Advantage Materials Letters (AML)

First draft received MAR 2019
Accepted August 2020
PJN.5202008
SNY: 000020200828
ONLINE VERSION

Vbbripress.org

www.Vbbripress.org

Flow and heat Modelling for Porous Region and Experimental Validation

Sandi W, Lisa N

Abstract —*Flow and heat transfer in porous media has in recent years gained considerable attention especially in high-temperature reactors. It is proposed in this study that light water reactors (LWRs) can be made safer by redesigning the fuel in the fuel assembly. The proposed design is aimed at increasing the safety level in LWRs by the use of fuel in the form of loose coated particles in a helium environment inside the nuclear fuel cladding tubes of the fuel elements. The coated particle fuel being a heat source forms a bed in the cladding tube closed at both ends, the heat from the particles is transferred to the gas in the tube, and the gas movement is due to natural convection. In this study, we investigate the flow and heat transfer characteristics inside a cladding tube containing packed beds of spherical particles by simulating a porous region whose medium properties are defined; that is, the geometrical model representing the packed bed is specified as a porous region. The finite volume method was used in solving the three-dimensional Navier-Stokes equation while the heat transfer coefficient h and the dimensionless numbers such as Ra f(Gr, Pr) and Nu are used in analyzing the results. Simulated results from this investigation were validated with experimental results. The discrepancy in the results may be due to uncertainties, experimental errors, numerical errors, and the consequence of the lump parameter effect in the porous region modeling approach. This approach may be considered a unique means of estimating heat transfer characteristics in porous media.*

Keywords — *Light water reactor, porous region, heat transfer characteristics.*

Note — *Some figures may be in color only in the electronic version.*

I. INTRODUCTION

Heat transfer in packed bed systems is an important operation in high-temperature nuclear reactor designs. A number of high-temperature reactors (HTRs) were developed with embedded fuel in graphite moderators randomly packed as fuel spheres in the core with helium gas as coolant flowing under the action of forced convection heat transfer around the spheres.

The heat transfer coefficient is an important parameter in the determination of the heat transfer performance in packed beds.[1,2] Considerable efforts through the use of

various experimental and theoretical techniques under either steady-state or unsteady-state conditions have been made to evaluate the heat transfer coefficient in fixed bed reactors.[3–5] Although much information is available for forced convection heat transfer in packed beds[6] under supercritical conditions, there is very little information on heat transfer in packed beds under natural convection.

It is proposed in this study that light water reactors can be made safer by redesigning the fuel in the fuel assembly. Experimental and theoretical study of fluid-toparticle heat transfer was carried out under natural convection to better understand particle-to-fluid heat transfer characteristics expected in the proposed new fuel design.
Knowledge gained through the study has assisted in establishing a theoretical relation required for particle-tofluid heat transfer performance in the cladding tube for the proposed design.

In recent years, the use of numerical simulation through computational fluid dynamics (CFD) offers the opportunity to predict the heat transfer and fluid flow phenomena thereby providing important gain in time, limiting the number of experiments, and accessing information at a large scale that may not be measurable with experimental methods, and it is also possible to take into account real physical properties such as high temperature and pressure conditions.[7] Two approaches can be adopted in simulating porous media. The first is the traditional method; it is a direct simulation of spherical packing permeated by fine-scale voids that permit the passage of fluids in a cylindrical vessel. Direct simulation of packed beds randomly arranged through the use of the discrete element method approach is effective when studying the characteristics of contacting particles interacting with each other and quantifying the properties of the porous medium.[8] The number of particles in the medium that can be simulated using this approach is limited due to the large computing capacity required.[9] The second is by simulating a porous region whose medium properties are defined; typically, the geometry model representing the packed bed is specified as a porous region. In many CFD simulations involving packing materials (bed particles), it is not the detail of the internal flow that is of interest but rather the macroscopic effect of the porous medium on the overall fluid flow and heat transfer.[10]

This paper presents the use of the second simulation approach in validating the heat transfer characteristics in a porous medium experimental investigation. The goal of the study is to evaluate the validation and suggest if the porous region modeling approach can adequately predict the heat transfer characteristics in the proposed new fuel design contained in a cladding considering the lumped parameter effect in the approach and other uncertainties. This approach has rarely been used; hence, its investigation is worthwhile.

