International Communications in Heat and Mass Transfer 97 (2018) 85–91

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

A model for heating and evaporation of a droplet cloud and its implementation into ANSYS Fluent

Timur S. Zaripov[a,b,*], Oyuna Rybdylova[b], Sergei S. Sazhin[b]

[a] Kazan (Volga region) Federal University, Kazan, Russia
[b] Advanced Engineering Centre, School of Computing, Engineering and Mathematics, University of Brighton, Brighton BN2 4GJ, UK

ARTICLE INFO

Keywords:
Droplets
Heating
Evaporation
Fully Lagrangian approach
ANSYS Fluent
n-dodecane

ABSTRACT

A model for heating and evaporation of a cloud of monocomponent droplets in air, taking into account the evolution of droplet number densities, is developed and implemented into ANSYS Fluent. Functionality testing of the new customised version of ANSYS Fluent is based on its application to the analysis of a droplet cloud in a two-phase back-step flow. It is shown that the effect of the droplet cloud needs to be taken into account when estimating the heat and mass transfer rates from the carrier phase to the droplets.

1. Introduction

The importance of modelling droplet heating and evaporation in various engineering and environmental applications is well known [1,2]. Although a number of advanced models of these processes have been developed (see [1,2] for the details), only very basic models are currently used in most commercial and research Computational Fluid Dynamics (CFD) codes.

The results of implementation of a previously developed model for monocomponent droplet heating and evaporation into ANSYS Fluent are described in [3]. In this model the effects of liquid finite conductivity and recirculation inside droplets were taken into account based on the Effective Thermal Conductivity (ETC) model. This model is based on the analytical solution to the heat conduction equation inside the droplet, assuming spherical symmetry of the processes. The results of implementation of a more general multicomponent droplet heating and evaporation model into ANSYS Fluent are described in [4].

One of the main limitations of the models described in [3] and [4] is that they are applicable to isolated droplets only, while in most realistic applications droplet clouds rather than isolated droplets are observed. The effects of heating and evaporation of droplet clouds could be investigated based on the conventional Lagrangian approach in which individual droplets are tracked along their trajectories. This approach, however, in most cases requires that calculations are made for prohibitively large numbers of droplets in order to perform reliable estimates of droplet number density in each computational cell.

As shown in a number of papers, including [5], a more efficient method of calculating droplet number density, when compared to the conventional Lagrangian approach, could be based on the fully Lagrangian approach developed in [6,7]. This approach is sometimes referred to as the Osiptsov method. The results of the implementation of this method into ANSYS Fluent are described in [8].

The main aim of this paper is to present a new model for heating and evaporation of a droplet cloud, based on a combination and further development of the approaches presented in [3], focused on heating and evaporation of individual droplets, and in [8], focused on the evolution of droplet clouds without heating and evaporation, and the results of the implementation of this model into ANSYS Fluent. We will use the same droplet heating and evaporation model as described in [3]. A two-phase back-step flow will be used for functionality testing of the combined model. As in [3] and [8], the effects of droplets on the carrier phase are ignored. The analysis is restricted to monosized and monocomponent droplets.

The mathematical models of the gas-droplet flow and their implementation into ANSYS Fluent are described in Section 2. In Section 3, the results of application of the new customised version of ANSYS Fluent to the analysis of a two-phase back-step flow are described. The main results of the paper are summarised in Section 4.

2. Basic equations

2.1. Gas (carrier phase) flow

The carrier phase is modelled as an incompressible or compressible

* Corresponding author at:University of Brighton, Brighton, UK.
E-mail address: T.Zaripov2@brighton.ac.uk (T.S. Zaripov).

https://doi.org/10.1016/j.icheatmasstransfer.2018.06.007

Available online 20 July 2018
0735-1933/