

Design considerations for the volutes of centrifugal fans and compressors

D Pan¹, A Whitfield^{2*} and M Wilson²

¹Department of Mechanical Engineering, Huainan Mining Institute, Huainan, Anhui Province, People's Republic of China

²Department of Mechanical Engineering, Faculty of Engineering and Design, University of Bath, UK

Abstract: The initial conceptual design of centrifugal fan and compressor volutes is considered and extended to accommodate overhung volute designs often used in process and turbocharger compressors. The initial passage design is then developed through the application of a commercial computational fluid dynamics (CFD) code. Based on the experimental data of a turbocharger compressor volute, three-dimensional, compressible, steady flow computations were carried out for alternative volute designs. Detailed internal flow data in both a conventional and a modified volute design, at both design and off-design flow conditions, are presented. The design investigation showed that enlarging the flow passage area near the tongue region, but without changing the exit-inlet area ratio of the volute, led to an improvement in the internal flow distribution at off-design flow conditions.

Keywords: fans, compressors, volutes, turbochargers

NOTATION

B	width of vaneless diffuser (m)
C	absolute velocity (m/s)
C_r	radial velocity component (m/s)
C_θ	tangential velocity component (m/s)
P	static pressure (Pa)
q_v	flowrate (m ³ /s)
Q_m	mass flowrate (kg/s)
r	radius of volute passage sectional circle (m)
R	radius from impeller axis to any arbitrary position in the volute passage (m)
R_c	radius from impeller axis to centroid of volute passage (m)
α	angle from the radial direction (deg)
θ	volute azimuth angle (deg)

Subscripts

d	volute discharge
3	vaneless diffuser inlet/impeller discharge
4	volute inlet/vaneless diffuser discharge

The MS was received on 8 September 1997 and was accepted after revision for publication on 2 June 1998.

* Corresponding author: Department of Mechanical Engineering, Faculty of Engineering and Design, University of Bath, Bath BA2 7AY, UK.

1 INTRODUCTION

A spiral-shaped volute which collects the flow from the diffuser and passes it to a single discharge pipe is a basic component of centrifugal compressors and fans. Assessment of the design and performance of the collecting volute has received only limited investigation. The volute is usually designed through the application of a one-dimensional analysis assuming a free vortex flow from the volute inlet to the centre of the volute passage [1]. The design objective is to achieve a uniform flow at volute inlet. This is usually attained at the design flowrate only; at off-design conditions the volute is either too small or too large and a pressure distortion develops circumferentially around the volute passage. At low flowrates the pressure increases with azimuth angle while at high flowrates the pressure decreases. These circumferential pressure distortions are transmitted back to the impeller discharge and have been observed at impeller inlet. The pressure distortions reduce the stage performance and have a direct impact on diffuser and impeller flow stability. Yao *et al.* [2] and Qi *et al.* [3] described a volute design technique that allowed a controlled distribution of the volute inlet flow angle, leading to the development of an 'optimum' volute design and improved fan performance.

In recent years three-dimensional viscous computational fluid dynamics (CFD) codes have been used to study the internal flows of volutes [4–6]. The computed results show satisfactory agreements with those measured.

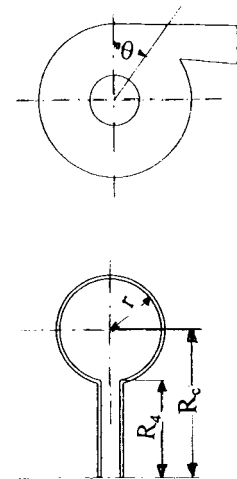
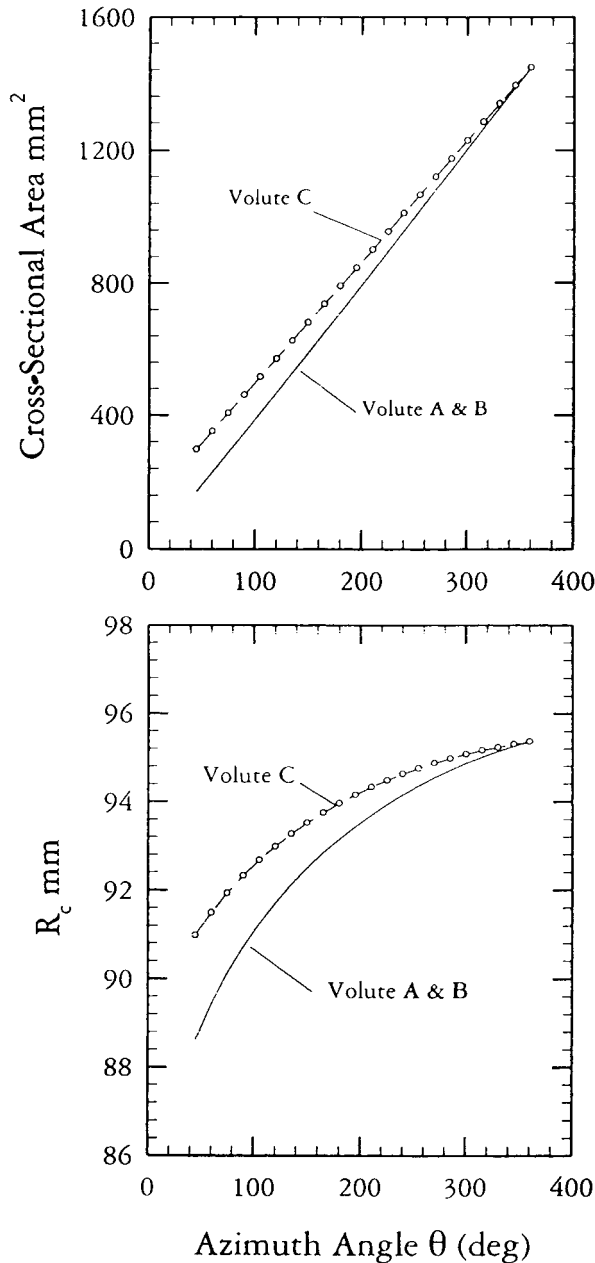
The purpose of the present study was to investigate the possibility of improving volute performance characteristics over a wider range of flowrates by modifying the volute design geometry. The performance of a turbocharger compressor with an overhung volute was fully tested and the pressure distributions around the volute and diffuser were measured in detail to provide fundamental data for CFD analysis. Based on the experimental data, three-dimensional, compressible, steady flow computations were carried out to develop the internal passage design of alternative volutes using a commercial CFD code.

