Toward CFD-Based Correlations for Single-Stage High-Pressure Transonic Turbine Stage

IYAD AKEL^{1,*}, WAGDI G. HABASHI^{2,†}, and HANY S. MOUSTAPHA^{3,‡}

¹Senior Aerodynamics Engineer, Newmerical Technologies, Inc., 680 Sherbrooke Street West, 7th floor, Montreal, Quebec, Canada H3A 2M7; ²Director of the Computational Fluid Dynamics Laboratory, Department of Mechanical Engineering, McGill University, Montreal, Quebec, Canada; ³Manager of P & WC Technology, Technical Education and Collaboration Programs, Pratt & Whitney Canada, Longueuil, Quebec, Canada

Correlations are of great importance in the preliminary design of a gas turbine engine. They are useful to determine the efficiency and thus the Specific Fuel Consumption (SFC) of the engine, before even defining blade geometries. Also, correlations play a very important role in the understanding of the behavior (performance) of the engine, under different operating conditions.

Since correlations are obtained from expensive rig tests and cascades, and since cascades cannot represent all situations in an actual engine, the necessity of finding more realistic and cost-effective representations of the actual flow phenomena is becoming more important.

A novel approach would be to use CFD as the experimental test cell to generate such correlations and even to extend their limited regime of applicability. In this paper, using a 3-D finite element viscous, compressible, turbulent code, CFD-based correlations for a high-pressure transonic turbine have been created and validated against Cold Flow Turbine Rig test data. This has been done by studying the effect of changing tip clearance, blade speed, stage pressure ratio and vane stagger angle on the performance characteristics of a turbine stage.

Keywords: CFD; Navier-Stokes; FEM; Turbine correlations; High-pressure transonic turbine

In the aerodynamic design of an efficient turbine, a good knowledge of its change in performance at different operating conditions is extremely important. This kind of

information can be obtained from correlated experimental results of many test cases run on turbines. Correlations are used to predict the efficiency of the turbine (and thus, the SFC of the engine). They are also useful in evaluating the behavior of an engine under varying conditions. They assist in predicting aerodynamic losses and in understanding the effect of design changes and off-design conditions on the efficiency of a power plant.

While Computational Fluid Dynamics (CFD) is supplanting such methods in the 3-D aerodynamic design of components, the fact is that correlations will still be needed for a long time to come for the 1-D (mean-line) and 2-D (Through-Flow) preliminary analysis of turbomachines. Such 1-D and 2-D design steps remain an essential feature of the overall design process of a turbomachine.

Correlations require extensive and costly experiments. If carried out on actual engines, this would be a prohibitively expensive proposition. Thus, most test data used for the creation of correlations are obtained from Cascade or Cold Flow Turbine Rigs (CFTR). Unfortunately, cascades predictions are not free from experimental errors, and they apply to cascades rather than to the actual turbines. The flow in a turbine, however, is much more complex than in a cascade. The real situation is one of three-dimensional flow, high radial pressure gradients, centrifugal and Coriolis forces, and unsteadiness and high turbulence levels.

Thus, the necessity of using less expensive tools with more comprehensive capabilities is very important. The tool that could complement and perhaps replace experiments at much lower cost is Computational Fluid Dynamics (CFD). This fact is becoming more true with time due to the significant progress and affordability of computational tools. While CFD has its own source of errors that should not be ignored, and its accuracy is dependent on the model complexity and the way the boundary conditions are applied, new approaches exist to guarantee its fidelity (Habashi et al., 1998).

Received in final form 2 March 2001.

*Corresponding author. Tel.: +1 (514) 398-2671, Fax: +1 (514) 398-8454. E-mail: iyad.akel@newmerical.com

†Tel.: +1 (514) 398-3747, Admin. Asst: 6005, Fax: 2203. E-mail: wagdi.habashi@mcgill.ca

‡Tel.: +1 (450) 647-7593, Fax: (450) 647-2506. E-mail: hany.moustapha@pwc.ca

This paper represent a joint effort between Pratt & Whitney Canada (P & WC), and Concordia's CFD laboratory to use 3-D unsteady viscous analysis codes for multistage turbomachines to:

- Confirm and verify existing correlations used in the gas turbine industry,
- Then build correlations outside the envelope of the existing experimental set-ups, thus edging towards a real-time numerical test cell.

