THE MODELLING OF FLOW IN THE VOLUTE AND VANE
OF A HIGH PRESSURE RADIAL INFLOW TURBINE

M. Lobo and R. L. Elder
Cranfield Institute of Technology
Bedford, United Kingdom

ABSTRACT

The flow in a vaned volute of a radial inflow turbine is analysed using PHOENICS, a very general three-dimensional viscous CFD code based on finite-volume pressure-correction techniques for solving the Navier-Stokes equations. The study involves two physically complementary but mathematically very different problems — flow modelling in the vaneless section of the volute and flow modelling in the vanes. Each of these problems is considered in turn — and each presents characteristic hurdles. Particular attention is paid to grid-generation and the process is carried out alongside the flow computation; the grids being modified in such a way as to facilitate convergence and accuracy.

Numerical results are presented in the form of vector plots for purposes of general description and tables for comparison with experiment. Agreement with experiment is good. However experimental results, for comparison are available only in the vanes. In the vaneless section of the volute, the converged solution depicts some interesting secondary flow features — in regions inaccessible to the current experimental measurements.

1. INTRODUCTION

Flows in radial inflow turbines have been of interest for more than two decades but, because of their relative complexity, they are even today much less well understood than axial turbines and in the 1970s, when both experimental and computer facilities were considerably less sophisticated than they are today, it was natural for researchers to concentrate their efforts on axial turbines which were in much wider use. But radial turbines possess certain advantages over small axial turbines such as ease of manufacture and compactness of installation; in the case of very small units, they have also been found to perform relatively better perhaps because of their reduced sensitivity to overtip leakage loss. With improvements in experimental and computer facilities during the 1980s, much more attention has been given to radial inflow turbines - but there is still much more to be learnt about the nature of the internal flow particularly where coupled with a volute.

Perhaps the most practicable method of analysing the flow through such radial inflow turbines is to deal separately with the flow through the volute casing, the nozzle guide vanes (if present), and the rotor. Evidently the flow through each of these components will influence the flow through the others and, for purposes of mathematical modelling, these influences may be reflected by inlet or outlet boundary conditions. Of course it would be ideal if the solution procedure could be carried forward in all the regions simultaneously but, as is the case with most current problems, the practical approach cannot come up to the ideal.

Experimental investigations into radial turbines date back to the 1960s, for example, Hiett and Johnston (1963), Benson and Scrimshaw (1965), Barnard and Benson (1966), Nusbaum and Kofskey (1969). By contrast, computational work only commenced in the late 1970s, for example, Hamed et al (1977, 1978) who restricted their considerations to the velocity potential equation. In the past decade, experimental technology has advanced to make use of measuring instruments such as the laser velocimeter; recent papers include Benisek and Strubel (1990), and Kitson et al (1990). At the same time, computer facilities have also advanced to the point of being able to handle the three-dimensional Euler and Navier-Stokes equations. Recent papers dealing with flow modelling in radial inflow turbines include that of Lymberopoulos et al (1988) and Zangeneh-Kazemi et al (1988). The former solves the Euler equations for the flow through a vanesless volute casing, the latter solves the Navier-Stokes equations for the flow through a rotor.

In terms of mathematical sophistication, the algorithm of Zangeneh-Kazemi (et al) is fairly advanced being the product of several years of research into time-marching techniques by workers at the Whittle Laboratory, Cambridge, commencing with the classic paper of Denton (1974). By contrast the algorithm of Lymberopoulos (et al) is relatively simple: only the two-dimensional Euler equations are solved, though allowance is made for varying streamtube thickness in the third dimension (hence the algorithm may be called quasi-3D). Having said this, it should be added that the flow in a volute is considerably more difficult to analyse numerically than the flow in a rotor. From a geometric standpoint, an impeller is akin to various other turbomachinery configurations - and today there exist CFD codes, notably that of Dawes (1988), which can be applied to a wide variety of such geometries including the rotor of a radial inflow turbine. However the volute of a radial inflow turbine has a completely different topological structure. Even from a two-dimensional standpoint, the numerical analyst is faced with several problems, not least of which is the generation of a suitable grid. There is no distinct outlet boundary — indeed not all the flow which enters at inlet escapes directly through the outlet, some of it makes a complete circulation, re-enters and mixes with the inlet mainstream flow.

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
Cincinnati, Ohio — May 24–27, 1993

Copyright © 1993 by ASME
The flow in this section of the volute - known as the 'tongue' - is highly non-uniform; it is difficult to measure experimentally - and equally difficult to compute numerically. And, of course, in the real three-dimensional standpoint, all these difficulties multiply.

The present paper describes some of the research into radial inflow turbine volutes carried out by the Turbomachinery Group at the Cranfield Institute of Technology during the past few years. It is restricted solely to numerical modelling, but a companion paper on experimental work, Hamid and Elder (1992), may appear separately. The configuration under consideration is the turbine volute depicted cross-sectionally in Fig 1.

Although researchers at Cranfield have developed several Navier-Stokes codes – based both on pressure-correction techniques, for example Lapworth (1987, 1988), Tourlidakis (1991, 1992), and on time-marching techniques, for example Tsay (1989), Cheng (1992) – the code used here is PHOENICS. The reason for this is that the research was carried out as part of a project, one of whose objectives was to test the ability of PHOENICS to deal with the complex flows in a turbocharger. The CFD code PHOENICS (an acronym for Parabolic, Hyperbolic or Elliptic Numerical Integration Code Series) is based on a solution technique now called 'pressure-correction', originally developed by Patankar and Spalding (1972) and expounded in some detail in the book by Patankar (1980). The basic ideas behind the pressure-correction technique apply to all kinds of problems in CFD and hold regardless of whether the governing equations are parabolic, hyperbolic or elliptic. PHOENICS, then, is a very general code which can take into account flows in one, two or three dimensions, which can be treated in Cartesian, polar or more general curvilinear coordinates and which can handle problems involving compressibility, turbulence, heat conduction, time-dependence, multiple phases etc.

The arrangement of the present paper in sections is quite simple and consists of two main sections - 2 and 3 - dealing with the flow modelling in the volute (Part I) and in the nozzle guide vanes (Part II) respectively. Each of these sections is divided into analogous subsections entitled "Grid Generation" and "Results and Discussion". Finally Section 5 concludes the paper with a summary and overall comments.

