International Journal of Mechanical Engineering (IJME)
ISSN(P): 2319-2240; ISSN(E): 2319-2259
Vol. 3, Issue 3, May 2014, 37-46

International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations

CFD ANALYSIS FOR LINEAR BLADE CASCADE OF A TURBINE

SURESH PITTALA & AWASH TEKLE TAFERE

School of Mechanical and Industrial Engineering, Hawassa University, Hawassa, Ethiopia

ABSTRACT

The main objective of this present investigation is computational study of the effect of pressure, velocity, temperature, kinetic energy, turbulence eddy dissipation, total temperature and total pressure on flows through a linear cascade. The main purpose of this project is investigation of flow characteristics and heat transfer around the blade. A 3D Navier-Stokes flow solver was applied to characterize flow, to support the flow phenomenon. The program highlights the flow analysis results and performance evaluation of a turbine blade design. The method of computational fluid dynamics (CFD) is used to investigate the flow in a linear cascade. CFD analysis is becoming a prerequisite event for design, testing and optimization of practical engineering systems. Engineers can make use of CFD tools to study about the flow, modeling the phenomenon in design and predict the system performance. The experiments were performed on models deriving from a stator of a high-pressure turbine. The flow understanding obtained by the current project can be used to guide the design of turbine blades at different flow conditions. Data include blade pressure distributions and velocity plots. CFD analysis of this design is carried out with an objective of assessing the blade performance.

KTYWORDS: Turbine, Linear Cascade, CFD

INTRODUCTION

Flow in turbines is complex and three dimensional. An important problems that arises in the design and performance of axial flow turbines is the understanding, analysis, prediction and control of flows. The flow behavior in a turbine blade cascade is explained with pressure, velocity, temperature and streamline plots

Axial flow turbines were developed in the late 19th century. Today's turbines for power generation are capable of delivering 600MW power or more. Their design is critically dependent upon advanced fluid mechanics and the cascade mode is an essential tool in turbine blade analysis. A small but consistent improvement in blade efficiency will have corresponding economic implications for the nation. Even efficiency improvements by a fraction of percent lead to considerable gains in the power output of turbines. Simultaneously, they lead to a reduction of costs of operation. Turbine manufacturers are constantly competing to produce more efficient, higher thrust and lighter engines. Correspondingly, the research and development of the turbo machinery components is directed for higher efficiency, higher power output and less weight. Mainly by the reasons given above, a great number of studies and experiments were performed in order to enhance the efficiency of turbo machines and to understand the loss creation phenomenon.

To achieve these goals, the study of flows is essential. Flow through turbines is highly complex and three dimensional due to three dimensional blading, boundary layer development on the blades and walls. A good understanding of flow physics help designer to improve efficiency and performance. Hence in turbines, the need of understanding the flow behavior is very important. The basic function of the blades is to turn the air to the required angle. Unlike an isolated airfoil for external flow application, blades of a turbo machines including compressor and turbine are used in a row and referred to as a “cascade”. The method of Computational Fluid Dynamics (CFD) is utilized in this project for the study of flow effects. A model of a linear cascade blade is built and the flow through the cascades is stimulated using a commercial CFD code, namely ANSYS-CFX 12. A flow analysis through a low pressure turbine cascade passage with medium of 5% turbulence intensity is presented in this work. The turbulence of the flow was modeled by the standard k-ε turbulence model. The standard wall function was applied. The flow was simulated as steady, incompressible flow.

Giel, P. W.; Thurman, D. R.; Lopez, I.; Boyle, R. J.; Van Fossen, G. J[1] have presented three dimensional flow field measurements for a large scale transonic turbine blade cascade. Flow field total pressures and pitch and yaw (rotation) flow angles were measured at an inlet Reynolds number of 1.0 x 10and at an isentropic exit Mach number of 1.3 in a low turbulence environment. Flow field data was obtained on five pitch wise or span wise measurement planes, two upstream and three downstream of the cascade, each covering three blade pitches. The large scale allowed for very detailed measurements of both flow field and surface phenomena.

