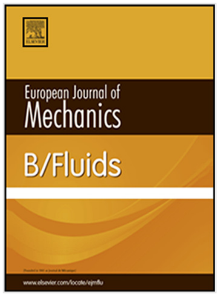
Accepted Manuscript



Investigation and modeling of two phase flow through a compressor stage: Analysis of film breakup

Niklas Neupert, Hassan Gomaa, Franz Joos, Bernhard Weigand

PII: S0997-7546(16)30381-8
DOI: http://dx.doi.org/10.1016/j.euromechflu.2016.09.007
Reference: EJMFLU 3056

To appear in: *European Journal of Mechanics B/Fluids*

Investigation and Modeling of Two Phase Flow through a Compressor Stage: Analysis of Film Breakup

Niklas Neupert[1]*, Hassan Gomaa, Franz Joos[1], Bernhard Weigand[2]

ISROMAC 2016

International Symposium on Transport Phenomena and Dynamics of Rotating Machinery

Hawaii, Honolulu

April 10-15, 2016

Abstract

For stationary gas turbines, the injection of water sprays into the compressor inlet is used to increase the power output, which can be used for rapid correction of power fluctuations appearing in power supply systems with today's increased use of wind and solar energy. However, to fully utilize this technique a comprehensive understanding of the underlying phenomena is needed. An inhouse CFD program was used at Institute of Aerospace Thermodynamics (ITLR) Stuttgart modeling a variety of two-phase phenomena which was validated on the transonic wind tunnel at Helmut Schmidt University (HSU) Hamburg. The present paper focuses on the water film on the blades formed by water deposition of impinged droplets. Experimental results using a reflective shadowgraphy method reveal the wall film pattern on the blade without disturbing the flow. A numerical model is presented predicting the position of film breakup based on the superposition of a small perturbation of the film height. The onset of film breakup is predicted to be at the position in which the perturbation is not dampened by the acting forces. The film breakup position is analyzed experimentally and numerically for three different water loads. The experimental analysis shows that the onset of film breakup is almost independent of the water load. The numerical model shows the same trends and indicates a film breakup at the same position.

Keywords

Two Phase Flow — Overspray Fogging — Experimental Investigation— Numerical Simulation

[1]*Laboratory of Turbomachinery (LSM), Helmut-Schmidt-University, University of the Federal Armed Forces, Hamburg, Germany*

[2]*Institute of Aerospace Thermodynamics, Stuttgart University, Stuttgart, Germany*

***Corresponding author**: Neupert@hsu-hh.de

INTRODUCTION

The increasing share of renewables in the electric energy producing sector increases the fluctuation of the power supply in the grid. To compensate this the requirements for conventional power plants move towards fast additional power and higher flexibilities [1]. The injection of a droplet spray into the inlet of the gas turbine, called inlet fogging, has shown to reduce compression work and increase mass flow by reducing gas turbine inlet temperature. This effect can be increased by injecting a higher fraction of water which is taken into the compressor and reducing the temperature rise throughout a compressor stage [2]. For instance this increase in flexibility helped to fulfill the stringent requirements of the UK grid code by providing an additional extra output of 18 MW for the Marchwood power plant within a short period of time [3]. Small droplets in the range of $\leq 10\mu m$ are favorable due to a better follow-up behavior and higher ratio of surface to volume thus improved evaporation behavior. Despite the very low average diameter of the spray normally produced by overspray fogging systems (mean diameter $D_{10} \leq 10\mu m$ larger droplets in the range of $50 - 100\mu m$ can evolve from droplet droplet interaction and the entrainment of previously deposited water [4]. These larger droplets show a significantly larger characteristic time τ_c and thereby a longer duration to adapt a change in motion. This results in interactions with the structure leading to erosion problems and the formation of small secondary droplets caused by splashing. Depending on the impact conditions a significant amount of water is deposited on the blades leading to the formation of a wall film and streaks which are driven towards the trailing edge. The contribution of water on the blade and shape of the water air interphase has been shown to influence the overlaying air flow [5]. The presence of water increases the friction coefficient of the surface [6] suggesting a positive effect for smaller fraction of water covered surface, e.g. rivulets instead of a continuous film. In contrary the maximum height of rivulets is larger compared to the film due to continuity reasons. This would lead to the conclusion that depending on the boundary layer thickness of the air flow either a continuous film or discrete rivulets will have a stronger influence on the surrounding airflow [5]. In any case a prediction of the water flow pattern on the surface is detrimental for the comprehensive description of a two phase flow in a compressor. This aspect of a two phase flow problem in a compressor is addressed in this paper.

A direct measurement of the wall film properties in a gas turbine compressor under overspray fogging conditions is extremely difficult or even impossible. This necessitates the use of numerical tools for the investigation of the problem.