DETAILED STUDIES ON THE FLOW FIELD AND HEAT TRANSFER CHARACTERISTICS INSIDE A REALISTIC SERPENTINE COOLING CHANNEL WITH A S-SHAPED INLET

Ken-ichi Funazaki
Iwate University
Morioka, Iwate, Japan

Hikaru Odagiri
Iwate University
Morioka, Iwate, Japan

Takeshi Horiuchi
Kawasaki Heavy Industries
Nishi Akashi, Hyogo, Japan

Masahide Kazari
Kawasaki Heavy Industries
Nishi Akashi, Hyogo, Japan

ABSTRACT

Accurate temperature prediction of turbine blades for gas turbine is very important to assure the life-span of the blade under a hostile hot gas environment and intense centrifugal force. Therefore, there have been a number of studies carried out to clarify the cooling performance of serpentine cooling channel inside a turbine blade for gas turbine, however, it remains to be quite difficult to make an accurate numerical prediction of the performance. Apart from the effects of disk rotation as well as large temperature gradient near the wall, such a poor predictability can be attributed to the complicated vortical motions caused by the rib-roughened cooling channel whose cross-sectional shape varies along the channel and by the existence of u-bends. Furthermore, since the cooling channel inside a real turbine blade usually has a curved or S-shaped inlet, which may induce flow separation as well as swirl developed in the inlet, it can be imagined that the flow and heat transfer inside the cooling channel is likely to become much more complicated than that with a straight inlet. Despite this situation, only few studies are made in order to examine the flow and heat transfer characteristics inside the cooling channel with s-shaped inlet. Accordingly, this study aims at detailed experimental and numerical investigations on the flow and heat transfer characteristics of a realistic serpentine rib-roughened cooling channel with an s-shaped inlet, which is modeled from an actual HP turbine blade for gas turbine. This study employs a transient TLC (Thermochromic Liquid Crystal) technique to measure the heat transfer characteristics, along with the flow visualization on the inner surface of the channel using oil mixed with titanium powder. Note that a special focus in this flow visualization is placed on the area of s-shaped inlet. As for the flow measurement, 2D-PIV (Particle Image Velocimetry) method is used to understand time-dependent vortical structures of the flow field that can have significant impacts on the heat transfer. RANS-based numerical simulation is also executed to predict the heat transfer distribution on the inner surface of the cooling channel.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dh</td>
<td>hydraulic diameter</td>
<td>[mm]</td>
</tr>
<tr>
<td>erfc</td>
<td>error function</td>
<td></td>
</tr>
<tr>
<td>h</td>
<td>heat transfer coefficient</td>
<td>[W/m² K]</td>
</tr>
<tr>
<td>Ha</td>
<td>dimensionless helicity</td>
<td>[-]</td>
</tr>
<tr>
<td>k</td>
<td>fluid thermal conductivity</td>
<td>[W/m K]</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
<td>[-]</td>
</tr>
<tr>
<td>u̇</td>
<td>streamwise velocity vector</td>
<td>[m/s]</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>[-]</td>
</tr>
<tr>
<td>T_w</td>
<td>wall temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>T_i</td>
<td>initial temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>T_m</td>
<td>main flow temperature</td>
<td>[K]</td>
</tr>
<tr>
<td>t</td>
<td>time</td>
<td>[s]</td>
</tr>
<tr>
<td>U</td>
<td>velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>τ</td>
<td>time delay</td>
<td>[s]</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>c</td>
<td>specific heat</td>
<td>[J/kg K]</td>
</tr>
<tr>
<td>λ</td>
<td>solid thermal conductivity</td>
<td>[W/m K]</td>
</tr>
<tr>
<td>μ</td>
<td>viscosity coefficient</td>
<td>[Pa s]</td>
</tr>
<tr>
<td>̇ω</td>
<td>vorticity vector</td>
<td>[1/s]</td>
</tr>
</tbody>
</table>

Abbreviation

SS  suction side
PS  pressure side
INTRODUCTION

In order to raise thermal efficiency of gas turbine, higher turbine inlet temperature (TIT) is needed and TIT tends to increase year by year. Since high pressure turbine blades are exposed to a hostile hot gas whose temperature exceeds the metal melting point, there is a high risk of blade failure or drastic decrease in blade life span. Therefore, many methods for cooling turbine blades have been studied for a long time to protect turbine blades.

