



**HAL**  
open science

# Hydraulic Effect on a Cooling Tunnel and First Stage of an Axial Compressor

Carlos Luis Moreno Negrin, Carlos Chacon, Miguel Asuaje

► **To cite this version:**

Carlos Luis Moreno Negrin, Carlos Chacon, Miguel Asuaje. Hydraulic Effect on a Cooling Tunnel and First Stage of an Axial Compressor. 16th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Apr 2016, Honolulu, United States. <hal-01516465>

**HAL Id: hal-01516465**

**<https://hal.science/hal-01516465v1>**

Submitted on 1 May 2017

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons CC BY 4.0 - Attribution - International License

# Hydraulic Effect on a Cooling Tunnel and First Stage of an Axial Compressor

Carlos Luis Moreno<sup>1\*</sup>, Carlos Chacon<sup>2\*\*</sup>, Miguel Asuaje<sup>2\*\*\*</sup>



## Abstract

One of the most effective methods for increasing a gas turbine thermal efficiency, is using “fogging” at the axial compressor first stage. That is why some investigations have been made by scientists all over the world, in order to create a model that represents fairly the hydraulic behavior of the water droplets flowing through the first stage of the axial compressor that forms part of a gas turbine. However, few works have been made for analysis of such behavior on systems conformed by both cooling tunnel and first stage of axial compressor. The main previous work was made by Moreno et al., where only a compressor hydraulic behavior was taken into account. It is important to notice that only first stage of rotor and stator was simulated using CFD techniques. The objective of this paper is to simulate the system conformed by cooling-tunnel and first stage of an axial compressor, when considering fogging technique, since on previous works on literature both components have been studied separately. Several variables like dry-bulb temperatures, droplet sizes, water injection rates, air mass flow rates and others were combinely evaluated, in order to enhance an improvement on axial compressors thermal efficiency. The numerical analysis was performed using CFX simulator. Also this study could help improve gas turbines performance, which implies lower costs of operation.

## Keywords

Compressor — Fogging — Droplets—Efficiency

<sup>1</sup>Department of Thermodynamics and Transport Phenomena, Simon Bolivar University, Caracas, Venezuela

<sup>2</sup>Department of Conversion and Transport of Energy, Simon Bolivar University, Caracas, Venezuela

\* [carlosmoreno@usb.ve](mailto:carlosmoreno@usb.ve) \*\* [carloschacon@usb.ve](mailto:carloschacon@usb.ve) \*\*\* [asuajem@usb.ve](mailto:asuajem@usb.ve)

## INTRODUCTION

One of the most effective methods for increasing a gas turbine thermal efficiency, is using “fogging” at the axial compressor first stage. That is why some investigations have been made by scientists all over the world, in order to create a model that represents fairly the hydraulic behavior of the water droplets flowing through the first stage of the axial compressor that forms part of a gas turbine. However, few works have been made for analysis of such behavior on systems conformed by both cooling tunnel and first stage of axial compressor. Among the previous works made on this researching area, Khang and Wang [1], helped to understand the water mist transport in the compressor flow. To help understand the water mist transport in the compressor flow passage, this study conducts a 3D computational simulation of wet compression in a single rotor-stator compressor stage using the commercial code CFX. An Eulerian-Lagrangian method is used to calculate the continuous phase and track the discrete (droplet) phase. Analysis on the droplet history (trajectory and size) with stochastic tracking is employed to interpret the mechanism of droplet dynamics under the influence of local turbulence, acceleration, diffusion, and body forces. The result shows that the local thermal equilibrium is not always achieved due to short residence time and high

value of latent heat of water. The erosion model predicts that the most eroded area occurs in the leading edge and one spot of the trailing edge of the rotor suction side.

On the other hand, Sun et al. [2] made numerical simulations of 3D compressible separated flows within a wet compression compressor stage. They carried out using a computational fluid dynamics (CFD) program. Numerical computations of flow fields in a compressor cascade with wet compression assume that a separated region exist in the corner of the rotor blade suction surface and hub surface in the case of dry compression. Under different operating conditions and with wet compression this study presents the changes in the extent of separated region on the flow channel surfaces, compression efficiency, pressure ratio and specific compression work, etc. Also effects of factors such as droplet size, droplet temperature, and injected water flow rate on the compressor stage performance and flow field within compressor stage passage had been investigated. The results show that wet compression could weaken and eliminate the flow separation and then the efficiency and pressure ratio maintain a high level.