2 VOLUTE DESIGN AND ANALYSIS

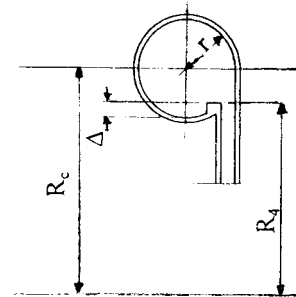
The conventional design theory for the volute assumes that the flow leaving the vaneless diffuser at radius R_4 (see Fig. 1) is uniform in the circumferential direction. With uniform static pressure and no wall friction it then follows from the conservation of the moment of momentum that the velocity throughout the volute is given by

$$RC_\theta = R_4 C_{\theta 4}$$

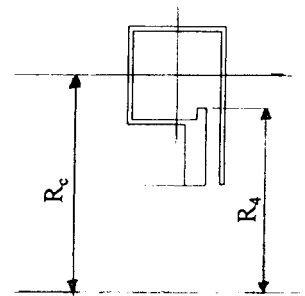
Based on the law of constant angular momentum, different relationships can be derived to calculate the



(a) Symmetrical volute



(b) Volute A



(c) Volute B & C

Fig. 1 Variation of cross-sectional area and radius to section centroid of the volutes

boundary curves of different types of volutes. For a circular cross-section volute it is possible to derive the volume flowrate through any cross-section by integrating the tangential component of velocity, given above. The radius of the cross-section is then

$$r = \sqrt{\frac{\theta R_c q_v}{360\pi C_{\theta 4} R_4} - \left(\frac{\theta q_v}{720\pi C_{\theta 4} R_4}\right)^2} \quad (1)$$

With the assumption of uniform flow conditions the volume flowrate q_v is given by

$$q_v = 2\pi R_4 B_4 C_{r4}$$

and equation (1) can be rewritten as

$$r = \sqrt{\frac{\theta B_4 R_c}{180 \tan \alpha_4} - \left(\frac{\theta B_4}{360 \tan \alpha_4}\right)^2} \quad (2)$$

where $\tan \alpha_4 = C_{\theta 4}/C_{r4}$.

For a symmetrical volute (Fig. 1a),

$$R_c - r = R_4 \quad (3)$$

and equations (2) and (3) can be solved for r and R_c at all azimuth angles. The radius to the outer profile is then given by $R_c + r$.

For many applications, particularly where installation constraints are important, overhung volute designs (Fig. 1b) are employed. In these cases equation (3) is not applicable and the amount of overhang, Δ , must be specified. The magnitude of the overhang was specified as a linear function of azimuth angle. As $\Delta = R_4 - (R_c - r)$ and R_4 is constant, $(R_c - r)$ is then a linear function of azimuth angle such that

$$(R_c - r) = a + b\theta \quad (4)$$

where a and b are constants.

For the turbocharger volute design used in the experimental investigation it was observed that the linear relationship

$$R_c - r = 0.0823 - \frac{8.4}{360 \times 10^3} \theta \quad (5)$$

accurately described the overhang of the volute. Solution of equations (2) and (5) with $B_4 = 5.2$ mm and α_4 assumed to be 65° yielded magnitudes for r and R_c which agreed, to within 1 per cent, with those for the actual volute design.

From equation (2) the flow angle α_4 is a critical parameter in the volute design as it determines both R_c and r . In traditional volute design methods the inlet flow angle α_4 is assumed to be a constant. The design objective is to achieve a uniform flow at discharge from the diffuser. This is usually attained at the design flowrate only; at off-design conditions a pressure distortion develops circumferentially around the volute passage. The measured data of a centrifugal fan volute show that for the actual three-dimensional flow in a volute the inlet

flow angle cannot be maintained constant near the tongue region at off-design flow conditions [6].

Qi *et al.* [3] stated that a fan volute designed with an assumed constant flow angle, α_4 , experimentally developed a flow angle that decreased with increasing azimuth angle. The design procedure was modified, therefore, to specify a flow angle which increased with azimuth angle so that the experimental distribution tended to become more uniform. The design procedure utilized a two-dimensional streamline curvature technique to develop the basic volute profile and dimensions. No experimental results were presented to show that a more uniform distribution in flow angle was achieved; the overall performance of the fans tested did, however, show a significantly improved efficiency and reduced noise levels. The original volute was designed with an inlet flow angle of 84° . As a consequence, small flow areas would be derived through equation (2). Modifying the flow angle so that it increased with azimuth angle from 78 to 88° led to an increase in the volute cross-sectional area over the first 180° of azimuth angle, and particularly near the tongue.

Pan *et al.* [6] carried out detailed internal flow measurements in two symmetrical fan volutes and did not find a significant variation of flow angle with azimuth angle at the design flowrate. The flow angle remained effectively constant with only a small deviation near the volute tongue. The most significant variation of flow angle occurred near the tongue region at off-design flow conditions. For the turbocharger design investigated here the basic design through equations (2) and (5) is based on a constant flow angle of 65° . As, based on the results of Pan *et al.* [6], the flow angle was expected to remain broadly constant with azimuth angle, it was decided not to follow the procedure described by Qi *et al.* [3]; instead the variation of cross-sectional area with azimuth angle was modified directly rather than indirectly through the specification of the flow angle. It

