

2

Tech Memo
P 1205

UNLIMITED

Tech Memo
P 1205

AD-A241 597



Technical Memorandum

April 1991

DTIC
SELECT
C

Predictions and Measurements of 3D Viscous Flow in
a Transonic Turbine Nozzle Guide Vane Row

by

G. C. Horton
S. P. Harasgama
K. S. Chana

91-12271

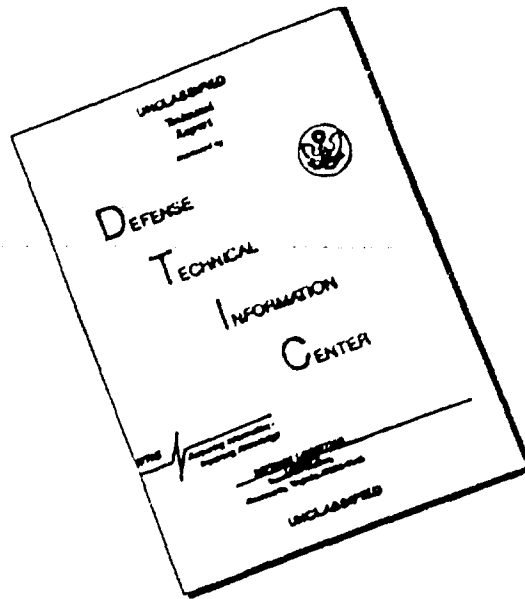


RAE, Farnborough, Hampshire

01 10 91

UNLIMITED

DISCLAIMER NOTICE



THIS DOCUMENT IS BEST
QUALITY AVAILABLE. THE COPY
FURNISHED TO DTIC CONTAINED
A SIGNIFICANT NUMBER OF
PAGES WHICH DO NOT
REPRODUCE LEGIBLY.

1416272

CONDITIONS OF RELEASE

108827

MR PAUL A FOBEY
SFC
Aircraft PDAC
Cement Station-Stop 5
Alexandria
12 22304 5145
USA

DRC J

COPYRIGHT to
1988
CONTROLLER
HMSO LONDON

DRC Y

Reports quoted are not necessarily available to members of the public or to commercial organisations.

UNLIMITED

DEFENCE RESEARCH AGENCY

Aerospace Division
RAE Farnborough

Technical Memorandum P 1205

Received for printing 25 April 1991

Submission for	
Final Check	<input checked="" type="checkbox"/>
Final Test	<input type="checkbox"/>
Approved	<input type="checkbox"/>
Justification	
By	
Distribution	
Availability Codes	
Availability	
Dist	Special

A-1

PREDICTIONS AND MEASUREMENTS OF 3D VISCOUS FLOW IN A
A TRANSONIC TURBINE NOZZLE GUIDE VANE ROW

by

G. C. Horton

S. P. Harasgama

K. S. Chana



SUMMARY

In recent years improvements in algorithms and computing power have allowed the regular use of three-dimensional viscous flow programs to analyse the flows in turbines. Before these programs can be used with confidence by the turbine designer, however, they must be validated by comparison with high quality experimental data taken at realistic conditions.

A transonic turbine nozzle guide vane has been tested in an annular cascade with two different end-wall geometries. The measurements were taken at engine-representative flow conditions and include surface static pressures and a downstream area traverse of total pressure.

The flow through these geometries has been modelled at the test conditions using a three-dimensional viscous flow program. The effects of different mesh densities and two turbulence models have been studied. Predictions of secondary flow and loss have been obtained and are compared with the experimental measurements.

This Memorandum is a facsimile of a paper to be presented at the 77th Symposium of the AGARD FEP, San Antonio, Texas, May 1991.

Copyright

©

Controller HMSO London
1991

UNLIMITED

LIST OF CONTENTS

	<u>Page</u>
1 INTRODUCTION	3
2 THE NGV FASTEN	3
3 THE HLPC FACILITY	3
4 ANALYSIS METHOD	4
5 MESH COMPARISON	4
6 TURBULENCE MODEL COMPARISON	5
7 COMPARISON WITH EXPERIMENT	5
8 COMPARISON OF ENVELOPE PROFILES	5
9 CONCLUSIONS	6
References	6
Illustrations	Figures 1-16
Report documentation page	inside back cover

S.A. Gorton, S.E. Braggins, K.S. Kana
 Royal Aerospace Establishment, Farnborough
 Southampton, Hampshire
 G2

SUMMARY

In recent years improvements in algorithms and computing power have allowed the regular use of three-dimensional viscous flow programs to analyse the flow in turbines. Before these programs can be used with confidence by the turbine designer, however, they must be validated by comparison with high quality experimental data taken at realistic conditions.

A transonic turbine nozzle guide vane has been tested in an annular cascade with two different endwall geometries. The measurements were taken at engine-representative flow conditions and include surface static pressures and a downstream area traverse of total pressures.

The flow through these geometries has been modelled at the test conditions using a three-dimensional viscous flow program. The effects of different mesh densities and two turbulence models have been studied. Predictions of secondary flow and loss have been obtained and are compared with the experimental measurements.

1. INTRODUCTION

The use of three-dimensional viscous flow programs to analyse the flow in turbine machinery blade passages has become widespread in recent years. In general these methods are currently used to analyse existing designs or to check specific features of a new design in the final stages of the design process. If these methods are to realise their full potential for radically improving turbomachinery design they must be used much earlier in the design process. This requires the designer to have confidence in the capability of the program and also a good understanding of the effect of the available options, such as mesh size and density, or different turbulence models.