2. PART I : FLOW MODELLING IN THE VOLUTE

A. Grid Generation

The turbine volute under consideration in the present paper forms part of an aircraft turbocharger capable of providing a compressor ratio of 6:1. There are 17 nozzle guide vanes and 15 blades. The mechanical design is similar to that of Benway (1987). Fig 1 depicts a cross-section of the turbine component of the turbocharger. A 'head-on' view of the nozzle guide vanes can be seen at the top inner corner of each cross-section.

The operational conditions for the turbine and the rig are discussed in the companion paper by Hamid and Elder (1993) and reviewed below:

\[ \text{N/rev} = 38,600 \text{ rpm (80% of the design speed)} \]

Turbine inlet temperature = 400°C

Turbine expansion ratio = 2.2

Mass flow \(\dot{M}/\dot{\theta}/\theta = 0.276 \text{ Kg/s}\)

These operating conditions were selected because the laser anemometry measurements to be taken limited the inlet temperature and thus the matching with the load compressor. In these circumstances, an expansion ratio was selected which gave the design turbine rotor incidence.

For the purpose of modelling the flow in the volute, the vanes are first assumed to be absent so that the cross-section depicted by Fig 1 would now appear as in Fig 2. Grid generation in the cross-sections - in the form of algebraic curves - is best carried out by dividing the cross-section into five regions as in Fig 3. Each region can now be fitted with a grid consisting of two families of curves. Of course care must be taken that the grid in each region merges smoothly with the grids of neighbouring regions.

Region A has the simplest grid, the elements of which are all rectangles. The elements of Region B are bounded by two families of orthogonal circles. Orthogonality of the grid is maintained in Region C where the curves of one family are ellipses and those of the other are governed by a transcendental equation. These grids are shown in Fig 4.

Region D has a semicircular outline near the inlet to the volute but gradually reduces to a semi-ellipse. It is possible to construct an orthogonal grid of the type shown in Fig 5a. However for reasons which will shortly be explained, this grid possesses certain inherent drawbacks and a grid structure such as shown in Fig 5b is to be preferred despite the existence of a few elements which are highly non-orthogonal (these elements occur only in corners and because the non-orthogonality is not in the plane of mainstream flow, the accuracy is not significantly affected).

Finally there is the problem of grid generation in Section E. The rapidly changing tangent of the outline can only be represented by a high order polynomial. An orthogonal grid is impracticable without a very high degree of refinement: this may be seen in Fig 6a where the grid density matches that of the neighbouring regions but which is obviously unacceptable (note that because the high order curves are replaced by straight lines in each individual cell, even the orthogonality may be spoilt in the coarse regions of the grid). On the other hand an evenly spaced grid (Fig 6b) would be highly non-orthogonal at some points. A compromise seems desirable as in Fig 6c. Here although non-orthogonality exists, it is not too severe and the grid spacing is still fairly uniform.

The complete cross-sectional grid formed by the union of the grids of Regions A, B, C, D and E, given by Figs 4a, 4b, 4c, 5b and 6c, is depicted in Fig 7. If the non-orthogonal grid of Fig 5b is replaced by the orthogonal grid of Fig 5a, the complete grid would appear as in Fig 8.

However this grid is not compatible with the operational structure of PHOENICS because one family of grid lines folds back over itself making it impossible to apply the correct boundary condition on the grid line which folds back into itself. Therefore further grid developments should be based on the grid of Fig 7.

For flow details near the walls, the grid must be refined as in Fig 9a. This applies to cross-section A of Fig 2. Likewise cross-section B may be discretized as in Fig 9b.

Having constructed the cross-sectional grids, the next step is to generate the full 3D grid for the volute - with the vanes assumed to be absent. At this point it is appropriate to look at the numbers which may be involved if one were to attempt to generate an accurate grid for the volute with all the vanes present. Fig 10 depicts the 17 vanes enmeshed in a grid which can be superimposed on the cross-sectional grids of Fig 9 (the tangential grid lines of Fig 10 are orthogonal to the two families of grid lines in Fig 4a). Clearly the grid of Fig 10 is quite inadequate for a detailed study of the flow in the 17 vane passages and yet, in combination with the (36 x 20) cross-sectional grid of Fig 9, it would lead to a three-dimensional grid of 170 x 720 = 122,400 points.

Because a body-fitted CFD run requires a considerable degree of storage for each grid cell (examples of quantities which require storage are coordinate transformation parameters, areas of grid-cell faces, volumes of grid cells, flow variables such as density, velocity and pressure, thermodynamic variables such as energy, enthalpy and temperature, nonlinear coefficients of these variables which may appear in the governing equations, arrays containing temporary approximations to these...
variables which may be required in the solution process, mass fluxes through each cell face, viscous stress terms, quantities associated with turbulence models, corrections to be added at each iterative step), a grid consisting of 122,400 points would require the use of a supercomputer. With the vanes assumed to be absent however, a reasonably accurate solution can be obtained using a much coarser grid. A grid of 64 x 25 x 10 was eventually chosen as optimum - with 64 cells from inlet to the point where the circulating flow rejoins the mainstream - 25 cells from the volute casing to the rotor inlet - and 10 cells in the third dimension (wall to wall in the volute interior and hub to tip in the region of the vanes). In the region where the circulating flow rejoins the mainstream (the 'tongue'), the grid cells on either side must be made to 'fit' each other.

The flow patterns obtained with the use of this grid will now be discussed.

**B. Results and Discussion**

Having generated the grid for the volute, the next problem is to impose the appropriate boundary conditions. Obviously the mass flow must be specified at inlet and standard log law wall functions can be imposed at all wall boundaries. At the 'tongue', where the flow having completed a full circle rejoins the mainstream, it is necessary to impose a piece of Fortran coding (into the GROUND subprogram of PHOENICS) which ensures that all flow properties (pressure, density, velocity components u,v,w, and turbulence parameters k,e) are correctly linked. Obviously the mass flow entering the vanes in this cross-section appears to come from the flowfield upstream of the vanes suggesting that little mass flow is actually entering the vanes in this area.

