The application of throughflow optimisation to the design of radial and mixed flow turbines

G D Cox
PCA Engineers Limited, UK
C Fischer, M V Casey
Institute of Thermal Turbomachinery, Stuttgart University, Germany

ABSTRACT

Radial and mixed flow turbines are important components of turbochargers in automotive engines. Their aerodynamic design is generally compromised by severe mechanical constraints, to deal with high temperature and unsteady operation, but also by the requirement of low inertia for rapid turbocharger response from low engine speed. Conventionally, the designer deals with these constraints in the preliminary design, using a high degree of empiricism, followed by extensive CFD analysis and geometry optimisation. This paper describes a new approach to the preliminary design using a quasi-3D throughflow method coupled to an optimiser, which allows a more rapid consideration of the design issues before moving on to 3D CFD analysis. The throughflow-based optimisation system was able to increase efficiency by over 3% at the same inertia or to reduce inertia by 20-30% at the same efficiency, compared to a baseline design.

NOTATION

U/C	The ratio of the wheel leading edge speed, at the casing, to the spouting velocity (non-dimensional)
R1	The radius of the leading edge at the casing
PF	Penalty Function
Np	Population size
h	Number of parameters
G	Current generation

1 INTRODUCTION

The design of turbines for automotive turbochargers is a balancing act between the requirement for high efficiency, for engine efficiency, and low inertia, for engine response. Low inertia drives the wheel diameter, and the hub line, down to a level that is lower than the designer would choose purely for aerodynamic reasons. The growing trend from car makers is to push the turbocharger manufacturers towards solutions that are optimised to reduce inertia for individual engines, so that modern designs operate in a region of high loading where the available empirical knowledge is limited.

The traditional process of turbine wheel design starts with 1D empiricism to establish the major features (e.g. inducer and exducer radii). Vane and passage shapes are developed from simple geometry. The empiricism is calibrated against test results, but the information is rarely available to establish the subtle design trade-offs needed in product optimisation at high loading. The modern process uses 3D CFD calculations to quantify the effects of subtle design changes, but the combination of a large number of geometric variables and significant CFD run-times means that it is difficult to ensure an optimum design within a realistic timescale.

The solution, that is the subject of this paper, is to use a quasi-3D (throughflow) calculation, which is extremely quick, to reveal key flow features. Some of these features are quantified as penalty functions that result in a score. By carefully establishing the scoring system, rival designs can be examined automatically and poor variants rejected. A large number of rival designs are generated by an optimiser that manages the process of rejection and creation of potentially better designs. The fact that the throughflow method acts on 3D geometry means that the inertia of each design is also calculated and a basic structural assessment can be carried out on the most promising designs. The final step, which is carried out after optimisation, is to carry out standard 3D CFD and FE analyses.

The title of the paper suggests that a choice has to be made between a radial and mixed flow turbine. Using the presently described method this distinction is not important; the appropriate style is allowed to emerge from the optimisation.

The elements of the design system are as follows,

- A parameter-based geometry definition system for radial and mixed-flow turbomachinery stages (based on Bezier curves and similar to that published by Casey (1)), including estimates of blade and disc inertia.
- An interface between the geometry system and a streamline curvature though-flow code, (known as Vista TF and described by Casey and Robinson (2)), but which now includes correlations for turbine deviation and losses, as described by Cox et al. (3)
- An optimization method for determining the best impeller, which is based on an evolutionary optimisation strategy known as differential evolution, as described by Storn and Price (4).
- Considerations with regard to the definition of the most suitable global objective function based on extensive experience of turbine impeller design and improved through the use of CFD simulations.

2 QUANTIFICATION OF TURBINE AERODYNAMIC PERFORMANCE FROM A Q-3D CALCULATION

2.1 Parameterised geometry definition method

The impeller geometry definition method described here is closely based on that published by Casey (1), where full details of the basic theory can be found. The use of the geometry method has already been described in some detail in connection with radial compressor optimisation by Casey et al. (5) so in the interest of space no extensive detail is given here and the reader is referred to the earlier papers. The method allows a meridional channel to be defined as series of Bezier patches in the meridional plane, whereby a series of dimensionless parameters are used to define the key skeletal dimensions and the curvature of the meridional walls. The implementation allows the meridional channels and blade rows of radial and mixed flow compressors and turbines to be defined as a set of parameters. Examples of typical designs and their adjustment in shape via the different parameters are given in later sections of the paper. Although there are about 50 parameters many of these are fixed by the boundary conditions (such as the length of the outlet channel or the radius ratio of the inlet nozzle) and many may be selected based on earlier experience (such as the curvature of the walls in the outlet duct).

