Thermal analysis and modeling of surface heat exchangers operating in the transonic regime

J. Sousa*, L. Villafañe, G. Paniagua

Aeronautics and Aerospace Department, von Karman Institute for Fluid Dynamics, Chaussée de Waterloo 72, 1640 Rhode-Saint-Genèse, Belgium

Abstract
Surface coolers for tightly packed airbreathing propulsion are proposed to evacuate the heat loads from the lubrication circuit. This paper demonstrates the capability of integrated bypass-flow surface heat exchangers to cope with the high cooling demands from innovative engine architectures. The present publication describes the experimental and numerical methodology to model the thermal performance of a surface cooler within an aero-engine. Experiments were carried out in a dedicated facility that reproduced transonic engine conditions, allowing the determination of surface temperature distribution with infrared thermography. The thermal convective process was characterized by means of an ad-hoc three dimensional inverse heat conduction approach. An unprecedented energy model was then developed to analyze the sensitivity of the heat exchanger capacity to different engine operating conditions. The results indicate that the investigated concept may provide up to 76% of the estimated lubrication cooling requirements during take-off of a modern gas turbine power plant.

1. Introduction
Lubrication heat load is of particular relevance in the design of any future energy related system. The architectural complexities of novel propulsion engines [1] and the growing importance of electronic devices [2] are drawing the current oil cooling capacity to its operational limit [3]. Hence, adequate thermal management of the engine system is mandatory to allow the continuous evolution towards more efficient and less polluting aero-transportation systems.

Future aircraft engine designs aiming to decrease specific fuel consumption, such as the open rotor, geared turbofan, and contra-rotating turbofan, are characterized by compact gas generators. The additional mechanical transmission components (gear boxes, bearings) increase the heat dissipation to the lubricant [1], hence saturating the currently available cooling sources. Fuel is the primary cooling source due to its high density and thermal conductivity, allowing the design of compact Fuel Cooled Oil Cooler heat exchangers. In addition fuel/oil coolers do not affect the aerodynamic performance of the engine and allow regenerative cycle modification. However, the capacity of the fuel as a cold source is limited by the maximum allowed temperature in the tank [4] which is a particular concern for future high speed and all-composite airframes [5]. When this limit is reached it becomes imperative to explore alternative heat exchange methods.

In the present work, the use of aerodynamically optimized air cooled oil cooler (ACOC) heat exchangers is considered as an alternative to ensure the correct operation of all the mechanical components, within the bypass duct of a turbofan engine during the flight envelope, keeping oil and fuel within its functional limits. Although, this concept has been proposed in various patents [6,7], it has been scarcely addressed in the public literature. In a similar context, Filburn et al. [8] presented an oversimplified analysis of the heat released by finned heat exchangers. The literature is profuse in heat exchangers with extended surface or finned heat exchangers. The literature is profuse in finned heat exchanger performances in high-speed flow applications, and references to finned heat exchanger performances in high-speed flows are scant. The present paper focuses on the aero-thermal process induced within a surface integrated finned air/oil cooler immersed in a transonic bypass flow, as shown in Fig. 1a).

The bypass flow downstream of the fan is characterized by transonic flow velocities and swirl angles ranging between 40 and 50°. The heat exchanger model was tested in a unique three dimensionally shaped transonic wind tunnel, represented in
Fig. 1b) and c), that duplicates the core/bypass flow separator (splitter) at real scale and capable of reproducing the streamlines curvature of the engine representative flow.

The literature dedicated to this type of aero-thermal process at transonic regimes is scarce in contrast with multiple studies performed at low Reynolds and Mach numbers [11–13]. Hence, the present transonic tunnel with infrared optical access allowed unprecedented analysis of the convective process in high speed flows.

The three dimensional shaped surface cooler model is depicted in Fig. 2a). During the 20–40 s test the internal heat conduction process is complex, where typical mono or bi-dimensional assumptions [14] would fail. For this reason, an innovative processing technique based on a transient three dimensional inverse heat conduction method (IHCM) had to be employed. This technique, coupled with robust finite element solvers and thermal imaging methods, models complex geometries by considering the full three dimensional (3D) heat transfer problem.

