Numerical Investigation of Aerodynamic Performance Parameters in Linear Blade Cascade for High-pressure Turbine

Nitish Kulkarni, D. Harish, R. D. Bharathan, Sharad Kapil, S. V. Ramana Murthy, R. Senthil Kumaran, and D. Kishore Prasad

Abstract The function of the turbine aerofoil is to turn the flow to a certain angle and provide smooth flow acceleration to a certain Mach number with minimum pressure loss. The flow field in the blade passage is complex, unsteady, transonic, and three dimensional. So, to aerodynamic designer, it is necessary to evaluate the profile losses and other aerodynamic performance parameters to improve the efficiency of the turbine stage. In the present study, CFD simulation has been carried out for the high-pressure turbine nozzle guide vane (NGV) profiles for design and off-design conditions. Parameters such as profile loss coefficient, surface Mach number distribution, and flow deflections are investigated. The results obtained from numerical analysis are compared with experimental data from transonic cascade tunnel.

Keywords CFD analysis · Pressure loss coefficient · Surface Mach no distribution · Flow deflection · Cascade test results

N. Kulkarni · D. Harish · R. D. Bharathan (✉) · S. Kapil · S. V. Ramana Murthy · D. Kishore Prasad
DRDO-Gas Turbine Research Estabilishment, Bengaluru, India
e-mail: bharathan.gtre@gov.in

D. Harish
e-mail: harishd.gtre@gov.in

S. Kapil
e-mail: sharadkapil.gtre@gov.in

S. V. Ramana Murthy
e-mail: ramanamurthy.gtre@gov.in

D. Kishore Prasad
e-mail: kishoreprasadd.gtre@gov.in

R. Senthil Kumaran
CSIR-National Aerospace Laboratories, Bengaluru, India
e-mail: rsk@nal.res.in

G. Sivaramakrishna et al. (eds.), *Proceedings of the National Aerospace Propulsion Conference*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-19-2378-4_9

Nomenclature

C	Blade Chord
C_{ax}	Axial chord
h	Blade height
HP	High Pressure
I	Incidence angle
k	Turbulence kinetic energy
M_2	Outlet Mach number
M_{is}	Surface Mach number
NGV	Nozzle Guide Vane
P_{01}	Inlet Total Pressure
P_{02}	Outlet Total Pressure
P_s	Outlet static Pressure
s	Pitch
SST	Shear Stress Transport
X	Axial distance from leading edge
Y_p	Pressure loss coefficient
γ	Specific heat ratio

1 Introduction

Gas turbine engines used in advanced fighters need high thrust to weight ratio and low-specific fuel consumption to meet high maneuverability, long range, and low-life cycle cost requirements. To meet high thrust to weight ratio and low-specific fuel consumption, aerogas turbine engines demand high-turbine entry temperature and compressor pressure ratio. A highly efficient turbine is required to maximize the work that can be extracted for a given pressure ratio. The flow in the turbine is highly complex, unsteady, transonic, and three dimensional. The reduction in efficiency is due to a variety of losses such as profile losses, secondary losses, tip clearance losses, and cooling losses. Hence, understanding the contribution from each of these losses is of prime importance.

During preliminary design, correlations are extensively used to predict the losses in the sections. Ainley and Mathieson [1] proposed correlations to calculate the pressure losses and deviation angle. The authors assumed that pressure loss correlations are independent of Mach numbers, and exit angle is independent of incidence angle. Dunham and Came [2] suggested Mach number and Reynolds number correction factors to earlier proposed correlations by Ainley and Matheson [1]. Kacker and Okapuu [3] further modified these correlations to improve the loss prediction. Denton [4] carried out a detailed study on losses in turbomachines and derived the correlation for increase in entropy due to losses. Mee et al. [5] experimentally investigated losses of transonic turbine blade cascade and provided the effect of Mach no. on

Table 1 NGV mean section parameters

Parameters	Values
h/C	2.9
S/C	0.73
Design Mach no.	0.98
Outlet angle	71.25°

boundary layer losses and also gave the effect of shock loss and mixing loss. At the midspan of the blade, the aerodynamic performance of a transonic turbine cascade for different exit Mach no. and for different incidence angles was studied by Jouini et al. [6]. Andrey Granovskiy et al. [7] examined both experimentally and numerically the influence of unguided turning angle and trailing edge shape on profile loss of transonic turbine blades. Over the last 70 years, these correlations have been updated based on the rig test data that are available to the authors. These studies suggest that the losses in high- and low-pressure turbine aerofoil are influenced by flow turning angle, blade loading, number of aerofoils, and inlet boundary layer profile.

