Applied Mathematical Modelling 37 (2013) 1208–1224

Contents lists available at SciVerse ScienceDirect

Applied Mathematical Modelling

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

A computational model of a transcritical R744 ejector based on a homogeneous real fluid approach

Jacek Smolka[a,*], Zbigniew Bulinski[a], Adam Fic[a], Andrzej J. Nowak[a], Krzysztof Banasiak[a], Armin Hafner[b]

[a] Institute of Thermal Technology, Silesian University of Technology, Gliwice, Poland
[b] Sintef Energy Research, Trondheim, Norway

ARTICLE INFO

Article history:
Received 31 March 2011
Received in revised form 17 March 2012
Accepted 21 March 2012
Available online 3 April 2012

Keywords:
Two-phase flow
Enthalpy formulation
Real fluid
CO_2
Ejector
Heat pump

ABSTRACT

A mathematical model of the compressible transonic single- and two-phase flow of a real fluid is discussed in this paper. The model was originally developed to simulate a refrigerant flow through a heat pump ejector. In the proposed approach, a temperature-based energy equation is replaced with an enthalpy-based formulation, in which the specific enthalpy, instead of the temperature, is an independent variable. A thermodynamic and mechanical equilibrium between gaseous and liquid phases is assumed for the two-phase flow. Consequently, real fluid properties, such as the density, the dynamic viscosity and the diffusion coefficient, are defined as functions of the pressure and the specific enthalpy. The energy equation formulation is implemented in commercial CFD software using subroutines that were developed in-house. The formulations was tested extensively for a single-phase flow of the R141b refrigerant, and for a two-phase flow of the R744 fluid (carbon dioxide) that occurred in a 3-D model of the ejector motive nozzle. In the model validation procedure, a satisfactory comparison between the experimental and computational results of the primary and secondary mass flow rates was obtained for both flow regimes. In addition, in the case of the R744 flow, the pressure distribution along the centre line of the ejector was accurately predicted as well. Furthermore, the results also shows that geometry modelling and measurement accuracy play an important in the final numerical results. As a result of the reasonable computational times, this method can be effectively used for the design of ejectors and also in geometric optimisation computations.

1. Introduction

This paper presents a mathematical model originally developed to predict the transcritical compressible flow of a real fluid occurring in an ejector for an environmentally friendly refrigerant (CO_2). This device replaces the throttling valve typically used in a heat pump cycle. In such a system, the supercritical pressure of the fluid from the gas cooler is reduced in the ejector's primary converging–diverging nozzle and is converted to the kinetic energy. In the nozzle, the evaporation of the working fluid occurs. Then the supersonic and low-pressure two-phase CO_2 that is ejected from the primary nozzle draws the low-pressure gas-phase CO_2 from the evaporator. During this process, the suction flow is accelerated because the flow area is gradually reduced from the secondary inlet to the mixing section. In addition, the speed of the gas-phase increases, and as a consequence, the pressure in the mixing section is lower than the suction pressure in the evaporator. In the mixing

* Corresponding author.
E-mail addresses: jacek.smolka@polsl.pl (J. Smolka), zbigniew.bulinski@polsl.pl (Z. Bulinski), adam.fic@polsl.pl (A. Fic), andrzej.j.nowak@polsl.pl (A.J. Nowak), krzysztof.banasiak@polsl.pl (K. Banasiak), armin.hafner@sintef.no (A. Hafner).

0307-904X/$ - see front matter
http://dx.doi.org/10.1016/j.apm.2012.03.044

Nomenclature

Latin symbols

$\boldsymbol{u}$	velocity vector, m/s
$C_{1\epsilon}, C_{2\epsilon}, C_{3\epsilon}$	empirical constants in transport equation for turbulent dissipation rate
G_K	generation of turbulence kinetic energy due to mean velocity gradients, kg/(m s^3)
G_b	generation of turbulence kinetic energy due to buoyancy, kg/(m s^3)
H	specific enthalpy, J/kg
K	turbulent kinetic energy, m^2/s^2
k	thermal conductivity, W/(m K)
p	pressure, Pa
T	temperature, K
u, v, w	velocity vector components in Cartesian coordinates, m/s
Y_M	fluctuating dilatation dissipation, kg/(m s^3)
S_ϕ	source term in a transport equation for scalar ϕ

Greek symbols

δ	Kronecker delta
ϵ	turbulent dissipation rate, m^2/s^3
Γ	diffusion coefficient, m^2/s
τ	stress tensor, N/m^2
μ	dynamic viscosity, kg/(m s)
μ_T	turbulent viscosity, kg/(m s)
ϕ	arbitrary scalar
ρ	density, kg/m^3
σ_ϵ	turbulent Prandtl number for turbulent dissipation rate
σ_K	turbulent Prandtl number for turbulent kinetic energy
σ_T	turbulent Prandtl number

Subscripts

c	calculated
eff	effective
i, j, k	indices of Einstein summation notation
m	measured

Other symbols

$\bar{n}$	Reynolds averaged
$\tilde{n}$	Favre averaged
UDS	User-Defined Scalar, a feature in Ansys Fluent to define a user transport equation
UDF	User-Defined Function, a feature in Ansys Fluent to customise/extend the software built-in models/capabilities

section, the working fluid that flows from the gas cooler through the primary nozzle and from the evaporator through the secondary nozzle mixes. As a result, a rise in pressure is observed. In the diffuser, the flow area gradually increases converting the kinetic energy to a region of higher pressure.

In the literature, the transcritical flow of CO_2 through an ejector is mainly investigated through experimental studies [1–3]. In these works, the most important operational and geometric parameters, e.g. the pressure in a primary nozzle or diffuser angle, were tested and their impact on the performance was studied. One could expect a similar number of contributions to a numerical model of the phenomena appearing in a transcritical CO_2 ejector. In the ejector literature, numerous papers describe the formulation of the mathematical models to simulate the flow through this type of device, but only few of these reports are devoted to the R744 (CO_2) flow.

In general, the models developed for flows through ejectors can be divided into 1-D and 3-D approaches in terms of the quality and functionality of the obtained results. The former models are mainly capable of predicting the global parameters of the analysed device [4–7], while the latter ones can additionally capture the local phenomena observed in all parts of the ejector considered [8,9].

The obvious cost of considering a 3-D problem is computational time. Therefore, most authors simplify their models to 2-D or 2-D axisymmetric geometries [7,10–12]. This assumption is true for motive flow. However, the potential inaccuracy of the results obtained lies in the simplification of the flow through a suction pipe followed by a secondary nozzle, which cannot be formally performed in this manner in most cases. In the literature devoted to supersonic single- and two-phase flows through ejectors, this problem has not yet reported, apart from the work of [11] on the water-steam ejector, in which is claimed that there are only small differences between the 2-D and 3-D solutions. In this study, the ejector geometry was