STUDY ON WATER EXTRACTION OF HOLLOW STATIONARY BLADE UNDER WET STEAM FLOW CONDITIONS

Wu Xiaoming*, Yang Jiandao**, Li Liang***

* Shanghai Turbine Works Co., Ltd., Shanghai, China, wuxm3@shanghai-electric.com
** Shanghai Turbine Works Co., Ltd., Shanghai, China, yangjd@shanghai-electric.com
*** Institute of Turbomachinery, Xi’an Jiaotong University, Xi’an, China, liliang@mail.xjtu.edu.cn

ABSTRACT

Numerical simulation and experimental investigation on water extraction of hollow stationary blade are presented under wet steam flow conditions. The deposition of fog and coarse droplets on a last-stage stationary blade is investigated based on the numerical simulations of multistage non-equilibrium condensation flow in a nuclear low-pressure cylinder. It indicates that the majority of depositional droplets are deposited in the front of suction surface, and a small quantity of droplets are deposited on the casing and in the rear of pressure surface. The water extraction performance of the suction slots on hollow stationary blade is measured on a wet steam test rig. A series of scaled blades are testing under different outlet Mach number and suction pressure difference, where the position, width and shape of slot are altered respectively. The rounded inlet corner of suction slot is preferred and the optimal slot width maybe is about 1mm.

KEYWORDS
Water extraction, hollow stationary blade, wet steam, numerical analysis, experimental research

NOMENCLATURE
C_x relative axial width

INTRODUCTION

Steam turbine plays an important role in worldwide power industry. With more advanced cold end and higher rotor blade of last stage in low-pressure cylinder, a series of problems caused by wet steam, such as efficiency and safety are more severe.

In addition to mechanical losses, such as interphase drag loss, braking loss and pumping loss, the presence of wetness in steam turbine can also lead to many additional losses in thermodynamics, profile, shock wave and blade end, etc. The additional thermodynamics loss is caused by non-equilibrium condensing flow where the rapid-expansion pure steam will be in supercooled state, and the heat transfer process is anisothermal. Many experimental and numerical investigations on non-equilibrium condensing flow have been conducted for years, where the classical condensation theory and the growth rate of water droplets have been validated and studied in Laval nozzles and turbine cascades. Moreover, experiments have been carried on in model or full-scale LP steam turbines although the actual wet steam flow is more complex, and related numerical simulations have been undertaken to provide more details. A full understanding of the wetness effect on turbine performance is helpful for the optimization of turbine flow path design.

The water erosion on rotor blade is another serious problem. Consequently, several active or passive techniques have been developed to prevent or mitigate the damage. The surface treatment on
the leading edge of suction surface is the most widely used passive technique, which can enhance the water erosion resistance of blade. The active techniques refer to reducing the amounts of the coarse droplets impacted on the leading edge, such as centrifugal drainage, steam extraction, long axial spacing. Especially, the water extraction via the suction slots on hollow stationary blade is one of the most efficient methods\textsuperscript{18}. The deposition of submicron particles on turbine blades was experimentally studied\textsuperscript{19}, and the inertial deposition of droplets was numerically investigated\textsuperscript{20-22}, where the turbulence effect is considered. It indicates that due to the weak effect of turbulence on the trajectories of coarse droplets, the numerical prediction of deposition rates is relatively accurate\textsuperscript{23}.

In this paper, the vapour-liquid two-phase flow in the LP cylinder of a 1000MW nuclear steam turbine is studied, where the trajectories and depositions of fog and coarse droplets are numerically predicted. The water extraction performance of the suction slots on hollow stationary blades is investigated on a wet steam test rig with a series of scaled blades, and the effects of various factors are discussed.

