A Numerical Investigation of Air/Mist Cooling through a Conjugate, Rotating 3-D Gas Turbine Blade with Internal, External, and Tip Cooling

Ramy Abdelmaksoud and Ting Wang
Energy Conversion and Conservation Center
University of New Orleans
New Orleans, Louisiana, USA

ABSTRACT
This paper describes a numerical investigation to study the effect of injecting mist (tiny water droplets) into the cooling air used to cool down rotating gas turbine blades. In this study, the conjugate heat transfer method is employed which consists of the simulation of the air/mist fluid flow inside and outside the blade as well as the heat conduction through the blade body. The complete 3-D blade with internal cooling passages and external film cooling holes on the surface and blade tip is simulated in a rotating, periodic sector of the blade. The discrete phase model (DPM) is used to simulate and track the evaporation and movement of the tiny water droplets. The rotation effect of the turbine blade is included in the CFD simulation by using the moving reference frame method. The effects of different parameters such as the mist/air ratio (10-20%) and the mist droplets size (20-40µm) on mist cooling enhancement are investigated. The results show that the mist cooling enhancements are about 10% to 25% on the outer surface of the blade and reach 50% in some locations inside the blade on the internal cooling passages walls. Most of the liquid droplets completely evaporate inside the internal cooling passages; only a limited amount of mist is able to escape from the film cooling holes to enhance the blade outer surface and blade tip cooling. The effect of 10% mist on enhanced cooling is also converted to an equivalent of a 30% reduction in cooling air flow.

Keywords: film cooling, mist, gas turbine blade.

NOMENCLATURE
C  Vapor concentration, (kg/m³) or Specific heat, (J/kg-K)
D  Droplet size, (m)
d  Mass diffusivity, (m²/s)
h  Convective heat transfer coefficient, (W/m²-K)
h_lg  Latent heat, (J/kg)
Nu  Nusselt number, (hL/k)
P  Pressure, (Pa)
ReD  Reynolds number, (u D/ν)
Sh  Sherwood number, (kL/D)
k_e  Mass transfer coefficient, (m/s)
Sc  Schmidt number, (ν/d)
G_k  Generation of turbulence kinetic energy, (m²/s²)
We  Weber number, (-)

Greek Letters
μ  Dynamic viscosity, (Pa-s) or microns, (10⁻⁶ meter)
ν  Kinematic viscosity, (m²/s)
ρ  Density, (kg/m³)
τ  Shear stress, (N/m²)
ε  Dissipation rate, (m²/s³)
λ  Heat conductivity, (W/m.k)

Subscripts
i  Initial, Term number, Tensor index (1, 2, 3)
t  Turbulent

INTRODUCTION
According to the IEA’s (International Energy Agency) report, the current world population of 7.65 billion is expected to reach more than 9 billion in 2040, increasing world energy demand by about one-third [1]. This is forcing the power generation industry to rapidly improve its present technologies to keep up with the predicted energy demand increase. One of the most important power generation systems is the industrial gas turbine. The gas turbine manufacturers are competing to increase both the power output and the thermal efficiency of gas turbines. One technique to increase the output power and thermal efficiency of gas turbines is to increase the turbine inlet temperature (TIT) and compressor pressure ratio. The turbine inlet temperature has been designed to reach 1600 °C (2912 °F) or beyond in modern gas turbines. One of the most pressing challenges is to enable the gas turbine vanes and blades (i.e., airfoils) to withstand such high temperatures. Different techniques have been employed to overcome this challenge, including improving the alloy composition of the gas turbine airfoils, spraying thermal barrier coatings, growing single crystal blades, and improving the airfoil cooling techniques. The research described in this paper addresses the improvement of the cooling techniques.

Generally, the air film cooling scheme is used to protect the outer surface of the blade from the extreme heating from the hot flue gas and the turbulated internal passages are used to allow cooling air to further cool down the airfoils from inside. There is abundant publicly available information on gas turbine airfoil cooling. For example, Ito et al. [2] studied the local film cooling produced by a row of jets on a gas turbine blade and its effect on the blade curvature. Bogard and Thole [3] studied the variables that have either a significant or an
inertial effect on the film cooling produced by a row of discrete film cooling holes. Bunker [4] gives a comprehensive discussion of the shaped holes used for film cooling in turbine blades. Regarding the internal cooling in gas turbine blades, Han and Dutta [5] described various internal cooling techniques as well as the effect of using ribs and pin-fin cooling. Dunn et al. [6, 7] and Takeishi et al. [8] studied the heat transfer in rotating turbine rotors. In all these previous studies air was used as the cooling fluid.