Boundary conditions must also be imposed at the outlet of the volute where the swirling flow enters the rotor. This is also not a straightforward problem. Ideally the influence of the rotor should be reflected in the solution; however, as mentioned in the introduction, a combined study of the flow in the volute and rotor is not practicable. For the present study, a uniform pressure condition is imposed at the exit boundary, its value being selected so as to match the specified total pressure at inlet. In practice this means doing a series of runs with varying values of the downstream static pressure; for each such run the total pressure upstream can be calculated from the formula $P_s = P_p + \frac{1}{2} \rho V^2$, where $P_p$, $\rho$ and $V$ represent the static pressure, density and velocity respectively at inlet. A downstream static pressure of 1.2 bars was found to yield the required inlet total pressure of 1.94 bars. Some vector plots of the solution at the termination of the 200th sweep are displayed in Figs 11 and 12. Fig 11a is a vector plot in the plane of mainstream flow whose intersection with the cross-section of Fig 3 is marked by the upper boundary of Region D. Fig 11b is a vector plot in the plane of mainstream flow whose intersection with Fig 3 bisects Region A horizontally. Note that in Fig 11b the vectors in the inlet pipe are not shown because the inlet pipe is below the level of this plane.

The vectors portrayed in Fig 11a are not of much interest but those of Fig 11b deserve some comment. The radial components of the vectors increase from periphery to rotor inlet - this is especially marked in the region about 270° ahead of the tongue. However the results in the rotor inlet region cannot be regarded as physically reliable because the presence of the vanes has been ignored. On the other hand the velocity field represented by the outer vectors is certainly of importance because they will form the basis of the flow modelling problem through the vanes. It is clear that the primary component of the velocity vectors in this region is tangential. On average the radial velocity components tend to be a whole order of magnitude less than the tangential components. However even these small velocities are of significance as they account for the mass flow entering the vanes.

Some vector plots of the solution at the termination of the 200th sweep are displayed in Figs 11 and 12. Fig 11a is a vector plot in the plane of mainstream flow whose intersection with the cross-section of Fig 3 is marked by the upper boundary of Region D. Fig 11b is a vector plot in the plane of mainstream flow whose intersection with Fig 3 bisects Region A horizontally. Note that in Fig 11b the vectors in the inlet pipe are not shown because the inlet pipe is below the level of this plane.

The vectors portrayed in Fig 11a are not of much interest but those of Fig 11b deserve some comment. The radial components of the vectors increase from periphery to rotor inlet - this is especially marked in the region about 270° ahead of the tongue. However the results in the rotor inlet region cannot be regarded as physically reliable because the presence of the vanes has been ignored. On the other hand the velocity field represented by the outer vectors is certainly of importance because they will form the basis of the flow modelling problem through the vanes. It is clear that the primary component of the velocity vectors in this region is tangential. On average the radial velocity components tend to be a whole order of magnitude less than the tangential components. However even these small velocities are of significance as they account for the mass flow entering the vanes.

**Fig 12** depicts secondary flow velocity vectors at different cross-sections of the volute. It is of interest to observe how the pattern of secondary flow changes at different cross-sectional planes around the volute. Some notes follow ($\theta$ is the angle made by the cross-section with the point where the inlet pipe joins the volute - taken anticlockwise).

- For $\theta = 0^\circ$ (Fig 12a), this is the region where the uniform flow in the inlet pipe undergoes a transformation to vortex flow and the flow which has swept through a complete circumference re-enters and mixes with the mainstream. This cross-section is marked by circulating flow just upstream of the vanes suggesting that little mass flow is actually entering the vanes in this area.

**Intersection results** show that downstream of the leading edge the circulation reduces in size while a second circulating zone appears at the SW part of the cross-section. In between the two zones of circulation there is a region of relatively large vectors which appears to form the major constituent of the mass flow entering the vanes in this cross-section - just before and under the leading edge. The anticlockwise circulating zone at the SW increases in diameter whereas the flow near the leading edge of the vanes begins to form a steadier pattern. The mass flow entering the vanes in this cross-section appears to come from the flowfield near the casing which is outside the circulating zone.

- For $\theta = 90^\circ$ (Fig 12b). The centre of the circulating zone has shifted upward so that part of the zone extends to the region before the leading edge of the vanes. This creates a region of low mass flow at the hub and high mass flow at the tip.

Further intersection results show that the centre of the circulating zone shifts slightly westwards (i.e. radially inwards). As a result much of the flow which was participating in the circulation at $\theta = 90^\circ$ now tends to escape at a tangent near the leading edge of the vanes and moves into the passage between the vanes. Thus the mass flow entering
vanes has significantly increased. The circulating zone then appears to be compressed to a somewhat elliptical region in the part of the volute directly below the vanes. Outside this region the flow appears to enter the vanes at a fairly steady rate though the vectors are still larger at the tip.

\[ \theta = 180^\circ \] (Fig 12c). This is the cross-section diametrically opposite the tongue. The circulating zone has now more-or-less disappeared. There is a general trend in the direction of the vanes. Cross-sectional streamlines in the inner part of the volute tend to converge near the hub-leading-edge junction resulting in an increase in size of velocity vectors in that area. From this point onwards, vectors begin to concentrate at the hub-leading-edge junction.

Further results show that the circulating region at the inner half of the cross-section completely disappeared and all vectors head straight for the hub-leading-edge junction where they increase in size. On the other hand the vectors at the outer half reduce in size and a small circulatory zone shows signs of emerging though it never really builds up. Another region of reduced velocities is that just downstream of the hub-leading-edge junction. This is due to the large velocity vectors at the junction itself which are directed upwards rather than inwards. Further downstream the overall pattern is essentially unchanged but the vectors at the hub-leading-edge junction become so large that they are comparable to the mainstream tangential components (perpendicular to the plane of the paper). Their presence causes an area of recirculation just downstream of the junction.

These secondary flow patterns are presented here mainly for general interest; no experimental comparisons are available, and for the purpose of flow modelling through the vanes, it is only the region directly upstream of the leading edge that is of importance. However the fact that the secondary flow patterns are clearly dependent on the position of the cross-section is of significance and it is clear that the mass flow entering each pair of vanes is not constant but increases from near zero at the tongue (\[ \theta = 0^\circ \]) to a maximum at about \[ \theta = 270^\circ \] after which it starts to decrease. This can also be observed from Fig 11b.

The experimental measurements in the present instance were taken in the vane passages corresponding approximately to the cross-section \[ \theta = 120^\circ \] where the secondary flow pattern indicates a steady mass inflow into the vanes (Fig 12e). For the flow modelling problem in this region, fairly precise inlet boundary conditions are imposed by interpolation from the converged velocity values upstream of the leading edge. Details of the flow modelling through the vanes will be given in the next section.