The impeller blade is defined as a ruled surface of straight lines joining points on the hub and the shroud contours. In the turbocharger turbine examples given here the ruled surface is oriented to give radial blade elements, but other orientations can also be selected. The blade shape is then defined uniquely by the distribution of the blade angles along the hub contour and the thickness along the hub and casing contour. The leading and trailing edge ellipses are defined as separate parameters. The complete numerical geometry definition of the blade and the hub then allows the mass and the inertia of the impeller to be determined, which are important parameters with regard to optimisation of turbocharger response. The geometry program includes several interface elements for transfer of geometrical data to other codes and to FEM, CAM and CFD tools, whereby the following were used in this study:

- A BGI file for transferring the geometry into ANSYS Workbench for subsequent mesh generation and 3D CFD analysis using ANSYS-CFX.
- A suitable geometry input file for reading into the throughflow code.
- Inertia calculation of the vanes and a solid hub positioned directly under the vanes.

2.2 Streamline curvature throughflow

The streamline curvature throughflow code used in this design tool is described in detail by Casey and Robinson (2), so only the features that are particularly relevant to its use in the turbine optimization procedure are briefly described here. The code is a general purpose code which is able to calculate most types of turbomachinery from water turbines to multistage axial compressors.

Key features of the code relevant to the optimisation of turbocharger turbines are:

- Highly curved annulus walls are allowed providing a simple definition of wall geometries required for radial and mixed flow turbine impellers.
- Any combination of blade row calculating stations, together with duct flow regions, can be used in the domain, so that in this application the domain includes the inlet nozzle duct, the impeller and the axial outlet channel.
- Internal blade row calculating stations are used. In impeller blade rows 15 internal planes are typically used and this allows an approximation for the blade-to-blade flow field to be calculated estimating the suction and pressure surface velocity distributions.
- Compressible fluids are possible, including supersonic relative flow in blade rows, such that transonic impellers may be calculated.
- The code includes a set of correlations for losses and deviation of radial and mixed flow turbines which have been derived from a range of CFD simulations. The deviation correlation is described by Cox et al. (3) and the loss correlation has not yet been published. Note that these correlations determine the spanwise variation of flow angle and entropy at impeller outlet taking into account the operating point, the geometry and the tip clearance flows.

The code also includes an option that allows a restart from a previously converged solution. This considerably reduces the effort for a new calculation with slightly changed geometry, which is particularly useful in combination with an optimization method. The flow field information of a converged iteration is stored on the basis of non-dimensional span-wise and meridional coordinates. These can be used to start a new simulation, even if the geometry has been changed, by mapping the values onto the new geometry.

The paper of Cox et al. (3) shows some examples of simulations with the throughflow method compared to 3D CFD calculations, which demonstrate that the throughflow method is adequate for preliminary design purposes.

3 THE OPTIMISATION SCHEME

3.1 Optimisation method

The basic principles of evolutionary methods in optimization processes are similar to the theory of natural selection of Darwin, whereby a population of individuals changes over several generations following laws of natural or artificial selection, involving reproduction and mutation of the fittest surviving individuals. In this case each individual is a different impeller design. Its chance of survival into the next generation (fitness) is related to how well it meets the user defined design objectives. Many different approaches for such optimisation problems exist which are difficult to compare with each other, and the choice of which method to choose depends to some extent on the problem it is applied to.

An earlier paper on the optimisation of centrifugal compressors, Casey et al. (5), used a breeder genetic algorithm for this purpose, and although this technique is still available within the current program, and gives similar results to those presented here, a more modern system is now generally used as it is quicker and more robust. This newer technique is known as differential evolution (DE) and is described in detail by Storn and Price (4). This is a reliable and versatile function optimiser which is very easy to use and, since its introduction, has proven itself in various international contests on optimisation methods. Further details can be found in (4) and (6).

