Influence of jets to wall distance on the heat transfer distribution at high flow velocity

MARCIN KUROWSKI[1*], RYSZARD SZWABA[1], JANUSZ TELEGA[1], PAWEL FLASZYNSKI[2], FERNANDO TEJERO[2]

[1] *Experimental Aerodynamic Department, IMP PAN, Fiszera 14, Gdansk, 80-231, Poland*
marcin.kurowski@imp.gda.pl

[2] *Aerodynamic Department, IMP PAN, Fiszera 14, Gdansk, 80-231, Poland*
pawel.flaszynski@imp.gda.pl

Introduction

This paper presents the results of experimental and numerical research on heat transfer distribution under the impinging jets at various distance to the wall and high jet velocity. The air jets flow out from the common pipe and impinge on a surface which is cooled by them, in this way all together create a model of external cooling system of low pressure gas turbine casing. Heat transfer is a critical issue in the development of aircraft engines. The external cooling systems of low pressure blades in modern aero engines are very complex and essentially rely on a series of circumferential feeding pipes. They have to be designed for high efficiency, long life cycles and safe operation in spite of being exposed to high aerothermal loadings of the engine. In such systems, impinging jets are directed towards the external turbine casing with the final aim of keeping the clearance between blade tip and casing as constant as possible under different engine operating conditions. The optimization of a low pressure (LP) turbine performance needs to be able to minimize the clearances between static and rotating parts, in order to avoid the by-pass flows around turbine blades and vanes. A typical method for controlling those clearances is to actively control the engine case temperature, and then the static parts radial displacement, during the different flight conditions. Impinging air jets is a commonly used technique to cool advanced turbines elements as it produces large convection enhancing the local heat transfer. Impinging is a complex phenomenon depending on many parameters: Reynolds number, flow velocity, nozzle to plate spacing, radial position from stagnation point.

The view of the measurement test stand is shown in Fig. 1. This test section is a model of turbine cooling casing system. Downstream of the inlet a pipe with a row of 26 jets is located. The jets outlet is connected to the vacuum tanks throughout the control valve. The test stand, apart outlet pressure control, has a possibility to control independently the inlet pressure. A camera for temperature measurement of the test wall was located beneath the test section. Measurements were carried out for the arrangement of twenty six in-line jets with orifice diameter of 0.9 mm and the distance between jets $l \approx 6d$. The outlet pressure correspond to aircraft cruise condition and was set to Pin=0.21 bar of absolute value. The mass flow was controlled to obtain two different Reynolds number (based on jet dia.), namely Re=2500 and 4000. Such flow conditions implies that isentropic Mach number at the jets exit was equal M=0.56 and 0.77 respectively. To investigate the heat transfer, liquid crystal (LC) sheet for temperature distribution measurements were placed on the surface under the jets, on the bottom wall of the test section.

In Fig. 2, the red dashed lines indicate 26 jets, yellow solid lines mark the centre of the jets row and region expanding 15 mm form the jets. The 15 mm distance was chosen as it is enough to embrace the region of significant activity of the impinging jet, and still sufficiently distant from the power supply buses, whose influence we neglect. The axial distance from extreme jets has been selected as twice the distance between the jets, which yields 10 mm. The temperature distribution field is calculated from the data like depicted in Fig. 2, by using the calibration curves of the liquid crystal.

A MATLAB code for determining the temperature from the LC colour, exploiting these curves, has been developed. Further, when the temperature field during the experiment is found, the Heat Transfer Coefficient is calculated.

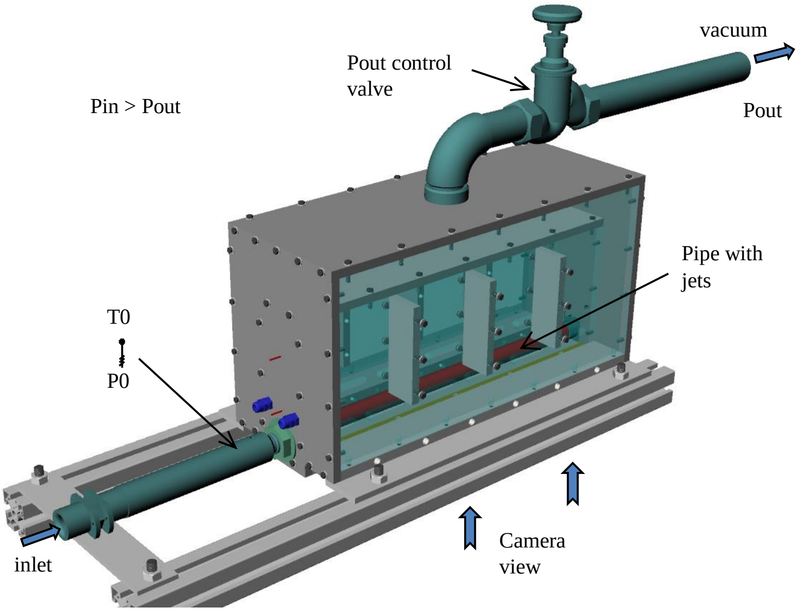

Fig. 1. View of the test stand.

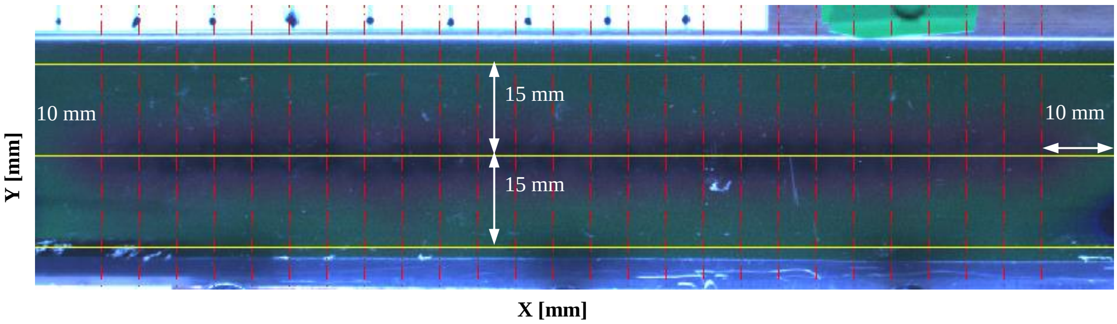

Fig. 2. Definition of the interest region.

Apart from the experimental investigations, numerical simulations were also carried out. Computational domain is created according to the test section configuration, but it includes one nozzle (one jet) only. Such simplification arises from assumption that flow conditions for each nozzle (jet) is the same.

The influence of various Reynolds and Mach numbers on cooled flat plate and jets to wall distance in terms of heat transfer effectiveness is presented in this paper. Experimental results were used for the Computational Fluid Dynamics (CFD) model development, validation and comparison with numerical results is presented as well.

This work is a part of the INNOLOT Program financed by National Centre for Research and Development.