Giel, P.W., Thurman, D.R., Van Fossen, G.J., Hippensteele, S.A, and Boyle, R.J. [2, 3] Turbine blade end wall heat transfer measurements are presented for a range of Reynolds and Mach numbers. Data were obtained for Reynolds numbers based on inlet conditions. Tests were conducted in a linear cascade at the NASA Lewis Transonic Turbine Blade Cascade Facility. The test article was a turbine rotor with 136^0 of turning and an axial chord of 13 cm. The large scale allowed for very detailed measurements of both flow field and surface phenomena. The flow field in the cascade is highly three dimensional as a result of thick boundary layers at the test section inlet. Endwall heat transfer data were obtained using a steady-state liquid crystal technique.

Garg, V. K., and Ameri, A.A. [4] Two versions of the two-equation *k*–ω model and a shear stress transport (SST) model are used in a three-dimensional, multi-block, Navier–Stokes code to compare the detailed heat transfer measurements on a transonic turbine blade. It is found that the SST model resolves the passage vortex better on the suction side of the blade, thus yielding a better comparison with the experimental data than either of the *k*–ω models. Computation of the production term in the turbulence equations for aerodynamic applications, and the relation between the computational and experimental values for the turbulence length scale, and its influence on the passage vortex on the suction side of the turbine blade were also addressed.

Wilcox, D.C. [5] straight forward modifications to the k-omega two-equation model of turbulence are proposed and tested for both wall-bounded and free-shear flows. The modifications eliminate the k-omega model's sensitivity to the free stream value of omega without destroying its accuracy for boundary layers in adverse pressure gradient, arid for transitional boundary layers. The revised model is shown to yield satisfactory agreement, with measurements for the far wake, the mixing layer and the plane jet.

Menter, F.R. [6] Two new two-equation eddy-viscosity turbulence models will be presented. They combine different elements of existing models that are considered superior to their alternatives. The first model, referred to as the baseline (BSL) model, utilizes the original *k-u* model of Wilcox in the inner region of the boundary layer and switches to the standard model in the outer region and in free shear flows. It has a performance similar to the Wilcox model, but avoids that model's strong free stream sensitivity. The second model results from a modification to the definition of the eddy-viscosity in the BSL model, which accounts for the effect of the transport of the principal turbulent shear stress. The new model is called the shear-stress transport-model and leads to major improvements in the prediction of adverse pressure gradient flows.

The NASA Glenn Research Center (GRC) linear transonic blade cascade of a largescale1-3 is a test case specially designed to provide a detailed high Mach number rotor blade flow and heat transfer data set to CFD code developers and users [7]. A brief description of the experimental facility and available data are as follows. The transonic turbine blade linear cascade with a large turning angle consists of 11 passages. A highly three-dimensional flow field was obtained in the blade passage by allowing the endwall turbulent boundary layer to develop in a long inlet section upstream of the cascade. To define the cascade inlet flow conditions, aerodynamic probe measurements were madeat a section located one axial chord upstream of the blade leading edge plane. For the endwall heat transfer measurements under conditions of approximately constant wall heat flux, power settings were ranged from 200 to 1200 Watts. The net surface heat fluxrate used to determine the Stanton number was the heater power corrected for conduction losses and for radiative heat transfer. The steady-state liquid crystal technique was used for surface temperature measurements. Local endwall heat transfer measurements were performed at eight combinations of the inlet Reynolds number, Re, the isentropic exit Mach number, Mex, and the free stream turbulence intensity. Additional details are given by Giel et al 2.Previously, the experimental data set obtained in the NASA GRC was used by Garg andAmeri4 to examine capabilities of two-equation turbulence models for prediction of blade heat transfer. Kalitzin et al 5 computed blade and endwall heat transfer using the Durbin four-equation v2-f model and the Spalart-Allmaras (S-A) one-equation model. Goriatchev et al 6 analyzed secondary flows and pressure losses using the S-A model. Ivanov et al 7 used different versions of S-A, k-e and k-w turbulence models with the same grids. All these studies were performed using Navier-Stokes codes of second order accuracy with block-structured computational grids consisting of about 350,000 to550,000 cells (for one half of the blade passage height, in compliance with the assumption of the time-averaged flow symmetry). Analysis of the computational data reported allows a conclusion that computations with grids of such a size produce grid.

