NUMERICAL SIMULATION OF FLOW FIELD IN AN ANNULAR TURBINE STATOR WITH FILM COOLING

Jun Zeng[*], Bin Wang[*], Yong Kang[**]
[*]**China Gas Turbine Establishment , Chengdu 610500, China**
[**]**Harbin Institute of Technology, Harbin 150001, China**

Keywords: *CFD, turbine, stator, film cooling*

Abstract

In the present paper steady three-dimensional numerical calculation is performed in order to investigate the effects of film cooling on turbine stator flow fields. A row of cooling holes is modeled as a slot. Film cooling flow in the stator has been simulated by using source terms. The effects of film cooling on the aerodynamic loss in terms of energy loss coefficient and the outlet flow angle are analyzed. The simulation results are agreed well with the experimental results.

1 Introduction

In order to obtain maximum thermodynamic cycle efficiency a high temperature level is required in the high pressure (HP) turbines of modern gas turbines. The inlet temperature level of turbine is usually by far higher than the maximum allowable temperature of even the most advanced materials. Therefore, every modern HP turbine needs a sophisticated cooling system. The most frequently used cooling method of today is film cooling. Relatively cool compressor air is injected through numerous holes and slots on the blade and endwall surfaces of a HP turbine. Apart from the desired influence of the injected cooling air on the heat transfer coefficients of the blade and endwall surfaces, the cooling jets have a considerable effect on the main flow as well. As a consequence, the effects of film cooling have to be taken into account in the aerodynamic design of a HP turbine. Modern commercial Navier-Stokes solvers provide the designer in the turbomachinery environment with a variety of options to simulate the flow inside the blade passage of a film cooling turbine. The CFD modeling of film cooling holes can be achieved by two primary numerical methods of different complexity.

The first numerical method is source term modeling technique which is the fastest and least complex method to introduce the effects of film cooling into a 3D Navier-Stokes calculation of a turbine. This method is computationally least expensive and easy to apply, making it well suitable for the fast turn-around times, which are required in the modern design processes. The cooling flow is taken into account by a distribution of various sources of mass, momentum and energy on the blade and endwall surfaces. Turner et. al using source term method studied flow fields of GE90 engine turbine which consist of 18 blade rows at the take off condition. Comparisons have been made to measurements, and good agreement has been demonstrated [1].

The second numerical method is full modeling of every single cooling hole represents the most complex approach. Using this method every cooling hole, including the cooling air plenum is discretized. Obviously, turn-around times and engineering efforts are by far higher if compared with the source term method. The reward of applying this method to a film cooling turbine is a large amount of very detailed flow information [2].

The present study is focused on investigation of flow fields in annular turbine stator with film cooling using source term method because full modeling of every single cooling hole doesn't allow it to effectively used in a design environment. An operating condition with film cooling is simulated using commercial CFD software CFX-TASCflow [3]. When simulating turbulence, the Shear-Stress-Transport (SST) turbulence model is adopted [4]. The predicted results are compared to the test data. These comparisons are used to evaluate effects of film cooling.

2 Annular turbine stator

The annular turbine stator under investigation is a transonic stator typically used in aero-engine turbines. Modern 3D design techniques, such as 3D CFD, compound lean and meridional contouring form are used to reduce secondary loss in the stator blade. In order to protect stator blade from the hot gas around it, the stator blade has many rows of film cooling holes on pressure and suction surface, and has a coolant slot on trailing edge. Besides, there are a lot of film holes on the lower and upper endwall, respectively (Fig.1).

3 Numerical simulation

3.1 Grid generation

In order to obtain reliable CFD solutions in a limited time frame, as usually required in practical situations, generating high quality mesh in a short time is as important as the performance of the solver itself. This is particularly true in the turbomachinery analysis. Besides, the mesh quality has directly effect on solution accuracy, speed and convergence behavior.

A structured multi-block grid system is generated with CFX-TurboGrid 1.6 software [5]. It possesses nine pre-defined library of templates which suit for various rotating machinery types, including axial, mixed-flow, radial devices, covering almost all the turbomachinery applications encountered in practice. For axial turbine stator blades, axial-high stagger blade template is used to generate high-quality grid (Fig.2).