Several experimental and numerical studies have been performed on air/mist and steam/mist cooling. Takagi and Ogasawara [9] performed one of the first experimental studies on air/mist flow. In their experiment the air was mixed with water droplets and then passed through a vertical rectangular tube to study the heat transfer in the tube. They found that the heat transfer coefficient increased in three cases: (a) when the droplet concentration increases; (b) when the air flow rate increases; (c) when the droplet size decreases. They also found that the heat transfer coefficient decreases as the wall temperature increases. Mori et al. [10] investigated experimentally the heat transfer of the air/mist cooling in a highly heated vertical tube. They found that the heat transfer along the tube axis could be divided into three main regions: the liquid film region, a dry-out region and a gas-phase forced convection region. Their results showed that when using mist, the heat transfer coefficient in the liquid film region is about 10 times higher than when mist is not used. One of the early experimental studies on applying air/mist in a turbine vane was conducted by Nirmalan et al. [11]. They found that by using the air/mist flow, the conventional only-air cooling flow can be reduced by 50% with maintaining the same overall cooling levels. However, they found that due to the strong water droplets spray, the leading edge region is overcooled. It seems like they conducted a spray cooling rather than a mist cooling.

An ongoing research effort has been conducted to investigate the feasibility of mixing the air with mist (micron-level water droplets) to enhance heat transfer. The potential of cooling enhancement due to using an air/mist mixture is based on the following factors: (a) the main flow temperature reduction is mainly due to droplet evaporation and the larger specific heats of water and water vapor as compared to that of air; (b) the increased turbulence mixing due to the droplets’ interaction with the main flow; (c) when a droplet impinges on a very hot wall, part of it will evaporate and expand in volume up to 600-1000 fold, causing a sudden propulsive momentum that will augment the turbulent mixing and enhance heat transfer; (d) the small period of time when the droplet touches a very hot wall will increase the heat transfer through direct conduction.

Guo et al. [12, 13] performed a comprehensive experimental study on the steam/mist cooling in a heated horizontal tube. They conducted a parametric study using a Phase Doppler Particle Analyzer (PDPA) to investigate the effect of droplet size, the wall heat flux, the mist mass flow rate ratio and the flow Reynolds number on heat transfer. Their results showed that the addition of mist to the main flow significantly improved heat transfer. It is found that by adding 5% (wt.) mist, an average heat transfer enhancement of 100% is achieved. Guo et al. [14] used a 180° tube bend as the test section in the same experimental setup as in [12] and performed a parametric study. It was found that the overall heat transfer cooling enhancement ranged from 40 to 300% with some local maximum enhancement of 500%. Zhao and Wang [15, 16] experimentally studied the air/mist film cooling on a flat plate with circular round holes. They found that for a blowing ratio of 0.6, the net cooling effectiveness enhancement reached a maximum local value of 190% and an overall value of 128% at the centerline. They also found that the wall surface temperature becomes more uniform, which is important in the reduction of wall thermal stresses.

Dhanasekaran and Wang [17] studied numerically the effect of air/mist cooling on a rotating blade with a single pair of film cooling holes. They found that the heat transfer coefficient increases with the blowing ratio, and that the cooling enhancement at the wall has an average value of 35%. Recently, Ragab and Wang [18 and 19] extended the film cooling experiment from circular round holes to fan-shaped holes. They discovered that by using the fan-shaped hole geometry with a blowing ratio of 0.66, the mist cooling enhancement improved remarkably in all the test section setups due to the lateral diffusion. Also, they found that the cooling in the region between two fan-shaped holes is much better than in the region between two circular holes since more droplets were thrown in the lateral direction by the divergent angle of the fan-shaped holes, leading to more uniformed lateral cooling than in the case of two circular holes.

Bian et al. [20] recently studied the effect of using mist/air flow on the pressure drop inside a turbine blade. They found that the pressure drops on the ribbed concave/convex surfaces are higher than that on the smooth surface in a gas turbine blade.

One of the practical concerns in the employment of mist cooling is associated with the question of "how to transport the mist to the turbine vanes and blades?" To answer this question, Ragab and Wang [21] first conducted a computational simulation for transporting mist to the vanes from the turbine outer casing. The results showed that under real Frame 7FA operating conditions, 50% of the mist can survive with an average droplet diameter of 10 – 20 µm if mist with 10% air mass ratio and 20-30µm in initial diameter is injected. They then conducted a further simulation in which high-pressure liquid was channeled through the turbine internal structure and then atomized near the preswirler, allowing mist to be transported to the root of the turbine blades through the rotational rotor disk cavity [22 and 23], as shown in Fig. 1. Their studies showed that using droplets with initial diameter ranging from 50-90 µm and a mist/air mass ratio greater than 15% is suitable in achieving droplet sizes in the range of 20-40 µm at the root (hub) of the turbine blades.

Continuing the previous studies, the present study assumes that the mist has been transported to the root of the blade and the computational domain starts at the root. The previous mist cooling studies have all been conducted in one part of a blade only for a simplified film cooling condition, or only in a simplified internal cooling passage. The objective of this study is to investigate the mist transporting phenomena and its cooling effectiveness in a complete, full turbine blade, including all the internal cooling passages and blade surface film cooling holes as well as trailing edge and tip squealer cooling. The blade is rotating with the external flow being computed via the periodic flow gas passage between two blades. Due to the large computational domain which includes both the external and internal flow calculation in complex.
conjugate air/solid geometry with many tiny film cooling holes, the intricacy of the turbulators in the internal passages is not included in the simulation.

