Numerical evaluation of the motion of fogging flow along the inlet duct of gas turbine engines

D G Kofar-bai

Department of Mechanical Engineering, Hassan Usman Katsina Polytechnic, Katsina State, Nigeria

Abstract. Computational fluid dynamic analysis of inlet duct fogging system was carried out to understand the characteristics flow of nozzles spray along the air inlet duct channel of Alstom GT13E2 gas turbine, the numerical work is validated using wind tunnel experimental data of previous study. Flow resistance on the nozzle were studied at specified reference locations. The results obtained have shown that, fogging spray possesses significant effects on mass flow rate due transfer of heat of evaporation caused by air to the liquid droplets in the inlet duct, which greatly influence the performance of the compressor and turbine. An adequately space between the nozzles array and compressor bell mouth produces mass flow rate evaporability up to 87.63% and temperature difference drop by roughly 6˚C. The pressure distribution becomes uniformly and squarely distributed when positioned at least 2.0 m away from the nozzle array. The current research offers an inclusive coverage on the droplets evaporative cooling effects, air flow resistance through pressure drop and their effects on the compressor performances, it is our desire that the work will be of added benefit to the gas turbine engine designers.

1. Introduction

The heat and mass transfer are of tremendous importance in many evaporation processes, to visibly moved the restrictions, a study of fluid flow and its characteristics heat transmission through small field[1]. Intense air motion on low pressure compressor blade of an aircraft boost turbine power and decreases the weight of an engine[2]. An experiment to understand the nature of flow field and heat transfer on vane type cooling system using various turbulence models was carried out[3]. An experiment to study rotor blade vibration on Darmstadt transonic compressor shows increment in aerelastic stability due to decrease of hindrance in the rotor tip section [4]. Modern type of gas turbine have relatively higher performance due to cooling system when compare to heavy duty industrial machines[5]. Influence of evaporative rate and the motion of the droplets on fogging system of gas turbine inlet duct has been study through experiment work [6]. Numerical method was employed to study the injection of water droplets in to the compressor [7].

The compressor of operational gas turbine consumes in consonance to rule two-third of useful expansion work[8]. Wet compression process has significant differences with conventional dry compression process due to latent heat being absorbed by the water droplets [9]. From the results obtained, the injection velocity and the arrangement of nozzles could result to the pressure drop and intake distortion which subsequently influence the operational performance of the compressor [10]. Air velocity and water droplets size have influence on the evaporation and cooling of the droplets, the efficiency of the process rely generally on the spray coverage area[11]. Among the important
dominant parameter in the investigation of evaporative rate of wet compression is that, there is low compression work at high evaporative rate and low temperature at the exit of impeller[12]. the standard k-ε turbulence model with an enhanced wall treatment produce consistent and reasonable results among various modelling scheme in cooling simulation [13]. High temperature and low relative humidity yield best performance during inlet cooling of industrial power plant[14]. Effective design of geometry of inlet duct channel was investigated as well as the effect of water injection through nozzles array[15]. From practical point of view on fogging droplets, measurements and verification of droplets size on gas turbine were analyzed and presented [16]. The fog gives cooling when it dissipates in the channel conduit of the air compressor [17].

Fogging system among other forms of Inlet cooling has been used to supplement the performance operation of industrial gas turbine for over decade, however, the technical justification of the mechanism theory has not been fully comprehended when it come to the prospective effects it has on the compressor capacity operation. Numerical simulation of fogging system based on the experimental work carried out by [18] is presented in the present report. Details overview of the complete inlet duct of the fogging system from the design to simulation is elaborated, pressure distribution and pressure drop on the nozzle manifold of the inlet duct and its characteristic behavior under the influence of fogging were examined, evaporation of the spray water and the temperature distribution due to the action of air flowrate were also studied and presented.

2. Model characterization

2.1 Outline of the inlet duct

Most conservative forms of industrial gas turbine outline provide acceptable time for highest amount of the vaporized droplets to evaporates before being compress in the compressor, inlet duct configuration is not an exceptional, thus necessitate selection of a reasonable length during erection of the fogging structure system. The components consist of filter unit with a wire mesh, fogging nozzles array, silencer and the compressor bell mouth intake as shown in the figure 1. Water from a pipe source is conveyed to the fogging nozzles while the suction side provides a room for the suction of the hot and dry ambient air. The byproduct of the inlet duct is moistened and humidified air in the compressor.

