

# Development And Validation Of A Bezier Curve Based Profile Generation Method For Axial Flow Turbines

S.V. Ramana Murthy, S. Kishore Kumar

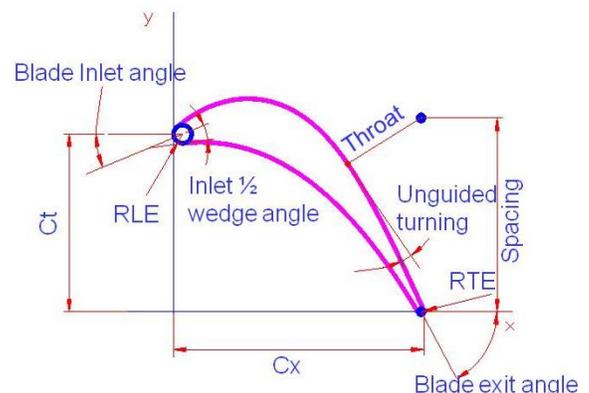
**Abstract:** The important step in the turbine aerodynamic design is generation of blade element profiles to achieve the required flow deflection and smooth flow acceleration with minimum losses. During profile generation, in addition to aerodynamic performance, additional constraints of thermal, structural and castability requirements need to be considered. Thus profile generation is a trade-off between contradicting requirements of aerodynamic performance, thermal and structural performance. In the present study, a method for profile generation based on Bezier curves is developed to meet the above conflicting requirements. These profiles are analyzed for the flow field using a commercial 3D N-S code for design and off-design conditions. These profiles are also tested in 2D transonic cascade tunnel. The profile pressure loss coefficient, blade exit angle and blade surface isentropic mach numbers are compared between CFD and test results.

**Index Terms:** Axial flow turbines, profile generation, Bezier curves, CFD analysis, 2D cascade testing, profile loss, Surface Mach number Distribution

## 1 INTRODUCTION

In general, military aero gas turbine engine high pressure and low pressure turbines are of single stage configuration with a view of light weight, low initial and maintenance cost due to reduction in number of components and expensive air cooled aerofoils [1],[2],[3],[4]. Though single stage offers compactness and simplicity, it results in high turbine stage work loading and expansion ratios. This leads to reduction in the turbine efficiency which in-turn results in higher SFC of the engine. However, lower SFC can be achieved by higher turbine efficiency, overall pressure ratio along with higher TET. But, higher TET's reduces turbine component's life. Hence design of single stage high efficient turbine with required life is a challenging task. The function of a turbine aerofoil is to deflect the flow and ensure smooth flow acceleration with minimum pressure losses. While generating the turbine blade element profiles, due care should be taken to meet the conflicting requirements from thermal, structural and castability considerations. Standard aerofoils like the NACA series aerofoils, British profiles like C4 and T6 and Russian axial flow turbine blade atlas have been used in the past to generate axial flow turbine profiles [5],[6],[7]. In this method, blade profiles are suitably modified to meet the specific requirements of the design like inlet angle, outlet angle etc. The standard aerofoils are defined as a camber line with a thickness distribution imposed on it to obtain the suction and the pressure surfaces.

However, the flow past a profile is felt by the suction and pressure surfaces and therefore it is desirable to have a direct control over the flow surfaces rather than the camber line. Kuzmin proposed a method in which the aerofoils are constructed using circular arcs and parabolic curves [8]. In this method, a specific aerofoil is defined by using a leading edge (LE) circle, trailing edge (TE) circle, a parabolic arc for the suction surface and a single circular arc for the pressure side. In this method, the LE and TE cusp are to be varied to obtain the desired  $T_{max}$  and the throat width. But control of Un-guided Turn (UGT) is not possible. Various commercial packages are also available for profile generation of turbine blades like BladeGen and VISTA from ANSYS [9], AutoBlade from NUMECA [10]. However, the algorithms used in these commercial packages are not clearly known. Utilization of these commercial packages to satisfy all the specific requirements of the design is not possible. In a eleven parameter profile generation method proposed by Pritchard [11], the profiles at different radii of the blade are generated by specifying the axial and tangential chord, throat, UGT, LE & TE radii, inlet and exit blade angles, and inlet and outlet wedge angles. The eleven input parameters translate into five fixed points on the aerofoil surface which result from: locating the leading edge and trailing edge circles, determining the suction surface and pressure surface tangency points and fixing the throat which are shown in Fig. 1.



