The American Society of Mechanical Engineers

Reprinted From
PD–Vol. 64-8.2, Design: Analysis, Synthesis, and Applications
Editors: I. I. Esat, A. Ertas, and D. Tilley
Book No. H912H2 – 1994

GLOBAL FLUID DYNAMICS DESIGN OF CENTRIFUGAL PUMPS

Carlo N. Grimaldi
Istituto di Energetica
Università di Perugia
Perugia, Italy

Giampaolo Manfrida
Dipartimento di Energetica (DEF)
Università di Firenze
Firenze, Italy

Roberto Pacciani
Istituto di Energetica
Università di Perugia
Perugia, Italy

ABSTRACT

The main features of a recently developed method for the computer-aided fluid-dynamic design of centrifugal pumps are presented and discussed. The method collects several design approaches, from the widely diffused design charts in terms of non-dimensional coefficients, to analytical models or empirical correlations.

Starting from basic expected performance data for the machine (flowrate, head, rotational speed), the method automatically produces a three-dimensional impeller geometry, with direct interface to CAD-CAM design workstations and to a 2-D finite element inviscid flow solver.

The fluid dynamics CAD process has been extended to the design of the volute. The local volute cross section is computed in a way that ensures circumferentially constant pressure, with reference to standard volute shapes. The procedure also allows to determine the local values of two velocity components (meridional+tangential) and pressure.

NOMENCLATURE

A	area, m^2
AR	area ratio
b	width, m
u	absolute velocity, m/s
f	friction coefficient
H	head, m
m	meridional coordinate
N	rotational speed, rpm
N_s	specific speed $N_s = N\sqrt{Q}/H^{3/4}$, rpm $(m^3/s)^{1/2} m^{-3/4}$
p	pressure, Pa
Q	volume flowrate, m^3/s
r	radial coordinate, m
Z	number of blades
z	axial coordinate, m
α	absolute flow angle, °
β'	impeller blade angle, °
δ	meridional angle of S1 stream surface (from radial direction),°
ε	correction factor for non radial volute $\varepsilon=1/\sin\gamma$
$\Delta\vartheta$	angular distance between consecutive volute sections $\Delta\vartheta=\vartheta_j-\vartheta_{j-1}$
ϕ	friction index
ϑ	tangential coordinate, °
ρ	density, kg/m^3
τ	shear stress, Pa

<u>Subscripts</u>

i	volute inlet
j	index of volute's cross sections
l	local
lim	limiting
m	meridional or midflow streamline
max	maximum
n	hub
r	radial
t	shroud
ϑ	tangential

INTRODUCTION

Nowadays the industrial design of centrifugal pumps is still generally carried out by recurring to conventional methods. These methods are in general limited to the 1-D approach, based on semi-empirical correlations, and need wide experimental data collections. Notwithstanding the apparently roughness of these methods, the obtainable results are usually satisfactorily, in terms of capability of performance prediction (flowrate, head, efficiency).

At the same time, the ever higher computing performance offered by modern machines allows to develop sophisticated numerical methods and to implement codes for the simulation of turbomachinery fluid dynamics.

In the last years an evolution in fluid dynamic analysis has occurred, ranging from codes based on potential or stream function formulations, to more general methods capable to solve viscous 3-D flows (e.g. Arnone and Stecco, 1991). Nevertheless, this powerful scientific development has not yet produced consistent effects on the industrial design. This difficulty in the diffusion of a fluid dynamic culture in industry can be charged to different causes, among which a non secondary one is the cost of machines, codes and qualified human resources. In general, it is not easy to match a specific industrial design experience with the capability to prepare data and run fluid dynamic codes, and to examine and to use the results. In fact, non common ability and specific experience are requested in order to extract useful informations from 2-D or 3-D analysis and to transfer them in effective design solutions.

It seems thus very important that researchers increase their efforts in diffusing a wider fluid dynamic knowledge and culture among turbomachinery industrial designers. To this end an effective aid can be represented by the development of a design tool which could integrate conventional approaches with modern fluid dynamics methods, allowing the designer to gain confidence with advanced techniques.

OUTLINE OF THE DESIGN METHOD

A first step in the development of a global interactive design tool for centrifugal pumps (PCAD) was presented by Grimaldi and Manfrida (1992).

PCAD allows to calculate the pump fundamental quantities (NPSH, efficiency, number of vanes, impeller inlet and outlet sections diameters and blade height, etc.) by selecting among several possible design choices based on conventional one-dimensional methods: Stepanoff (1957), Anderson (1980), Salisbury (1983), Peck (1968), Lobanoff and Ross (1986), Gongwer (1941), Pearsall (1973, 1978), Worster (1963).