In each generation, the DE method generates new parameter vectors by adding a weighted difference between two randomly selected members of the previous population to another member. In the basic strategy the difference is added to a random vector:

$$\mathbf{v}_i^{(G+1)} = \mathbf{x}_{r1}^{(G)} + F\left(\mathbf{x}_{r2}^{(G)} - \mathbf{x}_{r3}^{(G)}\right) \quad , \text{i} = 1(1)Np$$

The indices *r1*, *r2* and *r3* are mutually exclusive random indices and different from the current index *i*. To increase diversity in the population, crossover is introduced. Here, parts of the previous design vector are inherited by the new design, following a certain probability. The selection scheme is another distinctive difference to other evolutionary algorithms. Here fitness is tested against the direct predecessor.

One advantage of this optimisation method is that the mutation has a self adaptive behaviour. The differences adapt to the shape of the objective function and decrease when the population converges. The actual algorithm used is based on a newer modification based on local and global neighbourhoods as described in (7).

Typically the population size would be about 300, which is ten times the number of free parameters, but the optimiser was tuned to perform well with small populations so that a population size of 60 could be chosen. No advantage has been found in significantly increasing this number, despite the large number of geometry variables. In working with a low population size (relative to the number of variables) it is necessary that the geometry limits be set within fairly small ranges, but this is not a problem for radial turbine design.

The technique for dealing with this is to allow the optimisation process to continue until the penalty function reaches an asymptote, and then to examine the proximity of the geometry parameters to the limits. The optimiser is then re-run with displaced limits where the limits appear to be constraining the design. In this way a number of rival designs may be produced.

3.2 Objective functions

Ideally it would be best to optimize the efficiency or minimize the losses in the impeller to derive the optimal aerodynamic performance, but this would require the use of a more complex 3D viscous RANS CFD simulation, and is prohibited because of the time required. The throughflow code is extremely fast, but cannot calculate the losses accurately enough for this process. The losses in the impeller throughflow simulation are chosen to be compatible with the overall stage performance correlations of the preliminary design process so cannot form the basis of the optimization. Because of this, other results and features of the flow field from the throughflow analysis have to be included in the evaluation of the impeller.

In addition we need constraints for certain coefficients, which can be defined as targets having a high importance and therefore a high weighting factor. These targets are:

- Work coefficient (closely related to U/C)
- Incidence at tip and hub (generally with a low weighting)
- Relative velocity ratio across the impeller (de Haller number)

The following parameters were considered to have an effect on the efficiency and were included as objectives within the penalty function:

- Frictional losses as quantified by Denton's U^3 approach (8).
- Circumferential flow coefficient (exit)
- Meridional flow coefficient (exit)
- Peak blade loading at hub, mean and shroud
- Positive acceleration along the mean streamline

Not all the objectives are results from the throughflow calculation. An inertia limit and the minimal throat width are a result from the geometrical design tool but are balanced against the aerodynamic objectives. Both the inertia limit and the minimal throat width are targets defined by the user. In addition to the possibility to set a hard inertia limit it is also possible to create a pareto front consisting of the aerodynamic objectives (still including the hard penalty for the inertia limit) and the inertia itself. This creates a range of designs, which mostly are below the hard inertia limit.

4 STUDIES

The optimisation method is demonstrated by taking an existing baseline design of a turbine and attempting to improve it. This baseline, known as Geometry 4, was generated to support the development of the specific throughflow empiricism for this class of turbine. It is considered to be a typical turbocharger geometry except in that it is biased towards high efficiency rather than low inertia. Table 1 records the main geometric features and boundary conditions. The boundary conditions, especially the low U/C value, are typical for turbocharger design.