In this study, internal cooling inside the turbine blade is investigated experimentally and numerically. Generally, to promote heat transfer on the internal cooling channel surface, turbulence promoters or ribs are used there, which induce flow separation, reattachment and streamwise vortices in the cooling channel. Accordingly the internal flow becomes quite complicated, influencing the heat transfer characteristics. Therefore, a lot of researches have been done to elucidate the flow fields and heat transfer characteristics in internal cooling channel.

Han et al. [1] studied effects of rib angle, rib pitch, blockage ratio and aspect ratio of the channel using various types of straight ribbed channels. Fan et al [2] investigated the heat transfer performance of the turn region. Schabacker et al [3] examined the time-averaged flow field using PIV method in the 180 degree turn channel on which 45degree ribs were arranged. In the turn region, the secondary flow is generated, and influences heat transfer distributions in the turn regions. Farisco et al. [4] investigated the flow field and heat transfer in a two-pass internal cooling channel and compared those experimental data with numerical data obtained using different turbulence models. Jenkins et al. [5] studied the transient fluid temperature field in a ribbed internal cooling channel with high aspect ratio, and time-resolved results show that the dimensionless temperature field was time invariant, thus validating the assumption that the Nusselt number in transient heat transfer experiments is also time invariant over the duration of the experiment.

Funazaki [6] measured a heat transfer distribution using transient TLC method and nearwall flow using oil flow method over the pressure side in a serpentine cooling channel which had three passes arranged with inclined ribs. The result shows that oil traces on the test surface almost matched the heat transfer measurements. LeBlanc et al. [7] researched heat transfer distributions using transient TLC method in the leading edge channel and the three pass channels which were simplified so as to remove curved surfaces as seen in real designs and the results show 50-100% enhancement in heat transfer compared to the smooth surface.

Saha et al. [8] studied the entry geometries such as S-shape, twisted and 90 degree bend entrances, each of which was connected to a smooth channel or inclined ribbed channel. The results shows that the effect of entrance geometry was present for the first few hydraulic diameters only in the ribbed channel. Wright et al. [9] investigated the influence of entrance geometry on heat transfer under rotating condition in a rectangular channel. Test cases for fully developed flow, sudden contraction flow and partial sudden contraction flow conditions were compared under the stationary and rotating condition. The effect of the entrance condition was confirmed for both conditions. Sudden and partial contraction entrance provided higher heat transfer enhancement than the fully developed entrance in the channel with angled rib. However, the effect of rotation was more dominant than the effect of entrance geometry as the flow passed through the channel. Pearce et al. [10] reported the influence of rotation in a high aspect ratio serpentine channel and effect of three inlet velocity profile (Symmetric, PS bias and SS bias) using CFD. They concluded the inlet velocity profile could have a dominant effect on the flow structure and heat transfer in the first pass of a serpentine passage under stationary and rotating conditions, with lesser effects on the subsequent passes.

Most of these studies, however, dealt with simple models and detailed information on the flow and heat transfer in a realistic actual internal cooling channel with a realistic entrance geometry are few. It can be easily imagined that the flow field in a realistic channel, whose cross-section varies in the streamwise direction, becomes more complicated than that of simple model and it will accordingly become much harder to understand phenomena in the channel through the experiment and that may be the case even for very sophisticated CFD such as DES or LES. Apart from the realistic ribbed cooling channel, only few studies took into account the existence of S-shaped inlet which inevitably happens due to the accommodation of the cooling channel inside the turbine blade.