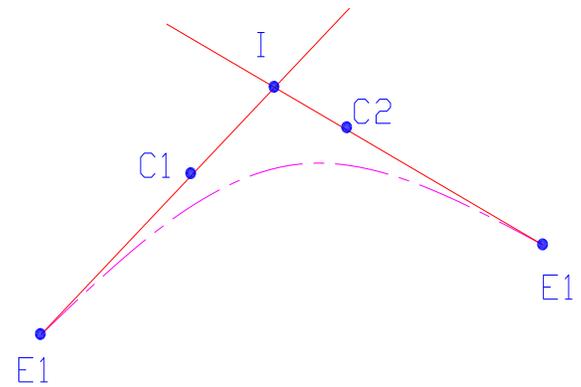
**Fig. 1.** Typical aerofoil of the 11 parameter method

- S V Ramana Murthy is Scientist in Gas Turbine Research Establishment and is currently pursuing doctoral degree program in Mechanical engineering in Visvesaraya Technological University, India, E-mail: [seepana.ramana@gmail.com](mailto:seepana.ramana@gmail.com)
- S Kishore Kumar is Scientist in Gas Turbine Research Establishment, Bangalore, India.

Then mathematical functions are used to connect these five points. Third order polynomials are used to connect the Leading edge suction surface tangent point to the throat point and Leading edge pressure surface tangent point to the trailing edge pressure surface tangent point. However, the eleven parameter method does not provide any control on the maximum thickness ( $T_{max}$ ) and the location of the maximum thickness ( $T_{max}/C$ ) which is very significant from structural and thermal requirements. This method specifies the leading edge as a circle which causes local acceleration. In the eleven parameter method, once the five key points are fixed using the input, the shape of the profile is fixed and there is no flexibility in changing the profile between these points. Therefore, the authors have developed a code to generate turbine blade element profiles by modifying the 11 parameter method using Bezier curves. The modified method overcomes the constraints of 11 parameter method by Bezier curves. The profiles are analyzed using 3D NS analysis for its performance in terms of profile pressure loss coefficient, flow exit angle and surface Mach number distribution. These profiles are also validated by testing the 2D cascades in TCT in NAL, Bangalore.

## 2 AEROFOIL PROFILE GENERATION BY BEZIER CURVE METHOD

Bezier curves are basically smooth curves which are drawn between two points using two or more control points which guide the path of the curve. The slope of the curve at the end points is defined by the control points. Since the control points are independent of one another, the relative position of these control points with respect to the end points only determines the shape of the curve. Therefore any number of curves can be fitted between the endpoints by changing the relative position of the control points [12],[13]. The modified aerofoil generation method uses the same five points of 11 parameter method, and these five points are connected using Bezier curves. Thus, the modified method keeps the advantages of the 11 parameter method also. The profiles which are generated by Bezier curves show better aerodynamic performance [14],[15]. The code modifying the 11 parameter method of profile generation is developed using third order Bezier curves for the portion of the aerofoil connecting the Leading edge tangent points and the throat point on the suction side and a single third order Bezier for the pressure side from the Leading edge tangent point to the trailing edge tangent points. The shape of these Bezier curves are controlled using the relative position of the control point with respect to the end points on the profile, thus providing a good amount of flexibility in the profile parameters like the maximum thickness and the location of the maximum thickness. A flexibility in the maximum thickness and its location in a turbine aerofoil helps to accommodate internal passages required for blade cooling and structural requirements. The mathematical formulation for implementing the control points and the Bezier curves are given below. Fig. 2 shows the control points for the Bezier generated profile.