To help prove this confidence, a turbine nozzle guide vane (ngv), designed for transonic flows, has been tested in annular cascade form with two different endwall shapes in the Isentropic Light Piston Cascade (ILPC) facility at RAE Farnborough. The same nozzle geometry has been analysed using a three-dimensional viscous flow program with different meshes and turbulence models. This paper reports the results of these computations and makes comparisons with the experimental results.

2. THE NGV DESIGN

The ngv tested was designed at the Royal Aerospace Establishment as part of a high work capacity, high blade speed turbine known as the High Rim Speed Turbine. The design philosophy and test performance have been

described in a previous AGARD paper (Ref 1) and blading aerodynamic and heat transfer measurements in Ref 2. The blade design is illustrated in figure 1. The design features a variation of exit flow angle along the span and a curved tangential stack of the trailing edge to control secondary flows by setting up spanwise pressure gradients. The design condition the exit Mach number is 1.65 and the Reynolds number (based on exit conditions and axial chord) is 1.75×10^6 . The original ngv was designed for cylindrical endwalls, but as part of an investigation of potential improvements to the turbine efficiency, profiled hub shapes were also considered.

The blading design has been tested with two of these endwall shapes, which are here termed the bellmouth and S-bend profiles (see figure 1). The aim of these endwall shapes was to reduce the secondary effects by re-energising the flow near the endwall. The distribution of curvature, and hence the local acceleration provided to the flow, varied between the two designs.

3. THE ILPC FACILITY

The Isentropic Light Piston Cascade, or ILPC, (Ref 3) is a short duration facility designed to allow high quality heat transfer and aerodynamic measurements to be taken for a full-size annular cascade of turbine vanes. Its design and operation has been described in an AGARD paper presented in 1985 (Ref 3). A major extension of the facility, to enable heat transfer data to be taken from a rotating rotor mounted downstream of the nozzle row, has been designed and will be installed in 1991. The aerodynamic conditions upstream of the cascade, to match the engine parameters of exit Mach number, Reynolds number and gas-to-wall temperature ratio, are obtained by the isentropic compression of air in a large pump tube. This compression is performed by a free piston driven along the tube by high pressure air. When the correct conditions are achieved, a fast-acting plug valve opens allowing the air to flow through the cascade giving steady operating conditions for approximately 0.5 second, during which time aerodynamic and heat transfer data can be acquired.

For this series of tests the instrumentation consisted of static pressure tappings at a total of 188 locations on the vane surfaces and endwalls. An area traverse of total pressure was performed on a plane 0.2 axial chord downstream of the trailing edge. This was built up from a series of 27 circumferential sweeps at different radii, each sweep requiring a separate run of the ILPC facility. The circumferential extent of a sweep was approximately 1.5 vane pitches. As it was desired to traverse through two vane wakes on each sweep and the vane wakes were

significantly curved, regular adjustment of the location of the traverse mechanism was necessary to produce a full-area traverse. As a result, producing the area traverse took a considerable time and so was done for only one endwall profile, the bellmouth profile. Results reported previously for the cylindrical endwall geometry (Ref 4) were obtained with a three-hole probe with high-response transducers mounted near the tip. These results suffered from a high degree of noise, much of it aerodynamic. For the current series of traverses a modified probe configuration was used. This configuration had three transducers with lower frequency response mounted 500 mm from the probe head. This distance between the probe head and the transducers appears to damp out the aerodynamic noise effectively while providing enough response to capture adequately the wake wakes. The natural frequency of the system is now approximately 400 Hz.

The manner of operation of the traverse mechanism presented some difficulties when measuring near the hub. To overcome these difficulties two different probe tips were used: a straight tip for the traverses above mid-height and a "swan-necked" tip for those below (figure 2). By using these two probe tips it was possible to traverse to within 7.5% wake height of both endwalls.

4. ANALYSIS METHOD

The method used to analyse the flow is the three-dimensional viscous flow program of Dawes (Ref 5). This solves the unsteady Reynolds averaged Navier Stokes equations for a steady solution using an implicit time marching algorithm. Turbulent stresses are modelled using an eddy viscosity concept. Two formulations of eddy viscosity are available: an algebraic method due to Baldwin and Lomax (Ref 6) and a one-equation method patterned after that of Birch (Ref 7). The latter model includes an additional transport equation for turbulent kinetic energy which is solved using a space-marching algorithm. In the Baldwin-Lomax model laminar-turbulent transition is modelled by specifying locations for the start and end of transition on both suction and pressure surfaces. A value of intermittency is then calculated for each mesh point by linear interpolation between these specified transition locations. For the current series of calculations transition on the suction surface was set to start at 25% axial chord (Cax) and end at 60% Cax. The corresponding figures for the pressure surface were 40% and 90%. The one-equation method models transition through a low Reynolds number damping term.

Spatial discretisation is achieved using a sheared, cell-centred H-mesh. Two different densities of mesh were employed; calculations with meshes 1 and 2 used 25 points circumferentially, 25 radially and 89 axially, a total of 55625 mesh points, while those with meshes 3 and 4 used a 33 x 33 x 119 mesh (129591 mesh points). In each case the mesh was refined considerably near solid surfaces

and also near the leading and trailing edges. For the bellmouth endwall shape a limited investigation of mesh distribution was performed by varying the refinement at the leading and trailing edges for the same overall number of mesh points. Results from all the different mesh densities and distributions are presented.

