Advances in Aircraft and Spacecraft Science, *Vol. 2, No. 4 (2015) 469-482*
DOI: http://dx.doi.org/10.12989/aas.2015.2.4.469

Assessment of the performances of a heat exchanger in a light helicopter

Antonio Carozza*

Dipartimento di Meccanica dei Fluidi, Centro Italiano Ricerche Aerospaziali, Capua, via Maiorise 81043, Italy

(Received June 16, 2015, Revised July 27, 2015, Accepted September 7, 2015)

Abstract. This study has the aim to develop a numerical design regarding the position and the inner performances of a heat exchanger in a light helicopter. the problem was to find first of all the best position of the heat exchanger inside the engine vane in order to maximize the air flow rate capable to pass through the heat exchanger section. It is to be said that the only air contribution in the vane comes from the opening present in the roof under the main rotor. The design has been performed by means of the commercial code Fluent and using the well known grid generator ICEM CFD. Different positions are first investigated so to establish the best one. Subsequently, different areas of the opening on the roof have been considered in order to maximize even more the flow rate in the heat exchanger that was not sufficient based on the first guess of velocity, as aforementioned. At the end interesting design results are presented and discussed by contours of fields and values.

Keywords: aerodynamics; CFD simulation; heat exchanger; mass flow rate

1. Introduction

In this ESPOSA European Project framework CIRA is involved in the engine integration for the configuration PU2 and TR1, and HE1. The research activities regarding the helicopter HE1 configuration are the oil-cooler air location design and the subsequent analyses at different flow rates at the inlet. Between the different oil-cooler critical operative conditions which the airplane experiments in cruise and ground environments the most critical one is the ground flow condition and the present study has been performed just on that one.

The commercial code Fluent is used as a solver. In particular, the numerical model used in this work is the porous medium, simulated by means of a negative momentum source. The porous media are widely investigated and simulated with a number of different numerical models like Latex-Boltzman methods or finite element methods or simplified correlations methods.

In the article by (Alshrae *et al.* 2009), numerical simulations of flow and heat transfer in a serpentine heat exchanger configuration are presented to demonstrate application of porous media techniques in heat exchanger analyses. The simulations are conducted using two different approaches. In the first approach, a porous continuum homogeneous model (PCM), or macroscopic model, is applied. The solid and fluid phases are modeled as a single, homogeneous

*Corresponding author, Ph.D., Research Scientist, E-mail: a.carozza@cira.it

http://www.techno-press.org/?journal=aas&subpage=7 ISSN: 2287-528X (Print), 2287-5271 (Online)