**NUMERICAL MODEL**

The condensing flow was simulated by the ANSYS CFX 15.0 software. With this solver, it indicates that the predictions of wet steam flow in a model LP steam turbine have good agreement with the experimental measurements\textsuperscript{16}. The predictions for a 300MW operating steam turbine seem to yield reasonable results\textsuperscript{17} although the comparison between CFX numerical predictions and filed test results is unavailable. From the thermodynamic viewpoint, wet steam can be treated as a homogeneous mixture of vapour and droplets that both interphase velocity slip and temperature difference are neglected. Moreover, the dispersed-phase information is considered in non-equilibrium condensing flow by additional conservation equations and source terms. Drainage of depositional water should reduce the mass flow rate and improve the enthalpy of mixture, which can be approximately simulated by adding a negative source term in the mass conservation equation of the liquid phase. The fog droplets are treated as mono-dispersed instead of actual poly-dispersed in spite of the fact that it has significant impact on the formation of coarse droplets. This simplified Eulerian-Eulerian frame system is acceptable for the macroscopic thermodynamic simulations.

Altering the surface tension value in the nucleation equation to bring numerical results in line with experiment is common practice in wet steam flow simulations, and a surface tension correction factor was adjusted at 0.85 which is based on the work by Li et al.\textsuperscript{17} The IAPWS-IF97 formulation is employed as state equation due to its accuracy in metastable supercooled region. The standard k-ε two-equation turbulence model and the mixing plane model are adopted for the simulations.

In order to obtain the position and fraction of depositional water, it is necessary to track the motion of both the fog and the coarse water droplets, which move in the turbine blade passages. The Lagrangian particle-tracking model in the CFX software is adopted, where the average behaviour of droplets is represented by a specified number of sample droplets. In the current work, the phase interactions is one-way coupling that the reaction of droplets on the flow field is neglected, and the force that the vapour acts on the droplets is comprised of the viscous drag, the buoyancy force, the virtual mass force, the pressure gradient force, and the centripetal and Coriolis forces in a rotating reference frame.

Figure 1 shows the comparison between numerical results and experimental data of deposition rate, where the experiment was conducted by Philips et al.\textsuperscript{23} for a waveplate demister. Obviously, when the droplet diameter is greater than 10μm the predicted deposition rate is much more accurate than that below 7μm. All efforts indicated that the deposition rate is overestimated by CFX for small droplets although different mesh sizes and turbulence models were tried. However considering that the coarse droplets dominate the water deposition on last-stage stationary blade and the deposition caused by turbulent diffusion is insignificant, the Lagrangian particle-tracking model is still applicative in the current work. Moreover, although the flow velocity in waveplate demister is lower than that in turbine passage, the depositional behaviour of coarse droplets is almost unaffected, which is mainly determined by inertial effects.
EQUILIBRIUM AND NON-EQUILIBRIUM CONDENSING FLOW IN A LP CYLINDER

The computational domain for a 10-stage LP cylinder is shown in Figure 2. The chamber of the last-stage hollow stationary blade is directly connected to condenser, and the extraction slots are arranged in a staggered pattern on each hollow blade. The bleed points are located on the casing after the third, seventh and ninth stages. Drainage of depositional water is applied at the last three stages. A multi-block structured hexahedron grid is generated for the simulation, and the total number of grid nodes is about eight million.

Figure 2: Computational domain of the LP cylinder of a 1000 MW nuclear steam turbine

The equilibrium and non-equilibrium condensing flow are simulated under the same boundary conditions which is comprised of a specified mass flow rate and total temperature at the inlet section, a constant back pressure at the outlet section, specified mass flow rates at the three bleed points and the draining steps. The rotational speed is 1500rpm.

Thermodynamic expansion lines from the equilibrium and non-equilibrium condensing flow calculations are shown in Figure 3, and each point on the lines represents a mass-averaged thermodynamic static state (except for the first and last one) on the enthalpy-entropy diagram. The initial point represents the inlet stagnation state, and the last two points indicate the leaving loss. The saturation line is crossed in the fifth stage, and in contrast to the supersaturated region, the two lines are nearly coincident in the superheated region. In the equilibrium calculation, the wetness increases gradually after the exit of the fifth stator, but condensation should not happen until near the sixth
stator in the non-equilibrium flow, where a rapid entropy increase occurs. Because of the different numerical treatments for the drainage process, the expansion lines obviously separate after the seventh stage. As mentioned before, drainage of depositional water can be approximately simulated by adding a negative source term in the mass conservation equation of the liquid phase, but wet steam was treated as a single phase in the equilibrium calculation. To avoid the convergence problem caused by choked mass flow condition, the mass flow rate of drainage should not be ignored and so it was added to the mass conservation equation of the mixture as a negative source term. Thus the comparison between the expansion lines is pointless after the seventh stage.