Table 1: Geometric features and boundary conditions of baseline turbine

Geometry	Style	Radial
	Number of vanes	8
	Casing exducer radius / R1	0.92
	Impeller length / R1	0.89
	Hub trailing edge thickness / R1	0.04
	Tip thickness / R1	0.02
	Tip clearance / R1	0.02
Boundary conditions	Expansion Ratio (t-s)	2.3
	$N/\sqrt{T}$ (rpm/$K^{0.5}$)	3991
	U/C	0.5

The baseline design was not originally defined with the parameterised geometry system used here, but could be recreated faithfully using this approach. The meridional geometry is shown in Figure 1 with the meridional velocity predictions from the throughflow and 3D CFD calculations. The CFD calculation is carried out on a mesh of 175,000 nodes. The throughflow calculation is carried out on a mesh of 198 nodes. The throughflow meridional velocity contours are considered to be acceptably close to the circumferentially averaged 3D CFD values.

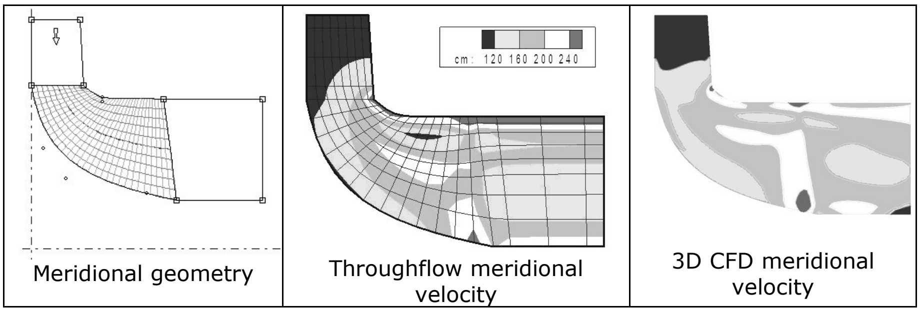

Figure 1: Geometry, throughflow and CFD solution of the baseline design.

The throughflow calculation runs to a defined mass flow. Loading in terms of the work coefficient is chosen to be the target rather than expansion ratio as the velocity triangles are better predicted than pressure, which is influenced by loss generation. The throughflow calculation is not designed to provide an accurate prediction of loss.

The design studies carried out with the optimiser included a number of geometric constraints for comparison against the baseline: the same casing inlet radius R1, the same number of vanes (8), the same vane thickness distribution, exducer radius at the casing limited to 95% of R1 and a back-swept trailing edge. In addition, a limiting throat width at the hub was defined to ensure an acceptable fillet radius. Within appropriate range limits the optimiser allows variation in the hub and casing geometry (as defined by the position of certain Bezier points) and the blade angle variation on the hub (as defined by another set of Bezier points). The blade angle variation on the casing is not required as the vane geometry is constrained to a radial-fibre definition.

The optimiser drives the designs towards the required work coefficient of 1.449 (as predicted by CFD for the baseline case) by severely penalising lower or higher values. The target inertia is achieved by penalising values higher than the target. Three other targets are set that are quite specific to this class of turbine: a minimum de Haller number of 1.5, maximum incidence at the hub and casing of 14° and 12° respectively. De Haller numbers below the target are severely penalised. Incidences above the targets are lightly penalised. Of course, an ideal turbine design would be expected to have incidence in the range –10° to –40°. At the low U/C typical of turbocharger turbines it is not realistic to expect negative incidence. The positive incidence values chosen are about as low as are expected from 1D design considerations. The deviation model and the loss distribution model used in the optimisation are the same as applied on the baseline model.

The ability of the optimiser to develop good designs has been split into three design phases, which, it is hoped, reveal something about good basic features of a turbine to suit the particular turbocharger specification and geometry features. Each design has been analysed using consistent CFD, as the baseline, to validate the penalty function methodology and to show whether better designs appear possible.

4.1 Optimiser with no inertia limit

The first study has the inertia target set to a value about 10 times higher than the baseline value so as not to influence the design. In the optimiser the turbine is thus assessed exclusively on aerodynamic features. It is found that with no constraint on inertia the turbine wheel extends in length to reduce the peak loading even though there is a small penalty for wetted area from the Denton U^3 model (8). Three cases were run with progressively higher upper limit on length.

The impeller lengths, penalty functions and inertias (over baseline) are recorded with the meridional shapes in Figure 2. Note that the length of the baseline impeller is 0.89 R1. Two features are evident: the hub line is high, which reduces the flow deceleration close to the hub; and, the inducer hub radius is at, or close, to the tip radius, which reduces the incidence on the hub.

