarse to resolve down to laminar sublayer scale.

In order to establish the accuracy of the method, the flow through Eckardt’s (1980) backswept impeller was computed by using a \(17\times73\times17\) computational grid. The predicted exit meridional velocity (non-dimensionalised by the tip speed) is compared to Eckardt’s measurements in Figure 1. There is a good qualitative correlation between the predicted and measured velocity distributions. A more thorough evaluation of this viscous method for analysis of flow through radial and mixed flow turbomachines can be found in Goto (1990) and Casey et al (1991). In terms of overall performance parameters Goto’s (1990) study indicates that the method can predict the Euler head (or the exit tangential velocity at exit from the impeller) quite accurately. However, both Goto and Casey et al show that the code fails to accurately predict the actual head (or pressure rise) in the impeller as it underestimates the amount of mixing losses in the impeller and the diffuser.

![Figure 1 Comparison of Viscous Calculation with Eckardt Measurements](image)

(b) Viscous Predictions

Figure 1 Comparison of Viscous Calculation with Eckardt Measurements

4. VISCOUS-INVISCID INTERACTION

In this section, we shall describe two different approaches for the introduction of the viscous effects into the design method. Both approaches introduce the effect of viscosity into the mean flow. One is based on the introduction of an aerodynamic blockage distribution which is computed from the viscous results. While in the second approach, the generation of vorticity due to the entropy gradients in the flow is modelled directly by introducing an extra vorticity term related to the entropy gradients.
4.1 Aerodynamic Blockage Approach

In external flow over aerfoils, it is usually possible to distinguish between the non-uniform, high loss, low momentum region inside the boundary layer and the uniform, low loss, high momentum region outside. Under these circumstances, the aerodynamic blockage can be based on the displacement thickness of the boundary layer. However, in the case of most compressors the non-uniform flow region can sometimes occupy the whole flow passage and there are therefore difficulties in defining the boundary layer thickness. Hence an alternative definition for the aerodynamic blockage is required. One definition which does provide a good measure of the extent of non-uniformity in the flow is the ratio of the area (pitchwise) averaged meridional velocity to the mass-averaged meridional velocity, i.e.

\[ B_{\text{aero}} = \frac{V_m \text{area}}{V_m \text{mass}} \]  \hspace{1cm} (19)

where

\[ \frac{\theta_s}{\theta_p} = \frac{2 \pi \int p V_m r d\theta}{2 \pi \int p V_m r d\theta} \]

and

\[ \frac{\theta_s}{\theta_p} = \frac{2 \pi \int p V_m r d\theta}{2 \pi \int p V_m r d\theta} \]

This definition of aerodynamic blockage is very similar to that suggested by Dring (1984) for processing of experimental measurements. For this application however, equation (19) is used to process the result of the 3D viscous flow calculation from which the aerodynamic blockage is calculated throughout the meridional geometry of the computational domain.

4.2 Vorticity Generation and Transport Approach

In section 2.1, it was mentioned that the effect of entropy gradients in the flow are to be represented by the mean vorticity component \( \Omega_t \). In this section a method will be presented for relating \( \Omega_t \) to the entropy gradients in the streamwise direction.

If has been shown by Hawthorne (1974) that the Crocco's equation of motion can be expressed as

\[ W \times \Omega = V \cdot f - \tau \cdot V s - F_b - F_t \]  \hspace{1cm} (20)

where \( \Omega \) is the absolute vorticity vector (which is the sum of the bound vorticity and the vorticity due to the entropy gradients), \( f \) is the body force distribution, \( F_b \) is the blade body force distribution. In order to obtain a consistent set of equations when entropy gradients are present, Horlock (1971) has shown that it is necessary to introduce a dissipative body force distribution \( F_t \) which acts in the streamwise direction, but opposes the motion. Now by substituting equations (8) and (9) into (20) and equating terms associated with the blade body force and dissipative force (see Dang and Wang (1992) and Roddis and Zangeneh (1993)) it is possible to show that:

\[ F_b = (W, \nabla r \nabla \theta ) \nabla \alpha - (W, \omega \alpha ) \nabla \alpha \theta \]  \hspace{1cm} (21)

and

\[ W \times \Omega_t = \nabla \cdot \tau \cdot V s - F_t \]  \hspace{1cm} (22)