SELECTED TEST CASES

The turbine analyzed in this work is a high-pressure transonic turbine, and it has the characteristics listed in Table I.

The selection of a limited number of test cases was made to gauge the ability of a 3-D viscous flow analysis code (NS3D) to reproduce the effect of some mean line design parameters and off-design parameters. The selection was based on the availability of experimental data, the ease of preparing the meshes and setting up the test cases, and most importantly, the usefulness of the chosen parameters to test the code and yield useful design correlations.

Four parameters were selected in order to study their effects on the performance characteristics of a high-pressure turbine stage:

- Tip clearance/span
- Blade speed (rpm)
- Stage pressure ratio
- Vane stagger angle.

TABLE I Turbine stage characteristics and boundary conditions

P.R.	4.04
$\Delta H/U^2$	1.83
C_x/U	0.64
Inlet total pressure (Pa)	96530
Inlet total temperature ($^{\circ}\text{K}$)	427.78
Inlet Mach number	0.15
Exit Mach number	0.5
Exit swirl (degrees)	16.8
$m\sqrt{Tt1/Pt1}$	4.98
$N/\sqrt{Tt1}$	668
Blade $Re \times 10^{-3}$	63
$AN^2 \times 10^{-10}$	7.9
Vane count	15
Blade count	70
Vane aspect ratio	0.5
Blade aspect ratio	2
Vane % TE blockage	6.8
Blade % TE blockage	9.3
Clearance/span (%)	1.5
α_1 (degrees)	0
η (%)	84.5

CFD SET-UP

Figure 1 shows the computational geometry of the vane and the blade. It is seen that the blade has a noticeably long downstream passage, due to the fact that the Inter-Turbine Duct (ITD) is included in the analysis, as was done in the experiments.

The same hexahedral, structured, grid is used for all the test cases, except the tip clearance test cases, since it is variable. Despite this, the number of nodes and the number of elements are kept the same. Table II shows a summary

FIGURE 1 Vane and blade geometry (See Colour Plate at back of issue.).

TABLE II Mesh details

	Vane	Blade	Blade tip	Total
Total number of I planes	125	145	49	319
L.E. at I plane	43	27		—
T.E. at I plane	91	75		—
Total number of J planes	27	27	5	59
Total number of K planes	35	30	6	71
Total number of nodes	118125	137025	1470	256620

of the three meshes used for each of the vane, the blade and the blade tip regions.

The 3-D viscous, compressible code used NS3D, solves the three-dimensional, Reynolds-averaged, Navier-Stokes equations. The steady version of the code is used with a $\kappa-\omega$ turbulent model.

EFFECT OF TIP CLEARANCE (CLEARANCE/SPAN) ON PERFORMANCE CHARACTERISTICS OF HIGH-PRESSURE TURBINE STAGE

Understanding the flow structure through the tip gap of axial flow turbomachinery blade rows is very important, since it may have interrelated adverse effects on the performance of the engine. The losses associated with the leakage flow account for a significant fraction of the losses in the endwall region. Also, leakage alters the blade loading in the tip region and thus affects the flow turning ability of the blade.

NS3D was used to analyze the effect of changing tip clearance on the performance of the HPT. Figure 2 plots the variation of turbine stage efficiency versus % blade tip

clearance/span. The results of NS3D and turbine correlation are shown. It is noticed that NS3D compares reasonably well with the mean line results. Both curves agree on the fact that a significant decrease in efficiency results from any increase of the tip clearance for the range shown.

Figure 3 represents NS3D predictions for vane mid span Mach number distribution. It shows that the change of the size of the tip clearance gap has no significant effect on the loading of the vane in the mid-span region. This is also the case for the vane loading in the tip region, as can be seen in Figure 4. Figure 5a shows that as the tip clearance increases, the blade loading at the LE area increases, whereas it decreases at the TE area.

To examine the effect of the tip leakage vortex on the blade loading, the blade surface Mach number distribution is plotted at different regions close to the tip in Figure 5. From this figure, it is clearly noticed that the no-tip-clearance case has practically no change in loading, whereas the other cases show an overall increase in loading. More specifically, the loading moves toward the TE, but no common behavior was found. This is probably due to the fact that the size, the strength and the number of the tip leakage vortices are different from one case to another (Yaras & Sjölander, 1989).

