Applied Thermal Engineering 99 (2016) 1048–1056

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Numerical investigation of heat exchanger performances for a two engine aircraft in pusher configuration by an u-RANS code

Antonio Carozza *

Dipartimento di Fluidodinamica, CIRA, Centro Italiano Ricerche Aerospaziali, Capua 81043, Italy

HIGHLIGHTS

- An unsteady RANS code has been upgraded with momentum and energy sources.
- The modification is suitable to reproduce the behavior of whatever heat exchanger performances.
- Both thermal exchanges and pressure losses can be simulated in a simplified manner.
- Times are noticeably reduced with respect to the commercial code Fluent ones.

ARTICLE INFO

Article history:
Received 14 April 2015
Accepted 31 January 2016
Available online 9 February 2016

Keywords:
CFD
Heat exchanger
RANS
Unsteady
Turbulent
Porous model

ABSTRACT

This paper deals with the numerical analysis of the effect a heat exchanger has on the flow field in an oil cooling system for aerospace applications. To this end, an important upgrade of the UZEN in house CIRA *Reynolds Averaged Navier–Stokes* solver has been carried out to account for the pressure drop and heat rejection that usually are present in a heat exchanger. This upgrade is based on the addition of a source term in the momentum and energy equations of balance, respectively, whose values are updated during the simulation. The correspondent code modification needs a proper validation before any concrete application. At this aim, a two dimensional channel has been firstly studied in order to test the accuracy of the UZEN code. Then a three dimensional circular duct has been studied with a porous insert to validate the porous approach robustness. After this preliminary campaign of assessment, a new generation two engine aircraft has been studied with particular emphasis on its nacelle cooling system. Velocity and pressure are compared using different turbulence models with the well known Fluent ones. The more interesting parameter for such an investigation is the mass flow rate through the cooling ducts as computed by UZEN that show a good agreement with the Fluent one.

1. Introduction

The investigation, numerical or experimental, was conducted on aeronautical systems in order to reduce the fuel temperatures of the engine case and can involve several parameters in some manner correlated. However, the main aerodynamic parameter influencing the efficiency of the engine's cooling is the air flow rate pushed on the oil responsible to refresh the fuel at the exit of the compressor. This variable is often optimized, introducing ducts with a certain section area size in accordance with the space available inside the engine cabin. Unfortunately performing simulations in these regions can be really difficult because of the numerous separations often present there. Both turbulence models and numerical schemes require considering these difficulties when they are implemented and used.

Several researchers have tried to develop simplified numerical models to overcome these drawbacks. Usually they have adopted a one-dimensional model or have done very strong assumptions on the flow. In some cases they have found that the best way to face that problem can be the use of existing numerical models belonging to completely different branches of the science research.

A number of numerical models are present in literature having the scope to simplify the simulation and the design of the aircraft engine thermal integration systems.

Most recent attempts divided the heat exchanger into thermal segments to find overall performance. Two popular distributed parameter methods are used currently in heat exchanger numerical simulation. These are the finite volume method and the moving boundary method. The first method divides the entire heat exchanger into a number of constant control volumes and all conservation equations are discretized into these control volumes to solve sequentially or simultaneously. The second method divides the heat exchanger into different sections according to the phase state of working fluids. The boundaries between different phase

* Corresponding author. Tel.: +39 0823 623648; fax: +39 0823 623335.
E-mail address: a.carozza@cira.it (A. Carozza).

http://dx.doi.org/10.1016/j.applthermaleng.2016.01.160
1359-4311/