Figure 3: Thermodynamic process lines of equilibrium and the non-equilibrium flows

Figure 4 shows the logarithmic nucleation rate of secondary nucleation under lower back pressure condition on the meridian plane of non-equilibrium flow. As shown in the figure, the secondary nucleation occurs in the tenth stage and near the hub of the ninth stator, however it will not have a material impact on the whole flow field because the nucleation rate is smaller than that of the primary nucleation by 5~6 orders of magnitude.

Figure 4: Distribution of the logarithmic nucleation rate of secondary nucleation

**DEPOSITION ON THE LAST-STAGE STATIONARY BLADE**

In the work above, only the fog rather than the coarse droplet can be described in non-equilibrium simulation, however the latter is an important source of depositional water. The coarse droplets are continually generated and move toward the blade tip under the action of stream and centrifugal force, then their spanwise distribution will be changed. Therefore, in order to analyze the distribution of depositional water on the hollow stator, it is necessary to determine the mass fraction and spanwise distribution of coarse droplets at the last-stage inlet, and the movement of coarse droplets in the upstream stages should be considered. Rather than complicated simulations of the information, deposition and motion, an empirical approach of Troyanovskii et al. was adopted, which can provide a reasonable distribution of the wetness caused by the coarse droplets.
Figure 5 shows the spanwise distribution of wetness at the last-stage inlet, and that of liquid mass flow rate is shown in Figure 6. The average diameter of the fog droplets 0.11 μm is obtained from the previous calculation, and that of the coarse droplets in spanwise is shown in Figure 7.

![Figure 5: Spanwise distribution of wetness at the inlet of the last stage](image)

![Figure 6: Mass flow rate of water droplets entering the last stage](image)

![Figure 7: Radius of the coarse water droplets entering the last stage turbine](image)

In the present work, the deposition rate for different categories of droplets is defined as the mass flow rate ratio of the depositional portion to the upstream corresponding-size droplets. The deposition rates near the last stator are summarized in Table 1. By contrast with nearly half of the coarse droplets (48.99%), only a small fraction of the fog droplets (0.39%) is deposited on the hollow blade.

<table>
<thead>
<tr>
<th>Position</th>
<th>Fog droplets</th>
<th>Coarse droplets</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade surface</td>
<td>0.39</td>
<td>48.99</td>
<td>13.51</td>
</tr>
</tbody>
</table>
The spanwise distributions of deposition rates on the blade are shown in Figure 8. Because of the influence of turbulent diffusion, the spanwise deposition rates of the fog droplets show randomness except near the tip zone. In general, the deposition rate of the coarse droplets increases along the spanwise, which should be affected by the upstream distribution as shown in Figure 6. Owing to the composition of depositional source, the total deposition rate is mainly dependent on that of the coarse droplets. In order to ensure the strength and stiffness of the hollow blade, it is reasonable to arrange the suction slots on only the upper half according to the water-erosion coefficient formula\(^2\) of rotor blade.