3. PART II: FLOW MODELLING IN THE NOZZLE GUIDE VANES

A. Grid Generation

Unlike the vane passages of the radial inflow turbine have already been depicted in Fig 10.

Unlike the modelling problem in the volute, the flow through the vanes can be studied to a reasonable degree of accuracy in two dimensions as the vane geometry does not change in the third dimension. It is true that the influence of the volute will effect some change in the flow features in the third dimension but this is very slight. A detailed examination of the vector plots of Fig 12 indicates that differences do exist but these are merely secondary flows and are, in general, a whole order of magnitude lower than the mainstream flow. Therefore, at least in the first instance, a two-dimensional study of the flow through the vanes ought to produce some useful results.

It is possible to generate an H-type grid between any pair of vanes by the straightforward process of connecting points with equal radii uniformly (Fig 13). However this grid has a major weakness - its streamwise gridlines very nearly reverse in direction at the leading edge of the pressure surface (marked by the point Q in the figure). A similar (though less severe) problem arises downstream of the trailing edge. Obviously such a grid would give rise to serious modelling difficulties in the awkward regions and the errors generated would eventually propagate over the rest of the flowfield. Therefore this is not a practical grid for modelling work.

One method of dealing with this difficulty, which involves only a relatively small amount of grid restructuring, is to ignore the region directly upstream of the leading edge so that the grid takes the form depicted in Fig 14. Here the change in direction of the streamwise grid lines at the leading edge is greatly reduced and is unlikely to cause major inaccuracy. A similar procedure may be followed downstream of the trailing edge. The drawback to this approach is that the boundaries of the domain upstream of the leading edge (AB and A'B') and downstream of the trailing edge (CD and C'D') are no longer analogous in their positions with respect to the vanes and it would not be strictly correct to apply the periodic boundary conditions. To rectify this, a section of grid upstream of the blunt leading edge may now be added as in Fig 15. Note that this is quite different (topologically) from the grid structure of Fig 13 because the grid lines upstream of the blunt leading edge now come to a halt at this edge so that no awkward change in direction of streamwise grid lines is involved. The situation may be pictured more clearly in Fig 16 where the grid lines upstream of the blunt leading edge may be imagined to continue into the solid body of the vane eventually to reappear downstream of the trailing edge. When executing the code, instructions should be provided to set the cells occupying the vane to 'zero porosity', that is, no flow can enter this region. In effect this group of cells will be treated as a solid body.

Although the grid of Fig 15 appears reasonable, the attempt to model the flow proved considerably more difficult than anticipated. Several numerical trials were carried out with changes in the relaxation factors and other control parameters but invariably the iterations were prone to divergence. Eventually the authors were forced to the conclusion that the skewness of the grid was affecting not just the accuracy but also the convergence potential. The process of grid generation was started afresh. The new grid - depicted in Fig 17 - is not completely orthogonal but is nearly so. The main area where the grid departs from orthogonality is the region upstream of the blunt leading edge but this is not severe enough to give rise to convergence difficulties.

A few comments on the grid should be made before proceeding to discuss results. Though it looks attractive, it is not without disadvantages. Consider Fig 18 (which depicts the same grid with the boundary broken up into segments). It may be assumed that AB and AB' are sufficiently far upstream of the vanes to be uninfluenced by their presence. Conditions here may therefore be imposed from the solution of the volute model. However BC and B'C' are directly upstream of the leading edges; the presence of the vanes will certainly affect the flow in this region and the imposition of boundary conditions from the volute solution would be of dubious accuracy. The answer is, of course, to impose periodic boundary conditions (this may be subject to slight inaccuracy because of the change in flow pattern around the volute but it may be assumed that this change is small in the 10° angular difference between a pair of vanes). But because the corresponding grid points in BC and B'C' do not fall on the same transverse grid line, the imposition of periodic boundary conditions is far from being a straightforward process. A similar difficulty arises in the case of the downstream periodic boundary conditions at DE and D'E'. This is the primary disadvantage of the orthogonal grid structure when compared to the skewed grid of Fig 15.
The technique employed to deal with this problem is to artificially 'link' the periodic boundaries by inserting coding sequences in the GROUND subprogram of PHOENICS. In order to implement the method, it is necessary to ensure that the solution procedure marches from one vane to the other and not from inlet to outlet as is customary. Consider Fig 19. A row of cells has been added below the periodic boundaries BC and DE (for reasons of consistency required by PHOENICS these rows must be extended from inlet to outlet). The coding sequences in GROUND set the velocity components (and other flow variables) in the cells below B'C' and D'E' to those in the corresponding cells below BC and DE respectively. Having imposed the other boundary conditions (inlet and outlet), the solution procedure then marches from the lower surface B'F' to the upper surface A'E'. Finally the periodic boundary conditions on the cells above BC and DE may be set from the values which have just been computed at the corresponding cells above B'C' and D'E'. The overall procedure is similar to the process of linking the flow at the tongue of the volute but rendered more complicated by the fact that the corresponding grid points do not fall on the same transverse grid lines. The additional patches below and above the periodic boundaries are necessary to ensure that boundary information is not merely exchanged (as would be the case if the flow at B'C' is set equal to the flow at BC; and later the flow at BC is set equal to the flow at B'C').

We conclude this section on grid generation by a look at the complete grid (Fig 20). The row of cells above the periodic boundary BC continues into the upper vane and emerges as the row of cells above the periodic boundary DE. The three rows of cells above the periodic boundary B'C' continue into the lower vane and emerge as the three rows of cells above the periodic boundary D'E'. All cells in the vanes are set to zero porosity.

B. Results and Discussion

We begin with a summary of the experimental results. Only those results which are of relevance for the purpose of comparison with the flow modelling are given here. Further details of the experimental investigations and results may be found in a complementary paper, Hamid and Elder (1993).

Fig 21 depicts the position of the points L1 ... L13, where experimental measurements were taken in the plane of mainstream flow (in essence this is the same as Fig 10 but viewed from the opposite side). The polar coordinates of the points of experimental measurement have been converted into the body-fitted coordinates of Fig 17 and the grid cells which most closely correspond to the points of measurement are shown in Fig 22.