The inverse heat conduction problem is mathematically ill-posed due to the instability created by the unavoidable noise in the measurements. Several mathematical regularization techniques were developed in the past to allow for a stable solution of these problems [15]. In the present research, an iterative regularization, proposed by Alifanov [16], was coupled with a three dimensional finite element solver (Comsol®) [17]. Prior to the final

---

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>coefficient of exponential fit</td>
</tr>
<tr>
<td>$b$</td>
<td>coefficient of exponential fit</td>
</tr>
<tr>
<td>$K_f$</td>
<td>thermal conductivity of the fluid (W/m K)</td>
</tr>
<tr>
<td>$h$</td>
<td>convective heat transfer coefficient (W/m² K)</td>
</tr>
<tr>
<td>$Ma$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandl number</td>
</tr>
<tr>
<td>$Q$</td>
<td>wall heat flux (W/m²)</td>
</tr>
<tr>
<td>$R^2$</td>
<td>quality of the linear fit</td>
</tr>
<tr>
<td>$R$</td>
<td>objective function</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature (K)</td>
</tr>
<tr>
<td>$Y$</td>
<td>measured temperature (K)</td>
</tr>
<tr>
<td>$V$</td>
<td>flow velocity (m/s)</td>
</tr>
</tbody>
</table>

**Greek symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\eta$</td>
<td>heat exchanger efficiency</td>
</tr>
<tr>
<td>$\rho$</td>
<td>fluid density (kg/m³)</td>
</tr>
</tbody>
</table>

**μ** fluid viscosity (N s/m²)

**Acronyms**

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACOC</td>
<td>air cooled oil cooler</td>
</tr>
<tr>
<td>IGV</td>
<td>inlet guide vanes</td>
</tr>
<tr>
<td>IHCM</td>
<td>inverse heat conduction method</td>
</tr>
<tr>
<td>IR</td>
<td>infrared</td>
</tr>
<tr>
<td>PIX</td>
<td>pixel</td>
</tr>
</tbody>
</table>

**Subscripts**

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>total condition</td>
</tr>
<tr>
<td>ad</td>
<td>adiabatic condition</td>
</tr>
<tr>
<td>b</td>
<td>base</td>
</tr>
<tr>
<td>exp</td>
<td>experimental</td>
</tr>
<tr>
<td>in</td>
<td>inlet</td>
</tr>
<tr>
<td>max</td>
<td>maximum temperature</td>
</tr>
<tr>
<td>out</td>
<td>outlet</td>
</tr>
</tbody>
</table>

**Fig. 1.** Illustration of the ACOC location with a detailed view of the investigated domain, a); artistic view of the three dimensional shaped test section, b); image of the experimental test section, c).
application, the methodology used to process the infrared experimental measurements was numerically validated and the expected uncertainties quantified.

The experimental data and the numerical model of the heat exchanger were then used to evaluate the increase in cooling capacity of the power plant at one specific operating point set by the experimental conditions. However, during an aircraft flight mission, the aero thermal conditions vary according to the altitude and flight Mach number and consequently affect the cooling capacity of the heat exchanger. Therefore, a turbo-fan engine model (available in GasTurb) was used to estimate the flow properties in the bypass section during a typical commercial aircraft mission. By scaling the convective properties of the flow to the estimated aero-thermal conditions the heat conduction process was numerically reproduced and the sensitivity of the surface cooler determined. Finally, different opportunities for future highly efficient power plants, offered by the present heat exchanger, were addressed.

2. Research methodology

The characterization of the convective heat transfer process on the fins was the target of the experimental research. The transient measurements of the wall temperature with infrared thermography and the corresponding numerical determination of the heat flux values with the inverse approach allowed the estimation of the adiabatic wall temperature and adiabatic convective heat transfer coefficient. These parameters are invariant descriptors of the convective process and therefore allow the numerical evaluation of the cooling performance at different engine flow conditions.

