Proceedings of ASME Turbo Expo 2002
June 3-6, 2002
Amsterdam, The Netherlands

2002-GT-30564

INLET FOGGING OF GAS TURBINE ENGINES - PART C: FOG BEHAVIOR IN INLET DUCTS, CFD ANALYSIS AND WIND TUNNEL EXPERIMENTS

Mustapha Chaker, Ph. D.
Director, Research and Development
Cyrus B. Meher-Homji
Chief Engineer
Thomas Mee III,
Chairman and CEO

Mee Industries Inc.,
Gas Turbine Division, Monrovia, CA, USA.

ABSTRACT

The inlet fogging of gas turbine engines for power augmentation has seen increasing application over the past decade yet not a single technical paper treating the physics and engineering of the fogging process, droplet size measurement, droplet kinetics, or the duct behavior of droplets, from a gas turbine perspective, is available. This paper along with Parts A and B provides the results of extensive experimental and theoretical studies conducted over several years coupled with practical aspects learned in the implementation of nearly 500 inlet fogging systems on gas turbines ranging in power from 5 to 250 MW. In part C of this paper, the complex behavior of fog droplets in the inlet duct is addressed and experimental results from several wind tunnel studies are covered.

NOMENCLATURE

C	Discharge Coefficient
CFD	Computational Fluid Dynamics
D_d	Droplet Diameter (m)
IGV	Inlet Guide Vanes
U	Flow Velocity ($m.s^{-1}$)
V_{rel}	Relative velocity of the droplet ($m.s^{-1}$)
We	Weber Number
Y	Expansion factor
β	Ratio of Open Area to Total Area
ΔP	Pressure Drop in the duct (Wg)
γ_w	Surface Tension of the water ($N.m^{-1}$)
ρ_a	Density of the air ($kg.m^{-3}$)

INTRODUCTION

Over the past decade, the application of inlet fogging for the power augmentation of gas turbines has become increasingly popular. It is estimated that approximately 700 gas turbines have fogging systems installed at this time including many modern F class machines. Part C of this paper focuses on fog behavior in inlet ducts while parts A and B, Chaker et al [1, 2], cover the area of modeling droplet evaporation, practical considerations, and nozzle testing.

Due to the wide range of duct configurations and arrangements, there are several issues that become of cardinal importance in designing and implementing fogging systems. Some of these issues include

- Ducts with complex geometry. For example, some ducts have multiple 90 degree bends.
- Ducts with short configurations that minimize fog residence time.
- Duct obstructions, blow in door interference, trash screens, silencer bull-noses, etc.
- Wall & floor wetting considerations.
- Drainage issues.
- Silencer issues and fog interaction.
- Optimal selection of fog nozzle manifold pitches.

In order to examine and understand several of these complex issues, a carefully designed and controlled testing facility was needed. For this reason a special variable speed wind tunnel was constructed to simulate different aspects of gas turbine intake ducts. The wind tunnel allows the real-world testing of nozzles, the development of optimal nozzle location and orientation schemes, and the testing of special drain and water removal approaches. The wind tunnel is shown in Figure 1.

Figure 1. Experimental wind tunnel used for studying fog behavior in gas turbine inlet air ducts: 11 meter (36 foot) long, capable of up to 25 m/s (5000 fpm) airflow velocity

EXPERIMENTAL SETUP DESCRIPTION

Wind Tunnel

The wind tunnel consists of a multi-sectioned duct approximately 11 meters in length (36 feet). It has the following sections which can be configured is various ways (dimensions are mm and inches):

- A & B: Two (2) long duct sections with clear polycarbonate walls; 2440 X 860 X 860mm (96 X 34 X 34 inches).
- C: One short duct section with clear polycarbonate walls and floors; 1220 X 860 X 305 mm (48 X 34 X 12 inches).
- D & E: Two reduction sections from stainless steel sheet metal; 1220 mm long, one end 860 X 860 mm, other end 305 X 860 mm (48 inches long, 34 X 34 inches and 34 X 12 inches).
- F: One short section with clear polycarbonate walls, which can serve as a 90° bend; 860 X 860 X 860 mm (34 X 34 X 34 inches).
- G: One fan section, from waterproof plywood; 1220 X 860 X 860 mm (48 X 34 X 34 inches).

Sections A, B and C are built from clear, 1/4 inch thick, polycarbonate sheet. This allows visual observation of the trajectories of the fog droplets in the wind tunnel. Section F can be used either as a 90 degree elbow section in the duct or to increase the length of the duct. This is useful for examining the flow of fog through bends that typical of intake ducts.

Another section that simulates the silencer in a gas turbine was built, also in clear polycarbonate. This section, with dimensions 860 X 860 X 127 mm (34 X 34 X 5 inches), was built to be used in the small section C. The air-velocity in the duct can reach 5.1 m/s (1000 feet per minute) in the big sections A, B, etc. and 15.2 m/s (3000 feet per minute) in the small section C without the silencer or 25.4 m/s (5,000 feet per minute) with the silencer section in place.

The reduction sections D and E are built from stainless steel sheet. Section G is built from wood and made waterproof using a special paint. Two trash screens were built, one at the beginning and one at the end of section G to protect the variable speed fan from foreign objects and to simulate and measure the size and trajectories of the droplets emitted from trash-screens in typical gas turbine ducts. The size of the mesh is 13 mm X 13 mm (1/2 inch by 1/2 inch) and the thickness of the wire is 16 mm (1/16 inch). The second trash-screen used is stainless steel with a 51 mm X 51 mm (2 inch by 2 inch) mesh dimension and 3.2 mm (1/8 inch) wire thickness.

High Pressure Water Atomizing System

The high pressure atomizing system, which generates the fog droplets, uses a variable speed drive Cat Pump that is capable of developing 207 barg (3000 psig). The atomizing nozzles used for testing purposes are either impaction-pin or swirl-type nozzles. The water is filtered and demineralized and the flow is measured with a flow meter before being injected into the duct.

The high-pressure water passes through a manifold and is divided into a number of lines each of which contains a number of nozzles. The configuration allows for modification of the distance between the nozzles, the distance between lines, and also the angle of the nozzles compared to the airflow direction. Single nozzles can also be tested as required.

Data Acquisition System

The laboratory is equipped with a high-speed data acquisition system, which consists of a DAS 1200 board with up to 16 input channels and 2 output channels. Daisy Lab software is used to manage the data acquisition. The DAS monitors

- Relative humidity in several locations.
- Air velocities.
- Fog water flow rate.
- Pressure drop in the duct.

The set up is flexible to allow for a wide variety of gas turbine fogging related experiments. The position of the sensors in the duct is shown in Figure 2.