**Fig. 2.** Typical Bezier curve showing the end points and control points

The location of the intersection point I ( $x_5, y_5$ ) of the lines drawn from the first end point E1 ( $x_1, y_1$ ) and the other end point E2 ( $x_2, y_2$ ) is calculated using

$$x_5 = \frac{[(y_1 - m_1 x_1) - (y_2 - m_2 x_2)]}{(m_2 - m_1)}$$

$$y_5 = y_2 + m_2 (x_5 - x_2)$$

Where  $m_1$  and  $m_2$  are the slopes of the lines from points 1 and 2 which are known. The control points of the Bezier curve i.e. C1 ( $x_3, y_3$ ) and C2 ( $x_4, y_4$ ) are calculated using the ratios  $V_1$  and  $V_2$  which represent the fraction of the lines from the points E1 to I and E2 to I on which the control points are to be located respectively.

$$x_3 = x_1 - V_1 (x_1 - x_5)$$

$$y_3 = y_1 + V_1 (y_3 - x_1)$$

$$x_4 = x_2 - V_2 (x_2 - x_5)$$

$$y_4 = y_2 + V_2 (y_4 - x_2)$$

Using these control points C1 and C2, and the end points 2 and 3, a third order Bezier curve (equation shown below) is fit between these two points. The co-ordinates of the points ( $x, y$ ) on the curve are obtained for different values of the parameter  $t$  in the equation below.

$$x = (1-t)^3 x_2 + 3t(1-t)^2 x_4 + 3t^2(1-t)x_3 + t^3 x_1$$

$$y = (1-t)^3 y_2 + 3t(1-t)^2 y_4 + 3t^2(1-t)y_3 + t^3 y_1$$

The value of  $t$  varies from (0 to 1). The NGV and rotor blade mean section aerofoils are generated using the above method. The location of the control points are finalized based on a 2D blade to blade flow analysis [16] for different exit Mach numbers and incidences. These finalized profiles are analyzed for the flow-field using a commercially available 3D Navier-Stokes code ANSYS CFX-12.0

## 3 COMPUTATIONAL METHODOLOGY

CFX-12.0 [17] is an integrated software capable of solving diverse and complex multi-dimensional fluid flow problems. The software solves the 3D Reynolds Averaged N-S equations in a fully implicit manner. It is a finite volume method and is based on a finite element approach to represent the geometry. The method retains much of the geometric flexibility of finite element methods as well as the important conservation properties of the Finite Volume

method. The software has four major modules a) CFX-Turbo grid – creates geometry and generates structured mesh tailored for turbo-machinery b) Pre-processor – sets the boundary conditions, the initial flow field conditions and solution numerics c) Solver Manager – solves the flow field based on the grid and boundary conditions d) Post processor – visualizes and extracts the results. It utilizes numerical upwind schemes to ensure global convergence of mass, momentum, energy and species. In the present study discretization of the convective terms is done by second order upwind differencing scheme. Local time stepping has been used to obtain steady state solutions. The present calculations are done using the SST model.

#### 4 GRID GENERATION

The computational grid is generated using ICEM-CFD software [18]. The grids are passage centered multi-block, structural hexahedrals. The grids are a combination of C and H grids. The grids are made finer at the regions where high gradients are expected in the flow field. The grid sizes for the stator and rotor are 0.5 million after the grid sensitivity analysis. Sufficient grid points are placed close to the aerofoil surfaces to resolve the boundary layer. The achieved skew angles are between 25 and 155 degrees, the aspect ratios are less than 100 and expansion ratios are less than 1.2. The achieved  $y^+$  value is less than 10. The outlet domain of the grid is placed one chord downstream of the exit plane as in the cascade tests. The passage centered grid for the NGV is shown below in Fig. 3.

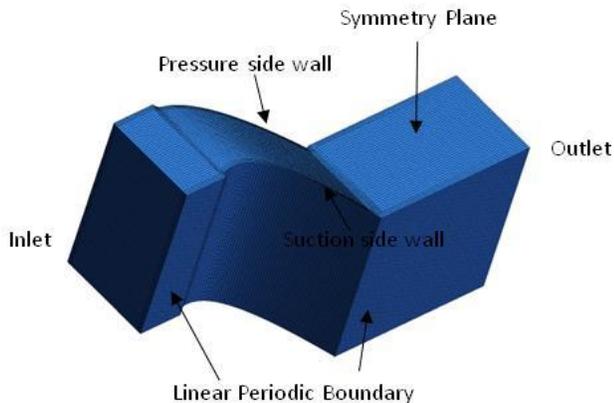


Fig. 3. Passage centered NGV grid

#### 5 BOUNDARY CONDITIONS

The analysis is carried out for both the NGV and the rotor cascades using the commercial 3D N-S solver ANSYS CFX-12.0 on IBM DX 360 parallel computing system. The boundary conditions are inlet total pressure, inlet total temperature, inlet flow angle and average exit static pressure. The walls are assumed adiabatic. Symmetry planes are defined in the top and bottom faces of the computational domain to simulate the cascade conditions. Translational linear Periodic boundary condition is applied in the blade to blade plane for one pitch.