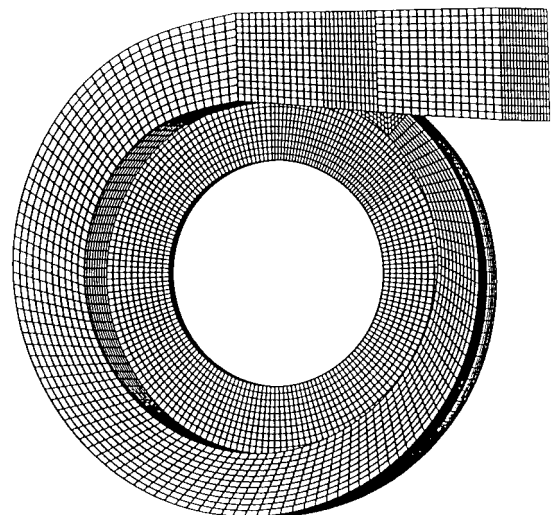


Fig. 2 Three-dimensional view of the discretized flow domain of the volute

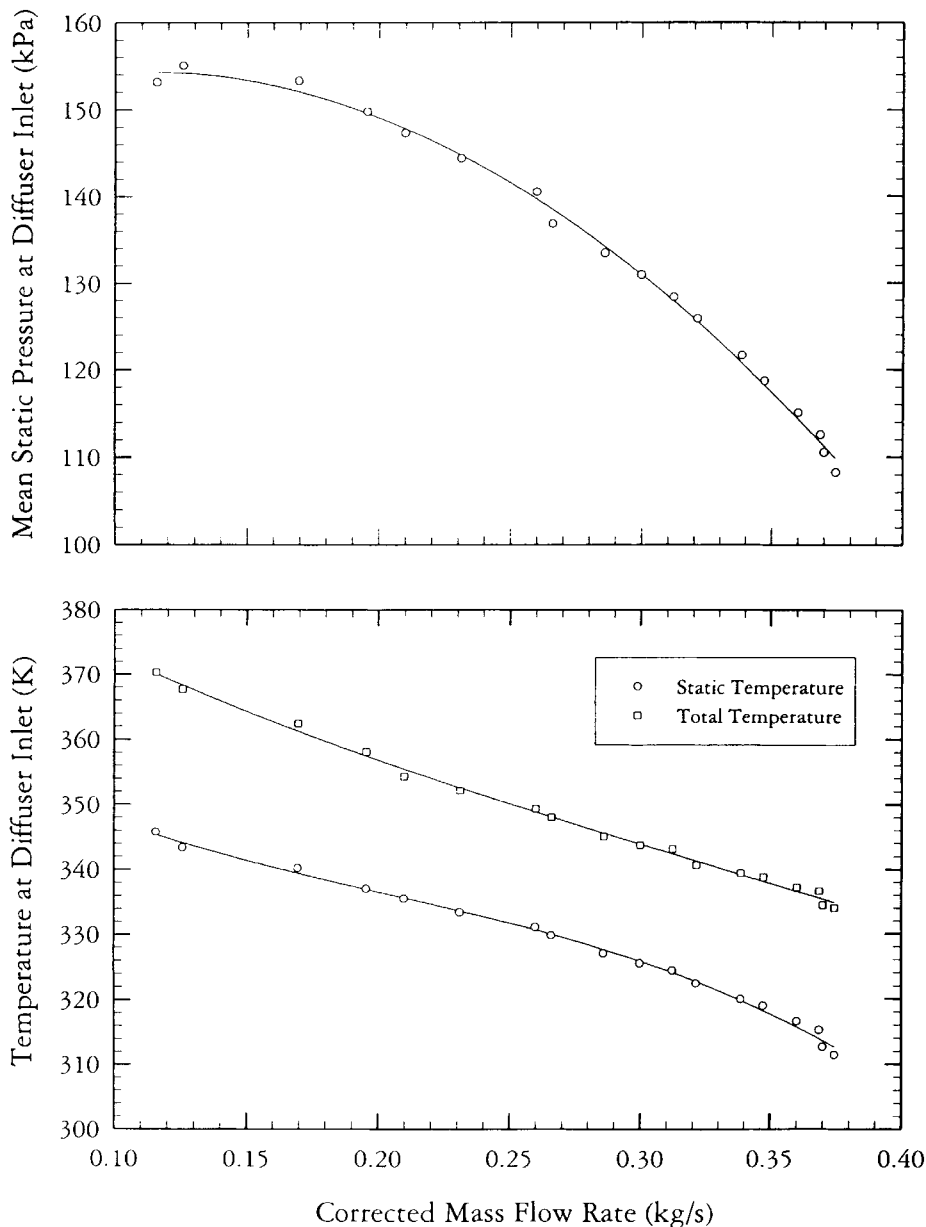


Fig. 3 Measured mean static pressure and temperature at diffuser inlet (impeller discharge)

is believed that the improvements shown by Qi *et al.* are primarily due to the increased flow area over the first 180° of azimuth angle, and particularly near the tongue, rather than improved flow angle distribution. The design procedure adopted, therefore, moved directly from the basic one-dimensional design described above to a three-dimensional analysis so that the effect of specific design modifications could be assessed.

2.1 Three-dimensional computational analysis

The commercially available computational fluid dynamics (CFD) code STAR-CD was used for the numerical investigation. The code solves discretized forms of the Reynolds-averaged Navier–Stokes equations for turbu-

lent flow using the finite volume method. The unstructured grid solution procedure is based on a variant of the SIMPLE pressure correction technique [7]. The standard form of the high Reynolds number $k-\epsilon$ turbulence model was employed for the calculations, with near-wall conditions supplied by the ‘wall-function’ conditions of Launder and Spalding [8].