![Deposition of the fog droplets](image1)

![Deposition of the coarse droplets](image2)

![Deposition of the coarse and coarse droplets](image3)

**Figure 8: Span-wise distributions of deposition rates**

The axial distributions of deposition rates are shown in Figure 9. Although the fog droplets are mainly deposited in the rear of the pressure surface, the deposition rate still shows some randomness as it on the suction surface. The deposition rate of the coarse droplets is much greater than that of the fog, and most of the depositional coarse droplets are deposited near the leading edge of the suction surface due to the small axial clearance between the stages and the great inertia of coarse droplet. Theoretically, most of the depositional water can be extracted into the chamber if the slots on the suction surface are arranged near the middle, and the slots on the pressure surface should be as close as possible to the trailing edge. However owing to the limitation of trailing edge thickness and the negative impact of the radial pressure gradient near trailing edge on water extraction performance, it is appropriate to arrange the slots in the relative axial width of 0.8~0.9 on the pressure surface.
WATER EXTRACTION PERFORMANCE WITH DIFFERENT SUCTION SLOTS

It is too complicated to numerically simulate the wet steam and water film two-phase flow near the slot, moreover the flow and its mechanism in the slot is not clear yet, therefore the structure parameter design is mainly determined by experimental data. Experiment on water removal from steam turbine stationary blades by suction slots was conducted by Tanuma et al.\textsuperscript{18} It indicates that the reasonable suction pressure ratio is about 0.9–0.92, and the step-shaped slot could more favorable to improve performance than the straight. However there are no more details can be referred, such as the influence of slot width and variable operating condition on water extraction performance.

The water extraction performance of the suction slots on hollow stationary blade is measured on a wet steam test rig. As shown in Figure 10, water flows through the softener (1) and into the four electrically-heating vapour generators (2), and then the generating saturated vapour flows into the surge tank (5) via the steam manifold (3) and the pressure regulating valve (4) in sequence. Meanwhile water is pumped out from the tank (8) by the vortex pump (9) and sprayed into the surge tank (5) by six sets of atomizing nozzles, and then the well-mixed vapour and water flows through the hollow blade passage in the test section (7). Wetness and droplet radius can be measured by Malvern laser particle size analyzer through the optical window (6). The chamber of hollow blade is connected with the collecting tank (15) where the extracted water is collected by vacuum. Typical water droplet diameters and spectra are shown in Figure 11.
1-water softener, 2-vapour generator, 3-steam manifold, 4-pressure regulating valve, 5-surge tank, 6-optical window, 7-test section, 8-water tank, 9-vortex pump, 10-regulating valve, 11-condenser, 12-circulating pump, 13-cooling tower, 14,16-vacuum pump, 15-collecting tank

Figure 10. Wet steam test rig system for water extraction performance

Figure 11. Typical water droplet diameters and spectra measured by Malvern Particle Size Analyzer

The testing straight blades are scaled from the 70% relative height section of the hollow stationary blade, and only one suction slot is arranged on the suction surface. The straight slot with a width of 0.7mm is located at the relative axial width of 0.36 in the original modeling blade. Seven sets of comparative blades have been designed for the study on the influence of slot’s structural parameter, such as position, width and shape, and there is only one structural parameter of each comparative blade different with the original. All the blades are measured under at least nine kinds of operating conditions, namely three kinds of outlet Mach number multiply by three kinds of suction pressure difference, which can be adjusted by altering the vacuum in condenser and collecting tank respectively. The evaluation index of water extraction performance is the collected water volume per unit time.

Three kinds of slot’s shape are shown in Figure 12, and the influences of the shape on the collected water volume are compared in Figure 13. For the original blade, with increasing outlet Mach number or suction pressure difference, the collected water volume reduces first and then increase, and the corresponding operating conditions of the minimum are about Mach number of 0.7 and pressure difference of 7.5 kPa. For the stepped-shape slot, the water extraction performance is similar to the
straight one when the Mach number is less than 0.7, however it sharply reduces when the Mach number is greater than 0.7. This could be due to the additional sharp corner, which can lead the adjoining water film easier to be evaporated, especially under the operating conditions of high Mach number. For the rounded inlet corner, the maximum and minimum of the collected water volume are all improved, and the collected water volume is nearly maximized in a wide range of the test operating conditions. It indicates that the rounded inlet corner can improve the water extraction performance significantly, and is appropriate for the operating conditions of high Mach number as the result of avoiding evaporation caused by local sharp acceleration.