#### 6 CASCADE TESTING

The experiments were carried out on 2D cascades of mean sections of NGV and rotor blade profiles to find out the

profile loss coefficients, exit flow angles and surface Mach number distribution for different exit Mach numbers and incidences at the Transonic Cascade Tunnel (TCT) of National Aerospace Laboratories, Bangalore. It is quasi-continuous blow down tunnel [19]. The size of test section is 153x500 mm. The schematic of the test section is shown in Fig. 4.

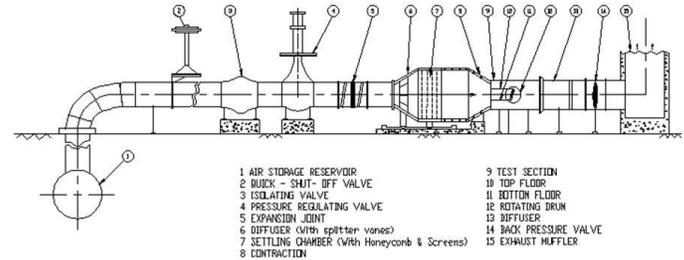


Fig. 4. Schematic of the TCT facility

Flow periodicity in the tunnel is ensured by adjusting the position of top and bottom walls of the test section. Inlet total pressures and temperatures are measured in the settling chamber using a Pitot probe and thermocouple respectively. Inlet static pressure is measured from the tappings on the test section sidewall. A calibrated five-hole wedge probe is used for wake measurements. Wake traverse of the exit flow from the cascade is made one chord downstream of the cascade plane. Five pitches of the blade are covered for recording the values. A two dimensional momentum averaging technique is used to calculate the profile loss and exit angle from the wake data. In the center of the cascade two blades are instrumented to measure the surface static pressure distribution. These measurements are carried out in two-scale model of the profiles after ensuring the similarity because of the inability to accommodate sufficient number of static pressure taps owing to the smaller chord of the profiles.

#### 7 RESULTS AND DISCUSSIONS

Fig. 5 & 6 show the Mach number contour plot and velocity vectors in the blade passage respectively.

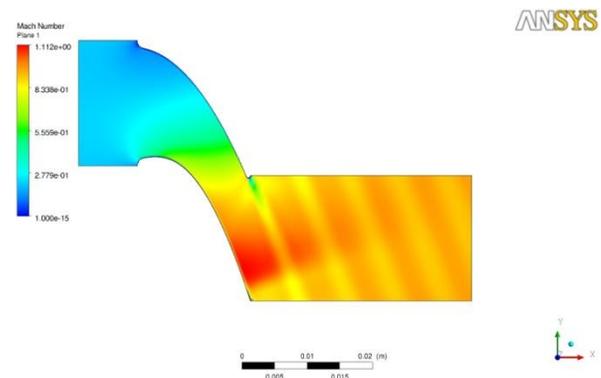
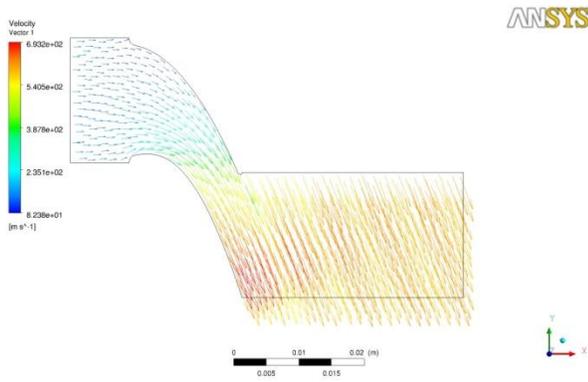