2.1. Wind tunnel test section

The experimental research was performed in a unique facility designed to reproduce the flow conditions within the by-pass section of a turbofan engine at the aircraft take off condition. Fig. 2b) illustrates the helicoidally shaped test section able to duplicate the radial and longitudinal flow gradients [18]. A set of inlet guide vanes (IGV) defines the radial distribution of the flow turning, identical to the fan outlet conditions. The walls were designed following flow streamlines, thus smoothly guiding the flow along the test section. The air under the splitter representing the engine core flow is exhausted to the atmosphere. The main flow stream evolves over the splitter, interacting with the array of fins composing the air heat exchanger, and it is later evacuated to the atmosphere after being decelerated in a diffuser.

The surface cooler model was manufactured of Aluminum alloy (6061-T6). It consists of 10 rectangular fins of 225 x 15 mm, with a thickness of 1 mm and a pitch of 10 mm. Five electrical heaters (with a resistance of 3.6 Ohm and an effective heating area of 43.2 cm² each) were attached to the inner surface of the model. The voltage was imposed by an external power supply and its magnitude was recorded in order to estimate the heating power. The surface of the investigated model in contact with the bypass flow was painted in black and instrumented with self-adhesive type T fine wire thermocouples. The model together with the surface thermocouple was introduced in a large electrical oven, equipped with bare thermocouples suspended in the vicinity of each surface thermocouple. This approach allowed the in-situ calibration of the surface thermocouples. The accuracy of the temperature measurements was ensured within ±0.5 K. On the other hand, the use of the surface thermocouples, together with the model electrical heaters, allowed an in situ calibration of the infrared camera [19] providing an estimated calibration uncertainty of ±1.5 K. As displayed in Fig. 3, a germanium window of 80 mm in diameter was assembled in the lateral wall of the test section to provide the required infrared optical access. The used infrared camera (FLIR SC3000) has a spectral range 8–9 μm with a thermal sensitivity of 20 mK (at 303 K) and a stated accuracy of ±1 K (for measurements up to 423 K). The images were acquired at a sampling rate of 50 Hz with a full resolution of 320 by 240 pixels (PIX). Additionally, total flow pressure and temperature were acquired at several axial locations by means of combined Pitot and thermocouple probes. Wall static pressures were recorded along the splitter center line upstream and downstream of the heat exchanger. The uncertainty on the pressure measurements was lower than 1 mbar, and total flow temperature measurements were accurate within ±0.3 K.

2.2. Convective research approach

Moffat [20] proposed the use of the adiabatic heat transfer coefficient (\(h_{ad}\)) as an invariant descriptor of the forced convection in high speed flows, equation (1):

\[
\frac{Q}{T_1 - T_0} = h_{ad} \cdot A
\]
The adiabatic heat transfer coefficient is only a function of the fluid dynamics dictated by the geometry and fluid properties, while being independent of the thermal boundary conditions of the problem [21]. Additionally, the adiabatic wall temperature \(T_{wall}\) provides an estimation of the local total temperature of the flow at the edge of the boundary layer. As illustrated in Fig. 4a), these two parameters can be experimentally determined by monitoring the surface temperature and computing the respective heat flux. Let us consider that the model is preheated (with an heat flux \(Q_b\) imposed by the heaters on the base surface) to an initial value above the flow temperature, such that when the test begins, with a sudden blow-down at the time \(t_0\), the model is at temperature \(T_{0}^{f}\), with a convective heat flux \(Q_{fl}\) established by the flow field. Immediately after, the heaters were turned off \((Q_b = 0)\), thus the model cooled down. Therefore, at instant \(t_2\), the surface temperature is lower and correspondingly the heat flux diminishes. Fig. 4b) represents the measured heat flux plotted in function of the wall temperature at any given instant. By performing a linear fitting and extrapolating the heat flux—temperature curve to a null heat flux value (adiabatic condition), the adiabatic heat transfer coefficient is determined by the gradient of the fitted curve, while the adiabatic wall temperature corresponds to the intersection with the abscissa. In the present application, the wide variation of the monitored wall temperature during a test was sufficient to perform a reliable linear fit.