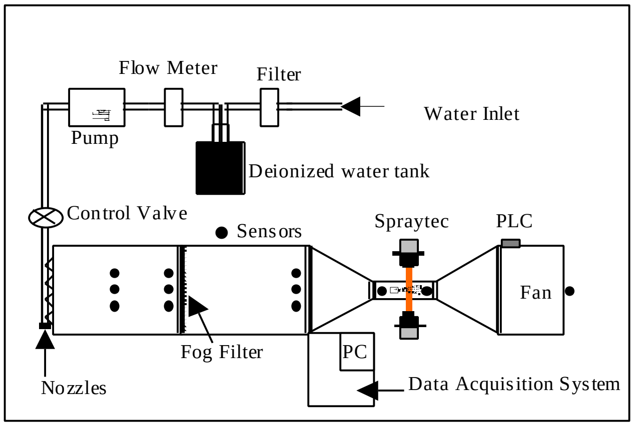

Figure 2. Instrumentation layout in the experimental wind tunnel

Instrumentation

All of the sensors were calibrated before conducting experiments.

Temperature And Relative Humidity
All temperature and relative humidity measurements are taken using VAISALA HMD60U/YO sensors. Accuracies are +/- 1% for the temperature and +/- 2% for the relative humidity.

Water Flow Meter
The water flow coming to the nozzles is measured using an Omega flow meter FLR 1011. The accuracy of this model is about +/- 3%. The sensors use a Pelton type turbine wheel and an electro-optical detection device to convert flow rates into a linear 0 to 5 VDC signal.

Air Flow velocity
The sensor measuring the velocity of the air in the duct is an Omega FMA-905-I air velocity transducer. Additional airflow measurement using a handheld turbine meter is also possible.

Duct Pressure Drop
The sensors measuring the pressure drop in the duct are differential pressure transducers, Omega type PX-274. The measured values correlated well when compared to calculated numbers.

Malvern Spraytec Laser Light Scattering Droplet Measurement System.
A sophisticated Malvern Spraytec droplet measurement system is used along with associated software to characterize and measure droplet distributions and sizes. The laser measurement system is shown in Figure 3 along with a nozzle undergoing tests. Further details are provided in part B of the paper.

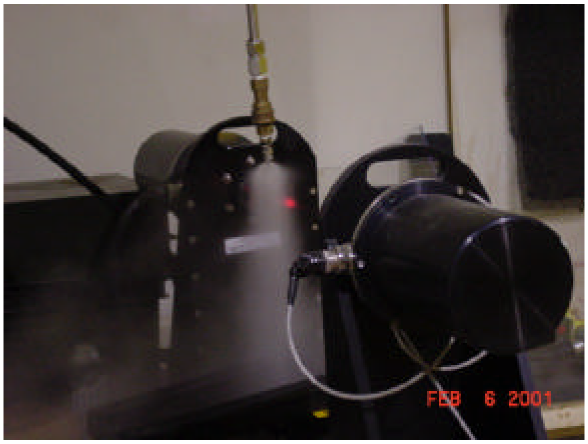

Figure 3. Malvern Spraytec Laser Light Scattering Droplet Measurement System

EXPERIMENTAL WORK USING THE WIND TUNNEL

Nozzle Location

Gas turbine inlet systems come in a wide variety of configurations and shapes. Some of the complexities that have to be addressed include:

- Multiple side entry configurations—two or three-side entry configurations where considerable care has to be taken to arrange nozzle manifolds in such a way has to have uniform coverage.
- Configurations with steeply curved roofs that require progressive changes in the nozzle spray angle to avoid roof wetting and impaction on nearby nozzle manifolds.
- Short duct configurations where residence time is minimal. In these situations special patterns of the fog nozzles may have to be used to optimize the temperature distribution across the airflow. In most cases, horizontal lines are used but in some cases lines may have to be vertical (three-side entry filter systems, for example) to allow nozzle angular changes to be made in a vertical plane.
- Complexities relating to unusual duct obstructions, such as blow-in doors, reheat manifolds, duct support trusses and even generator cooling ducts, which in some turbines pass through the main inlet air duct.

It is important to note that the guidelines presented ahead are based on Mee Fog nozzles, which generate a fog with a Dv90 less than 20 microns[1] and, due to the wide range of intake duct configurations, the optimization of nozzle locations and spray angles is often based on experience. When the situation warrants, computational fluid dynamics (CFD) studies can be performed but even CFD studies must be modified by experience. Analytical tools are useful but lessons learned from *actual installations* contribute immeasurably to a successful final selection of the nozzle array configuration. Figure 4 shows a configuration for a V shaped air filter house that required a nozzle array as shown. A steep duct roof is shown in Figure 5 and the nozzle angular orientation is shown in Figure 6.

The decision to locate the nozzle array either before or after the silencer depends on several factors. A residence time of 1 to 2 seconds of unobstructed flow, is ideal but rarely exists unless a special duct modification is made or it is a new gas turbine installation where extra duct sections can be incorporated at the design stage.

---

[1] As commercially available nozzles made by other makers may produce larger droplets, the guidelines presented here may not apply.

Figure 4. V-shaped duct configuration requiring a special nozzle array configuration

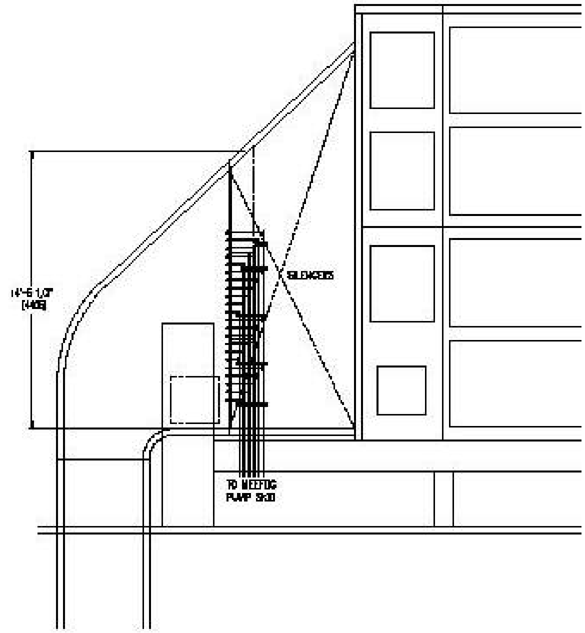

Figure 5. Nozzle array configuration for a steep-roofed duct

Residence time is an important consideration that must be carefully evaluated. As shown in Figure 7, droplets attain the airflow velocity in a few milliseconds due to the large drag forces[2]. This figure shows the response time for a droplet to attain air stream velocity as a function of droplet size.