II. EXPERIMENTAL SETUP

The schematic diagram and experimental setup for this investigation are displayed in Figs. 1a and 1c. The setup consists of a cylindrical enclosure containing stainless steel (ANSI 304) balls (particles); the cylinder is manufactured from a highly insulating solid PTFE (polytetrafluoroethylene) material. The setup is made up of a cylindrical container placed on an induction heater, pressure transducers with transmitters, sets of calibrated T-type thermocouples connected to a data logger, a helium gas cylinder, a computer system, and some other accessories. A PolyScience Circulating Bath PD20R-30A12E was used for performing the calibration of the T-type thermocouples used in this experimental study before the test particles and particle test sample were instrumented with the thermocouples. The calibration is carried out to determine the error reading associated with each thermocouple. The readout of the thermocouples is compared to the readout of a secondary standard at the same conditions. Using regression, a slope and intercept are determined in order to adjust by appropriate difference to reflect the same readout as the secondary; this is carried out to ensure accurate data measurement in the experiment. The 34970A Agilent Data Acquisition/Switch Unit is used as a data logger and is connected to a computer system to acquire data (see Fig. 1c). It makes use of three sets of 20-channel Armature Multiplexer cards onto which the thermocouples and pressure transducers are connected. The pressure transducers used in the experiments are calibrated to secondary standards as well.

The container has removable top and bottom lids, an inductively heated bottom steel plate lying on the inside floor of the bottom lid, an inlet valve fixed to the cylinder wall, and a pressure relief valve fitted to the top lid; a wire mesh for suspending the bed in the tube to allow for convectional fluid flow current was placed a little distance vertically above the bottom plate inside the cylinder. The particles were poured randomly into the cylindrical container; nine selected particles were instrumented with thermocouples in such a way as to measure temperatures at the surface and central point inside the particle.

Thermocouples were also placed in the interstices to measure the average gas temperature close to these particles; silicon tubes connected to pressure transducers were placed below and above the bed to measure possible pressure drop across the bed. The air in the cylinder was initially flushed out before the cylinder was later finally filled with helium gas and sealed off by the inlet valve. Thermal energy enters the packed bed by means of the inductively heated steel

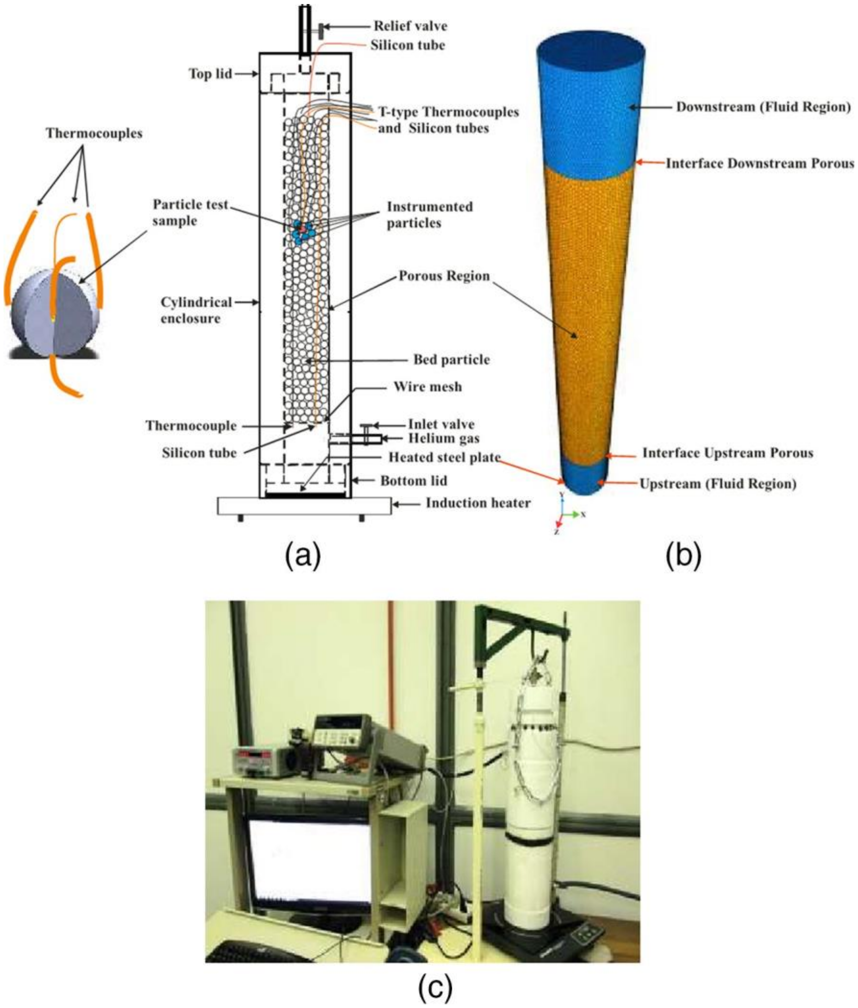

plate inside the cylinder that transfers heat to the helium gas immediately above the plate. Buoyancy-driven convection flow then takes place in the medium. The particle test sample was raised to 450 K by the heated helium gas taken at 6 h. Thermomechanical properties of the particles are taken directly from tables for an ANSI 304–type stainless steel sphere at room temperature (see Table I). Data were collected using a data logger connected to a computer system. The data collected were analyzed before application in theoretical formulations developed for determining the heat transfer phenomenon.