The convergence of the program is accelerated using a multigrid algorithm. Previous experience indicated that good convergence was obtained after 2000 time steps and so this number was used for the results presented here. At this point the rms value of axial momentum residue was 2.6×10^{-4} on mesh 4. The program was run on a Stardent 1500 mini-supercomputer and 2000 time steps on mesh 4 required approximately 48 hours of cpu time.

5. MESH COMPARISON

The four meshes employed for modelling the bellmouth profile are compared in figures 3 and 4. The different mesh refinements at leading and trailing edges and near solid surfaces resulted in the mesh spacings shown in table 1. The endwall boundary layer at the first plane, one axial chord upstream of the leading edge was set to have a thickness of 6% annulus height. This resulted in 6 points in this inlet boundary layer for meshes 1 and 2, and 8 points for meshes 3 and 4.

The mid-height distributions of isentropic Mach number predicted with the different meshes are compared in figure 5. The main variations occur on the pressure surface between the leading edge and 60% axial chord (Cax) and on the suction surface at about 60% Cax. The latter feature corresponds to the impingement of the trailing edge shock on to the suction surface and the differences are probably due to the different modelling of the trailing edge flow with the various meshes. This shock is modelled best with mesh 4, which has the finest spacing at the trailing edge. Contours of Mach number near the trailing edge are illustrated for meshes 2 and 4 in figure 6. The pressure surface boundary layer upstream of the trailing edge is considerably thinner with the finer mesh leading to delayed separation and higher peak Mach number at the trailing edge. This produces the stronger shock evident where it impinges on the suction surface. On the forward pressure surface the isentropic Mach number is higher with the finer meshes than the coarser. The differences appear to be associated with the finer cross-stream mesh rather than the axial refinement, suggesting that they are due to the improved modelling of the pressure surface boundary layer as a result of the finer spacing of meshes 3 and 4 in this region.

Figure 7 compares the secondary flows near the suction surface for meshes 1 and 4. The manner in which the endwall crossflow was swept onto the suction surface can be clearly seen in both cases. The finer mesh (4) has

resolved two separate points of impingement onto the suction surface for each endwall at approximately 50% Cax and 65% Cax with the flow nearly parallel to the endwall between the two. The coarser mesh has modelled the first impingement very similarly to the finer mesh but has not picked up the second.

The development of mass-averaged loss along the passage is shown for each mesh in figure 8. The results from the finer meshes have a lower overall loss than those from the coarser meshes reflecting the improved modelling of the blade and endwall boundary layers with the finer meshes. Also they exhibit a more realistic growth of loss through the blade passage; the loss with the coarser meshes grows from the leading edge to about 30% Cax, then diminishes before growing again towards the trailing edge. At the trailing edge mesh 4 shows a very rapid growth of loss with little further growth downstream whereas the other results, particularly meshes 1 and 3, show less growth at the trailing edge but significant further growth downstream. This is attributed to the increased mixing of the flow at the trailing edge with mesh 4 due to the finer mesh spacing in this region. The total losses at the exit plane (1.0 axial chord downstream of the trailing edge) for meshes 1 and 4 are 0.081 and 0.052 respectively.

Contours of total pressure on a cross-stream plane at 120% axial chord are compared for all four meshes in figure 9. The distinctive curved wake is due to the curved trailing edge and the radial variation of exit angle. In each case the peak loss (minimum total pressure) occurs in a thick region around mid-height, the level of this peak loss being similar for all the meshes. Meshes 1 and 4 give narrower wakes than (respectively) meshes 2 and 3 suggesting that the finer mesh in the trailing edge region has changed the modelling of the flow in the mixing region downstream of the trailing edge. Also, the wake with mesh 4 is narrower than with mesh 1 indicating that the improved modelling of the suction and pressure surface boundary layers due to the finer near-wall mesh has resulted in thinner boundary layers being predicted at the trailing edge.

6. TURBULENCE MODEL COMPARISON

The above results were all produced using the Baldwin-Lomax turbulence model with the specified transition locations. Calculations have also been performed using the one equation turbulence model on meshes 1 and 4. The development of the mass averaged loss along the vane passage is compared for these meshes with both turbulence models in figure 10. There are only small differences between the results with the two turbulence models, particularly with the coarser mesh. This is consistent with the findings of Dawes (Ref 8) which he attributes to most of the loss being generated very close to solid surfaces, within a laminar sub-layer region which is common to both models.

7. COMPARISON WITH EXPERIMENT

The total pressure traces obtained from the centre hole of the three hole probe for the experimental sweeps at 10% and 50% vane height are shown in figure 11. A small amount of smoothing has been applied to the traces to remove some high frequency noise. The vane wakes can be seen clearly in both plots, the wake at 50% height being considerably wider than that at 10% height. Also evident are some oscillations in the traces between the wakes. These are due to fluctuations with time in the total pressure upstream of the vanes arising from piston velocity variations during the operation of the rig. Because of the manner in which the traverse is performed, a variation with time appears as a variation with position when the trace is plotted. These oscillations can be largely removed by further analysis to allow for the variation in upstream total pressure; the results at 10% and 50% height after this additional processing are shown in figure 12.

The individual experimental sweeps, after processing, have been combined to form a complete area traverse and contours of total pressure are shown in figure 13 together with the predictions using both turbulence models on mesh 4. The predictions both show wakes which are wider at mid-height than near the endwalls. The wake in the prediction with the one-equation model is wider (23% pitch at mid-height) than that with the Baldwin-Lomax model (20%); the experimental results show a wake which is slightly wider than either prediction (25%). The one-equation model predicts a similar deficit of total pressure in the centre of the wake to the experimental value but the Baldwin-Lomax model does not capture the full deficit. In common with the experimental results neither prediction has identifiable regions of secondary loss though the Baldwin-Lomax results do exhibit some thickening of the wake near the hub. The experimental results show greater curvature of the wake near mid-height and a more acute angle between the wake and the endwall, particularly near the tip, than either prediction, with again the one-equation model giving a slightly closer result.