This study aims at detailed experimental and numerical investigations on the flow and heat transfer characteristics of a realistic serpentine rib-roughened cooling channel with an S-shaped inlet, which is modeled from an actual HP turbine blade for gas turbine. This study employs a transient TLC (Thermochromic Liquid Crystal) technique to measure the heat transfer characteristics, flow visualization on the inner surface of the channel using oil mixed with titanium powder, 2D-PIV (Particle Image Velocimetry) method to understand time dependent vortical structures. RANS-based numerical simulation also executed to predict the heat transfer distribution on the inner surface of the cooling channel.

EXPERIMENTAL SETUP

Test Model

Fig.1 shows two types of test models used in this study for heat transfer measurement for PIV measurement. These were made of acrylic resin. Internal geometries of the two channels are identical. They were equipped with 60 degree inclined ribs, curved surfaces, two bend corners and S-shaped inlet. The reason for employing the test model was to make PIV measurement much easier by engineering the external shape of the test model so as to increase the visibility of the model inside.
**Heat Transfer Measurement**

Fig. 2 shows an experimental apparatus for the heat transfer measurement. A laminar flow meter and a heater were installed downstream of the blower and a three-way valve was set just after the heater. A test model was connected to one side of the three-way valve, while an adjusting valve was connected to the other side of the three-way valve for pressure loss adjustment. Thanks to this adjustment, even when the three-way valve was switched to divert the flow direction, the flow rate in the test model was almost unchanged. In addition, two screens and a honeycomb were provided in the connecting pipe between the test model and the three-way valve in order to reduce the turbulence level as well as the swirling flow caused by the bend of the pipe.

A digital video camera (SONY HDR- PJ 800) was set in front of the test model to capture the color change of TLC. In the experiment, to take clear movies of TLC, the test model was illuminated by two halogen lamps covered with tracing papers to make the light more uniform. In addition, to avoid the reflection of light from the surroundings, sun-shade curtains were used around the test model so that unnecessary light affecting the color image capturing was almost shut out.

In this study, heat transfer coefficient inside the channel was measured by transient TLC method developed by Vedula and Metzger [11]. The inner surface of the channel inside the model for heat transfer measurement was coated first with black paint then with TLC, whose nominal color band was from 30 to 32 degree Celsius. In this experiment, the heated mainstream air instantaneously entered the test model after the three-way valve was switched from a warm-up mode to the measurement model, and the swift switching of the valve guaranteed a suitable step-like temperature change over the channel surface of the test model.

Assuming that the surface of semi-infinite object with initial temperature $T_i$ is abruptly exposed to the flow with constant temperature $T_g$, the surface temperature $T_s(t)$ of the object follows the expression

$$T_s(t) = T_g - T_i \left[1 - \exp\left(\frac{\beta(t)}{\rho c\lambda}\right)\right]$$

$$= T_g - T_i \left[1 - \exp\left(\frac{h_i t}{\rho c\lambda}\right)\right]$$

(1)

In reality, it is difficult to realize step-like temperature change over the surface. To cope with such a situation, Duhamel's theorem can be applied to have the expression of Eq. (2), by
approximating a gradual increase in temperature into a summation of small step-like temperature rise with the time delay $\tau$ from the start of the measurement,

$$T_w(t) - T_i = \sum_{j=1}^{N} \left\{ 1 - \exp(\beta(t - \tau_j)) \right\} \text{erfc}\left(\sqrt{\beta(t - \tau_j)} \right) (T_{g,j} - T_{g,j-1}) \quad (2)$$

According to Ishizawa's research [12], the division number $N$ of the temperature rise curve was selected to be 40.

The main flow temperature $T_g$ in the test model was measured by the thermocouples as shown in Fig.1. Since the main flow temperature decreased in the flow direction due to the endothermic action of the test model, the obtained temperature data was linearly interpolated to calculate mainstream temperature at an arbitrary position in the channel. The wall temperature $T_w$ was calculated from the captured images of TLC by use of the calibration curve between hue and temperature.

