A Two Dimensional Flow Analysis Model for Designing a Nozzle-Less Volute Casing for Radial Flow Gas Turbines

H. O. OWARISH
RAM Research and Engineering Ltd
The Lord Rank Research Centre
M. ILYAS
Senior Lecturer
F. S. BHINDER
Professor
School of Engineering
Hatfield Polytechnic
Hatfield, Herts., AL10 9AB. U.K.

INTRODUCTION

The isentropic efficiency of an inward flow radial (IFR) gas turbine fitted with a nozzle-less volute casing tends to be a few percentage points lower than that fitted with a casing incorporating nozzles. However, because of its lower cost and wider operating range the nozzle-less volute casing is widely used in small turbochargers for the automotive class diesel engines. However, it has been seen that the performance handicap of the nozzle-less casing can be overcome by using better design techniques. The aim of this paper is to describe the details of a two dimensional flow analysis model which may be used to improve the quality of design of such casings.

Radial flow turbines are suitable for a variety of applications including auxiliary power units for space craft. Hence considerable research effort has gone into improving their performance and a large number of papers have been written on the aero-thermodynamics, design and performance of these machines. However, much of the published work deals with the rotor. In contrast, the casing does not appear to have excited the imagination of researchers and design engineers to the same extent. This is in spite of the fact that flow in the casing is very complex being three dimensional, compressible, viscous and, in the case of turbochargers, also unsteady.

The available design methods for the casing invariably assume the flow to be steady and one dimensional. Hamid and Bashkarone [1] developed a three dimensional flow analysis method which made a valuable contribution but perhaps it was too advanced for the design office use. A detailed review of the published literature is omitted from here for the sake of brevity, but the reader's attention is drawn to the work by Husain [3] for further study.

DESIGN REQUIREMENTS

The constructional features of a typical single entry nozzle-less volute casing are shown in Fig.
1(a). In addition to housing the rotor, the casing must be designed to perform the following aero-thermodynamic functions:

a. to convert some of the static enthalpy of the working fluid into kinetic energy and hence accelerate the fluid to the desired rotor entry Mach number;

b. to direct the high velocity fluid streams into the rotor at the prescribed absolute flow angle and to achieve the congruence of the velocity triangles around the periphery of the rotor;

c. to achieve (i) and (ii) with minimum loss of stagnation pressure in the casing.

Pressure taps, 1.5 mm dia.
15 deg. spacing, at centroid along the volute length.

Fig. 1(a) Constructional features of a nozzle-less volute casing.

Fig. 1(b) The expansion process in the casing on the Temperature - Entropy plane

The expansion process that takes place in the casing is shown on the Temperature - Entropy diagram in Fig. 1(b). The flow in the casing is influenced by the detailed design of the following sections:

(i) the entry region, in particular the volute tongue;
(ii) the cross sectional area vs the azimuth angle distribution;
(iii) the cross-sectional shape of the volute.

The important performance parameters that must be optimized are: the stagnation pressure loss factor $\zeta$, total to static isentropic efficiency $\eta$ and the variation of the absolute flow angle $\alpha$ at rotor entry with the azimuth angle $\phi$. The loss factor and efficiency are defined as follows:

Stagnation pressure loss factor $\zeta = \frac{P_{01} - P_{02}}{P_{01}}$

Total to static isentropic efficiency

$\eta_{t-s} = \frac{T_{01} - T_{2}}{T_{01} - T_{2}'}$

(2a)

Since the stagnation temperature is assumed to remain constant, i.e. $T_{02} = T_{01}'$, the equation (2a) can be written as follows:

$\eta_{t-s} = \frac{1 - T_{2} / T_{02}}{1 - T_{2}' / T_{02}}$

From Fig. 1(b)

$\eta_{t-s} = \frac{1 - \frac{P_{2}}{P_{02}} Y - 1}{1 - \frac{P_{2}'}{P_{02}'} Y - 1}$

(2b)

This equation can be written also in terms of $\zeta$. 
TWO DIMENSIONAL FLOW MODEL

By considering the forces acting on a fluid element shown in Fig. 2(a), a set of equations can be derived to calculate the step changes in the following fluid properties across the element:

- (a) absolute static pressure
- (b) absolute flow angle
- (c) radial velocity
- (d) absolute static temperature
- (e) fluid density.