Jeanty et al. [3] made 3-D simulations using CFD techniques, in order to model a system conformed by a cooling-tunnel where water droplets were sprayed, and variables like relative humidity, humidity ratio, dry-bulb and wet-bulb temperatures, heat transfer rates and bubble diameter sizes were

characterized. So it was probed the importance of the fogging technique, prior to the first stage of fogging systems. Later, Perez et al. [4], analyzed a fogging system on a natural gas air cooler, using CFD techniques. As a result, a simplified geometry with such aim was made. Grid was validated using GCI consideration. Heat and mass transfer were taken into account for simulations. Variables like water droplets velocity, droplet size, humidity ratios, relative humidity, dry-bulb temperatures and wet-bulb temperatures were characterized. It was showed the effectiveness of the fogging method for these purposes. This work could guide the management of fogging technique simulation, considering Lagrangian focus.

The main previous work was made by Moreno et al. [5], where only a compressor hydraulic behavior was taken into account. It is important to notice that only first stage of rotor and stator was simulated using CFD techniques. Finally, the objective of this paper is to simulate the system conformed by cooling-tunnel and first stage of an axial compressor, when considering fogging technique, since on previous works on literature both components have been studied separately. Several variables like dry-bulb temperatures, droplet sizes, water injection rates, air mass flow rates and others were combinely evaluated, in order to enhance an improvement on axial compressors thermal efficiency. The numerical analysis was performed using CFX simulator. Also this study could help improve gas turbines performance, which implies lower costs of operation.

## 1. METHODOLOGY

The geometrical model consists on three pieces: Cooling-Tunnel, Rotor (one blade) and Stator (two blades). This combination was chosen so that the contact area between tunnel, rotor and stator coincide most. This model is presented on Fig. 2.

For the simulations, two fluids were taken in account (two-phase flow): wet air (variable composition mixture) and water. The former is the continuous phase and the latter the dispersed phase. It was considered Eulerian-Lagrangian non-homogeneous point of view, and particle model for water, since the objective of this work is mainly to study the hydraulics.

The well known transient governing equations for conservation of mass, momentum and energy were used. Also the heat interchanging between air and water was taken into account. Mass transfer was considered since the wet air is being saturated with water that proceeds from a fogging system where droplets were previously injected. The turbulence model for both phases was SST, since it combines the K- $\epsilon$  and K- $\omega$  models advantages.

### 1.1 Case studies

The scheme that corresponds to the simulations made are shown on Table 1.

Table 1. Case Studies

Inlet Water Temp.(K)	Drop Diameter ( $\mu\text{m}$ )	% Load		
		80	100	120
285	10	80	100	120
288	20	80	100	120
255	30	80	100	120
288	10	80	100	120
285	20	80	100	120
288	30	80	100	120

### 1.2 Flow Domain

The compressor considered was taken from an electric power plant located in Los Teques, Venezuela. The domain is shown on Fig.1. The complete flow system including the cooling-tunnel is presented on Fig.2.

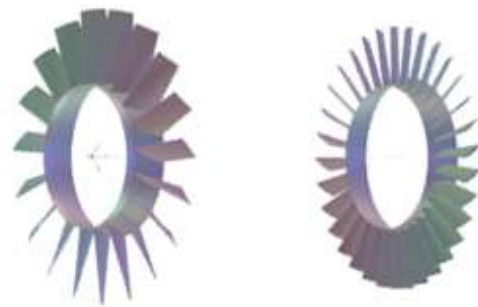


Figure 1. Rotor and Stator

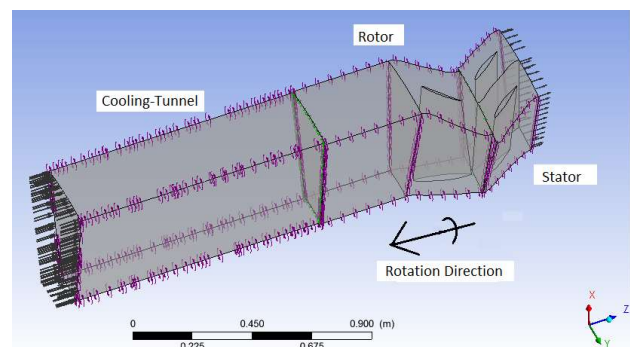


Figure 2. Complete Flow Domain

The entire compressor of the gas turbine is model W501D5, which possesses 19 rotor blades and 32 stator blades, and the nominal rotational velocity is 3500 R.P.M. The pressure ratio at 100% load for the entire compressor is and the entire mass flow is 337 Kg/s.