![Figure 12. Three kinds of slot’s shape](image)

The influences of slot’s position on the collected water volume are compared in Figure 13, where 0.36C<sub>x</sub> indicates that the center of slot is located at the relative axial width of 0.36, and so on. Overall, the collected water volume gradually increases with the moving downstream of slot’s position. This should be related with the water volume deposited on the blade surface before the slot, namely the water film thickness, and it also indicates that the water film on the suction surface has not separated in the range of test operating conditions. For the slot located at 0.24C<sub>x</sub>, owing to the greater shear force of faster steam flow on the water film, the thinner water film tends to cross over the slot in the case of high Mach number and low pressure difference, corresponding to the minimum value region shown in the upper left of Figure 13(a). When the Mach number is identical, the collected water volume reduces with the pressure difference increased from 4kPa to 8kPa. It indicates that evaporation occurs near the inlet corner of slot and intensifies with increasing pressure difference. However, the collected water volume increases with the pressure difference increased from 10kPa to 14kPa. It may be due to the condensation heat release from the saturated vapour around the inlet corner, and the evaporation of water film is instead suppressed by the increasing pressure near the corner.

For the slot located at 0.42C<sub>x</sub>, as shown in Figure 13(c), the collected water volume increases with increasing suction pressure difference until 10kPa due to the relative lower local pressure near the inlet of slot, where the water film is not easy to be evaporated, and it also indicates that the water film is likely to be evaporated when the suction pressure difference is greater than 10kPa. The thicker water film is harder to cross over the slot, thus the collected water volume slightly reduces with increasing outlet Mach number only in the upper left of contour, namely in the case of high Mach number and low pressure difference.

![Figure 13. Influences of slot’s shape on the collected water volume](image)
The influences of slot’s width on the collected water volume are compared in Figure 15. For the narrowest slot of 0.35mm, as shown in Figure 15(a), the collected water volume is small and reduces with increasing outlet Mach number when the suction pressure difference is low (about 4 to 6kPa), and it indicates that the water film is likely to cross over the slot in the case of high Mach number and low pressure difference. The collected water volume increases in the case of moderate pressure difference (about 6 to 12kPa), and the influence of Mach number is also obvious. The maximum value region corresponds to the Mach number about of 0.65 to 0.8, moreover as the Mach number reducing from 0.65 the water film get slow and thick, so it is likely that the inner layer is extracted into the narrow slot and the outer layer crosses over. As the pressure difference further increases (about 12 to 14kPa), the collected water volume reduces probably due to evaporation.

As the width increases to 1mm (Figure 15c), both the maximum and minimum of the collected water volume increase significantly. Since the water film can turn into the wider slot with smaller curvature, as a result the possibility of evaporation is reduced. However as the width further increases to 2mm (Figure 15d), the collected water volume reduces more than 20% compared with that of 1mm, although the distribution characteristics are similar. It may be due to the excessive wide slot where some vapour is extracted into, and then the pressure distribution at the slot outlet along the blade height becomes more uneven. As a result, the water extraction performance reduces, and the stage power also reduces due to the loss of working medium.
CONCLUSIONS

Numerical simulation and experimental investigation on water extraction of hollow stationary blade are presented under wet steam flow conditions. The numerical simulations show that the majority of depositional droplets are deposited in the front of suction surface, and a small quantity of droplets are deposited on the casing and in the rear of pressure surface. The experimental investigation shows that the rounded inlet corner of suction slot can improve the water extraction performance significantly, and the stepped inlet corner is likely to reduce the performance in the case of high Mach number. The water film thickness is not only influenced by the depositional quantity, but also influenced by the local Mach number, and only the water film with appropriate thickness can be extracted into the chamber effectively and adequately. The evaporation of the water film turned into slot should be inhibited, thus the suction pressure difference should not be excessive high. The excessive wide slot is likely to lead some vapour to be extracted into the chamber, and the optimal slot width maybe is about 1mm.

REFERENCES


