International Journal of Thermal Sciences 65 (2013) 170–177

Contents lists available at SciVerse ScienceDirect

International Journal of Thermal Sciences

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

Thermal mixed convection past a sphere in an assisting flow

Cédric Le Bot [a,*], Gilles Bouchet [b]

[a] Université de Bordeaux, I2M-Département TREFLE, UMR CNRS 5295, 16 avenue Pey-Berland, 33607 Pessac Cedex, France
[b] Laboratoire IUSTI, UMR 6595 CNRS, Aix-Marseille University, 5 rue Enrico Fermi, 13453 Marseille Cedex 13, France

ARTICLE INFO

Article history:
Received 17 October 2011
Received in revised form
18 October 2012
Accepted 23 October 2012
Available online 2 December 2012

Keywords:
Mixed convection
Numerical simulation
Recirculation

ABSTRACT

This paper presents the numerical study of the fluid flow and thermal transfer between a molten metal bath and a spherical metallic particle maintained at a constant cold temperature and submitted to mixed convection. The fluid velocity is imposed and the particle is fixed. For a given Reynolds number, a study of the influence of the Richardson number on heat transfer and fluid flow is proposed in order to determine the effect of particle–fluid temperature difference on the wake and the heat transport.

1. Introduction

The aim of this study is to understand the link between dynamic behaviours and heat transfer of a cold particle submitted to a hot ambient fluid flow with a given velocity. In many natural or industrial processes, a molten medium is composed of a fluid phase with solid inclusions. The presence of these particles, made of the same material that the melt phase or not, may lead to different properties compared to the expected ones. For industrial processes, it may alter the quality of the final product. For example, during metal products manufacturing, the material is melted and is formed under this liquid phase. Then, a cooling system leads to solidification to obtain the final product. The presence of solid particles in the bulk molten phase (due to a bad melting) may create defaults: the particles can be transported by the flow generated by thermal buoyancy or an external force, and aggregate. It may create clusters of solid in the molten metal, which potentially modifies the structure of the final product, and weakens it. To avoid these difficulties, it is necessary to understand how a particle evolves in a liquid phase, considering fluid flow and heat transfer. It will give information in order to control the casting process.

We have decided to study a single particle, as many phenomena occur during the movement of only one particle in a fluid. In an isothermal configuration, Schouveiler and Provansal [1] have shown experimentally that for a fixed particle submitted to a forced flow, the wake exhibits various structures, depending on the Reynolds number. Physically, coupling thermal transfer and fluid and particle dynamics implies a broad range of behaviours. In an isothermal flow, Archimedes forces imply the movement of the particle, upward or downward depending on the density of both the fluid and the particle. The resulting fluid velocity implies a laminar or turbulent behaviour, determined by several parameters (diameter, densities, viscosity…). Some dimensionless numbers (Froude, Galilei numbers) correlate the characteristics and the fluid flow. When a cold particle evolves in a hot fluid, natural convection accelerates or decelerates the particle movement, depending on the configuration: assisting or opposing flows. An assisting flow corresponds to the situation where Archimedes forces are superposed to the fluid flow, whereas for the situation corresponding to an opposing flow, the Archimedes forces are opposed to the fluid flow. When the particle reaches a high velocity, the mixed convection replaces the natural convection. The temperature difference between the particle and the fluid is of importance in the resulting velocity field. The thermal effect of buoyancy may become negligible compared to forced effects which adds difficulties in the analysis of phenomena. Some authors ([2–6]) have shown, by experimental and numerical analysis of settling or ascension of a particle, that a transition appears between a low velocity, for which a linear behaviour of the particle is observed, and a high velocity for which an oscillating movement occurs. For a metal medium, an experimental study of the movement of a particle and heat transfer is not possible. It is hard (or impossible) to

* Corresponding author. Tel.: +33 (0) 540 006 192; fax: +33 (0) 540 006 668.
E-mail addresses: lebot@enscbp.fr (C. Le Bot), Gilles.Bouchet@univ-provence.fr (G. Bouchet).

1290-0729/$ – see front matter
http://dx.doi.org/10.1016/j.ijthermalsci.2012.10.015

visualize the phenomena because of the medium opacity, and the detection of particles does not allow the use of standard techniques (PIV, …). Numerical simulation is then essential to lead an analysis of the behaviour of solid inclusions in the metal bulk. Some works have been carried out on the numerical simulation of particles settling. Pianet and Arquis [9] have presented the numerical simulation of the isothermal 2D particle settling under Faxen conditions (low Reynolds number) in order to compare the terminal velocity to the theoretical solution, with a numerical code based on a finite volumes method. Another work has been carried out by Maunoury et al. [10] on the transport of a cold 2D particle in a hot fluid to show the difference with the isothermal case. Moreover, a numerical study has been carried out on a sphere falling or ascending freely in a Newtonian fluid [11]. The results have shown that vortices occur downstream from the particle, which influences heat transfer with the fluid.

