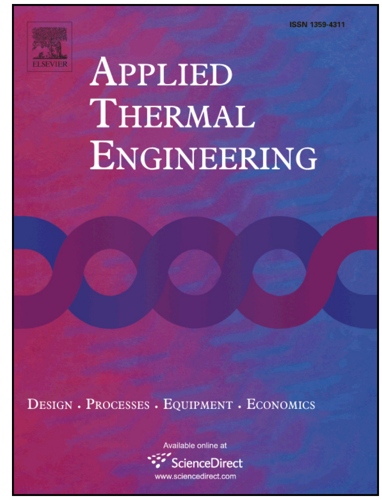
Accepted Manuscript



Numerical Investigation on Various Heat Exchanger Performances to Determine an Optimum Configuration for Charge Air Cooler, Oil and Water Radiators in F1 Sidepods

Pierre Salmon, László Könözsy, Clive Temple, Stuart Grove

PII: S1359-4311(17)30785-8
DOI: http://dx.doi.org/10.1016/j.applthermaleng.2017.02.026
Reference: ATE 9907

To appear in: *Applied Thermal Engineering*

Received Date: 18 June 2016
Revised Date: 1 February 2017
Accepted Date: 5 February 2017

Numerical Investigation on Various Heat Exchanger Performances to Determine an Optimum Configuration for Charge Air Cooler, Oil and Water Radiators in F1 Sidepods

Pierre Salmon[a], László Könözsy[b,*], Clive Temple[a], Stuart Grove[a]

[a] *Centre for Advanced Vehicle Engineering, Cranfield University, Cranfield, Bedfordshire, MK43 0AL, United Kingdom*
[b] *Centre for Computational Engineering Sciences, Cranfield University, Cranfield, Bedfordshire, MK43 0AL, United Kingdom*

Abstract

The present work focuses on a three-dimensional CFD approach to predict the performance of various heat exchangers in conjunction with non-isothermal transitional flows for motorsport applications. The objective of this study is to determine the heat transfer, pressure drop and inhomogeneous flow behaviour for distinct heat exchangers to identify an optimum configuration for the charge air cooler, water and oil radiators placed in the sidepods of a formula one (F1) car. Therefore, a comprehensive analysis of various heat exchanger configurations has been carried out in this work. In order to assess the reliability of the obtained results, a mesh sensitivity study along with additional parametric investigations have been performed to provide numerical parameters predicting accurately a) the heat transfer rate at the fluid-solid interface and b) the sporadic separation. As a result of the performed validation procedure in this study, the aerodynamic- and thermal boundary layer development along with the convective characteristics of the air flow have been captured accurately near to the heated surface. The characterization of a heat exchanger core and a core configuration in a closed domain is also possible with this procedure. The presented three-dimensional CFD approach could overcome the difficulties of macroscopic heat exchanger and porous media methods for F1 applications, because it can be used to predict the heat transfer and pressure drop related to the mass flow rate correlation curves. The contribution of fins to the total heat transfer rate has been predicted theoretically, and application benchmark test cases have been presented to analyze five different heat exchanger configurations in accordance with the 2014 formula one technical regulations. The numerical data extracted directly from three-dimensional CFD simulations can be used in the sidepod design process of the external cooling system of F1 engines.

Keywords: heat exchangers, non-isothermal transitional flows, formula one, charge air cooler, sidepod, fin

1. Introduction

Heat exchangers in automotive and motorsport applications often operate at flow velocities corresponding to Reynolds numbers indicative of the laminar-to-turbulent transitional flow regime [1]. Therefore, robust and accurate numerical simulations are required to predict gradients of all flow field variables correctly. High-resolution meshes and advanced engineering turbulence modelling approaches are capable of capturing sporadic separations and adverse pressure gradients in the boundary layer of heated surfaces. In the field of computational fluid dynamics (CFD), traditional numerical procedures usually employ the porous media method where the implementation of the heat transfer to mass flow rate curves are pre-determined [2, 3]. Due to the constraint of semi-empirical correlations, the porous media approach could loose its accuracy related to those real world applications where sporadic separation and/or non-adiabatic pressure drop is observed. The non-linear nature of heat transfer and pressure drop requires a more accurate simulation approach than the use of semi-empirical correlations or the porous media method. Therefore, the present work focuses on an advanced computational engineering procedure to determine performance curves of various heat exchanger configurations relying on direct three-dimensional CFD heat transfer simulations for a formula one (F1) car sidepod.

An intensive development on different heat exchanger design including air-cooled ones was carried out over the past fifty years. Application of direct CFD methods is playing an important role, because the performance of modern computers is highly increased. Bhutta et al. [4] carried out a detailed review on advanced CFD applications in various heat exchangers design. Wang et al. [5] focused on the investigation of the data reduction method for air-side performance of fin-and-tube heat exchangers, because this method for the air-side heat coefficients were not relying on a consistent approach in the literature. Xie et al. [6] performed numerical parametric studies and obtained multiple correlations on air-side heat transfer and friction characteristics when large number of large-diameter tube rows were considered. Borrajo-Peláez [7] carried out a three-dimensional CFD study to compare the air-side model with the air/water model for a plain fin-and-tube heat exchanger. Koplow [8] proposed a fundamentally novel approach to air-cooled heat exchangers highlighting that CFD data would be very useful in the transitional flow regime for device optimization purposes. Fugmann et al. [9] evaluated the performance of air-based heat rejection system related to simulation methods. Erek et al. [10] presented CFD simulation data on the influ-

*Corresponding author
Email address: laszlo.konozsy@cranfield.ac.uk (László Könözsy)