Taking curl of both sides of equation (22) we find:

\[ W (\nabla \cdot \Omega_t) - \Omega_t \nabla \cdot W + (\Omega_t \cdot \nabla ) W - (W, \nabla ) \Omega_t = -\nabla \times (\nabla \circ V s) - \nabla \times F_t \]  \hspace{1cm} (23)

Using the fact that vorticity vector is solenoidal together with vector identity for the first term on the RHS it is possible to derive the following expression for \( \Omega_t \):

\[ (W, \nabla ) \Omega_t = \Omega_t \nabla \cdot W - (\Omega_t \cdot \nabla ) W + (\nabla \circ \nabla \circ V s) + \nabla \times F_t \]  \hspace{1cm} (24)

Now by considering the tangential component of (24) and noting that the radial component of \( \Omega_t \) is \( \frac{1}{r} \frac{\partial \mathbf{V} \theta}{\partial r} \) so that:

\[ \Omega_{tr} = \varepsilon_r \left( \frac{\Omega}{r} - \Omega_t \right) = 0 \]

and similarly the axial component of \( \Omega_t \) is \( \frac{1}{r} \frac{\partial \mathbf{V} \theta}{\partial z} \) so that:

\[ \Omega_{tz} = \varepsilon_z \left( \frac{\Omega}{r} - \Omega_t \right) = 0 \]

we can derive the following PDE for \( \Omega_t \):

\[ \nabla \frac{\partial \Omega_t}{\partial z} + \frac{\partial \Omega_t}{\partial r} + \frac{\partial \Omega_t}{\partial z} \frac{\partial \Omega_t}{\partial r} = \left( \frac{\partial \Omega_t}{\partial r} + \frac{\partial \Omega_t}{\partial z} \right) \frac{\partial \Omega_t}{\partial r} \]

\[ \frac{\partial \Omega_t}{\partial z} = \frac{\partial \Omega_t}{\partial r} \]

\[ \frac{\partial \Omega_t}{\partial r} = \frac{\partial \Omega_t}{\partial z} \]

\[ \frac{\partial \Omega_t}{\partial r} = \frac{\partial \Omega_t}{\partial z} \]

This equation governs the transport of vorticity and its generation due to entropy gradients in the flow. For given values of mean velocity, temperature and entropy it is possible to integrate equation (25) for \( \Omega_t \) once the values of the dissipative body force components \( F_{tr} \) and \( F_{tz} \) are known.

To calculate the dissipative body forces, following Horlock (1971), let us consider the streamwise component of equation (22), whereby:

\[ W (W \cdot \Omega_t) = W, \nabla \cdot \tau \cdot W, \nabla s - W, F_t \]

As mentioned in section (2.2), if the flow is adiabatic and uniform at inlet (i.e. homenergetic) then, inspite of the dissipative effects, the energy equation can be reduced to a constant rothalpy condition. As a result the first term on the RHS of equation (26) vanishes and so we are left with the following equation for \( F_t \):

\[ W (W \cdot \Omega_t) = W, \nabla \cdot \tau \cdot W, \nabla s - W, F_t \]

or

\[ F_{tr} = -\nabla \cdot \left( \frac{\Omega_t}{r} \frac{\partial s}{\partial r} + \frac{\Omega_t}{r} \frac{\partial s}{\partial z} \right) \]

Now since \( F_{tr} \) acts in the direction of \( W \) then \( W \times F_t = 0 \) so that the components of \( F_t \) can be found from:

\[ F_{tr} = -\frac{\Omega_t}{r} \frac{\partial s}{\partial r} \]

\[ F_{tz} = -\frac{\Omega_t}{r} \frac{\partial s}{\partial z} \]

\[ F_{tr} = -\frac{\Omega_t}{r} \frac{\partial s}{\partial r} \]

\[ F_{tz} = -\frac{\Omega_t}{r} \frac{\partial s}{\partial z} \]

The form of equation (26) in the transformed plane is presented in the Appendix. This equation is then solved by using the Crank-Nicholson method. The initial value of \( \Omega_t \) at the upstream boundary is set to zero, as the flow is irrotational at inlet.