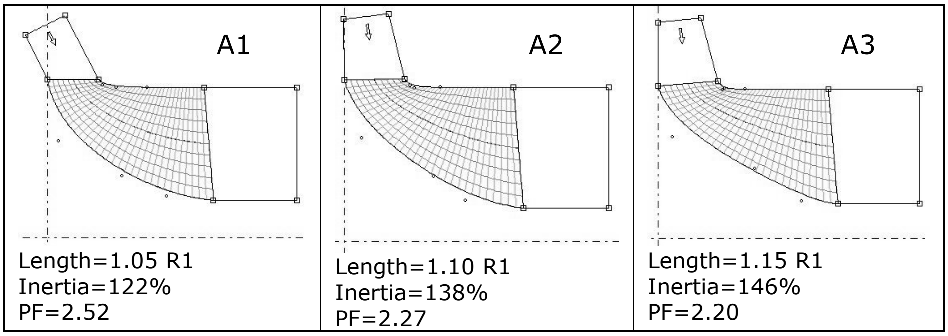

Figure 2: Optimiser results with no constraints on inertia.

4.2 Optimiser with progressively decreasing inertia limit

Starting at the baseline inertia, the inertia target is set progressively lower, at 90%, 80%, 70% and 60% of the baseline value, to simulate the effect of designing a turbine wheel with potentially superior transient response. The resulting designs are split into two groups: those that sit on the front rank, as identified in Section 4.4, and those that are less efficient for the same inertia. The front rank designs are identified in Figure 3. The consequence of limiting inertia, by a high penalty function weighting, is that the aerodynamic penalty functions are forced to rise as the inertia target is dropped. Physically, the impeller length reduces and the hub line reduces. The hub radius at the inducer remains a high fraction of the casing value. There is some variation in the exducer outer radius, but the optimiser clearly favours keeping this parameter high rather than the impeller length.

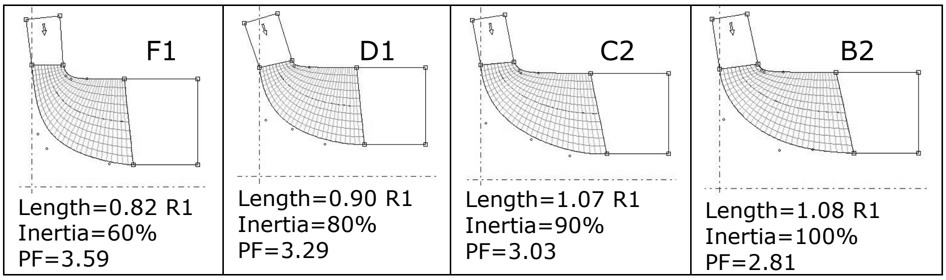

Figure 3: Optimiser results with progressively lower inertia limits.

4.3 Optimiser with decreasing inertia limit and enforced mixed-flow

A frequently asked question is whether a mixed-flow design is superior to a radial design for turbocharger applications. With reference to figures 2 and 3, above, it could be argued that most of the optimised designs have elements of a mixed-flow classification, i.e. a component of axial velocity at the inducer. Another classification is whether the inducer is normal to the radial direction or not. Again, most designs have an observable deviation from pure radial, although most readers would probably describe these designs as more "radial" than "mixed-flow".

Some of the geometric limits have been adjusted in the optimiser to enforce a more mixed-flow style. At the inducer the hub and casing lines are constrained to be between 30° and 60°. Figure 4 identifies a mixed-flow design for each inertia value. None of the designs appear promising because of the Penalty Functions that are higher than the comparable designs in Figure 3. The hub is more substantial. This implies that the wheel has to be shorter for the same inertia. The inducer radius on the hub is also much reduced. This will cause the incidence to be higher in general.