In this study, numerical investigation is carried out to estimate the pressure loss coefficient and aerodynamic parameters like surface Mach number distribution, exit flow deflection for midspan section of the HP turbine NGV for different outlet Mach numbers. These parameters are estimated for −10, 0, and 10° incidences. Also computational results are compared with the experimental results obtained for the same profile in transonic cascade tunnel (TCT).

2 Methodology

2.1 Modeling

The fluid domain has modeled in CAD software and same geometrical dimensions maintained as per aerofoils tested in TCT, National Aerospace Laboratories (NAL). The geometrical features for the NGV are given in Table 1.

The fluid domain consists of two NGVs which are placed at 1 pitch distance to define proper boundary condition and for ensuring adequate flow development, inlet and outlet are extended one axial chord from leading and trailing edge, respectively.

2.2 Grid Generation

The computational grid is generated using ICEM-CFD 16.0 software. The grid is passage centered, multiblock, structured, and hexahedral. The grids are made finer at the regions where high gradients are expected in the flow field. Grid independence analysis is carried out, and the variation of profile loss with grid size is shown in Fig. 1. At 1.5 million grids, the solution becomes grid independent. Therefore, it is finalized for the NGV domain.

Sufficient grid points are placed near the aerofoil to resolve the boundary layer. The skewness achieved for the grid is between 25 and 150°, and $y+$ value for finalized NGV grid is maintained below 1. The aspect ratio is less than 100, and expansion ratio is less than 1.2. The passage centered grid for NGV is as shown in Fig. 2.

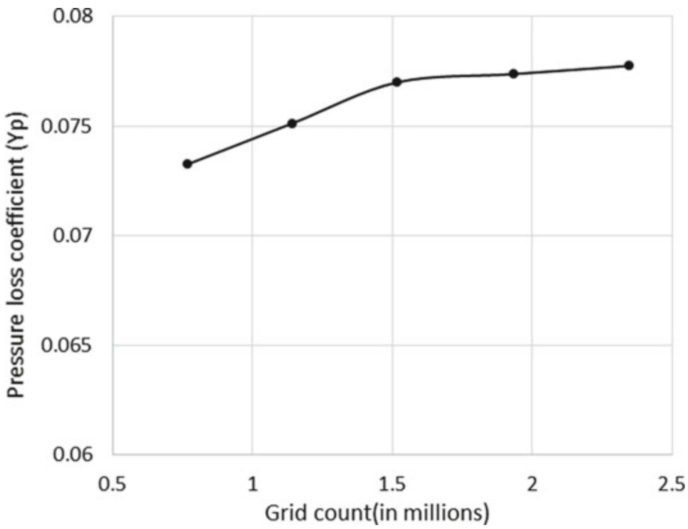


Fig. 1 Grid independency study for NGV fluid domain

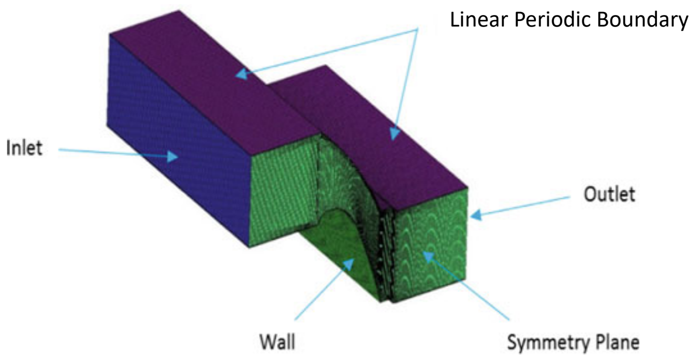


Fig. 2 Computational domain for NGV

2.3 CFD Solver and Boundary Condition

Ansys CFX-16.0 solver is used to solve the Navier-Stokes equation. CFX solver is based on finite volume approach and is capable of solving complex multidimensional fluid flow problems. The turbulence effect is modeled by using $k{-}\omega$ SST turbulence model. The advantage of two equation $k{-}\omega$ SST turbulence model is that it predicts highly accurate flow separation at adverse pressure gradient conditions.