Figure 1 Suggested schematic of the mist cooling injections and paths in a gas turbine [19]

NUMERICAL SIMULATION

Geometry
The computational domain presented in this paper consists of two subdomains; a fluid domain and a solid domain. The hot air, cooling air and the mist move through the fluid domain. The solid domain represents the gas turbine blade solid body. The computational domain is a sector of the rotor of the first turbine stage in a typical industrial gas turbine. Since there are 70 blades in the first turbine stage, the computational domain is chosen as an axisymmetric sector of 360/70 = 5.143° in the circumferential direction. To avoid using proprietary geometry and dimensions of a real modern gas turbine blade, a generic blade configuration similar to the one presented by Han and Dutta [5] is used in this study. No 3-D twist or optimum design is considered to make this blade more aerodynamically efficient or more cooling effective.

The blade has a height of 42 cm (16.54 inches) and an axial chord of 20 cm (7.78 inches). The distance from the hot external gases inlet to the blade leading edge is 75% of the chord length. The blade has 7 rows of film cooling holes: 3 on the pressure side, 3 on the suction side and one at the trailing edge as shown in Fig. 2a, 2b, and 2e, respectively. The first row near the leading edge has 23 film cooling holes, the second row has 48 film cooling holes, and the third row has 23 film cooling holes. Both the pressure side and the suction side have the same number of film cooling holes per row. The trailing edge row has 23 holes. Figure 2d shows the blade tip, squealer wall, and its 6 tip cooling holes. All the film cooling holes have a projected diameter of 2.5 mm except the ones at the trailing edge which are 2 mm in diameter. The tip squealer is 1 cm (0.39 inch) high. There is a clearance of 0.5 cm between the blade tip and the shroud. The blade has three separate internal cooling passages with openings at the bottom to allow cooling air to enter as shown in Fig. 2c. A cut-off view for the blade in Fig. 2e shows the internal cooling passages. When the cooling air enters the first internal cooling passage near the leading edge (see the solid blue arrows), it will flow through this first passage radially outward (upward direction in Fig. 2e).

Figure 2 (a) Pressure side, (b) Suction side, (c) Bottom view from the hub showing the cooling air inlets, (d) Top view of tip cooling holes and squealer, and (e) Cut-off view for the blade showing the internal cooling air passages
During its flow to the blade tip, part of the cooling air is gradually diverted horizontally through 23 holes to impinge on the inside wall of the blade leading edge, and finally the cooling air exits through two rows of film cooling holes on either side of the blade. The second internal cooling passage consists of a three-pass serpentine arrangement of cooling channels between the leading and trailing edges. The dashed blue arrows show the path of the cooling air entering from near the trailing edge, going through three passages toward the leading edge, and finally exiting the blade through the film cooling holes on both sides of the blades. The third internal cooling passage is the one closest to the trailing edge. The dotted blue arrows show the path of the cooling air. The cooling air enters and moves radially outward with part of the flow gradually depleted by exiting through film cooling holes on both sides of the blade. In addition, in each internal cooling passage, the cooling air also exits through the tip holes with two tip holes through the first passage, three tip holes through the second passage, and one tip hole through the third passage.

Mesh

The conjugate computational domain which consists of the blade’s solid wall and several fluid domains inside and outside the blade of a very complex design. Hence, the computational domain is constructed using unstructured tetrahedral cells. Higher mesh density is employed near the walls to give a value of $30 < Y^+ < 100$ for using the standard wall functions for the selected turbulence model which will be discussed later in more detail. Higher mesh cell density is also used in and surrounding the film cooling holes. Four meshes, consisting of 1.6, 2.4, 3.4 and 4.4 million elements respectively, are chosen to conduct the mesh sensitivity analysis.

![Figure 4 The 3.4 million unstructured tetrahedral mesh](image)

**Figure 4** The 3.4 million unstructured tetrahedral mesh

### Mathematical model

The mathematical model consists of the governing equations given below and the associated boundary conditions. In the numerical study presented in this paper, there are two fluids (air and water) and one solid. Since the mist is very tiny water droplets and its volume fraction is less than 0.1% of the air volume fraction, the trajectories of these tiny water droplets are calculated by using the discrete phase model (DPM). The continuous phase is the air and the dispersed phase is the droplets. The two phases are coupled through source terms in the governing equations. The blade rotation effect (3600 rpm) is modeled using a moving reference frame. The solver software used in this numerical study is the commercial CFD software ANSYS FLUENT 16.0 from Ansys, Inc. The pressure based steady-state compressible flow solver is used. The SIMPLE algorithm is applied for the pressure-velocity coupling. The PRESTO scheme is selected for the pressure interpolation. Second order upwind schemes are selected for the spatial discretization for all the other convective terms.