A vector plot of the experimental measurements in a plane midway between the hub and tip of the vanes is depicted in Fig 23. Hamid and Elder (1993) depict vector plots on other planes - halfway between hub and centre - and halfway between centre and tip. These plots are very similar to that of Fig 23 indicating that there is no significant change in flow properties in the third dimension (at least this may be assumed to be true in the section of the volute where the experimental measurements were taken). This serves to confirm what was said in the previous subsection - that the flow through the vanes studied can be modelled to a reasonable degree of accuracy in two dimensions. For the sake of completeness, however, a three-dimensional extension of the present work was also carried out. As expected, the vector plots generated at the different planes were very similar to each other as well as to the results of the two-dimensional analysis.

Before proceeding to compare the two-dimensional modelling results with those of the experiments, some comments should be made about the nature of the modelling and the numerous parameters which may affect the final result. Even if the entire investigation is restricted to the grid of Fig 17, there is more than one way in which the inlet and outlet boundary conditions may be implemented - there is more than one way in which the flow variables at the periodic boundaries may be linked - and there are numerous permutations of relaxation factors and other solution control switches.

Consider first the problem of imposing inlet and outlet boundary conditions. The first decision to be made is whether mass flow should be specified at inlet or outlet. Note that the mass flow entering the inlet boundary AB', in Fig 18, will differ from the mass flow leaving the outlet boundary EF' because much of the flow will actually leave the computational domain through the boundary AB and will not therefore enter the passage through the vanes. It may seem more natural to specify mass flow and velocity components at inlet (to be obtained from the Part I analysis) and perhaps a constant pressure boundary condition at outlet. But it is equally plausible to specify the mass flow downstream. One way by which this may be done is to divide the overall mass flow entering the volute by the number of vanes in the turbocharger (in this case 17). However this involves the (inaccurate) assumption that the mass flow through each pair of vanes is the same. A more accurate figure may be obtained from the Part I results and involve no net gain or loss of mass into (or from) the computational domain.

Having specified the mass flow, the velocity components must be specified at the upstream boundary. There is more than one way by which this can be done - three of which are noted below:

Case 1: The flow may be assumed to be uniform in the entire region ABB' (shaded in Fig 24a) - considered as sufficiently far upstream of the vanes to be uninfluenced by their presence. The uniform values of the velocity components may be obtained by interpolation and averaging from the results of the Part I analysis. These uniform velocity components will only be imposed at the boundaries AB and AB' but, effectively, the flow in the entire triangular region will be more-or-less the same.

Case 2: Non-uniformity in the region ABB' may be taken into account. In practice this would involve the specification of a number of different small boundary 'patches' (another term of PHOENICS jargon) in each of which different values of the velocity components will be imposed. Naturally this may be expected to yield more accurate results but the process of defining each small patch and imposing the different velocities may be quite time-consuming and would necessitate a lengthy input file.

Case 3: The region ABB' may be ignored altogether and the inlet boundary conditions imposed on the circumferential arc BB'. Similarly the constant pressure condition downstream may be imposed on the arc EE' (Fig 24b). There are obvious advantages to this formulation chief among which is the fact that the downstream constant pressure condition is now more physically correct. The disadvantages are less obvious but they are present nevertheless. One disadvantage is that the circular arcs BB' and EE' are in fact being approximated by stepped boundaries (Fig 24c). Moreover the shaded regions ABB' and EE' cannot be ignored altogether. For reasons of consistency, PHOENICS requires that all grid points be specified; the regions ABB' and EE' can later be set to zero porosity (just like the cells which make up the solid body of the vanes). This procedure is rather laborious.

All three cases were tested and convergence to a reasonable degree was found to be attainable in 300 sweeps (a comparison of the results after 300 sweeps and after 600 sweeps for a particular case showed the difference to be less than 1%). A set of vector plots, corresponding to Cases 1, 2 and 3 are depicted in Figs 25, 26 and 27. In all these cases, the mass flow has been specified upstream but it may equally well have been specified downstream. Although the vector plots of Figs 25 and 26 look the same, careful examination reveals the non-uniformity in velocity.
components at the upstream boundaries $AB$ and $AB'$; this difference will of course reflect on the solution in the entire flowfield but the results are not significantly altered. In Fig 27, the inlet velocity components are specified on the circular arc $BB'$ (approximated by the stepped cells). Note that these components are not the same as those specified on the boundaries $AB$ and $AB'$ for Case 1. The uniform velocity components specified on the boundaries $AB$ and $AB'$ in Case 1 are averaged values obtained at different radii from the Part 1 calculation. The uniform velocity components specified on the arc $BB'$ in Case 3 are also obtained from the Part 1 calculation - but only at the constant radius corresponding to the arc $BB'$.

The actual values of the streamwise velocity components at the points $L_1$, $L_2$, $L_3$, and $L_4$ (depicted in Fig 22) are given in Table 2 for each of these three cases and compared with those of the experimental measurements. Table 3 is the corresponding tabulation of normal velocity components and Table 4 of turbulence levels. The experimental values of velocity were obtained by converting the velocity measurements reported by Hamid and Elder (1992) from polar to body-fitted coordinates. The experimental values of turbulence were obtained directly from Hamid and Elder (1992).

In all three tables, the points $L_1$, $L_2$, $L_3$, and $L_4$ have been arranged in order of distance from inlet (increasing $Y$ coordinate). In Table 3, the points $L_1$, $L_2$, and $L_3$ have been omitted as numerical approximations in this region make the comparison unreliable.

A few general comments on Tables 2 and 3 are called for. Consider first Table 2. Agreement between modelling and experiment is generally good though in all three cases the modelling tends to overestimate the velocity near the leading edge (especially at $L_2$) and underestimate it near the suction surface ($L_3$, $L_4$, $L_5$). As between the three cases themselves there is little to suggest that any one is markedly superior to the other two in terms of accuracy: Case 3 gives better results than Cases 1 and 2 at $L_2$, $L_3$, and $L_4$, but fails altogether at $L_1$. At all other points the three cases appear to be about equally accurate and agree with experimental measurements just about as closely as may reasonably be expected.

Similar remarks may be made about Table 3. Agreement with experiment is once again reasonable though appreciable differences do exist at $L_1$, $L_2$, and $L_3$. However this is perhaps not unexpected as the transverse component of velocity is small anyway. In absolute terms, the differences at $L_3$, $L_4$, and $L_5$ are not large - and, indeed, the agreement at other points is remarkably good. As with Table 2, there is little to choose.