### 1.3 Numerical Approach and Convergence

The compressor performance feature conditions are shown in table 2.

For all cases, the flow was assumed to be at steady state, two-phase flow with water droplets injection. Turbulence effects were modeled using the SST turbulence model with standard wall functions and adiabatic wall boundary conditions. The walls were considered smooth. At solid walls, the no-slip boundary condition was imposed in the appropriate frame of reference. Furthermore, the interfaces between the tunnel, rotor and stator were set up as stage interface. (Rotor moves at 3500 RPM, while stator and cooling-tunnel are fixed).

For the discretization of the advection term, the Upwind was used. The convergence criteria were to look at the final boundary balances and the RMS values. The process was considered converged when the maximum residual error for any parameter was reduced to less than  $1E-04$ . With regard to the computational domain of the cooling tunnel-Compressor, it was possible to take advantage of the axisymmetry of the impeller, stator and cooling-tunnel. For that reason, only one blade for rotor and and two blades for stator were simulated for this turbomachine.

On the other hand, as in any CFD problem, a sensibility analysis was performed to guarantee the independence of results with respect to the grid. Some simulations were made changing the number of elements, calculating the power for each cases. The grids used were: 4,588,732 elements, then 5,965,352 elements. Following 7,754,957 elements and then finally 10,081,445 elements. A power change of less than 5 percent was accomplished.

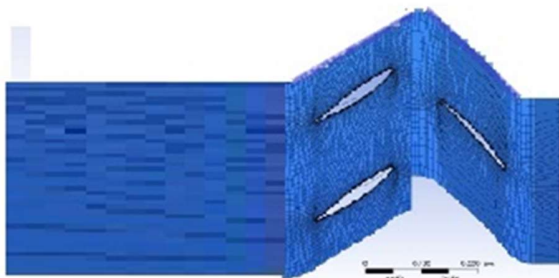


Figure 3. Final Mesh

The boundary conditions are indicated on table 2.

Table 2. Boundary Conditions

Diameter ( $\mu\text{m}$ )	Inlet			Outlet
	Water Temp. (K)	Air Total Temp.(K)	Air Total Press. (kPa)	%Load
10	285	302.27	107.75	80/100/ 120
20	285	305.37	111.37	80/100/ 120
30	285	309.14	115.77	80/100/ 120
10	288	302.27	107.75	80/100/ 120
20	288	305.37	111.37	80/100/ 120
30	288	309.14	115.77	80/100/ 120

The global system is conformed by three meshes: Rotor with one blade (Structured with 276,480 Hexahedron elements), Stator with two blades (Structured 467,200 Hexahedron elements) and cooling-tunnel (Unstructured 9,337,765 Tetrahedron and Prism elements). The 3-D mesh is presented on Fig. 4.

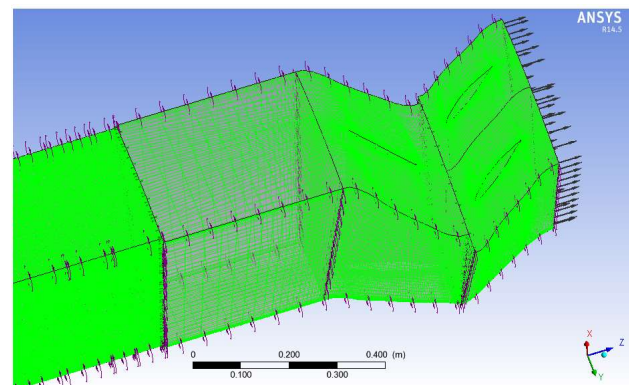


Figure 4. Complete Mesh

## 2. RESULTS AND DISCUSSION

The total number of runs was 21 (18 for water-injection and 3 for only air). The procedure matrix was already shown on table 1. It can be seen that the variables to evaluate are Inlet water temperature (285 K and 288 K), and the inlet water droplet diameter (10  $\mu\text{m}$ , 20  $\mu\text{m}$  and 30  $\mu\text{m}$ )

## 2.1 Validation

Regarding the validation of the model, some simulations were made considering only humid air without water droplets. For nominal point (100% load), the pressure ratio resulting from the simulation was 1.29, which is close to 1.33 (Pressure ratio at nominal performance). On the other hand, the comparison between CFD and experimental results are shown on Fig.5, showing good agreement. It can be noticed that only a few performance data is available.