5 RESULTS

In the first application of the method we shall use it to redesign the blade geometry of Eckardt's (1980) backswep centrifugal impeller. This impeller's blade geometry may not be representative of...
some of the more advanced current designs. But it does provide a well understood impeller of relatively high specific speed. The design was based on identical design point, i.e. a rotational speed of 14000 rpm, mass flow rate of 4.54 kg/s, blade number of 20 and backsweep angle of 30° from radial.

The meridional geometry and the tangential thickness distribution used for the design were also identical to that of the Eckardt’s impeller. A (73x25) meridional grid was used for the design. The grid points were clustered near the endwalls in order to resolve the severe entropy gradients, see Figure 6(b).

To design for comparable overall loading the exit tangential velocity was set to 203.5 m/s. The specified distribution of \( \dot{r}V_\theta \) was determined by specifying the loading distribution shown in Figure 2 on the hub and shroud and then interpolating linearly to find the \( \dot{r}V_\theta \) distribution throughout the meridional domain. For this particular design, the blade was stacked at the trailing edge plane. In this case no attempt was made to control secondary flows in the impeller. However, the method has already been successfully applied to control secondary flows in radial turbomachinery the result of which will be reported in near future. More information on the choice of \( \dot{r}V_\theta \) distribution and the stacking condition is given in Zangeneh (1992).

\[ \text{Figure 2 Specified Loading Distribution} \]

These input specifications were then used to compute the blade geometry by using the Actuator Duct (axisymmetric) and three dimensional inviscid versions of the design method (Runs C0AD and C0 respectively). For the three dimensional design 15 harmonics of the Fourier series, representing the periodic flow, were used. The corresponding CPU times for the Actuator Duct and 3D runs were 30 seconds and 53 minutes on a VAX station 4000 VLC, respectively. The mean blade angles (or wrap angles) computed by the two methods are compared in Figure 3. The difference between the wrap angles computed by the 3D and axisymmetric methods become very significant in regions of high loading near the trailing edge. The resulting Mach number distribution on the shroud streamline of the 3D designed blade is presented in Figure 4. Also shown in Figure 4 are the Mach number distributions for the original Eckardt’s impeller. These Mach number distributions were computed by Denton’s (1983) inviscid 3D Euler solver code using a (17x69x17) grid. Figure 4 shows that the specified loading distribution results in a smooth Mach number distribution particularly on the shroud suction surface, where little diffusion can be observed.

In order to validate the inviscid version of the design method, the flow through the designed impeller was computed by using Denton’s 3D Euler code, using a (17x73x17) grid. The resulting \( \dot{r}V_\theta \) distribution on the shroud predicted by the Denton code is compared with the specified distribution in Figure 5. This excellent agreement between the predicted and specified \( \dot{r}V_\theta \) distribution could be observed along all the streamlines.

The flow through the blade geometries calculated by the Actuator Duct (Run COAD) and 3D method (Run CO) were then computed by using the 3D viscous method of Dawes (1988) using a (17x73x17) computational grid. In each case, the static pressure at outlet, which was located at \( R/R_2 = 1.3 \), was varied until the design mass flow rate was obtained. In these computations the tip clearance effects between the impeller tip and the casing was not considered. To achieve convergence for a typical run about 2500 time steps were required. A typical calculation requiring about 11 hours of CPU time on VAX station 4000VLC. The overall results of these two viscous calculations are tabulated in Table 1. The predicted losses were 4.2% and 4.3% respectively for geometries COAD and C0. These losses compare favourably with a predicted loss of 5.5% by the same viscous method for the original Eckardt impeller. The main difference between the geometries designed by the axisymmetric and the 3D method is in the values of exit tangential velocity. The blade geometry computed by the actuator duct method results in an exit tangential velocity lower than the design value (196 m/s as opposed to 203.5 m/s). As this is a design method (in which the mean tangential velocity rather than the blade geometry is specified) the lower actual tangential velocity implies that the flow field does not fully follow the blade shape computed by the Actuator Duct method. We can therefore conclude that there is a certain amount of slip (using this term in its widest sense) in the impeller geometry computed by the AD method. The flow through the 3D designed geometry, however, shows little slip as the exit tangential velocity is 205.3 m/s; very slightly higher than the design value.