In order to connect 1-D design with 2-D or 3-D analysis the generation of the three-dimensional geometry of the impeller is needed. Suitable algorithms -developed in a previous work (Grimaldi and Manfrida, 1992) for the determination of the midflow S1 (with reference to Wu, 1957) streamsurface coordinates-have been extended to the generation of any number of surfaces, corresponding to different sections of the meridional channel. This code's section represents an automatic interfacing procedure to fluid dynamics codes, which in this work was used to generate the input file for a 2-D finite element inviscid flow solver on the blade-to-blade (S1) surface.

The impeller geometry data can also be transmitted, as a draft impeller configuration, to CAD-CAM workstations. From these, the design geometry can be also transmitted back to PCAD for the interfacing to fluid dynamic codes.

The fluid dynamic CAD process has also been extended to the design of the volute. For this component the main design requirement concerns the minimization of circumferential pressure distortion at nominal operating conditions. In the present approach, based on a direct integration of conservation equations, uniform flow is considered in the tangential direction and the radial variation of the flow rate delivered by the generic section, together with friction effects, are taken into account.

The local volute cross section is thus computed, with reference to standard volute shapes and assigned orientation direction, which can range from radial to axial. The design process also allows the determination of the local values of the meridional and tangential components of the velocity, while the meridional component of the momentum equation is used to establish the meridional pressure profile in the volute section.

GENERATION OF COMPLETE 3-D IMPELLER GEOMETRY

In the industrial design of centrifugal pumps, the impeller shape is usually determined by means of graphical methods. The analytical approach presented by Grimaldi and Manfrida (1992) was extended in this work, regarding the definition of both the flow channel shape on the meridional surface and the impeller curvature distribution on the blade-to-blade surface.

With reference to the first aspect, several standard shapes of the meridional flow channel are examined: as a function of N_s, the midflow streamlines are described by the nondimensional parameter $z(m)/r_2$ and by the meridional angle $\delta(m)$, defined as the angle between the vertical axis (r) and the tangent to the streamline (Figure 1). Data files of these "standard shapes" are then used for each new design to calculate the r-z coordinates of the midflow streamline, from the geometrical data calculated by one-dimensional design at the inlet and outlet impeller sections. The selection of a representation of $\delta = f(z/r_2)$ has proven to be very general, and has the advantage of leaving substantial freedom in the choice of the impeller eye diameter D_{1t} by anyone of the selected approaches.

After the midflow line coordinates are determined, the local cross-section area can be specified at the design stage by a simple analytical law:

$$(1) \qquad A(m) = A_1 + (A_{\lim} - A_1) \cdot \left[\frac{m - m_1}{m_{\lim} - m_1} \right]^{n_b}$$

where the choice of n_b allows the designer to select proper area distributions along the meridional coordinate m. The section lim is the first of the purely radial shape of the impeller, from there on, the shape of the meridional flow channel is determined by the hub/shroud taper (Figure 1).

The coordinates of the hub and shroud lines on the meridional surface can then be calculated at each section from the midflow line coordinates, the $\delta(m)$ angle and the area $A(m)$.

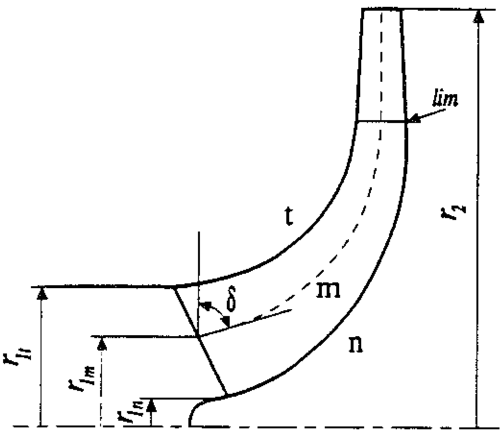

FIGURE 1 - SCHEMATIC OF PUMP IMPELLER.

The shroud coordinates are determined by

$$(2) \qquad r_t(m)=\sqrt{r_m^{\,2}(m)+\frac{A(m)\cdot sin[\delta(m)]}{2\cdot\pi}}$$

and

$$(3) \qquad z_t(m)=z_m(m)-\frac{r_t(m)-r_m(m)}{tan[\delta(m)]}$$

Equation (3) is applied until section *lim*, beyond which

$$(4) \qquad z_t(m)=z_m(m)-\frac{\pi\cdot r_m(m)}{A(m)}$$

Similar equations are applied in order to determine the hub coordinates $r_n(m)$ and $z_n(m)$.

The shape of the impeller on the S1 surface is determined by the blade curvature, for which the matching is needed between the blade angle values at the inlet (β_1') and at the outlet (β_2'). This design step needs wide flexibility in order to improve possible mechanical and fluid dynamic problems arising from the highly three-dimensional aspect of some impellers.

Typically, one can specify $\beta'(m)$ *as:*

$$(5) \qquad \beta'(m)=\beta_1'+(\beta_2'-\beta_1')\cdot\left[\frac{m-m_1}{m_2-m_1}\right]^{n_\beta}$$

were the n_β exponent is chosen by the designer.