Re number, defined by Eq. (3), was 25,000, where the hydraulic diameter $D_h$ of the cooling channel at A - A in the Fig.1(a) was used as characteristics length.

$$Re = \frac{\rho D_h U}{\mu} \quad (3)$$

Nusselt number was defined as follows,

$$Nu = \frac{hD}{k} \quad (4)$$

The method of Saabas et al. [13] was used to evaluate uncertainty of the heat transfer coefficient. It was found that the heat transfer coefficient includes 14% uncertainty on the pressure side and 13% on the suction side.

**Calibration of TLC**

It is necessary for the temperature measurement to establish a relationship between surface temperature and color or hue in this study. Fig.3 shows the calibration device, which consisted of a Peltier device and a copper plate. The copper plate, which was first painted with Black paint and TLC, was covered with an acrylic plate as shown in this figure. A hole for thermocouple insertion was opened in the copper plate and the temperature just beneath the copper plate surface was measured by using a K-type thermocouple. It was confirmed that the temperatures of the copper plate surface and the inside temperature were almost the same. A temperature controller connected to the Peltier devices changed the temperature at regular intervals over the color range of TLC. Temperature was measured with the thermocouple, simultaneously pictures of TLC were taken. As a result, a correlation between the hue value of TLC and its temperature was obtained. During the calibration test, the measurement equipment and their arrangement were carefully placed so as to be as similar as possible with those of actual measurements. Also, since the illumination arrangement for the pressure side measurement was slightly different from that for the suction side measurement, the calibration test was performed for each case.

Fig. 4 shows the calibration curve between temperature and hue value for the pressure side case. The calibration test was carried out 6 times for each case under its illumination condition, and each of the correlation curves was obtained by averaging those data. Also, since it was difficult to approximate these curves with a single curve, they were approximated by combining three polynomial curves.

**Fig. 3 Calibration devices**

**Fig. 4 Calibration curve (Pressure side)**

**Oil Film Method**

To understand the flow near the surface, which influences the heat transfer distribution, oil film method was carried out. In this method, S-shaped inlet channel was focused on. As with the heat transfer measurement, the main flow was flowed into the test model by changing the three-way valve. In order to prevent the oil film from dropping due to gravity, video shooting was performed as shown in Fig.5. The mixing ratio of the oil film is shown in Table 1.
To understand the vortex structure in the channel, 2D-PIV measurement was carried out. Experimental setup of the PIV measurement is shown in Fig.6. Using a high-speed camera (Vision research Phantom v1210) with camera lens (TAMRON Model B01N), particle images were taken. The Schémpflug principle was employed in this study to capture particle images because the image sensor of the camera was not parallel to the laser sheet and there existed some regions that could not otherwise be properly focused on. As a laser sheet device, 15W YVO4 laser (Kato koken G15000K) was used. It is continuous wave laser whose wavelength is about 532nm. Using atomizer (Dantec Dynamics saftex fog generator), tracer particles were generated from 10T10 solution (Dantech Dynamics). The average size of particles is about 1 μm cited from the maker catalog.

Table 2 Camera setting

<table>
<thead>
<tr>
<th>Image size</th>
<th>512 × 512[px]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shutter speed</td>
<td>30000[fps]</td>
</tr>
<tr>
<td>Exposure time</td>
<td>32[μs]</td>
</tr>
<tr>
<td>F-number</td>
<td>F3.5</td>
</tr>
</tbody>
</table>

A laminar flow meter was installed downstream of the blower to measure the main flow mass flow rate. The main flow temperature was measured using a K-type thermocouple at the upstream side of the entrance of the test model for PIV measurement. In the connection pipe upstream of the test model, two rectifying screens and a flow-regulating honeycomb were installed, and the turbulence flow including the swirling flow caused by the bend of the pipe was reduced. Three-way valves were installed at three locations as shown in Fig. 6. A pressure adjustment valve was installed on one side of three-way valve ①, and an atomizer was placed on one side of three-way valve ②. Since the pressure inside the test model was high and it was difficult to supply the tracer particles into the chamber continuously, at first the particles was injected into the chamber through the valve ② to accumulate some amount of particles enough for PIV measurement, while the valve ③ was open to the atmosphere to release the air including the particle outside. After the accumulation of the particle was confirmed, the valve ③ was turned to supply the air with the particles into the test model. Reynolds number in PIV measurement was the same as that of the heat transfer measurement.

