NUMERICAL ANALYSIS OF THE 3D SWIRLING FLOW IN CENTRIFUGAL COMPRESSOR VOLUTES

E. Ayder
R. Van den Braembussche
von Karman Institute for Fluid Dynamics
Chausée de Waterloo 72
Rhode-Saint-Génèse
BELGIUM

ABSTRACT

The improvement of centrifugal compressor performance and the control of the radial forces acting on the impeller due to the circumferential variation of the static pressure caused by the volute require a good understanding of the flow mechanisms and an accurate prediction of the flow pattern inside the volute. A 3D volute calculation method has been developed for this purpose.

The volute is discretized by means of hexahedral elements. A cell vertex finite volume approach is used in combination with a time marching procedure. The numerical procedure makes use of a central space discretization and a four step Runge-Kutta time stepping scheme. The artificial dissipation used in the solver is based on the fourth order differences of the conservative variables. Implicit residual smoothing improves the convergence rate. The loss model implemented in the code accounts for the losses due to internal shear and friction losses on the walls.

A comparison of the calculated and measured results inside a volute with elliptical cross sectional shape reveals that the modified Euler solver accurately predicts the velocity and pressure distribution inside and upstream of the volute.

LIST OF SYMBOLS

- \( \bar{Q} \) vector of residuals
- \( S \) area
- \( t \) time
- \( T \) temperature
- \( \mathbf{u}, \mathbf{v}, \mathbf{w} \) cartesian components of the velocity
- \( \mathbf{V} \) velocity
- \( \mathbf{W} \) vector of conservative variables
- \( \alpha \) ratio of through flow velocity to radial velocity
- \( \beta \) ratio of axial velocity to radial velocity
- \( \gamma \) ratio of specific heats
- \( \rho \) density
- \( \epsilon \) implicit residual smoothing coefficient
- \( \Omega \) volume
- \( \tau \) shear stress

Subscripts

- \( S \) swirl component
- \( T \) through flow component
- \( R \) radial
- \( Z \) axial

Superscripts

- \( o \) stagnation conditions
- \( - \) averaged value

INTRODUCTION

The large number of parameters influencing the performance of a centrifugal compressor volute prohibit systematic experimental investigation because of the time and cost involved in the manufacturing and testing of the complex three-dimensional geometries. A reliable prediction method...
is therefore of great help in determining the influence of the different design parameters on the volute flow and losses.

Most prediction methods presented in the literature are based on simplified models assuming one or two dimensional flow (Iversen et al, 1960, Kurokawa, 1980, Badie et al, 1992). However, the experimental results of Muller (1973), Van den Braembussche and Hande (1990), and Ayder et al (1991,1992) show that the flow inside the volute is highly three-dimensional, and that the swirling velocity component has an important influence on the cross-wise and circumferential variation of the static pressure and velocity distribution. One- and two-dimensional methods are therefore of limited interest, and are unable to provide a reliable prediction of the circumferential pressure distortion and performance of three-dimensional volutes.

The way the volute flow is built up of layers of non-uniform total pressure and temperature (Van den Braembussche and Hande et al, 1990), in addition to the high shear forces at the center of the volute, results in a rotational flow. The methods dealing with the solution of the potential flow equations developed to analyse the flow in turbine volutes (Hamed et al, 1983) are of no use here since they start from the irrotational flow equations.

Inviscid analytical methods lead to non-physical gradients of density, velocity, pressure and temperature at the center of the vortex (Mayer and Powell, 1990).

The numerical solutions of the Euler equations, however, seem to exhibit no problems in the calculation of vortices. All show regular solutions, with the swirling velocity going to zero at the center of vortices, and with finite values for the pressure, density and axial velocity (Rizzi and Erikson, 1984, Powell, 1990). The last one concludes that the artificial dissipation required to solve the discretized Euler equations regularizes the solutions near the vortex center, where the flow is governed by shear flow.

NUMERICAL METHOD

Experimental results have shown that the flow in volutes is affected more by the losses in the core than by the wall boundary layer. Increased turbulent mixing on the concave walls (Johnston, 1970), limits the growth of the boundary layer which are absorbed in any case by the core flow after each rotation. Boundary layer blockage can therefore be neglected, and its influence may be limited to the shear forces on the walls.

Based on these observations, it has been decided to solve the problem by adding second order dissipation and wall shear forces to an Euler solver, rather than resorting to a more expensive and time consuming solution of the Navier-Stokes equations.

