GPPS-CH-2020-0062

Numerical Study on the Impact of Air-Side Inflow Condition and Oil-Side Thermal Boundary Layer on a Cooling Performance of Surface Air Cooled Oil Cooler

Tsukasa Ishii, Toshinori Watanabe, Takehiro Himeno
The University of Tokyo
ishii@aero.t.u-tokyo.ac.jp
Bunkyo-ku, Tokyo, Japan

Susumu Tomida
Sumitomo Precision Products Co.
Amagasaki, Hyogo, Japan

ABSTRACT

Surface Air Cooled Oil Cooler (SACOC) is a widely used oil cooler for turbofan engines. While SACOC has a simple configuration of air-cooling fins on oil passages, the cooling performance is not sufficiently clarified because the coolant air is of high-Reynolds number. In the present study, a numerical analysis was conducted to identify the impact of inflow swirl angle and inflow turbulence intensity on the cooling performance of SACOC. From the results, it was found that the cooling performance was not sensitive to either swirl angle or turbulence intensity. A simple numerical method for a crossflow type heat exchanger was also developed in this study to avoid simulating several hundreds of air-side and oil-side passages. With the developed method, the thermal resistance of the oil passage of SACOC was investigated, which was usually ignored in industrial investigations, and a possibility of improving the cooling performance was found in oil-side devices.

INTRODUCTION

Thermal management is one of the critical technology issues in developing modern turbofans. In recent years, the introduction of a fan drive gear drastically increased the amount of dissipated heat. The severe situation is thought to be more challenging in the future; new electric systems require additional cooling capacity (Morioka et al., 2018).

In a turbofan, lubricating oil works as a heat transfer medium and is finally cooled by oil coolers. A turbofan usually has two types of oil coolers: an air-cooling system and a fuel-cooling system. For the air-cooling system,

Surface Air Cooled Oil Cooler (SACOC) is installed in a modern turbofan. Figure 1 shows the schematics of SACOC. SACOC is a crossflow heat exchanger with air-cooling fins, and oil flows in the plate-like structure with corrugated fins inside. The system is installed at the downstream of fan exit guide vanes.

SACOC is widely used because of its advantages of less aerodynamic losses and lighter weight than compact heat exchangers. However, there is an emerging issue, that is, modern turbofan engines are equipped with so many SACOC's almost to the limit of available areas in the fan duct. Moreover, because the reduction in fuel consumption limits the capacity of the fuel cooling system, the air-cooling system has to manage more heat load than ever. Therefore, the cooling characteristics of SACOC should be clarified precisely to develop an efficient thermal management system for future engines.

There are a few previous reports about the cooling fins for SACOC. Outirba & Hendrick (2013) showed that the thermal performance of cooling fins was not effectively improved by the refinement of fin shape under high velocity flow situation. Sousa et al. (2014) tested the fin array in a wind tunnel with spiral passage to reproduce

Figure 1 Schematics of SACOC,
Left: Installation Position, Right: Flow Passages

swirling at the position of a fan splitter. Based on the experimental results, they simulated the performance dependency on the cruise condition of an aircraft. Villafañe & Paniagua (2018) discussed the disturbance of airflow owing to the existence of fins using the same wind tunnel as Sousa et al. (2014) used. Liu et al. (2019) conducted numerical analysis on the cooling fins of SACOC and showed the geometric impacts on the mass flow rate in the fin channel. Tomoda et al. (2013) clarified that the inlet separation and the flow leakage from the fin channel characterize the heat dissipation performance of SACOC. In response to this knowledge, Sekoguchi et al. (2015) tested cooling fins with devices to suppress the boundary layer development. Their study showed that the cooling performance is improved by increasing the mass flow rate in the fin channel. Ishii et al. (2019) studied the potential of flow separation for heat transfer enhancement, and they concluded that turbulence promotion is not suited to SACOC.

By virtue of these previous studies, the characteristics of SACOC have been gradually understood. However, there is no report for the performance sensitivity on the inflow condition of the air, whereas actual SACOC experiences highly turbulent and swirling inflow at the off-design condition. Besides, no previous study considered the thermal boundary layer in the oil passages, which inevitably affects the cooling performance.