Table 2 shows the video shooting conditions. DynamicStudio v4.00 (Dantec Dynamics) was used for processing data obtained by the PIV measurement. In the image processing, adaptive PIV processing was performed in which the interrogation area was dynamically changed from 32 × 32 pixels to 16 × 16 pixels. Overlap was set at 50%. The number of particle image to be processed was 3000 images for each condition.

**Numerical Setup**

RANS simulation was carried out using a commercial solver ANSYS CFX ver15.0. Fig.7 shows the computational domain. Also Table 3 shows boundary conditions. For each boundary condition value, the corresponding experimental value obtained in the heat transfer test were specified. Unstructured mesh was generated by Ansys ICEM ver16.0.

Prism meshes were allocated near the wall surface, and the y+ of the meshes on the wall surface was less than 1. Total number of the nodes used in this study was determined from the mesh dependency check shown in Fig.8. The averaged heat transfer coefficient of the suction and pressure side calculations was used as an evaluation index of the mesh dependency check, where heat transfer coefficient was calculated using Eq. (5)

\[
h = \frac{q}{(T_v - T_s)}.
\]

The adapted number of nodes was about 12.5 million, which yielded only 2% difference in the averaged the heat transfer coefficient from the heat transfer coefficient attained by the finest mesh condition in this investigation. To compare experimental results and computational results, the definition of \( T_g \) [K], mainstream temperature, was the same for experiment and computational simulation when calculating heat transfer coefficient.
RESULT AND DISCUSSION

Heat Transfer Measurement

Experimental findings

Figs. 9 and 10 show the Nusselt number distribution on PS and SS, respectively. In both cases (a) was obtained by the experiments, and (b) was the calculated result by RANS. The measured surface in heat transfer measurement was the wall surface separated by ribs, where ribs were not included.

Looking at the Nusselt number distribution (Fig. 9 (a), Fig. 10 (a)) obtained by the experiments for both the SS and PS planes, higher Nusselt number regions were identified at the downstream of the ribs area where the ribs and the neighboring side wall formed acute angle, which could be attributed to flow reattachment. It was also found that those high Nusselt number regions happened to have some angle to the rib. This tendency was quite similar with the previous studies on the rectangular straight channel that had opposing parallel inclined ribs [1].

In the 1st pass, the level of Nusselt number was different between PS and SS, where SS values were higher than PS values. In a rectangular straight channel having opposed parallel inclined ribs, it is known that a pair of streamwise vortices generated in the channel are symmetrical between SS and PS. Therefore, one of the reasons for this difference could be attributed to the incoming flow through the S-shaped inlet, which was found to generate asymmetric flow structure before entering ribbed section of the 1st pass, as will be shown later. In addition, the fact that the channel itself was curved could have some influence on the observed difference.

In the 180 degree turn from the 1st pass to the 2nd pass, both PS and SS had regions with high Nusselt number near the dividing wall in the channel. This was probably because the flow field became very complicated due to the rib-based vortices developed in the 1st pass as well as secondary flows.
generated in the first turn region. However, similar tendency was not observed in the 180 degree turn from 2\textsuperscript{nd} pass to 3\textsuperscript{rd} pass.