FIGURE 2 NS3D predictions for the variation of stage efficiency vs. blade tip clearance.

FIGURE 3 NS3D predictions for vane mid-span Mach number distribution, for all the tip clearances (See Colour Plate at back of issue.).

FIGURE 4 NS3D predictions for vane Mach number distribution at 99% span, for all the tip clearances (See Colour Plate at back of issue.).

(a)

(b)

FIGURE 5 NS3D predictions for blade Mach number distribution at different span locations, for all the tip clearances. (a) Blade loading at mid span; (b) Blade loading at 86% span; (c) Blade loading at 96% span (See Colour Plate at back of issue.).

FIGURE 5 (Continued) (See Colour Plate at back of issue.).

FIGURE 6 Views of the tip leakage vortex, as predicted by flow solver for the three blade tip clearances (See Colour Plate at back of issue.).

In order to understand the characteristics of the tip leakage vortex, the power of the CFD tool is put to the test. Figure 6 shows the simulation of the tip leakage

vortex for the three tip clearances tested. As expected, a good examination of the figures confirms the fact that as the tip clearance gap increases, the vortex

FIGURE 7 NS3D predictions for averaged stage reaction, for all the tip clearances.

becomes bigger and stronger another (Yaras & Sjölander, 1989).

The degree of reaction of the turbine stage is calculated and shown for different tip clearances in Figure 7. It is noticed that while no significant change is found in the stage reaction for the four tip clearances modeled, a relatively lower reaction is found for the 2.5% tip clearance case.

EFFECT OF BLADE SPEED (RPM) ON PERFORMANCE CHARACTERISTICS OF HIGH-PRESSURE TURBINE STAGE

An aircraft engine has a wide operating envelope (take-off, climb, cruise, descent and landing), and thus does not always run at its design speed. Also, engine deterioration and variation in atmospheric conditions are other important reasons causing the engine not to run at its design speed. This has an important impact on the engine (turbine) efficiency, and thus on the fuel consumption. All of that makes it necessary to understand the effect of variations of blade speed on the efficiency of the turbine.

The 3-D predictions of NS3D, as well as turbine correlations and experimental data are presented in Figure 8. This graph plots the change in efficiency, normalized with respect to the design point value, versus blade speeds at constant pressure ratio. The graph shows that for speeds below the design speed, all three curves agree on the fact that efficiency drops significantly. Whereas for speeds above the design value, a continuous increase in efficiency is shown by the experimental and NS3D curves. Turbine correlations show peak efficiency at 110% design speed, and then a decrease in efficiency is noticed for the 120% design speed case. Comparison of

these results with those presented by Vlasic et al. (1996) and Woinowsky-Krieger et al. (1998) shows similar trends.

These results demonstrate the impact of running below the design speed (ex: running 5% under the design speed results in around 1% decrease in stage efficiency, see Figure 8).

From NS3D results, surface Mach number distribution plots for the vane and blade were obtained at mean radius for the different speeds, and they are shown in Figures 9 and 10. These plots show that the variation of blade speed, which had caused changes in the flow conditions on the blade, has resulted in some variations in the vane back pressure which in turn affected the Mach number profile in the vane exit area. This is well depicted in Figure 9.

It has been found that the vane exit Mach number varies from 1.24 at 70% $(N/\sqrt{T})_{des}$ down to 1.14 at 120% $(N/\sqrt{T})_{des}$. Furthermore, Figure 10 shows that the variations in blade speed had a severe impact on the blade loading. NS3D predicted significant incidence changes in the blade leading edge area, which were found to be for the 70% $(N/\sqrt{T})_{des}$, 10.6° increase in incidence from 100% $(N/\sqrt{T})_{des}$, and a drop of 13.7° for the 120% $(N/\sqrt{T})_{des}$ from 100% $(N/\sqrt{T})_{des}$. This variation of incidence had a pronounced effect on the Mach number in the inlet region. The inlet peak Mach number reaches a value as high as 1.88 (for the 70% $(N/\sqrt{T})_{des}$). Whereas for the 100% $(N/\sqrt{T})_{des}$ speed and higher, the inlet Mach number was completely subsonic. The drop in blade speed had also resulted in a significant increase of Mach number in the exit region. The peak exit Mach number is found to drop from 1.68 at 70% $(N/\sqrt{T})_{des}$ to around 1.4 at 120% $(N/\sqrt{T})_{des}$. This drop in Mach number gives a reasonable explanation of the increase of stage efficiency for the speed range studied.