The present paper aims to clarify the impacts of the swirling inflow, the inlet turbulence of the air, and the thermal boundary layer development in the oil-side passages to the performance of SACOC. A novel method to estimate the overall performance of a crossflow system with the simulation results of several passages was developed. Because the oil property is highly dependent on temperature, not only the airflow but the flow field of the oil was numerically simulated.

METHODOLOGY

Computational Scheme

In this paper, CFD analysis was conducted with the in-house code incorporating a conjugate heat transfer technique. The target system consisted of air, aluminum, and lubricating oil. These domains were discretized with a finite volume method. The interface temperatures between the domains of different solvers were given explicitly. The diffusion terms in all the domains were evaluated with the second order central differentiation.

In the air domain, Reynolds-averaged Navier-Stokes equations were solved in combination with the two equations of $k - \omega$ turbulence model (Wilcox, 1988). The inviscid terms were evaluated with Simple High-resolution Upwind Scheme (Shima & Jounouchi, 1997) and the third order MUSCL interpolation.

In the oil domain, Navier-Stokes equations were solved by use of fractional steps. CIP scheme (Yabe & Takei, 1988) was used for the convective terms. The

pressure term was evaluated with the second order central differentiation.

In the solid domain, the thermal diffusion equation was utilized. The solid domain was discretized with relatively coarse meshes compared to the fluid domain.

Heat transfer to a cell on the interface between the solid domain and the fluid one was calculated as the sum of all the transferred heat from the facing cells. Figure 2 is a schematic of the interface cells. The cell faces do not match at the interface because of the difference of mesh sizes. There exist overlapping cell faces, A_i , between every pair of facing cells. The heat through the area, Q_i , is then calculated from the following equation with the temperature T , the distance between the cell center and the interface Δx , and the thermal conductivity κ .

$$Q_i = A_i \left(\frac{T_s - T_f}{\Delta x_s / \kappa_s + \Delta x_f / \kappa_f} \right)_i \quad (1)$$

The subscripts “s” and “f” denote the value of the solid and the fluid domain, respectively. The temperatures T_s and T_f were given by linear interpolation. Further details on the coupling were described by Ishii et al. (2019).

Target System Geometry

A simplified model of SACOC illustrated in Fig. 3 was adopted in this study. The curvature of the structure was ignored, i.e. the rectangular air-cooling fins were mounted on a flat floor, and the oil-side corrugated fins were replaced by the rectangular passages.

The geometry of the air-cooling fins and the oil passage is summarized in Table 1. Note that all the schematics in the present paper are geometrically scaled for visibility. The geometry was determined to simplify the configuration previously tested by Sumitomo Precision Products.

Figure 2 Heat Transfer between Facing Cells

Figure 3 Simplified Crossflow Model of SACOC

Table 1 Target Geometry

	Air cooling fin		Oil passage	
Length	L	200 mm	W	600 mm
Thickness	t	0.8 mm	t_{oil}	0.45 mm
Pitch	s	4 mm	s_{oil}	1.43 mm
Height	H	25 mm	H_{oil}	5.95 mm
Base thickness	H_{base}	2 mm		

Computational Condition

Analysis without Oil Passages

The impacts of inflow turbulence and inflow swirling were studied by simulating the airflow and the thermal diffusion in the air-side fin. Figure 4 shows the schematic of the computational domain. The pitchwise widths were set as a whole pitch for the swirl study, and half a pitch for the other studies to save computational costs.

The total pressure was fixed at the inlet, and the static pressure was fixed at the outlet boundary. No boundary layer was set at the inlet boundary for ease of comparison with wind tunnel tests of clean inflow in which the boundary layer is controlled, and the turbulence length scale at the inlet condition was set with the reference to the thickness of the boundary layer which developed with the entrance region of 200 mm long, not to the wakes of actual fans. The upper boundary was assumed as a slip wall, while the bottom boundary and the fin surface were all assumed as non-slip walls. The bottom walls at the inlet and the outlet region were adiabatic. The bottom wall of the fin passage was assumed to be isothermal in accordance with the heating from hot oil. For the turbulence intensity study, the flow field was assumed to be periodic and symmetric, and the pitchwise boundaries were set as slip walls. The boundary conditions of the swirl study were the same as those of the turbulence study excluding the pitchwise condition; the pitchwise boundaries were connected to be periodic.

The computational condition of these analyses is shown in Table 2. The turbulence intensity was changed up to 15%, assuming large turbulence at a fan exit. The swirl angle was changed up to 25° according to the off-design operation in a flight.

