Application of numerical modelling for determining the optimal position of the suction slot in the steam turbine vane cascade

V G Gribin 1, A A Tishchenko 1, R A Alekseev 1, I Yu Gavrilov 1, V V Popov 1 and V A Tishchenko 1

1 National Research University "MPEI", Russia, 111250 Moscow, Krasnokazarmennaya, 14
GavrilovIY@mpei.ru

Abstract. The numerical investigations of liquid film development in the last stages of steam turbines were performed. This data was used to suggest the position of the suction slot used to evacuate the liquid phase from the turbine flow path. The object of study is a periphery 2D section of vanes cascade. The movement of coarse droplets in the blade passage was simulated. Their collision with the blade surfaces results in the appearance of additional sources of mass and momentum for liquid film. The influence of theoretical Mach number on the features of liquid film formation was studied. The distributions of main characteristics, reflecting the formation of the film, have been obtained. They were used to analyse the liquid phase flow along the surfaces and to point out base recommendations about the location of the suction slot.

1. Introduction
Erosion wear of the rotor blades – one of the main problems should be solved during the design process of the last stages of the steam turbines. Coarse droplets moving as discrete particles in a wet steam flow are involved in this phenomenon. They are generated in the turbine flow path during the separation and breakup of liquid films. Nowadays the interchannel suction systems are used to evacuate the liquid film from the vane blades surfaces. The question about the suction slots effectiveness is very important. Besides the geometry features and operation conditions, it is important to estimate the optimal position of the suction slot on the blade surface. Comprehensive studies on the experimental and real-size turbines such as [1, 2] allow analyzing the features of wet steam flow in the last stages. As a result, currently, we have a theoretical understanding of liquid film development in the turbine flow path. Based on this knowledge, numerical models of the liquid film formation have been developed [3, 4].

In the current paper, we present the study of the liquid film formation process in the passage of the vane cascade. For this purpose, the thin liquid film numerical model was used [4, 5]. This computational method was verified by the experimental investigations in the test rig. The main goal of this work is to determine the position of a suction slot which provides the best abilities of the liquid film evacuation from the turbine flow path.

2. The object of study and numerical method
The study was performed for the flat vane cascade presented in figure 1(a). The design of the blade profile was developed according to the data of liquid particles movement in a wet steam flow [5]. The
main feature of the design – effective accumulation of a liquid film (with minimized secondary droplets generation) on the surfaces for its further suction through the slot. The numerical method described in [4] used to get the data about the liquid film parameters distribution along the blade surfaces. This technique utilizes the one-way coupling mechanism of modelling the continues steam flow and discrete coarse liquid droplets movement. The CFD results of wet steam flow (the media of gaseous steam and a “fog” of fine liquid droplets) in the blade passage used in calculating the liquid droplets trajectories and their interaction with the blade solid surfaces. So, this method does not consider the effect of coarse droplets movement on the continuous phase. As a result of the liquid particle impact on a surface, the part of droplet mass leaves the surface as secondary droplets, another one is involved in the liquid film formation. Momentum and mass conservation equations are solved for liquid film flow on the curvilinear surfaces. The complex process of liquid film separation, breakup and coarse droplets formation is taken into account by the implementation of empirical separation model. The described method validated in the experimental conditions on the flat turbine casades [4, 6].

Figure 1. The geometry of studied vane cascade (a) and the instantaneous structure of droplets streams in the cascade passage (b).

Figure 1(b) presents the instantaneous structure of the discrete phase in the blade passage and downstream the cascade during the modelling process. Green points – primary droplets came from the inlet of the computational domain; red points – secondary droplets generated after the impact of a primary droplet on the surface; blue points – droplets formed after the liquid film separation and breakup.

The described technique used to model the liquid film formation on the vanes at different flow regimes. The studies performed for 3 values of theoretical Mach number downstream the cascade: 0.76, 0.95 and 1.06. The inlet boundary conditions correspond to the upstream parameters of the steam turbine last stage. They were kept constant for each case. The studied 2d section of the vanes cascade (see figure 1(a)) located on the periphery of the blades (see figure 2(a)) where the mass concentration of coarse droplets is maximum due to the upstream rotor blades centrifugal effect [7]. The inlet boundary conditions in the current study for all cases: initial wetness $y_0 = 4\%$, total pressure $p_0 = 28$ kPa, coarse droplets mass fraction: 50% (relative to the total mass fraction of the liquid phase). The primary coarse droplet size spectra are presented in figure 2(b). This distribution corresponds to results obtained by measurements on the full-scale steam turbine [8]. The droplet’s inlet velocity was set as a linear function according to the diameter – their slip coefficients ($v = \frac{c_d}{c_s}$, where $c_d$ – the droplet velocity and $c_s$ – the main flow velocity) distribution is presented in figure 2(b).
3. Results and discussion