![Figure 1](image1.png)

Figure 1. Illustration of inlet duct fog system

2.2 Numerical Method and description of the fogging system

A Commercial software package ANSYS CFX is chosen for this research, figure 5 shows the model channel for the inlet air to move over sequence of some wire mesh before it reaches on the nozzle arrays. The inlet duct channel configuration is design to fit the existing Alstom GT13E2 gas turbine, the mesh of the entire inlet duct channel is shown in Figure 2.
The wire mesh intake filter presented is highly significant and sensitive zone, it is also a zone on to which bulk of the pressure drop occurs.

The fogging system is divided into primary/normal and secondary/overspray zones. The primary nozzle zone moistens the air closely to 80% relative humidity while overspray zone is to supersaturate the inlet air.

A pipe which acquiesce easy access to air flow and of size 12.7mm diameter length is used for the fogging system channel, an impaction-pin nozzles type is selected throughout the course of the research and is attached at specified positions on the manifold pipes. Water at a substantial speed of 25 m/s to 50 m/s operates at high control pressure of 13785 pa were allowed to pass through the nozzle cavity at 19 microns of droplets for a total of 1890 nozzle. The grid number of mesh for nozzles was found to be 12,100,000.

Silencer is an essential device and component of fogging system, it main purpose is to weaken the noise [19]. Rectangular transitional type of silencer is chosen in the throughout the simulation research.

2.3 Equations and turbulence model

Fluid motion in porous media is computed through typical exemplary in ANSYS CFX for 3D phase flow or with Euler-Lagrange method for two dimensional. The momentum loss model is available in the form of fluid dominions, whereas the complete porous loss model is only feasible in porous domains.

The general momentum equation of a fluid domain is given by:

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = \frac{\partial p}{\partial x_i} - \rho g_i + \frac{\partial \tau_{ij}}{\partial x_j} + S^M_i
\]

(1)

The momentum source \( S^M_i \) is express by the equation:

\[
S^M_i = -C^{R1} U_i - C^{R2} |U| U_i + S^{spec}_i
\]

(2)

From equation 2 \( C^{R1} \) and \( C^{R2} \) stand for the linear resistance coefficient and quadratic resistance coefficient respectively. \( S^{spec}_i \) contain other momentum sources (which may be directional) while \( U_i \) and \( U_j \) are the superficial velocities.
During the computational processes, moderate fragment diameter of the droplet has to be chosen before the commencement of the computational analysis, and in the present research a value of 19 microns is selected. The dispensation of the disintegrated particle diameter obeys Rosin-Rammler distribution function.

Turbine manufacturers have their own peculiar air intake arrangement subject to the turbo-machine design or power plant configurations. Injected air must be conveyed in the most aerodynamic manner towards the compressor air intake.

In the present work, a quality test on several models was carried out and the results agree with the findings in reference [13]. Among the model selection the k-ε turbulence model prove most powerful and is fitted for the current research [17]. The two-phase flow is computed with Euler-Lagrange method in the computations.

Having defined air in the form of continuous phase while droplet as dispersed phase. The k-ε turbulence model is enabled the task to deal with turbulent air-flow. Turbulence interaction with particle trajectory estimation has additionally been incorporated into the CFD simulation.

The standard k-ε turbulence transport equations model are:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \mu_j \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{ib} + \rho \varepsilon
\]

(3)

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu_j + \frac{\mu_j}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 3} P_{ib} \right)
\]

(4)

Where, \( C_{\varepsilon 1} \) and \( C_{\varepsilon 2} \) are constant terms, \( \sigma_k \) stands for Prandtl number for \( k \) and \( \sigma_\varepsilon \) is the turbulent Prandtl number for \( \varepsilon \). \( P_{ib} \) and \( P_{ib} \) stands for the effects of buoyancy forces. \( P_k \) is the turbulence generation by virtue of viscous forces.

3. Results and discussion

Figure comprises results from a wind tunnel experiment [18] and computational results on the flow resistance through pressure drop. In both wind tunnel experiment and computational fluid dynamic inlet duct length of about 12.7m with capacity up to 25m/s airflow velocity were put to use. The results show consistency in the resistance of the pressure drop on the nozzles.

\[ \text{Figure 3. Validation with Numerical work} \quad \text{Figure 4. Reference Positions for fogging evaluation} \]

In an effort to study the pressure dispersion along the nozzle channel some distinctive positions where selected at different length interval between the nozzle arrays and the compressor bell mouth,
extensively, mass flow rates and evaporation capacity where equally study at the various selected positions.

