International Journal of Heat and Mass Transfer 77 (2014) 321–334

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

# Investigation of high-speed bypass effect on the performance of the surface air–oil heat exchanger for an aero engine

Sangkeun Kim [1], June Kee Min [1], Man Yeong Ha *, Changmin Son [1]

*School of Mechanical Engineering, Pusan National University, Busandaehak-ro 63 beon-gil, Geumjeong-gu, Busan 609-735, Republic of Korea*

ARTICLE INFO

*Article history:*
Received 11 December 2013
Received in revised form 10 May 2014
Accepted 13 May 2014

*Keywords:*
Extended surface
Surface cooler
Heat exchanger

ABSTRACT

In the present study, the influence of an air–oil heat exchanger (SAOHE) location and orientation on engine performance is investigated using numerical predictions with a range of geometry options that match experimental data. The airflow in the unit-fin domain of a SAOHE was modeled with rotational periodic boundary conditions. Using the standard $k$–$\varepsilon$ turbulence model, the compressible Reynolds-averaged Navier–Stokes (RANS) equations and energy equation were solved numerically. In order to validate the numerical method, experimental measurements of the velocity profile and the distribution of the pressure and heat transfer coefficient were compared to numerical results. The pressure drop, overall heat transfer coefficient, and velocity profile downstream of the heat exchanger were taken into consideration as performance metrics. An efficient numerical procedure for the installation study of a cooler having a bypass duct was conducted, and important design variables for SAOHE were clearly identified.


## 1. Introduction

Recently, air transportation has expanded rapidly owing to the increasingly globalized world economy. Hence, environmental concerns have also attracted public attention. These circumstances demand a reduction in the operational costs of airplanes in addition to an increase in fuel consumption efficiency of aero gas-turbine engines. For these reasons, various types of heat exchangers are incorporated into gas-turbine engines, such as an intercooler, recuperator, and cooling air cooler [1]. Oil coolers are also used in engines to decrease the temperature of the electric generators and gearboxes located in transmission systems [2].

Extended surfaces have been used in various cooling systems, such as internal turbine blade passages, external gas turbine cooling systems, compact heat exchangers, and electronic component heat sinks. In general, previous studies of extended surfaces sought to investigate the enhancement of the forced convective heat transfer coefficient in channels. Many of these also focused on the effect of the extended surface shape change, the fin arrangement, and the fin dimension on the cooling performance.

Several researchers have investigated the effects of various extended surface shapes and fin arrangements of the duct flow on the cooling performance both experimentally and numerically [3–10]. Robertson et al. [3] carried out an experimental study of the performance of external fins for gas turbine engine component cooling using a transient liquid–crystal technique in a low-speed wind tunnel. These authors considered interrupted and corrugated fin geometries in addition to a flat continuous fin in order to enhance the heat transfer coefficients by disrupting the thermal and velocity boundary layers that develop along the length of the fin. Neely et al. [4] measured the distribution of local heat transfer coefficients on a smooth cylinder and selected cylindrical finned geometries using transient liquid crystal techniques in a heat transfer tunnel with a new mesh heater device. These authors considered the effect of the relative sizes of the fin diameter to the fin array on the heat transfer characteristics. Sparrow and Ramsey [5] carried out an experimental study to investigate the detailed row-by-row heat transfer and pressure drop characteristics of a staggered array of circular cylinders attached perpendicularly to a principle wall and placed in a cross flow of air in a flat rectangular duct. They applied an analogy between the heat and mass transfer to obtain the heat transfer coefficient from mass transfer coefficients measured via naphthalene sublimation techniques. Mutlu and Al-shemmeri [6] carried out experiments to investigate the pressure drop and heat transfer characteristics in continuous and interrupted fins with and without a cross flow of air across a test section. They determined the average heat transfer coefficients and fin efficiencies and reached conclusions regarding the influence of the cross flow on the pressure drop and heat transfer rate

* Corresponding author. Tel.: +82 51 510 2440; fax: +82 51 515 3101.
*E-mail addresses:* sami99@pusan.ac.kr (S. Kim), jkmin@pusan.ac.kr (J.K. Min), myha@pusan.ac.kr (M.Y. Ha), changmin.son@pusan.ac.kr (C. Son).
[1] Fax: +82 51 515 3101.