The computational domains consist of a H-type grid block in INBLOCK ($11\times67\times31$), an O-type grid block surrounding the blade ($354\times8\times31$ MAIN), a C-type grid block around the O-type grid from leading edge to trailing edge ($324\times8\times31$ CGRID), two H-type grid blocks extending from the trailing edge to a plane region on the pressure and suction sides ($45\times8\times31$ TEDPS and $25\times8\times31$ TEDSS) and a H type grid block in OUTBLOCK ($17\times35\times31$) (Fig.3).

3.2 Computational code

CFX-TASCflow versions 2.12.01 were used for the calculations [3]. CFX-TASCflow utilizes the 3-D Navier-Stokes equations to calculate primitive flow variables such as velocity and pressure over the flow domain. The solver uses finite element-like finite volume discretization to formulate and solve algebraic equations over the entire grid or node distribution in the flow domain. With the attachment of boundary surfaces, specification of the boundary and initial conditions, and several control parameters, the solver resumes computing for the solution. The capacity of the computing facility, the degree of grid refinement in the model, the quantity of residual for termination of computation, etc. determines the time required for convergence.

Different numerical schemes are available varying from first order upwind difference, to a mass weighted one, to modified linear profile, to a truly second order linear profile skew.

For the advective terms, the modified linear profile with the PAC (Physical Advection Correction) scheme has been employed.

3.3 Turbulence modeling

In order to obtain accurate solutions it is essential to use advanced turbulence models, which reproduce the physics of turbulence. This is a key item for most engineering application.

In most CFD codes the k-ε turbulence model in combination with standard wall function is widely used. These wall functions assume that the first near-wall grid node is within the universal logarithmic law of the boundary layer. In practice, however, it is often difficult to generate a mesh beforehand, which satisfies this condition, and the violation of this requirement leads to poor prediction of flow fields and machine performances. Additionally, this model is known to have the tendency to overestimate the shear stress and heat transfer at walls. This is problematic, because it results in an optimistic prediction of machine performance, with flow separation being underestimated.

The k-ω model is substantially more accurate than k-ε in the near wall layers and has therefore been successful for flows with moderate adverse pressure gradients but fails for flow with pressure induced separation. In addition the ω equation show a strong sensitivity to the values of ω in the free stream outside the boundary layer. The free stream sensitivity has largely prevented the ω equation from replacing the ε equation as the standard scale-equation in turbulence modeling despite its superior performance in the near wall region.

In order to avoid this problem a combination of the k-ω model near the wall and k-ε model away from the wall has been proposed by Menter [4] leading to the SST model. By this approach, the near wall performance of the k-ω model can be used without the potential errors resulting from the free stream sensitivity of that model.

It is also important in turbulence modeling to have the numerical treatment of the equations done properly in regions close to walls. The formulation determines the accuracy of the wall shear stress and heat transfer predictions and has influence on the development of boundary layers. For the SST model, the new wall boundary treatment exploits the simple and robust near-wall formulation of the k-ω model. It switches automatically from a low-Reynolds number formulation to a wall function treatment, based on grid density. This automatic wall treatment avoids deterioration of the results typically seen if low-Reynolds models are applied on under-resolved grids.

The comparison with experimental data demonstrates the superior predictions obtained with SST model in comparison with models purely based on the ε equation. This is particular true for flow with adverse pressure gradients [6,7]. Therefore the SST is adopted as the standard model in the present paper.

3.4 Boundary condition

The boundary conditions employed are based on the experimental data. For the inlet region, total pressure, total temperature, flow angle, inlet turbulence intensity and eddy viscosity ratio are prescribed. For the outlet region, hub static pressure is prescribed. The blade, hub and shroud wall regions are assumed as adiabatic and the non-slip condition is applied. For the pitch wise boundaries, periodic boundary condition is applied.