FIGURE 8 Flow solver predictions for the variation of stage efficiency vs. blade speed.

FIGURE 9 NS3D predictions for vane mid-span Mach number distribution, for all blade speeds (See Colour Plate at back of issue.).

FIGURE 10 NS3D predictions for blade mid-span Mach number distribution, for all blade speeds (See Colour Plate at back of issue.).

FIGURE 11 NS3D predictions for averaged stage reaction, for all blade speeds.

Figure 11 shows predicted stage reaction versus speed. From this figure it is noticed that the reaction is minimum near 80% $(N/\sqrt{T})_{des}$, thereafter, increasing on either side with a steeper rising slope on the low speed side.

EFFECT OF PRESSURE RATIO (PR) ON PERFORMANCE CHARACTERISTICS OF HIGH-PRESSURE TURBINE STAGE

As mentioned previously, since a gas turbine engine operates at different conditions, thus, the components of the engine such as the turbine will have a different operating condition (such as the stage pressure ratio) than the design one. That makes the understanding of the effect of running the turbine at an off-design PR a very important step in the process of predicting the engine overall performance.

NS3D has been used to analyze the effect of variation of the turbine stage pressure ratio. Figure 12 plots the change of efficiency normalized with respect to the design point value versus a range of PR at a constant blade speed (the design speed). Despite the fact that the range of PR studied is very large, the prediction of NS3D gives reasonable results for PR higher than the 90% PR_{des} . For PR lower than this value, however, the code is less sensitive to the variation of PR than are the experimental data. The figure also shows that NS3D predictions compare more favorably to the test results than those obtained by the turbine correlation predictions. This figure reveals the fact that for any increase or decrease of the stage pressure ratio from the design value (100% PR_{des}), the efficiency will drop, with this drop in the stage efficiency being more important for high-pressure ratios.

Figures 13 and 14 present the predicted NS3D vane and blade Mach number distributions for various PR at the

design speed. Figure 13 shows that the vane is choked. Therefore, the vane exit swirl is practically constant and the exit Mach number is varying slightly. The vane exit Mach number is found to vary from 1.12 at 64.5% PR_{des} up to 1.17 at 76.3% PR_{des} , whereas, it is practically constant at 1.18 for 94% PR_{des} and higher.

Thus, since the vane is choked, the inlet conditions to the blade remain almost unchanged (particularly for the 94% PR_{des} and higher cases), and as seen in Figure 14, all the additional loading resulting from the increase in PR is taking place entirely on the uncovered suction surface.

Figure 15 plots the variation of the blade inlet and exit relative swirl. It indicates clearly that the variation in incidence on the blade is very small. It also shows that the blade exit relative swirl drops from its peak at 94% PR_{des} for any change in PR, but it is more significant for PR higher than 94% PR_{des} . This is very well translated into the big variation of the blade exit Mach number as seen in Figure 14. The peak Mach number, in the blade exit area is found to vary from 1.1 at 64.5% PR_{des} to around 1.7 at 113% PR_{des} . For the case of 121% PR_{des} , the very high value of the peak exit Mach number found (around 2.6 from Figure 14) is believed to be too high and thus non-physical. It is most probably due to the mesh skewness in that region, so as PR increases, the over-shoot in Mach number becomes more significant, and thus, non-physical.

The prediction of the stage mean reaction is shown in Figure 16. The reaction increases from 11% for 64.5% PR_{des} to around 28% for 121.3% PR_{des} . This figure shows that PR has a significant effect on reaction, and it is more important than that of the speed (as seen in the previous section). This is due to the fact that at very low PR's, the vane takes up most of the available static pressure drop. For higher PR's, however, the vane becomes choked, and, therefore, the blade increases its static pressure drop, which yields an increase of the reaction.

