Turbine Blade Cascade Heat Transfer Analysis Using CFD –A Review

MD.Hasheer.Sk
Assistant Professor
Department of Mechanical Engineering
RVR&JC College of Engineering

Srinivas Kolla
Professor
Department of Mechanical Engineering
RVR&JC College of Engineering

Abstract

Heat transfer analysis on a turbine blade cascade using CFD is reviewed in this paper based upon the literature available. The flow conditions across turbine blades are complex because of three dimensional configuration of the blade. Axial flow turbine blade design is hectic because understanding of control flows, prediction and analysis is a tedious task. The flow behaviour in a turbine blade cascade can be understood to some extent by analyzing pressure, velocity, temperature and streamline plots. The fluid flow simulation coupled with heat transfer analysis is a common practice in CFD, to study how behaviour influences heat transfer. By optimizing design variables more efficient systems can be designed. A 3-D Navier-Stokes flow solver was applied to characterize flow which supports the flow phenomenon.

Keywords: CFD Analysis, Heat Transfer, Turbine Blades.

I. INTRODUCTION

Turbines have been considered energy workhorses for generations. Regardless of the type of fuel used, turbines are at the heart of almost all of the world's electricity generating systems. Their design is critically dependent upon advanced fluid mechanics and the cascade mode is an essential tool in turbine blade analysis. To achieve these goals, the study of flows is essential. It is well known that three-dimensional secondary flows in blade rows can dramatically affect performance of gas turbines. Consequently, there is a strong need for computational models/tools that would allow accurate predictions of the secondary low effects both on the pressure losses and heat transfer. 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 used to study the flow effects.

Measurements were made in a linear cascade facility at the NASA Glenn Research center[1-2]. A turbine blade with 136° of turning, an axial chord of 127mm and a span of 152.4mm was tested in a highly three-dimensional flow field resulting from thick inlet boundary layers. Data were obtained by a steady-state technique using a heated, isothermal blade.

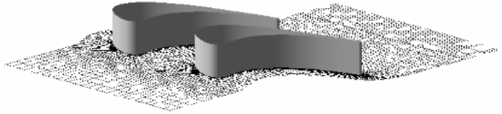

Fig. 1: Blade Passage[1-2]

Levchenya and E. Smirnov[3] have presented results of numerical simulation of three-dimensional turbulent flow and end wall heat transfer in a transonic turbine cascade by employing several turbulence models, an analysis of Computational Fluid Dynamics (CFD) predictability was done in comparison with measurements in a linear cascade at the NASA Glenn Research Center transonic turbine blade cascade facility. It has been concluded in particular that rather fine computational grids are needed to get grid-independent data on the end wall local heat transfer controlled by complex 3D structure of secondary flows. Giel. P. W., Thurman. D. R., Lopez, I.; Boyle, R. J[1].; Van Fossen, G. J[2] have presented three dimensional flow field measurements for a large scale transonic turbine blade cascade. Flow field total pressures and pitch and yaw flow angles were measured at an inlet Reynolds number of 1.0 x 10(exp 6) and at an isentropic exit Mach number of 1.3 in a low turbulence environment. Flow field data was obtained on five pitch wise/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[1-2]. 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 12.7 cm.

The flow field in the cascade is highly three dimensional as a result of thick boundary layers at the test section inlet. End wall heat transfer data were obtained using a steady-state liquid crystal technique[4].

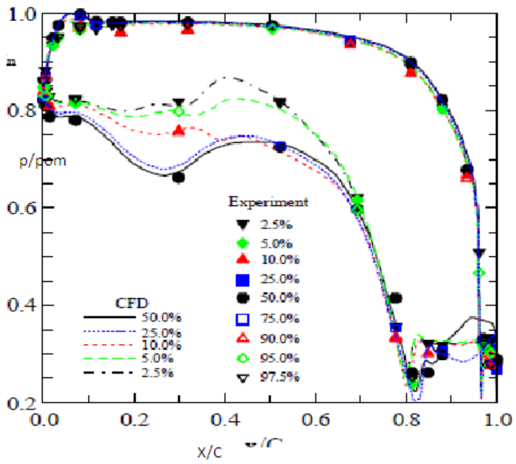

Fig. 2: Comparison of Computed And Measured Static Pressure Distribution Over The Blade Surface For Various Span Positions[3]

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. Wilcox, D.C[7]. Straightforward 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[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. Preference for the k-ω model stems from its robustness and absence of the distance to the wall in its formulation. Which makes it the original k-ω model[5] were the k-ω model and Menter's SST model, Menter's SST model among these performed the best. Additional details are given by Giel P.W et al [2]. Previously, the experimental data set obtained in the NASA GRC was used by Garg and Ameri[4] to examine capabilities of two-equation turbulence models for prediction of blade heat transfer[7] computed blade and end wall heat transfer using the Durbin four-equation v2-f model. Ivanov et al[8] used different versions of S-A, k-e and k-w turbulence models with the same grids.