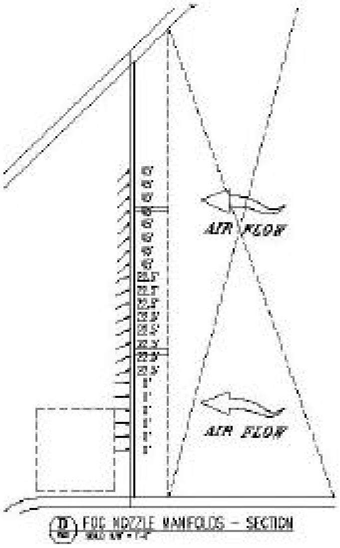

Figure 6. Optimization of nozzle angular orientation

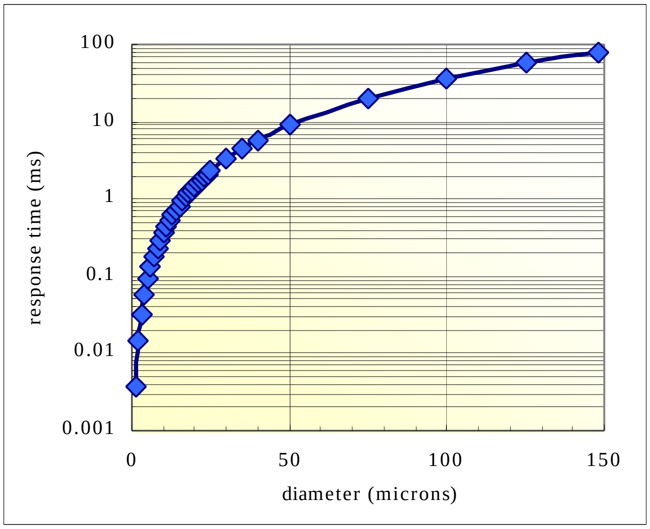

Figure 7. Response time for droplets to attain air stream velocity as a function of droplet size

Even very small droplets can collide with items in the ducts (Figure 8), including silencers, walls at duct bends, structural supports, blow-in doors and trash-screens, of course the problem is much worse with nozzles that produce larger droplets. Such collisions can lead to pooling of water on duct floors or the creation of larger droplets if fog water collects on surfaces and is then re-entrained into the high-velocity airflow. These factors make the proper design of the nozzle arrays, and the water collection and drainage systems, a very important aspect of proper fog system design.

---

[2] As droplet Reynolds number is very small, the Coefficient of drag is exceedingly large and the droplet accelerates very rapidly.

Figure 8. Typical obstruction found in an inlet air duct

Nozzle Orientation and Fog Distribution

Once a suitable location for the nozzle arrays has been established, one must consider the questions of nozzle orientation and distribution, i.e. the spacing of the nozzles on the nozzle manifolds. The fundamental factor driving the design of nozzle orientation and distribution is to obtain uniform droplet distribution in the intake duct, and to thereby ensure that the airflow is evenly cooled. Uneven distribution of nozzles, or large gaps between the nozzle manifold tubes, can result in poor mixing of the fog with the inlet airflow. This leads to longer evaporation times and temperature distortions at the compressor inlet.

The nozzle orientation angle chosen depends on

- Airflow velocity.
- Distance between nozzle lines.
- Operating pressure and spray pattern of the nozzles.
- Duct wall and roof shape constraints.
- Overall duct geometry.

Several studies have been done by CFD modeling and by empirical testing (conducted using the wind tunnel and in actual installations) to verify and optimize nozzle layouts. The nozzle angle with respect to the inlet airflow may vary between 0 degrees (co-flow) and 90 degrees (perpendicular to the flow). Nozzle orientation perpendicular to the flow can be used if the plume does not interact with the duct walls or the other nozzle lines. Orienting the nozzles into the airflow generally results in droplet impaction on the nozzle manifold tubes themselves, and is therefore not recommended. At 90 degrees, the increased angle gives a marginally longer residence time, as compared to the co-flow position. However, due to practical considerations, 90 degrees is rarely used. Angles varying from 0 to 60 degrees have been successfully employed.

The effect of nozzle angle with respect to flow is shown in Figure 9, 10 and 11. Figure 9 shows the shape of the plume in a co- flow position (i.e., the nozzle spraying in the direction of the airflow). The plume diameter stays relatively constant in the axial length. This is an advantage when the distance between the nozzles is not too large as it provides a homogenous pattern across the duct. However in cases where there is a large spacing between nozzles, the 90 degree orientation may be advantageous because, as can be seen in Figure 10,

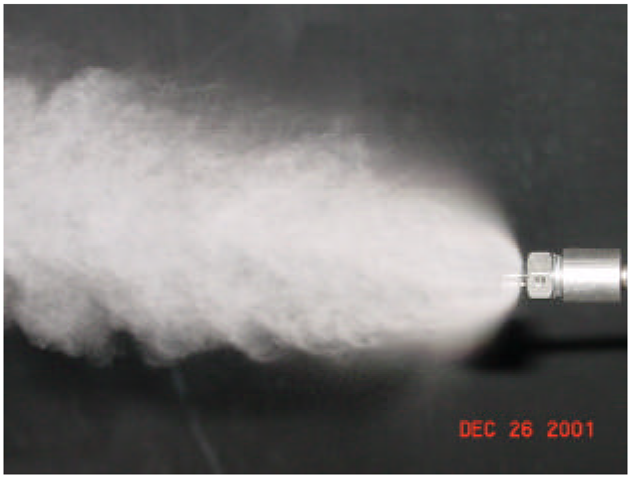

Figure 9. Co–flow nozzle orientation of a fog nozzle in wind tunnel; airflow velocity is 4 m/s (800 ft/min), Operating Pressure is 138 barg (2000 psig)

Figure 10. Ninety-degree orientation of a fog nozzle in the wind tunnel; airflow velocity is 4 m/s (800 ft/min), operating Pressure is 138 barg (2000 psig)

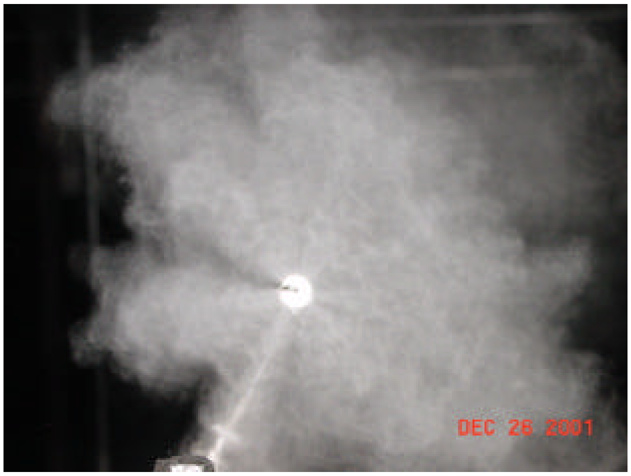

Figure 11. Counter-flow orientation of a fog nozzle in the wind tunnel; airflow velocity is 4 m/s (800 ft/min), operating Pressure is 138 barg (2000 psig)

the plume diameter is much larger and consequently covers more of the duct cross section. The counter-flow position is shown in Figure 11. The counter flow position is never used for the reason stated above.