FIGURE 12 Flow solver predictions for the variation of stage efficiency vs. stage pressure ratio.

FIGURE 13 NS3D predictions for vane mid-span Mach number distribution, for the six PR's studied (See Colour Plate at back of issue.).

FIGURE 14 NS3D predictions for blade mid-span Mach number distribution, for the six PR's studied (See Colour Plate at back of issue.).

FIGURE 15 NS3D predictions for blade in and out relative swirl, for the range of stage pressure ratio studied.

FIGURE 16 NS3D predictions for average stage reaction, for the range of stage pressure ratio studied.

EFFECT OF VANE STAGGER ANGLE ON PERFORMANCE CHARACTERISTICS OF HIGH-PRESSURE TURBINE STAGE

The stagger angle is the angle between the airfoil axial chord and the airfoil chord. In this chapter, the term “closing” the vane is used to indicate that the vane stagger angle is increased, and therefore closing the throat

(reducing the throat area), from which the term “closing” comes.

Experience has shown that manufacturing in general introduces some imperfections in the stacking of the airfoils of the vane (and in the blade) and thus the stagger angle. So, some testing and adjustment to the stagger angle has to be done before sending the component to a full production. The design parameter used as a criteria to determine if the

obtained vane is matching the requirement of the design, is known as the 'Flow Capacity' (Q), which is represented by $m\sqrt{Tt1}/Pt1$.

Thus, the stagger angle of the vane has an important impact on the performance of the turbine stage, since it influences the inlet Mach number to the blade and certainly the incidence. As will be discussed later, the losses in a turbine stage will vary (increase or decrease) for even a minimal change in the vane stagger angle.

In order to study the effect of the change of the vane stagger angle, four tests have been conducted. The first two

are without closing the vane and at two different blade speeds (70% $(N/\sqrt{T})_{des}$ and 100% $(N/\sqrt{T})_{des}$) and the other two were with a closed vane for the same blade speeds of the first two cases. Thus, the effect of the speed could also be studied.

With the vane stagger angle closed by only 0.8° , interesting results are found. NS3D predictions of the stage isentropic efficiency for the four test cases are plotted in Figure 17 (Due to the lack of experimental data from the CFTR used, only the results of NS3D are presented). The code predicts that by closing the vane by only 0.8° , a slight

FIGURE 17 NS3D predictions of stage efficiency for the effect of closing vane stagger angle by 0.8° .

FIGURE 18 NS3D predictions for vane mid-span Mach number distribution for the two speeds and vane stagger angles (See Colour Plate at back of issue.).

increase in stage efficiency was noticed for the 70% $(N/\sqrt{T})_{des}$, whereas, a significant drop in efficiency was detected for the 100% $(N/\sqrt{T})_{des}$.

Figure 18 shows that closing the vane results in a higher vane exit Mach number for the two different blade speeds. For the 70% $(N/\sqrt{T})_{des}$, the vane exit Mach number has increased from 1.21 to 1.24 whereas

it has increased from 1.15 to 1.18 for the 100% $(N/\sqrt{T})_{des}$. The change of the blade incidence resulting from closing the vane is: an increase of around 1.1° for the 70% $(N/\sqrt{T})_{des}$ and an increase of 2.2° for the 100% $(N/\sqrt{T})_{des}$. As a consequence, the flow entering the blade has been affected, and an increase in Mach number is noticed in Figure 19. A much bigger change is

FIGURE 19 NS3D predictions for blade mid-span Mach number distribution for the two speeds and vane stagger angles (See Colour Plate at back of issue.).

FIGURE 20 NS3D predictions for averaged stage reaction for the two speeds and vane stagger angles.

detected in the blade exit Mach number, however, especially for the 70% $(N/\sqrt{T})_{des}$. The peak Mach number in the blade exit area has increased from 1.55 to 1.68 for the 70% $(N/\sqrt{T})_{des}$, and has decreased from 1.5 to 1.42 for the 100% $(N/\sqrt{T})_{des}$. Thus, for this studied turbine, it was found that at low blade speeds (such as the 70% $(N/\sqrt{T})_{des}$), a small closing in the vane stagger angle results in a steep increase of exit Mach number. Whereas for high blade speeds (such as the 100% $(N/\sqrt{T})_{des}$), the same change in the vane stagger angle results in a small decrease of exit Mach number.