2.3. Numerical data processing

The heat flux distribution was estimated by means of a three dimensional IHCM [17]. The algorithm uses an iterative regularization procedure where an optimization technique [22, 23] performs the minimization of the objective function as defined by equation (2):

\[
R(Q|x, y, t|) = \int_{t_1}^{t_2} \sum_{x} \sum_{y} [T(x, y, t) - Y(x, y, t)]^2 dt
\]

Equation (2) quantifies the squared difference between the measured temperature \(Y(x, y, t)\) and the estimated one \(T(x, y, t)\), from the beginning \((t = t_1)\) to the end of the test \((t_2)\). At each iteration the solver computes a new heat flux \(Q(x, y, t)\) that minimizes the objective function \(R(Q|x, y, t|)\). The corresponding estimated wall temperature \(T(x, y, t)\) is then obtained by computing the direct problem. The iterative process is repeated until a certain convergence threshold \([R(n_i) - R(n_i - 1)]/R(n_i) < \epsilon\) is reached, where \(n_i\) is the last iteration value. For the present study \(\epsilon\) was chosen to be equal to \(2.5 \times 10^{-4}\), as a compromise between computational time and accuracy of the minimization procedure.

As sketched in Fig. 5, the slight curvature of the model was considered negligible in the numerical domain to simplify the post-processing procedure. The infrared measurements were performed on the central fin where the flow and the heat transfer process were considered to be insensitive to the presence of the lateral walls of the wind tunnel. Hence, only one of the fins was modeled and periodic boundary conditions were imposed on the lateral adjacent surfaces. An adiabatic condition was imposed in the base wall (since the heaters were turned off following the blow down). Fig. 5b) depicts a raw infrared image and the result of the transformation, which is then used as an input to the inverse method. Due to the constrained visibility, shown in Fig. 5b), the objective function could only be evaluated over the visible part of the domain, restricting accurate heat flux results to this region.

3. Results and discussion

3.1. Numerical validation

Prior to the final application, the accuracy of the proposed data processing technique was evaluated by means of a numerical validation. A direct transient heat conduction problem was set up by imposing an initial wall temperature (initial condition) and convective cooling over the wet surfaces (imposing \(h_{exact}\) and \(T_0\) in Comsol\(^8\)). A preliminary and rather simplified experimental data processing allowed a fast estimation of the heat flux levels and respective spatial distribution. These values provided the basis for the numerical inputs. Fig. 6 displays the numerical flow temperature \((T_0 = 280 \text{ K})\) and heat transfer coefficient \((h)\) imposed on the wet surfaces. Strong spatial gradients were imposed to test the behavior of the data processing approach. Besides the periodic and wet surfaces the remaining boundaries were considered as adiabatic.

The numerical solution provided transient temperature distribution along the visible surface. In order to reproduce the experimental procedure, these temperatures were assumed to be the ones measured by the infrared camera and therefore used as an input to the IHCM solver. The estimated transient heat flux values by the inverse method and the temperature maps allowed, as described in Section 2.2, the determination the convective properties \((h = h_{ad})\) and adiabatic wall temperature \((T_{wall})\) of the heat transfer process.

---

**Fig. 4.** Temperature evolution of the model during the test, a) linear fitting of the heat flux \((Q)\)–temperature \((T)\) curve, to compute the adiabatic heat transfer coefficient \((h_{ad})\) and adiabatic wall temperature \((T_{wall})\), b).
The uncertainty quantification was based on the difference between the estimated values \( (h, T_0) \) by the inverse approach and exact values \( (h_{\text{exact}}, T_{0\text{exact}}) \) imposed in the direct numerical problem, as expressed by equation (3). Fig. 7 presents the convective heat transfer coefficient along the lateral surface. The exact heat transfer coefficient distribution \( (h(x,y)_{\text{exact}}) \) is depicted in Fig. 7a). The estimated values, obtained by the inverse approach \( h(x,y)_{\text{estimated}} \), is plotted in Fig. 7b) following equation (3). The flow temperature was computed with the method described in Section 2.2 with accuracy within 4 K. The uncertainty of the present method \( (h(x,y)_{\text{error}}) \) was only increased at the vicinity of the boundary at \( X = 70 \) PIX. The results on the non-visible area \( (X > 50 \) PIX) were discarded since the lack of input information did not allow for the correct estimation of the heat transfer coefficient. Additionally, the results for \( X > 50 \) Pix were also ignored given the higher uncertainty induced by the proximity of the non-visible boundary \( (X = 70 \) PIX). However, it was proved that the lack of information in the non-visible area did not affect the remaining part of the computed results in the visible area which is related with the fact that the conductive heat flux travels mainly in the \( Y \) direction. The same procedure was used to compute the convective characteristics in the bottom surface. It was conclude that the error of \( h \) on this wall was approximately 10%, while the estimated flow temperature presented a maximum deviation of 2 K.