Alternatively, the $\beta'(m)$ distribution can be taken by generalization of any analytical law for 2-D blade profiles (e.g. NACA correlations for compressor airfoils or wings). The blade angle values at the inlet, proposed by the one-dimensional calculation, can also be adjusted by choosing different incidence values at the midflow, hub and shroud respectively. The angular coordinate θ can then be calculated according to the approach presented in Grimaldi and Manfrida (1992).

Complete 3-D geometry (r-z-ϑ) of the three main sections of the impeller can then be supplied to a structural CAD workstation as a trial impeller configuration. More detailed information is also obtainable directly at PCAD level by considering intermediate sections on the meridional surface, defined by streamlines which divide the impeller in constant-percentage cross-section channels. On these surfaces, the same procedure is applied as that described for hub and shroud in order to calculate the blade geometry at that intermediate sections. This procedure can be a valid alternative to the running of S2 meridional codes, especially for low N_s pumps.

After the structural CAD analysis of global construction constraints, the possibly modified geometry can be transmitted back to the PCAD. The data referring to original PCAD configuration -and/or to the modified ones- can be transmitted also to a fluid dynamic solver for the analysis of the flow. An automatic interfacing procedure with a 2-D solver on the blade-to-blade surface is included in PCAD; it should be remarked that PCAD produces a complete three-dimensional geometry suitable for input to any 2-D and/or 3-D code.

The blade-to-blade code also allows to study the effect of flow prerotation at the inlet, as can naturally arise at low flowrates, or can be artificially induced by pre-swirl vanes.

VOLUTE DESIGN

Starting from an assigned cross sectional shape, the volute design procedure allows the determination of the cross section area in a number of preselected positions in a way that ensures circumferentially constant pressure.

The flow in the volute is assumed to be two dimensional and the basic equations for the design procedure are provided by the conservation of mass (Vinokur, 1974):

$$(6) \qquad \frac{\partial(rbu_r)}{\partial r}+u_\vartheta\frac{\partial b}{\partial\vartheta}=0\,.$$

and of angular momentum:

$$(7) \qquad \frac{1}{r}\frac{\partial}{\partial r}\left[bru_r(u_\vartheta r)\right]+u_\vartheta^2\frac{\partial b}{\partial\vartheta}=-2\frac{\tau_\vartheta}{\rho}r$$

where the local width b of the volute is let to be a function of both r and ϑ (Figure 2) and:

$$(8) \quad \tau_\vartheta=\frac{1}{2}\rho f u_\vartheta^2$$

is the shear stress in the tangential direction.

The integration of equation (6) between two subsequent cross sections leads to the following relation:

$$(9) \qquad Q_j=Q_{j-1}+Q_i$$

where:

(10) $$Q_i = u_{ri} b_i r_i \Delta\vartheta$$

(11) $$Q_j = \int_{r_i}^{r_{jmax}} u_\vartheta \varepsilon b \, dr$$

Equation (9) can be used to determine the maximum radius r_{jmax} of the local volute's section j, provided that the distribution of tangential velocity is available. Such distribution can be obtained by integrating the conservation equation (7) of the angular momentum.

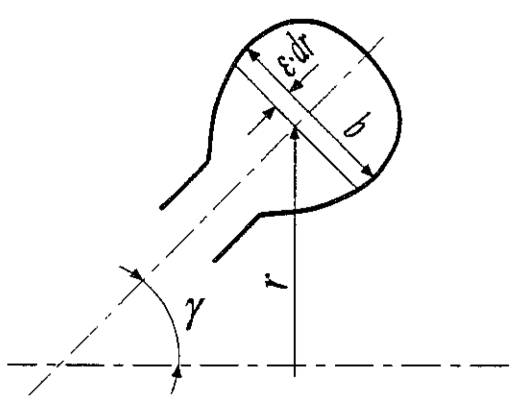

FIGURE 2 - SCHEMATIC OF VOLUTE GEOMETRY

In the inviscid case, equation (7) leads to the well known free-vortex law for the tangential component of velocity:

(12) $$u_\vartheta r = u_{\vartheta i} r_i$$

If friction is considered, an approximate "viscous" correction can be applied to (12), to account for the effects of the shear forces. For a two dimensional flow in a vaneless diffuser of constant width b, the integration of equation (7) provides:

(13) $$u_\vartheta = \frac{u_{\vartheta i} r_i}{r}\left[1 - f\frac{u_{\vartheta i}}{u_{ri}}\frac{r_i}{b}\left(\frac{r}{r_i}-1\right)\right]$$

For the case of the volute, where b is generally variable depending on the cross section shape, equation (13) is no longer valid. In this case, following the approach suggested by Byron Brown and Bradshaw (1949), the effect of friction can be taken into account by assuming that:

(14) $$u_\vartheta = \frac{u_{\vartheta i} r_i}{r}\left[1 - \varphi\left(\frac{r}{r_i}-1\right)\right]$$

where the friction index φ is assumed constant for each single volute section and can be determined by the conservation of angular momentum on each volute segment. According to numerical tests conducted during this work, the conservation of mass and angular momentum can be ensured, in the design procedure, following the method proposed in this paper. A locally variable friction index is computed from the relation:

(15) $$\varphi_i = f\frac{u_{\vartheta i}}{u_{ri}}\frac{r_i}{b}$$

and a mean value φ_j based on a mass flow average:

(16) $$\varphi_j = \frac{1}{Q_j}\int_{r_i}^{r_{jmax}} \varphi_i\left(\frac{dQ_j}{dr}\right)dr$$

is used, in equation (14), to express the tangential velocity distribution in the generic section j. The well known Colebrook - White formula is used to compute the local value of the friction coefficient f.

Starting from the assigned values of inlet geometric and flow data (b_i, r_i, u_{ri}, $u_{\vartheta i}$), equation (9) is solved iteratively, in each volute segment, for r_{jmax}. At each iterative step the actual cross section geometry is obtained by rescaling and positioning the chosen non dimensional section's shape, to accommodate the value of the maximum radius r_{jmax}. Discretized versions of equations (11) and (16) are used to update the value of Q_j and φ_j respectively. Once convergence is reached, the cross section area is computed in any of the selected tangential positions. As a result of the design process the local streamline slope is also computed from the relation:

(17) $$\tan\alpha = \frac{1}{r}\frac{dm}{d\vartheta}$$

The meridional component of the velocity is then:

(18) $$u_m = u_\vartheta \tan\alpha$$

and the shear stress:

(19) $$\tau_m = \frac{1}{2}\rho f u_m^2$$

The meridional pressure distribution can be computed by integrating the meridional component of the momentum equation:

(20) $$\frac{1}{\rho}\frac{dp}{dm} = \frac{u_\vartheta^2}{\varepsilon r} - u_m\frac{du_m}{dm} - \frac{\tau_m}{\rho b}$$

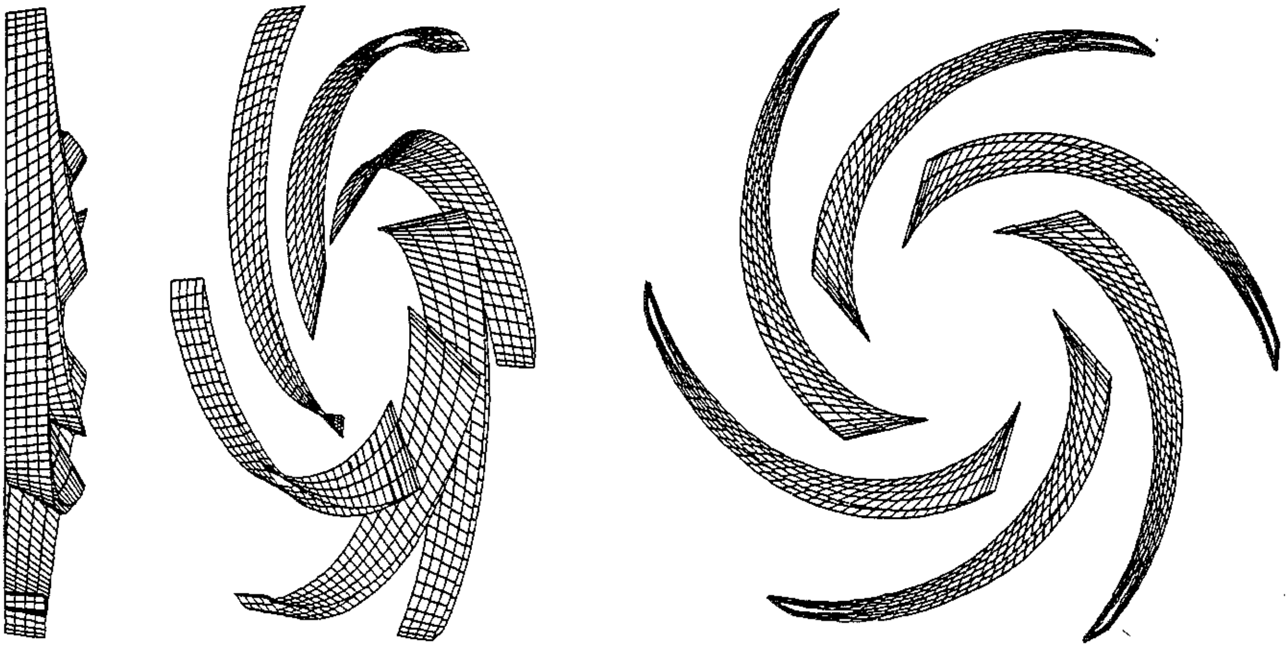

FIGURE 3 - THREE-DIMENSIONAL VIEWS OF THE IMPELLER

Since the design procedure leaves undefined the volute geometry, various cross section shapes and orientation angles can be tested to satisfy particular design requirements. The possibility offered by the proposed method, within the limits of the two-dimensional approach, of predicting the meridional distribution of the velocity components and of the pressure, represents, in the authors' opinion, a substantial improvement with respect to elementary (one-dimensional) volute design methods which assume uniform flow over each section and a linear increase in cross section in tangential direction (cfr. Lobanoff and Ross, 1986; and Stepanoff, 1957).