Fig. 5. Blade to blade Mach number plot for the NGV

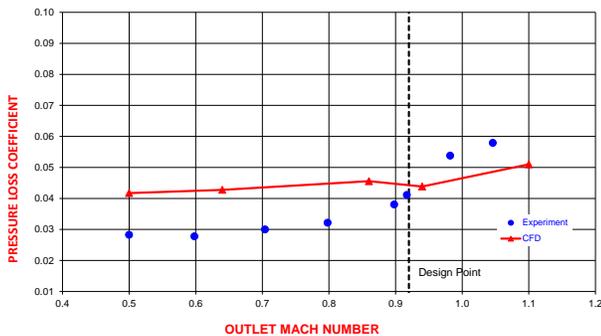


**Fig. 6.** Blade to Blade velocity vectors for the NGV

It can be observed from the figures, that there is no separation in the passage and the trailing edge wake is captured. The results of the cascade tests and the 3D NS simulations carried out at the same conditions are compared and studied for both the NGV and rotor mean section of the turbine stage under consideration. The pressure loss coefficient ( $Y_p$ ) is defined as the total pressure loss ( $P_{01} - P_{02}$ ) across the cascade to the outlet dynamic head of the flow ( $P_{02} - P_2$ ).

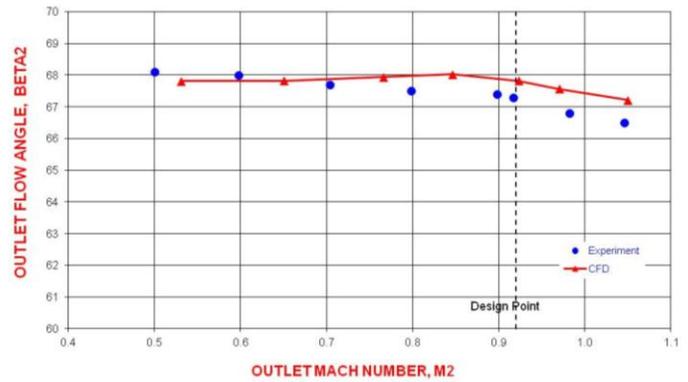
$$Y_p = \frac{(P_{01} - P_{02})}{(P_{02} - P_s)}$$

Fig. 7 shows the comparison of the pressure loss coefficients of the NGV mean section for different exit Mach numbers obtained from CFD analysis and experiments.



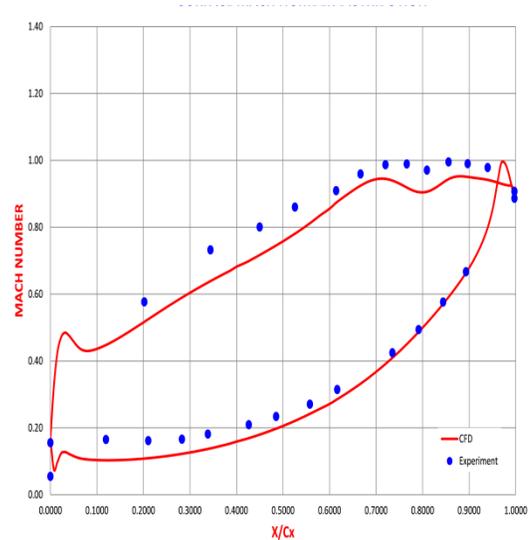
**Fig. 7.** Variation of NGV Pressure loss coefficients for different exit Mach numbers

The 3D NS analysis shows similar trend as the cascade experiments. At the design exit Mach number, there is a good agreement between CFD and the experiment results. Fig. 8 shows the variation of the flow exit angles from the NGV cascade for different Mach numbers obtained from CFD analysis and experiments.



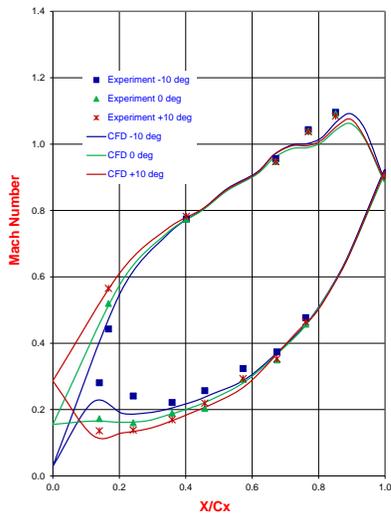
**Fig. 8.** Variation of NGV exit angles for different exit Mach numbers

It can be observed that there is a good agreement between the CFD and experimental results. Fig. 9 shows the comparison of the surface Mach number distribution measured from CFD analysis and experiment for the NGV mean section at the design exit Mach number.