As regards turbulence comparisons, one general comment which may be made is that the modelling, in all cases, predicts higher values of turbulence near the surface of the vanes. The points in question are $L_1$, $L_2$, $L_3$, $L_4$, and $L_5$. However this is only to be expected and it is perhaps a little surprising that the experimental measurements at these stations do not exhibit a significant increase in turbulence levels. At the other points the agreement is reasonably good though at $L_6$ and $L_7$, the modelling values are lower than experiment. It should be noted that in order to generate these turbulence levels (using the k-e model), an assumption must be made about the turbulence level at inlet. For Cases 1 and 2, a turbulence level of 2% was assumed at the inlet boundaries $AB$ and $AB'$; for Case 3 a level of 3% was assumed at the inlet arc boundary $BB'$. By trying different assumptions it is possible to obtain better agreement at some of the points but at the expense of a wider disagreement at other points.

As with Tables 2 & 3, there is little to choose between the three cases. Case 3 agrees better with experiment at $L_4$, $L_5$, $L_7$, and $L_9$, but is significantly worse at $L_1$, $L_2$, and $L_3$. At all other points the three cases appear about equally accurate (or inaccurate) agreeing with the experimental predictions at $L_1$, $L_2$, $L_3$, $L_4$, $L_5$, $L_6$, $L_7$, $L_9$, and $L_10$, and disagreeing with them at $L_1$, $L_2$, and $L_3$.

If a choice must be made between Cases 1, 2 and 3, perhaps Case 1 should be adopted because of its relative simplicity.

We conclude this section with a brief look at the three-dimensional solution. The model adopted is a straightforward extension of Case 1. Figs 28 a,b,c depict vector plots of the solution at hub, midspan and shroud respectively. All plots are similar indicating that the three-dimensional influence in this region of the volute is not significant. The main difference is that the normal component of the velocity decreases from hub to shroud.

It is true that the picture may be somewhat different further downstream of the volute where the axial component of the flow increases near the junction of the hub and leading edge (as seen in Figs 12 c-d, $\theta = 180^\circ-270^\circ$), so that the three-dimensional aspects of the flow may be expected to become more significant. This study has not been carried out here, partly for reasons of time, partly because no experimental comparisons are available in this region, and finally because the results of the volute flow analysis are in any case unreliable very close to the leading edges (as the vanes were assumed to be absent in the volute flow modelling). A satisfactory analysis of the flow in this area would probably require a combination of volute and vane modelling (simultaneously) together with a much finer grid. This does not seem feasible with the computer facilities currently available but it may form the subject of a future study.

**4. CONCLUSIONS**

To summarise, a fairly detailed flow analysis has been carried out in the volute and vanes of a high pressure radial inflow turbine. These have been treated as two separate problems linked only by the fact that the results of the volute analysis are incorporated into the boundary conditions for the flow analysis through the vanes. The two modelling problems are very different in nature - the flow in the volute is wholly three-dimensional and cannot be meaningfully approximated in any two-dimensional manner ... although the flow in the vanes is also three-dimensional, the axial flow component (i.e. in the hub-tip direction) for the vaned region studied is very small and a two-dimensional analysis yields results which are very nearly as accurate as the fully three-dimensional study and allows a considerable saving in computer time. There are also other differences between the two problems of modelling the flow in the vane and volute. The flow in the volute is more physically complex involving mainstream vortex flow over the axis of the rotor as well as secondary vortex flow in the different cross-sections ... the flow in the vanes is relatively uniform, the overall velocity showing a monotonic increase from leading edge to trailing edge and the static pressure decreasing correspondingly. The boundary conditions for the volute flow modelling reflect its complex 3D structure; the inlet boundary marks the beginning of one set of grid lines, the outlet boundary marks the end of another (orthogonal) set of grid lines. The boundary conditions for the vane modelling are better matched especially for the H-grid where inlet and outlet boundaries mark the beginning and end of the streamwise grid lines, whereas periodic and wall boundaries mark the terminal elements of the transverse grid lines.

Despite the differences in the two problems, all of which indicate that the modelling of the flow in the volute is the much more difficult of the two, each problem presented characteristic hurdles. When all these were taken into consideration, the flow analysis through the vanes proved just about as difficult a task as the flow analysis through the volute.

Perhaps the biggest hurdle in the vane flow-modelling was the high degree of skewness of the H-grid which necessitated a fresh start to the grid generation procedure. The orthogonal grid presented hurdles of its own, notably 'cornered' inlet and outlet boundaries (that is, inlet and outlet boundaries extending along two mutually orthogonal directions) and non-matching periodic boundaries. These hurdles were dealt with in different ways and a trial and error procedure was adopted to see which method yielded the best results.
In the modelling of the flow in both volute and vanes, it was necessary to interact with PHOENICS by inserting Fortran coding sequences in its subprogram GROUND for the purpose of 'linking'. In the case of the volute, these sequences are executed at the tongue where the flow having completed a full circulation rejoins the mainstream. In the case of the vanes, they are executed at the periodic boundaries, the procedure being rendered more difficult by the fact that these boundaries do not match in their transverse coordinates.

In the modelling of both volute and vanes, a constant-pressure boundary condition was imposed at the downstream boundary - which roughly coincides with the radius of the rotor. Admittedly this condition is not strictly accurate, because some amount of non-uniformity is inevitable, especially in the region about the tongue. A practicable - but time consuming - option is to attack the problem iteratively. Thus, the non-uniformity at the end of one complete solution cycle, incorporates this as a new boundary condition and start the solution procedure afresh. For reasons of time, this approach was not attempted here.

The overall solution procedure, for the modelling in both the volute and the vanes, takes account of all the important flow features. Together with the experimental investigations of Hamid and Elder (1992) - with which they agree well - they make a complete study providing details of the flow at all points of the volute-vane configuration. Extension of the analysis into the rotor may form the subject of a future study.

Another possible subject of a future study is the flow analysis between those vanes in the region of the volute where the axial flow component significantly increases at the hub ($\theta = 180^\circ-270^\circ$, Figs 12 g..j). This study would undoubtedly present many new hurdles and would involve grid-generation and modelling techniques quite different to those used in the current study, but it should prove an interesting challenge.

Let us conclude with a statement which brings out the essence of the approach that has been adopted for this paper. Accurate results are no doubt highly satisfying, but the price of accuracy is often impracticability. In this paper attention has been focused, first and foremost, on the design of a workable method of solution to this complex problem - a goal which has been reached. Nor has accuracy been sacrificed unduly. It is hoped that the method will prove a useful addition to the toolbox of the turbocharger designer.

