A RAPID AERODYNAMIC LOADING PROCEDURE FOR CENTRIFUGAL IMPELLER DESIGN

J. H. G. Howard
Department of Mechanical Engineering
University of Waterloo
Waterloo, Ontario, Canada

Colin Osborne and David Japikse
Concepts ETI Incorporated
Norwich, Vermont

ABSTRACT

A crucial aspect of the design process for centrifugal impellers is the establishment of specific blade shapes. A rapid inviscid flow analysis procedure was developed for incorporation within a geometry manipulation code. Using a single streamtube model, a single-pass computation technique was generated. A two-zone model ensures that key features of the passage flow physics are incorporated.

Several examples of industrial design problems are employed to demonstrate the capabilities of the rapid loading method and its use in a geometry design procedure (used by some 20 industrial design groups worldwide). Comparisons with a quasi-three-dimensional method are included. The rapid loading method is most accurate when the meridional stream paths have similar shapes to those for the hub and shroud contours. The technique is useful within a geometry generation program since rapid aerodynamic screening of candidate configurations is allowed with sufficient accuracy to avoid the need for quasi-three-dimensional approaches. If required, the final design may be analyzed using three-dimensional viscous flow calculation methods.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S$</td>
<td>entropy</td>
</tr>
<tr>
<td>$t_n$</td>
<td>blade normal thickness</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$W$</td>
<td>relative velocity</td>
</tr>
<tr>
<td>$x_1, x_2, x_3$</td>
<td>distances in the directions 1,2,3 (Fig. 1)</td>
</tr>
<tr>
<td>$\beta$</td>
<td>angle of the relative velocity from the meridional</td>
</tr>
<tr>
<td>$\theta$</td>
<td>coordinate in tangential direction</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\phi$</td>
<td>angle between the meridional velocity and the axis (Fig. 1)</td>
</tr>
<tr>
<td>$\omega$</td>
<td>angular velocity of rotation</td>
</tr>
<tr>
<td>hub,shroud</td>
<td>mean values along hub or shroud surface</td>
</tr>
<tr>
<td>suction</td>
<td>value on blade suction surface</td>
</tr>
<tr>
<td>pressure</td>
<td>value on blade pressure surface</td>
</tr>
</tbody>
</table>

INTRODUCTION

The traditional design process for a centrifugal impeller consists of two steps. In the first, the overall geometry is chosen, employing extensive empirical data incorporated in a meanline or one-dimensional technique. The second step of the process seeks to establish the specific blade shape and the hub and shroud contours. It is necessary to ensure that these are optimum, within the constraints of the overall geometry already chosen. Some form of analysis of the impeller passage flow is needed in this procedure, in order to verify the optimum. It has become traditional to employ a quasi-three-dimensional inviscid flow analysis for this purpose. An example of such an analysis is the
Howard and Osborne (1977) technique, which employed a two-zone model (then called jet-wake) as part of a streamline curvature analysis method. The two-zone model, or equivalent, is needed to ensure that the principal features of impeller flow physics are included.

The drawback of this approach is that, on the one hand, the accuracy of the analysis does not match that of a fully three-dimensional viscous flow analysis, even when combined with some form of boundary layer modelling, while on the other hand, the procedure is sufficiently cumbersome to slow down the process of geometry change followed by flow analysis, and thus reduce the number and complexity of geometry changes evaluated.

One obvious solution was to introduce an approximate flow solver within the program controlling the geometry changes. This was tried by Holbrook and Brasz (1984), where the purpose was to provide an intermediate step in the analysis process and reduce the number of required cases which had to be evaluated by a quasi-three-dimensional procedure. The velocities predicted by their method did not correspond closely to the quasi-three-dimensional results so that the objective was only partially achieved.

The motivation for the approach described here is to develop a rapid inviscid passage flow analysis procedure which can be incorporated within a geometry manipulation (design) code, and also be sufficiently accurate to replace the need for a full quasi-three-dimensional analysis as part of the geometry optimization procedure. It is important to ensure that any technique must include the effects of the actual flow physics, mainly the presence of the secondary flow region which models the viscous flow characteristics. This is true both for the traditional quasi-three-dimensional methods as well as the simpler, rapid loading technique to be described. The former role of the boundary layer plus inviscid flow analysis as a major confirmatory tool is now rapidly being replaced by the fully three dimensional, viscous flow analysis. The complexity and extended computer run times for the latter, however, mean that they are not useful in the role of general optimization of impeller shape. The motivation for the rapid loading technique is really to enable advanced designs to be produced in short order. This requires not only easy to use and fast geometry generation programs, but also aerodynamic techniques that allow rapid screening of candidate configurations. The desire is to produce good designs simply from the application of the rapid loading technique. Also, for those cases where truly advanced designs are required, the rapid loading technique should allow screening of the candidate configurations prior to more comprehensive three-dimensional viscous flow evaluations.