Analysis with Oil Passages

Figure 5 shows the schematic of the computational domain. The oil-side passages were connected with each other to form a U-shape circuit. The passages in $L/2 < x < L$ were inflow passages, and the passages in $0 < x < L/2$ were returning passages. Several passages were chosen to conduct CFD in the air-side passages, the oil-side inflow passages, and the oil-side return passages, respectively. The passages were determined to be evenly spaced and to cover the whole area.

The paired passages had the same pressure distribution, the same temperature distribution, and the velocity distribution flipped in the y -direction at the y -axis positive boundary. Parabolic velocity distribution was given at the oil inlet to simulate developed inflow. The walls on the oil passages were set as non-slip walls, and the bottom floor of the oil passages was set as an adiabatic one. The flow field was assumed to be almost symmetric, and half a pitch of each passage was simulated. The other thermal boundary condition was given based on the assumptions in the following section.

The analysis condition is listed in Table 3. The Reynolds number in the oil-side is based on the hydraulic

diameter. The air-side condition was the same as that of the turbulence intensity analysis. The oil property was set as a commercial turbine oil whose property meets the British requirement of DEF STAN 91-100.

Figure 4 Computational Domain for the Air-only Analysis

Table 2 Flow Condition for the Turbulence Intensity Study and the Swirl Study

	Turbulence intensity study	Swirl study
Inlet Mach number	0.4 (referential)	
Inlet total pressure	101,325 Pa	
Reynolds number (Re_L)	1.4×10^6	
Inlet total temperature	348.15 K	
Fin root temperature	403.15 K	
Swirl angle of the inflow air	0°	0°, 5°, 15°, 25°
Turbulence intensity of the inflow air	1%, 3%, 7%, 15%	1%

Figure 5 Computational Domain for the Crossflow Analysis

Table 3 Flow Condition for the Crossflow Analysis

Inlet Mach number (referential)	0.4
Total pressure of the inflow air	101,325 Pa
Air-side Reynolds number (Re_L)	1.4×10^6
Total temperature of the inflow air	348.15 K
Swirl angle of the inflow air	0°
Turbulence intensity of the inflow air	1%
Total mass flow rate of the oil	2.5 lb/s
Average velocity at the oil inlet	3.0 m/s
Oil-side Reynolds number	1.4×10^3
Total temperature at the oil inlet	403.15 K

Assumptions on the Thermal Characteristics in the Crossflow System

In the present study, the following assumptions were made to enable the performance estimation of a crossflow heat exchanger without simulating all the passages.

Assumption A: the temperature distributions of neighboring passages are almost identical both for air and oil. Hence, the heat transfer between neighboring passages was ignored. Heat transfer through the plane $x/L = 0.5$, where the inflow oil passage is located next to the returning oil passage, was also ignored in accordance with some structural elements.

Assumption B: from the air-side domain, the oil-side temperature can be referred to as linear, and vice versa. This assumption is based on the fact that a blazing plate is placed between the air-side and the oil-side structures, and the thermal diffusion in solid is relatively simple.

Because of the difference in the lengthwise scale and the pitchwise scale, the cell widths in the x - and the y -directions have quite different geometries between the air- and the oil-side. Therefore, the local temperature variation over a passage in its pitchwise direction was averaged. Figure 6 exemplifies the reconstruction of the oil-side temperature. Firstly, the temperature of the surface cells of an oil passage was averaged in the x -direction. With the averaged values, the oil-side temperature was interpolated linearly to be referred to from the air-side.

Grid Convergence

Grid convergence in air-side

Grid convergence in the air-side was investigated using the same computational domain as the turbulence intensity study. Inflow air had a turbulence intensity of 1%. Three grids are tested; the respective minimum grid widths on the fin sidewall were 1.2×10^{-6} m, 1.5×10^{-6} m, and 2.0×10^{-6} m. Figure 7 shows the coarsest mesh. The local heat dissipation from the air-side fin, $Q'(x)$, was compared among the three cases to check the grid independency.

$$Q'(x) = \frac{\int_{\text{bottom}} q \, dA + \int_{\text{side}} q \, dA + \int_{\text{tip}} q \, dA}{\Delta x \cdot W} \quad (2)$$

where q is heat flux, Δx is the width of the integration area, as shown with shade in Fig. 8, and subscripts “bottom”, “side”, and “tip” denote the passage bottom, the fin sidewall, and the tip, respectively.