The boundary conditions are inlet total pressure, inlet total temperature, and outlet average static pressure. The top and bottom surfaces are symmetrically placed at one pitch distance and provided with translational periodicity to obtain similar flow condition as linear cascade test. The walls are assumed to be adiabatic and no slip walls.

2.4 Linear Cascade Testing

Linear cascade testing is the simplified experimental method for evaluating aerodynamic performance of turbo machinery aerofoils where Coriolis forces and curvilinear effects are ignored. Three dimensional flows can be simplified to two dimensional flows by eliminating the radial component of velocity using linear cascade. A linear cascade is an array of aerofoils stacked at uniform pitch and stagger representing a section of turbo machinery blade rows.

The present study is carried out in TCT, NAL, Bangalore. TCT is a quasi- continuous blow down tunnel. The size of test section is 153 × 500 mm. The height of the test section is adjustable from 500 to 200 mm.

The schematic view of the mean cascade assembly in the transonic cascade tunnel is shown in Fig. 3. Flow velocity in the test section can be varied by controlling the

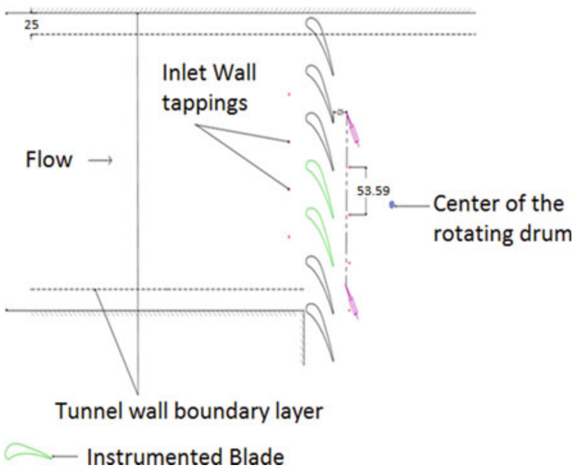


Fig. 3 Schematic view of NGV mean cascade in TCT

pressure in the settling chamber with the help of an automatic pressure regulator valve. Inlet total pressure and total temperature are measured in a settling chamber using a pitot tube and thermocouples, respectively. Inlet static pressure and outlet static pressure are measured from test section side wall tappings. A five-hole wedge probe is mounted downstream of the blade trailing edge at the expected outlet flow angle to measure the wake flows. One aerofoil is instrumented along the suction side for surface static pressure measurements, while another aerofoil is instrumented on the pressure side. Scanivalve system and electronic pressure scanners are used for blade surface pressure measurement. The pressures were measured with an accuracy of ± 172 N/m^2, and the flow angle was measured with an accuracy of $\pm 0.2°$. Maximum uncertainty in the Mach number was ± 0.02.

3 Results and Discussion

In this section, the effect of incidence angle and exit Mach no. on the NGV cascade has been numerically predicted and compared with the experimental data.

There are many different loss coefficient definitions to study the turbine losses, but the pressure loss coefficient is most commonly used in cascade studies, because it is easy to calculate from cascade test data. The stagnation pressure loss coefficient for turbine is defined as

$$Y_p = \frac{(P_{01} - P_{02})}{(P_{02} - P_s)} \tag{1}$$

Figure 4 shows the variation of pressure loss coefficient for mean section of the cascade with the exit Mach no. ranging from 0.6 to 1.1 and for incidence angles -10, 0, and 10°. The pressure loss coefficient at midspan of the NGV is calculated using Eq. (1) by obtaining area-averaged total and static pressures at cascade exit. The results obtained from CFD investigation have been compared with experimental data, and the experimental results closely match the numerical results for subsonic exit Mach number 0.6–0.9.