The computational analysis has been applied to a square cross-section volute (volute B) which has the same cross-sectional area and the same radius to centroid as the experimental volute (volute A). The variation of the cross-sectional area with azimuth angle was then systematically modified and the alternative volute designs analysed. The variation of the cross-sectional area and radius to centroid of volutes A, B and C is shown in Fig. 1. The cross-sectional area for volute C was increased beneath the

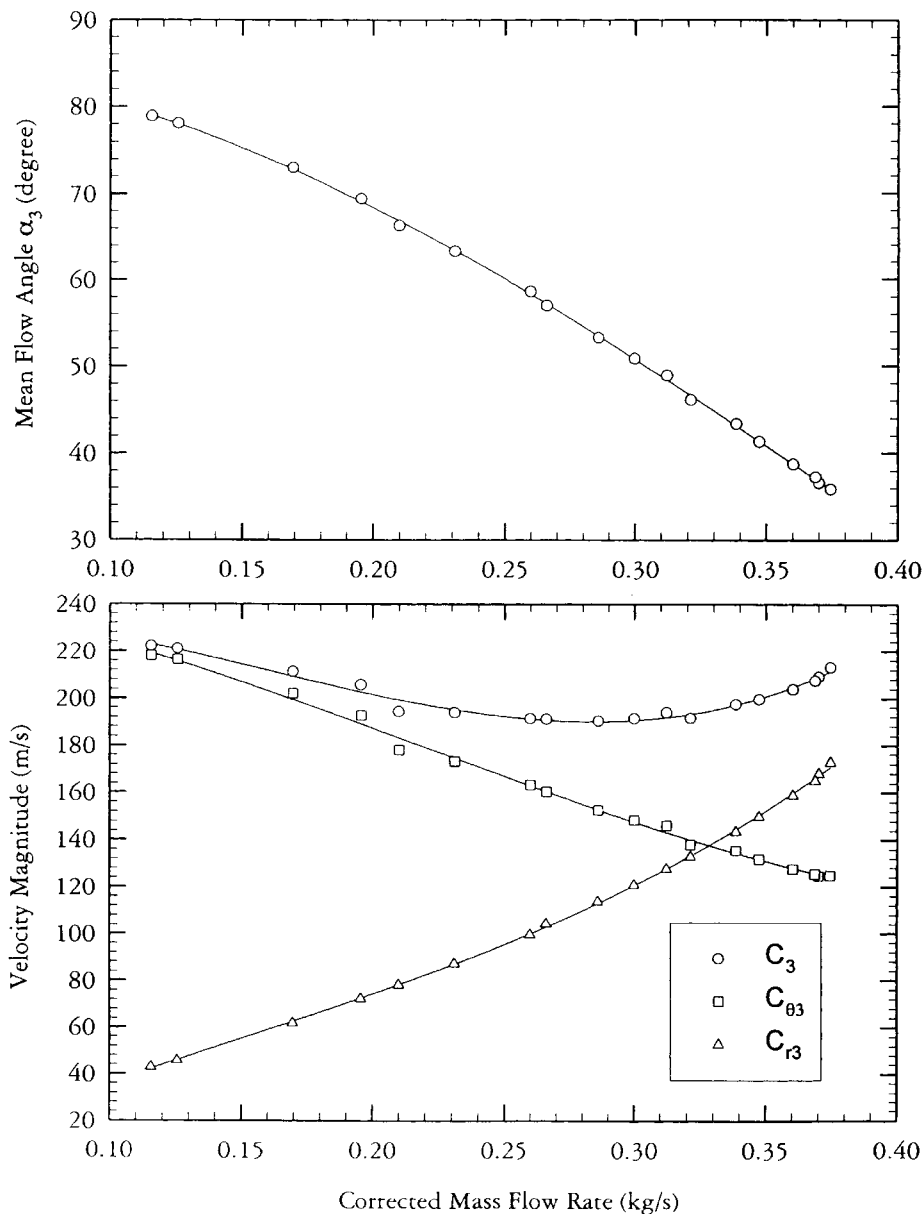


Fig. 4 Mean flow angle and velocities at diffuser inlet (impeller discharge)

tongue by 75 per cent, compared with that for volutes A and B, and then increased linearly with azimuth angle to the same discharge area as that for volutes A and B (see Fig. 1). The exit–inlet area ratio was the same for all three volutes as they all had the same discharge area and the same vaneless diffuser discharge (volute inlet) area.

The grids were generated from known analytical descriptions of the volute geometries, and each volute contained over 45 000 fluid cells in total. Typically, the vaneless diffuser was discretized by 14×7 grid points on each of the 130 cross-sections and the volute passage and exit duct were discretized by 13×15 grid points in 180 cross-sections. Cross-sections were concentrated near the tongue region to obtain a more detailed flow description. The number of cells used was systematically increased until no cell dependence was observed. A

three-dimensional view of the discretized flow domain of a volute is shown in Fig. 2.

The inflow boundary conditions used for the diffuser inlet were velocity, flow direction, static temperature and turbulence intensity. Uniform velocity profiles were prescribed at the diffuser inlet, according to different mass flowrates, by specifying radial and tangential velocity components separately. The measured averaged static pressure was used to specify a diffuser inlet reference pressure at the position of azimuth angle $\theta = 225^\circ$ for each flowrate. The volute outlet was defined as an outflow boundary.

The iterative solution was deemed to be converged when the normalized absolute error over the mesh had reduced to 10^{-5} for each variable. The computation time for each case was typically 24 h on a Silicon Graphics R4600PC workstation with 64Mb memory.