ACKNOWLEDGEMENTS

The research on which this paper is based was sponsored by a consortium, the members of which were (listed alphabetically): British Aerospace plc, Department of Trade and Industry, Holset Engineering Ltd, Ministry of Defence, Noel Penny Turbines Ltd, Nomalair Garrett Ltd and Rolls Royce plc. Acknowledgements are also due to CHAM Ltd for providing the software package and - in particular - to John Edwards of CHAM for his valuable counsel.

REFERENCES

Barnard M C S and Benson R S (1968)

Benisek E and Strawle A G (1990)
"Laser Velocimeter measurements in a Turbocharger Turbine"
In Turbomachinery and Turbochargers, Inst Mech Engrs pp 1-6

Benson R S and Scrimshaw K H (1965)
"An experimental investigation of non-steady flow in a radial gas turbine"

Benway R B (1987)
"Design and Development of a light weight high pressure ratio aircraft turbocharger"
S.A.E. Technical Paper Series 871041

Cheng C P (1992)
"The Modelling of 3-D Flows in Turbomachinery using Time-marching Techniques"
Ph.D. Thesis, Cranfield Institute of Technology

Dawes W N (1988)
"Development of a 3D Navier-Stokes solver for application to all types of turbomachinery", ASME Paper 88-GT-70

"A Time Marching Method for Two and Three-Dimensional Blade to Blade Flows"
ARC - RM - 3775

Hamid A, Abdallah S and Tabakoff W (1977)
"Flow study in the cross sectional plane of a turbine scroll"
AAIA Paper 77-714 (10th Fluid and Plasma Dynamics Conference, Albuquerque)

Hamid S and Elder R L (1992)
"Investigation of Flow in a High Expansion Ratio Radial Turbine at 400°C inlet temperature using Laser Anemometry" (submitted for publication)

Hiett G F and Johnston I H (1963)

"The computational experiment applied to aerodynamic design and analysis of turbomachinery", S.A.E. Paper 900360

Lapworth B L (1987)
"Three-Dimensional Elliptic Computations for Viscous Turbomachinery Flows"
Ph.D. Thesis, Cranfield Institute of Technology

Lapworth B L (1988)
"Examination of Pressure Oscillations arising in the computation of cascade flow using a boundary fitted coordinate system", Int. J. Num. Meth. in Fluids, Vol 8, pp 387-404

"Flow Modelling in the Volute of a Radial Inflow Turbine" (in preparation)

Lymberopoulos N, Baines N C and Watton N (1988)

"Effects of specific speeds on experimental performance of a radial inflow turbine"
NASA TN-D-6605

Patankar S V (1980)
Table 1

<table>
<thead>
<tr>
<th>Position Coordinates</th>
<th>(P_0) (bars)</th>
<th>(P_s) (bars)</th>
<th>(p) (kg/m²)</th>
<th>(V) (m/sec)</th>
<th>(P_T = P_s + \frac{1}{2}pV^2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(L_1)</td>
<td>1.0</td>
<td>1.4665</td>
<td>0.819</td>
<td>316.5</td>
<td>187,670</td>
</tr>
<tr>
<td>(L_2)</td>
<td>1.1</td>
<td>1.488</td>
<td>0.828</td>
<td>316.0</td>
<td>190,140</td>
</tr>
<tr>
<td>(L_3)</td>
<td>1.2</td>
<td>1.5465</td>
<td>0.8505</td>
<td>305.0</td>
<td>194,209</td>
</tr>
<tr>
<td>(L_4)</td>
<td>1.3</td>
<td>1.5985</td>
<td>0.871</td>
<td>298.0</td>
<td>198,524</td>
</tr>
</tbody>
</table>

Table 2

A Comparison of Experimental and Numerical Results for the Streamwise Velocity Component (metres/sec)

<table>
<thead>
<tr>
<th>Position Coordinates</th>
<th>(Y)</th>
<th>(Z)</th>
<th>Experiment</th>
<th>Modelling (Case 1)</th>
<th>Modelling (Case 2)</th>
<th>Modelling (Case 3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(L_1)</td>
<td>15</td>
<td>9</td>
<td>144</td>
<td>148</td>
<td>143</td>
<td>204</td>
</tr>
<tr>
<td>(L_2)</td>
<td>19</td>
<td>13</td>
<td>142</td>
<td>174</td>
<td>177</td>
<td>165</td>
</tr>
<tr>
<td>(L_3)</td>
<td>21</td>
<td>10</td>
<td>164</td>
<td>149</td>
<td>147</td>
<td>173</td>
</tr>
<tr>
<td>(L_4)</td>
<td>23</td>
<td>17</td>
<td>132</td>
<td>148</td>
<td>153</td>
<td>125</td>
</tr>
<tr>
<td>(L_5)</td>
<td>25</td>
<td>14</td>
<td>189</td>
<td>191</td>
<td>195</td>
<td>182</td>
</tr>
<tr>
<td>(L_6)</td>
<td>26</td>
<td>10</td>
<td>190</td>
<td>170</td>
<td>171</td>
<td>185</td>
</tr>
<tr>
<td>(L_7)</td>
<td>29</td>
<td>15</td>
<td>217</td>
<td>225</td>
<td>227</td>
<td>210</td>
</tr>
<tr>
<td>(L_8)</td>
<td>30</td>
<td>11</td>
<td>233</td>
<td>214</td>
<td>216</td>
<td>220</td>
</tr>
<tr>
<td>(L_9)</td>
<td>33</td>
<td>13</td>
<td>256</td>
<td>261</td>
<td>261</td>
<td>253</td>
</tr>
<tr>
<td>(L_{10})</td>
<td>37</td>
<td>11</td>
<td>325</td>
<td>332</td>
<td>328</td>
<td>327</td>
</tr>
<tr>
<td>(L_{11})</td>
<td>39</td>
<td>1</td>
<td>350</td>
<td>347</td>
<td>340</td>
<td>342</td>
</tr>
<tr>
<td>(L_{12})</td>
<td>41</td>
<td>8</td>
<td>375</td>
<td>377</td>
<td>375</td>
<td>375</td>
</tr>
<tr>
<td>(L_{13})</td>
<td>43</td>
<td>11</td>
<td>393</td>
<td>394</td>
<td>393</td>
<td>393</td>
</tr>
</tbody>
</table>

Table 3

A Comparison of Experimental and Numerical Results for the Transverse Velocity Component (metres/sec)