In the 2\textsuperscript{nd} pass, the Nusselt number tended to gradually decrease as the flow proceeded downstream, which was partially because the channel cross-section area became wider and the flow velocity decreased in the second pass. In the 3\textsuperscript{rd} pass, while PS and SS had different heat transfer distributions near the bend, PS and SS tended to have similar heat transfer distributions.

**Comparison with CFD**

The experimental results for PS were compared with the RANS simulation, as shown in Fig.9. The experimental data was well reproduced by CFD in the 1\textsuperscript{st} pass. However, the heat transfer distribution was clearly different from the numerical analysis at first turn regions and 2\textsuperscript{nd} pass.

In turn region, unsteady flow which is composed of a secondary flow generated by ribs, a corner vortex, a swirling flows, reattached by ribs and separation induced tip wall between channels is presumed to be exist and interfered each other. In the 2\textsuperscript{nd} pass, especially heat transfer distribution on PS is significantly different between experiment and CFD. This is because the cross section shape is like trapezoidal and flows come from turn is complicated. These complex unsteady flow and characteristic channel shape could make it quite difficult to predict such flows by the steady-state flow analysis in first turn and the 2\textsuperscript{nd} pass.

In the 3\textsuperscript{rd} pass, there is a discrepancy between CFD and the experimental results near the second turn in terms of the peak positions of Nusselt number immediately behind the rib as well as the areas of high heat transfer region. Downstream of the second turn, the numerical results gradually reproduced the features of experimental data, especially in terms of the appearance of high Nusselt distribution between the ribs. A following scenario is possible to explain the above-mentioned phenomena, that is, although the unsteady flow passing through the 2\textsuperscript{nd} pass and the second turn made it difficult to predict the flow field around the second turn and its neighborhood by RANS simulation accurately, the flow field downstream of the second turn tended to be greatly influenced by the rib-based streamwise vortices so that the flow in the 3\textsuperscript{rd} pass became relatively steady as it went downwards, which eventually made RANS-based flow prediction more reliable.

Similar comparison was made in Fig.10 for SS. The heat transfer distribution obtained by CFD resembled the experimental results to some extent, but it is also obvious that there is a deviation in the peak and the area of high Nu at the first and second turns, as seen in PS.

Fig.11 shows area-averaged Nusselt number, where the averaging area are depicted in Fig.12. Nusselt number distributions on the centerline of each channel are shown in Fig.13, 14 and 15, respectively. Fig.11 exhibits comparisons of the area average Nusselt numbers obtained by CFD and the experiments on both PS and SS. Although the heat transfer levels were different between CFD and the experiments, especially for the 1\textsuperscript{st} and 2\textsuperscript{nd} passes, overall features of the experimental heat transfer distributions were predicted by CFD. It may be possible that the flow separation was excessively evaluated in RANS simulation, resulting in underestimation of Nusselt number by RANS simulation. Further discussions on the heat transfer distributions in comparison with the numerical results are described in the following.

![Fig.11 Area averaged Nusselt number distributions](image)

In the 1\textsuperscript{st} pass, the area-averaged Nusselt number on PS and SS increased towards the first turn, reaching its first peak at area 3 for PS and area 4 for SS. In addition, the averaged values on SS were larger than those on PS. These trends were well predicted by CFD. Similar behaviors can be seen in the center line data shown in Fig.13. The level of the area-averaged Nusselt number was underestimated by CFD shown in Fig.11, however CFD was able to reproduce the local behaviors of the experimental data qualitatively. Several ways of reasoning can be made for the pronounced difference of the level of Nu between PS and SS, for example a slight influence of 3D heat conduction effects in the experiment especially near the ribs. In addition, as will be discussed later, effects of S-shaped inlet seem to have contributed to the difference. Note that a sharp increase in the averaged Nusselt number on PS appeared in the area 6 and area 7, around the first turn, which was also well predicted by CFD.