AERODYNAMIC LOADING MODEL

The purpose is to achieve a simplified form of quasi-three-dimensional inviscid flow analysis in the impeller, with the inclusion of a two-zone flow model to represent approximately viscous effects.

Traditional approaches employ analysis across several streamtubes between hub and shroud. However, available data and the form of geometry established in the course of the design process suggest the suitability of a single streamtube model. Along the mean stream sheet, the hub-to-shroud velocity gradients are established and assumed constant over the single streamtube at a chosen calculation station (or quasi-orthogonal). Knowledge, or assumption, of flow direction enables mean velocity to be derived from continuity, and the velocity gradient prediction leads to hub and shroud velocities. At the shroud and the hub, the prediction of blade-to-blade velocity gradients allows relative velocity distributions on the suction and pressure surfaces to be produced. Loading coefficients arise naturally as part of the velocity gradient calculations.

The assumption of a single streamtube should be most appropriate in those geometries where internal streamline curvatures are well represented by the curvatures of the hub and shroud surfaces. This includes low aspect ratio centrifugal impellers with smoothly distributed axial to radial bends. The method is expected to be less accurate where significant streamline curvature occurs without associated and similar hub and shroud curvatures.

Aerodynamic Loading Theory

To derive velocity gradients, the vector form of the Euler equation for compressible, inviscid flow, which is steady with reference to a rotating reference frame, is written as:

\[-\frac{\nabla p}{\rho} + \frac{\nabla (\omega^2/2)}{2} = \nabla \cdot \nabla \cdot \vec{W} + 2\vec{\omega} \times \vec{W} \] (1)

where \( W \) is the relative velocity.

From the thermodynamic relationship

\[(1/\rho)\nabla p = \nabla h - T \nabla S \]

together with the identity:

\[\vec{W} \cdot \nabla \vec{W} = \nabla (W^2/2) - \nabla \cdot (\nabla \times \vec{W})\]

and, since rothalpy is

\[ H = h + (\omega^2/2) - \omega^2 r^2/2 \]

Eq. (1) becomes:

\[\nabla H = \nabla \cdot (\nabla \times \vec{W}) + \nabla \times (2\omega) \] (2)

To carry out the required analysis, Eq. (1) or (2) must be applied along the mean stream sheet, that is, along the meanline of the passage, except where the two-zone model dictates a different mean streamline position. To do this, a curvilinear coordinate system is established as shown in Fig. 1. The \( x_i \) direction lies along the relative streamline, the \( x_2 \) direction is formed by the intersection of the plane normal to \( x_1 \) with a meridional plane, and \( x_3 \) is the third orthogonal direction. The relationship between these directions in terms of the blade angle, \( \beta \), and the meridional plane flow angle from axial, \( \phi \), has been derived by Fraser, et al. (1983). If, for the present case, assuming that no flow component exists normal to the \( x_1 \) direction, and making allowance for the differences in \( \beta \) definition and
velocity gradients in the passage arise directly from Eq. (4). Assuming linear velocity gradients, Eq. (4) becomes:

$$\frac{W_{\text{shroud}} - W_{\text{hub}}}{W} = C_2$$

$$(W_{\text{axial}} - W_{\text{pressure}})$$

along $x_1$ direction

When the rothalpy and/or the entropy has a gradient from hub to shroud at the entry, the relative velocity gradient from hub to shroud must be suitably modified based on the constancy of $H$ and entropy along hub and shroud relative streamlines. It should be noted that the blade-to-blade loading, $C_b$, is in the direction normal to the passage and thus is not the same as the tangential loading parameter $\Delta W/W$ frequently employed (see Fig. 2).

The Rapid Loading method uses Eq. (5) and the definitions of $C_b$ and $C_t$, together with continuity, to determine the velocity field, as described in Calculation Procedure.