8. COMPARISON OF ENDWALL PROFILES

Measurements of surface static pressure were taken for both the bellmouth and S-bend endwall profiles. These are shown on figure 14 in the form of isentropic Mach number distributions at 10% and 50% span, together with predictions for both endwall profiles, using mesh 1 spacings. At 10% span the S-bend hub profile has increased the Mach number on the early part of the suction surface and slightly increased the strength of the shock at approximately 70% Cax. The effect on the pressure surface is small. At 50% span there is very little difference between the two sets of experimental results, though the S-bend does still have a slightly higher Mach number over the early suction

surface. Each of these differences has been well modelled by the flow program, though neither prediction captures the full shock strength. The effect of the endwall profile on the predicted secondary flows is illustrated in figure 15 by the velocity vectors near to the suction surface. It appears that the passage cross-flow on the hub impinges on to the suction surface slightly further downstream with the S-bend profile but that it does so at a greater angle resulting in a greater spanwise extent of the vane being affected by the secondary flow. The effect on the losses is small as is shown by contours of total pressure on the plane at 120% Cax in figure 16. There is very little difference between the shape of the vane wakes or the total pressure deficits for either endwall profile. The total loss is 0.081 of exit dynamic head for the bellmouth profile and 0.084 for the S-bend.

9. CONCLUSIONS

A transonic turbine nozzle guide vane has been tested in an annular cascade facility with two different end-wall profiles. Vane surface static pressures have been measured and a traverse of the total pressure field downstream of the trailing edge has been performed for one of the profiles.

The new area traversing technique has given good results. It has been found necessary to process the results to remove the effect of piston oscillation.

The Dawes three-dimensional viscous flow program has been used to model the flow through the nozzle. A variety of mesh densities and distributions have been investigated as have two different turbulence models. Good agreement has been obtained for the pressure distributions and realistic secondary flows have been predicted. It was found that the finer meshes gave better predictions of the static pressure distributions and reduced levels of loss as a result of the improved modelling of the vane surface boundary layers.

The different turbulence models were found to have little effect on the predicted overall loss though there were differences in the distribution and the shape of the vane wakes. The one-equation turbulence model produced wakes which had similar levels of total pressure deficit to the experiment and had a more similar shape than those with the Baldwin-Lomax algebraic model.

REFERENCES

1. R. C. Kingcombe, J. D. Bryce and N. P. Leversuch, "Design and Test of a High Blade Speed, High Work Capacity Transonic Turbine", AGARD CP-421, 1987.
2. R. C. Kingcombe, S. P. Haragana, N. P. Leversuch and E. T. Wedlake, "Aerodynamic and Heat Transfer Measurements on Blading for a High Rim-Speed Transonic

Turbine", ASME Paper 89-GT-223.

3. A. J. Brooks, D. E. Colbourne, E. T. Wedlake, T. V. Jones, M. L. G. Oldfield, D. L. Schultz and P. J. Loftus, "The Isentropic Light Piston Annular Cascade Facility at RAE Pyestock", AGARD CP-390, 1985
4. G. C. Horton, "Secondary Flow Predictions for a Transonic Nozzle Guide Vane", AGARD CP-459, 1990.
5. W. N. Dawes, "A Numerical Method for the Analysis of 3D Compressible Flow in Turbine Cascades; Application to Secondary Flow Development in a Cascade With and Without Dihedral", ASME Paper 86-GT-145, 1986.
6. B. S. Baldwin and H. Lomax, "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows", AIAA Paper 78-257 1978.
7. N. T. Birch, "Navier-Stokes Predictions of Transition, Loss and Heat Transfer in a Turbine Cascade", ASME Paper 87-GT-22, 1987.
8. W. N. Dawes, "A Comparison of Zero and One Equation Turbulence Modelling for Turbomachinery calculations", ASME Paper 90-GT-303 1990.

Copyright

©
Controller HMSO London
1991

	Mesh 1	Mesh 2	Mesh 3	Mesh 4
Axial spacing at leading edge (% axial chord)	0.35	0.43	0.30	0.10
Axial spacing at trailing edge (% axial chord)	0.35	0.43	0.30	0.10
Spanwise spacing at endwall (% blade height)	0.35	0.49	0.24	0.24
Tangential spacing at blade surfaces (% pitch)	0.39	0.39	0.24	0.24