\[
 h(x,y)_{\text{error}} = \frac{|h(x,y) - h(x,y)_{\text{exact}}|}{h(x,y)_{\text{exact}}}
\]  

(3)

3.2. Experimental convective results

The model was initially heated to 318 K by imposing a total heating power of approximately 1 kW/m², to maximize the heat flux values and the respective wall temperature variation during the test. Fig. 8 displays the recorded transient test conditions. The flow temperature was computed with the method described in Section 2.2 with accuracy within 4 K. The uncertainty of the present method \( (h(x,y)_{\text{error}}) \) was only increased at the vicinity of the boundary at \( X = 70 \) PIX. The results on the non-visible area \( (X > 70 \) PIX) were discarded since the lack of input information did not allow for the correct estimation of the heat transfer coefficient. Additionally, the results for \( X > 50 \) Pix were also ignored given the higher uncertainty induced by the proximity of the non-visible boundary \( (X = 70 \) PIX). However, it was proved that the lack of information in the non-visible area did not affect the remaining part of the computed results in the visible area which is related with the fact that the conductive heat flux travels mainly in the \( Y \) direction. The same procedure was used to compute the convective characteristics in the bottom surface. It was conclude that the error of \( h \) on this wall was approximately 10%, while the estimated flow temperature presented a maximum deviation of 2 K.

\[
 h(x,y)_{\text{error}} = \frac{|h(x,y) - h(x,y)_{\text{exact}}|}{h(x,y)_{\text{exact}}}
\]  

(3)

The data processing was started by performing the infrared image perspective transformation and the respective camera calibration. Fig. 9 illustrates the evolution of the processed temperature map at three distinct instants.

The inverse solver was fed with the initial conditions and the temperature distribution measured over the visible part of the lateral and bottom surfaces. The transient three dimensional inverse computations were performed for a time step of 0.05 s on a domain containing 9450 tetrahedral elements. The iterative process was stopped after \( n_f = 29 \) iterations. The convergence value was equal to \( R(n_f) \leq 2 \times 10^{-6} \). Fig. 10 displays the quality of the minimization process by plotting the measured temperature \( Y \) and the estimated temperature \( T \) computed based on
the guessed heat flux at the last iteration, for two different locations in the domain.

The adiabatic conditions were computed with a linear fitting between the local temperature and heat flux at each point. Fig. 11a shows the adiabatic heat transfer coefficient distribution on the visible part of the lateral wall. It is possible to observe a gradual decrease of adiabatic heat transfer coefficient in the streamwise direction due to the development of the boundary layer. Fig. 11b represents the adiabatic wall temperature surface distribution. One observes a decrease of the adiabatic wall temperature in the streamwise direction with a maximum variation of 4 K. At the leading edge, the value is similar to the total flow temperature measured at the inlet of the test section (286 K).

Fig. 12 shows the axial distribution of $h_{ad}$ along the flow direction at three different heights over the lateral wall. Experimental results are represented by the discrete symbols, while dashed lines illustrate an interpolation based on an exponential fitting of the measured data ($h_{ad} = a x^2$) in accordance with a generally accepted Nusselt number ($Nu$) dependence with the Reynolds number ($Re = N u = Re^{1/2}$) [24]. The fitted curve was extrapolated to the full length of the fin (225 mm). The results were consistent with the typical streamwise dependence of $h$ over a flat plate ($h \sim X^{-0.2}$, [24]) in the presence of a turbulent boundary layer. At the mid height of the fin the value of the heat transfer coefficient is 1250 W/m² K close to the stagnation region where the boundary layer starts to develop. It decreases exponentially ($\sim X^{-0.16}$) down to 700 W/m² K after 75 mm due to the thermal isolation caused by the increase of the boundary layer thickness. Because of the higher flow velocities, the heat transfer coefficient increases along the height of the fin.