\[ \text{Figure 3 Contours of Mean Camber Angle - } f \]

\[ \text{Figure 4 Comparison of Mach Number Distribution} \]
The results of this viscous calculation for the 3D geometry (Run C0) was then used to compute the aerodynamic blockage distribution (using equation (19)) and the mass-weighted tangentially averaged entropy distribution in the impeller as shown in Figure 6. The results in Figure 6(a) indicate that the aerodynamic blockage is negligible everywhere apart from a relatively small region near and on the shroud starting from the meridional bend. This picture is further confirmed by the distribution of mass averaged entropy, which shows an accumulation of low momentum, high loss fluid in this region. The main mechanism for the accumulation of low momentum fluid in this region can be explained by considering the velocity vector plots on the pressure and suction surfaces. The entropy vectors on the suction surface are presented in Fig. 7. The velocity vectors on the pressure surface are almost identical to that shown in Fig. 7 and therefore are not presented. This plot indicates that the flow is fully attached, but there is a strong secondary flow moving low momentum fluids towards the shroud, thereby resulting in the accumulation of low momentum fluids in this region.

The aerodynamic blockage distribution and the entropy gradients, shown in Fig. 6 were then used in the corresponding versions of the design program to compute the blade shape. The CPU time for the blockage method is similar to that required for the inviscid method. However, the introduction of the vorticity term due to entropy gradients, does slow down the convergence rate and more blade iterations are required to reach convergence. In this case 2 hours and 10 minutes of CPU time on VAX 4100 was required.

In Figure 8, the mean camber angles (or wrap angles) computed by the aerodynamic blockage and entropy gradient methods are compared with the inviscid results. The wrap angles computed by the two former methods are similar at the hub, but differ considerably near the tip. The meridional velocity at the trailing edge as predicted by the different versions of the design method is compared with the mass-averaged velocity computed by the 3D viscous method in Fig. 9. The circles represent the exit meridional velocity as predicted by the Dawes code for the impeller C0. As the entropy gradients and aerodynamic blockage distribution obtained from run C0 were specified to compute the geometries C1-Entropy and C1-Blockage, we can conclude that the entropy gradient method can predict the exit flow pattern with a very good degree of accuracy. While the aerodynamic blockage approach results in a fairly uniform increase in meridional velocity across the span.

In order to establish whether the flow field and the blade shapes computed by these methods are affected by extra viscous/inviscid iterations, the flow through impellers designed by the entropy gradient method (C1-Entropy) and blockage method (C1-Blockage) were computed by using the viscous method and the whole process was repeated. The effect of extra viscous/inviscid iterations on the flow field and blade shape was found to be relatively small. For example in Figure 9(b) the exit meridional velocity for the entropy gradient designs C2-Entropy and C1-Entropy are compared, where only a small difference between the exit velocity can be observed. As expected, the extra iteration has resulted in a better correlation between the Dawes prediction of the exit meridional velocity and that predicted by the design method. A summary of the results of the iterations is presented in Table 1. A solid modelling of the geometry of impeller C2-Entropy is presented in Figure 10. All the data presented in table 1, were obtained from the 3D viscous method. The results indicate that in this low pressure application, the incorporation of viscous effects does not seem to make a very significant contribution to the design of the blade geometry, and they may perhaps be neglected.