In the 2\textsuperscript{nd} pass, the area-averaged Nusselt number decreased towards the second turn as can be seen in Fig.11. This tendency can be confirmed in the centerline data in Fig.14. As will be shown in Fig.16, the variation of the streamwise flow velocity along the channel was responsible for the decrease to some extent. The experimental values on SS were reasonably predicted by CFD, while there were still large discrepancies between CFD and the experiments on PS, which
could be closely connected to the sharp increase in Nusselt number around the first turn as mentioned above.

In the 3rd pass, the area-averaged Nusselt number became almost leveled off until the mid of the 3rd pass, followed by moderate increase towards the channel outlet. This tendency was also confirmed in the results of Fig.15. From Fig.15, in the area-averaged Nusselt number, CFD and the experimental values roughly agreed with each other, although some differences were observed in peak position of Nusselt number for both SS and PS. However, at the downstream position, the peak positions of CFD gradually matched the experiments. This may be because secondary flow induced by rib effect became stronger than the effect of turn effect.

Fig.13 Nu number on center line of 1st Pass

Fig.14 Nu number on center line of 2nd Pass

Fig.15 Nu number on center line of 3rd Pass

Fig.16 shows averaged x velocity component on the channel cross sections obtained by CFD. The area-averaged x velocity component on each cross section was normalized by the values at X=4.2 D_h. In the 2nd pass, since the main flow flows in opposite to the X axis direction, the absolute value of the X component velocity is used. The position of each channel cross section is displayed in Fig.17. From Fig.11 and Fig.16, it can be stated that there is a correlation between the variations of the area-averaged x component velocity and the change of the Nusselt number in each channel, that is, the Nusselt number increased as the averaged x component velocity increased, and vice versa.

Fig.16 X velocity component on channel cross section

Fig.17 Definition of the channel cross section
Fig. 18 Dimensionless helicity distribution on the cross-sections

Fig. 18 shows the dimensionless helicity contours on the channel cross-sections obtained by CFD. The position of each of the cross-sections appears in Fig. 17. The dimensionless helicity was calculated by,

$$H_n = \frac{\bar{u} \cdot \bar{\omega}}{\bar{u} \cdot \bar{u}}$$

In the 1st pass, it can be inferred from the dimensionless helicity contours at 6D_h and 7.5D_h that there existed the influences of the secondary flow generated in the S-shaped inlet channel as mentioned latter. Going downstream from 9D_h to 12D_h, dimensionless helicity contours tended to be almost symmetric, which was because the secondary flow induced by ribs was becoming dominant over the inlet flow effect. Near the turn from 13.5D_h to 15D_h, the symmetry of the dimensionless helicity structure was gradually getting collapsed.

As mention before, in the 2nd pass, the measured Nusselt number on SS was larger than that on PS. As can be seen from Fig. 18, the cross-sectional shape in the 2nd pass is rather trapezoidal, and the dimensionless helicity near the SS became dominant, resulting in a clear difference in heat transport capability in the 2nd pass between SS and PS.

In the 3rd pass, the difference in the area-averaged Nusselt number between PS and SS became small, which was partially because the shape of the channel cross section was almost rectangular and the dimensionless helicity by the PS and the SS were relatively symmetrical.

Oil film patterns

Oil film method was applied to the area of the S-shaped inlet. Fig. 19 shows a flow pattern on PS of the S-shaped wall. Looking at (a) in Fig. 20, it was found that the flow towards the bottom of the channel changed its direction slightly toward the upper side of the channel as it approached the first rib at the beginning of 1st pass. This behavior was also observed in the calculated limiting streamlines on the wall. In addition, the oil film was accumulated just upstream of the rib at the beginning of 1st pass. In the results of CFD, a separation line was observed as well.

Fig. 20 shows the results of the flow field near the S-shaped wall on the SS. (a) is the experimental result of the oil film method and (b) is the result of the CFD wall surface streamline. As shown in Fig. 21, the results obtained by the oil film method and the CFD result were roughly in agreement, implying the movement of the oil film on the upper surface and the bottom surface of the channel. In the region sandwiched between those two regions, there appeared a region with little movement of the oil film. Separation was also seen from the result of the streamline of CFD.