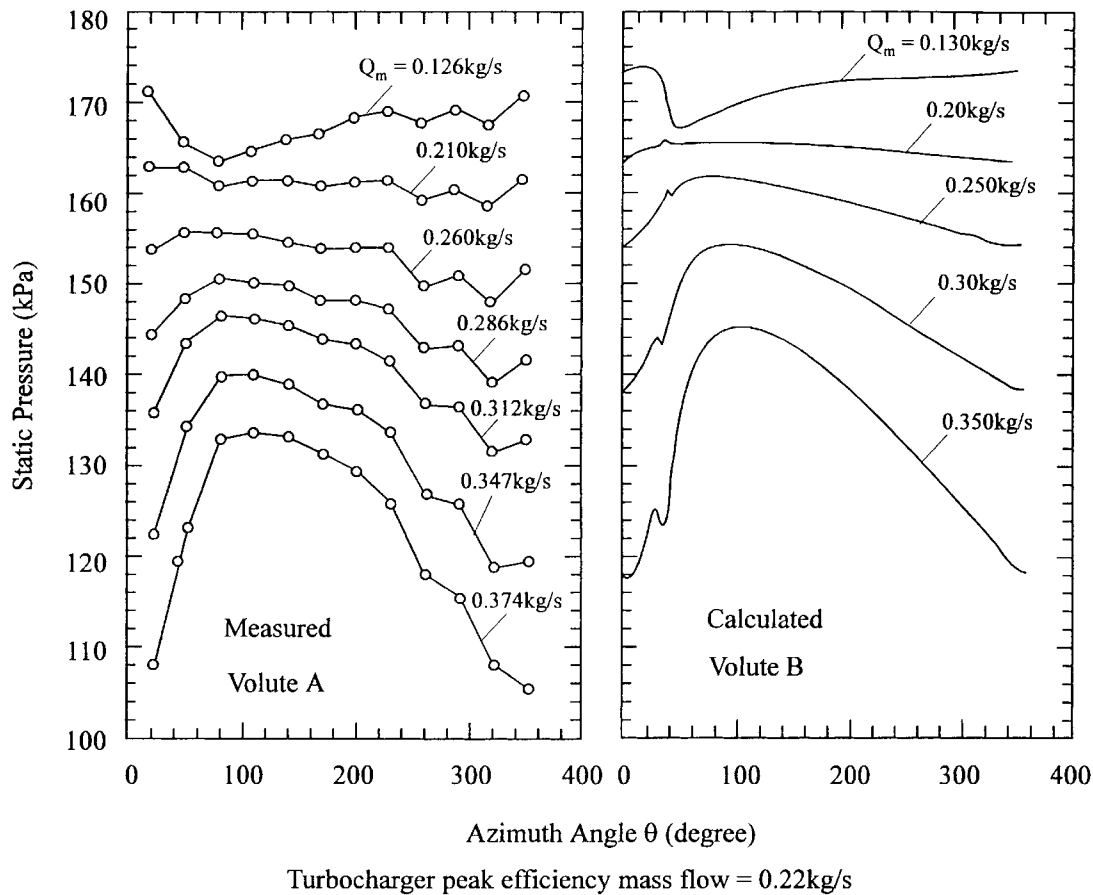


Fig. 5 Comparison of measured and calculated static pressure variations around the discharge of the diffuser

3 EXPERIMENTAL TEST FACILITY

The experimental investigation was carried out with a turbocharger compressor which had a 98 mm diameter impeller with a backward swept blade design and seven full length and seven splitter blades, a parallel walled vaneless diffuser (radius ratio 1.65 and width 5.2 mm) and a collecting volute (volute A).

The basic performance of the compressor was obtained through the measurement of static pressures and total temperatures at inlet and discharge, the mass flowrate being obtained through the application of an inlet nozzle. The diffuser and volute performance was obtained through the measurement of static pressures at inlet to and discharge from the vaneless diffuser together with the discharge duct static pressure. There were 8 pressure tapings around the inlet of the diffuser and 12 around the outlet on the shroud side. All tests were carried out at an impeller speed of 60 000 r/min.

The measured average static pressure and temperature at the diffuser inlet plotted against a corrected mass flowrate are shown in Fig. 3. Based on these measured static pressures, temperatures and mass flowrates, the mean velocity components, flow angle and the density of air at impeller discharge were calculated. Figure 4 shows variations of flow angle and velocity at the impel-

ler discharge against corrected mass flowrate. The data of Figs 3 and 4 were used to specify the diffuser inlet boundary conditions for the CFD calculations.

Due to the small size of the diffuser and volute (diffuser width, $B_4 = 5.2$ mm) no attempt was made to measure the detailed internal flow structure. A large-scale rectangular model of the volute is being constructed at the Science University of Tokyo so that detailed internal flow measurements can be made. In order to investigate the effect of volute design on turbocharger performance it is intended to fabricate alternative volute designs and a square or rectangular cross-section will facilitate this fabrication.

4 PRESENTATION AND DISCUSSION OF RESULTS

The measured static pressure profiles around the inlet of volute A are compared with the numerical results for volute B, at different flowrates, in Fig. 5. It can be seen that the trend of the pressure profile against flowrate is well predicted by the numerical method. Both experimental and numerical data give the same trend of pressure variation with azimuth angle. At low flowrates the static pressure increases with azimuth angle, thereby