The method solves the following integral form of the Euler equations:

\[
\int_{\Omega} \int_{\partial \Omega} \mathbf{G} \cdot \mathbf{n} \, dS + \int_{\partial \Omega} \mathbf{H} \cdot \mathbf{n} \, dS = 0
\]

applied to a control volume \( \Omega \) of surface \( \partial \Omega \). \( \mathbf{n} \) is an outer normal to the surface element \( dS \). The vectors of this equation are defined by:

\[
\mathbf{W} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{pmatrix} \quad \mathbf{F} = \begin{pmatrix} \rho u \\ \rho u^2 + P \\ \rho u v \\ \rho u w \\ \rho u(E + P/\rho) \end{pmatrix}
\]

\[
\mathbf{G} = \begin{pmatrix} \rho v \\ \rho v u \\ \rho v^2 + P \\ \rho w v \\ \rho w(E + P/\rho) \end{pmatrix} \quad \mathbf{H} = \begin{pmatrix} \rho w \\ \rho w u \\ \rho w v \\ \rho w^2 + P \\ \rho w(E + P/\rho) \end{pmatrix}
\]

and the system can be solved by means of following closure condition:

\[
p = (\gamma - 1) \left[ \rho E - \frac{1}{2} \rho (u^2 + v^2 + w^2) \right]
\]

All quantities in these equations are nondimensionalized by means of \( l \) (Characteristic length of the channel), \( R \) (Gas constant), \( T^0 \) (total temperature), \( P^0 \) (total pressure), \( \rho^0 \) (total density) measured at the exit of the compressor.

Discretization

Elementary control volumes are defined by four sided (quadrilateral) elements on a number of cross sections of the volute and upstream diffuser (Figure 1). The same number of quadrilateral cells is used for each cross section in order to obtain a structured grid and to enhance the resolution of the flow patterns in small cross sections. Connecting the corresponding elements of each cross section to one another produces hexahedral finite volume cells. The discretization of the flow domain in the tongue region requires special attention. The first discretized cross section of the volute is a line in space coinciding with the tongue. This leads to the triangular prism type of volumes between the tongue and first cross section.

The method makes use of a cell-vertex approach in which the unknowns (the components of \( \mathbf{W} \)) are defined at the vertices of the hexahedron. This provides an easy implementation of the solid wall boundary conditions, since discretization points are on the wall.

A control volume around the point \((i, j, k)\), on which the integral form of the Euler equations will be applied is defined by the eight hexahedrons containing the point under consideration. This leads to overlapping control volumes, which is allowed in the finite volume approach.

Time discretization of the unsteady Euler equations is done by using well known four step Runge-Kutta scheme proposed by Jameson et al (1981, 1983).
Since the volute flow is subsonic, and no shocks are expected, the dissipative flux $D_{i,j,k}$ will be proportional to the fourth order differences of the conservative variables in all three index directions and second order dissipation will not be required to stabilize the equation. The evaluation of the fourth order differences presents some problems near the boundaries. In the present study, the approach suggested by Kroll and Jain (1987) is used to determine the artificial dissipation at the boundary points. The dissipation term is only calculated in the first step of the Runge-Kutta time integration and is kept constant for the other three steps.

**Boundary Conditions**

A theoretical analysis of the three dimensional Euler equations indicates that four physical flow quantities have to be imposed at the inlet of the calculation domain.

In addition to the total pressure and temperature, the third and fourth physical conditions impose the ratio of the through flow velocity to the radial velocity ($\alpha$) to determine the through flow velocity

$$V_{T,i} = \alpha \cdot V_{R,i}$$

and the ratio of the radial velocity to the axial flow velocity ($\beta$) to determine the axial velocity

$$V_{z,i} = \beta \cdot V_{R,i}$$

The unknown radial velocity is determined as a function of the internal flow by a numerical boundary condition by which the radial component of the velocity of the first interior point ($i+1$) is extrapolated to the inlet boundary point ($i$). The following relation is used in the extrapolation procedure to account for the change in area

$$V_{R,i} = V_{R,i+1} \frac{S_{i+1}}{S_i}$$

All other nondimensionalized variables at the inlet can then be calculated by means of the thermodynamic relations.

The downstream static pressure is not uniform because it results from a radial equilibrium with the centrifugal forces, and its definition requires the calculation of the local streamline curvature radius.

A more practical method consists of imposing the static pressure at one point of the outlet plane and to calculate the static pressure in all other points by imposing the pressure ratio, calculated between the reference points,

$$\text{Ratio} = \frac{P(i_{ref}, j_{ref}, k_{max})}{P(i_{ref}, j_{ref}, k_{max} - 1)}$$

to all corresponding points of the two planes.

$$P(i, j, k_{max}) = \text{Ratio} \cdot P(i, j, k_{max} - 1)$$

The three velocity components and density can then be calculated from the Euler equations discretized on the boundary elements. Together with the static pressure, they are used to determine the conservative variables.
The solid wall boundary condition result from imposing that no mass or other convective flux can penetrate the solid wall.

\[ V_n = 0 \]

As a consequence, all convective flux components through the solid wall vanish and only the pressure contributions at the solid walls remains in the flux vectors \( \vec{F}, \vec{G} \) and \( \vec{H} \).