APPLICATION

As an example of PCAD capabilities, an application is presented of the design of a pump with the following expected performance:

$Q = 170\ \mathrm{m^3/h} \qquad H = 80\ \mathrm{m} \qquad N = 2900\ \mathrm{rpm}$

For the one-dimensional design, the example refers to the approach of Lobanoff and Ross for the impeller inlet, and to the area ratio principle of Anderson for the outlet, with AR=1.0.

Two different volutes have been generated with the following geometric characteristics:

a) radial orientation and circular cross section;
b) axial orientation and rectangular cross section with constant height.

The global output data are:

$NPSH_r = 6.4\ \mathrm{m} \qquad N_s = 1219\ \mathrm{rpm}\ (\mathrm{m^3/s})^{1/2}\mathrm{m}^{-3/4} \qquad Z = 6$

$r_{1t} = 0.0523\ \mathrm{m} \qquad r_2 = 0.128\ \mathrm{m}$

$\beta_{1t}' = 20.5\ ^\circ \qquad \beta_2' = 25\ ^\circ$

In Figure 3 three-dimensional, wire-frame visualizations of the impeller geometry are presented, as obtained from the PCAD-2 output data.

A part of the graphical output from the blade-to-blade code, applied with automatic impeller grid generation and direct PCAD interfacing, is shown in Figure 4. The S1 streamlines are represented for the three main sections: hub(n), midflow (m), and shroud (t)- and for two intermediate sections, at 25% and 75% respectively. For the same sections, velocity vectors are plotted in Figure 5.

The effect of a 40% prerotation ($u_{\vartheta 1}/(\omega r_{m1})$ =.4) of the inlet flow is shown in Figure 6, where the expected reduction of the incidence effects can be noted with respect to Figure 4m and Figure 5m.

