Propulsion and Power Research 2014;3(2):82–89

http://ppr.buaa.edu.cn/

Propulsion and Power Research

www.sciencedirect.com

ORIGINAL ARTICLE

Effect of surface roughness on the aerodynamic performance of turbine blade cascade

Tao Bai*, Jingyuan Liu, Weihao Zhang, Zhengping Zou

National Key Laboratory of Science and Technology on Aero-Engine Aero-Thermodynamics, 2011 collaborated innovation center of advanced aero-engine, School of Energy and Power Engineering, Beihang University, Beijing 100191, China

Received 29 September 2013; accepted 24 April 2014
Available online 10 July 2014

KEYWORDS

Turbine blade;
Surface roughness;
Boundary layer;
Reynolds number;
Aerodynamic loss

Abstract The effect of surface roughness on the boundary development and loss behavior of turbine blades is investigated with different Reynolds numbers in this paper. The result shows that the velocity profile in boundary layer is plumper on rough surface than on smooth blade. The aerodynamic loss is lowered at low Reynolds number, but becomes significantly large at high Reynolds number. The total pressure loss coefficient of cascade can reach a top increase of 129% for rougher blades comparing with smooth blades at $Re=300000$.

1. Introduction

Surface roughness increases significantly due to erosion, corrosion, and deposition during operation under high pressure and temperature condition [1] and even exists on new made blades. After several thousands of in-service hours, roughness heights would typically drop into the range of 20–150 μm centerline on average [2]. The efficiency of turbines will be seriously affected by surface roughness. Bammert and Sandstede's [3] measurement indicated that sand grain surface roughness on turbine blade with K_s/C ranging from 10^{-3} to 10^{-2} could decrease the stage efficiency by 7 to 14 percent comparing with smooth blades. Boynton et al. [4] found that decreasing the surface roughness from 10.16 μm to 0.76 μm would cause a 2.5 percent increase on the efficiency of a high pressure fuel turbopump for a rocket engine. Yun et al. [5] found that in the fully rough regime (400 μm), normalized efficiency decreased by 11% with roughness only on stator vanes, 8% with roughness only on rotor blades, and 19% with roughness on both the stator and rotor blades.

Surface roughness results in aerodynamic loss through its interaction with the boundary layer. The results of

*Corresponding author.

E-mail address: 921036859@qq.com (Tao Bai).

Peer review under responsibility of National Laboratory for Aeronautics and Astronautics, China.

Production and hosting by Elsevier

2212-540X
http://dx.doi.org/10.1016/j.jppr.2014.05.001

Nomenclature

K_s	equivalent sand grain roughness
R_a	centerline average roughness
S	dimensionless surface length
V_s	free stream velocity
H	shape factor
y/y_s	dimensionless distance from wall
V/V_s	dimensionless velocity
C_f	friction coefficient
Y_p	total pressure loss coefficient
C_d	dissipation coefficient
Re	Reynolds number

Subscripts

f	fluid domain
in	inlet
s	solid domain
w	wall

Boyle et al's [6,7] experiment showed that the effect of roughness on turbines is closely related to Reynolds number. Denton's [8] research indicated that low roughness has no effect on boundary layer when Reynolds number is low, while the aerodynamic loss becomes prominent at high Reynolds number. M. Montis et al. [9,10] found that the boundary layer upstream of the separation point will become thinner with the presence of roughness on blade surface, so the loss becomes significant at high Reynolds number. Q. Zhang et al. [11–13] analyzed the effect of surface roughness and turbulence intensity on aerodynamic losses. Results showed that the effects of changing the surface roughness condition on IAL values are substantial, whereas the effects of different inlet turbulence intensity levels are relatively small in general. Roughness height and roughness distribution on blade are clearly important. Suder et al. [14] found the roughness distribution on blade leading edge and the front half of the suction surface contributed to most of the aerodynamic loss. Roughness on these regions accounted for more than 70% of the performance degradation found in fully coated blade. Kind et al. [15] showed that roughness on the suction surface could cause large increase in profile loss while roughness on the pressure surface had relatively small effect.

In previous foreign studies, it has been demonstrated that the impact of roughness on the turbine cannot be ignored. Some related research of surface roughness also has been done in domestic. J. Yao et al. [16] in Tsinghua University found that the roughness effect on off-design incidence is highly significant, but different Reynolds numbers are of no sensitive. J. Wen's [17] study indicated that unsmooth blades will change the development of vortex and lower the aerodynamic loss comparing with smooth ones. However, in domestic journals, there is limited work dealing with the effect of surface roughness on boundary layer development.

In this article, simulation methods are validated at first. The boundary layer's development at different Reynolds numbers with different roughness height is analyzed in detail in the following parts.

2. Object and numerical methods

The simulation is carried out on the two-dimension cascade. The blade geometry parameters are listed in Table 1. The size of roughness (K_s) researched are 26 μm, 53 μm, 110 μm designated as SL_1, SL_2, SL_3. K_s and K are two parameters which describe the roughness on wall surface. K is defined as the actual roughness geometry, while K_s is a modeling parameter, defined to characterize and quantify the roughness on the surface. The quantity of K_s represents the size of sand grains which give the same skin friction coefficients in internal passages as the roughness being evaluated.

Three-dimensional steady viscous Reynolds-averaged Navier-Stokes (N-S) equations are solved to simulate the flow in the turbine cascade using the commercial computational fluid dynamics (CFD) software CFX. The flow region is discrete using a finite volume method and convection terms are analyzed using a second-order accurate upwind scheme. The flow time is discrete using second-order rear different Euler format. Shear stress transport (SST) two-equation turbulence model, and $\gamma-\theta$ transition model are used throughout the whole work. The total temperature, total pressure and flow angle are specified as inflow boundary condition and the outlet static pressure is specified as outflow condition in this investigation. K_s is used to correct the wall function in the simulation, and the function of sand grain height K is pointed out to correct the transition equation to consider the roughness effect on the transition process. The outlet Mach number is 0.75.

Sixty-four thousand grid cells in computational domain are adopted. The maximum wall normal distance of the first cell center to the blade surface is less than 1, and the grid extension near the wall is less than 1.2. The computation mesh is illustrated as Figure 1.

Table 1 Cascade parameter.

Parameters	Value
Axial chord/mm	17.3
Blade pitch/mm	18.32
Leading edge radio/mm	0.63
Trailing edge radio/mm	0.2
Inlet metal angel/(°)	4.78
Outlet metal angel/(°)	67.96
Flow turning angel/(°)	72.74