NUMERICAL METHODS

Numerical Solution

The numerical simulation of the blade-to-blade flow field in the current research was fulfilled by using available ANSYS software, SOLID WORKS, ANSYS CFX 12.2. This was developed by ANSYS CFX was specifically developed for aerodynamic and/or heat transfer analysis of modern turbo machinery configurations, with the capability of predicting both steady state and time-dependent flow fields by solving Euler or Navier-Stokes equations. It allows either serial execution or parallel execution on massively parallel or workstation cluster computing platforms from a single source. The code utilizes a finite-volume, four- stage Runge-Kutta time-marching numerical procedure in conjunction with a flexible multiple grid block geometric representation to permit detailed aerodynamic simulations for complex configurations. The code had been verified for both turbo machinery and non-turbo machinery based applications. In addition to being used for analyzing the steady and unsteady aerodynamics of high-bypass ducted fans involving multiple blades rows, CFX possesses many features which make it practical to compute a number of other flow configurations, such as the cascade flow in the current investigation.

Solution Procedure Sequence

Step 1 involves selecting geometry and flow conditions, defining which specific results are desired, and specifying whether steady state or time-dependent data are required, whether in viscid calculation (Euler equations) is sufficient or whether a viscous flow solution (Navier-Stokes equations) is needed. In Step 2 and 3, the geometry is specified (such as the blade profile) and the flow domain is represented by the computational grid. Grid generation can be handled through commercial packages. In this work, ansys work bench was used. Most of the work to fulfill a desired execution of the code is included in the next step. The standard input file controls operations specific to a particular run of the code. Operations such as the number of iterations, flow conditions, and input/output control are governed by the values specified in the input file. All the boundary condition information is provided by the boundary data file, in which numerous values need to be defined and/or chosen according to the code syntax. The next step is to run the code. During the computation, some real-time output can provide information about the progress of the code running. The convergence history is also available for review. Commercial post-processing software's can be used by to read the output files and create the plots of the flow field.

Problem Definition

The geometry of the linear cascade is that available as shown below in figure 1. A fragment of the cascade is illustrated in the Figure 1, together with a slice of the computational domain. Table 1 covers blade and cascade dimensions and the basic flow parameters at the design inlet flow angle.

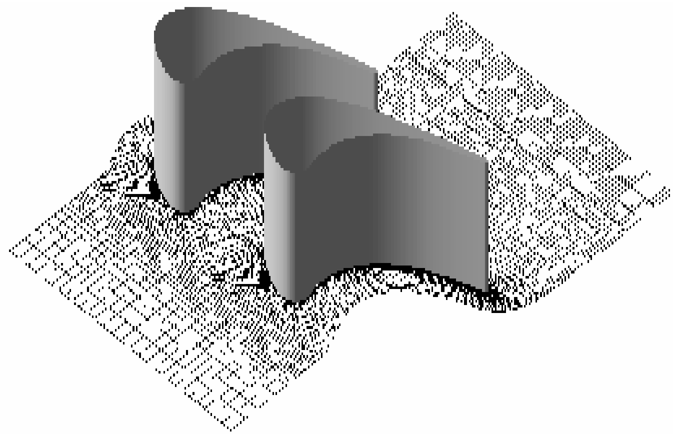


Figure 1: Blade Passage and Slice of the Computational Domain

CFD Procedure of the Current Work

Numerical investigation was performed on the cascade flows for the cascade, using ANSYS CFX. Numerical results fully characterized the flowfield, providing detailed flow information such as flow speed, flow angle, pressure, axial chord, true chord, design inlet flow angle, etc. The flowfield information from CFD simulation was then used to help elucidate the flow physics.