Fig. 13 depicts the heat transfer coefficient ($h_{ad}$), the adiabatic wall temperature ($T_{ad}$) and the respective quality of the linear fitting ($R^2$) at the bottom surface. One can notice a continuous increase of the adiabatic wall temperature in the streamwise direction. The adiabatic heat transfer coefficient augments slightly in the vicinity of the fin's leading edge and decreases within the fins passage. Due to the lack of visibility the results in the inter-fins space could not be measured and therefore no convective results are presented for this region. Additionally, the numerical validation proved that the results predicted in the visible part were not affected by the non-existent information in the non-visible part of the model.

4. Integration analysis of the surface heat exchangers in compact power plants

4.1. Cooling capacity and thermal performance of surface heat exchanger at aircraft take off condition

Based on the adiabatic convective heat transfer coefficient, a numerical analysis was performed to evaluate the cooling capacity of the heat exchanger operating at engine conditions, in particular when the aircraft takes off. The total flow temperature was set to 364 K while a typical lubrication oil temperature value of 433 K [4] was considered. The simulations assumed the occurrence of forced convection on the lubricant surface, considering an oil velocity of 1 m/s ($h \sim 2000 W/m²K$). The oil system assumptions, regarding the flow velocity and pipe diameter, are based on the recommended values from Neale [25], while the heat transfer coefficient is based on the correlation ($Nu_0 \sim 0.023Re^{0.5} Pr^{0.3}$) from Incropera [24]. The steady state case was computed using as boundary conditions the previously determined convective heat transfer coefficients both on the lateral and bottom surfaces (Fig. 12).

Fig. 14a displays the local temperature difference of the model, relative to the maximum value obtained in the domain ($T_{max} = 396 K$), at a cut plane along the center line of the fin. Fig. 14b shows a detailed view of the leading edge temperature distribution including the internal heat conduction vectors. Similarly, Fig. 14(c) presents a transversal cut of the fin displaying a three dimensional heat conduction process. The temperature plots show a considerable reduction of the fins temperature with the height, attaining a maximum difference of about 25 K. This phenomenon induces an important decrease of the fins thermal efficiency ($\eta$).

The fin’s thermal efficiency is defined as the ratio between the actual heat flux ($q$) released by the surface and the hypothetical one if the entire fin surface would have a constant temperature equal to the base surface ($T_b$), resulting in the largest possible heat flux value [26]:

$$\eta = \frac{Q}{(T_b - T_0)h} = 46\%$$

The bottom temperature was obtained by averaging the value over the full bottom surface (at $y = 0$ in Fig. 14), while $T_0$ is the total flow temperature. The present case revealed an averaged heat flux through the oil surface of about 82 kW/m². For the studied ACOC configuration, this represents a total lubrication cooling capacity of 65 kW. According to Streinfinger [4], the heat to oil load of an 80 kN thrust engine is approximately 55 kW at take off. The Bearings
(40%) and the gear boxes (32%) represent the most demanding components. Additionally, one has to account for the electrical systems such as the integrated drive generator with a reported heat rejection of typically 30 kW [4]. Therefore, it is estimated that the proposed air cooler concept would provide 76% of the total oil cooling demand of the exemplified turbofan engine, accepting the applied oil boundary conditions.

4.2. Sensitivity of the air cooler to the aircraft flight conditions

The efficiency and cooling capacity of the heat exchanger are functions of the aero-thermal properties of the flow across the by-pass channel. These air flow characteristics vary according to the aircraft flight envelope due to the variation of altitude and flight velocities. Hence, the sensitivity of the surface cooler to the different flight conditions must be evaluated to ensure compliance with the cooling requirements.