Figure 5 and Figure 6 presents the contours of pressure dispensation at different designated positions downstream of the nozzles manifold with and without silencer respectively. Each designated position has a peculiar flow field trend, and the trends uniformity increases with positions downstream the manifold. At position $X_1=0.5m$ from the nozzle there is relatively bumpy pressure dispensation, the bumpiness reduces at a position distance $X_2=1.5m$, until it reaches a distance $X_3=2.5m$ where the flow field is progressively uniform.

However, the pressure dispensation in figure 5 appear indelicately rough compare to cases in figure 6 due to the nature and obstacle of the turbulent flow of the evaporated droplets, silencer provides reduction to the obstacle in the flow field which in turns offers uniformity of the mass flow rate of the evaporates, it also, escalate residence time for the evaporation and evaporation capacity as presented in Table 1.

![Figure 5](image)
**Figure 5.** Contours of pressure dispensation at selected positions downstream the nozzles manifold without silencer

![Figure 6](image)
**Figure 6.** Contours of pressure dispensation at selected positions downstream the nozzles manifold corelative to silencer

During fogging process, the system is sub-divided in to two zones namely the primary zone and the secondary zone with each serving it purpose. The main classifications purpose is to provides the operator with options during time of critical power need. The primary zone which often referred as normal zone gives adequate moisture to the airflow with adequate evaporation capacity along the inlet duct while the secondary zone is often used when the airflow is highly dehumidify.

Table 1 show fogging evaporation of some selected positions along the inlet duct and downstream of nozzles manifold subjected to primary zone fogging. The mass flow rate increases with fogging
system and moves consecutively along the selected positions until near the outlet when it suddenly reduces, the sequence of the motion of the mass flow rate and evaporation capacity is similar to the contour distribution of the pressure along the inlet duct as explained in Figure 5.

Table 1. Evaporation of fogging along the inlet duct with primary nozzle

| Position          | Mass flow rate without silencer (kg/s) | Evaporation capacity (kg/s) | Evaporability |
|-------------------|----------------------------------------|----------------------------|---------------|
| Inlet             | 514                                    | -                          |               |
| Position x₁       | 515.033                                | 1.033                      | 77.49         |
| Position x₂       | 515.152                                | 1.152                      | 86.42         |
| Position x₃       | 515.168                                | 1.168                      | 87.62         |
| Near the outlet   | 515.047                                | 1.047                      | 78.54         |
| Outlet            | 515.274                                | 1.274                      | 95.57         |

Position x₃ is the best position with evaporability of 87.62% that would generate maximum mass flow rate at the outlet position and without effect on the compressor blades.

![Temperature Contours](image)

(a) Horizontal view along the inlet duct  
(b) Top view of the inlet duct

**Figure 7** Contours of temperature along the inlet duct pipe

During fogging process moisture is induced in to hot and dry air to lower the air temperature before it reaches the intake of the compressor bell mouth. The trajectory of the temperature on the targeted positions along the inlet duct is a decisive moment to understand the points at which temperature changes most decisively due to optimum evaporation. Figure 7 show the air temperature scale drops from inlet ambient temperature of 32°C to around 26°C at the toward the intake of the compressor, the temperature difference of roughly 6°C coincides with the experimental values of 6.1°C [chaker]. Nevertheless, from the top view in figure 7b the drops in the temperature along the inlet duct is proportional to the positions downstream the nozzles array before it suddenly reaches it lowest at the compressor bell mouth.

4. Conclusions

A new technique for fogging system evaluation through numerical simulation was suggested and presented base on the wind tunnel experimental work [18]. From the evaluation results, the proposed fogging inlet duct was model successfully and implemented to augment the performance of the compressor of gas turbine. Variation of air flow resistance (pressure drop) with inlet velocity on the nozzles manifold were verified, the flow field of pressure distribution in the inlet duct channel at selected distances between the nozzles manifold and compressor bell mouth were evaluated, distances
between 2.0 m to 2.5 m are highly recommended and are characterized with uniform pressure distribution and adequate evaporation capacity of the mass flow rate.

Acknowledgments
The authors wish to thank the financial support of National Natural Science Foundation of China (Grant No. 51409067) and the support of Aeronautical Science Foundation of China (Grant No. 201410P6003). And Turbomachinery laboratory of Harbin Engineering University, China.

