Hydraulic Effect of Water Droplets on a System compound by Cooling Tunnel and First Stage of an Axial Compressor

Carlos Luis Moreno, Department of Thermodynamics and Transfer Phenomena., Simon Bolivar University. G-10 Caracas, Venezuela.
Carlos Pacheco, Department of Conversion and Transport of Energy., Simon Bolivar University. Caracas, Venezuela
Miguel Asuaje, Department of Conversion and Transport of Energy., Simon Bolivar University. Caracas, Venezuela

Long Abstract

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 compresor. 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 FLUENT. 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 comprobed the importance of the fogging technique, prior to the first stage of fogging systems.

Later, Perez et al. [4], analized 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. Methods

The simulations of this study are performed using three basic geometries silmutaneously : cooling-tunnel, compressor rotor and stator (First stage). In order to construct the entire geometry, the number of control volumes for rotor and stator on first stage, corresponded to the minimum number, when using the periodic surfaces. The focus of the water-air two-phase flow was Euler-Lagrange. An RMS of 10^{-4} was used. The variables to be considered are mass flow rates, water injection rates and droplet sizes, mainly. Steady-state was considered. For the results of the simulations, blade-blade planes were made in order to analize the behavior of water droplets, residence time and dry-bulb temperatures. Also 3-D streamlines were represented. Then the compressor thermal efficiency was calculated, and the results show that such efficiency depends on those factors. It influences the gas turbine efficiency. The compressor rotor and stator in question are presented as follows.

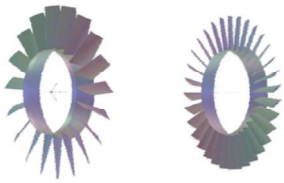

Figure 1. First Stage Rotor and Stator of Compressor

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

,

.