In order to simulate film cooling flow, a source term modeling techniques is used. A row of cooling holes is actually modeled as a slot because the grid is not fine enough to capture the effect of each discrete film hole. The area of slot is equal to sum of a row of film cooling holes. Sources of mass, momentum, energy and the turbulence quantities are specified in each cell adjacent to a surface with film injection. Several inputs are required to specify the source terms. These include the coolant mass, coolant flow angle, the coolant supply temperature, the turbulent intensity and eddy viscosity ratio of the coolant. With this information, the mass flux, energy flux, turbulent kinetic energy flux, the turbulent frequency flux and the total momentum flux can be determined.

4 Results and discussions

The computation was carried out on a single processor SGI Origin O2000 at 250 MHz. The computation refers to the test condition is that specified by the designed exit Mach number at midspan and the coolant mass flow rate.

4.1 Blade pressure distributions

The blade pressure distributions in hub (5% span), midspan (50% span), tip (95% span) are shown in Fig.4. The sudden changes in pressure occur at the location of the film cooling holes. When using fully modeling of every single cooling holes, Hildebrandt et. al found pressure over- and undershoots originate in a quasi stagnation of the main flow immediately in front of the cooling jet. After a severe deceleration, the main flow is forced around the cooling jet resulting in a strong acceleration [2]. But the pressure peaks aroud the film cooling jets are less obvious in present source term modeling results.

4.2 Flow field details

The velocity vector around leading edge and trailing edge film cooling slot at midspan are show in Fig.5, 6, respectively. The coolant ejected from slot can be clearly identified.

The Mach number contour is show in Fig.7 for blade to blade at midspan.

Fig. 8 shows total pressure contour at 2/5th axial chord downstream of trailing edge. It is observed that the stator wakes near endwall are strongly affected by film cooling flow from hub and tip wall. In addition to the clear loss core, there is higher loss at hub than tip region.

The distribution of the secondary velocity vector in the same plane is presented in Fig.9. According to the vectors the secondary vortices are clearly identified.

4.3 Performance parameter distributions along span

The spanwise variations of total pressure recovery coefficient on outlet are shown in Fig.10. At 20% ~ 80% span, the agreement between computation and test data is well. There exist differences between computational and measured results near endwall. The cause may be inlet boundary layer has not been accounted in present computation.

Computational flow angle on outlet is in very well agreement with test data at almost the whole span (Fig. 11).

5 CONCLUDING REMARKS

The commercial CFD software CFX-TASCflow has been used to study flow fields in annular turbine stator with film cooling. Film cooling flow in the stator has been simulated by using source terms modeling method. The computational results are agreed well with the measured data. The source terms modeling method, which need a little time in modern computer, is very suitable in daily design work.

ACKNOWLEDGMENTS

The authors would like to thank the personnel from the turbine department and turbine laboratory of GTE who provided support throughout the design and testing phases of the program.

References

[1] Turner,M.G., Vitt,P.H., Topp,D.A. et. al , Multistage simulations of the GE90 turbine, NASA CR-1999-209311.

[2] Hildebrandt,Th, Ettrich, J.,Kluge, M., et. al., Unsteady 3D Navier-Stokes calculation of a film-cooled turbine stage, Part 2–cooling flow modeling via discrete cooling holes, 2003.

[3] CFX-TASCflow user documentation, Version 2.12, AEA Technology Engineering Software Ltd., Waterloo, Ontario, Canada, 2002.

[4] Menter F. R. Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal* Vol. 32, No. 8, pp.269-289, 1994.

[5] CFX-TurboGrid user documentation Version 1.6, AEA Technology Engineering Software Ltd., Waterloo, Ontario, Canada, 2001.

[6] Vieser, W., Esch, T. and Menter, F. R., Heat transfer predictions using advanced two-equation turbulence models, CFX-VAL10/0602, AEA Technology ,2002.

[7] Menter F.R. Kuntz M. and Langtry R., Ten years of industrial experience with SST turbulence model, Turbulence Heat and Mass Transfer 4 Begell House Inc., 2003.