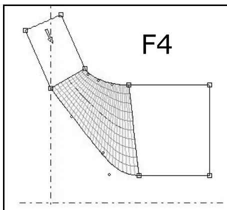	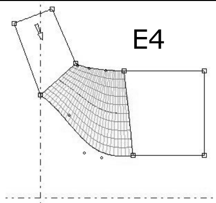	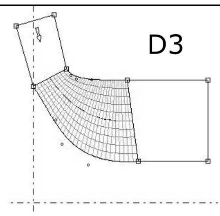	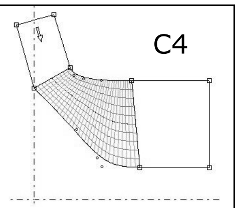
Length=0.66 R1 Inertia=60% PF=4.29	Length=0.68 R1 Inertia=70% PF=4.61	Length=0.79 R1 Inertia=80% PF=3.87	Length=0.80 R1 Inertia=90% PF=3.86

Figure 4: Optimiser results with enforced mixed flow.

4.4 CFD predictions

The quality of the designs generated by this process is ultimately judged by carrying out 3D CFD. For each design a mesh has been generated using ANSYS-TurboGrid, having consistent mesh settings with the baseline, and a CFD model has been constructed and solved. In each case the inlet vector has been adjusted so as to give the same predicted mass flow as the baseline. The predicted efficiencies are plotted on Figure 5 against the inertia, in both cases non-dimensionalised against the baseline. Considering first the unconstrained geometries, a near straight line relationship is seen to exist for the best combinations of efficiency and inertia. It is evident that most designs are within 2% of the optimum efficiency at a particular inertia. Those that are on the "front rank" have been described in Sub-sections 4.1 and 4.2. This "front rank" indicates a significant performance increase over the baseline. The designs having "mixed-flow" style geometric constraints have consistently lower efficiencies than their unconstrained equivalents.

The validation of the optimised throughflow process is taken to be the calibration between CFD efficiency prediction and the throughflow based penalty function. Figure 6 reveals a strong correlation between the two. The unconstrained geometries have, almost universally, higher penalty functions and higher efficiencies than the enforced mixed-flow designs, but there is clearly a common trend. There is more scatter in the data as penalty function increases, where the aerodynamics are less predictable using empirical methods.

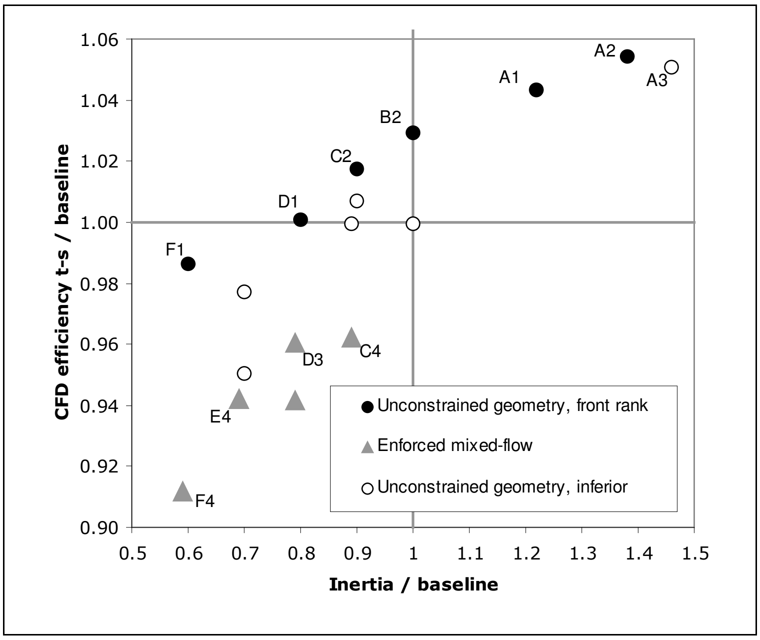

Figure 5: Predicted efficiency versus inertia limits for optimiser designs.