Table 1. Mesh spacings for different meshes

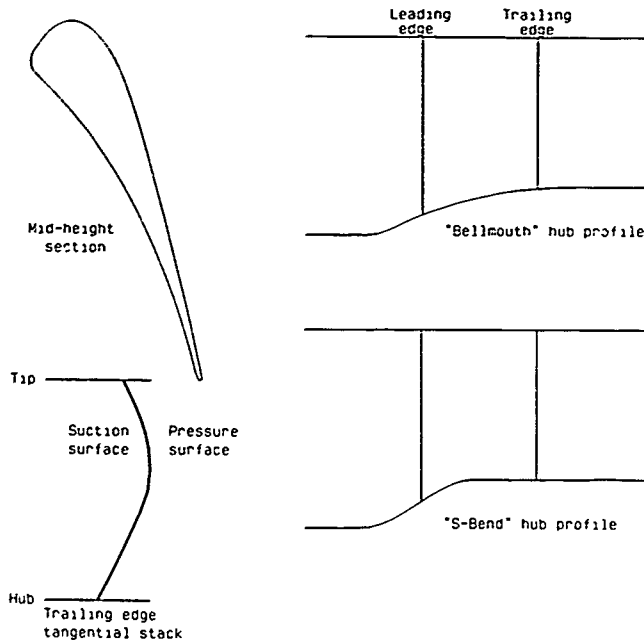


Fig 1 Illustration of ngv design

Fig 2&3

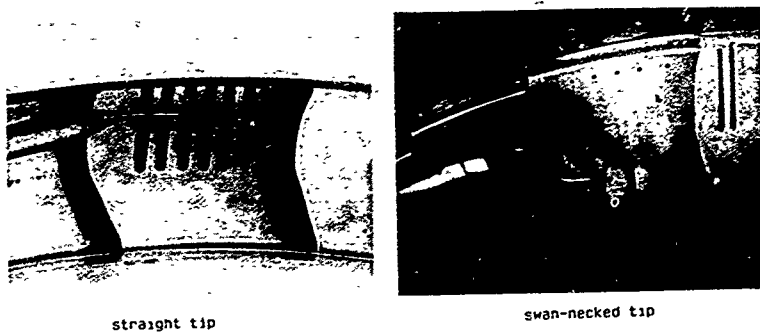


Fig 2. The two traverse probes

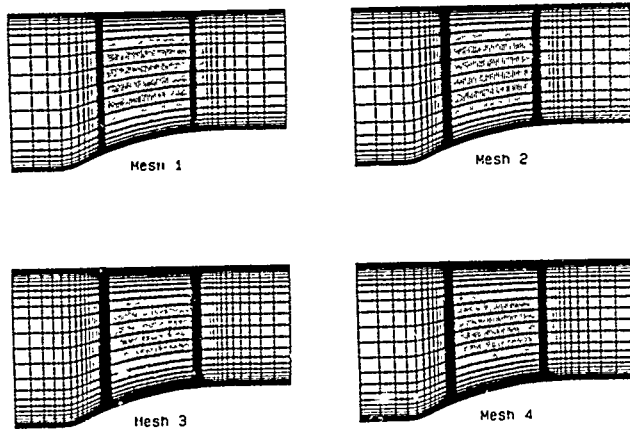
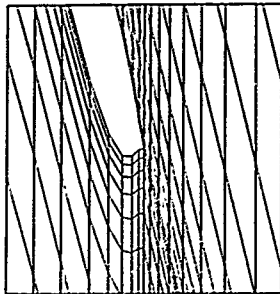
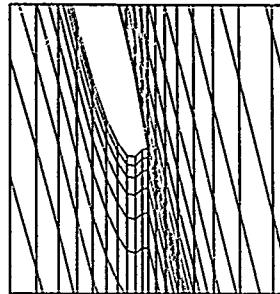


Fig 3 Details of calculation mesh

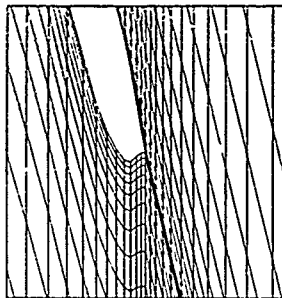
Fig 4



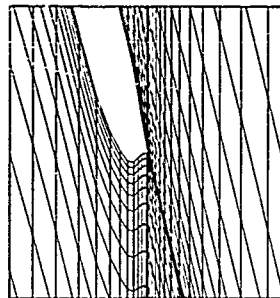
Mesh 1



Mesh 2



Mesh 3



Mesh 4

Fig 4 Details of mesh near trailing edge

Fig 5&6

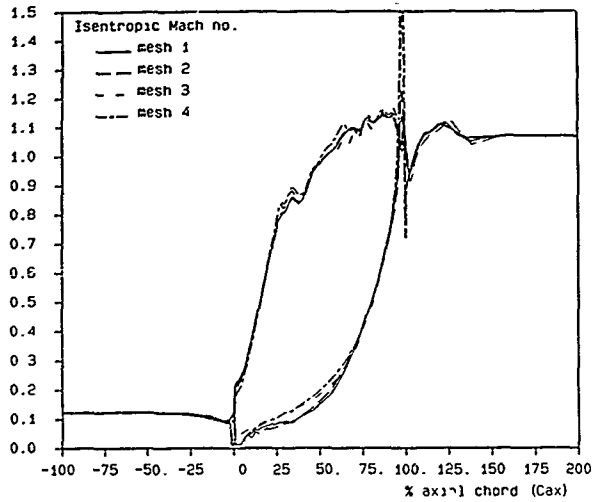


Fig 5 Comparison of mid-height isentropic Mach number distributions with different meshes