Two Zone Model

The flow through the impeller passage is considered to be comprised of two zones. The primary zone flow is isentropic, while the non-isentropic secondary zone is located toward the suction side of compressor or pump impeller blade passages. This secondary zone location is not intrinsic to the two-zone model but is adopted to assist in the calculation of velocities on the blade surfaces. The flow analysis is carried out only in the primary zone. The exit blockage associated with the secondary zone, the exit flow angle in the primary zone and the exit mass flow fraction in the secondary zone are determined as described by Japikse (1985).

Calculation Procedure

To complete the aerodynamic analysis, the following procedure

numbering of the directions by Fraser, then Eq. (1) becomes, using equations from Fraser, et al. (1983):

$$\nabla p - \frac{v(\omega^2r^2)}{2} = \frac{i_1 \frac{\partial}{\partial x_1} (W^2/2)}{r} + \frac{i_2 C_1 W^2}{\Delta x_2} + \frac{i_3 C_1 W^2}{\Delta x_3}$$

$$C_2 = \Delta x_2 \left( \frac{\cos^2 \beta}{r_m} - \frac{\sin^2 \beta \cos \phi}{r} - \frac{2 \omega \sin \beta \cos \phi}{W} \right)$$

and:

$$C_3 = \Delta x_3 \left( \frac{\sin \phi \sin \beta}{r} + \frac{\partial \beta}{\partial x_1} + \frac{2 \omega \sin \phi}{W} \right)$$

where:

- $r_m = \cos \beta dx/d\phi$ is the streamline radius of curvature in the meridional plane
- $\phi$ is defined in Fig. 1.
- $\Delta x_2$ is the passage height (normal to the passage streamline), and
- $\Delta x_3$ is the passage width (normal to the passage streamline)

Similarly, Eq. (2) becomes:

$$\nabla H_r - TVS = i_1 \left[ \frac{\partial}{\partial x_2} (W^2/2) - \frac{W^2 C_2}{\Delta x_2} \right] + i_2 \left[ \frac{\partial}{\partial x_3} (W^2/2) - \frac{W^2 C_3}{\Delta x_3} \right]$$

When the impeller entry flow has constant total head and a free vortex swirl distribution, and when isentropic flow conditions are presumed in the impeller, $\nabla H_r - TVS = 0$, and the
The shroud and hub contours and the blade angle and normal thickness distributions are described by Bernstein-Bezier polynomials. Unlike a similar method by Casey (1983), a single segment of adjustable order is employed to specify the hub and shroud geometry, while the blade angle (β) and normal thickness (t) are related to meridional distance along the hub and shroud surfaces. The single segment definition assists in easy adjustment of the curves and distributions. The mean streamline at the hub or shroud surface is assumed to follow both the slope φ and the blade angle β, adjusted to account for the secondary zone and/or entry incidence effects.

The mean streamline location is positioned at the root-mean-square radius at each quasi-orthogonal. Linear distributions between hub and shroud are assumed for (tanβ/h), (1/r_n), and tan(φ - angle between x and radial). The mean meridional component of velocity is calculated from the continuity equation at the RMS radius. The area employed is the annulus area generated by sweeping a quasi-orthogonal about the axis. The blockage due to the blades and the growing secondary zone is subtracted and the area reduced to account for the difference in the slope of the quasi-orthogonal and the normal to the slope φ at the mean streamline. The mass flow is that in the primary zone and this is reduced from the entry value with growth of the secondary zone. For compressible flow, this continuity determination may lead to a choke prediction. If so, the calculations proceed at a reduced mass flow.

The mean relative velocity, W, is calculated using the flow direction on the mean streamline, β. At the entry plane, an alternative method is used as described in (g). The shroud and hub relative velocities are determined by first calculating C_2 on the mean streamline, using Eq. (3), then using Eq. (5). A correction for non-zero VH is employed when a gradient of H is present at the entry.

The blade-to-blade velocity gradient is generated at the shroud and at the hub by evaluating C_2 on each surface (Eq. 3). The velocities determined by Eq. (5) are located opposite each other on a line which is normal to the mean passage direction. This is in contrast to the Stanitz and Prian (1951) method which has a similar degree of approximation, but where the blade-to-blade velocity gradient is in the tangential direction. The geometry of the two approaches are illustrated in Figure 2.

The meridional position, m, at which the values of W_{ambient} and W_{stream} occur, is calculated by tracing the normals to the passage centerline from there to their intersections with the suction and pressure surfaces of the passage (see Fig. 2).