Fig. 1. (a) Schematic diagram of experimental setup (for illustrative purposes). (b) Meshed geometry model diagram of experimental setup boundary face wall for particle generating heat source porous medium. (c) Experimental setup.

The three-dimensional (3-D) computer-aided design geometrical model used in this simulation is a cylindrical enclosure representing the upstream fluid region where the heated bottom plate is located, the porous region, and the downstream fluid region (see Fig. 1b). The detailed drawing of the experimental setup prepared in Solidworks was imported into the commercial STAR-CCM package where surface repair was carried out on the two in-place boundary interfaces (see Fig. 2) to allow surface and volume meshing to proceed correctly due to the close proximity of neighboring boundaries. An in-place interface was required to link together two different regions (fluid and porous) by imprinting two boundaries on one another so that

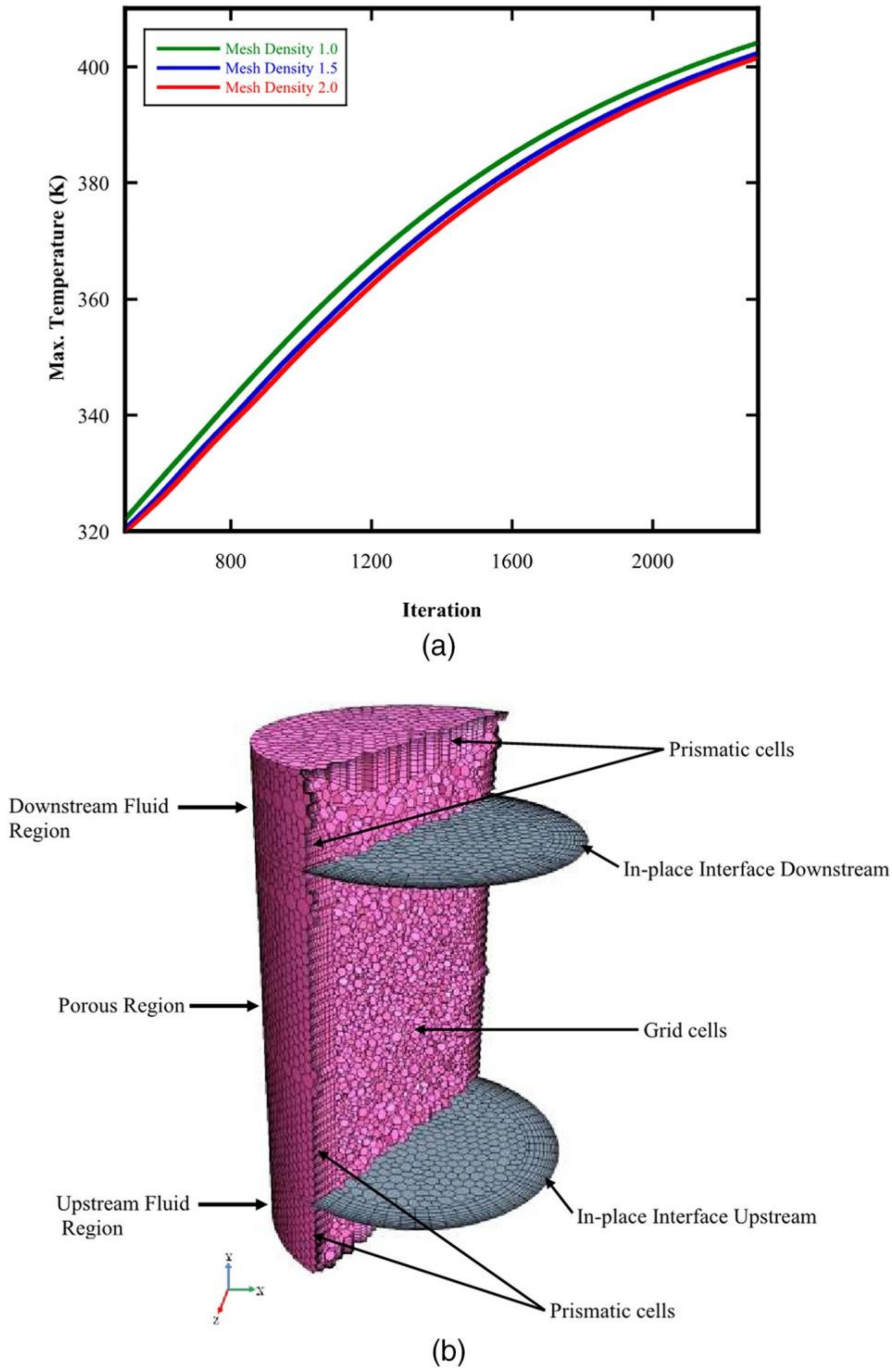

Fig. 2. (a) Mesh sensitivity analysis. (b) Threshold of a generated quality mesh of experimental model.

a conformal mesh is created at the common interface planes during the surface and volume meshing process.