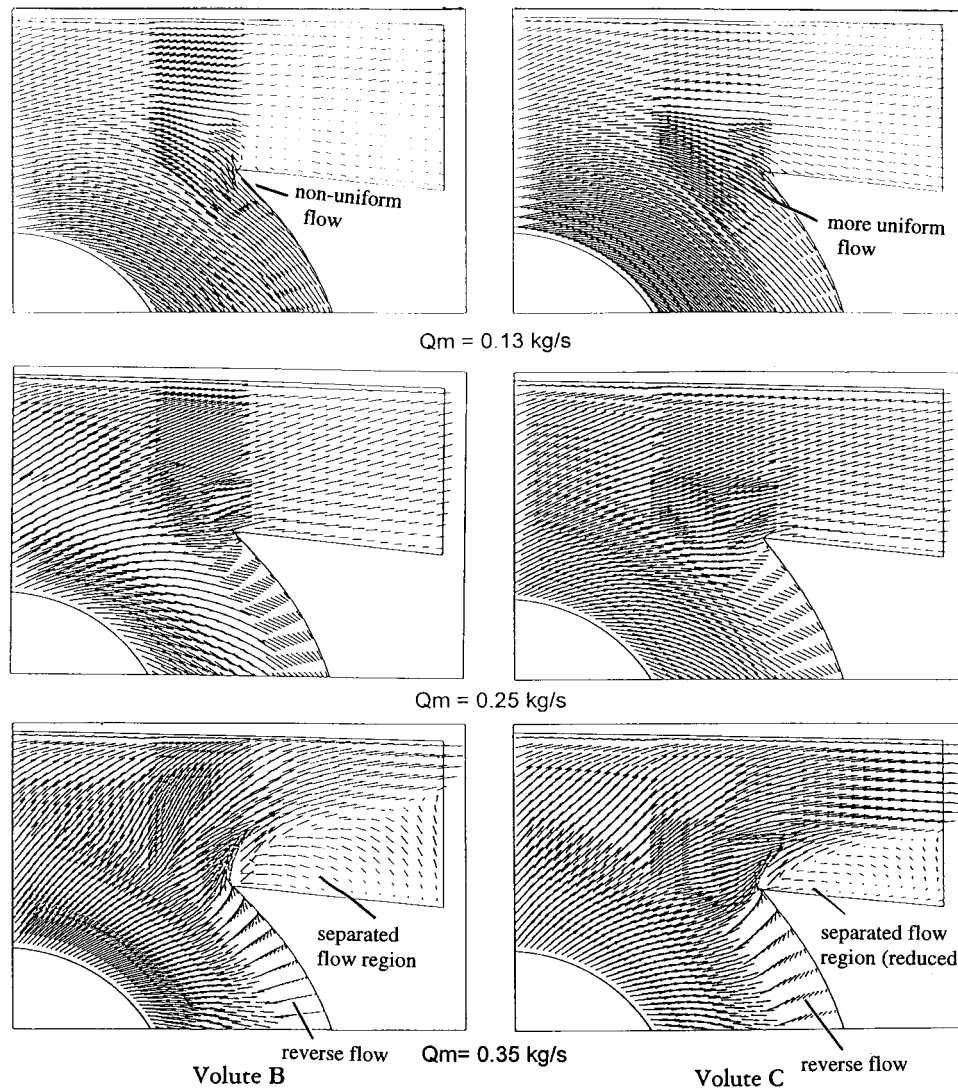


Fig. 6 Calculated velocity vectors near the volute tongue (volute C has a larger flow area under the tongue than volute B)

showing flow deceleration, and at large flowrates the static pressure decreases, showing a flow acceleration. The peak efficiency of the compressor occurred at a flowrate of 0.22 kg/s; this flowrate is close to that which gave a uniform pressure distribution around the volute. The magnitudes of the predicted pressures are always larger than those measured at the same flowrates.

The numerical results revealed the direct impact of the volute designs on the internal flow pattern. Figure 6 shows that, at low flowrates, the small radial and large tangential velocity components result in a negative incidence on to the volute tongue. Some air enters the volute again at the cut-off point and recirculates in the volute, and the flow is distorted by the narrow passage under the tongue. At large flowrates, the large radial velocity component at inlet to the volute results in a high positive incidence on to the tongue and a subsequent flow separation in the exit duct. The sharp circumferential pressure rise (Fig. 5) in the tongue region at large flowrates

pushes the air backwards through the tongue gap. This also results in a flow separation in the exit duct. Similar flow phenomena were observed in a pump volute by flow visualization [9] and in a centrifugal fan volute [10]. The computational results also show that by enlarging the flow passage area under the tongue the internal flow structure of the volute was improved at off-design flow conditions. Figure 6 shows that in volute C at low flowrates the flow under the tongue became more uniform and at large flowrates the size of the separation region in the exit duct was reduced.

The effect of modifying the volute passage design on the flow distribution with azimuth angle is shown in Fig. 7. Calculated variations of flow angle, flow velocity and static pressure along the central line of the volute inlet for different flowrates are shown in Figs 7a, b and c. At low flowrates, because of the recirculation in the volute, the flow angle increases as the flow approaches the tongue and the flow becomes more tangential

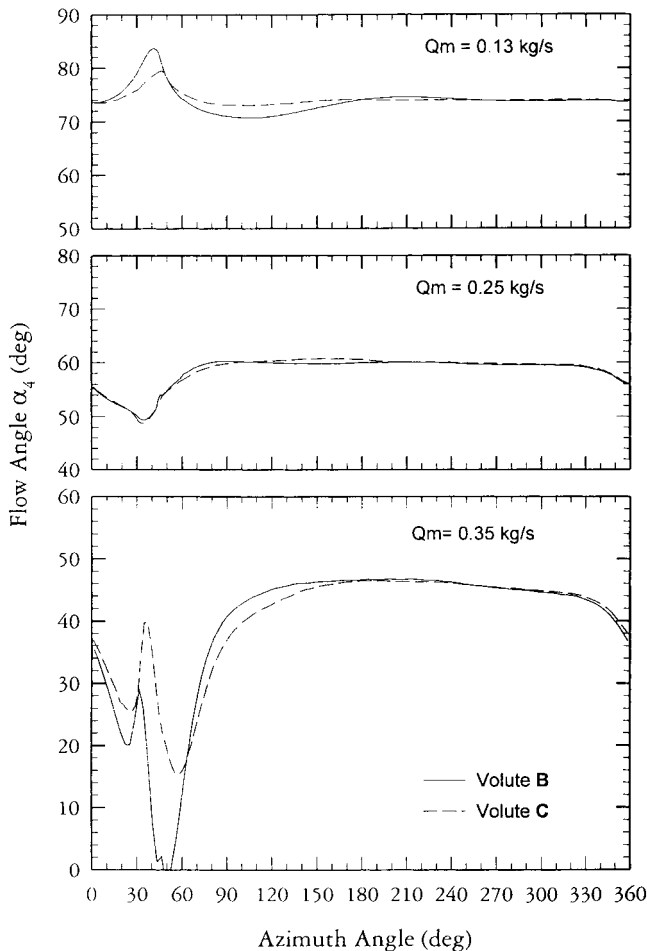


Fig. 7a Calculated flow angle profiles along the central line of the volute inlet

(Fig. 7a). There is a rapid increase in velocity and a sharp pressure drop just after the cut-off point (Figs 7b and c). At large flowrates, because of the separation in the exit duct and the reverse flow under the tongue, the flow angle and the velocity magnitude decrease before the cut-off point and increase after it. The relative variation of both flow angle and velocity magnitude within the tongue region is over 50 per cent for volute B at large flowrates. At the same time there is a sharp increase in static pressure within the tongue region. The predicted flow angle, α_4 , remained constant over a large range of azimuth angle ($90\text{--}360^\circ$), even at off-design flowrates, and it was only in the tongue region that fluctuations occurred. It is believed that the flow distortion and fluctuation caused by the tongue at off-design flow conditions will exert a negative effect on the overall performance of the compressor.