With respect to fog distribution, Figure 12 shows two lines with multiple nozzles in the wind tunnel, operating at 4 m/s (800 fpm). The distance between the nozzle lines is 20.3 cm (8 inches) . It can be seen that the plumes make contact approximately at an axial distance equal to two times the line pitch. Figure 13, shows a different view of the same setup. The distance between nozzles is 4 inches (10.1 cm). In this case we see that the point of plume contact is approximately equal to the nozzle pitch.

Figure 12. Side view of nozzle array; airflow velocity is 4 m/s (800 ft/min), operating pressure is 138 barg (2000 psig)

Figure 13. Face view of Nozzle Array; airflow velocity is 4 m/s (800 ft/min), operating pressure is 138 barg (2000 psig)

It is important to note that if swirl-jet nozzles were installed in this same way, the conical shape of the swirl jet plume and the existence of much larger droplets in the edges of the plume, may result in coalescence where the cones intersect[3]. In some situations, where larger pitches are required, it may be beneficial to use a triangular configuration as shown in Figure 14.

This configuration results in a very well distributed fog pattern. Several experiments were performed to measure the possibility of droplet collision and coalescence in the intersection of two or more fog nozzle plumes. These experiments show that there is no measurable coalescence of droplets in this region. Numerical analysis confirms that the statistical probability of coalescence in this area is very small. As discussed in Part B of this paper, collision and coalescence do play a roll in droplet size near the nozzle orifice, where both relative velocities and droplet density are much higher.

It is important to understand the velocity and pressure profiles in the intake duct and to recognize that certain areas will have accelerated flow and pressure gradients due to duct bends and turns. Nozzles may have to be specially oriented to accommodate these patterns. In real-world fogging applications, ideal duct configurations hardly ever occur, so proper design of fog nozzle orientation and manifold location is a critical factor for fog system design.

Number of Nozzles and Nozzle Pattern

The number of nozzles, and their spray pattern, should be appropriately designed so they provide uniform fogging of the gas turbine inlet duct. There is a tradeoff between nozzle flow rate, spray pattern, number of nozzles and pressure drop across the nozzle manifolds. Having a fewer number of higher flow rate nozzles can result in larger spaces between the plumes and less homogeneity of the fog distribution. Furthermore, larger flow nozzles, as a general rule, produce larger droplets and fewer nozzles lead to uneven temperature distribution, especially during part-load fogging.

The nozzle pattern itself should be such as to cover the maximum cross-sectional area of the airflow.

Fog Droplet Behavior at Silencers

In practice, nozzle arrays are often best located upstream of the silencers as this location gives the longest residence time and best possible mixing. Therefore it is important to understand by actual experimentation and observation.

- How the fog interacts with the flow field around the silencers.
- Wetting effects on the silencer.
- Ideal distance between fog nozzles and silencers.
- Special approaches such as locating the nozzles between the silencer baffles (this is rarely done but has worked successfully in some cases).
- Effect of different silencer nose profiles on fog droplet behavior and wetting.

---

[3] Part B of this paper discusses this subject at some length.

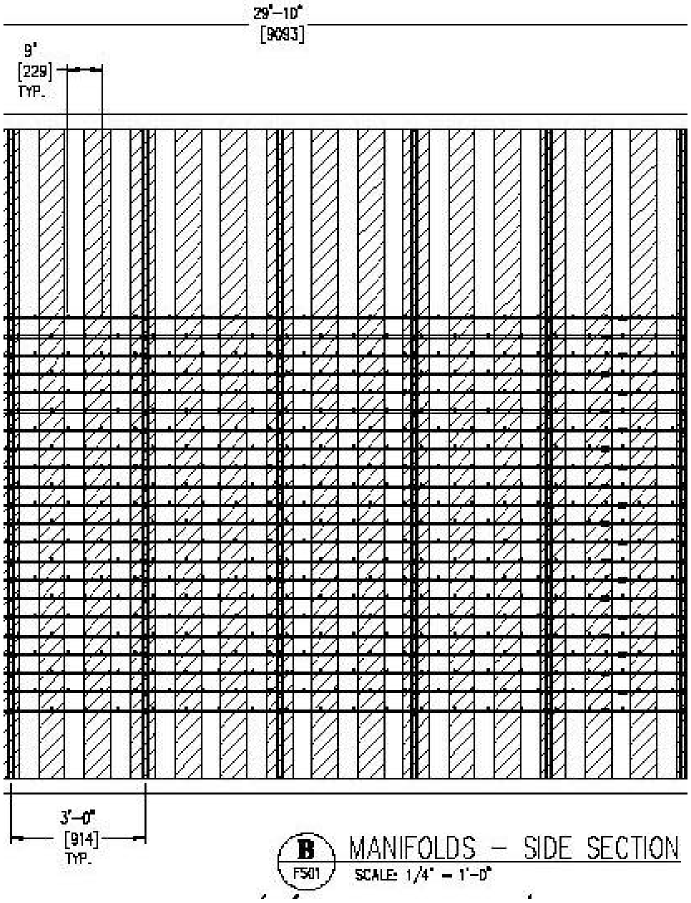

Figure 14. Triangular pattern nozzle arrangement

Several studies were been done using the instrumented wind tunnel. A transparent Polycarbonate model of a silencer section was built and tests were conducted under varying fog and air velocity conditions. Figure 15 shows the silencer section model. The silencer model was also modified with different nose profiles to examine the affect on flow behavior.

Figure 15. Silencer section model used for wind tunnel studies

The flow through silencers was also modeled by CFD techniques as presented in the section ahead and experimental verification was conducted with the wind tunnel. This was done by locating the model at different positions near the wall of the droplet measurement section of the wind tunnel. This allowed visual observation of the flow of fog and also measurement of the droplet sizes at the various silencer locations and under various fogging conditions. Drain water flow from the silencer nose, with different nose profiles, was also measured. These tests resulted in the development of beneficial design tools that allow fairly accurate predictions of quantity fog impaction that can be expected on silencer noses when fog droplets are very small, and under favorable airflow velocities, most of the fog droplets will follow the flow lines around the nose of the silencer.

Fog Interaction with Trash-Screens

As most fogging systems have to operate upstream of a trash screen, it is important to experimentally study the large droplet creation effects from the trash screen. This was done experimentally using the wind tunnel.