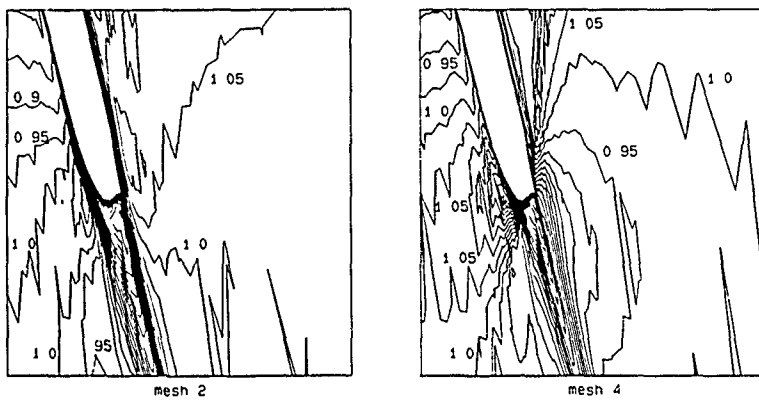


Fig 6 Contours of Mach number near trailing edge

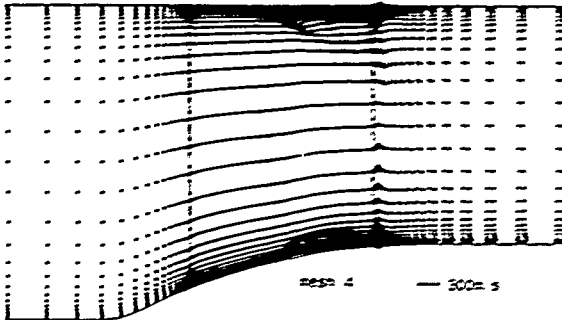
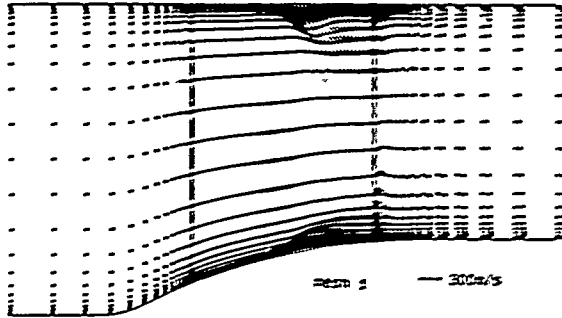


Fig 7. Suction surface flows with meshes 1 and 4

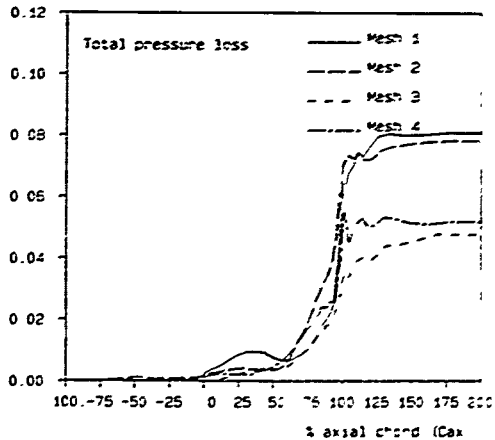


Fig 8 Development of total pressure loss with different meshes

Fig 9

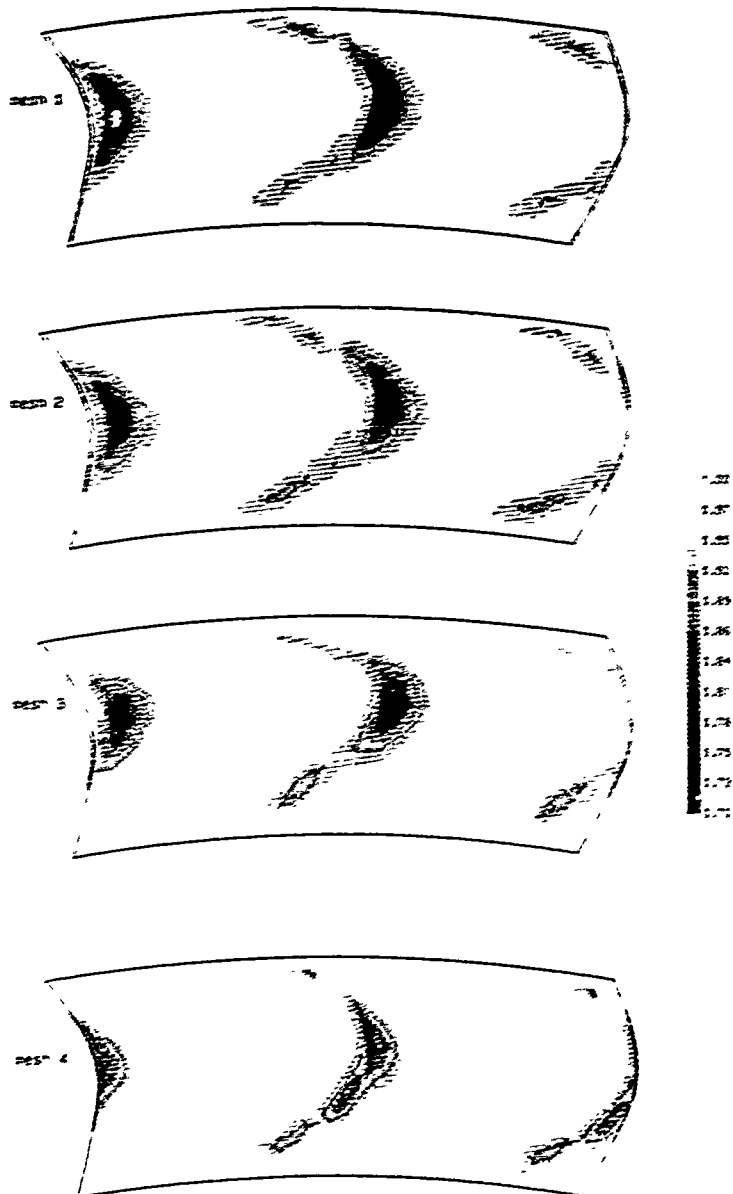


Fig 9 Contours of total pressure on a plane at 120% axial chord
Comparison of results with different meshes