Figure 7 shows clearly that the modified volute design has led to a reduction in the flow distortion within the tongue region at both low and large flowrates. For example, the difference between the peak and minimum magnitudes of pressure, velocity and flow angle along the centre-line of volute C have all reduced by over 50 per cent

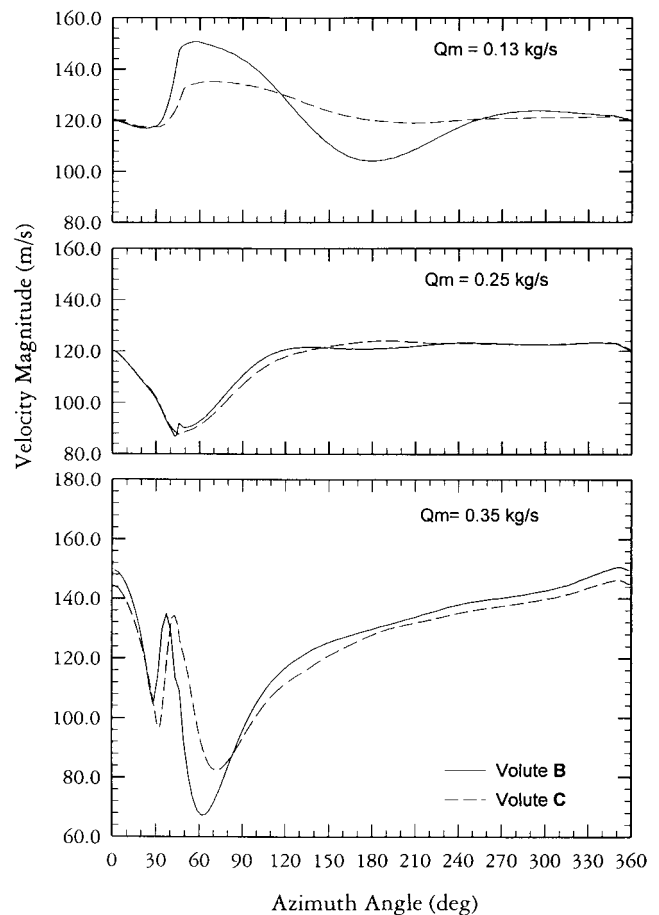


Fig. 7b Calculated flow velocity profiles along the central line of the volute inlet

at low flowrates compared with volute B. At the middle flowrate, the flow structure inside the volute remained unchanged and the flow angle remained constant except in the region of the tongue. At the high flowrate there was a substantial reduction in the variation of both velocity and flow angle in the tongue region.

The basic performance of the volute and diffuser is presented in terms of the averaged static pressure rise across the diffuser and volute in Fig. 8. Both measured and calculated performances of the different volute designs are shown. Compared with the experimental data the pressure rise across the vaneless diffuser was overestimated while the pressure rise across the volutes was underestimated by the numerical method over the entire flow range investigated. At off-design flow conditions the impeller discharge flow is not uniform either in the circumferential or axial directions; the uniform velocity assumption at the diffuser inlet, therefore, has added a favourable factor for pressure rise calculation in the numerical investigations. For the volute the square section design will lead to increased flow losses and a reduced pressure rise across the volute. Despite these quantitative discrepancies the trend of the performance of the volute and diffuser was well predicted by the numerical method. The objective of the design and

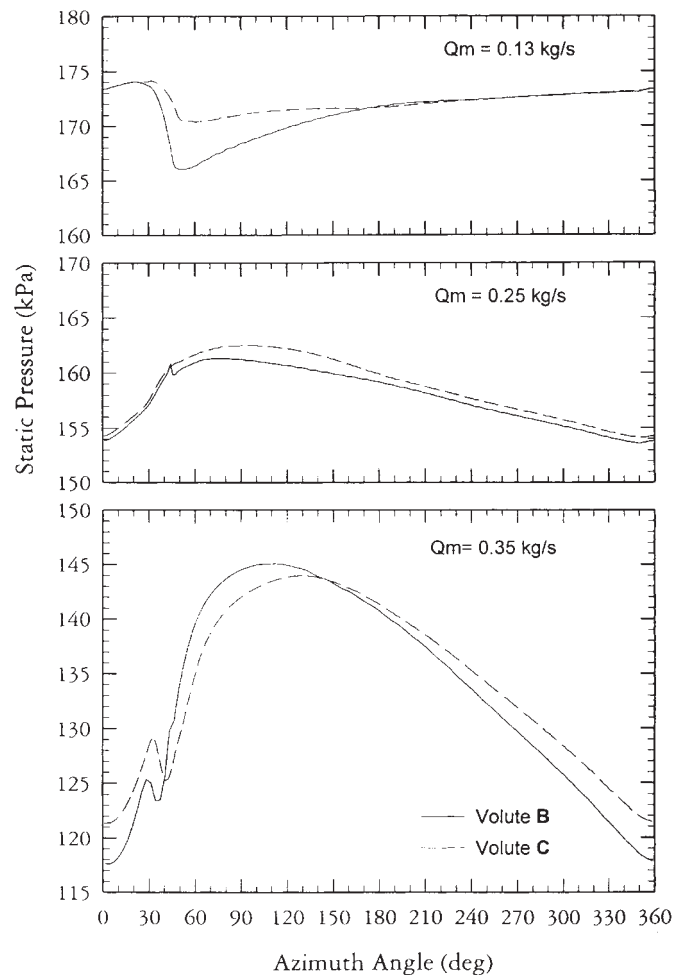


Fig. 7c Calculated static pressure profiles along the central line of the volute inlet

analysis procedure was to identify improved designs, not necessarily to predict accurately the performance of specific components.