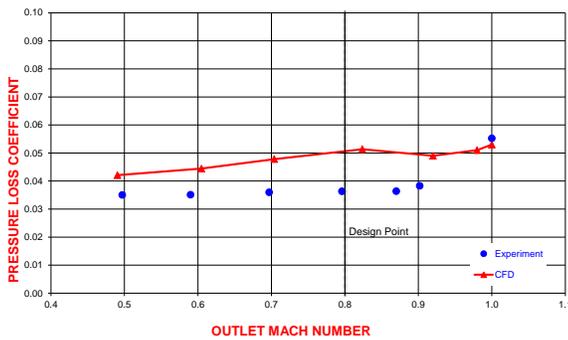
**Fig. 9.** Surface Mach number distribution of NGV at the design point

The results of the cascade tests and the CFD results show a good agreement. It can also be observed that, there is smooth flow acceleration in the aft-loaded blades. Fig. 10 shows the effect of incidence on the Surface Mach number distribution on the NGV aerofoil.



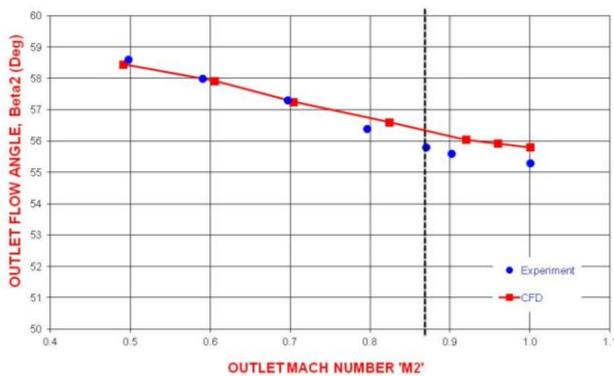
**Fig. 10.** Effect of incidence on the Surface Mach number distribution

Both the CFD analyses and experiments show good match. Incidence changes the acceleration pattern of the flow near the leading edge because of the change in the stagnation point due to inlet incidences. Fig. 11 shows the variation of pressure loss coefficients of the rotor mean section aerofoil as obtained from CFD analysis and experiments.



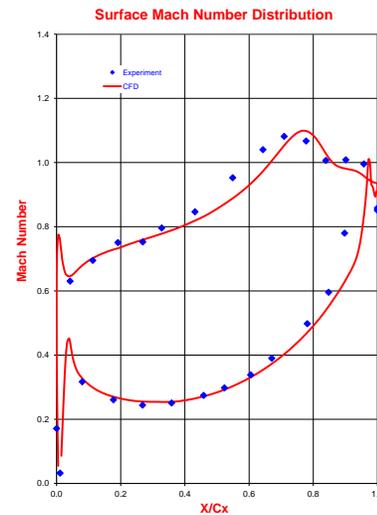
**Fig. 11.** Variation of rotor Pressure loss coefficients for different exit Mach numbers

There is a good agreement between the CFD results and cascade experiments. Fig. 12 shows the variation of the flow exit angles from the rotor cascade for different Mach numbers obtained from CFD analysis and experiment.



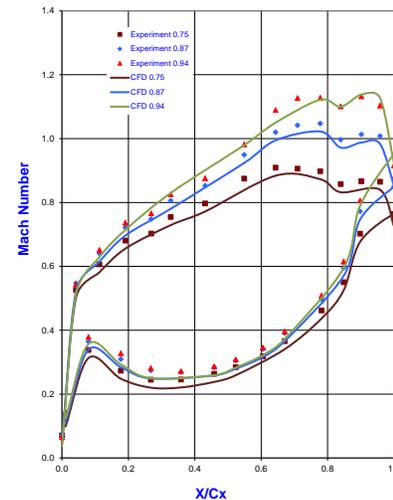
**Fig. 12.** Variation of NGV exit angles for different exit Mach numbers

It can be observed that there is a good agreement between the CFD and experimental results. Fig. 13 shows the comparison of the surface Mach number distribution measured from experiment and CFD analysis for the rotor mean section at the design exit Mach number.