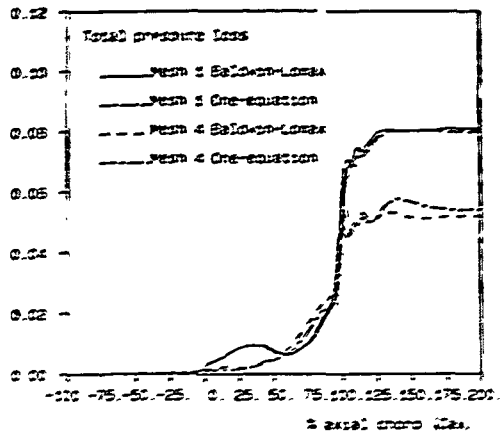


Fig 10. Development of total pressure loss with different turbulence models

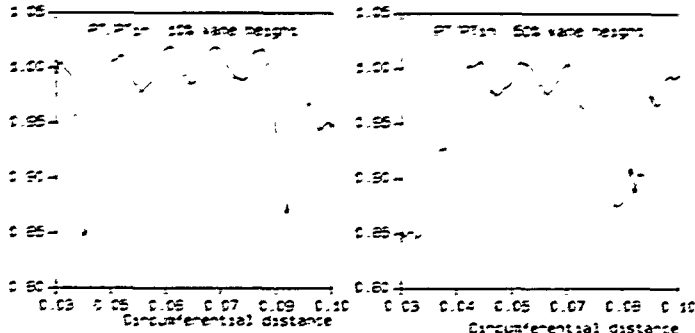


Fig 11. Total pressure traverse sweeps before processing

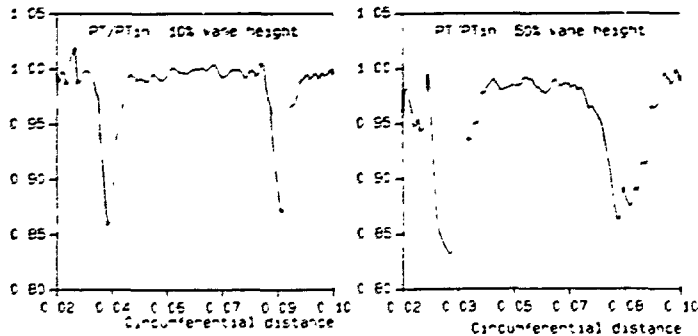


Fig 12. Total pressure traverse sweeps after processing

Fig 13

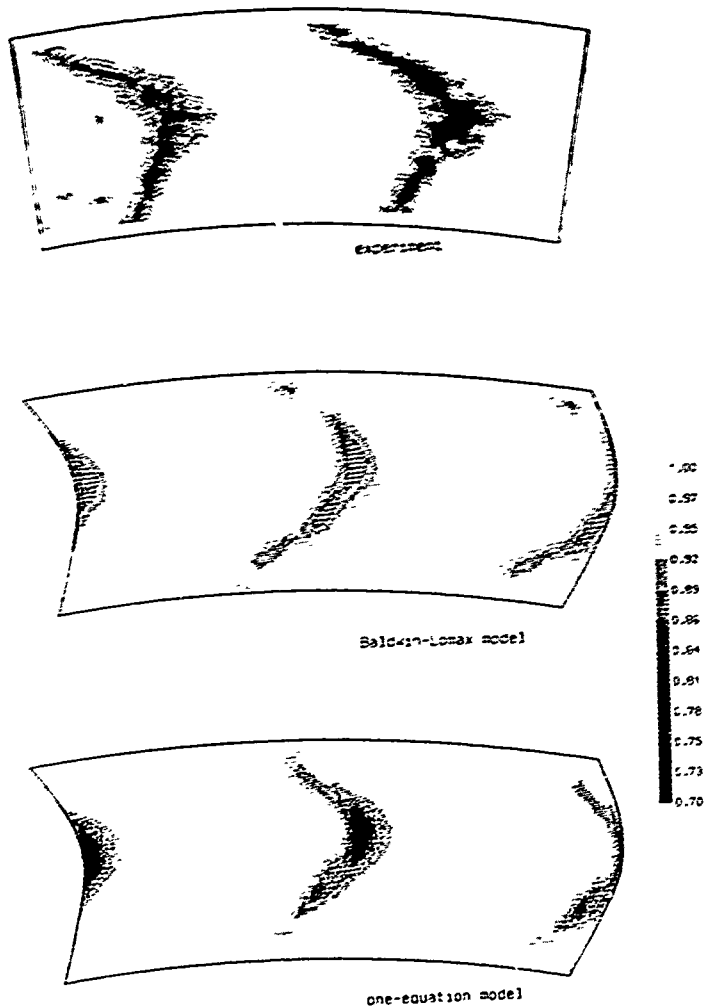


Fig 13 Total pressure contours on a plane at 120% axial chord
Comparison of different turbulence models with experiment