Table 1: Comparission between Classical and New Model Cascade Dimensions and Flow Parameters

Parameter	Classical Model	New Model
Axial chord, C_x, cm	12.7	13
Pitch,cm	13.0	13.5
Span, cm	15.24	16.24
True chord, cm	18.42	19.42
Design inlet flow angle. Degrees	63.6^0	60.6^0
Total turning(at inlet flow angle) degrees	136^0	156^0
Prandtl number, P_r	0.72	0.82
Inlet turbulence intensity,%	0.5	0.7
Inlet turbulence length scale, cm	0.127	0.135

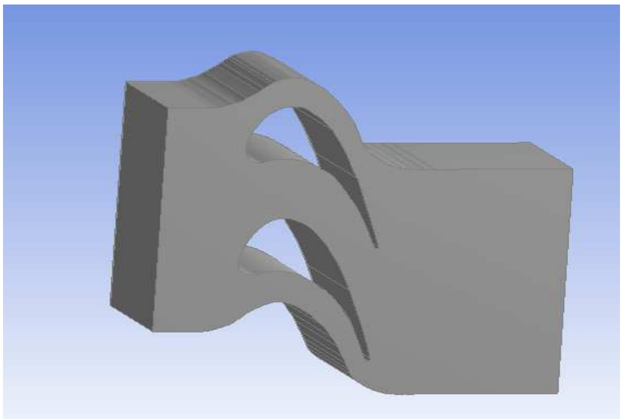


Figure 2: Computational Domain

Meshing

Ansys cfx provides unstructured meshes in order to reduce the amount of time spent generating meshes, simplifying the geometry modeling and mesh generation process, model more complex geometries than can be handled with conventional, multi-block structured meshes, and let the mesh to be adapted to resolve the flow-field features. Ansys cfx can also use body-fitted, block-structured meshes and is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, and wedge elements (or a combination of these) in 3D. This flexibility allows picking mesh topologies that are best suited for particular application. All types of meshes can be adapted in ansys cfx in order to resolve large gradients in the flow field, but the initial mesh (whatever the element types used) outside of the solver, using ansys work bench, T Grid, or one of the CAD systems for which mesh import filters exist must always be generated. The geometry is created by using ansys work bench Design Modeler and the extruded geometry is meshed by cfx mesh. Volume mesh is generated by choosing Generate Volume mesh. The meshed domain is shown in the figure 3: with the mesh details

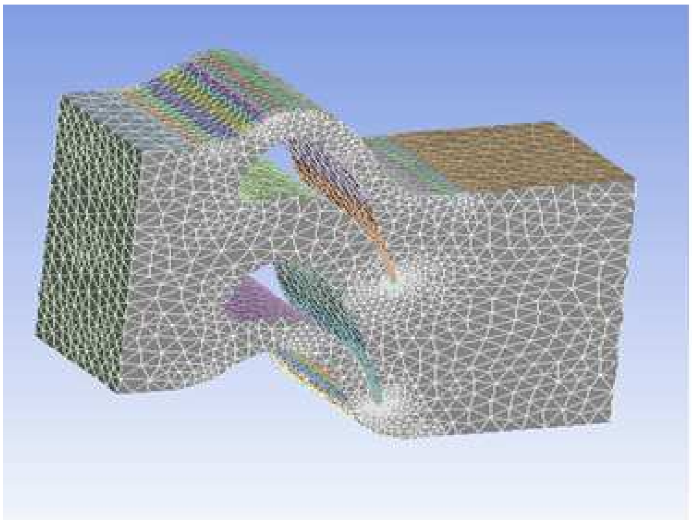


Figure 3: Meshed Domain

Table 2: Mesh Details

Domain	Nodes	Elements	Tetrahedra
Default Domain	417682	2136739	2037489

Boundary Conditions, Boundary Sources and Fluid Properties

To solve the problem, it is necessary to specify appropriate initial and boundary Conditions. The grid is imported in to ANSYS CFX PRE and then domain is selected. The boundary conditions are specified and then the solver control is set up.

Table 3: Default Domain Conditions

Name	Location	Material Type	Models
Default domain	B208	Fluid at ideal gas	Heat Transfer Model = Total Energy Turbulence Model = Shear stress transport Turbulent Wall Functions = Scalable Buoyancy Model = Non Buoyant Domain Motion = Stationary

CFX Pre Process

At the present computations, the fluid (air) is treated as a perfect gas with the specific heat ratio *g* = 1.4. The governing equations are the Reynolds-averaged Navier-Stokes equations and the energy equation written for the total enthalpy. A power-law is adopted to account for the dependency of viscosity on temperature, *m* ~ *T* 0.76.In order to define proper boundary conditions at the 3D computational domain inlet section placed one axial chord upstream of the blade leading edge. At inlet total pressure of 130000 Pa and at outlet Static pressure of 90000Pa is considered as boundary conditions to define the total temperature, total pressure and velocity vector angle and turbulence parameters distributions over the inlet plane of the 3D blade cascade computational domain.