The results of liquid film numerical simulation are shown in figure 3(a) and figure 3(b), where the distributions of thickness and velocity on the blade surfaces are pointed out. The x-axis corresponds to the passage’s nondimensional axial coordinate. It should be noted that the behaviour of liquid film development is unsteady. Here the time-averaged results are presented. One can see that the film development scenario differs depending on the blade surface where it located. On the suction side, for each regime, the liquid film separates in the area near the leading edge (at $C_x = 0.2$, where $h = 0$ and $u = 0$). In this region, film conditions and blade geometry properties meet the separation criteria – a ratio of the inertial forces to the surface tension on the convex wall [9]. As a result, after the breakup process, the liquid film forms the droplets moving along the suction side. Following the blade passage geometry, the rest suction side surface located in the aerodynamic shadow for the drops (see figure 1(b) Zone I) [6]. So, the intensity of the droplets impact here is low and there is no source of mass and momentum for film development. Other processes are observed on the pressure side. This surface is prone to the primary droplets impact along its entire length (see figure 1(b) Zone II). As a result, the intensive growth of the liquid film flow rate is observed. As one can see from figure 3(a), the film thickness increases up to axial coordinate $C_x = 0.6$ for each considered regime. At the same time, the average velocity of the liquid film on the pressure side curve section $C_x = 0 – 0.6$ increases slowly (see figure 3(b)). So, the mass flow rate of the liquid phase on this site of the wall increases due to the growth of the film thickness. This effect is imposed by the features of the main (steam) flow. As one can see from Figure 3(c), the isentropic Mach number along the pressure side increases slowly in the range of $C_x = 0.05 – 0.60$. In this zone flow conditions match for each considered regimes – acceleration tends to zero (pressure gradient tends to zero) in a couple with relatively low values of the main flow velocity (shear stress between the main flow and liquid film is low). At $C_x > 0.6$ the main flow acceleration is observed (see figure 3(c)), as a result, increases the liquid film average velocity and the ratio of its growth. At this range of $C_x$ with the growth of theoretical Mach number downstream the vanes row ($M_{f1}$) the film velocity increases. It leads to decreasing of the liquid film thickness (see figure 3(a)). Near the blade trailing edge ($C_x = 1.0$) the film velocity decreases dramatically, and the local extremum of thickness is observed (with further dramatical decrease). At this position the process of liquid film separation takes place. The results of numerical simulations have shown that this is an unsteady phenomenon. It consists of several stages – accumulation of liquid mass and it’s further separation.
Figure 3. Liquid film thickness distribution (a), liquid film velocity distribution (b) and isoentropic Mach number distribution (c) along the studied blade profile surfaces.

As was mentioned above, the mass flow rate of the liquid film increases along the entire pressure side surface. This is seen from figure 4(a), where the ratio between the primary droplet's mass interacted with the surface at current \( \bar{C_x} \) (\( m_1 \)) and mass of interacted droplets with the entire surface is presented (\( m_2 \)). The extremum of droplets impact intensity on the pressure side is situated at \( \bar{C_x} = 0.7 \). In the area of this position several coarse droplet streams impact on the blade surface. Besides the primary liquid particles from the inlet, secondary droplets, originating from the blade leading edge (see Zone III in figure 1(b)), interact with the pressure side here. This “fountain” of secondary drops has a significant effect on the liquid film development process. As mentioned in [1], it is important to provide the suction of the liquid film downstream the impact zone of this stream. It is important to note that after the interaction with the blade wall, the part of droplet mass leaves the surface in the form of secondary liquid particles. Figure 4(b) shows the distribution of ratio between generated secondary droplets mass (\( m_2 \)) and impacted primary droplets mass (\( m_1 \)). As one can see, over most of the length of the blade pressure side, less than 50% of impacted droplets mass is involved in the film development. Figure 4(c) represents the Sauter mean diameter distribution of the droplets impacting on the surface. The behaviour of the obtained curves reflects the features of flow-particles interaction. Different forces (drag, virtual mass, etc) acting on the droplet from the side of the steam flow cause the change of particle velocity vector and trajectory. So, according to the droplet movement equation (Newton’s second low) [6], with the increase of liquid particle diameter, its trajectory becomes straight and deviates from the steam flow pathline, while decreasing the droplet size leads to the approach of its direction to the main flow pathlines. As a result, the mean droplet diameter decreases with the increase of axial coordinate \( \bar{C_x} = 0.6 - 0.8 \) (see figure 4(c)) because of interaction between main flow and liquid particles. But in the range of \( \bar{C_x} = 0.6 - 0.8 \), where the impact of secondary droplets from “fountain” (see Zone III in figure 1(b)) takes place, the increase of droplets mean diameters been observed. As was mentioned above, these drops are generated due to the interaction of primary particles with the leading edge. Generally, as one can see from figure 4, the considered wet steam flow regimes don’t affect the features of liquid droplets interaction with the surfaces. This is due to the high inertia of droplets with considered sizes. The mechanism of particles impact on the blade surface with varying other parameters (attack angle and total pressure) is considered in [10].
Figure 4. Primary droplets impact on the pressure side mass ratio (a), secondary droplets mass generation ratio on the pressure side (b) and impact primary droplets Sauter mean diameter distribution (c).