There is steep rise in pressure loss in the transonic outlet Mach no. regime due to formation and strengthening of the shock wave, but surprisingly, the loss coefficient observed from computational studies is lower compared to experimental results for Mach no. 1.0 and 1.1. There is no variation observed in pressure loss coefficient for different incidence angles between Mach no. 1.0 and 1.1. This shows that shock losses dominate over the incidence losses in this regime.

The blade to blade Mach number distribution for mean section of the NGV at the exit Mach number 1.0 is shown in Fig. 5. From the contour plot, it is clearly observed that there is no flow separation in the flow passage. Also, wake and shock formation downstream of the blade passage is captured in this contour plot.

The surface Mach number plot shown in Figs. 6 and 7 is obtained using surface static pressure measurement as well as upstream total pressure measurement as shown in Eq. (2).

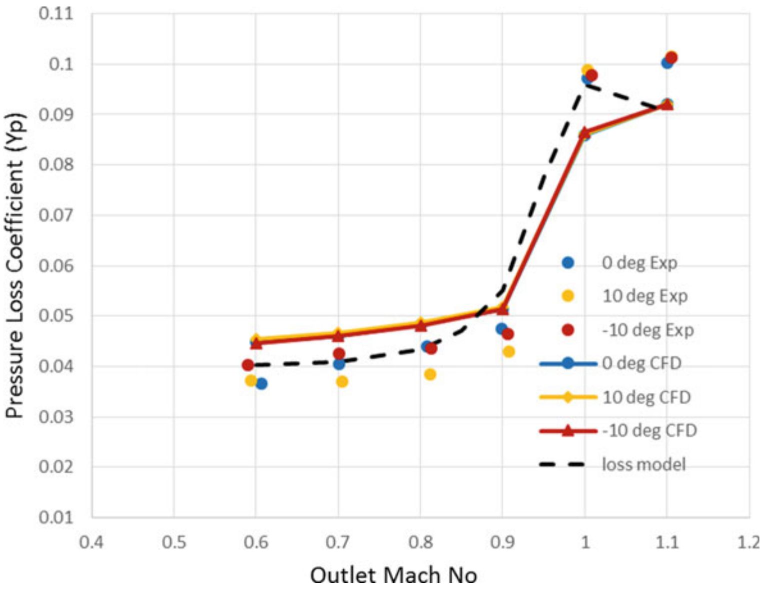


Fig. 4 Variation in pressure loss coefficient for different outlet Mach number

Fig. 5 Mach number contour for the NGV $M_2 = 1.0$

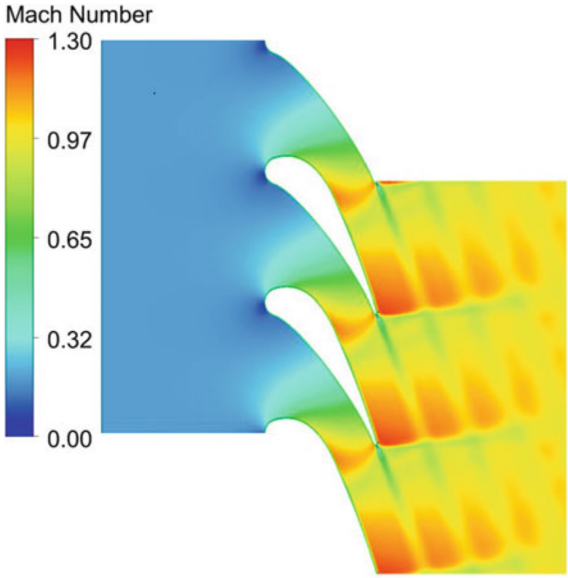


Fig. 6 Surface Mach number distribution of NGV for $M_2 = 0.6$

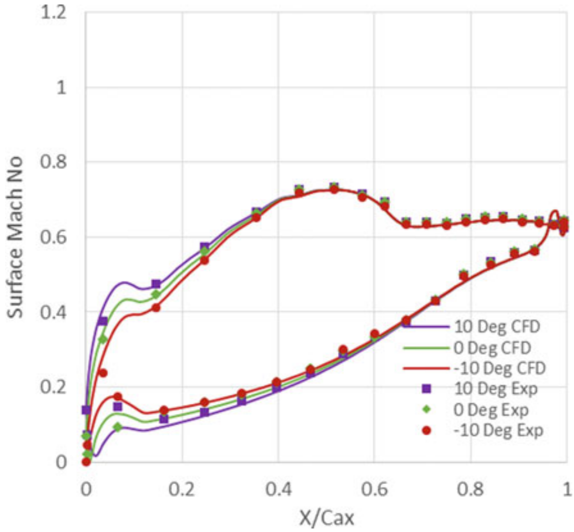
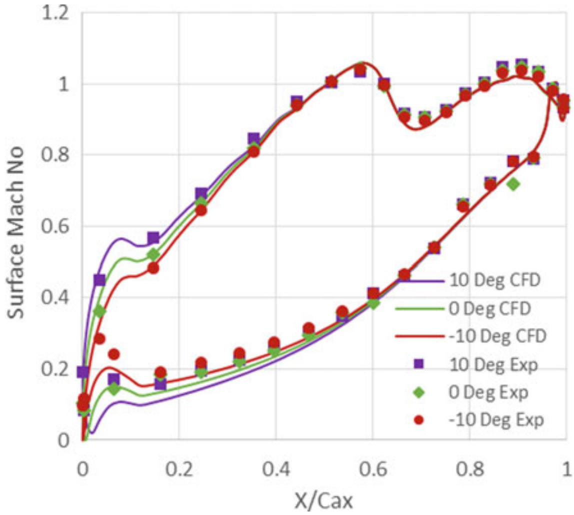


Fig. 7 Surface Mach number distribution of NGV for $M_2 = 0.9$



$$M_{is} = \sqrt{\frac{2}{(\gamma - 1)}\left\{\left(\frac{P_{01}}{P_s}\right)^{\left(\frac{\gamma-1}{\gamma}\right)} - 1\right\}} \tag{2}$$