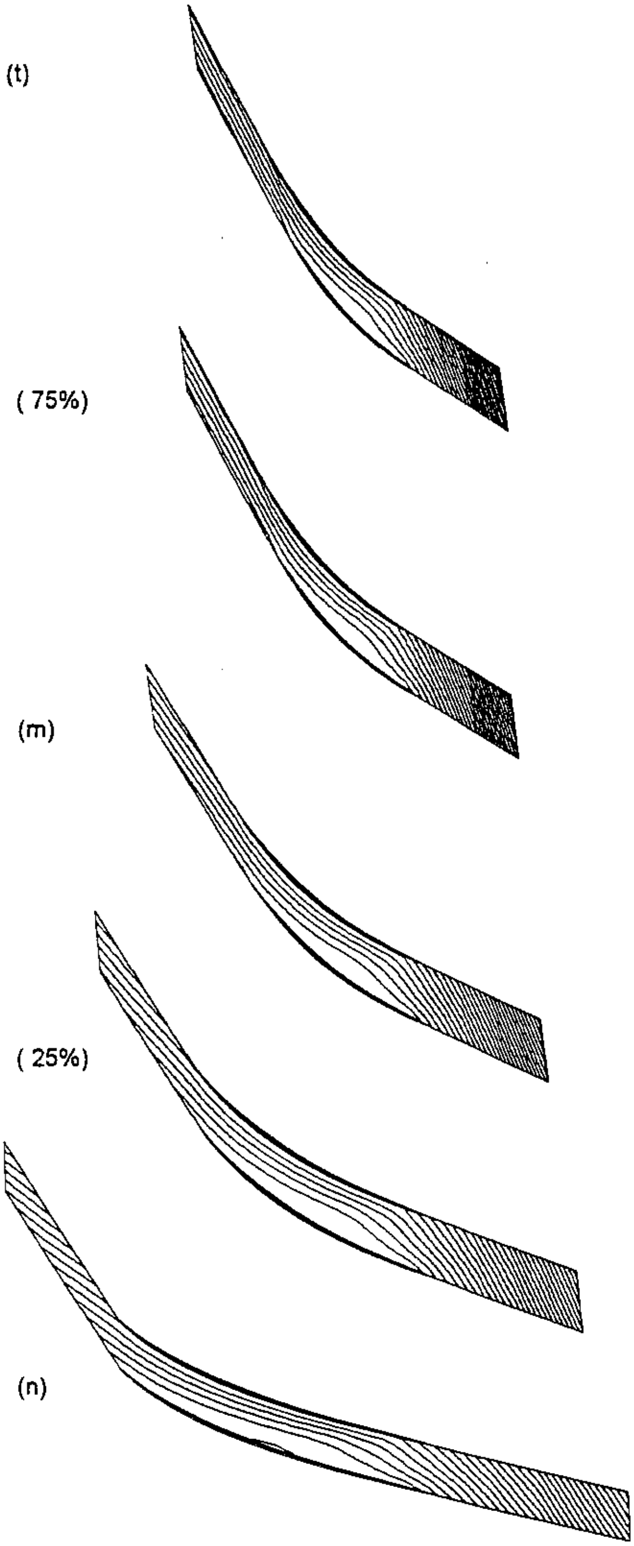

FIGURE - 4 - STREAMLINES ON S1 SURFACE.

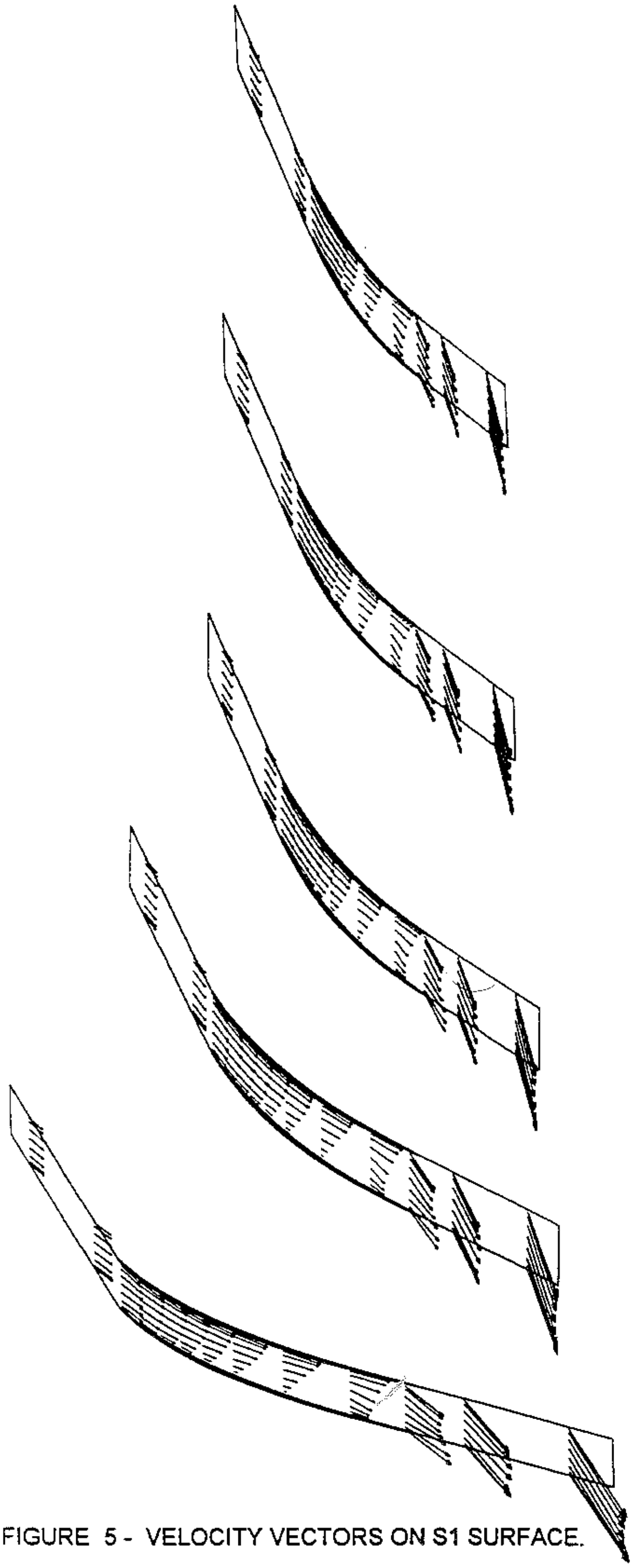

FIGURE 5 - VELOCITY VECTORS ON S1 SURFACE.