Figure 16 shows a photograph of trash-screen sections used in the wind tunnel.

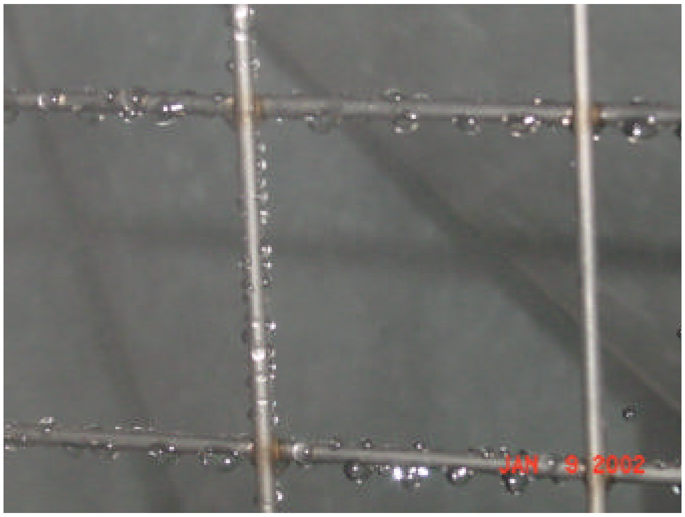

Figure 16. Fog collection on a trash-screen resulting in larger droplet formation

Trash screens are particularly good collectors of small fog droplets because the cross section of the screen wires is so small that droplets cannot easily follow the air stream around them. Experimental results indicate that the droplet sizes formed from fog collection on trash-screen wires were found to be proportional to the wire diameter and in some cases droplets as large as 2 mm (0.08 inches) were noted. This may be of critical importance when trash screens are very close to the compressor inlet. However when the trash screens are located near the filter house, prior to the elbow section, the problem is less severe as good drainage systems can collect any large droplets.

Fog collection on Intake Cones

It is also possible that the intake cones of cold-end drive machines will collect some moisture and some of this may then progress inwards to the compressor, covering the intake struts. Observations and calculations have shown however that the Weber number effect will come into play here, as the relative velocity differences between a droplet on the cone and the flow as it accelerates into the bellmouth will cause the droplet to shatter. More details are provided in the section ahead on the Weber number effect. Figure 17 shows the intake cone of a large gas turbine.

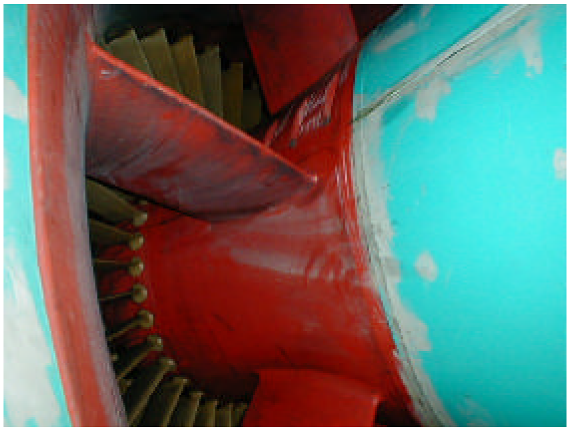

Figure 17. Intake cone and struts at an axial compressor inlet

Droplet Eliminator Application and Testing

Due to the requirements of certain retrofit applications, several studies have been conducted in the wind tunnel to study droplet eliminator behavior under fogging conditions and to optimize the angular location of the eliminator. An experimental setup is shown in Figure 18.

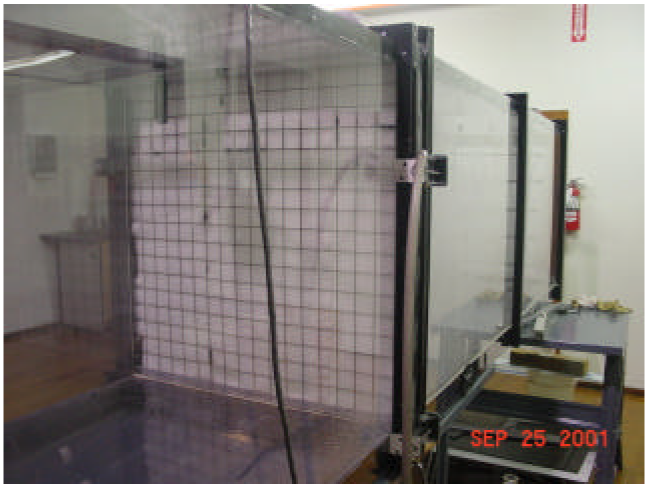

Figure 18. Typical fog droplet eliminator (as used in an HVAC humidification application)

The droplet eliminator induces a large pressure drop that is, as expected a strong function of the flow velocity as shown in Figure 19. Further, as can be seen in the figure, the pressure drop increases significantly when the eliminator gets wet and saturated with water. Our experience with numerous installations has indicated that the use of droplet eliminators is not beneficial for gas turbine applications because of the inlet pressure drop penalty, which detracts from performance year round.

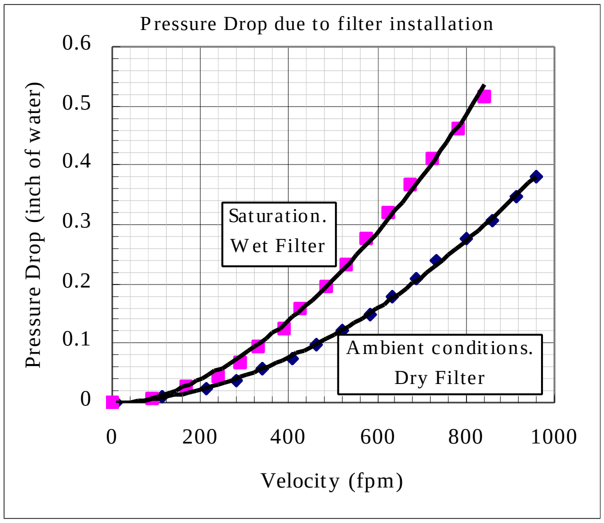

Figure 19. Pressure drop of a droplet eliminator under both dry and wet conditions

SIGNIFICANCE OF WEBER NUMBER FOR INLET FOGGING

The Weber number was covered in Part B of the paper with respect to droplet behavior near the fog nozzle. The Weber number is the ratio of aerodynamic forces to surface tension forces and is given by the equation,

$$We = \frac{\rho_a V_{rel}^2 D_d}{\gamma_w} \qquad (1)$$

Studies have shown that shattering of droplets occurs when the inertial forces overcome the surface tension forces, which happens when the Webber number is less then 13 [3]. Details on aerodynamic breakup of liquid droplets are provided in Suzuki and Mitachi [4] and Samenfink et al [5].