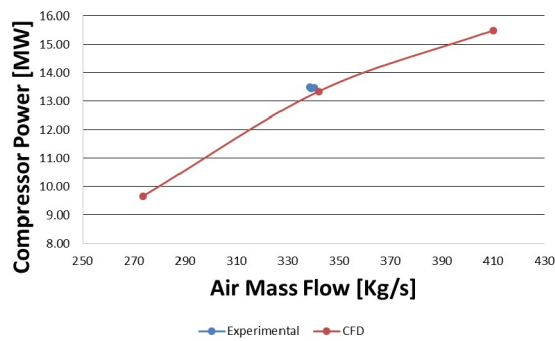


Figure 5. Complete Mesh

### 2.1.1 Efficiency

First, the comparison between the only air and the air-water simulations regarding efficiency can be seen from Figure 6 to 11. (Corresponding to 80%, 100% and 120% load).

It can be seen that in all cases the efficiency considering fogging is greater than for the only air case. Particularly the best increase is presented for 120% Load, 10  $\mu\text{m}$  water droplet, 285K. (Increase of 4.05%). It could be explained as follows: As droplet diameter diminishes, the evaporation is more effective since it takes less time to occur. Also, If inlet water temperature is less, the humid air will cool more easily.

On the other hand, the least efficiency increase for fogging corresponds to for 80% Load, 30  $\mu\text{m}$  water droplet, 288 K. (Increase of 0.21%).

Both water velocity and temperature profiles for those two cases can be observed on Fig.20 to Fig.23.