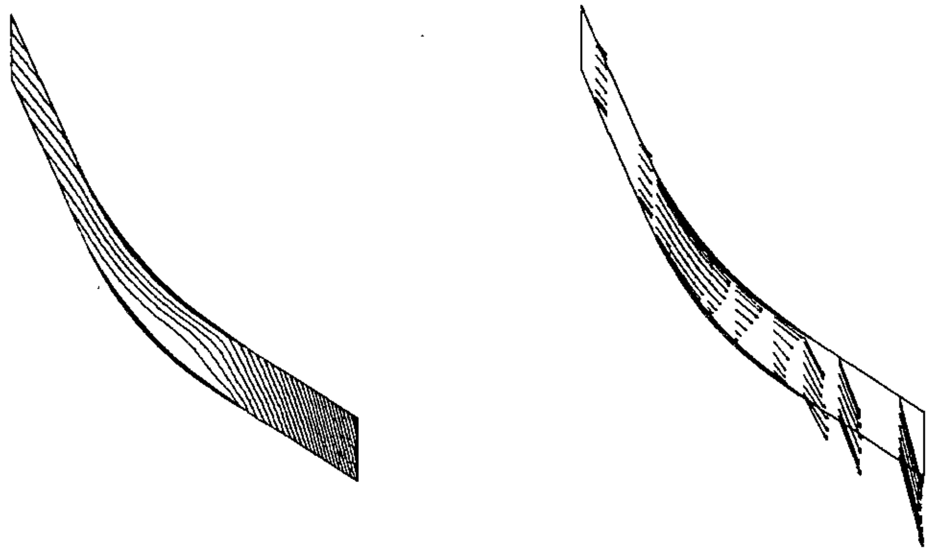

FIGURE 6 - STREAMLINES AND VELOCITY VECTORS ON S1 SURFACE FOR THE MIDFLOW LINE, WITH 40% PREROTATION.

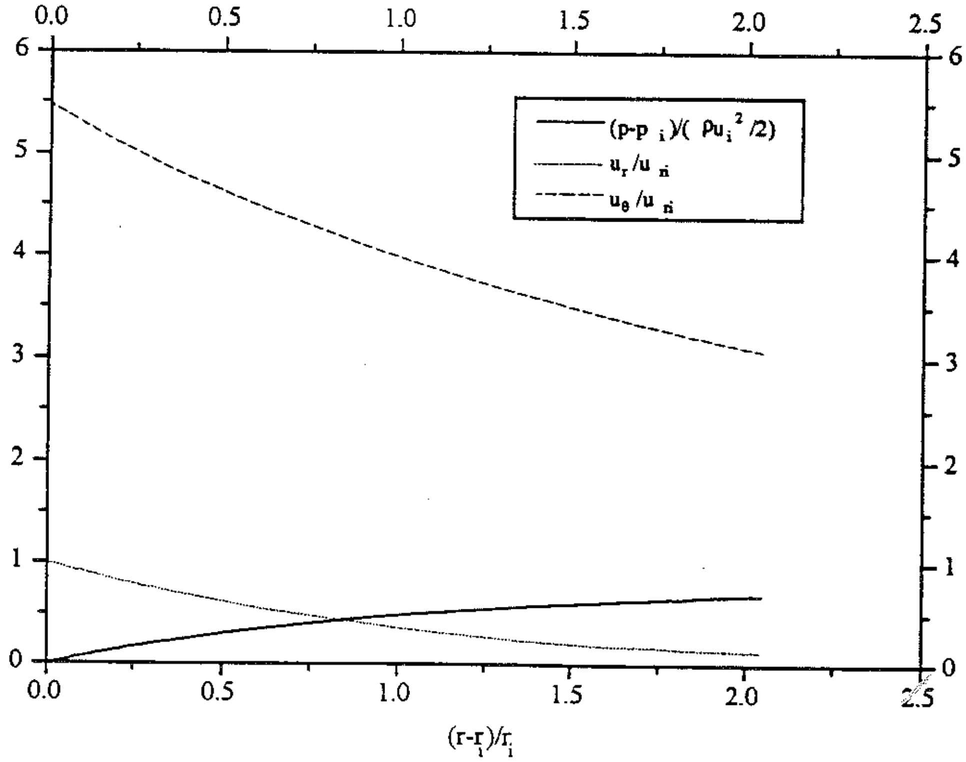

FIGURE 7 - RADIAL DISTRIBUTION OF NON-DIMENSIONAL COMPONENTS OF VELOCITY AND PRESSURE IN THE EXIT SECTION OF THE RADIAL VOLUTE.

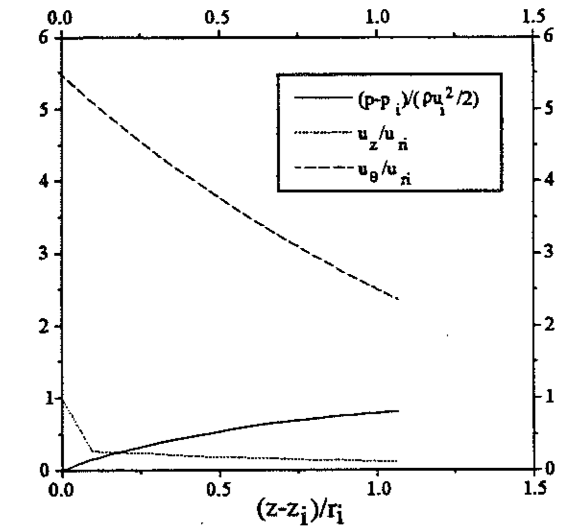

FIGURE 8 - AXIAL DISTRIBUTION OF NON-DIMENSIONAL COMPONENTS OF VELOCITY AND PRESSURE IN THE EXIT SECTION OF THE AXIAL VOLUTE.