**The Governing Equations**

The governing equations of mass, momentum, energy and species which are based on time averaged steady state conditions are:

$$\frac{\partial}{\partial x_i}(\rho u_i) = S_m$$

$$\frac{\partial}{\partial x_1}(\rho u_i u_j) = \rho g_j + \frac{\partial \rho}{\partial x_1} + \frac{\partial}{\partial x_i}(\tau_{ij} - \rho \overline{u_i u_j}) + F_j$$

$$\frac{\partial}{\partial x_1}(\rho c_p u_i T) = \frac{\partial}{\partial x_i}(\lambda \frac{\partial T}{\partial x_i} - \rho c_p \overline{u_i' u_i'}) + \mu \Phi + S_h$$

$$\frac{\partial}{\partial x_1}(\rho u_i C_i) = \frac{\partial}{\partial x_i}(\rho d_i \frac{\partial C_i}{\partial x_i} - \rho \overline{u_i' C_i'}) + S_j$$

The source terms ($S_m$, $F_j$ and $S_h$) are added to the mass, momentum and energy conservation equations respectively to couple the discrete phase with the continuous phase and to show the impact of the dispersed phase. The symbol $\lambda$.
represents the heat conductivity, the term $\mu\Phi\xi$ is the viscous dissipation, $C_i$ is the mass fraction of the species (j) in the mixture, and $S_i$ is the source term for this species. $d_i$ is the diffusion coefficient. Two species (air and water vapor) are used to simulate the mixture. The terms $\rho \overline{u_i u_j}$, $\rho C_i T$ and $\rho \overline{u_i u_j}$ in the equations above represent the Reynolds stresses, turbulent heat fluxes, and turbulent concentration (mass) fluxes respectively, which should be modeled by using a suitable turbulence model. Dhanasekaran and Wang [24, 25] made a comparison between different turbulence models to validate the mist cooling experimental data. They concluded that both the Reynolds Stress Model (RSM) and the standard k-ε turbulence model predict data which is more consistent with the experimental data than other turbulence models. Since the standard k-ε turbulence model is more robust and needs significantly less computational time than RSM, it was selected with the standard wall functions to model the turbulence effect in this numerical study. The k-ε turbulence model is a two-equation turbulence model. The turbulent kinetic energy (k) and the dissipation rate ($\varepsilon$) equations are,

$$\frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_i}\left[(\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_i}\right] + \rho \varepsilon + G_k - \rho \frac{\partial}{\partial x_i} \left(\frac{\partial \varepsilon}{\partial x_i}\right)$$  (5)

$$\frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}\left[(\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon} \frac{\varepsilon}{k} \frac{\partial k}{\partial x_i} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$  (6)

The term $G_k$ denotes the generation of turbulence kinetic energy due to the mean velocity gradients. The turbulent viscosity, $\mu_t$, is calculated from the equation

$$\mu_t = \rho C_t \frac{k^2}{\varepsilon}$$  (7)

and the effective heat conductivity ($\lambda_{eff}$) and the effective diffusion coefficient ($D_{eff}$) are calculated using following two equations, respectively.

$$\lambda_{eff} = \lambda + \frac{c_p \mu_t}{\Pr_t}$$  (8)

$$D_{eff} = d + \frac{\mu_t}{Sc_t}$$  (9)

The values chosen for the constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{\mu}$, $\sigma_k$ and $\sigma_\varepsilon$ are: $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_{\mu} = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$ [26]. The turbulence Prandtl number, $Pr_t$, is set to 0.85, and the turbulence Schmidt number, $Sc_t$, is set to 0.7.

**Discrete Phase Model (DPM)**

The DPM is a multiphase model where the dispersed phase is tracked in a Lagrangian reference frame. There are several factors that affect the motion of a water droplet in the continuous phase such as the gravity, drag, Brownian force, thermophoretic force, Saffman’s lift force and the virtual mass force. All the aforementioned forces are incorporated in this study. All these forces are used in the droplet equation of motion which applies Newton’s 2nd Law. More details about these forces and their relations can be found in [27-29]. The heat and mass transfer between the droplet and the continuous phase are controlled by another set of equations. The energy equation for each droplet can be written as:

$$m_p c_p \frac{dT}{dt} = \pi D^2 h(T_{\infty} - T) + \frac{dm_p}{dt} h_{fg}$$  (10)

where $h_{fg}$ is the latent heat. The Nusselt number empirical correlation given in Eq. 11 can be used to obtain the convective heat transfer coefficient [30, 31].

$$Nu = \frac{h D}{k} = 2 + 0.6 Re^2 Pr^{rac{1}{3}}$$  (11)

The droplets will experience two heat transfer modes such as evaporation and boiling. The evaporation depends on the water vapor partial pressure and the concentration difference between the droplet surface and the air is shown in Eq. 12.

$$- \frac{dm_p}{dt} = \pi D^2 k_s (C_s - C_{\infty})$$  (12)

where $\frac{dm_p}{dt}$ is the evaporated mass rate of change, $C_s$ is the vapor concentration at the droplet surface and $C_{\infty}$ is the vapor concentration of the bulk flow. The value of $C_{\infty}$ can be calculated by solving the transport equation in the computational cell. $k_s$ is the mass transfer coefficient and its value can be obtained from the Sherwood number correlation in Eq. 13.

$$Sh = \frac{k_s D}{d} = 2 + 0.6 Re^2 Sc^3$$  (13)

where Sc is the Schmidt number which is defined as ($\nu/d$) where d is the mass diffusion coefficient of the water vapor in the bulk flow.