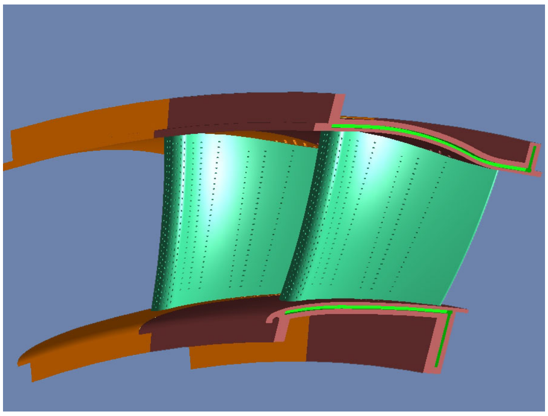

Fig.1 Compound lean stator with film cooling

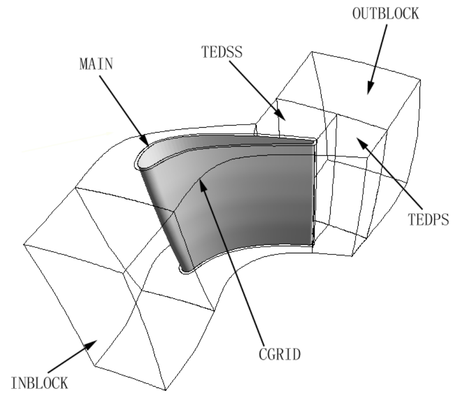

Fig.2 Grid block configuration

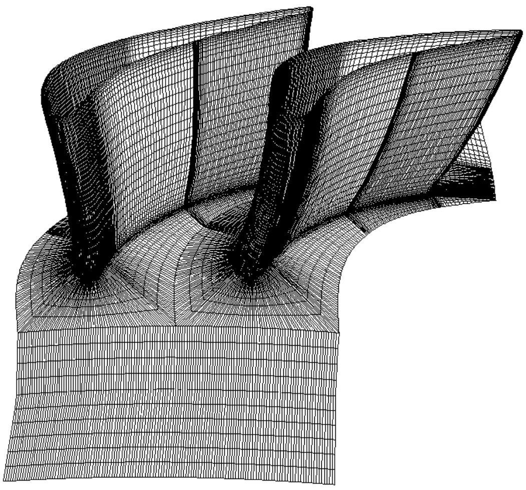

Fig.3 Computational grid

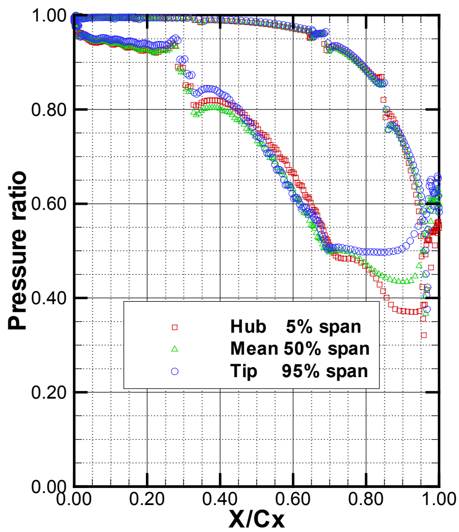

Fig. 4 Balde surface pressure distributions

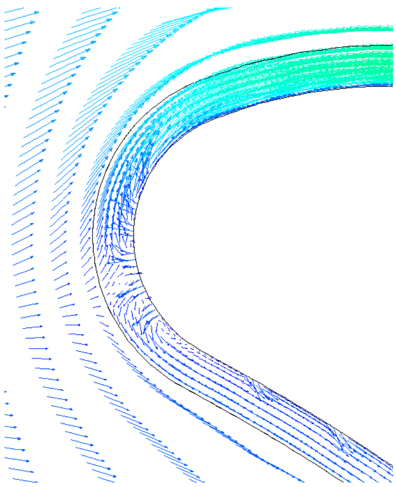

Fig. 5 Velocity vector near leading edge at midspan

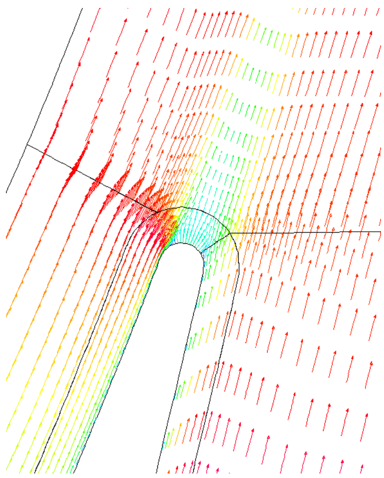

Fig. 6 Velocity vector near trailing edge at midspan