The stage reaction predictions are plotted in Figure 20. It is clear from the figure that closing the vane stagger angle by 0.8° resulted in a decrease in reaction for the two-tested blade speeds. A slightly higher decrease in reaction is found for the 70% $(N/\sqrt{T})_{des}$. This leads to the conclusion that closing the vane by such small value has the same relatively negative effect (causing a drop) on the stage reaction, independent of the blade speed.

CONCLUSION

Using a 3-D viscous, compressible code (NS3D) that solves the Reynolds averaged Navier-Stokes equations, CFD-based correlations have been produced for a single stage high-pressure transonic turbine. Four design and off-design parameters were studied, and their effects on the performance characteristics of the turbine were discussed. Each of the above mentioned parameters was studied in detail and relevant phenomena, such as the tip leakage vortex, were analyzed. All of the results obtained from NS3D were compared to an existing turbine correlation and to experimental data, except the tip clearance and vane stagger angle cases for which no test data are available from the CFTR test case used.

In general, the code compared favorably to the experimental values except for the PR cases, where the code was found to be insensitive to the variations of pressure ratios lower than 90% PR_{des} . This is due to the relatively coarse mesh used, since the code is still under development and since these runs were the first obtained from the code for turbines. It can also be explained by the implementation of the boundary conditions (constant profile imposed at inlet and imposed static pressure at the hub of the exit plane). Nevertheless, the results obtained from the code are still considered satisfactory. No tests were conducted on the effect of denser mesh on the solution, since much longer running times would have been required, but experience has shown that a denser mesh (to a certain limit) will improve the results in general. A better alternative is to use Mesh Adaptation Method in 3-D. (MOM3D). This is a future consideration.

Acknowledgments

The authors wish to thank the personnel from the Numerical Applications Group and the Turbine Component Center at Pratt & Whitney Canada who provided support throughout this work, and more specifically, Dr. M. Sleiman and Mr. J.-P. Lavoie for their technical expertise.

NOMENCLATURE

C_x	axial velocity, m/s
M	Mach number
$m\sqrt{T_t}/P_t$	flow capacity
N	rotational speed (rpm)
$N/\sqrt{T}$	normalized speed (rpm/ $\sqrt{^\circ K}$)
Nozzle	stator (see vane also)
NS3D	the 3-D viscous flow solver used
P	static pressure (Pa)
PR	pressure ratio
$^\circ K$	degrees Rankine
SFC	specific fuel consumption
T	static temperature ($^\circ K$)
tpclr	tip clearance
U	rotational speed, m/s
α	absolute gas angle
γ	ratio of specific heats
η	isentropic efficiency

$$= \frac{T_{in} - T_{out}}{(T_{in} - (P_{out}/P_{in})^{(\gamma-1)/\gamma})}$$

Subscripts

des	design
r	radial
t	total
x	axial

Superscripts

$^\circ$	degrees
----------	---------

REFERENCES

- Akel, I. (1999) Toward CFD-Based Correlations for Single-stage High-pressure Transonic Turbine Stage, *M. A. Sc. Thesis*, Concordia University, Montréal.
- Habashi, W. G., Dompierre, J., Bourgault, Y., Fortin, M. and Vallet, M.-G. (1998) Certifiable Computational Fluid Dynamics through Mesh Optimization, Invited Paper in Special Issue on Credible Computational Fluid Dynamics Simulation, *AIAA Journal*, **36**(5), 703–711.
- Moustapha, S. H., Aerodynamics Research on Transonic and Small Aspect Ratio Turbines, *International Conference of Fluid Mechanics – ICFM5*, 5th, Jan. 2–5, 1995, Cairo, Egypt.