The derivations of the basic equations are given in the following:

(a) Change in Static Pressure The relationship between the mean static pressure and radius can be obtained by considering the forces acting on the control volumes in the radial direction, Figs. 2(a) and 2(b).

\[ -P_{A1} = \left( p + \frac{\delta p}{2} \right) \left( A_{r1} - A_1 \right) + (p + \delta p)A_1 \]

\[ + C_r \rho v_r^2 \sin(\alpha) = \rho V_a r \]

(b) Change in Absolute Flow Angle The relationship between the absolute flow angle and the radius can be obtained by differentiating the equations for radial and tangential velocities with respect to radius and substituting the result into the equation for momentum in the tangential direction. The derivation is given below:
\[ p \cdot V \cdot a_t = \left[ P + \frac{\delta P}{2} \right] A_2 - \left( P + \frac{\delta P}{2} \right) A_2, \]
\[ + \left[ P - \frac{\delta P}{2} \right] \left( A_1 - A_2 \right) - F_t \]
\[ = - F_t \]  

where \( a_t \) is the acceleration and \( F_t \) the force due to friction, both in the tangential direction given by the following expressions:

\[ a_t = \frac{C_r \rho C_r}{r} + C_r \frac{\delta C_r}{\delta r} \]  
\[ F_t = C_r C^2 A_0 \cos(\alpha) \]

The relationship between the tangential and the radial components of the absolute velocity \( C \) is given by the following:

\[ \tan(\alpha) = \frac{C_r}{C_w} \]  

Differentiating this equation with respect to the radius \( r \):

\[ \tan(\alpha) \frac{\delta C_r}{\delta r} - \frac{1}{r} \frac{\delta C_r}{\delta r} = \frac{\delta C_w}{\delta r} \]  

Combining equations (7), (8) and (10b) and simplifying the expression for change in the absolute flow angle with respect to radius can be obtained.

\[ \delta(\tan \alpha) = \tan \alpha \left( \frac{\delta C_r}{C_r} + \frac{F_t}{C_r C_w \rho V} + \frac{1}{r} \right) \frac{\delta r}{\delta r} \]  

(c) Change in Radial Velocity

The change in the radial velocity as a function of the radius can be found by differentiating the equation for mass flux in the radial direction. Assuming mass flux in the radial direction is a function of the radius:

\[ \frac{1}{\rho} \frac{\delta p}{\delta r} + \frac{1}{\rho} \frac{\delta \Delta}{\delta r} + \frac{1}{\rho} \frac{\delta C_r}{\delta r} \frac{\delta r}{\delta r} = \frac{\delta}{\delta r} \left( C(\tau) \right) \]  

If mass flux in the radial direction is assumed to be constant, the above equation simplifies to the following:

\[ \frac{\delta C_r}{\delta r} = \left( \frac{\delta p}{\rho} + \frac{\delta \Delta}{\Delta} \right) C_r \]

(d) Change in Static Temperature

The change in static temperature with respect to radius can be found by differentiating the energy equation with respect to radius as follows:

\[ T_0 = T + \frac{1}{2} p \left( \frac{C^2}{C_w} + \frac{C^2}{C_r} \right) \]  

Differentiating with respect \( r \)

\[ \frac{\delta T_0}{\delta r} = \frac{\delta T}{\delta r} + \frac{1}{p} \left( \frac{\delta C_w}{\delta r} + C_r \frac{\delta C_r}{\delta r} \right) \]  

Assuming adiabatic flow and substituting for \( C_w \) this equation can be written as:

\[ R_b = k_1 \phi + k_2 \phi^2 + k_3 \phi^3 + \ldots \]  

Fig. 3(a) Calculation Procedure for flow analysis

Main Program

\[ \frac{\delta T}{\delta r} = - \frac{1}{p} \left( \frac{\delta p}{\delta r} + \frac{1}{\Delta} \frac{\delta \Delta}{\delta r} + \frac{1}{\rho} \frac{\delta C_r}{\delta r} \right) \]  

(e) Change in Density

Since static pressure and temperature are known, the density of the working fluid can be calculated directly from its equation of state as follows:

\[ \frac{\delta \rho}{\delta r} = \rho \left( \frac{\delta p}{\delta r} + \frac{1}{\rho} \frac{\delta \Delta}{\delta r} + \frac{1}{\rho} \frac{\delta C_r}{\delta r} \right) \]  

COMPUTER PROGRAMS

A suite of programs has been written which included flow analysis as well as design. The calculation procedure is shown in flow diagrams of the programs in Figs. 3(a), 3(b), and 3(c). The program requires the following input data:

(a) total pressure at inlet
(b) total temperature at inlet
(c) mass flow rate at inlet
(d) geometric parameter to be used to calculate the casing dimensions

The execution of the program starts with the fitting of a polynomial equation to a given set of data relating the outer radius of the volute to the azimuth angle. Using a least square fit method, an equation of the following type is obtained.

\[ R_b = k_1 \phi + k_2 \phi^2 + k_3 \phi^3 + \ldots \]  

The main program shown in Fig. 3(a) uses the sub-routine S1 to calculate the flow properties across
the segments spanning each incremental azimuth angle \( \delta \phi \). A relationship between the radial velocity at the outer wall and the mass balance across the segment is satisfied. The sub-routine \( S2 \) is used to establish the mass balance.