Figure 9 shows the local heat dissipation acquired from each grid. The distribution of the dissipated heat almost coincides with each other. On this basis, the grid dependency was considered small enough, and the grid with the minimum width of 1.5×10^{-6} m was adopted for the following analysis.

Grid convergence in oil-side

Figure 10 shows the oil passage and grid topology. The passage had the same length as the U-shape circuit in Fig. 5. Based on the preliminary study on the crossflow

system, the temperature on the top of the solid structure was given linearly from 400 K at the inlet to 370 K at the outlet. The other boundaries were set to be adiabatic. Half pitch analysis with five grids was conducted; the respective minimum grid widths were 1.0×10^{-6} m, 2.0×10^{-6} m, 3.0×10^{-6} m, 4.0×10^{-6} m, and 8.0×10^{-6} m.

Figure 6 Reconstruction of Oil-side Temperature

Figure 7 Grid Topology of Air-side Domain, Left: Overall View, Right: Streamwise Slice in the Vicinity of the Fin

Figure 8 Control Surface on Cooling Fin for Calculation of Local Heat Dissipation

Figure 9 Comparison of Local Heat Dissipation with respect to Each Minimum Grid Width

Figure 10 Configuration of Oil-side Grid Test, Left: Geometry, Right: Streamwise Grid Slice

Figure 11 shows the bulk temperature, T_{bulk} , of each mesh. The distribution is seen to converge monotonically. Considering the tradeoff between the computational cost and the precision, the grid with the minimum width of 3.0×10^{-6} m was adopted.

Passage Number Convergence for the Crossflow Analysis

On the basis that a more precise solution is obtained with the more analysis passages, the convergence for the number of passages was checked. Table 4 shows the tested passage numbers. The passage number of the oil-side is the sum of the number of the inflow and the returning passages. The convergence was investigated by checking the local heat dissipation from the air-side fins located at the y -axis positive end (denoted as “+ y ”) and the negative end (denoted as “- y ”), which were the analysis targets regardless of the cases. The local heat dissipation of each case is shown in Fig. 12. The differences between the distributions obtained from Case A and Case B are small enough. In the latter part, the results of Case A are shown.

RESULTS AND DISCUSSION

Impact of the Air-side Inflow Condition

Impact on the flow field

As Tomoda et al. (2013) showed, the inlet separation is one of the characteristic phenomena of SACOC. Figure 13 shows the turbulence energy distribution and the

Figure 11 Bulk Temperature (Grid Dependency)

Table 4 Passage Numbers

	Case A	Case B	Case C	Case D
Air-side	6	5	4	3
Oil-side	10	8	6	4

— - y , Case A — - y , Case B — - y , Case C — - y , Case D
 - - + y - - + y - - + y - - + y

Figure 12 Local Heat Dissipation (Effect of Passage Number)

streamlines in the vicinity of the fin leading edge on the plane $z/H = 0.5$. The turbulence intensity and the swirl angle are denoted as I and θ , respectively. It is found that the vortex size or the turbulence energy distribution is not drastically changed by the inlet turbulence, and the inlet turbulence intensity does not affect the flow phenomena. On the other hand, the flow swirling generates a large separation on the “suction side” of the fin and generate large turbulence energy, which contributes not only to the aerodynamic loss but also to the deterioration of the cooling performance as discussed later.

The previous study by Sekoguchi et al. (2015) showed the reduction in the mass flow rate is not preferable. The mass flow rate, $\dot{m}(x)$, is calculated as the sum of the mass flux through the control surface at each x -position as shown in Fig. 14.

$$\dot{m}(x) = \int \rho u \, dA/W \quad (3)$$

Figure 15 shows the mass flow distribution in the streamwise direction for various turbulence intensity cases. The inflow turbulence makes the boundary layer on the fin sidewall thick and reduces the mass flow rate. However, the mass flow rate becomes insensible to the inlet turbulence intensity when $I \geq 7\%$. This tendency at highly turbulent conditions is possibly dependent on the turbulence characteristic length.