Due to the high relative velocities that would occur in the event that water droplets collected on the intake cone or on the Inlet Guide Vanes (IGVs), the Weber number effect is of some importance.

Figure 20 shows a graph that shows what air velocities and droplet sizes would result in droplet shattering. The velocities indicated are relative velocities between the droplet and the airflow. For the cases where we have droplets adhering to stationary surfaces, we can assume that the droplets themselves are at a very low velocity and so the *relative* velocity becomes high, almost equal to the air stream velocity. It is important to note that this figure should only be applied to larger agglomerated droplets that may be present near the compressor.

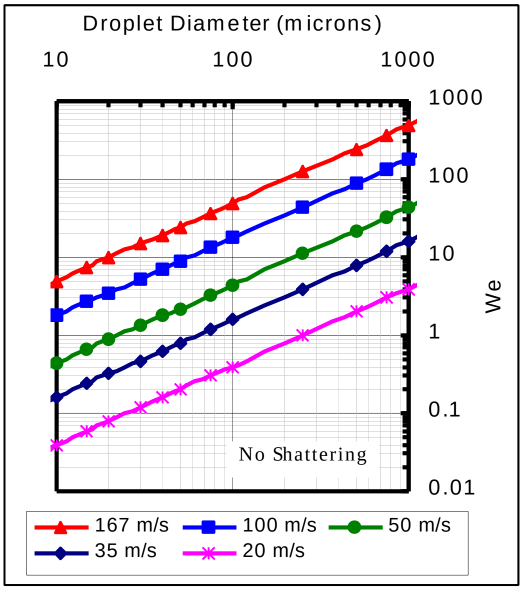

Figure 20. Graph showing the Weber number for varying droplet sizes. Velocities shown are relative velocities between the droplet and the airstreams

CFD ANALYSIS FOR FOG FLOW IN GAS TURBINE INLET DUCTS

CFD analysis has been extensively used for

- Examinations of flow and pressure fields in intake ducts.
- The study of droplet behavior around obstructions.
- To optimize nozzle array locations and the arrangement and orientation of nozzles on the arrays.
- To evaluate droplet trajectories for different droplet diameters and airflow velocities so that the potential for collision and coalescence of droplets can be studied.

The CFD model uses a Lagrange-Euler framework for two-phase flows, which is appropriate for the modeling of fog behavior. Flow patterns for the fog for different types of ducts have been examined. Many configurations that have been modeled by CFD studies have been experimentally verified using wind tunnel and real-world studies. In this way, the accuracy of the models can be constantly tuned and validated. Some typical applications are depicted ahead.

The trajectories of fog droplets emitted from a nozzle in an elbow section in the duct are shown in Figure 21. This has been done for relatively small droplet sizes. It is possible to vary the airflow and to see how the droplet trajectory will follow the duct bends. Figure 22 shows the velocity distribution in the duct over silencer sections in two dimensions. Figure 23 shows the distribution of the velocity in the duct in 3D. Figure 24 shows the velocity profile in a duct bend in two dimensions.

CFD analysis can be applied to establish the best possible location for fog nozzle manifolds in both retrofit situations and new units. The ideal location for nozzle manifolds is the location that gives the maximum residence time in the duct for the fog droplets prior to the airflow entering the compressor (to give maximum evaporative efficiency and ensure a minimum size for any droplets entering the compressor) and that results in the least amount of fog-impaction and collection on obstructions in the duct.

As mentioned before, establishing the best location for nozzle manifolds is as much an art as a technology and requires extensive experience with real-word fog applications. Duct configurations and the types of obstructions present in ducts vary widely and it is, therefore, important that operators who are considering installing an inlet fog system provide detailed drawings and, where possible, photographs of the inside of the air filter house and the inlet ducts to the fog system supplier. A detailed design justification should also be requested from the fog system supplier. The design justification should include a detailed description of the proposed design and information about the design, including:

- The design justification for the chosen fog nozzle manifold location with information about residence time and expected size of any droplets that might enter the compressor.
- Information about potential for fog impaction on obstructions, silencers, duct walls and floors, etc. and methods proposed for mitigation of excessive pooling.
- Information about drains, drain channels and other water collection devices.
- Calculations for nozzle array strength and methods for avoidance of airflow-induced vibration and FOD.

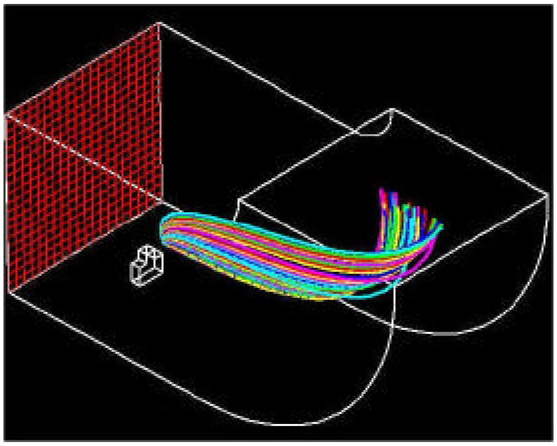

Figure 21. CFD representation of fog flow streamlines in a duct bend

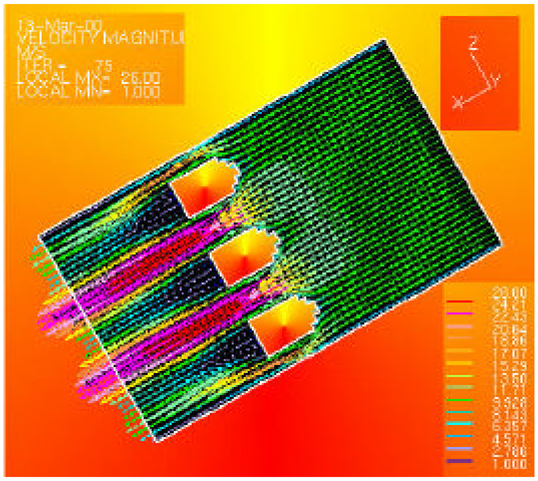

Figure 22. Flow Pattern velocity field around silencer section

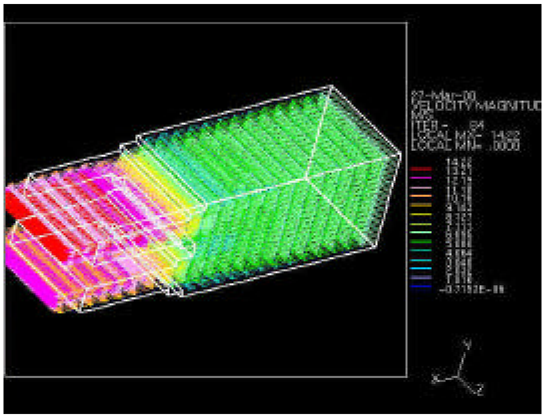

Figure 23. Three-dimensional visualization of flow velocity field over a silencer section