However, the flow through this low pressure impeller was fully attached, as velocity vector plots in Figure 7 indicates. Therefore, in order to investigate the case where significant viscous effects, including gross boundary layer separation, are present a generic high pressure ratio transonic compressor was designed. This compressor had exactly the same meridional geometry and tangential thickness distribution as the original Eckardt impeller, however, the rotational speed was increased to 23000 rpm and all velocity triangles were scaled by the ratio of blade tip speed. The design mass flow rate through the impeller was now 7.156 kg/s, the Mach number based
on tip speed and inlet stagnation temperature was 1.42 and the stagnation pressure ratio of 4.4. The same leading distribution as the low speed case was specified and the value of \( V_g \) at exit was set to 332 m/s.

The above inputs were then used in the inviscid versions of the design method to compute the geometry. The maximum relative Mach number on the shroud suction surface was predicted to be 1.15 by the inviscid method. As before, the flow through this geometry was computed by the 3D NS method. The computed aerodynamic blockage and mass-averaged entropy distribution are presented in Figure 11. The overall distributions are similar to the low speed case. However, in this case in a region on the suction side of the shroud the blockage factor is as low as 0.2, i.e. only 20% of flow passage is available to the flow. Furthermore, the maximum entropy is now 150 J/K in this region. The Mach number contours predicted by the 3D Navier-Stokes method, shown in Figure 12, indicate that there is a strong shock system near the leading edge, which causes a gross boundary layer separation, which is the main cause of the dramatic increase in losses. Summary of the results of this calculation is presented in Table 2 (Run HS0), where we can see that the losses in the impeller alone amount to almost 19%. Furthermore, the exit tangential velocity predicted by the viscous method is about 13% less than the designed value (288 m/s vs 332 m/s). This is particularly important because in the low speed case, where the flow was fully attached, the predicted \( V_g \) was almost the same as the specified value indicating that very little slip was present. In this case, however, the gross boundary separation has resulted in a significant amount of slip in the impeller.

The blockage distribution and entropy gradients described above were then used in the corresponding versions of the design program. The wrap angle distributions computed by each method are presented in Figures 13 (a) and (b). Also shown in these figures are the wrap angles distribution computed by the inviscid version of the design method. The corresponding exit meridional velocities are compared with 3D Navier-Stokes results in Figure 14. The flow through these geometries were then computed by the 3D NS method, the overall results of which are summarised in Table 2. Despite the differences in geometry both method seem to be to able to account for the viscous losses, so that the actual (or predicted) exit swirl velocity is now very close to the specified values. An interesting feature of the results for impeller HS1-Entropy is the reduction in loss from 19.1%
to 14 %. This reduction in loss can be contributed to the changes in the geometry near the tip which have reduced the shock strength and therefore the resulting shock losses. However, the viscous results still indicate that a shock induced boundary layer separation is present near the inlet to the impeller. From practical point of view, it is of course possible to eliminate the shock wave by appropriate adjustments to the blade loading.

Figure 13 Comparison of Mean Camber Angle - f

Figure 11 Contours of Aerodynamic Blockage and Entropy

(b) Mass Averaged Entropy

Figure 12 Predicted Mach Number Distribution Near the Shroud

Table (1) Summary of the Results of Viscous/Inviscid Interactions.

<table>
<thead>
<tr>
<th>Run</th>
<th>m (kg/s)</th>
<th>$V_\theta$ (m/s)</th>
<th>Loss (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C0 AD</td>
<td>4.57</td>
<td>196</td>
<td>4.2</td>
</tr>
<tr>
<td>C0</td>
<td>4.59</td>
<td>206.3</td>
<td>4.3</td>
</tr>
<tr>
<td>C1 - Entropy</td>
<td>4.5</td>
<td>209.2</td>
<td>4.7</td>
</tr>
<tr>
<td>C2 - Entropy</td>
<td>4.51</td>
<td>210.8</td>
<td>4.8</td>
</tr>
<tr>
<td>C1 - Blockage</td>
<td>4.61</td>
<td>210.8</td>
<td>4.8</td>
</tr>
<tr>
<td>C2 - Blockage</td>
<td>4.49</td>
<td>214.3</td>
<td>4.7</td>
</tr>
<tr>
<td>C3 - Blockage</td>
<td>4.53</td>
<td>210.1</td>
<td>4.9</td>
</tr>
</tbody>
</table>

Table (2) Summary of Results of Viscous/Inviscid Interactions.

