International Communications in Heat and Mass Transfer 73 (2016) 84–99

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Double-diffusive natural convection in an enclosure including/excluding sloshing rod using a stabilized ISPH method☆

Abdelraheem M. Aly

Department of Mathematics, Faculty of Science, South Valley University, Qena, Egypt

ARTICLE INFO

Available online 8 March 2016

Keyword:
Anisotropic porous media
Double-diffusive
ISPH
Natural convection
Non-Darcy flow
Sloshing rod

ABSTRACT

In this paper, double-diffusive natural convection in an enclosure is introduced by an incompressible smoothed particle hydrodynamics (ISPH) method. Two different cases of an enclosure have been studied. In the first case, the non-Darcy model for natural convection and heat and mass transfer in an enclosure saturated with anisotropic porous media has been investigated numerically by a stabilized ISPH method. The second case including sloshing rod inside an enclosure filled with free fluid has been studied numerically by a stabilized ISPH method. In the ISPH algorithm, a semi implicit velocity correction procedure is utilized, and the pressure is implicitly evaluated by solving pressure Poisson equation. The results are presented with flow configurations, isotherms, concentration contours and average Nusselt and Sherwood numbers for different Darcy numbers from 10^{-4} to 10^{-2}, porosity values from 0.5 to 0.9, permeability ratio from 0.1 to 10, inclination angle of permeability from 0° to 90° and Rayleigh numbers from 10^3 to 10^5. The results demonstrate the effects of the parameters such as Darcy number, porosity, permeability ratio and inclination angle in both of the heat and mass transfer rate and the flow regime. Adding sloshing rod with initial condition inside enclosure affects clearly in heat and mass transfer and the flow characteristics inside the enclosure. The results from this investigation are well validated and have favorable comparisons with previously published results.

1. Introduction

The natural convection in an anisotropic porous medium is an important area of research due to its wide range of applications including thermal insulation, flow in mushy region of a solidifying alloy [1] and flow past heat exchanger tubes [2]. The non-Darcy effects (i.e. the Forchheimer term) on natural convection in porous media have also received significant attention. The divergence in the heat transfer results has been reviewed in detail in Cheng [3] and Prasad et al. [4], among others. Thus, extensive efforts are being made to include the inertia and viscous diffusion terms in the flow equations and to examine their effects in order to develop a reasonably accurate mathematical model for convective transport in porous media. Detailed accounts of the research into non-Darcy convection have been reported in Tien and Hong [5], Cheng [6], Prasad et al. [7], and Kiadias and Prasad [8]. Nield and Bejan [9] provided an excellent summary of the subject regarding porous media models. The numerical studies of the natural convection flow in anisotropic porous media were conducted by use of Brinkman equation [10] or Brinkman–Forchheimer equation with permeability tensor [11]. They demonstrated that their formulations were accurate in predicting the flow and heat transfer for various inclinations of the principal permeability direction, permeability ratios, and Darcy numbers. The natural convective flow and heat transfer in a fluid saturated anisotropic porous medium have been investigated using the generalized non-Darcy models Nithiarasu et al. [12].

On the other hand, double-diffusive convection refers to buoyancy-driven flows induced by combined temperature and concentration gradients. The cases of cooperating thermal and concentration buoyancy forces where both forces act in the same direction and opposing thermal and concentration buoyancy forces where both forces act in opposite directions have been considered in the literature. Double diffusion occurs in a wide range of scientific fields such as oceanography, astrophysics, geology, biology and chemical processes Beghein et al. [13]. Ostrach [14] and Viskanta et al. [15] have reported complete reviews on the subject. Lee and Hyun [16] and Hyun and Lee [17] have reported numerical solutions for double-diffusive convection in a rectangular enclosure with aiding and opposing temperature and concentration gradients. Their solutions were compared favorably with reported experimental results. Mamou et al. [18] have reported an analytical and numerical study of double diffusive convection in a vertical enclosure.

In the recent years, the SPH method had been applied into compressible and incompressible viscous fluid flow problems [19,20]. The SPH was originally developed in compressible flow, and then some special treatment was required to satisfy the incompressible condition. One approach is to run the simulations in the quasi-incompressible

☆ Communicated by W.J. Minkowycz.

E-mail address: abdelreheam.abdallah@sci.svu.edu.eg.

http://dx.doi.org/10.1016/j.icheatmasstransfer.2016.01.008
0735-1933/

Nomenclature

C'	concentration of species
C	dimensionless species concentration
C_p	specific heat
Da	Darcy parameter
d_0	particle size
F	Forchheimer coefficient
g	gravitational acceleration vector
H	enclosure height
K	permeability
k	thermal conductivity
Nu	Nusselt number
N	Buoyancy ratio
Le	Lewis number
P	pressure
Pr	Prandtl number
Ra	Rayleigh number
Sh	Sherwood number
t	time
T'	temperature
T	dimensionless temperature
U,V	dimensionless velocity components
$\mathbf{V}$	velocity vector
W	enclosure width
x,y	Cartesian coordinates
X,Y	dimensionless coordinates
Greek symbols	
α	thermal diffusivity
β_T	thermal expansion coefficient
β_C	compositional expansion coefficient
ε	porosity
μ	viscosity
υ	kinematic viscosity
σ	ratio of heat capacities
ρ	density
τ	dimensionless time
∇^2	Laplacian operator