Figure 16 shows the mass flow distribution in the streamwise direction for several swirl angles. The circles show the products of the mass flow variation of the case $\theta = 0^\circ$ and $\cos 25^\circ$. The mass flow rate is reduced by flow swirl not only through the decrease in the lengthwise velocity component of the uniform flow, which is proportional to $\cos \theta$, but also through the blockage of the deformed separation vortex.

Figure 13 Turbulence Energy and Streamlines on the Mid-height Slice at the L. E.

Figure 14 Control Surface for Mass Flow Rate Calculation

Figure 15 Mass Flow Rate Distribution (Impact of Turbulence Intensity)

Figure 16 Mass Flow Rate Distribution (Impact of Swirl Angle)

Impacts on the performances

Figure 17 shows the local heat dissipation on the cooling fins for various turbulence intensity cases. The cooling performance is almost the same for all the cases because the heat capacity of the air in the fin channel keeps similar value as shown in Fig. 15. Due to the small fin pitch, the flow cannot keep strong shear in the channel. Moreover, the large fin length makes the flow insensitive to any local impacts. Consequently, no characteristic flow phenomenon is found in the fin channel among the cases, and the inflow turbulence intensity has little impact on the cooling performance.

Figure 18 shows the local heat dissipation for different swirl angles. The differences among the cases are visible in the downstream region. As Fig. 13 shows, the air flows along the fin almost all over the channel, even near the fin tip with the swirled inflow. Hence, only the heat capacity reduction in Fig. 16 is the potential cause. The difference is still small, and the swirling is also thought to have little impact on the cooling performance.

The impact of inflow condition on the cooling performance and the aerodynamic performances are evaluated in the following part. Total heat dissipation, Q_{tot} , is calculated as the integral of the local heat dissipation.

$$Q_{tot} = \int_0^L Q'(x) dx \quad (4)$$

For the evaluation of aerodynamic performance, the total pressure loss and the drag to the cooling fins are used for the swirl study and the turbulence intensity study, respectively. The total pressure loss is calculated as

$$\Delta(\rho u P_t) = (\int_{in} \rho u P_t dA - \int_{out} \rho u P_t dA)/W \quad (5)$$

where ρ is density, u is streamwise velocity component, and P_t is total pressure. The subscripts, “in” and “out”, denote the inlet and outlet boundaries of the

Figure 17 Local Heat Dissipation (Impact of Turbulence Intensity)

Figure 18 Local Heat Dissipation (Impact of Swirl Angle)

Table 5 Thermal and Aerodynamic Performance (Impact of Turbulence Intensity)

I	1%	3%	7%	15%
Q_{tot} [W/m]	27424	27320	27183	27064
vs. $I = 1\%$	-	-0.4%	-0.9%	-1.3%
F_x [N/m]	136.0	139.6	145.5	157.0
vs. $I = 1\%$	-	+2.6%	+7.0%	+15.4%
$(F_x)_n$ [N/m]	50.7	54.4	60.4	70.8
$(F_x)_f$ [N/m]	85.2	85.2	85.1	86.2

Table 6 Thermal and Aerodynamic Performance (Impact of Swirl Angle)

θ	0°	5°	15°	25°
Q_{tot} [W/m]	27424	27393	27139	26419
vs. $\theta = 0^\circ$	-	-0.1%	-1.0%	-3.7%
$\Delta(\rho u P_t)$ [J/m s]	15526	15570	15793	15907
vs. $\theta = 0^\circ$	-	+0.3%	+1.7%	+2.5%

computational domain, respectively. The drag on the fin, F_x , is evaluated as the sum of the pressure drag, $(F_x)_p$, and the shear drag, $(F_x)_f$. The drag is calculated in the x -direction, which is the momentum direction of the exhaust jet. The drag is not used for the evaluation of swirling, because the streamwise velocity component of the airflow was not kept constant. Likewise, the pressure loss is not used for the turbulence intensity study, because the turbulence kinetic energy affected the total energy.

Table 5 shows the total heat dissipation and the drag for the turbulence intensity study. Unlike the cooling performance, the drag on the fin shows a deteriorating tendency with increasing the pressure drag. Strong turbulence is not preferable for SACOC therefore.

Table 6 shows the total heat dissipation and the total pressure loss obtained in the swirl study. The deterioration of the total heat dissipation is smaller than 4% with the swirl angle $\theta=25^\circ$. On the other hand, the evaluation

value of the total pressure loss increases, although the evaluation value of Eq. (5) is dependent on the streamwise velocity component. The cooling fins should be installed along the flow direction from the aerodynamic point of view.