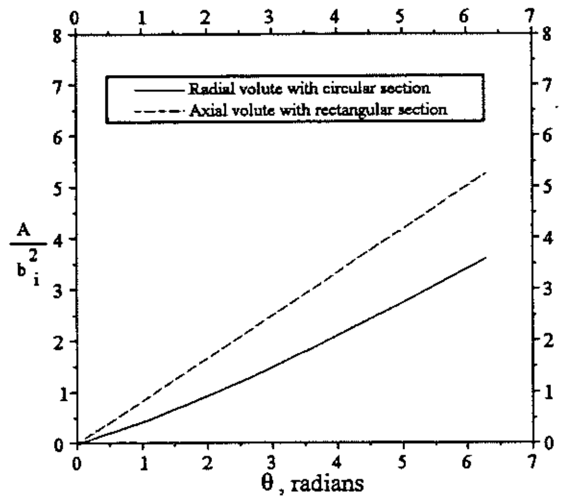

FIGURE 9 - NON-DIMENSIONAL AREA DISTRIBUTION OF THE TWO VOLUTES

The results of the volutes design procedure are synthesized in Figures 7, 8, 9, in terms of non-dimensional quantities. It can be seen that the meridional distributions of the velocity components and pressure (Figures 7 and 8) depend substantially on the volute shape and orientation. It can also be noticed that the condition of uniform flow in tangential direction can produce a non linear streamwise growth of the cross section area (figure 9) due to the meridional variation of the local flow rate. This suggest that traditional (one-dimensional) volute design methods, which assume a linear cross section area increase in the tangential direction (Lobanoff and Ross, 1986; Stepanoff, 1957), can lead to relevant circumferential pressure distortions even at nominal operating conditions.

ACKNOWLEDGEMENTS

A part of this work was developed under a research contract with Lowara S.p.A. (a division of Goulds Pumps Company).
The staff of the Research Division of Lowara S.p.A. is gratefully acknowledged for the helpful discussions and suggestions.

LIST OF REFERENCES

Anderson, N.H., 1980, "Centrifugal pumps", Trade and Technical Press, London.

Arnone, A., Liou, M.S., and Povinelli, L.A., 1991, "Multigrid calculation of three-dimensional Viscous cascade flows", AIAA paper 91-3238, 9th Applied Aerodynamics Conference, Baltimore.

Arnone, A., Stecco, S.S., 1991, "Multigrid calculation of incompressible flows for turbomachinery applications", XXIV IAHR Congress, Madrid.

Byron Brown, W., Bradshaw G. R., 1949, "Design and Performance of Family of Diffusing Scroll with Mixed Flow Impeller and Vaneless Diffuser", NACA Report 936

Gongwer, C. A., 1941, "A theory of cavitation flow in centrifugal-pump impellers", Trans. ASME, January, 29-40.

Japikse, D., and Brennen, Ch., 1991, "Centrifugal pump design and performance", Concepts ETI, Inc., Norwich, Vermont.

Lobanoff, V.S., and Ross, R.R., 1986, "Centrifugal pumps:design and application", Gulf Pub. Co.

Manfrida, G., Buzzi, P., and Michelassi, V., 1989, "Computer-aided fluid-dynamic design of centrifugal and mixed-flow pumps", Hydroturbo 89, Brno.

Manfrida, G., and Martelli, F., 1983, "Flow calculation in pump impellers by finite elements", 7th Conference on Fluid Machinery, Budapest, Hungary.

Pearsall, I.S., 1973, "Design of pump impellers for optimum cavitation performance", Proc. Inst. Mech. Engrs., 187, pp. 667-678.

Pearsall, I.S., 1978, "Off-Design performance of pumps", VKI LS 1978-3.

Peck, J.F., 1968, "Design of centrifugal pumps with computer aid", Proc. Inst. Mech. Engrs., 183, pp. 321-351.

Salisbury, A.G., 1983, "Current concepts in centrifugal pump hydraulic design", Proc. Inst. Mech. Engrs., 197, pp. 221-231.

Stepanoff, A.J., 1957, "Centrifugal and axial flow pumps", Wiley, New York.

Yedidiah, Sh., 1971, "Effect of inlet vane design on cavitation in centrifugal pumps", ASME Cavitation and Multiphase Flow Forum, New York.

Vinokur, M., 1974, "Conservation Equations of Gasdynamics in Curvilinear Coordinate Systems", Journal of Computational Physics, vol. 14.

Worster, R.C., 1963, "The flow in volutes and its effect on centrifugal pump performance", Proc. Inst. Mech. Engrs., 177, pp. 843-861.

Wu, C.H., 1957, "A general theory of the two-dimensional flow in subsonic and supersonic turbomachines of axial, radial and mixed-flow type", NACA TN 2604.