limit, that is, by selecting the smallest possible speed of sound which still gives a very low Mach number ensuring density fluctuations [19,20]. This method is known as the weakly compressible smooth particle hydrodynamics (WCSPH). A new model of the SPH method with numerical diffusive terms, called δ-SPH, has been introduced by Antuono et al. [21]. The δ-SPH is built on the assumption that a fluid is weakly-compressible, in which it allows the implicit fulfillment of the dynamics boundary condition along the free surface, as described by Colagrossi et al. [22]. Marrone et al. [23] adapted a δ-SPH scheme with an improved boundary treatment to simulate violent impact flows. They achieved accurate and robust predictions for global and local loads of impact flows on structures. Macià et al. [24] provided an in depth analysis of the most representative mirroring techniques used in SPH to enforce boundary conditions along solid profiles. Antuono et al. [25] combined the δ-SPH method with an enhanced treatment of solid boundaries to simulate 2D gravity waves generated by a wave maker and propagating into a basin. Marrone et al. [26] applied a 2D + t approach to study the wave pattern generated by high speed slender ships with a sharp stem. They described the body deformation by a proper modeling of the solid boundaries. Colagrossi et al. [27] studied, in detail, the assumptions of the SPH method such as (i) surface integral terms on the boundary of the interpolation kernel support are neglected and (ii) free-surface conditions are implicitly verified. Landrini et al. [28] used hybrid BEM-SPH to study the fluid mechanics of splashing bow waves on ships. Antuono et al. [29] discussed the use of numerical diffusive terms in weakly-compressible SPH schemes. They added diffusive terms to the continuity equation in order to reduce the spurious numerical noise that affects the density and pressure fields in weakly-compressible SPH schemes. Federico et al. [30] simulated 2D open-channel flows through an SPH model. Colagrossi et al. [31] also proposed a particle packing algorithm. This algorithm allows for a drastic reduction of the numerical noise due to particle resettlement during the early stages of flow evolution. Moreover, it can be easily derived starting from whatever SPH scheme, and applies under the hypothesis that a fluid is weakly-compressible or incompressible as well.

A proposal for developing an incompressible SPH (ISPH) model has been introduced, which pressure is implicitly calculated by solving a discretized pressure Poisson equation at every time step. Cummins and Rudman [32] introduced a new formulation for enforcing incompressibility in Smoothed Particle Hydrodynamics (SPH). The method uses a fractional step with the velocity field integrated forward in time without enforcing incompressibility. The resulting intermediate velocity field is then projected onto a divergence-free space by solving a pressure Poisson equation derived from an approximate pressure projection. Lee et al. [33] presented comparisons of a semi-implicit and truly ISPH algorithm with the classical WCSPH method, showing how the ISPH model could resolve some problems encountered in incompressible flow simulation by using WCSPH. Khayyer et al. [34,35] proposed a corrected incompressible SPH method (CISPH) based on a variation approach to ensure the angular momentum conservation of ISPH formulations to improve the pressure distribution by improvement of momentum conservation and the second improvement is achieved by deriving and employing a higher order source term based on a more accurate differentiation. Hu and Adams [36–38] introduced angular-momentum conservative smoothed particle dynamics for incompressible viscous flows and they adapted ISPH method for multi-phase flow. They proposed projection method combining the divergence of velocity plus density invariance conditions and thus solving two Poisson equations. Shao and Lo [39] proposed projection method which consists keeping density invariance condition only. Asai et al. [40] introduced the stabilized incompressible SPH method by relaxing the density invariance condition. Aly et al. [41–43] adapted the stabilized incompressible SPH method to simulate multi-fluid problems, fluid–structure interaction and fluid–soil–structure interactions. Xu et al. [44] proposed a stabilizing method for the ISPH model based on keeping divergence-free velocity field, which makes it possible to accurately estimate the pressure while keeping computational time smaller than WCSPH. This method consists in slightly shifting the position of the particles at each iteration so as to avoid highly anisotropic particle spacing. This method was improved by Lind et al. [45], who proposed an expression for the position shift based on Fick's law of diffusion. They also extended the shifting method to free-surface flows.

Numerical modeling of transient natural convection by using SPH method has also been investigated. Chaniotis et al. [46] proposed a remeshing algorithm based on weakly compressible flow approach and performed a comprehensive study for non-isothermal flows. Remeshing procedure was tested for various benchmark problems for fluid and energy transport, which include 1-D shock-tube problem, 2-D Taylor–Green flow, 2-D double shear layer, lid-driven flow in a square cavity, natural convection in a differentially heated cavity and mixed convection in a driven cavity. From the results, it was found that remeshing improves the accuracy of simulations since uniform particle spacing was conserved in each time step. SPH simulation of flow and energy transport using SPH was performed by Szewc et al. [47]. In their study, natural convection in a square cavity problem with a Boussinesq and a non-Boussinesq formulation was performed. They introduced a new variant of the Smoothed Particle Hydrodynamics (SPH) simulations of the natural convection phenomena. Danis et al. [48] modeled the transient and laminar natural convection in a square cavity using SPH method with a discretization tool on uniform Eulerian grids. Recently, Aly [49] modeled the multi-phase flow and natural