Fig. 21 shows the contour colored with dimensionless helicity on the isosurface of the Q value in the S-shaped channel near SS obtained by the CFD. As seen in Fig. 21, vortices were generated at the upper side and the lower side. The position of these vortices are considered to correspond to the position where the oil film moved in the experiment. Also, it was found that three vortices were formed immediately upstream of the 1st pass. It can be imagined that the heat transfer is promoted by the existence of incoming vortices from this S-shaped inlet, and it was confirmed by the measurement and the calculation that the heat transfer actually increased on the SS of 1st pass.

Fig. 22 shows the result of oil film at the 1st pass. From Fig. 22, the reattachment area was different between PS and SS. Looking at PS, it can be seen that the separation line was located on the downstream side of each of the ribs in comparison with the SS case. This delayed reattachment caused the decrease in heat transfer on PS wall near S-shaped inlet.

From the results of this survey, it is revealed that S-shaped inlet effects heat transfer distribution in the realistic channel, indicating a possibility that the expected cooling performance of the ribbed channel may not be attained. On the other hand, turbine cooling designers may be able to control the heat transfer at least in the 1st cooling channel by changing the shape of S-shaped inlet channel. This implies necessity for further investigation on the influencing factors in a S-shaped inlet on the heat transfer distribution (for example, aspect ratio, length of inlet channel, radius of filet, twisting of the channel and so on.).
Carrying out 2D PIV method, vortical structures in the cooling channel were examined. In this method, a special focus was on the 180 degree turns regions because the heat transfer distribution is likely to differ between experiment and CFD. First, the flow field in the serpentine channel was experimentally investigated by PIV and the measured flow fields were compared with the flow field obtained by RANS. Note that the particle images in the vicinity of the wall surface could not be photographed due to the complicated flow path shape and there was no data near the wall.

Fig.23 shows that a flow field in the second 180 degree turn of the result of PIV (time averaged) and CFD (RANS). From Fig.23, the size and position of the pronounced vorticies obtained by PIV were different from the results of CFD. Also, as for the X direction velocity, and CFD overestimated the velocity in comparison with the experimental result. This was due to the impingement flow to the outer wall side and the influence of the centrifugal force etc. in the turn region, and the flow field with strong swirl could be predicted in the RANS simulation.

In order to understand the unsteady phenomena, Fig.24 shows the X component velocity contours and vector maps obtained by PIV for every 10 frames (every 0.33 ms). The position is as same as Fig.23.

In order to have a further understanding of the detailed flow phenomena, we will conduct unsteady flow analyses based on LES and compare them with the time-resolved flow fields by use of PIV.
Fig. 23 Time averaged X component velocity and vector map at 2nd turn

Fig. 24 Snapshots of X component velocity and vector map at 2nd turn obtained by PIV
CONCLUSIONS

Experiment and computational analysis were carried out in a realistic serpentine channel with s-shaped inlet. The conclusions in this study are itemized as follows,

1. The prediction accuracy of the numerical analysis was confirmed to some extent in the entire serpentine channel. Although the heat transfer levels were different, a similar trend was observed in the experiment and steady flow analysis.

2. Heat transfer distribution characteristic of the serpentine channel was different between PS and SS due to the difference in cross-section shape of the channel along with flow direction. Also it was found that there was a correlation between the variations of the area-averaged x component velocity and the change of heat transfer in each channel position.

3. It was revealed that S-shaped inlet geometry affected the heat transfer distribution, particularly in the 1st pass of the realistic cooling channel.

4. In the turn region, time-dependent flow behaviors were identified, and the flow field changed with time and various secondary flows interfered with each other, which made RANS flow prediction quite difficult.

ACKNOWLEDGMENTS

In carrying out this research, we express our gratitude for the cooperation of Mr. Terabe, former student of Iwate University, who has contributed to the study.

REFERENCE