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.05.025
0017-9310/

**Nomenclature**

| | |
|---|---|
| $Bi$ | Biot number based on the fin thickness |
| $C_p$ | specific heat |
| $d_h$ | hydraulic diameter of bypass-duct |
| $d_t$ | thickness of fins |
| $h$ | heat transfer coefficient |
| $k_f$ | thermal conductivity of air |
| $k_s$ | thermal conductivity of fin |
| $Nu$ | Nusselt number |
| $P$ | pressure |
| $Q$ | heat transfer rate |
| Re | Reynolds number |
| $T$ | temperature |
| $TI$ | turbulent intensity ($u'_{rms}/U$) |
| $U$ | overall heat transfer coefficient |
| $u, v, w$ | dimensionless velocity components in $x$, $y$ and $z$ direction |
| $x, y, z$ | Cartesian coordinates |

*Greek symbols*

| | |
|---|---|
| $\rho$ | density |
| $\mu$ | viscosity |
| $\theta$ | location angle |

*Sub/superscripts*

| | |
|---|---|
| $i, j$ | tensor notation |

of interrupted fins. Sparrow and Liu [7] investigated the heat transfer and pressure drop for a laminar airflow through arrays of in-line or staggered plate segments from numerical solutions of the fluid flow and energy equations. They compared the performance of two types of segmented-plate arrays and with a parallel-plate channel. They found that a staggered array yields better performance than an in-line array under most conditions. Dogruoz et al. [8] gained physical insight into the behavior of square, in-line pin–fin heat sinks from both experimental and numerical research. They developed a two-branch bypass model and compared this experimental data with that obtained from friction and heat transfer coefficients available in the literature for infinitely long tube bundles with a circular cross-section. Neely et al. [9] experimentally investigated the performance of extended fin surfaces for the forced convective cooling of a range of engine component geometries with a cross-flow. They measured the surface heat transfer coefficient distributions of the external finning around non-cylindrical geometries with the cooling performance/mass ratio maximized in aviation gas turbines. They compared their measured data with those for equivalent smooth geometries and also with empirical calculations from the literature. They found that finned geometries are better than those without fins because the increased surface area of the fins more than outweighs the decrease in the local heat transfer coefficient on the fin surface as compared to smooth geometries. Yang and Peng [10] numerically examined a pin–fin heat sink with a non-uniform fin height and confined impingement cooling. They considered the effects of the fin shape of the heat sink on the junction temperature and on the heat transfer near the center of the heat sink. They also demonstrated an optimized non-uniform fin height design.

Other research has considered heat transfer bypass effects [11–15]. Chapman et al. [11] carried out a comparative thermal performance evaluation using aluminum heat sinks made with extruded fins, cross-cut rectangular pins, and elliptical shaped pins in environments characterized by a low air flow. They compared the performance of three fin designs. The overall thermal resistance of a straight fin was lower than those of the other two designs owing mainly to the combined effect of enhanced lateral conduction along the fins and lower flow bypass characteristics where the heat source is localized at the center of the heat sink base plate. Sata et al. [12] carried out a numerical simulation to calculate the flow and temperature fields around a plate fin array subjected to a uniform flow for different values of fin lengths to the half-pitch of the fins and the Reynolds number. They compared their predicted friction resistance and heat transfer values of the fins with those for a developing flow between parallel plates with a uniform inlet flow velocity. They found excellent agreement between these two conditions. They also developed a prediction technique for the cooling performance of a fin array. Jonsson and Palm [13] investigated the influence of the fin height while taking into account the plate fin and strip-fin heat sinks in both inline and staggered arrays. They also tested various widths and heights of wind tunnels for duct Reynolds numbers ranging from 2000 to 14,000. These authors estimated the fraction of the total airflow passing through the heat sink and compared their results to those from a simple physical bypass model based on experimental data. Morega and Bejan [14] numerically examined the appearance of hot spots when the plate fin thickness and height are allowed to increase along the flow direction. They considered fin conduction uncoupled from external convection using a two-dimensional model and conjugate fin conduction and external convection using a three-dimensional model in order to maximize the thermal contact between the package and the coolant. Jeng [15] numerically investigated the effects of the pin–fin thickness and width of a rectangular channel on a forced convection flow with a laminar side-bypass effect. The author proposed an analytical porous model, the Brinkman–Forchheimer model, for a fluid flow and a two-equation model for heat transfer in order to decrease the computational time. The optimal cooling configuration in these analytical models was determined.

As mentioned above, many studies of heat exchangers focused on heat transfer enhancement technology in the incompressible flow regime, typically by introducing novel shapes of extended surfaces. Also, previous studies considered a bypass stream having a relatively low Reynolds number compared to the high-speed flow considered in this study. For the surface air–oil heat exchanger [16] considered in this study, however, the installation location of the heat exchanger also has a considerable influence on the engine "total performance", as it uses the bypass stream for the cooling. As a result, one should carefully design the installation location of the heat exchanger, even with a highly efficient heat exchanger. The main purpose of this paper is to propose efficient and systematic numerical and experimental methods for this type of application. In addition, due to its high speed and large Mach number ($Ma \approx 0.6$), the analysis should be carried out for a compressible flow regime.

In the present study, the influence of the location and orientation of a SAOHE installation on the engine performance, the SFC in this case, is investigated using numerical predictions of a range of geometry options which match experimental data. The pressure drop, overall heat transfer coefficient, and velocity profile downstream of the heat exchanger were taken into consideration as performance metrics. In order to validate the numerical method, an experimental study of a baseline case was carried out and results of the velocity profile, pressure variation, and the distribution of