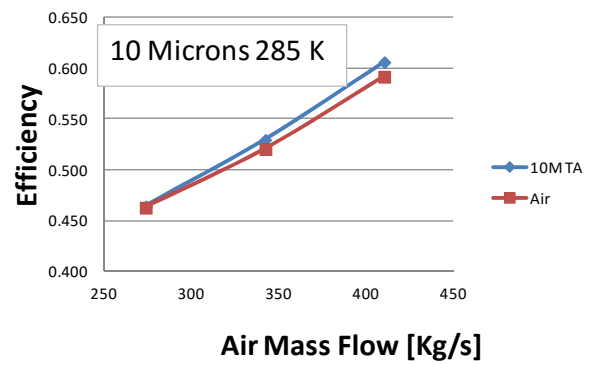


Figure 6. Efficiency for 10  $\mu\text{m}$  285 K Water Temperature

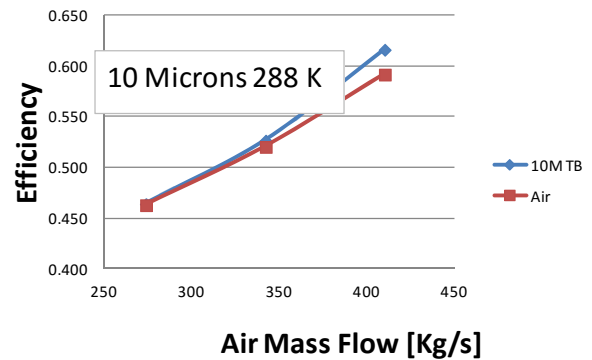


Figure 7. Efficiency for 10  $\mu\text{m}$  288 K Water Temperature

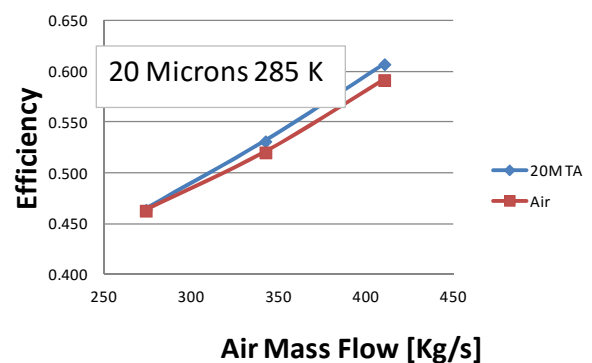


Figure 8. Efficiency for 20  $\mu\text{m}$  285 K Water Temperature

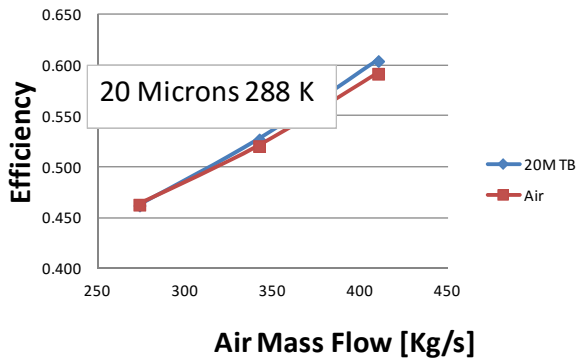


Figure 9. Efficiency for 20 μm 288 K Water Temperature

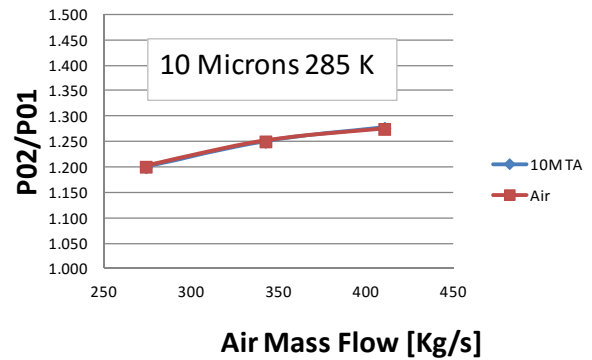


Figure 12. Total Pressure Ratio for 10 μm 285 K Water Temperature

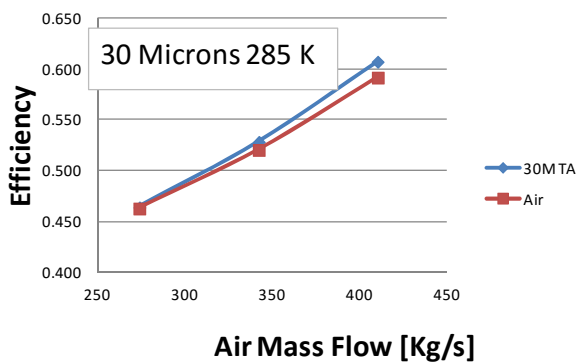


Figure 10. Efficiency for 30 μm 285 K Water Temperature

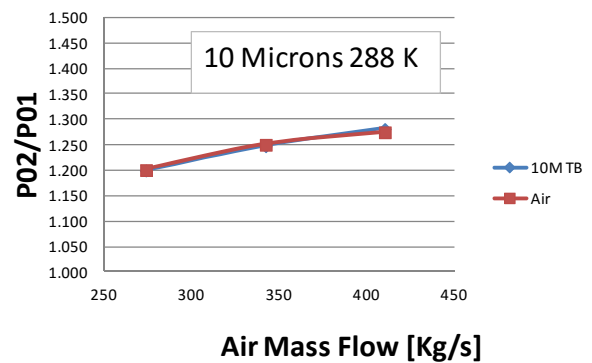


Figure 13. Total Pressure Ratio for 10 μm 288 K Water Temperature

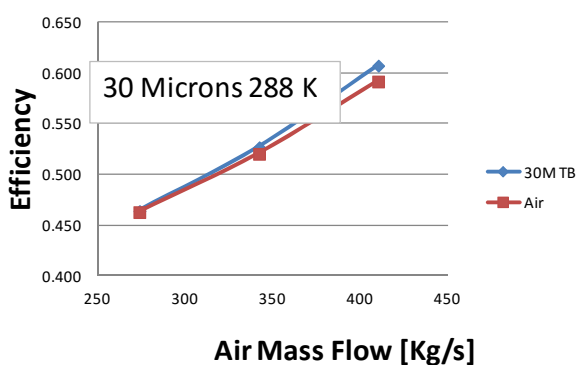