- Peeters, M. F., Habashi, W. G., Nguyen, B. Q. and Kotiuga, P. L. (1992) Finite Element Solutions of the Navier-Stokes Equations for Compressible Internal Flows, *AIAA Journal of Propulsion and Power*, **8**, 192–198.
- Sjölander, S. A. and Amrud, K. K., Effects of Tip Clearance on Blade Loading in a Planar Cascade of Turbine Blades, *ASME, International Gas Turbine Conference and Exhibit*, 31st June 8–12, 1986, Duesseldorf, West Germany.
- Sleiman, M. (1999) Simulation of 3-D Viscous Compressible Flow in Multistage Turbomachinery by Finite Element Methods, *Ph.D. Thesis*, Concordia University, Montréal.
- Sleiman, M., Tam, A., Robichaud, M. P., Peeters, M. F. and Habashi, W. G. (1999) Turbomachinery Multistage Simulation by a Finite Element Adaptive Approach, *ASME Journal of Fluids Engineering*, **121**(2), 450–459.
- Vlasic, E. P., Girgis, S. and Moustapha, S. H., The Design and Performance of a High Work Research Turbine, *ASME, International Gas Turbine and Aeroengine Congress and Exposition*, June 5–8, 1995, Houston, TX.
- Woinowsky-Krieger, M., Lavoie, J.-P., Vlasic, E. P. and Moustapha, S. H., Off-Design Performance of a Single Stage Transonic Turbine, *ASME, International Gas Turbine and Aeroengine Congress and Exposition*, 43rd June 2–5, 1998, Stockholm, Sweden.
- Yaras, M. and Sjölander, S. A., Development of the Tip-leakage Flow Downstream of a Planar Cascade of Turbine Blades – Vorticity Field, *ASME, Gas Turbine and Aeroengine Congress and Exposition*, June 4–8, 1989, Toronto, Canada.

Vane geometry			Blade geometry		
x	suction	pressure	x	suction	pressure
–4.54	16.30	–1.78	–1.69	–3.80	0.22
–2.75	14.25	–4.51	–0.91	–3.69	0.41
0.84	7.76	–10.55	0.66	–3.00	1.02
2.63	3.34	–13.88	1.44	–2.39	1.45
4.42	–1.53	–17.42	2.23	–1.59	1.96
6.21	–6.74	–21.19	3.01	–0.63	2.57
9.80	–17.94	–29.40	4.58	1.60	4.05
11.59	–23.83	–33.85	5.37	2.81	4.91
13.38	–29.85	–38.55	6.15	4.05	5.83
16.07	–39.17	–46.08	7.33	5.99	7.29
16.96	–42.38	–48.78	7.72	6.66	7.81
17.50	–44.32	–49.76	7.96	7.08	8.13
17.86	–45.67	–49.88	8.11	7.36	8.33
18.22	–46.98	–49.82	8.27	7.64	8.39
18.58	–48.24	–49.55	8.43	7.93	8.32
18.76	–49.02	–49.02	8.50	8.14	8.14

APPENDIX I

Airfoil Mean Section Geometries

All co-ordinates are in mm.

Vane geometry			Blade geometry		
x	suction	pressure	x	suction	pressure
–17.08	14.91	14.91	–7.19	0.43	0.43
–16.72	16.49	13.36	–7.03	–0.13	0.87
–16.36	17.09	12.78	–6.87	–0.45	0.99
–15.29	18.06	11.61	–6.40	–1.25	1.03
–13.50	18.83	9.66	–5.62	–2.22	0.60
–12.60	19.07	8.65	–5.23	–2.58	0.42
–10.81	19.19	6.56	–4.44	–3.13	0.18
–9.91	19.11	5.47	–4.05	–3.34	0.11
–8.12	18.64	3.19	–3.26	–3.64	0.06

Gaspath Geometries

All co-ordinates are in mm.

x-hub	r-hub	x-tip	r-tip
–82.81	122.66	–80.44	167.22
–72.02	121.03	–75.94	160.18
–66.56	120.94	–72.45	157.81
–55.64	120.58	–64.44	155.15
–50.22	119.91	–60.31	154.25
–44.80	119.24	–56.12	153.74
–39.35	118.79	–51.94	153.12
–28.42	118.66	–43.73	151.11
–17.50	118.76	–35.56	148.91
–1.19	117.71	–22.99	147.39
4.10	116.34	–18.81	147.92
8.65	115.20	–14.63	148.46
20.97	113.90	–6.18	148.53
40.78	116.05	11.08	148.60