<table>
<thead>
<tr>
<th>Run</th>
<th>m (kg/s)</th>
<th>$V_\theta$ (m/s)</th>
<th>Loss (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>HS0</td>
<td>7.183</td>
<td>288</td>
<td>19.1</td>
</tr>
<tr>
<td>HS1 - Entropy</td>
<td>7.10</td>
<td>326</td>
<td>14.8</td>
</tr>
<tr>
<td>HS2 - Entropy</td>
<td>7.22</td>
<td>329.7</td>
<td>16.1</td>
</tr>
<tr>
<td>HS1 - Blockage</td>
<td>7.16</td>
<td>321.8</td>
<td>19.4</td>
</tr>
<tr>
<td>HS2 - Blockage</td>
<td>7.12</td>
<td>320</td>
<td>18.9</td>
</tr>
</tbody>
</table>

Figure 14 Comparison of Exit Meridional Velocity
6. CONCLUSION

A method for the 3D design of centrifugal compressors has been developed, in which the viscous effects are accounted for by using an inviscid/viscous procedure. The viscous effects are incorporated by either specifying an aerodynamic blockage distribution or the entropy gradients in the machine. The results indicate that the entropy gradient approach can fairly accurately represent the viscous effects in the machine, including the effect of accumulation of low momentum fluids by secondary flows and loss generation by shock waves.

The application of the method to a low pressure ratio compressor, indicates that by specifying the loading distribution it is possible to design impellers with controlled diffusion, especially on the shroud suction surface. This in turn has resulted in an impeller which is free from flow separation and the results indicate that in this case inclusion of the viscous effects does not result in any obvious improvement in overall parameters such as efficiency and pressure rise. An interesting feature of the results is that the inviscidly designed impeller in fact shows very little slip, i.e. the actual exit swirl velocity is very close to that specified.

In the case of a high pressure ratio (transonic) compressor, however, the exit tangential velocity for the impeller designed by the 3D inviscid method was found to be 13% lower than the specified value. This was because the strong shockwave/boundary layer interaction resulted in a gross boundary layer separation. By including the viscous losses, it was possible to account for these effects and thereby design an impeller with an exit tangential velocity very close to that specified.

In conclusion this inverse design method not only enables one to optimise the Mach number distributions in the impeller but it can also be used to design impellers with very high slip factors (approaching unity) and thereby provide compact designs with higher performance. Furthermore, the method can also be used to control secondary flows in the impeller. One such successful application of this method for controlling secondary flows will be reported in the near future.

ACKNOWLEDGEMENTS

The author is indebted to Professor J. D. Denton and Dr W. N. Dawes of the Whittle Laboratory at Cambridge University, who made their codes available for the present study and Professor W. R. Hawthorne for his comments on section 2.4. The help of Mr M Roddis in obtaining a solid model of the designed impeller is gratefully acknowledged.

REFERENCES


APPENDIX

Equation (25) in the transformed (ξ, η) curvilinear computational plane takes the following form:

\[ \psi_\eta T_\eta - \psi_x T_x + \nabla' \left[ \psi_R R - \psi_x T_x \right] \]

\[ + \left(\psi_x R - \psi_x T_x^2 - \psi_x^2 \right) + \frac{1}{r} \left( \psi_T \frac{\partial \psi_T}{\partial \tau} \right) \]

\[ = \left( \frac{\rho}{\rho} \right) \left( \psi_T \right) - \psi_x \left( F_x \right)_x + \psi_y \left( F_y \right)_x + \psi_x \left( F_x \right)_x - \psi_x \left( F_y \right)_x \]

where R = ln(p/\rho) and \( F_x \) and \( F_y \) are the radial and axial components of the dissipative body force.