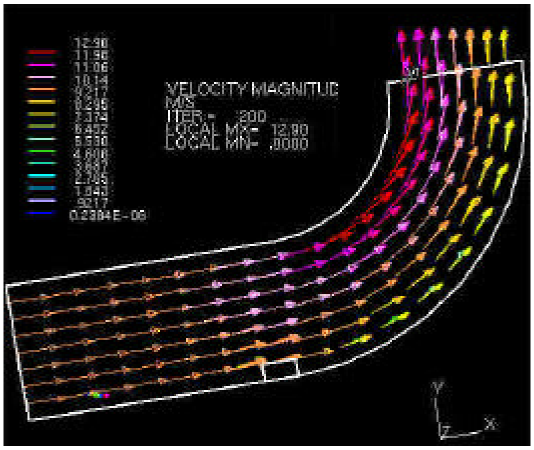

Figure 24. Velocity flow vectors in turning duct flow

MECHANICAL DESIGN OF THE FOG NOZZLE MANIFOLD

Vortex Shedding And Vibration Considerations

Fog nozzle manifolds are located downstream of the filter systems so proper care has to be taken to ensure their mechanical integrity. The nozzle manifolds and their supporting structures will exhibit resonant oscillations under certain velocity conditions, as a consequence of vortex shedding. Bluff bodies in an air stream shed vortexes at an oscillating rate that is a function of the flow velocity and the shape of the structure. If the vortex shedding frequency is equal to, or a harmonic function of, the natural resonant frequency of the manifold tube or support structure, the oscillations can reach an amplitude sufficient to cause catastrophic failure of the array. There are several design approaches relating to strengthening of the structure and the use of vibration absorbing clamps but it is important to carefully evaluate the natural frequency behavior of the nozzle array. The array should be studied under the range of velocities that can occur for the given turbine. The compressor ingests a constant volume of air (Assuming a fixed speed single shaft design) so that changing atmospheric conditions result in changes in the air velocity in the duct. Furthermore, if the turbine is operated off-design for extended periods of time, the airflow velocity in the duct must be considered under these conditions. Nozzle manifolds must be evaluated under both the wet and dry conditions as water in the tubes will change their natural frequency.

To avoid the occurrence of resonance, calculations must be made of the natural frequencies of the tubes and the struts using analytical formulae. The forcing frequencies (vortex shedding frequencies) due to the airflow past the cylindrical or bar-shaped (tube or strut) are calculated using formulae and dimensionless numbers—the Reynolds (**Re**) and Strouhal (**St**) numbers. The resonant frequencies of tubes and struts depend on the material of construction and dimensions. Knowing the range of possible airflow velocities in the intake duct, it is possible to select the right nozzle line configuration in order to ensure the avoidance of resonance. Details on flow-induced vibration may be found in Thomson [6], Barbi et al [7], Stansby [8] and Ongoren et al [9].

Based on our standard tubes and struts, a design guide has been developed to evaluate the structural properties of nozzle arrays. Figure 25 provides design envelope guidelines based on the first mode of vibrations for tubes running dry for a range of velocities. Figure 26 provides the same guideline for tubes running wet.   These curves are plotted with error bars of ± 7.5%. They show the interaction for only the first mode of oscillation, the second and the third mode only occur at a tube length of more then one meter (39 inches).

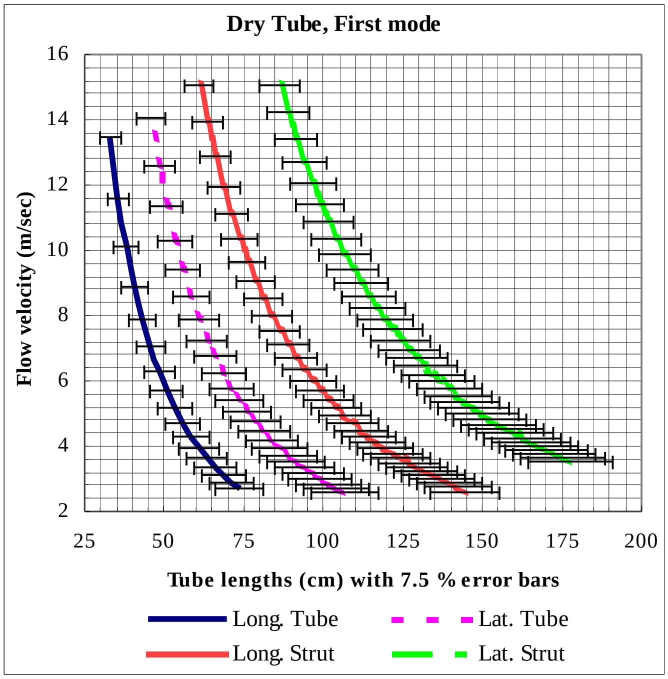

Figure 25. Resonant frequency inteference diagram for dry operation of nozzle array tubes

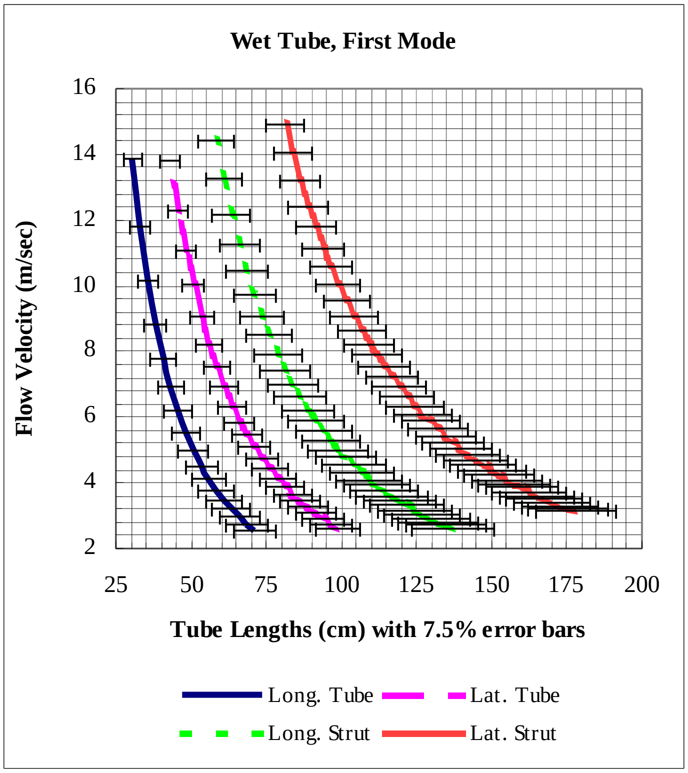

Figure 26. Resonant frequency inteference diagram for wet operation of nozzle array tubes