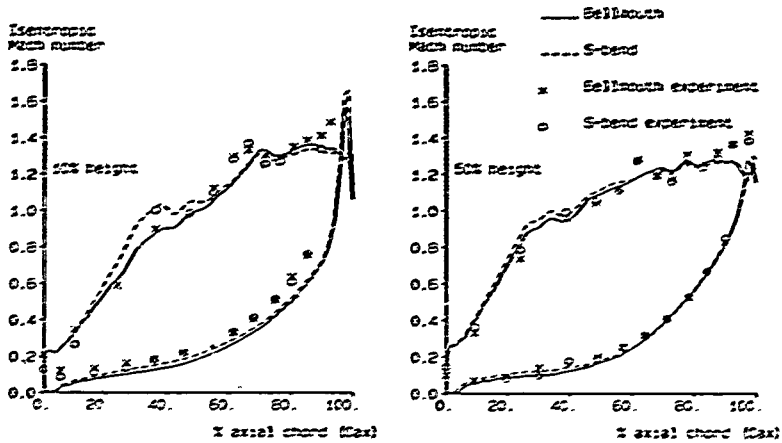


Fig 14. Mach number distributions with different end-wall profiles

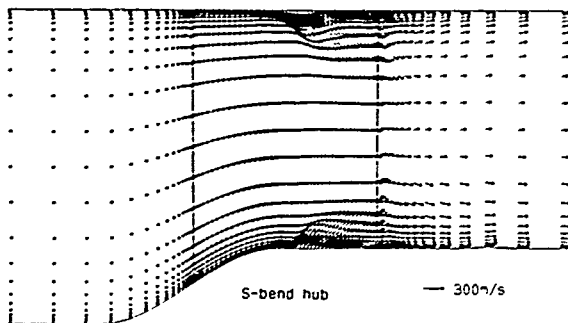
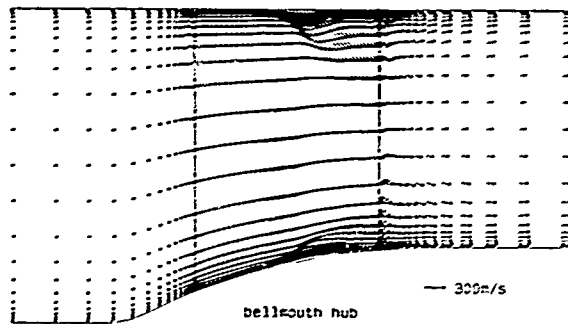


Fig 15 Suction surface flows with bellmouth and S-bend hub profiles

Fig 15

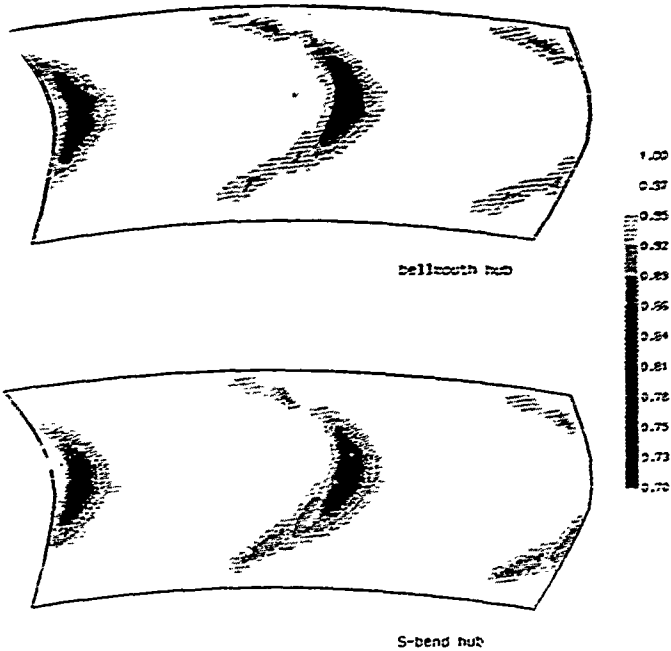


Fig 16 Total pressure contours on a plane at 120% axial chord
for bellmouth and S-bend hub profiles

REPORT DOCUMENTATION PAGE

Current security classification of this page

RESTRICTED

As far as possible this page should contain only essential information. If it is necessary to give any "red information", the form should be marked to indicate the classification, e.g. Restricted, Confidential or Secret.

1. DRIC Reference (to be added by DRIC)	2. Originator's Reference RAE TR P 4265	3. Agency	4. Report Security Classification/Marking RESTRICTED
5. DRIC Code (for Classification) 767A2002	6. Originator (Company's Author) Name and Location Royal Aerospace Establishment, Pyestock, Farnborough, Hants, UK		
5a. Sponsoring Agency's Code	6a. Sponsoring Agency (Contract Authority) Name and Location		
7. Title Predictions and measurements of 3D viscous flow in a transonic turbine nozzle guide vane row			
7a. (For Translations) Title in Foreign Language			
7b. (For Conference Papers) Title, Place and Date of Conference			
8. Author I. Surname, Initials Horton, G.C.	9a. Author Harasgama, S.P.	9b. Author's I. S. ... Ghana, K.S.	10. Date Pages Refs. April 1991 21 8
11. Contract Number	12. Period	13. Project	14. Other Reference Nos.
15. Distribution statement (a) Controlled by - (b) Special limitations (if any) - If it is intended that a copy of this document shall be released overseas refer to RAE Leaflet No.3 to Supplement 6 of MOD Manual 4.			
16. Descriptors (Keywords) (Descriptors marked * are selected from TEST) Turbines*. Viscous flow.			
17. Abstract In recent years improvements in algorithms and computing power have allowed the regular use of three-dimensional viscous flow programs to analyse the flows in turbines. Before these programs can be used with confidence by the turbine designer, however, they must be validated by comparison with high quality experimental data taken at realistic conditions. A transonic turbine nozzle guide vane has been tested in an annular cascade with two different endwall geometries. The measurements were taken at engine-representative flow conditions and include surface static pressures and a down-stream area traverse of total pressure. The flow through these geometries has been modelled at the test conditions using a three-dimensional viscous flow program. The effects of different mesh densities and two turbulence models have been studied. Predictions of secondary flow and loss have been obtained and are compared with the experimental measurements.			

11/10/10

END

FILMED

DATE:

10-91

DTIC