Figure 11. Efficiency for 30 μm 288 K Water Temperature

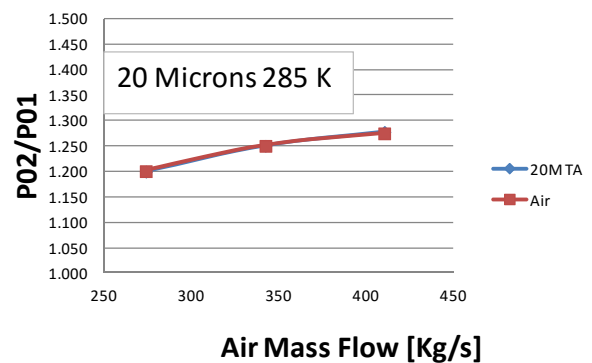


Figure 14. Total Pressure Ratio for 20 μm 285 K Water Temperature

### 2.1.2 Total Pressure Ratio

On the other hand, the total pressure ratio comparison is shown from Fig. 12 to Fig. 17. It can be noticed that the total pressure ratio behavior for water injection and only air cases is very similar. It indicates that the most important variable that influences the efficiency is the outlet temperature. (The effect of water droplets is more effective).

The consequence lies on the fact that the excess of temperature in the humid air flowing through the compressor is related with the increase in entropy, so that the efficiency increase because of fogging, is not as good as the case when the exit temperature is lower.