Pressure Drop Considerations

The air in the duct behaves as a compressible fluid when it goes through an orifice; the flow expands adiabatically as the pressure in the air falls. In this case the downstream static pressure will be less than that measured upstream. The equation to calculate the pressure drop in this condition is:

$$\Delta P = \frac{\left(1-b^4\right)\times U^2 \times r}{2\times Y^2 \times C^2} \qquad (2)$$

Where ΔP is the pressure drop, β is the ratio of the open area to the total area: $\beta = A_{open}/A_{tot}$. U is the flow velocity through the orifice throat, ρ is the fluid density measured at the upstream position, C is the discharge coefficient, it is equal to 0.6 when the Reynolds number is higher than 10000, which is always the case in our conditions, and Y is the expansion factor, which depends only on the ratio of specific heat at constant pressure (cp) to that at constant volume (γ).

The pressure drop through the typical nozzle manifold, as calculated and corroborated by measurement, are exceedingly small (on the order of <2.5 mm water gauge, <0.1 inch water gauge). It is in fact so small that it is difficult to measure[4]. To understand the reason for the small pressure drop, we note that β is relatively big for a typical inlet air fogging array and U is also small in the location just after the filters.

DUCT DRAIN STUDIES AND RECOMENDATIONS

The importance of proper and efficient drainage has been covered in Part A of this paper. Drains are often required at several locations in the inlet duct as well as at the bellmouth region. It is inevitable that some water runoff will occur due to wetting of duct walls and floors, and the presence of bends and other obstructions. Drains should be either P trap designs or flapper-style check valves. All drains provided must be open and operating during periods of inlet fogging. P trap arrangements are not preferred because of the risk of the water seal evaporating causing the trap to run dry and allowing ingestion of untreated air into the engine. P traps must also be drained during freezing conditions and fitted with a valve that can be closed during extended shutdown of the fog system, when the trap will be empty.

Several experimental and in-the-field tests have been conducted on drain systems to study:

- Optimizing drain locations.
- Optimizing the drain sizes.
- Optimizing guttering that can be used to channel water into the drains.
- Development of special wall-drain gutters.

---

[4] We refer only to the pressure drop due to the nozzle manifold. The overall filter house pressure drop is a function of the filter differential pressure that is a square function of the flow velocity.

Figure 27 shows an experimental setup developed to actually quantify how much water could be evacuated by different line sizes under different negative pressure conditions. While this can be evaluated theoretically, we felt that it was important to actually test different drain configurations under realistic conditions. In this test apparatus, a viewing window was located to examine the flow behavior of the water that was being evacuated.

Figure 27. Test rig for drain studies

This sort of study ensures that the drains installed have the capability of accommodating the total flow under array leak conditions that might result in a large amount of water being dumped. However, it is important to note that, with a well-designed fog system, drains handle only 2 - 5 % runoff flow under normal conditions. Furthermore, even very small leak in the nozzle system would be immediately detected by a properly designed fogging control system and an immediate shutdown would occur.

The use of viewing windows is recommended both at the nozzle array location and at the inlet bellmouth location. This is valuable in evaluating the fogging system performance and for tuning purposes. A viewing window installed on a large heavy-duty gas turbine located at the intake section to the axial compressor is shown in Figure 28.

Figure 28. Viewing window at the compressor inlet section of a large gas turbine. Upper window is for lighting arrangement

CLOSURE

This paper has provided details of wind tunnel testing and CFD analysis for inlet air fogging in gas turbine ducts. In this paper along with Parts A and B, only some of the key issues could be briefly covered, in order to help gas turbine users in implementing fogging systems. Further papers will treat specific topics in greater detail. For an understanding of fogging behavior, it is imperative that theoretical analyses be validated by laboratory and when possible, by in-the-field testing.

ACKNOWLEDGMENTS

The lead author would like to acknowledge the contributions of Allen Reinholtz of Mee Industries' controls group for his work in helping set up the wind tunnel measurement systems and Conrad Klemzak, MeeFog R&D technician, for his help with the experimental setups. We also acknowledge and thank the large number of Meefog system users who's technical inputs and support has been most valuable.

REFERENCES

[1] Chaker, M., Meher-Homji, C.B., Mee T.R. III, (2002) *"Inlet Fogging of Gas Turbine Engines-Part A: Fog Droplet Thermodynamics, Heat Transfer and Practical Considerations,"* Proceedings of ASME Turbo Expo 2002, Amsterdam, The Netherlands, June 3-6, 2002, ASME Paper No: 2002-GT-30562.

[2] Chaker, M., Meher-Homji, C.B., Mee T.R. III, (2002) *"Inlet Fogging of Gas Turbine Engines-Part B: Fog Droplet Sizing Analysis, Nozzle Types, Measurement and Testing,"* Proceedings of ASME Turbo Expo 2002, Amsterdam, The Netherlands, June 3-6, 2002, ASME Paper No: 2002-GT-30563.

[3] Faeth G. M., Hsing L.P., Wu P.K., (1995) "*Structure and Breakup Properties of Sprays*", Int. J. Multiphase Flow, Vol. 21, P99.

[4] Suzuki, T. and Mitachi, K., (2000) " *Experimental Study on Aerodynamic Breakup of Liquid Droplets in Time Dependent Relative Velocity Fields*," Eight International Conference on Liquid Atomization and Spray Systems, Pasadena, CA, USA, July 2000.

[5] Samenfink, W., Hallmann, M, Elsaber, A and Eittig, S. (1994) "*Secondary Breakup of Liquid Droplets: Experimental Investigation for a Numerical Description*, " ICLASS-94, Rouen, France July 1994.

[6] Thomson W. T, "*Theory Of Vibration With Applications*", 1988, Prentice Hall, Third edition, New Jersey. Page 223.

[7] Barbi C., Favier D.P., Maresca C.A., Telionis D.P; "*Vortex Shedding And Lock-On Of A Circular Cylinder In Oscillatory Flow*"; J. Fluid Mech. 1986, vol. 179, P. 527.

[8] Stansby P.K, " *The Locking-On Of Vortex Shedding Due To The Cross-Stream Vibration Of Circular Cylinders In Uniform And Shear Flows*"; J. Fluid Mech. 1976, vol. 74. P. 641.

[9] Ongoren A,. Rockwell D; "*Flow Structure from an Oscillating Cylinder. Part 1. Mechanisms of Phase Shift and Recovery in the Near Wake*," J. Fluid Mech. 1988, vol. 191, P. 197.