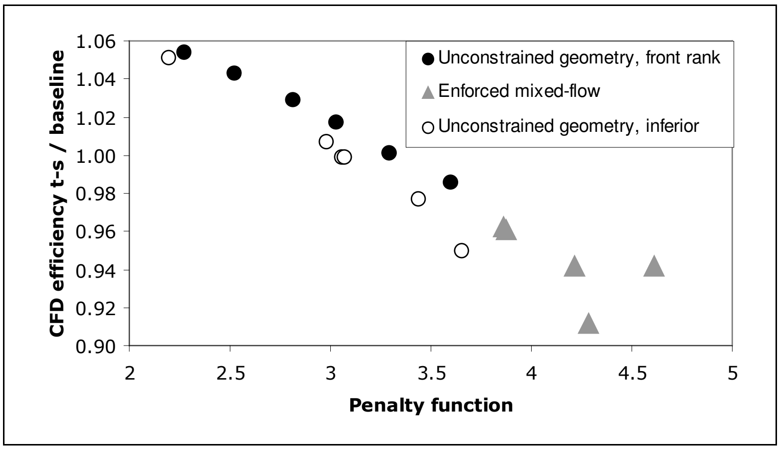

Figure 6: Efficiency predictions versus penalty function.

5 CONCLUSIONS AND DISCUSSION

On the basis of CFD prediction, the throughflow-based optimisation system was able to increase efficiency by over 3% at the same inertia or to reduce inertia by 20-30% at the same efficiency, compared to the baseline. However there has been no modification to the vane thickness, so it is still necessary to confirm the structural similarity.

The optimisation system works reasonably well because the penalty function based on the throughflow results is a generally reliable indicator of CFD efficiency. The validity of the throughflow depends on its empiricism, i.e. the loss distribution and the deviation. Where the penalty function is low it can be assumed that the deviation model is generally reasonable, but at high values of penalty function the flow is likely to be less ordered and the deviation could be different. This probably explains why there is increasing spread on the CFD efficiency vs. penalty function trend. At low penalty functions there is no guarantee of optimum CFD efficiency so it is still advisable to generate a number of variant designs created from slightly different variable limit ranges.

For the case considered, designs having a strong mixed-flow appearance do not perform as well as the more radial "unconstrained" designs, but only a single combination of inlet radius, work coefficient and vane number has been investigated. A full design study should include variation of these independently. It should also include allowance for the structural performance of the turbine wheel.

Aerodynamic targets and penalty weightings were tuned for this particular turbine class. It is expected that a different combination would be required for less highly loaded ideal turbines (U/C = 0.7).

The method has the option of calculating basic structural characteristics of the vane (stress parameters and natural frequencies). By allowing some of the vane thickness terms to be variable, alternative designs are generated having various combinations of aerodynamic and structural penalty functions. The optimiser keeps solutions that sit on the pareto front. A small selection can then be submitted to formal FEA and CFD analysis. This study would produce a different set of "optimum" designs, but the effect on the basic, meridional geometry is expected to be small.

6 REFERENCE LIST

(1) Casey, M.V., "A computational geometry for the blades and internal flow channels of centrifugal compressors", ASME Journal of Engineering for Power, Vol.105, April 1983, pp 288-295.
(2) Casey, M.V., Robinson, C.J, "A new streamline curvature throughflow code for radial turbomachinery", ASME TURBOEXPO 2008, Berlin, ASME GT2008-50187
(3) Cox, G., Roberts, A. and Casey, M.V., "The development of a deviation model for radial and mixed-flow turbines in thoughflow calculations" ASME GT2009-59921, ASME Turbo Expo 2009, June 8-12, 2008, Orlando, Florida, USA
(4) Storn, R.; Price, K., "Differential Evolution - A simple and efficient adaptive scheme for global optimization over continuous spaces", Tech. Rep. TR-95-012, International Computer Science Institute, Berkeley, 1995
(5) Casey, M.V., Gersbach, F., Robinson, C.J, "A new optimisation technique for radial compressor impellers", ASME TURBOEXPO 2008, Berlin, ASME GT2008-50561
(6) Price, K.V.; Storn, R.M.; Lampien, J.A., "Differential Evolution - A Practical Approach to Global Optimisation", Natural Computing Series, Springer, 2005
(7) Das, S., Abraham, A., Chakraborty, U.K., Konar, A. "Differential Evolution Using a Neighborhood-based Mutation Operator", IEEE Transactions on Evolutionary Computation, IEEE Press, USA, Volume 13, Issue 3, pp. 526-553, 2009.
(8) Denton, J.D., "Loss mechanisms in turbomachines", The 1993 IGTI Scholar Lecture, Trans. ASME, Journal of Turbomachinery, October, Vol. 115, pp. 621-656. 1993