In subroutine \( S2 \), starting with an assumed value of radial velocity at the outer wall which has been passed across from \( S1 \), a function is defined relating the difference between the assumed and the new calculated values of the radial velocity at the control volume exit. In order to find the step change across the control volume, the resulting transcendental equation is solved by using a NAG routine. Other flow properties at the exit of the control volume and the mass flow rate in the tangential direction are also calculated. The process is repeated until the flow properties from the outer wall to the rotor inlet and the total mass flow in the tangential direction have been determined.

The details of the program dealing with the calculation of the geometrical dimensions are omitted from here for the sake of brevity, but the reader's attention is drawn to reference [6].

**VALIDATION OF THE PROGRAMS**

The programs were validated on an existing single entry nozzle-less volute casing for which comprehensive experimental data were available. Space restrictions do not allow full design details of the casing and of the test procedure to be given here, but they may be found in reference [3]. Here it should suffice to mention that the cross-sectional shape of the casing was circular and the geometrical dimensions were measured from the drawings. Results of the validation are discussed in the following.

**Static Pressure Calculations** Comparison between the calculated and measured values of static pressure

---

**Fig. 3(b) Calculation of the total mass flow in the radial direction - Subroutine S1**

**Fig. 3(c) Calculation of the total mass flow in the tangential direction - Subroutine S2**
ratio along the centroid of the volute, close to the rotor entry for \( \phi \) ranging from 0 to 360 degrees, and at different radius ratios at \( \phi = 0 \) are shown in Fig. 4(a), 4(b) and 4(c) respectively. The agreement along the centroid is not very good, but it must be remembered that the position of the actual centroid may differ widely from that obtained from the draught because of the casting inaccuracies.

**Tangential Velocity Calculations** Figures 5(a), 5(b) and 5(c) show the comparison between the calculated and measured values of the tangential velocity ratio along the centroid, close to the rotor entry and at different radius ratios at \( \phi = 0 \) respectively. In this case the difference between the calculated and measured values at the rotor entry increases as \( \phi \) approaches 360 degrees. This may be due to the influence of the tongue because towards the tail end of the volute the centre of the volute moves close to the rotor.

**Radial Velocity Calculations** The calculated and measured values of the radial velocity at the centroid and close to the rotor entry for \( \phi \) ranging from 0 to 360 degrees and at different radii for \( \phi = 70 \) degrees are shown in Fig. 6(a), 6(b) and 6(c) respectively. The model appears to overestimate the radial velocity along the centroid, but the trends shown in Fig. 6(a) as well as in 6(b) are very similar to those observed experimentally. The agreement between the measured and calculated values at \( \phi = 0 \) is quite good.

**Absolute Flow Angle Calculations** Figures 7(a), 7(b) and 7(c) show the comparison between the measured and calculated values of the absolute flow angle at the centroid, close to the rotor entry for values of \( \phi \) ranging from 0 to 360 degrees, and at different radius ratios for \( \phi = 70 \) degrees. Here again the trends shown by the analysis are similar to those given by the experimental data, but the agreement between the theoretical and experimental results is rather poor. However, the theory as well as the experiment show quite clearly that a volute casing of circular shape fails to meet the design criteria.

---

**The Design Method**

In spite of the fact that the flow in the volute casing is very complex, the two-dimensional model described in the paper appears to represent the flow reasonably well. Hence it may be used to study the effect of cross-sectional shape on performance and thus optimize the design. The procedure for design is shown in the form of a flow diagram in Fig. 8.
(1) The details of a two dimensional flow model that takes account of friction, area change and the cross-sectional shape of the volute has been described. The agreement between the theoretical results and measurements is not very good but the trends shown by theory are similar to those observed by experiment.

(2) The conformity of a manufactured casing with the design is very difficult to ascertain. The differences between the calculated and measured results could well be due to the fact that there were differences between the actual casing and its manufacturing drawing.

(3) The results show quite clearly that a casing of circular cross-section does not satisfy the design criteria identified in the paper. There is
Fig. 6(b) Comparison between the graphs of calculated and measured values of radial velocity vs the azimuth angle close to the rotor entry.

Fig. 6(c) Comparison between the graphs of calculated and measured values of radial velocity vs radius ratio at zero azimuth angle.

A significant variation of the absolute flow close to the rotor periphery. This would have quite a serious effect on the performance, particularly its efficiency. For best efficiency the absolute flow angle measured from the tangential direction should be constant and close to 17 degrees.

Fig. 7(a) Comparison between the graphs of calculated and measured values of absolute flow angle vs the azimuth angle at the centroid.

Fig. 7(b) Comparison between the graphs of calculated and measured values of absolute flow angle vs the azimuth angle close to rotor entry.
Fig. 7(c) Comparison between the graphs of calculated and measured values of absolute flow angle vs the radius ratio at 70° azimuth angle

ACKNOWLEDGEMENTS

The authors wish to thank British Aerospace for sponsoring this work and to the Directorate of Hatfield Polytechnic for providing the facilities.

REFERENCES

(5) Mager A. Transformation of the compressible turbulent boundary layer Marquardt Aircraft Co. Van Nuys, California.