At higher altitudes, the air flow temperature decreases; however, the lower air density \((\rho)\) and pressure reduce the convective capacity of the fluid. By contrast, flying at higher speeds augments the flow temperature \((T_0)\) and velocity \((V)\) in the by-pass section. Fig. 15 shows the competing effects of flight Mach number and altitude on the efficiency and cooling sensitivity of the heat exchanger. The flow characteristics in the by-pass duct were computed with a turbo-fan engine model, with a by-pass ratio of 6, provided by GasTurb. This model computes the aero-thermal characteristics of the flow at different sections of the engine both for design and off design conditions as encountered during an aircraft mission. By imposing a certain flight envelope, the flow conditions at the bypass section were estimated. The convective heat transfer coefficient was corrected based on a simplified ratio between the \(h\) obtained in the experiments and the one corrected to the different analyzed flow conditions as shown in equation (5).

Following the results obtained in Fig. 12, a turbulent Nusselt

![Fig. 10](image1)

Fig. 10. Comparison between the measured \((Y)\) and the estimated \((T)\) temperature profiles at the bottom surface, a); and at the lateral wall b) at the final iteration.

![Fig. 11](image2)

Fig. 11. Experimental results for the adiabatic heat transfer coefficient \((h_{ad})\), a); adiabatic wall temperature map \((T_{ad})\) measured at the lateral wall, b).

![Fig. 12](image3)

Fig. 12. Exponential fitting of the heat transfer coefficient in accordance with the development of a turbulent boundary layer \((h_{ad} \sim X^{-0.2})\).

![Fig. 13](image4)

Fig. 13. Evolution of the adiabatic heat transfer coefficient \((h_{ad})\), adiabatic wall temperature \((T_{ad})\) and quality of the linear fitting \((R^2)\) at the bottom surface.
correlation, \( Nu \propto Re^{4/5} Pr^{1/3} \) \([24]\) was proposed to build up the estimated ratio:

\[
\frac{h_{\text{engine}}}{h_{\text{exp}}} = \frac{\left( \frac{\nu V}{T} \right)^{4/5} Pr^{1/3} K_l}{\left( \frac{\nu V}{T} \right)^{4/5} Pr^{1/3} K_l} \tag{5}
\]

The thermal simulations were repeated for each flight condition, by imposing the expected \( h_{\text{engine}} \) and the flow temperature at the bypass, while the oil assumptions \( (T_{\text{oil}} \text{ and } h_{\text{oil}}) \) were kept constant. The sensitivity was obtained by comparing the heat removed from the oil surface. The results were normalized by the maximum cooling capacity obtained. At sea level, the cooling penalty was clear when flying at higher speeds. The results show an estimated deficit of 30% at \( M_a = 0.8 \) where the increase of the total flow temperature played a major role. However, this negative gradient perished at higher altitudes, maintaining an almost constant capacity from 0.2 to 0.85 flight Mach number. The increase in flow temperature with the higher flight Mach number was counteracted by an increase of the mass flow rate \( (\nu V) \). In Fig. 15b) one can observe that the variation of the efficiency is mainly governed by the altitude, since the consequent decrease of \( h \) led to an important decrease of the thermal gradients. Hence, the model presents a temperature distribution closer to the bottom surface.

The present analysis proves the limited sensitivity of the proposed surface cooler configuration during a commercial flight envelope \( (0 [m] \sim 0.2 [M_a]; 6000 [m] \sim 0.5 [M_a]; 12,000 [m] \sim 0.8 [M_a]) \) varying in a limited range of 10% relative to its maximum capacity. The minimum cooling is reached at the aircraft take off where it was previously shown to provide 76% of the total cooling demand of the engine.

4.3. Opportunities offered by surface heat exchangers to future power plants

The possibility to use a surface air cooler assembled at the bypass section of a turbofan engine provides a considerable thermal advantage for the lubrication cooling system. From an aerodynamic point of view, the measured averaged total pressure losses increased approximately 1% relative to the clean configuration (without the presence of the fins array). These losses were estimated for the full annular section of a turbofan with a by-pass ratio equal to 6. The considerable advantage of the proposed heat exchanger concept becomes even more evident when possible increments in propulsion efficiency are dependent on improved cooling strategies.