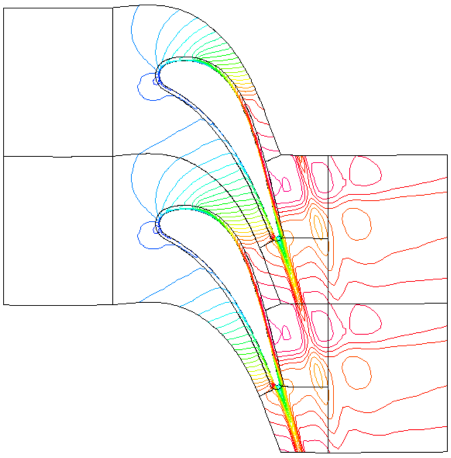

Fig.7 Mach number contour at midspan

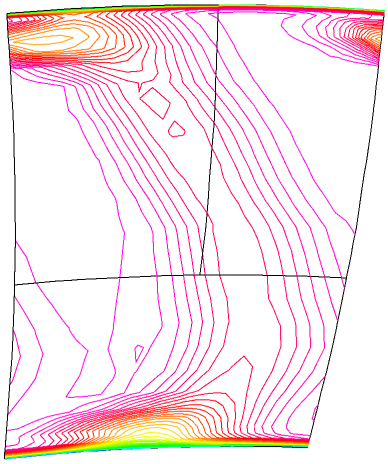

Fig.8 Total pressure contour on outlet

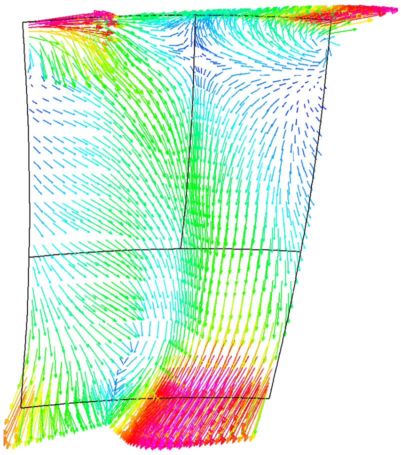

Fig.9 Secondary flow velocity vector on outlet

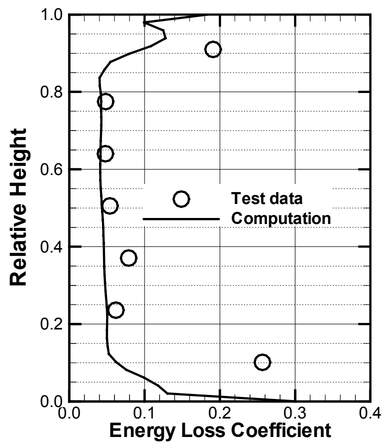

Fig.10 Radial distribution of the pitchwise mass-averaged energy loss coefficient

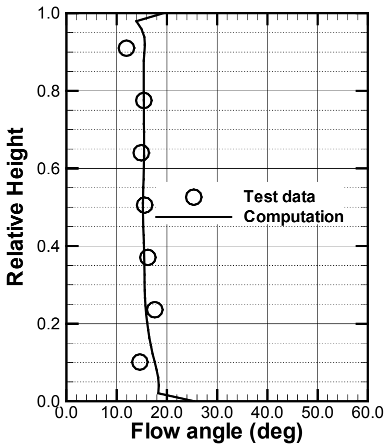

Fig.11 Radial distribution of the pitchwise mass-averaged flow angle