On the other hand, the boiling phenomenon depends on the boiling temperature of the liquid, subject to the pressure of the air flow. When the droplet temperature reaches the boiling point, Eq. 14 will be used to obtain the evaporated mass change rate.

$$- \frac{dm_p}{dt} = \pi D^2 \left( \frac{\lambda}{\rho} \right) (2.0 + 0.64 Re^{0.5} + \frac{\ln(1+ (\frac{c_p \alpha T_{\infty} - T)}{c_p})}{c_p})$$  (14)

where $\lambda$ is the gas/air heat conductivity and $c_p$ is the specific heat of the bulk flow. The *stochastic particle tracking* method is used to simulate the instantaneous turbulent effect on droplet dispersion by predicting the turbulent dispersion of particles by integrating the trajectory equations for individual particles, using the instantaneous fluid velocity.

**The Moving Reference Frame Method**

The moving reference frame method is used to model the effects of rotation by adding additional terms to the conservation of momentum equation. The use of steady-state time-averaged governing equations is justified because the effect of rotation is just to add additional terms related to the rotational speed to the momentum equation without actually rotating the mesh itself. The two acceleration terms are Coriolis acceleration $-\left(2\vec{\omega} \times \vec{v}_r\right)$ and centripetal acceleration $(\vec{\omega} \times \vec{\omega} \times \vec{r})$. $\vec{\omega}$ is the angular velocity and $\vec{v}_r$ is the relative velocity i.e. the velocity viewed from the moving reference frame.
The Conjugate Heat Transfer Technique

This technique is used to solve a computational domain which consists of both solid and fluid subdomains. In the solid region, the conduction heat transfer relation is used to calculate the heat transfer. The two domains are coupled to permit heat transfer from the fluid to the solid or from the solid to the fluid.

Boundary Conditions

The computational domain is shown in Fig. 5. It consists of a 1/70th axisymmetric sector of a hot flue gas passage, the cooling air and mist inlets, the convective hub, and shroud. A rotational periodic boundary condition is employed.

![Figure 5 The boundaries of the computational domain geometry](image)

Hot Gas Flow—The hot flue gases are modeled as hot dry compressible air with a total mass flow rate of 440 kg/s, about 6.3 kg/s (440÷70) passing through this computational sector. The hot gas inlet temperature is 1645K (2501°F) and the pressure is 13 bar (188.5 psia). The turbulence intensity is assumed to be 5% and the turbulence length scale, based on the hydraulic diameter of the inlet is about 17.337 cm. The air properties are a function of the temperature. The specific heat is calculated by using a polynomial function and the other properties are calculated by using the linear piecewise function where the properties values are calculated by interpolating a set of values range including the minimum and maximum values in the domain.

Cooling Air Flow—The blade cooling air rate is given as 2% of the total gas turbine mass flow rate. Each blade receives a cooling air mass flow rate of 0.126 kg/s. This amount is divided between the three inlets at the bottom of the blade in proportion to the size of the inlet's cross-sectional area: inlet 1 (60%), inlet 2 (30%) and inlet 3 (10%). The cooling air is given a temperature of 700K (800°F) and a pressure of 15 bars (217.5 psia) at the blade bottom inlet.

Mist—Two mist mass flow rates, (10% and 20%) of the cooling air mass flow rate (i.e., 0.2% and 0.4% of the total flue gas mass flow rate), and two droplet sizes (20 and 40 μm) are studied. Based on the study of Wang and Ragab [23], the mist droplet temperature is assigned as 420K (296°F)−this is still subcooled liquid under 15 bar. In the case of a dry wall, the droplets have three major regimes, including reflect, break-up, and trap. Watchers et al. [32] found that the regimes depend on the incoming Weber number of the droplet where the Weber number is the ratio of kinetic energy of the droplet to its surface tension energy (\( We = \rho v^2 \sigma \)) where \( \sigma \) is the surface tension. Since the blade wall is very hot and by taking into consideration the Weber number value, the “reflecting” wall boundary condition for the discrete phase is selected.

The Shroud and Hub Endwall—Both the shroud and the hub endwall are as insulated walls.

Solution Procedure

Five cases are studied and their corresponding conditions are listed in Table 1.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mist mass flow rate (%)</td>
<td>Air only</td>
<td>10%</td>
<td>10%</td>
<td>20%</td>
<td>20%</td>
</tr>
<tr>
<td>Droplet size (µm)</td>
<td>—</td>
<td>20</td>
<td>40</td>
<td>20</td>
<td>40</td>
</tr>
</tbody>
</table>

As a first step, the computation is performed for the continuous phase until a converged solution is obtained. Then the dispersed phase tracking is calculated by alternating the iterations between the continuous and dispersed phases until a converged solution is obtained. Convergence is declared when the mass residuals are less than \( 10^{-3} \), the energy residuals are less than \( 10^{-6} \), and momentum and turbulence kinetic energy residuals are less than \( 10^{-4} \). These residuals are the summation of the imbalance for each cell, scaled by representative flow rate. An additional procedure is adopted to ensure that the solution is converged by crosschecking the local convergence via monitoring the values of local entities such as velocity or static temperature at multiple selected points of interest in the domain.