Figures 6, 7, and 8 show the surface Mach number plot for exit Mach number of 0.6, 0.9, 1.0 and different incidence −10°, 0°, 10°. It can be observed that the drop in local Mach number passed the throat point on the suction side becomes increasingly sharp with increase in exit Mach number. At around 0.6 Mach number, the slight drop

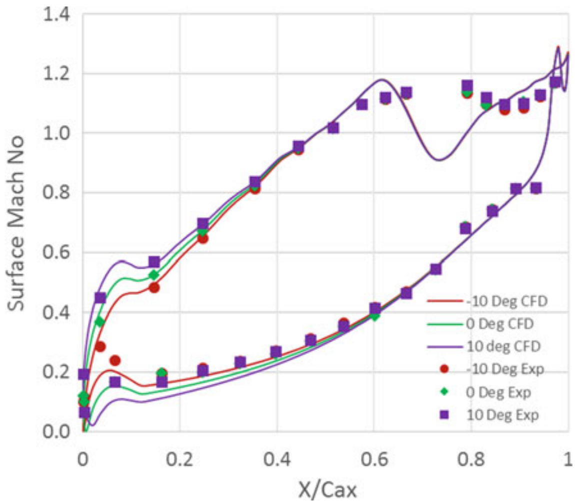


Fig. 8 Surface Mach number distribution of NGV for $M_2 = 1.0$

in local Mach number at the throat is due to flow not being guided by the neighboring aerofoil. At around Mach number 1.0, a shock is observed (see Fig. 5) near the throat region which results in a sharp decrease in local Mach number. The surface Mach number is calculated for mean section of the NGV cascade using Eq. (2). There is good match found between computational and experimental results. It can be seen that CFD captures the initial acceleration and the peak Mach number on the suction side of the aerofoil with reasonable accuracy for the Mach number range of 0.6–0.9. In Fig. 7, at around 60% of axial chord, the surface Mach number reaches 1.

The effect of incidence on the surface Mach no. distribution can be seen from Figs. 6, 7, and 8, near the leading edge of the NGV. And its effect on the surface Mach number reduces toward the throat section. For the aerofoil used in this study, the inlet blade angle is 0°. The incidence of +10° shifts the stagnation point move toward the pressure side of the aerofoil. As a result, the flow has to cover more distance over the leading edge circle and experiences strong acceleration. Because of this, the surface Mach number over the pressure side is less compared to incidence of 0°.