The flow direction, β, is based upon the passage direction. However, the two-zone model requires that a secondary zone builds up along the suction surface, modifying the effective mean angle, β. The conditions at the impeller exit are set by values previously calculated by the preliminary design method, where the parameters for the two-zone method have been established by empirical data. The commencement and distribution of the secondary zone may be selected, but a cubic distribution is generally employed.

At the blade leading edge, the flow angle is not known. Instead, the angular momentum is given as input and effective blade blockage is accounted for. Thus equation (3) can be re-expressed in terms of the absolute tangential velocity, in order to establish C_2. To establish the velocity, flow angle and incidence distribution from hub to tip at the plane of the leading edge, the single step approach is bypassed and the calculation is carried out over 8 steps from the mean location to the shroud and 8 steps from the mean to the hub. The flow direction at the entry is smoothly merged with that determined under (f), over a selectable number of quasi-orthogonals.

The impeller designer is usually interested in the surface velocity and Mach number distributions as well as blade-to-blade and hub-to-shroud loadings. Various designers may be expected to have different criteria in terms of the amount and position of diffusion along the hub and shroud contours as well as to the level and distribution of various loadings. The values of C_2 on the meanline and the value of C_3 on the shroud can be used directly to judge the probability of severe flow separation in a passage. It has been suggested elsewhere that C_3 should not exceed 0.7 to 0.8, based on the use of the two-zone model. Where a single zone model is employed, alternative levels of C_3 may have been developed to yield good impeller designs. The rate of deceleration in W has been observed to be influential. If W is not decreasing along the shroud suction surface, C_3 values from 1.4 to 2 have been observed not to lead to suction surface separation in a very low specific speed pump impeller (see Abramian and Howard, 1993, and Howard, et al., 1987).

In the design iteration process, the hub and shroud meridional contours and the blade angle distributions (and to a lesser extent the blade thickness distributions) are manipulated to meet desired loading levels and their respective locations. First, the meridional contours are developed (a) to provide close control of the curvature distribution including the value and location of peak curvatures and (b) to yield a passage width distribution leading to the desired diffusion schedules (as well as acceptable values for the hub-to-shroud loading C_2). Second, the blade angle distributions are manipulated to meet desired loading levels (particularly the blade-to-blade value C_3, and to a lesser extent C_2), while simultaneous iterations are required with the meridional contours to maintain appropriate diffusion schedules.

The entire procedure takes approximately one-half a second on a 486 computer and is included within a design program called CCAD.
COMPARISON WITH QUASI-3-DIMENSIONAL FLOW ANALYSIS

The aim is to use a Rapid Loading procedure to provide quick aerodynamic screening of different candidate geometries in a geometry generation program prior to using more advanced calculation methods. The technique should be sufficiently accurate so that there is no need to perform a quasi-three-dimensional analysis but that, if needed, calculations of a three-dimensional viscous nature can be carried out. The technique must be justified by comparison with an established procedure. The method described by Howard and Osborne (1977) was chosen because it employs the same two zone analysis. It is a streamline curvature method in the hub to shroud plane, while in the blade-to-blade direction, a linearized analysis (Stanitz and Prian, 1951) is employed.

Examination of the approach suggests that certain impeller geometries lend themselves to more accurate analysis by this procedure. The flow model assumes that the streamline curvature in the meridional plane is determined solely by the local curvatures in the hub and shroud profiles. Thus, ideally, the impeller should have a low aspect ratio and the wall profiles near entry and exit should have curvatures representative of upstream and downstream wall curvatures, respectively. An impeller with a sharp bend in the shroud profile would be expected to lead to lower accuracy in the rapid loading calculations.

Four compressor impeller examples are examined here. They are all highly competitive commercial products or product prototypes. One example is a two-dimensional radial impeller, while another includes splitter blades.

Example 1

The meridional cross-section and axial view of the impeller for this air compressor are portrayed in Figure 3. The impeller has 10 main blades and 10 splitter blades, with exit blade angle of 40 degrees from meridional. The stage design total-to-total pressure ratio is 4.6, while the entry tip mean relative Mach number is predicted to be 1.16 (1.17 by the Quasi-3-D method).