Impact of Oil-side Thermal Boundary Layer

Thermal boundary layer in the oil passage

As shown in Fig. 19, the simulated passages in the air-side are named A ~ F from the $-y$ end to the $+y$ end, and the oil-side circuits are named I ~ V (Roman numerals) in the order from the outside.

Figure 20 shows the temperature distributions of the inner oil circuit and the outer oil circuit. The figure shows the slices at the $-y$ boundary of the inflow passages (denoted as 0% in the figure), the $+y$ boundary of the inflow passages (50%), and the $-y$ boundary of the returning passages (100%). At the outlet of the outer circuit, the temperature difference between the wall and the passage center reaches almost 30 K, which makes the viscosity of the oil 60% larger on the wall. The thermal dependency of the oil property makes the thermal behavior difficult to be simplified, and the result shows the necessity of the numerical resolution of the oil passages. The figure also shows that the passage center remains uncooled. This result implies that mixing techniques may improve the thermal performance of SACOC effectively.

Estimation of the overall cooling performance

Figure 21 shows the local heat dissipation on the air-side cooling fins. Due to the uniform temperature at the oil inlet, Passage A shows the obvious difference in the local heat dissipation. In an actual configuration, the oil is thought to have a thermal boundary layer developed to some extent through a passage from heat sources, and the heat transfer near the inlet port is possibly suppressed than the distribution of Passage A in Fig. 21.

To estimate the overall cooling capacity, the local heat dissipation of the area between simulated passages was assumed as the averaged distribution of neighboring passages. Based on the assumption, the total heat dissipation of the system was calculated as 17812 W/m.

Contribution of the air, the oil, and the solid structure to the overall thermal resistance

The amount of transferred heat in a heat exchanger is generally calculated as follows.

$$Q = \psi KA \Delta T_{lm} \quad (6)$$

where ψ is an experimental coefficient, K is overall heat transfer coefficient, A is an installation area, and ΔT_{lm} is logarithmic mean temperature difference. Ignoring the spatial distribution, the inverse of the heat transfer coefficient is estimated as the sum of the thermal resistances of the air, the solid structure, and the oil.

Figure 19 Schematic of Flow Passages, A - F: Air-side Passages, I - V: Oil-side Passages

Figure 20 Streamwise Development of Thermal Boundary Layer in the Oil-side Passages

Figure 21 Local Heat Dissipation (for Channels A - F)

$$\frac{1}{K} \sim TR_{air} + TR_{solid} + TR_{oil} \quad (7)$$

where TR denotes thermal resistance. The contribution ratio of each TR to the overall thermal resistance is estimated by comparing the value of $\Delta T_{lm}/Q$ ($\propto 1/K$) of the following cases: the crossflow system without thermal resistance in the oil or the solid structure, the system without thermal resistance in the oil, and the system with finite thermal resistance.

The total heat dissipation of each case is shown in Table 7. The air-side outlet temperature is calculated as the averaged total temperature at the trailing edge of the fin channel. For the cases without the thermal resistance of the oil, the logarithmic mean temperature difference is calculated on the assumption that the oil is kept at 403.15 K throughout the oil passage.

Finally, the contribution ratio to the overall thermal resistance of each component is calculated as shown in

Table 7 Heat Dissipation with Partial Thermal Resistance

Ignored TR		None	TR_{oil}	$TR_{solid}&TR_{oil}$
Q_{tot} [W/m]		17812	26640	43159
T_{air} [K]	In	(348.15)	(348.15)	(348.15)
	Out	357.53	362.19	369.62
T_{oil} [K]	In	(403.15)	(403.15)	(403.15)
	Out	398.73	(403.15)	(403.15)
ΔT_{lm} [K]		48.06	47.64	43.38

() ... given as boundary condition

Table 8 Estimated Ratio of Thermal Resistance

TR_{air}	TR_{solid}	TR_{oil}
37.3%	29.0%	33.7%

Table 8. The table indicates that the oil has thermal resistance comparable to the air, whereas oil is generally considered to be thermally conductive. From Table 8, there is a potential in improving SACOC with oil-side devices. It should be noted that the thermal resistance of the solid part in the table takes the fin efficiency into account; its smaller value does not necessarily mean better performance.