References
[1] S Batzdorf, T Gambaryan-Roisman and P Stephan 2017 Direct numerical simulation of the microscale fluid flow and heat transfer in the three-phase contact line region during evaporation. *J. Heat Transfer* 140 032401-032401-10
[2] P Bear, J M Wolff, A Gross, C Marks and R Sondergaard 2017 Experimental investigation of total pressure loss development in a highly loaded low pressure turbine cascade *J. Turbomach.*
[3] E Laroche, M Fenot, E Dorigneac, J J Vuillerme, L E Brizzi, and J C Larroya 2017 A combined numerical and experimental investigation of the flow and heat transfer inside a turbine vane cooled by jet impingement *J. Turbomach.*
[4] D Möller, M Juengst, H P Schiffer, T Giersch and F Heinichen 2017 Influence of rotor tip blockage on near stall blade vibrations in an axial compressor rig *Journal of Turbomachinery*
[5] R Bhargava and C B Meher-Homji 2005 Parametric Analysis of Existing Gas Turbines With Inlet Evaporative and Overspray Fogging *J. Eng. Gas Turbine Power* 127 145-158
[6] M A Chaker, C B Meher-Homji and I I I T Mee 2006 Inlet fogging of gas turbine engines: experimental and analytical investigations on impaction pin fog nozzle behavior *J. Eng. Gas Turbine Power* 128 826-839
[7] A J White and A J Meacock 2004 An evaluation of the effects of water injection on compressor performance *J. Eng. Gas Turbine Power* 126 748-754
[8] Q Zheng, Y Sun, S Li and Y Wang 2003 Thermodynamic Analyses of Wet Compression Process in the Compressor of Gas Turbine *Journal of Turbomachinery* 125 489-496
[9] A Mohan, P K Chidambaram, A Suryan and H D Kim 2016 Thermo-fluid dynamic analysis of wet compression process *J. Mech. Sci. Technol.* 30 5473-5483
[10] H Zhang, M Luo, X Pan and Q Zheng 2016 Numerical analysis of gas turbine inlet fogging nozzle manifold resistance *PI Mech. Eng. A-J. Pow.* 230 63-75
[11] Z G H G A Alkhedhair, I Jahn and S He 2014 Experimental Study on Inlet Air Cooling by Water Spray for Natural Draft Dry Cooling Towers Enhancement *19th Australasian Fluid Mechanics Conference, Melbourne, Australia* 8-11 December 2014.
[12] J S Kang, B J Cha and S S Yang 2006 Thermodynamic and aerodynamic meanline analysis of wet compression in a centrifugal compressor *J. Mech. Sci. Technol.* 20 1475-1482
[13] X Li and T Wang Effects of various modeling schemes on mist film cooling simulation *J. Heat Transfer* 129 472-482
[14] G Barigozzi, A Perdichizzi, C Gritti and I Guaiatelli 2015 Techno-economic analysis of gas turbine inlet air cooling for combined cycle power plant for different climatic conditions *Appl. Therm. Eng.* 82 57-67
[15] M Bianchi, M Chaker, A D Pascale, A Perotto and P R Spina 2007 CFD Simulation of Water Injection in GT Inlet Duct Using Spray Experimentally Tuned Data: Nozzle Spray Simulation Model and Results for an Application to a Heavy-Duty Gas Turbine 629-642
[16] M Chaker, C B Meher-Homji and I I I T Mee 2002 Inlet Fogging of Gas Turbine Engines: Part B — Fog Droplet Sizing Analysis, Nozzle Types, Measurement and Testing 429-441
[17] J R Khan, T Wang and M Chaker 2011 Investigation of cooling effectiveness of gas turbine inlet fogging location relative to the silencer *J. Eng. Gas Turbine Power* 134 022001-022001-9
[18] M Chaker, C B Meher-Homji and I I I T Mee 2004 Inlet Fogging of Gas Turbine Engines—Part
III: Fog Behavior in Inlet Ducts, Computational Fluid Dynamics Analysis, and Wind Tunnel Experiments J. Eng. Gas Turbine Power 126 571-580

[19] P Rosin 1933 The laws governing the fineness of powdered coal J. Inst. Fuel. 7 29-36
[20] B E Launder and D B Spalding 1972 Lectures in mathematical models of turbulence