The comparisons between predicted relative Mach number derived from the quasi-3-D and the present method are presented in Figure 4 for the pressure and suction surfaces along the hub and shroud. The comparison is seen to be very good except for some erratic predictions in the region of the splitter blade leading edge in both methods and in the predictions at the leading edge of the main blades. The differences in that region arise mostly from the blade-to-blade predictions and how they are distributed along the blades (see Fig. 2). The distribution of the blade-to-blade loading coefficient (C_l) for the stream surface which is mean between hub and shroud is shown in Figure 5. In the regions near the leading and trailing edges, neither method is highly accurate in the blade-to-blade direction. In addition, the Rapid Loading method seems to slightly underpredict the hub to shroud loading in a region at and just downstream of the highest shroud curvature.
Example 2

The impeller chosen for the second example has greater back-sweep at the exit (47 degrees), with no splitters and a lower entry Mach No., predicted at 0.85 for the shroud mean (0.88 by the Quasi-3-D method). The geometry is presented in Figure 6, in meridional cross-section and axial view. This impeller has 18 blades and a stage design total-to-total pressure ratio of 3.66. Figure 7 presents a comparison of the predictions of pressure and suction relative Mach numbers along the hub and shroud. The comparisons between the Rapid Loading technique and the Quasi-3-D method are very good, probably due to the smooth distribution of curvatures and flow angles.

Example 3

An example is presented here of an impeller operating near choke conditions with a high inlet swirl with free vortex distribution. The impeller is small and rotates at nearly 140,000 rpm, with a stage design total-to-total pressure ratio of 4.03. There are no splitter blades and the exit blade angle is 32 degrees. Geometry is portrayed in Figure 8 and the velocity distribution in Figure 9. Modest discrepancies are observable between the Rapid Loading and the Quasi-3-D methods in the entry region, which is believed to be associated with rapid changes of flow angle and the subsequent meridional positioning of the wall velocities. Discrepancies close to the impeller exit are not as yet explained.
The reverse loading is clearly apparent at entry hub and shroud although, as seen in Figure 10, the Quasi-3-D analysis does not predict negative loading on the mean stream surface. The strong distribution of flow conditions associated with the swirl result in some prediction difficulties for the Rapid Loading method near the leading edge, but the method outlined in paragraph (g) above results in a reasonably accurate distribution of flow angle and incidence, the latter shown in Figure 11. Incidence is defined as the difference between the blade angle and the flow angle including blade blockage effects.

**Example 4**

This is a two-dimensional impeller with a bladeless bend from axial to radial preceding it. The geometry, shown in Figure 12, features an impeller with 20 blades, an exit angle of 45°, a stage pressure ratio of 1.51 and an entry Mach number of 0.6 (both methods). The velocity distribution (Figure 13) for the impeller shows the expected almost identical conditions for the hub and shroud velocities and a significant entry loading, associated with incidences of 5° at the hub and 6° at the shroud. The blade to blade loading (Cₚ) for this impeller varies from 0.25 at the entry to a maximum of 0.32 at about 56 percent of the meridional distance and reduces to a value of 0.14 at the exit. The velocity
distribution of the non-swirling flow through the bend upstream of the impeller is seen in Figure 14 and reveals a shortcoming of the Rapid Loading method. At the bend outlet, the Quasi-3-D technique, with the aid of two downstream quasi-orthogonals to improve the accuracy of the curvature determination, predicts a slightly different hub to shroud velocity distribution. Such differences may be significant in the design of the important impeller entry region, and the effect is expected to be increasingly strong where the bend from axial to radial has higher curvature.

**CONCLUSIONS**

A rapid inviscid passage flow analysis procedure has been developed for incorporation within a geometry manipulation (design) code. Through the use of a single streamtube model and by defining the streamtube slopes and curvatures in relation to the hub and shroud contour, a single-pass computation was made possible. A two-zone model ensures that key features of the passage flow physics are included. The rapid loading technique is useful in that it can be incorporated into a geometry generation program that will allow rapid aerodynamic screening of candidate configurations.

The rapid loading method is most accurate when the meridional stream paths have a similar shape to that of the hub and shroud contours. At impeller entry, the presence of strong positive or negative loading is best monitored by direct observation of the loading parameters generated by the method, and by noting local incidences. Special care may be necessary when strong upstream swirl is present. The demonstrated accuracy of the technique is sufficiently good so that advanced designs can be generated without the need to go to some of the more complex techniques,
including the traditional quasi-three-dimensional approaches sometimes employed. However, for really advanced machines, the designs that are generated with the rapid loading technique can be taken directly into the latest three-dimensional viscous flow calculation methods without the need for quasi-three-dimensional calculations.

REFERENCES