CONCLUSIONS

In the present study, a numerical analysis was conducted on the simplified system of SACOC to understand the impact of air-side inflow condition and oil-side boundary layer on the performance of SACOC. A simple method was also developed to estimate the performance of a crossflow system by simulating several passages.

Neither inflow turbulence intensity nor swirl angle was found to have a considerable impact on the thermal performance. On the other hand, the aerodynamic performance was deteriorated with highly turbulent or swirled inflow. To achieve optimal performance, the fin array must be installed along the flow field of the operating condition whose aerodynamic losses affect the fuel consumption the most in a flight.

The results showed the validity and the necessity of the newly developed method for a crossflow system. The local heat dissipation was confirmed to converge in the simulation of only several passages. The large temperature gradient was found in the oil passages, which made the viscosity of the oil on the wall about 60% larger than the passage center. The validation of the developed method in comparison with experiments should be a future work.

In the simulation including the oil passages, the oil-side boundary layer was found to have a substantial thermal resistance. On the other hand, most of the oil was found to keep the high temperature at the outlet, which implies that the mixing in the oil-side or other oil-side heat transfer promotion must be effective for improving the performance of SACOC.

ACKNOWLEDGMENTS

This research has been funded by the European Commission (EC) and New Energy and Industrial Technology Development Organization (NEDO), Japan, through the research project “Surface Heat Exchangers for Aero Engines 2 [SHEFAE 2]”, the support of which is gratefully acknowledged by the authors.

REFERENCES

- Ishii, T., Watanabe, T., Himeno, T., Sakuma, Y., and Tomida, S. (2019). Shape Effect of Surface Air Cooled Oil Cooler on Heat Exchange Characteristics. In: *International Gas Turbine Congress 2019*. Tokyo: GTSJ, IGTC-2019-076.
- Liu, J., Peck, J., Yazawa, K., Fisher, T. S., and Shih, T. (2019). Bypass, Loss, and Heat Transfer in Aircraft Surface Coolers. *Frontiers in Mechanical Engineering*, 5, 46. <https://doi.org/10.3389/fmech.2019.00046>
- Morioka, N., Oyori, H., Seki, N., Fukuda, T., and Suzuki, F. (2018). Thermal Management System for the MEE and Engine Embedded Electric Machine. In: *ASME Turbo Expo 2018*. Oslo: ASME, GT2018-76356.
- Outirba, B. and Hendrick, P. (2013). Development and Testing of Advanced Surface Oil Coolers for Aero Engines. In: *International Symposium on Air Breathing Engines*. Busan: Curran Associates Inc., pp. 477–486.
- Sekoguchi, N., Watanabe, T., Himeno, T., Inoue, C., Uzawa, S., Sakuma, Y., Tomida, S., and Watanabe, K. (2015). Numerical Study on Heat Exchange Characteristics of Surface Air Cooled Oil Cooler Focused on Flow Field around Cooling Fin. In: *International Gas Turbine Congress 2015*. Tokyo: GTSJ, pp. 1187–1193.
- Shima, E. and Jounouchi, T. (1997). Role of CFD in Aeronautical Engineering (No. 14) - AUSM Type Upwind Schemes -. *Special Publication of National Aerospace Laboratory*, 34, pp. 7–12.
- Sousa, J., Villafañe, L., and Paniagua, G. (2014). Thermal analysis and modeling of surface heat exchangers operating in the transonic regime. *Energy*, 64, pp. 961–969.
- Tomoda, T., Watanabe, T., Himeno, T., Inoue, C., Tomida, S., and Watanabe, K. (2013). Numerical Study on Radiation Characteristics of Surface Air Cooled Oil Cooler Focused on Flow Field around Radiation Fin. In: *Annual Conference of the GTSJ*. Okinawa: GTSJ, pp. 66–68. (in Japanese)
- Villafañe, L. and Paniagua, G. (2018). Aerodynamic impact of finned heat exchangers on transonic flows. *Exp. Therm. Fluid Sci.* 97, pp. 223–236.
- Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*, 26(11), pp. 1299–1310.
- Yabe, T. and Takei, E. (1988). A New Higher-Order Godunov Method for General Hyperbolic Equations, *Journal of the Physical Society of Japan*, 57, pp. 2598–2601.