To get the isentropic Mach number required a proper value of static pressure was specified at the outlet boundary located one axial chord downstream of the blade trailing edge. At the solid surfaces of the cascade the no-slip condition was imposed. The constant temperature, *Tw*, of 350 K was specified on the end wall. Remaining walls were treated as adiabatic Wall boundary condition was used in the pitch wise direction. For computational purposes, the flow between the two cascade blades is considered assuming the symmetry.

Inlet Boundary Conditions

Table 4: Inlet Boundary Conditions

Domain	Name	Location	Type	Settings
Default Domain	Inlet	Inlet	Inlet	Flow Direction = Normal to Boundary Condition Flow Regime = Subsonic Heat Transfer = Static Temperature Static Temperature = 350 [K] Mass And Momentum = Total Pressure Relative Pressure = 208000(p) Turbulence = Medium Intensity and Eddy Viscosity Ratio

Outlet Boundary Conditions

Table 5: Outlet Boundary Conditions

Domain	Name	location	Type	Settings
Default Domain	Outlet	Outlet	Outlet	Flow Direction = Normal to Boundary Condition Flow Regime =Supersonic

Wall Boundary Conditions

Table 6: Wall Boundary Conditions

Domain	Name	Location	Type	Settings
Default domain	Default domain default	F209.208, F210.208, F211.208, F212.208, F213.208, F2...	Wall	Heat Transfer = Adiabatic Wall Influence On Flow = No Slip Wall Roughness = Smooth Wall

CFX Solver

For the simulation run, inlet flow conditions (Pt1, Tt1, α1) were specified. Outlet static pressure was chosen to achieve the desired inlet Reynolds number. Wall boundary condition was used in the pitch wise as well as span wise direction. The simulation run had stable convergence history.

CFX Post Processor

In this the results such as contours, Vectors and Streamlines can be viewed. In the results velocity contour, total pressure contour, total temperature contour, velocity stream line, velocity vectors, turbulence kinetic energy contour are shown.

RESULTS AND DISCUSSIONS

Pressure Contours

By Taking Inlet Pressure 250(K)

In the pressure contours shown in figure 4. There is a gradual decrease in the pressure between the blades from leading edge to the trailing edge which shows that there is no shock formation. The inlet pressure used for the pressure output graph is 208000. It is possible to distinguish the region of the blade end wall starting from the low pressure point on the suction side. Since the boundary conditions are applied on the end wall and there being a sharp curve, we observe there is sudden increase in pressure in the figure. It can also be interpreted as the Velocity being high and the boundary conditions being given on the end wall there might be a sudden increase in pressure.

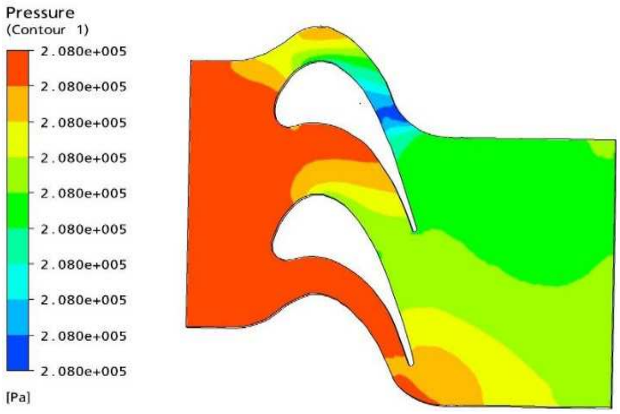


Figure 4: Total Pressure Counter

Temperature Contours

Two different total temperatures are taken those are 350(k)and 315(k).There isno drastic variation observed in 315(K),but there is variation in 350(k)temperature.When local temperatures were used, we found a small vatiation near theinlet.