We propose in this study to carry out numerical simulations on a fixed particle submitted to a fluid flow at controlled velocity, taking into account thermal effects, and gravity effect. Some authors [7] have proposed to study the mixed convection for a heated particle and have shown the relation between the thermal conditions and the generated flow. They have shown that the Nusselt number depends on the Richardson and Reynolds numbers (the range of the Reynolds number being 50–1400). The same kind of study has been carried out by Bhattacharyya and Singh [8], the authors propose a Lattice Boltzmann method to compute the motion of a particle in a fluid. They have shown that for three Richardson numbers ($Ri = 0$, $Ri = 0.5$ and $Ri = 1$), the Nusselt number increases with the Reynolds number (the range of the Reynolds number is 0–200). These studies only concern water and air flows with a Prandtl number respectively of $Pr = 7$ and $Pr = 0.72$. No information on metal settling is available in literature, and such a study could be helpful to control the final metal product shape and quality. In this article, we do not consider phase change, even if in the case of particles made of the same material than the fluid, this phenomenon occurs. We will lead a study of Nusselt and eddies evolutions with the Richardson number, for $Re = 309$ and $Pr = 0.11$. Such a study has not been found in literature. The modelling of the solid–liquid interface during phase change (melting) will be reported in a future work, the present study focuses on the behaviour of an undeformable cold solid particle in a hot fluid.

In the present work, the particle is maintained at a fixed position and a fluid velocity is imposed upstream from the particle. The aim of this study is to highlight the relationship between the flow regime and thermal transfer between the cold particle and the hot fluid (thanks to appropriate dimensionless numbers which quantify and qualify thermal and dynamic changes). It provides information about the energy transferred between the particle and the fluid, the temperature and velocity fields of the fluid, for an assisting flow. This will be useful to predict the behaviour of metal while it is forming. To this way, we use two numerical codes based on different methods, the first one is called Thétis and is developed at I2M Institute of Bordeaux. The second one is called IMFS and is developed at IUSTI of Marseille and Institut de Mécanique des Fluides et des Solides of Strasbourg (France).

In the first part, we present the physical configuration, with the corresponding equations and assumptions. In the second part, we describe the numerical methods of the two codes. In the last part, we present the results from simulations for various Richardson numbers and a constant Reynolds number ($Re = 309$) and we comment the results from the numerical simulations of the 3D particle in terms of Nusselt evolution and particle wake as functions of the Richardson number.

2. Physical modelling

The problem that we study consists of a fixed cold solid particle submitted to a hot fluid flow. The domain is large enough (16 times larger than the particle radius), so that the assumption of an infinite medium can be considered (no interaction with walls). The fluid infinite velocity and temperature are set at a given constant value. The flow is considered incompressible, the fluid is Newtonian and its thermal conductivity, heat capacity and dynamic viscosity are temperature-independent variables. Only the fluid density depends on temperature (the Boussinesq approximation is used). The particle density, heat capacity and conductivity are constant. The two codes we describe in this work are based on two different manners.

2.1. Thétis modelling approach

The velocity field is modelled by the incompressible Navier–Stokes equations, with the Boussinesq approximation. A Brinkman term (where K is the permeability) is added to model the presence of a solid particle:

$$\nabla\cdot\mathbf{u} = 0$$
$$\rho_0\left(\frac{\partial\mathbf{u}}{\partial t}+\mathbf{u}\cdot\nabla\mathbf{u}\right) = \rho g - \nabla p + \nabla\cdot\left(\mu\left(\nabla\mathbf{u}+\nabla^T\mathbf{u}\right)\right) - \frac{\mu}{K}\mathbf{u} \quad (1)$$

The numerical code Thétis is based on a One Fluid Model. The distinction between the fluid and the particle is obtained by a high value of permeability K in the fluid ($K = 10^{20}$ m^2) and a low value in the particle ($K = 10^{-20}$ m^2). So, in the particle, $u = 0$, and the fluid does not penetrate it. The distribution of K in the domain is directly imposed with the geometry and the position of the particle defined as a sphere. So the permeability field is discontinuous, like density and heat capacity.

The heat transfer is modelled by the energy equation:

$$\rho_0 C_p\left(\frac{\partial T}{\partial t}+\mathbf{u}\cdot\nabla T\right) = \nabla\cdot(\lambda(\nabla T)) + B(T-T_p) \quad (2)$$

To take into account the thermal buoyancy, the fluid density is supposed to be linear with temperature.

$$\rho = \rho_0(1-\beta(T-T_0)) \quad (3)$$

In equation (3), the density ρ_0 corresponds to the density at $T = T_0$.

In order to limit the domain size, and considering a moderate Reynolds number, lateral boundary conditions are symmetry planes. The particle centre is set on the Y-axis. Imposed velocity and temperature are set on the inlet boundary condition and a Neumann adiabatic condition is proposed at the outlet condition. The particle is kept at a cold temperature. The fluid is initially at steady state with a given temperature, and the particle is kept at a cold temperature T_p during all the simulation, thanks to a penalization term expressed in the energy equation (2): the term B is locally (in the particle) imposed to a high value to obtain $T = T_p$ in the sphere region.

2.2. IMFS modelling approach

The method of the code IMFS consists in taking into account only the fluid phase, so that the sphere surface corresponds to a part of the boundary conditions. The previous equations are expressed under a dimensionless form, where d, V_i, d/V_i and ρV_i^2 are respectively the length, velocity, time and pressure scale references. The dimensionless temperature is expressed as