Van Fossen et al[1-2] also found that the turbulence length scale affects the stagnation region heat transfer. Even when the experimental length scale is known, such as in the present case, the computational value may not be the same. The turbulence length scale also affects the passage vortex on the suction side of the blade-an essentially three-dimensional phenomenon. It is therefore essential to use a correct value for the turbulence length scale.

II. OVERVIEW OF HEAT TRANSFER AND FLUID FLOW SIMULATION

The process of using computers to study fluids that are in motion, and how the fluid flow behavior influences heat transfer in the systems numerically is called computational fluid dynamics (CFD) analysis

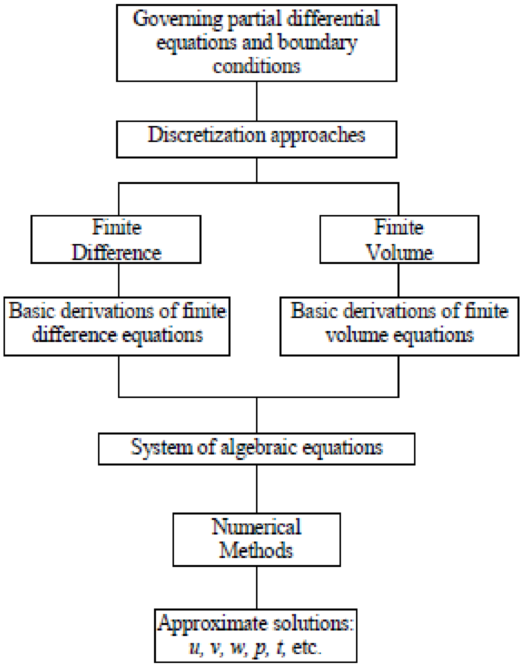

Fig. 3: Overview Process of The Computational Solution Procedure

A. Governing Equation:

The governing equations used for the above study are the unsteady, three-dimensional Navier-Stokes equations, and in the non-dimensional, vector form, are given by

$$\frac{\partial Q}{\partial t}+\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y}+\frac{\partial H}{\partial z}=0 \qquad (1)$$

Where Q is a vector of dependent or unknown variables, which can be written as

$$Q = [\rho, \rho u, \rho v, \rho w, E_t]^T \qquad (2)$$

The quantities F, G, and H in Eq. are the flux vectors that can be expressed as

$$F=\begin{bmatrix} \rho u \\ \rho u^2+p-\tau_{xx} \\ \rho uv-\tau_{xy} \\ \rho uw-\tau_{xz} \\ (E_t+p)u-u\tau_{xx}-v\tau_{xy}-w\tau_{xz}-q_x \end{bmatrix} \qquad (3)$$

$$G=\begin{bmatrix} \rho v \\ \rho uv-\tau_{yx} \\ \rho v^2+p-\tau_{yy} \\ \rho vw-\tau_{yz} \\ (E_t+p)v-u\tau_{yx}-v\tau_{yy}-w\tau_{yz}-q_y \end{bmatrix} \qquad (4)$$

and

$$H=\begin{bmatrix}\rho w\\ \rho uw-\tau_{xz}\\ \rho vw-\tau_{yz}\\ \rho w^2+p-\tau_{zz}\\ \left(E_t+p\right)w-u\tau_{xz}-v\tau_{yz}-w\tau_{zz}+q_z\end{bmatrix} \quad (5)$$

Where ρ is the density, u, v and w are the Cartesian components of velocity, and E_t is the specific total energy. The total energy of the fluid, is expressed in terms of internal energy and kinetic energy as

$$E_t=\rho\left(e+\frac{u^2+v^2+w^2}{2}\right) \quad (6)$$

The components of shear stress that appear in the expressions for the flux vectors are given by

$$\tau_{xx}=\frac{2}{3}\frac{\mu}{\mathrm{Re}}\left(2\frac{\partial u}{\partial x}-\frac{\partial v}{\partial y}-\frac{\partial w}{\partial z}\right) \quad (7)$$

$$\tau_{yy}=\frac{2}{3}\frac{\mu}{\mathrm{Re}}\left(2\frac{\partial v}{\partial y}-\frac{\partial u}{\partial x}-\frac{\partial w}{\partial z}\right) \quad (8)$$

$$\tau_{zz}=\frac{2}{3}\frac{\mu}{\mathrm{Re}}\left(2\frac{\partial w}{\partial z}-\frac{\partial u}{\partial x}-\frac{\partial v}{\partial y}\right) \quad (9)$$

$$\tau_{xy}=\frac{\mu}{\mathrm{Re}}\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right) \quad (10)$$