- The possible increase of the bypass ratio with the increment of fan diameter requires the integration of an additional gear box \([1]\) to permit the optimal aerodynamic operation of the components (fan, compressor and turbine) connected with the same shaft. The first generation of geared turbofan from Pratt & Whitney \([27]\) uses a gear reduction of 3:1, with a resulting by-pass ratio of 12. Such design provides 16% decrease in fuel consumption compared to conventional fan engines. The new Airbus 320neo engine (110–150 kN of thrust) has an additional 22.4M W (30,000 hp) gear box with a heat dissipation of 67 kW to the lubricant \([27]\).

- The increase of the bypass ratio can also be attained by decreasing the core size. However, this design would diminish the available wet area on the engine casing. As shown by Reulet et al. \([28]\), the cooling of the casing relies on ventilation flow and jet impingement originated from the by-pass flow. Based on the provided engine data \( (T_{\text{vent}} \text{ and } T_{\text{jet}}) \) and the convective cooling performance of the engine casing \([28]\), it was possible to estimate that a core size reduction of 80% would impose an estimated cooling deficit of 3.5 kW assuming a temperature casing of 770 K.

- The use of active core cooling based on a dedicated internal circuit \([29]\) would allow the cooling of crucial components that are not actively controlled and are sized for the hottest engine condition, decreasing the efficiency during less demanding flight conditions. Additionally, the use of internal cooling circuits can grant a reduction of the total cooling mass flow extracted from the last stage of the compressor. Such coolant (up
to 30% of the core mass flow in current aero-engines) bypasses the combustor and therefore it is not able to deliver work over the following turbine stages.

- The reduction of the coolant temperature (purge from the compressor) by taking profit of the secondary flow as proposed in Ref. [30], would allow a considerable gain in cooling potential over the engine hot parts [29]. This could permit a reduction of the total coolant mass flow extracted from the compressor or an increase of the cooling capacity for the same extracted mass flow rate. To evaluate this possibility it was considered an internal coolant flow, purged from the compressor, with a certain forced convection (h = 100 W/m² K) over the internal part of the heat exchanger with a mass flow rate of ~0.4 kg/s and a flow temperature of 590 K. The numerical heat conduction solution has shown a reduction of the coolant temperature of 40 K (using as boundary conditions the experimentally determined convective coefficients). This situation represents an important increase (~20%) of the coolant performance over the combustion and turbine stages (blades, disk) or a decrease of the total coolant mass flow.

The AOC technology is a suitable option to dissipate the required cooling demands, of future compact energy systems design to minimize the specific fuel consumption, with short aerodynamic penalties.

5. Conclusions

This paper presents the experimental procedure and the respective thermal analysis of a surface cooler operating under transonic conditions. The model was tested in a unique facility design to reproduce the flow conditions at the bypass section a turbofan engine. The convective results were obtained by means of infrared thermography and processed with a three dimensional inverse heat conduction methodology.

The experimental results show that the investigated concept provided 76% of a current aero-engine lubricant cooling demand at the aircraft take off condition. A detailed turbofan engine model was used to compute the aerothermal variation of the flow properties in the bypass section during a typical commercial aircraft mission. These results show that the cooling fluctuations were below 10% during the full flight range, revealing the low sensitivity of the heat exchanger to the flight envelope. It was also demonstrated that the investigated heat exchanger concept provides sufficient cooling power to cope with the additional heat generated by state-of-the-art technologies required in the future design of more efficient propulsion systems.

Acknowledgments

The research leading to these results has received funding from the European Community’s Seventh Framework Programme (FP7/2007-2013) for the Clean Sky Joint Technology Initiative under grant agreement n° CSUJ-GAM-SGO-2008.

References


[26] Langston LS. The geared turbofan may be a revolutionary jet engine, but it’s just the latest device to employ precision gearaging. Mech Eng ASME 2013;1: 45–51.