The computational results predict that the flow improvement in volute C will lead to an improvement in the vaneless diffuser performance rather than in the volute itself. The adoption of volute C, which is predicted to generate a reduced flow distortion with azimuth angle at off-design flow conditions, leads to an improvement in the predicted pressure rise across the vaneless diffuser.

5 CONCLUSIONS

For the initial design of a volute the one-dimensional free vortex procedure, combined with a linear variation with azimuth angle of the specified overhang, led to a satisfactory design. Based on both the experimental and CFD investigation it was found that the volute inlet flow angle did not vary significantly with azimuth angle except at off-design flowrates and in the vicinity of the tongue. As a consequence, the procedure of specifying a flow angle variation with azimuth angle, as

suggested by Qi *et al.* [3], was unsatisfactory for further modifying the design in detail.

The detailed passage design was developed by systematically modifying the flow area and predicting the internal three-dimensional flow structure using a commercial CFD package. This was found to provide satisfactory prediction of the mean pressure distributions in the diffuser and volute despite the assumption of uniform diffuser inlet flow conditions. The predicted internal flow distributions near the tongue showed satisfactory qualitative agreement with the results of Pan *et al.* [6] with a symmetrical volute design.

The CFD investigation has shown that the internal flow distribution could be improved by enlarging the flow area near the tongue. This led to reduced pressure and flow fluctuations with azimuth angle and a predicted improvement in the performance of the vaneless diffuser upstream of the volute.

Due to the small size turbocharger volute, detailed internal flow measurements were not attempted to provide direct comparison with the predicted results. Detailed internal flow studies are to be carried out on a large-scale model of the volute at the Science University

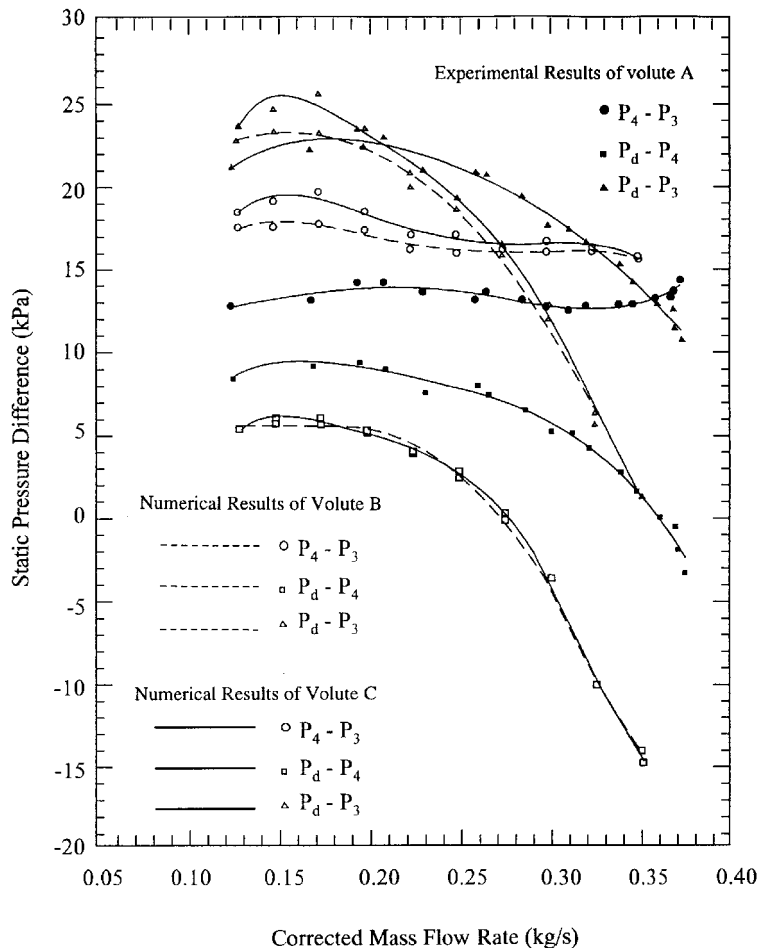


Fig. 8 Comparison of experimental and theoretical static pressure variations through the vaneless diffuser volute interface

of Tokyo and complementary turbocharger volute studies are being planned at the University of Bath.

REFERENCES

- 1 Eck, B. *Fans*, 1973 (Pergamon Press, Oxford).
- 2 Yao, C. F., *et al.* Study on the scroll of the fan with high efficiency and low noise (in Chinese). *Fluid Engng*, 1991, **19**(7), 2–6.
- 3 Qi, D. T., *et al.* A new approach to the design of fan volute profiles. *Proc. Instn Mech. Engrs, Part C, Journal of Mechanical Engineering Science*, 1996, **210**(C3), 287–294.
- 4 Flathers, M. B., *et al.* Aerodynamically induced radial forces in a centrifugal gas compressor—part 2: computational investigation. ASME paper 96-GT-352, 1996.
- 5 Ayder, E., *et al.* Numerical analysis of the three-dimensional swirling flow in centrifugal compressor volutes. *Trans. ASME, J. Turbomachinery*, **116**, 462–468.
- 6 Pan, D., Sakai, T., Whitfield, A. and Wilson, M. A computational and experimental evaluation of the performance of a centrifugal fan volute. In IMechE Conference on *Fan Systems*, November 1997.
- 7 Patankar, S. V. *Numerical Heat Transfer and Fluid Flow*, 1980 (Hemisphere, New York).
- 8 Launder, B. E. and Spalding, D. B. The numerical computation of turbulent flow. *Comput. Meth. Appl. Mech.*, 1974, **3**, 269.
- 9 Elholm, T., *et al.* Experimental study of the swirling flow in the volute of a centrifugal pump. ASME paper 90-GT-49, 1990.
- 10 Sakai, T. Laser sheet flow visualisation in a fan volute. Personal communication.