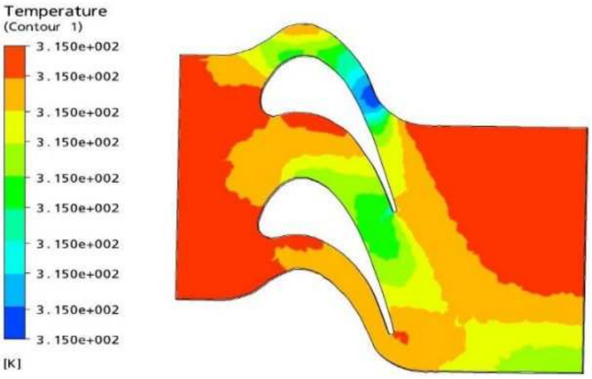


Figure 5: Total Temperature Counter

Velocity and Stream Lines

Here it is observed that there is a flow separation on the suction side at the trailing edge which results in Kutta condition near the exit. The recirculation in the Figure 6 clearly shows the variation in velocity streamline. There is a pressure difference in the suction and pressure side of the blade and hence we get a Lift force for the blade.

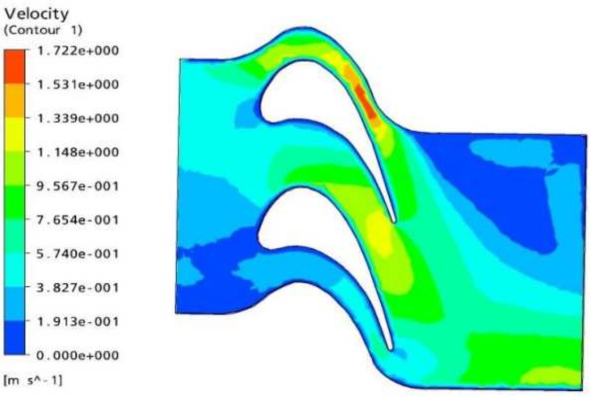


Figure 6: Velocity Streamlines 3D

Turbulence Kinetic Energy

Because of the pressure gradient from leading edge to trailing edge on the suction side near the wall as there is a constrictive passage between the wall and the blade side, the zone is subjected to high flow velocity as shown in the figure 7. Analysis of the computational data reported allows a conclusion that computations with grids produce wall pressure distribution, but the grid sensitivity of local heat transfer however remains questionable.