RESULTS AND DISCUSSION

CFD Scheme and Droplet Model Validation

Because of the unavailability of experimental studies on air/mist film cooling in a rotating gas turbine blade, the validation of the computational method and mist model is performed with one of the experimental studies on steam/mist film cooling in a horizontal tube performed by Guo et al. [12, 13]. Two cases are selected for validation: one case has only the steam flow and the other case has the mist. The following conditions are specified: the steam is saturated at 1.6 bar, the Reynolds number is 10,000, the mist/steam mass flow ratio is
5.1%, the water droplet size is uniform at 6 µm, and the wall is heated uniformly with a heat flux of 8000 W/m². Figure 6 shows a comparison between the CFD results and the experimentally measured wall temperatures where the x-axis is the normalized test section length (length of the test section divided by its diameter) and the y-axis is the temperature values in K. The differences are within 5-10%, which is deemed acceptable for the current study.

Figure 6 Validation of the CFD model against experimental data from Guo et al. [12 and 13] in a horizontal tube.

Rotating Blade CFD Results for the Five Cases

Pressure and suction sides

Figure 7 shows the temperature contours for the pressure and suction sides of the blade for each of the five cases. The temperature contours of the pressure side for the five cases are shown in Fig. 7a to 7e, which correspond to Cases 1-5 (air-only, mist=10% & 20 µm, mist= 10% & 40 µm, mist= 20% & 20 µm and mist =20% &40 µm) respectively. The temperature contours of the suction side for the five cases are shown in Fig. 7f to 7j, which correspond to Cases 1-5 respectively. Note that the red arrows represent the direction of the hot flue gas. There are four graphs surrounding the temperature contour plots; two for the pressure side and two for the suction side. Each one shows a comparison between the temperature values for the five cases at a certain vertical line (the x-axis of the graph represents the height of the blade in cm where H= 0 is the bottom of the blade and H=42 cm is at the blade tip) and the y-axis represents the temperature value in K). The particular graph border line has the same style as the vertical line on the contour plots where the temperature values are obtained. For example, Graph I has a solid line border and it shows the temperature profile at the vertical solid line near the trailing edge of the pressure sides of the five cases (Figs. 7a to 7e).

The temperature contours of the pressure side for the five cases show the cooling enhancement when mist is injected. In Fig. 7b (Case 2: mist=10% & 20 µm), we notice that the pressure side is cooled near the leading edge, in the middle region and near the trailing edge compared to Case 1 (air-only) in Fig. 7a. For Case 3 (mist=10% & 40 µm) in Fig. 7c, changing the mist droplet size from 20 to 40 µm with the same mist mass flow rate ratio does not cause a noticeable overall cooling enhancement but a slight local enhancement is observed in the upper film cooling holes near the leading edge. In Fig. 7d (Case 4: mist=20% & 20 µm) where the mist mass flow rate ratio is doubled and the droplet size is 20 µm again, we notice that the pressure side has better cooling near the leading edge, in the middle region and near the trailing edge when compared to Case 2 (mist=10% & 20 µm). In Fig. 7e (Case 5: mist=20% & 40 µm), the droplet size is doubled from 20 to 40 µm at the same mist/air ratio, there is no noticeable overall enhancement on the pressure side except a slight enhancement in the upper film cooling holes near the leading and trailing edges.

From the above analysis, we conclude that the effect of mist/air ratio on cooling enhancement is more dominant than the effect of increased droplet size. To allow more quantitative comparisons, two vertical lines are marked on the temperature contours of the pressure sides—one near the trailing edge and another near the leading edge. The vertical temperature distributions along these vertical lines in the five cases are presented in Graphs I and II in Fig. 7. It is shown that a cooling enhancement of 40 to 60 K with 10% mist/air ratio and a cooling enhancement of 60 to 100 K with 20% mist/air ratio are achieved along most of the blade height except the squealer region at the top of the blade, where only a very minimal amount of mist can reach in this study (to be shown later in Fig. 9 below). The effect of droplet size on cooling enhancement is small but noticeable, as shown by overlapped lines with a temperature difference around 5-8 K. This is because larger droplets (40µm) can reach the tip.

The results of the suction side are similar to those on the pressure side, as shown from Fig. 7f to 7j. Adding 10% mist in Case 2, causes a cooling enhancement in the middle region of the blade and the upper film cooling holes of the leading edge (Fig. 7g) as compared to the air-only Case 1 in Fig. 7f. Adding more mist up to 20% in Case 4 shows (in Fig. 7i) a better enhancement as compared to Case 2. Increasing the droplet sizes from 20µm (Case 3) to 40 µm (Case 5) does not show a noticeable cooling enhancement compared to Cases 2 and 4 respectively except for a slight cooling enhancement in the tip cap region.