The variables other than the normal velocity are calculated by solving the discretized Euler equations on a one sided control volume.

**Acceleration Technique**

The implicit residual averaging provides a better convergence rate compared to the explicit one and is therefore used in the present study (Jameson 1983). The residual averaging is done at the second and fourth steps of the Runge-Kutta scheme.

**Loss Calculation**

Determination of the shear stresses on all inner volumes of the volute channel requires a flow adapted grid where the volume surfaces coincide with the stream surfaces, or are perpendicular to the velocity. This is very difficult to achieve in all volutes of different cross sectional shapes and varying flow conditions. Shear forces will therefore be calculated only on the walls and applied to the volumes having a surface in common with the wall. The internal friction is approximated by a viscous term.

The additional momentum caused by wall friction is included by adding a term \( \bar{L}_{i,j,k} \) to the momentum equations.

\[
\Omega_{i,j,k} \left( \frac{d}{dt} \vec{W}_{i,j,k} \right) + \vec{Q}_{i,j,k} - \vec{B}_{i,j,k} - \bar{L}_{i,j,k} = 0 \quad (5)
\]

Neglecting the displacement thickness of the sublayer and assuming that the surface streamline lies at the edge of the laminar sublayer, the wall shear stress can be defined by the following relation

\[ \tau_{wall} = C_f \frac{1}{2} \rho V^2 \quad (6) \]

resulting in following components on each element of the wall

\[
\tau_x = -\frac{V}{V} \tau_{wall}, \tau_y = -\frac{V}{V} \tau_{wall}, \tau_z = -\frac{V}{V} \tau_{wall}
\]

Due to the nonuniformity of the total temperature at the inlet of the volute, the simplification of the energy equation suggested by Denton (1986), cannot be used and the energy dissipation due to wall friction

\[
-\tau V (S_x + S_y + S_z)
\]

must be added to the loss vector \( \vec{L} \).

\[
\vec{L} = \begin{pmatrix}
0 \\
\tau_x (S_x + S_y + S_z) \\
\tau_y (S_x + S_y + S_z) \\
\tau_z (S_x + S_y + S_z) \\
\tau V (S_x + S_y + S_z)
\end{pmatrix}
\]

As the loss vector is calculated only on the solid wall surfaces, it influences only the vertices of the volumes adjacent to the solid walls and creates a local velocity gradient perpendicular to the wall.

**FIG. 3:** Calculated static pressure isolines (a) and velocity (b) over the mid plane of the vaneless diffuser at high mass flow.
The viscous energy dissipation, at all regions of non-zero velocity gradient (at the wall and at the vortex center) is achieved by the second order dissipation

\[
k^2 (W_{i-1,j,k} + W_{i+1,j,k} + W_{i,j-1,k} + W_{i,j+1,k} + W_{i,j,k-1} + W_{i,j,k+1} - 6W_{i,j,k})
\]

The coefficient \(k^2\) is adjusted empirically to simulate the viscous terms in the Navier Stokes equations. Actual calculations are done with a \(k^2\) value of 0.002.

**RESULTS**

The computer program has been validated by calculating the three dimensional swirling flow inside a volute of elliptic cross section for which detailed experimental data have been obtained by Ayder et al (1992). The flow field is extended upstream to the impeller outlet in order to be able to study the flow around the tongue, and to minimize the influence of the inlet conditions on the tongue and on the volute flow. However, the inlet boundary conditions (total pressure, total temperature and the ratios of tangential and axial velocity over the radial velocity) have been measured only at the volute inlet. The values used at the inlet of the calculation domain are derived from the measured ones by correcting for the change in radius between diffuser inlet and volute inlet, taking into account the circumferential shift due to the peripheral velocity in the diffuser. Vaneless diffuser losses and the axial variation of the flow over the diffuser width are neglected.

The static pressure at the outlet equals the atmospheric pressure and is fixed at one point of the solid wall in the last plane of the exit cone.

The vaneless diffuser is discretized by \(5 \times 15\) grid points on each of the 59 cross sections. The volute and exit cone is discretized by \(15 \times 15\) grid points in 74 cross sections. Cross sections are concentrated near the tongue region to obtain a more detailed flow description. A three dimensional view of the geometry is shown in Figure 1. Variation of the cross sectional area along the volute channel, starting from the volute tongue to the volute exit, is shown on Figure 2.