The heat flux terms in can be written as

$$q_x=-\frac{\mu}{\left(\gamma-1\right)M_{ref}^2\,\mathrm{Re}\,\mathrm{Pr}}\frac{\partial T}{\partial x} \quad (11)$$

$$q_y=-\frac{\mu}{\left(\gamma-1\right)M_{ref}^2\,\mathrm{Re}\,\mathrm{Pr}}\frac{\partial T}{\partial y} \quad (12)$$

$$q_z=-\frac{\mu}{\left(\gamma-1\right)M_{ref}^2\,\mathrm{Re}\,\mathrm{Pr}}\frac{\partial T}{\partial z} \quad (13)$$

Using the definition of Mach number, temperature can be calculated as

$$T=\frac{\gamma M^2 p}{\rho} \quad (14)$$

The molecular viscosity is computed using Sutherland’s law, given by

$$\mu=\frac{\left(1+\overline{T}\right)}{T+\overline{T}}T^{3/2} \quad (15)$$

Where $\overline{T}$ is Sutherland’s constant, which is equal to 110 K/Tref.

III. CONCLUSIONS

Computational study of the effect of pressure, velocity, temperature, kinetic energy, turbulence eddy dissipation..etc on flows through a linear cascade is reviewed. It is observed that the above parameters varies from leading edge to the trailing edge.

It is suggested, the study can be further carried out with secondary flows by 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., “*Endwell heat transfer measurements in a transonic turbine cascade,*” ASME Paper 96-GT-180 (1996)

[3] Alexander M, Levchenya and EvgueniM. Smirnov “cfd-analysis of 3d flow structure and end wall heat transfer in a transonic turbine blade Cascade: effects of grid refinement” CD-ROM Proc. West- East High Speed Flow Field Conference - WEHSFF'07, Moscow, 12 p

[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] 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] Kalitzin, G., and Iaccarino, G., “*Computation of heat transfer in a linear turbine cascade*”, Center for Turbulence Research Annual Research Briefs, 277-288 (1999)

[8] Ivanov, N. G., Levchenya, .., Ris, V. V., and Smirnov, E. M. [2002]. “*Calculation of threedimensional flow and heat transfer in an experimental model of a blade cascade on the base of one- and two equation turbulence models”*. Proc. 3rd Russian National Heat Transfer Conf., MPEI Publishers, Moscow, Russia, Vol.2, pp. 147-150 (in Russian)

[9] 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)

[10] Yahya,S.M.,” *Turbines, Compressors and fans* “,Satya Prakashan,New Delhi,1972. CFX help manual.www.ansys.com.

[11] Ameri, A. A., Arnone, A. [1994]. “*Prediction of turbine blade passage heat transfer using a zero and two-equation turbulence models”*. ASME paper. No. 94-GT-122, 8 p

[12] Graziani, R. A., Blair, M. F., Taylor, J. R., and Mayle, R. E. [1980]. “*An experimental study of end wall and airfoil surface heat transfer in a large scale turbine blade cascade”*. ASME J. Eng.Power, Vol. 102, pp. 257-267

[13] Holley, B. M., Langston, L. S. [2006]. “*Surface shear stress and pressure measurements in a turbine cascade*”. Proc. ASME Turbo Expo 2006, GT2006-90580, 10 p

[14] Langston, L. S., Nice, M. L., Hooper, R. M. [1977]. “*Three-dimensional flow within a turbine cascade passage”*. ASME J. Eng. Power, Vol. 99, pp. 21-28

[15] Lee, H. G., Yoo, J. Y. [1997]. “*Numerical simulation of turbulent cascade flows involving high turning angles”*. Computational Mechanics, No.20, pp. 247-260.

[16] Levchenya, .., Ris, V. V., and Smirnov, E. M. [2006]. “*Testing of turbulence models as applied to calculations of 3D flow and end wall heat transfer in cascades of thick vane blades”*. Proc. 4th Russian National Heat Transfer Conf., MPEI Publishers, Moscow, Russia, Vol.2, pp.167-170 (in Russian)

[17] Menter, F. R. [1994]. “*Two equation eddy-viscosity turbulence models for engineering applications”*. AIAA Journal. Vol. 32. pp. 1598-1605

[18] Smirnov, E. M., and Zaitsev D. K. [2004]. “*The finite-volume method in application to complexgeometryfluid dynamics and heat transfer problems”*. Scientific and Technical Bulletin of theSt.-Petersburg State Technical University, No. 2(36), pp.70-81 (in Russian)

[19] Wilcox, D.C. [1993].” *A two-equation turbulence model for wall-bounded and free-shear flows”*. AIAA Paper, AIAA-93-2905

[20] Boyle,R.J.,Giel,p.,1992. Three-dimensional Navier -stokes heat transfer prediction for turbine blade rows. AIAA Paper 92-3067 Chima. R.V.,1996. A k–ω model for quasi-three-dimensional turbo machinery flows. AIAA Paper 96-0248

[21] Vinícius Guimarães Monteiro et al 2012 “Numerical Simulation of Performance of an Axial Turbine First Stage” J. Aerosp. Technol. Manag., São José dos Campos, Vol.4, No 2, pp. 175-184, Apr.-Jun., 2012