Two vertical lines are marked on the temperature contours of the pressure sides—one near the trailing edge and another near the leading edge—to provide a more quantitative comparison of temperature profiles along those vertical lines in Graphs III and IV. The 10% mist cases (Cases 2 and 3) achieve a cooling enhancement of 20 to 40 K in wall temperature reduction, while the 20% mist cases (Cases 4 and 5) a larger cooling enhancement of 40 to 75 K in wall temperature reduction in most of the blade height except the squealer region at the top of the blade. Again, the effect of droplet size on cooling enhancement is small but noticeable, which is shown by overlapped lines with a temperature difference around 5-8 K because larger droplets (40µm) can reach the tip.
Figure 7 Comparison of temperature contours of five cases on the pressure and suction sides. The temperature distributions from the hub to the tip of selected locations are shown in four satellite graphs I, II, III, IV. The border line type of each graph corresponds to the location where the vertical line of the identical type is placed on the contour plots.
Tip cap and squealer area
The temperature contours of the tip cap and the squealer top surface are shown in Fig. 8. In summary, addition of 10% mist reduces the wall temperature around 50-90 K in the tip cap area (comparing Fig. 8b with Fig. 8a). Adding more mist up to 20%, a further reduction of 50-80 K can be achieved (comparing Fig. 8d with Fig. 8b). The effect of droplet size on cooling enhancement is noticeable, about 5-15 K, by comparing Fig. 8b with 8c for 10% mist or about 10-25 K by comparing Fig. 8d with 8e for 20% mist. This is because the larger droplets injected at 40µm can reach the tip cooling holes as evidenced in Fig. 9, which is not attainable by the 20 µm droplets.

The Droplet Trajectories
Figure 9 shows the water droplet tracks for the four cases where the mist is injected, colored by the droplet size. In summary, we see that increasing the mist/air ratio from 10% to 20%, the mist lasts longer (comparing Fig. 9a with 9c for 20µm droplets and Fig. 9b with 9d for 40µm droplets). But the difference of the droplet tracks due to increased mist ratio is not as significant as due to the increased droplet size from 20µm to 40 µm. This explains why tip cap cooling is better in the cases with 40µm droplets, as shown in Figs. 8c and 8e and Figs. 9b and 9d.

The Pressure Drop in The Two-Phase Region
The mist concentration mainly existed in the internal passages which affected the pressure losses due to the drag caused by the droplets. The pressure drop of the single-phase flow (cooling air only) in the internal passages is smaller than that of the two-phase (mist/air) flow in the four cases that had mist. The pressure drop increases with increased mist ratio. However, changing the mist droplet size from 20 to 40 µm did not affect the pressure drop significantly. The pressure drops in Cases 2 and 5 were about 7% and 12% higher compared to Case 1.

To further analyze the effect of other parameters on the simulated cooling results, including the rotational effect, non-uniform droplet distribution, and blowing ratios, Case 3 (10% mist with 40µm) was selected for these parametric studies.

Uniform Vs. Non-Uniform Droplet Distributions
All the cases discussed above were simulated with a uniform droplet size for the controlled studies. We now examine the effect of non-uniform distributed droplet size employing a Rosin-Rammler particle distribution. In this
distribution method, a range of droplet sizes is chosen (for this study, the range is from 30 to 50 µm) divided into an adequate number of discrete intervals which are represented by a mean diameter for which trajectory calculations are performed. More details about this method can be found in [33]. Case 6 is assigned to this non-uniform droplet study with 10% mist and a Rosin-Rammler droplet distribution with the mean droplet diameter at 40µm. Figure 10 shows the temperature profiles at the two vertical lines locations shown in Fig. 7. It is found that there is negligible temperature difference (about 3-5 K) between Case 3 and Case 6.

Rotating Vs. Non-Rotating Effect

The same selected Case 3 mentioned above is compared to the non-rotating version of itself to observe the effect of rotation on the heat transfer. Generally, the rotational motion exerts a noticeable effect on the flow field and heat transfer due to the Coriolis and centrifugal forces. These forces are expected to have a larger effect on the droplet trajectories.

Figure 11 Temperature profiles with and without the rotation effect along a vertical line near the leading edge on the (a) pressure side (PS) and (b) suction side (SS), respectively

Figure 11 shows the temperature profiles along the vertical line near the leading edge on the pressure and suction sides respectively of Case 1 (air-only) and Case 3 (10% mist with 40 µm); both are with and without rotation. In Fig. 11b, on the suction side, we see that when using the mist without rotation, the cooling enhancement is significantly better than that with rotation. This is believed to be caused by the effect of Coriolis force, \(-2\omega \times \vec{v}\), during rotation, which pushes the droplets towards the pressure side of the blade with the internal cooling flow moving from the hub toward the tip, leading to a lack of cooling on the suction sides of the cooling passages.

Figure 12 shows the droplets piling up on the pressure side in the cavities due to the rotation effect. Although the Coriolis force will push the flow and droplets toward the suction side when the internal cooling flow moves from the tip toward the hub in the second leg of the internal passage, the effect of Coriolis force on the droplets would be small because barely any mist survives there, as can be seen in Figures 9 and 12. In other words, without rotation, both the air and the mist can approach all sides of the internal passage walls more uniformly, achieving better overall heat transfer.