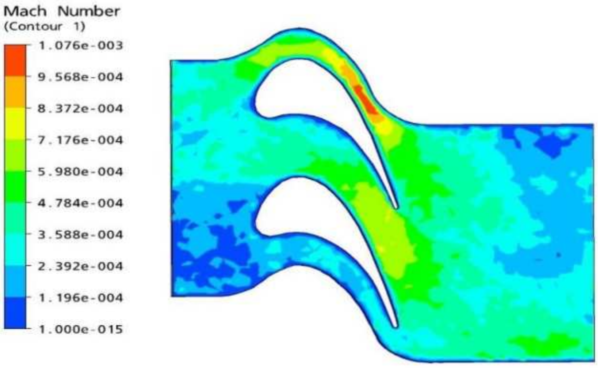


Figure 7: Mach No Contour

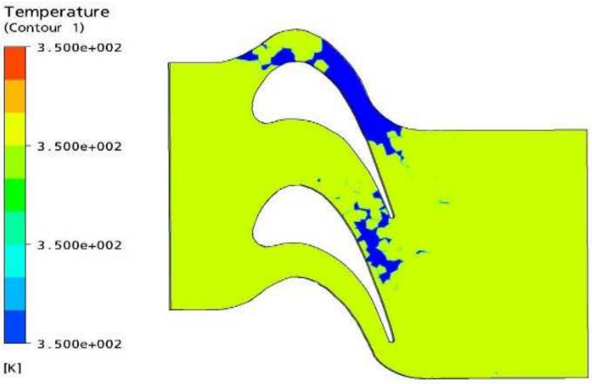
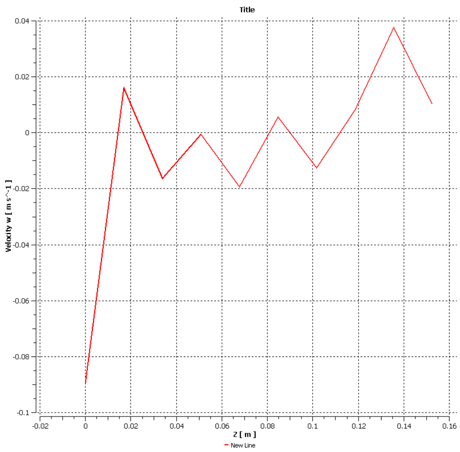
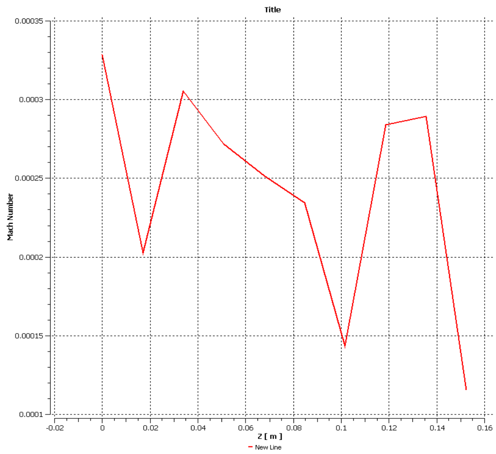


Figure 8: Temperature Contour

Graphs



Velocity (u) vs Length (Z)



Mach Number vs Length (Z)

CONCLUSIONS AND SUGGESTIONS

A CFD analysis methodology on a turbine cascade has been presented in this work. By taking same inlet pressure and different total temperatures the variations of resuts are observed. The effects of parameters like pressure, temperature, velocity and turbulence kinetic energy ,mach number distribution are considered.Results are presented through flow field contours. The flow was examined using contours of velocity, pressures, velocity vectors and stream lines.

Some of the Key Findings of this Work Are Listed as follows

- Because of the drastic pressure gradient from leading edge to trailing edge on the suction
- Side near the wall as there is a constrictive passage between the wall and the blade side, the zone is subjected to high flow velocity.
- Regions of high turbulence intensity are identified.
- Here we observe that there is a flow separation on the suction side at the trailing edge which results in Kutta-condition.

In general, the developed CFD model provided useful understanding of flow with in turbine. The model will be used as a basis for further work on considering flow with in turbine. The study enhances our knowledge on secondary flows for future turbine. Further progress on secondary flows is dependent on a deeper understanding of the physics of end wall boundary layers.

REFERENCES

1. Giel, P. W., Thurman, D. R., Lopez, I., Boyle, R. J., Van Fossen, G. J., Jett, T.A., Camperchioli, W. P., and La, H., "Three-dimensional flow field measurements in a transonic turbine cascade," ASME Paper 96-GT-113 (1996).
2. Giel, P. W., Thurman, D.R., Van Fossen, G.J., Hippensteele, S.A, and Boyle, R.J., "Endwall heat transfer measurements in a transonic turbine cascade," ASME Paper 96-GT-180 (1996)
3. Giel, P. W., and Gaugler, R.E., "NASA Blade 1. Endwall heat transfer data. Version 1," NASA-Glenn Research Center, Turbine Branch, CD ROM (2001).
4. Garg, V. K., and Ameri, A.A., "Two-equation turbulence models for prediction of Heat transfer on a transonic turbine blade," Int. J. Heat and Fluid Flow, Vol. 22, 593-602 (2001).
5. Kalitzin, G., and I accarino, G., "Computation of heat transfer in a linear turbine cascade", Center for Turbulence Research Annual Research Briefs, 277-288 1999).Wilcox, D.C., "A two-equation turbulence model for wall-bounded and free-shear flows", AIAA Paper 93-2905 (1993).
6. Menter, F. R., "Two equation eddy-viscosity turbulence models for engineering Applications", AIAA Journal, Vol. 32, 1598-1605 (1994)
7. Numerical Calculation of the flow in a centrifugal blower impeller using Cartesian grid procedure of 2nd WSEAS int. Conference on applied and theoretical mechanics, Venice, Italy, November 20-22, 2006, According to john S. Anagnostopoulos.