The calculations have been performed for operating points corresponding to high, medium and low mass flows. Because of the limited space, detailed results will be shown only for high and low mass flows. Calculation time is 16 hours on a Silicon Graphics IRIS 4D/35 computer.

**Diffuser Flow**

The variation of velocity and static pressure over the mid-plane of the diffuser is shown in Figure 3. As can be expected at high mass flow, the static pressure decreases along the circumference of the diffuser in the direction of the rotation of the impeller up to an abrupt increase in static pressure around the tongue (Figure 3a) resulting in a strong perturbation of the local velocity vectors (Figure 3b). Almost no fluid enters the volute in the region of high pressure. This phenomenon is similar to what has been observed by Elholm et al (1991).

The radial velocity (\(V_R\)) and static pressure (\(P\)) distribution at the diffuser inlet (and as a consequence, also at the volute inlet) are a result of the calculation. Spanwise averaged calculated values are compared with experimental values in Figure 4 and shows a very good agreement for all flow conditions.

**Volute Flow**

The variation of the measured and calculated swirl velocity (\(V_s\)), through flow velocity (\(V_T\)), total pressure (\(P^0\)) and static pressure (\(P\)) over the sections corresponding to the section 1, 5 and 7 of the experimental study (indicated on Figure 1), are shown in Figure 5 for high mass flow, and in Figure 6 for low mass flow.

The mechanisms defining the change of flow quantities
FIG. 5a: Calculated (left) and measured (right) swirl velocities at high mass flow

FIG. 5b: Calculated (left) and measured (right) through flow velocities at high mass flow (m/sec)

FIG. 5c: Calculated (left) and measured (right) static pressure at high mass flow (Pascal)

FIG. 5d: Calculated (left) and measured (right) total pressure at high mass flow (Pascal)

over the cross section have been explained in detail by Ayder et al (1992) and will not be repeated.

The circumferential variation of the calculated flow quantities is very similar to the one observed in the experimental study.

The magnitude and variation of the calculated swirl and through flow velocities over the cross sections and along the volute channel are in agreement with the measurements. (Figures 5a,b and 6a,b).

Measured results indicate high total pressure losses at the center of the first cross section (Figure 5c) due to the dissipation of the swirl kinetic energy. The calculations underestimate the concentration of low total pressure at the center of the first cross section. However the calculated total pressure outside the viscous core is in agreement with the measured one (Figure 5c).

The calculated magnitude of the total pressure and static pressure over the cross section 5 are slightly higher than the measured ones for high mass flow (Figure 5c,d). The reason for this is unknown. However, the measured and calculated
total and static pressure over the cross section 7 are different. It can be seen that the high total pressure losses, occurring at high mass flow in the exit cone, are underestimated. Measurements at section 7 being restricted to a small portion of the cross section and it is not possible to find out what is the origin of this high losses. It may be due to separation at the tongue.

In case of the low mass flow, the experimental results do not exhibit large total pressure gradients caused by the viscous effects. The calculated total and static pressures therefore shows better agreement with the measured ones (Figure 6c,d).

Calculated results are used to trace particles inside the diffuser and volute. The traces start at four circumferential positions at the diffuser inlet. The results obtained at medium mass flow are shown in Figure 7. One observes that the fluid entering the volute at a position close to the tongue (A) remains in the center of the volute until the compressor exit. Fluid entering the volute further downstream (B,C,D) wraps around the previous one which confirms the volute...
FIG. 7: Particle tracing inside the vaneless diffuser and volute at medium mass flow


CONCLUSIONS

To our knowledge, this is the first time that the complex flow in a volute is calculated using a fully 3D method. Taking into account the complexity of the geometry and the uncertainty of the experimental data, results are assumed to be very satisfactory.

Comparisons between the calculated and measured flow fields inside the volute show a good prediction of the cross sectional variation of the swirl and through flow velocity. The total and static pressure show a more qualitative agreement, but the internal shear loss mechanism, typical for swirling flows, is correctly captured.

The circumferential distortion of the static pressure and radial velocity at the volute inlet, required for impeller response calculations, is accurately predicted.

The calculated flow patterns around the tongue, and leakage flow, are in agreement with the flow visualizations.

The calculation of flow patterns by particle tracing also confirms the validity of the volute flow model proposed by Van den Braembussche and Hande (1990).

The proposed combination of an Euler solver with a volute adapted loss model has shown to be an accurate alternative to the solution of the full Navier Stokes equations of which one can expect that the computational effort will be much larger.

REFERENCES


DENTON, J.D., 1986, “The use of a distributed body force to simulate viscous effects in 3D flow calculations”, ASME Paper 86-GT-144