The External Flow Field Near the Film Cooling Holes

Figure 13 shows three temperature planes with the velocity streamlines on them. It is shown that the temperature in the near-wall area is lower than that of the main external flow due to the cooling air coming out from the film cooling holes which can be noticed through the streamlines coming out from the holes. The temperature of the air coming out from the film cooling holes is around 1500 K while the main external flow has a temperature of 1650 K. The velocity streamlines show that the flow is flowing smoothly along the suction side of the blade. The flow is relatively slower in the near-wall region.
Blowing Ratio of Film Cooling

One of the important parameters for studying turbine blade film cooling is related to the blowing ratio, defined as:

$$\text{BR} = \frac{\rho \text{ } \langle v \rangle_{\text{hole}}}{\rho \text{ } \langle v \rangle_{\text{main stream}}}$$  \hspace{1cm} (15)

where the velocities and densities are the local values. The representative blowing ratio values at the mid-span in the first row near the leading edge on the pressure side is 1.513 and those values are 0.65 and 0.5 in the second and third rows, respectively. These values are in line with the operation condition experience of real blades.

Temperature and Pressure Distributions at Mid-Span

The temperature and pressure distributions at mid-span along the pressure and suction sides are shown in Fig. 14. The mid-span curve on the pressure and suction sides can be divided into three main regions; (from the leading edge to 25% of the axial chord, 25% to 75% of the axial chord and 75% of the axial chord to the trailing edge), respectively. The pressure side is cooler than the suction side in the first region (from LE to 25% of the axial chord). The suction side is cooler in the second region. The pressure side is cooler again in the third region. The mid-span curve where the temperature and pressure distributions are obtained, passes through some film cooling holes, hence the curves are not smooth due to the cooling air blowing through the surface.
Equivalent Cooling Air Reduction

The cooling air used to cool down the turbine airfoils is extracted from the compressor which results in a reduction in net output power and efficiency. By using air/mist cooling, the cooling air in the conventional cooling method (air-only) can be reduced which would help to increase the net efficiency. The equivalent cooling air reduction due to enhanced mist cooling is estimated by performing several air-only cases by increasing the cooling air flow until the similar cooling enhancement is achieved as Case 3 (10% air/mist ratio & 40µm). The results of two representative, additional air-only cases (Cases 7 and 8) were performed. Case 7 has 2.2% cooling air (10% more than Case 1 which has 2% cooling air) and Case 8 has 2.4% cooling air (20% more than Case 1). The temperature profiles used for comparison, were obtained at two vertical lines locations near LE (same as shown in Fig. 7). Both Cases did not reach the same cooling enhancement as Case 3. Due to the extensive computational time needed to perform additional cases, an extrapolation formula was applied to predict the extra air percentage needed by extrapolation of the results of Cases 7, 8 and 3. It was found that approximately 30% extra air is necessary to give a similar cooling enhancement as that in Case 3.

CONCLUSIONS

A computational study of the effect of injecting water droplets (mist) into a rotational blade under real industrial gas turbine operating conditions was performed. The simulated blade includes convective cooling through three internal passages, impinging cooling inside the leading edge, film cooling over the surfaces and tip cooling in the squealer. The turbine blade wall is simulated with conjugate heat transfer. Two mist/air ratios, 10% and 20%, and two droplet sizes, 20µm and 40µm, were simulated. The conclusions are:

- By using a mist mass flow rate ratio of 10% with a droplet size of 20 µm, the blade surface temperature of the pressure side decreased 40-60 K showing noticeable cooling enhancement in the region along the film cooling holes near the leading edge, in the middle region and the upper film cooling holes near the trailing edge as compared to the air-only case. On the suction side, the blade surface temperature decreased by 20-40 K in the middle region and around the upper film cooling holes at the leading edge and trailing edge.
- Adding more mist up to 20%, more cooling enhancement is achieved as evidenced by a wall temperature reduction of about 60-100 K in some regions on the pressure side and 40-75 K on the suction side compared to the air-only case.
- Increasing droplet size from 20µm to 40µm doesn't noticeably affect the cooling in most of the blade, but it does enhance the cooling in the tip cap area by around 4-8 K because the larger droplets (40µm) last longer and can reach the tip cap area, whereas the smaller droplets (20µm) can't.
- Using non-uniform droplet diameters with a Rosin-Rammler distribution between 30µm and 50 µm does not show a noticeable change in the results. This insensitivity to the non-uniform droplet distribution may be due to the selected narrow diameter range (30-50 µm). A comparison between non-rotating and rotating condition indicates that rotation of the blade drives the droplets towards the pressure side, reducing the opportunity for droplets to provide more uniform cooling to all the sides, and thus reducing the cooling enhancement. This may imply that mist cooling could be more effective implemented in the turbine vanes.
- Using a 10% mist/air ratio can reduce the cooling air extracted from the compressor by roughly 30% resulting in an increase in net efficiency.

In summary, if 40-60 K reduction of turbine wall temperature in addition to usage of the current air cooled turbine blades is significant to the designers, then application of a 10% mist/air ratio with 20µm droplet size could do the job. If further reduction of wall temperature (60-100 K) is needed, adding more mist up to 20% of the cooling air with 40µm droplets could achieve the goal.

ACKNOWLEDGEMENT

The authors want to thank the support from the Louisiana Board of Regents.

REFERENCES


