Numerical Simulation of 3D Unsteady Heat Transfer at Strongly Deformed Droplets at High Reynolds Numbers

Matthias Hase and Bernhard Weigand

Institute of Aerospace Thermodynamics, University of Stuttgart, Pfaffenwaldring 31, 70569 Stuttgart, Germany, matthias.hase@itlr.uni-stuttgart.de

Summary. The dependency of the heat transfer on an initial deformation of droplets has been investigated at high droplet Reynolds numbers. The two-phase flow has been computed with an inhouse 3D DNS program (FS3D) using the Volume-of-Fluid method. For the droplets initial prolate and oblate shapes with an axial approaching flow has been studied. In addition, a spherical shape has been used as reference. The initial droplet Reynolds number for the present study has been \( Re_0 = 660 \) for all investigated cases. Due to the fact that the steady droplet velocity for the considered droplets has been much lower than the initial velocity of the droplets, the droplet velocity is decreased during the simulation. To gain more knowledge about the influence of deformation on the heat transfer, the time dependent, spatial averaged Nusselt number \( Nu_t \) and the time and spatial averaged Nusselt number \( Nu_m \) has been matched by the temperature and velocity field around a deformed droplet. By this comparison the oscillation phase with the largest heat transfer has been observed. The simulations have been performed on the Cray T3E/512-900 at the HLRS with 32 processors. The parallel performance in dependency of the number of processors has been investigated.

1 Introduction

The rate of heat transfer from the surrounding gas to droplets in sprays is a critical design parameter of many technical spray systems as for instance in automotive engines or gas turbines. In these processes the considered droplets respectively liquid ligaments are in many cases strongly deformed and the droplet velocity is heavily unsteady. Additionally due to the high velocities the flow around the droplet is transient and fully 3D. Because of this difficulties strongly deformed droplets have been studied rarely in the past, neither numerically nor experimentally.

In the present study it has been assumed that the droplets are deformed initially due to the primary breakup. During this breakup process strongly deformed liquid ligaments emerge which are approximated by two idealized...
droplet shapes in this study as a first step. The numerical investigation has been performed at high droplet Reynolds numbers ($Re > 270$), which means that the flow is fully 3D and time dependent. The initial droplet Reynolds number $Re_0 = D_0 U_0 / \nu_L$ of the considered droplets is $Re_0 = 660$, where $D_0$ is the diameter of a spherical droplet with the same volume, $U_0$ the initial droplet velocity and $\nu_L$ the kinematic viscosity of the droplet fluid. To take the flow character into account the investigation has been performed fully 3D. Additionally no restrictions on the deformation of the droplets are assumed. The programs efficiency and reliability for the computation of strongly deformed two-phase flow has been presented already in [1].

2 Analysis and numerical method

The inhouse 3D CFD program FS3D (Free Surface 3D) has been developed to compute the Navier-Stokes equations for incompressible flows with free surfaces. The equations are solved without using a turbulence model by direct numerical simulation (DNS). The governing conservation equations for momentum and mass are

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot ((\rho u) \otimes u) = -\nabla p + \nabla \cdot \mu [\nabla u + (\nabla u)^T] + \nabla \cdot T$$

(1)

$$\nabla \cdot u = 0$$

(2)

where $T$ is the capillary stress tensor which adds the surface tensor force to the momentum equation. Furthermore $u$, $\rho$, $\mu$ and $p$ are the velocity vector, the density, the dynamic viscosity and the pressure, respectively. In addition, the energy equation is solved. For the above mentioned incompressible flow and for a fluid with constant fluid properties in each phase the energy equation is decoupled from the equations of motion. Therefore, the energy equation

$$\frac{\partial (\rho c_p T)}{\partial t} + \nabla \cdot (\rho c_p u T) = \nabla \cdot (k \nabla T) + \Phi$$

(3)

can be solved after the computation of the flow field, where $T$ is the temperature, $c_p$ the specific heat and $k$ the heat conductivity. The dissipation term $\Phi$ can be neglected for all mentioned flows due to the low Eckert number. The implementation and validation of the energy equation has been described in [2].

In two phase flows additional information about the interface position between the disperse and the continuous phase are needed. In FS3D a Volume-Tracking method, well known as the Volume-of-Fluid (VOF) method, is used [3]. In the VOF-method an additional transport equation

$$\frac{\partial f}{\partial t} + \nabla \cdot (uf) = 0$$

(4)

for the volume fraction of the disperse phase is solved.