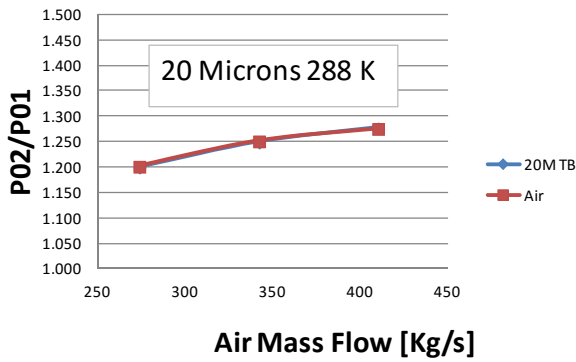


Figure 15. Total Pressure Ratio for 20 μm 288 K Water Temperature

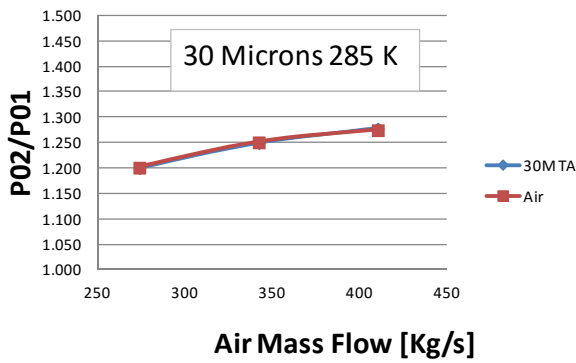


Figure 16. Total Pressure Ratio for 30 μm 285 K Water Temperature

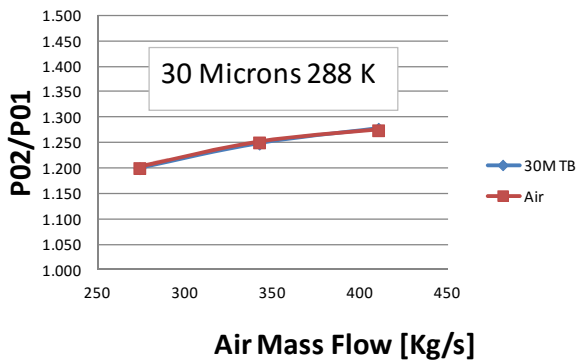


Figure 17. Total Pressure Ratio for 30 μm 288 K Water Temperature

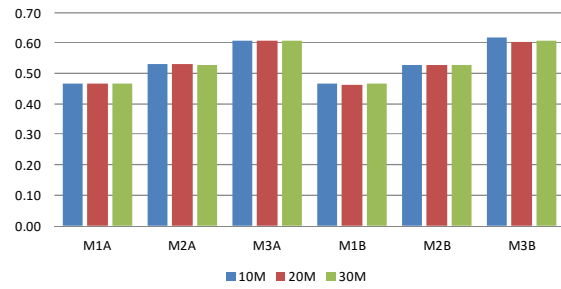


Figure 18. Efficiency Comparison

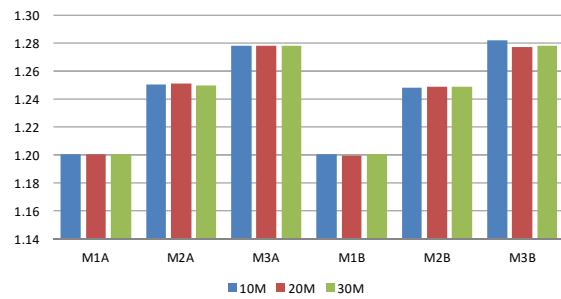


Figure 19. Total Pressure Ratio Comparison

### 2.1.3 Water hydraulics

As seen on the results regarding efficiency and pressure ratio, the best and worst efficiency corresponded to 10μm with 285K water temperature, and 30μm with 288K respectively. Let us see the for a moment the water droplets trajectory from the cooling-tunnel to the rotor and stator. (See Fig.20 and Fig.21). It can be seen that for the first case the water droplets practically enter the compressor rotor in the entire height. For the second case the water droplets enter the rotor at 75% of the height, and with less velocity relative to blades. It causes a better air-cooling effect for the former case, because the convection heat transfer coefficient is greater.

The comparison between each case for “fogging” technique can be observed for Fig.18 and Fig. 19.