4. Optimal suction slot position estimation

The obtained numerical results may be used in choosing the optimal position of the liquid film suction slot. For the considered regimes, the blade pressure side becomes the object of interest. Two main features of the liquid film development should be pointed out: the mass flow rate and average velocity of the liquid film increase during the entire length of the surface; secondary droplets, forming the “fountain”, have a significant effect on the film development process. As one can note – the best position of the suction slot is close to the trailing edge. In this case, the most amount of liquid film can be evacuated from the blade passage. The range of possible positions of the suction slot is limited by the technical possibilities – it is impossible to arrange the slot close to the trailing edge due to the small thickness of the blade rigid body. The geometry of the studied profile was developed in accordance to provide the position of “fountain”, impacting on the pressure side surface as far from the trailing edge as possible [5]. So, according to described demands the position $C_s = 0.76$ for the current study was chosen. As one can see, the slot is located downstream of the extremum which corresponds to the “fountain” mass source (see figure 4(a)). The vane blade sector with the suction slot and its chamber is presented in figure 5(a).

Figure 5(b) shows the comparison of the liquid film mass flow rate per blade unit length at the suction slot section ($g_{slot}$) and near the trailing edge ($g_{max}$). As one can see, the possible removed from the blade passage amount of water ($g_{slot}$) 2 times lower than the maximum mass flow rate, which is
achieved near the trailing edge. The zone of the pressure surface near the trailing edge interacts with relatively small droplets (see figure 4(c) $C_x > 0.8$). As a result, their collision with the wall proceeds at low impact energy. It leads to the reduction of secondary droplets generation process (see figure 4(b) $C_x > 0.8$), while the liquid film mass flow rate growth intensity increases.

5. Conclusions
In this paper, the numerical investigations of liquid film development were performed for different theoretical Mach numbers. The results of modelling have shown that the suction surface is not involved in the active film formation, while the pressure side is impacted by the primary droplets along the entire length. As a result, the mass flow rate of the liquid film on this surface increases from the leading to the trailing edge. Film development process along the surface is distributed uniformly. It is connected as with the difference of conditions of primary droplets impacting on the wall, as with the stream of secondary droplets originated from the leading edge (“fountain”) and colliding with the surface too. According to specified features, the position of the suction slot should be located as close to the trailing edge as possible, but also it should be placed downstream the zone, where “fountain” impacts the surface. In this study presented the method, providing the estimation of the optimum position of the suction slot for certain operating conditions.

Acknowledgments
This study was supported by Russian Science Foundation (project No. 16-19-10484).

References

[1] Hoznedl M, Kolovratník M, Bartoš O, Sedláč K, Kalista R and Mrózek L 2018 Proc IMechE Part A: J. Power and Energy 232(5) 515-524
[2] Hoznedl M, Kolovratník M, Tajč L, Weiß A and Mrózek L 2018 Proceedings of ASME Turbo Expo (Oslo) GT2018-75049
[3] Simon A, Marcelet M, Hérard J, Dorey J and Lance M 2016 Proceedings of ASME Turbo Expo (Seoul) GT2016-56148.
[4] Tishchenko V A and Alekseev R A 2019 Thermal Engineering 66(11) 830-838
[5] Alekseev R A, Gribin V G, Tishchenko A A, Gavrilov I Yu, Tishchenko V A and Popov V V 2019 Journal of Physics: Conference Series 1359(1) 012038
[6] Tishchenko V A, Alekseev R A and Gavrilov I Yu 2018 Thermal Engineering 65(12) 885-892
[7] Young J B, Yau K K and Walters P T 1988 Journal of Turbomachinery 110(2) 163-172
[8] Cai X, Ma L, Tian C, Li J, Ning D and Xu W 2012 Proceedings of Baumann Wet Steam Centenary Conference (Cambridge) BCC-2012-16
[9] Friedrich M A, Lan H, Wegener J L, Drallmeier J A and Armaly B F 2008 Journal of Fluids Engineering 130(5) 051301
[10] Tishchenko V A and Alekseev R A 2018 Journal of Physics: Conference Series 1128(1) 012137