The inability of the NGV to turn the flow to the required outlet angle is a measure of the losses in the NGV. Figure 9 shows the outlet flow angle variation for exit Mach number of 0.6–1.1 and different incidences of −10°, 0°, 10°, and it is observed that there is a good agreement between CFD and experimental results. The NGV profile was designed for 71.25°. The CFD and experimental results closely match the design blade angle for exit Mach number below 0.9. Kibsey [8] experimentally and computationally investigated the effect of exit Mach number on outlet flow deflection for different linear blade cascades and stated that outlet flow deflection reduces with increase in exit Mach number beyond the sonic region. Similarly, in present study, outlet flow angle remains more or less constant for the range of exit

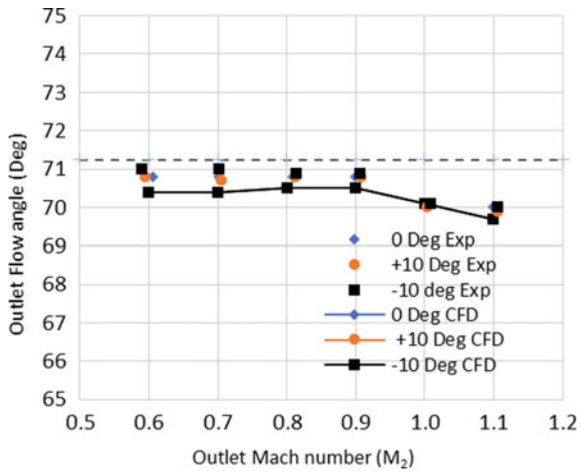


Fig. 9 Outlet flow angle variation along the outlet Mach number

Mach no. 0.6–0.9 but it reduces continuously beyond the sonic condition due to formation of shock.

The flow deflection also is shown with respect to different incidence angle in Fig. 9. Very minor variation is observed for different incidences.

4 Conclusions

Computational analysis for NGV cascade has been carried out in commercial CFD software and compared with cascade test results. There is good agreement found between CFD and experimental results for pressure loss coefficient, surface Mach no. distribution, and deviation angle for different incidence angles.

The loss coefficient is lower at subsonic regime but increases steeply in supersonic regime. This is due to formation of shocks which can be observed from Mach number contour plots.

The surface Mach number distribution predicted from numerical analysis shows good match with experimental results. It is observed that the surface Mach number locally exceeds 1 on the suction side in the throat region (minimum area) for exit Mach number of 0.9. CFD captures the shock patterns downstream of the throat.

The deviation angle fairly remains constant at low Mach numbers but it reduces for exit Mach number higher than 0.9.

References

1. Ainley DG, Mathieson GCR (1951) A method of performance estimation for axial-flow turbine. Issue 2974 of Report and memoranda, Aeronautical Research Council
2. Dunham J, Came PM (1970) Improvements to the Ainley-Mathieson method of turbine performance prediction. ASME J Eng Power 92(3):252–256
3. Kacker SC, Okapuu U (1982) A mean line prediction method for axial flow turbine efficiency. ASME J Eng Power 104(1):111–119
4. Denton JD (1993) Loss mechanisms in turbomachines. ASME Int Gas Turbine Aeroengine Congr Exposition 115:621–656
5. Mee DJ, Baines NC, Oldfield MLG, Dickens TE (1992) An examination of the contribution to loss on a transonic turbine blade in cascade. ASME Int Gas Turbine Aeroengine Congr Exposition 114:155–162
6. Jouini DBM, Sjolander SA, Moustapha SH (2001) Aerodynamic performance of a transonic turbine cascade at off-design conditions. ASME Int Gas Turbine Aeroengine Congr Exhibition 123:510–518
7. Granovskiy A, Gribin V, Lomakin N (2018) Experimental and numerical study of transonic cooled turbine blades. Int J Turbomach Propul Power
8. Kibsey MD (2015) Influence of mach number on profile loss of axial-turbine blade cascade. Thesis, Carlton University, Ottawa, Ontario