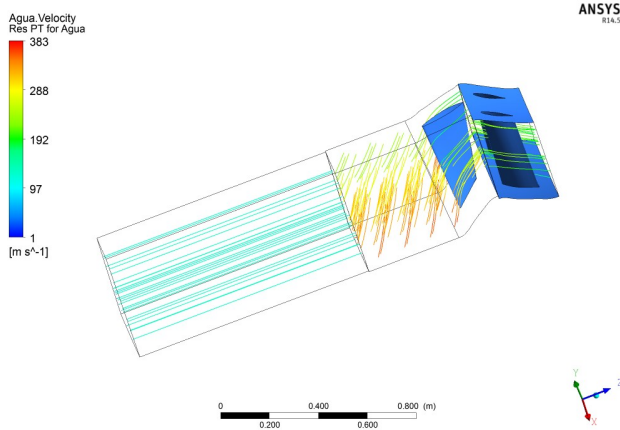


Figure 20. Water Stream Lines for 10 μm 285 K Water Temperature

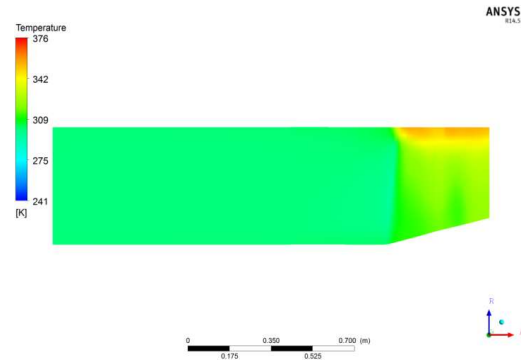


Figure 22. Temperatures for 10 μm 285 K Water Temperature

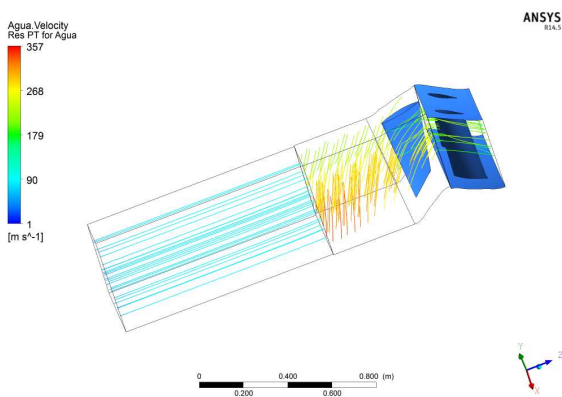


Figure 21. Water Stream Lines for 30 μm 288 K Water Temperature

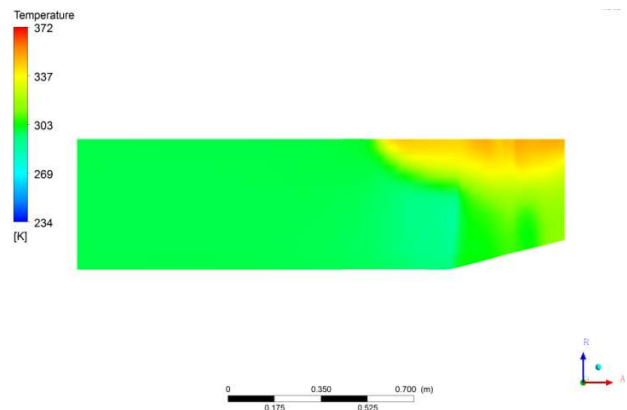


Figure 23. Temperatures for 30 μm 288 K Water Temperature

## 2.2 Temperature Profiles on Meridional Plane

The temperature profiles for the cases of best and worst efficiency, are shown on Fig. 22 and Fig.23, respectively . It can be noticed that for the 10μm with 285K case, the temperature of the air for the stage is less than for the case of 30μm with 288K case. It agrees with the numerical results of the efficiency and pressure ratio graphs already shown.

So, efficiency increase is better as air temperature diminishes more, because of the water droplets trajectory.

## CONCLUSIONS

- A system compound by a compressor from a gas turbine, considering impeller, stator and cooling-tunnel, using “fogging” technique was modeled.
- Grid and system validations were made. Four grids were used for the convergence index method. A few experimental performance data is available, mainly around the best efficiency point.
- For a range of performance, considering three water droplet diameters and two inlet water temperatures, several simulations were made.
- It was demonstrated that as both inlet water temperature and droplet size diminishes, the increase in efficiency is better.

- The maximum increase in the efficiency lies on the fact that the outlet temperature is lower.
- The efficiency considered is thermal, applied for this sort of turbomachines.

## ACKNOWLEDGMENTS

Special thanks are given to the Department of Conversion and Transport of Energy since it was possible to make the simulations on that location in the Simon Bolivar University.

## NOMENCLATURE

P01.....Stagnation Pressure at Compressor Inlet [KPa]  
 P02.....Stagnation Pressure at Compressor Outlet [KPa]  
 Mij.....Air Mass Flow [Kg/s]

### Subindex:

i.....Percentage of Load  
     1: 80%  
     2: 100%  
     3: 120%  
 j.....Inlet Droplets Water Temperature  
     A: 285 [K]  
     B: 288 [K]

## REFERENCES

- [1] J. Khan and T. Wang, "Three-Dimensional Modeling for Wet Compression in a Single Stage Compressor Including Liquid Particle Erosion Analysis," vol. 133, no. 012001, 2011.
- [2] L. Sun et al., "Understanding Effects of Wet Compression on Separated Flow Behavior in an Axial Compressor Stage Using CFD Analysis" Journal of Turbomachinery. vol. 133, No. 031026, 2011.
- [3] F. Jeanty et al., "Numerical Analysis of a Fogging System in a Gas Turbine", GT2012-68808, 2012
- [4] J. Perez, "Analysis of a Fogging System on Natural Gas Air Cooler using CFD Techniques" , Thesis, Simon Bolivar University, 2015
- [5] C. Moreno et al., " Hydraulic Behavior of Air and Water Droplets Flow on First Stages of an Axial Compressor ", ISROMAC15, 2014
- [6] C. R. Cortes and D. E. Willems, "Gas Turbine Inlet Air Cooling Techniques: An Overview of Current Technologies," in POWER-GEN, Las Vegas, Nevada, 2003.
- [7] S. Sanaye, H. Rezazadeh, M. Aghazeynali, M. Samadi, D. Mehranian, M K. Ahagaran, "Effects of Inlet Fogging and Wet Compression on Gas Turbine Performance" ASME Turbo Expo 2006: Power for Land, Sea and Air. Paper No. GT2006-90719.
- [8] A.S Moning and A. M. Yaglom, "Statistical Fluid Mechanics. Mechanics of Turbulence" , Dover, USA, 2007
- [9] A. Lecuona and J. Nogueira, "Turbomaquinas. Procesos, Analisis y Tecnologia" , Ariel, Spain, 2000
- [10] Royce Brown, "Compressors. Selection and Sizing" , Gulf Publishing, USA, 1998
- [11] ANSYS, "ANSYS CFX Introducción" , Albatros-ESSS, USA, 2011