<table>
<thead>
<tr>
<th>Position Coordinates</th>
<th>(Y)</th>
<th>(Z)</th>
<th>Experiment</th>
<th>Modelling (Case 1)</th>
<th>Modelling (Case 2)</th>
<th>Modelling (Case 3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(L_1)</td>
<td>15</td>
<td>9</td>
<td>7.1</td>
<td>6.0</td>
<td>7.0</td>
<td>5.1</td>
</tr>
<tr>
<td>(L_2)</td>
<td>19</td>
<td>13</td>
<td>12.4</td>
<td>21.0</td>
<td>23.3</td>
<td>24.2</td>
</tr>
<tr>
<td>(L_3)</td>
<td>21</td>
<td>10</td>
<td>3.8</td>
<td>5.2</td>
<td>3.7</td>
<td>3.3</td>
</tr>
<tr>
<td>(L_4)</td>
<td>23</td>
<td>17</td>
<td>12.8</td>
<td>12.1</td>
<td>12.4</td>
<td>15.9</td>
</tr>
<tr>
<td>(L_5)</td>
<td>25</td>
<td>14</td>
<td>-16.4</td>
<td>-17.9</td>
<td>-18.9</td>
<td>-8.4</td>
</tr>
<tr>
<td>(L_6)</td>
<td>26</td>
<td>10</td>
<td>-10.8</td>
<td>-10.1</td>
<td>-10.1</td>
<td>-4.9</td>
</tr>
<tr>
<td>(L_7)</td>
<td>29</td>
<td>15</td>
<td>-4.1</td>
<td>-4.4</td>
<td>-4.3</td>
<td>-2.8</td>
</tr>
<tr>
<td>(L_8)</td>
<td>30</td>
<td>11</td>
<td>-11.2</td>
<td>-5.6</td>
<td>-6.5</td>
<td>-1.4</td>
</tr>
<tr>
<td>(L_9)</td>
<td>33</td>
<td>13</td>
<td>-11.2</td>
<td>-3.6</td>
<td>-4.0</td>
<td>-7.0</td>
</tr>
<tr>
<td>(L_{10})</td>
<td>37</td>
<td>11</td>
<td>-13.5</td>
<td>-14.2</td>
<td>-14.2</td>
<td>-13.7</td>
</tr>
</tbody>
</table>

Table 4

A Comparison of Experimental and Numerical Results for the Turbulence Level (%)

<table>
<thead>
<tr>
<th>Position Coordinates</th>
<th>(Y)</th>
<th>(Z)</th>
<th>Experiment</th>
<th>Modelling (Case 1)</th>
<th>Modelling (Case 2)</th>
<th>Modelling (Case 3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(L_1)</td>
<td>15</td>
<td>9</td>
<td>4.1</td>
<td>7.2</td>
<td>7.1</td>
<td>7.1</td>
</tr>
<tr>
<td>(L_2)</td>
<td>19</td>
<td>13</td>
<td>4.4</td>
<td>4.3</td>
<td>4.2</td>
<td>4.1</td>
</tr>
<tr>
<td>(L_3)</td>
<td>21</td>
<td>10</td>
<td>3.9</td>
<td>6.6</td>
<td>7.2</td>
<td>5.5</td>
</tr>
<tr>
<td>(L_4)</td>
<td>23</td>
<td>17</td>
<td>3.4</td>
<td>3.6</td>
<td>3.5</td>
<td>3.7</td>
</tr>
<tr>
<td>(L_5)</td>
<td>25</td>
<td>14</td>
<td>4.8</td>
<td>5.8</td>
<td>3.7</td>
<td>3.6</td>
</tr>
<tr>
<td>(L_6)</td>
<td>26</td>
<td>10</td>
<td>4.3</td>
<td>5.9</td>
<td>6.5</td>
<td>4.5</td>
</tr>
<tr>
<td>(L_7)</td>
<td>29</td>
<td>15</td>
<td>4.2</td>
<td>3.5</td>
<td>3.4</td>
<td>3.9</td>
</tr>
<tr>
<td>(L_8)</td>
<td>30</td>
<td>11</td>
<td>2.8</td>
<td>4.4</td>
<td>4.4</td>
<td>3.4</td>
</tr>
<tr>
<td>(L_9)</td>
<td>33</td>
<td>13</td>
<td>4.5</td>
<td>3.4</td>
<td>3.3</td>
<td>3.2</td>
</tr>
<tr>
<td>(L_{10})</td>
<td>37</td>
<td>11</td>
<td>4.9</td>
<td>4.3</td>
<td>4.3</td>
<td>3.3</td>
</tr>
<tr>
<td>(L_{11})</td>
<td>39</td>
<td>1</td>
<td>6.2</td>
<td>9.8</td>
<td>10.7</td>
<td>12.7</td>
</tr>
<tr>
<td>(L_{12})</td>
<td>41</td>
<td>8</td>
<td>4.5</td>
<td>7.4</td>
<td>7.8</td>
<td>6.9</td>
</tr>
<tr>
<td>(L_{13})</td>
<td>43</td>
<td>11</td>
<td>4.7</td>
<td>4.4</td>
<td>4.4</td>
<td>3.5</td>
</tr>
</tbody>
</table>
Fig 1: Volute cross-section
Fig 2: Cross-section with vanes absent
Fig 3: Different gridding regions
Fig 4: Gridding in Regions A, B & C
Fig 5: Gridding in Region D
Fig 6: Gridding in Region E
Fig 7: Complete cross-sectional grid
Fig 8: Grid incompatible with PHOENICS
Fig 10: The seventeen vanes

Fig 9: Refined Grid

Fig 11: Mainstream vector plots

Fig 12: Flow Solution at Various Stations

θ = 0°
(a)

θ = 90°
(b)

θ = 180°
(c)

θ = 270°
(d)
Fig 15: Smoothened H Grid with periodic extension

Fig 16: "X-ray" View of Fig 15

Fig 17: A near-orthogonal grid

Fig 18: Different mathematical boundaries

Fig 19: Extension to facilitate linking

Fig 20: "X-ray View" of Fig 19

Fig 21: Points of experimental measurements

Fig 22: Grid cells corresponding to Fig 21

Fig 23: Experimental vector plot

Fig 24a: Inlet region of the grid

Fig 24b: Bounding circumferential arcs for Case 3
Fig 25a. Case 1 solution

Fig 26c. Case 2 solution

Fig 27c. Case 3 solution

Fig 28. Three-dimensional solution