**Fig. 13.** Surface Mach number distribution of rotor at the design point

There is also good agreement between both the experiment and CFD results. Fig. 14 shows the Surface Mach number distributions for different exit Mach numbers of the rotor mean section. This is obtained by varying the exit static pressure.



**Fig. 14.** Surface Mach number distribution of rotor for different exit Mach numbers

### 8 CONCLUSIONS

The aerofoils are generated using the Bezier curve method which takes care of the conflicting aerodynamic, thermal, structural and castability requirements. These profiles are analyzed using 3D N-S code and the results are compared with the 2D Cascade test results. There is a good agreement in terms of the profile loss coefficient, exit flow angles and

isentropic blade surface Mach number distribution between the CFD analysis and the test results.

## 9 ACKNOWLEDGMENTS

The authors acknowledge GTRE and NAL for supporting in publishing of this paper. The authors express their thanks to Mr. David John, Scientist for assistance rendered during the preparation of the manuscript.

## 10 REFERENCES

- [1] Bernard L.Koff; "Gas Turbine Technology Evolution: A Designer's Perspective", Journal of Propulsion and Power, Vol. 20, No. 4, July –August 2004.
- [2] P Hermann; "The EJ200 – Engine Development and Full Certification for Production Release with Eurofighter 2000", AIAA 97 # 99 2661, 35th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 20 – 24 June 1999, Los Angeles, CA.
- [3] Martin Jones, Phil Curnock, Stephen John Bradbrook, Nigel Birch; "Evolutions in Aircraft Engine Design and a Vision for the Future", ISABE – 2002 – 1014.
- [4] J E Allen, F W Armstrong, R M Denning; "Evolution of aviation and propulsion systems: the next fifty years", Proceedings of Institution of Mechanical Engineers, Vol. 209, IMechE 1995.
- [5] Glassman J, "Turbine Design and Application", Volume One and Two, NASA SP-290, 1973.
- [6] Horlock.J.H.; "Axial Flow Turbines," Butterworths, 1966.
- [7] Deich, M.E; Filippov, G.A; and Lazarev,L.Ya; "Atlas of Axial Turbine Bladecharacteristics" 4563 C.E Trans., Mashinostroenie Publishing House, 1965.
- [8] Kuzmin, G.A; "Aircraft Engine Design", FTD - MT – 64, Ohio.
- [9] Blade Gen 12; Visual Turbo-machines Analysis - Vista; ANSYS CFX release 12.1,2009
- [10] User Manual, Auto Blade v8.7, Blade Modeler; Documentation v8.7a, NUMECA release, July 2010.
- [11] Pritchard, L.J; " An Eleven Parameter Axial Turbine Airfoil Geometry Model" ASME paper 85-GT-219.
- [12] Hartmut Prautzsch, Wolfgang Boehm, Marco Paluszny; "Bezier and B-spline Techniques" , ISBN 3-540-43761-4, Springer Verlag, 2002.
- [13] Daniel Karpowitz; "Bezier Curve Fitting method for existing Turbine Blade Design", Journal of Applied Engineering Mathematics, April 2005, Vol. 1
- [14] Fazil, J and Jayakumar,V; " Investigation Of Airfoil Profile Design Using Reverse Engineering Bezier Curve" , ARPJN Journal of Engineering and Applied Sciences, Vol 6, July 2011
- [15] Kyoungwoo Park, Byeong Sam Kim, Juhee Lee, and Kwang Soo Kim," Aerodynamics and Optimization of Airfoil Under Ground Effect", World Academy of Science, Engineering and Technology 52, 2009
- [16] Ramana Murthy, S.V ; Kishore Kumar, S ; "Parametric Study Of Axial Flow Turbine For Mean-Line Design And Blade Elements", GTIndia-2012-9589, Proceedings of the ASME 2012 Gas Turbine India Conference, IIT Bombay, Mumbai.
- [17] CFX-12.0; "User Documentation," ANSYS India Inc.
- [18] ICEM-CFD; "User Documentation," ANSYS India Inc.
- [19] MS Ramachandra, KMM Swamy, MR Narasimhaswamy; " Facility for Transonic Cascade Testing at NAL", NAL – TM – PR – 001 (007) – 1 – 83, September 1983.