A mesh is the discretized representation of the computational volume, which the physics solvers use to provide a numerical solution. STAR-CCM provides meshers and tools that are used to generate a quality mesh (as can be seen in Fig. 2) for various geometries and applications. Mesh quality does influence the quality of the numerical results, regardless of the setup. On a mesh of given quality and sufficient fineness, higher-order schemes yield more accurate results than a lower-order scheme. Grid skewing is an important contributing factor (much more than grid stretching) to the loss in nominal accuracy of the solution. Grid design (distribution of cell size, local refinement using feature edges, boundary region, or volume shapes) is important in maximizing the accuracy for a given effort. When solving steady-state problems on fine meshes, start with a much coarser mesh and then successively refine the mesh by increasing the mesh density; refinement enables better resolution of the flow. Meshes should be well designed to resolve important fluid flow features that are dependent upon flow condition parameters such as the grid refinement inside the wall boundary layer.

Grids can be either structured (hexahedral) or unstructured (tetrahedral) mesh types depending on the type of discretization scheme; the finite volume method (FVM) was used in this study because of its robustness and unconditional stability[11] and the nature of geometric configuration used. Concerning the mesh sensitivity analysis, the test performed consisted of increasing the mesh density of the geometrical model in order to properly capture the boundary layer associated problem. Three grid refinement levels are carried out in this study, from coarse to finer mesh. This is done by varying the mesh density to ensure that the CFD solution does not change with further mesh refinement as the aim of grid refinement is basically for numerical accuracy that leads to a solution closer to the exact solution of a system of equations that implies considerable physical approximations. For a 3.5-mm base size used in this study, results obtained for the last two finer meshes are almost identical (see Fig. 2a); hence, it can be established that simulations have reached an asymptotic solution, and the geometry with the last grid refinement level of finer mesh density 2.0 is used to build the fixed bed model. The chosen geometry has 319 057 grid cells, 1 701 035 faces, and 1 227 886 vertices; from experience, this should be adequate to capture heat transfer phenomena within the porous region. Figure 2b depicts the mesh generated for the experimental model. In this case, a surface remesher was initially applied on the geometry to retriangulate an existing surface in order to improve the overall quality of the surface and optimize it for the volume mesh models. A polyhedral mesh in conjunction with a prism layer mesh model was used to generate orthogonal prismatic cells next to wall boundaries; this is indicated in Fig. 2b. This layer of cells is necessary to allow the solver to resolve near wall flow accurately. This is critical in determining not only the forces and heat transfer on walls but also flow plus features such as separation. Other details on the geometry and mesh design are shown in Table II.

III. CFD Modeling

Modeling is the mathematical problem formulation of the physics involved in terms of a continuous initial boundary value problem is an essential to understand the methods of analysis, where an initial modeling was conducted using SolidWorks Flow Simulation. It was found the velocity and pressure of the flow through the porous materials can be obtained effectively using the results of CFD simulation, but the porous medium was suppressed to find the heat transfer during flow conditions due to the limitations of such software packages to accurately modeling the thermal contact resistance between the porous materials and flow particles. However, the physics should first be set up for the model. Essentially, the physics models define the primary variables of the simulation and what mathematical formulation will be used to generate the solution. In the physics model for this study,

1. The fluid was a gas.
2. The gas was assumed to behave as an ideal gas, and the ideal gas law was used for the density temperature dependency.
3. Sutherland’s law[10] was used for the thermal conductivity and viscosity temperature dependencies.
4. The flow was steady and laminar.
5. The space model selected was 3-D because themesh was generated in three dimensions.
6. The gas specified was helium; its property valuesare available in the STAR-CCM database.

7. As the medium is under natural convection, acoupled solver (iterative) was selected to control the solution due to its robustness for solving flow with dominant source terms, such as buoyancy.

The same experimental boundary conditions (given in Secs. III.D and III.E) were applied to boundaries at the walls and interface in the model. A no-slip shear stress specification was applied at the walls.

III.B.1. Model Analysis

The 3-D geometry model in Fig. 1b consists of an upstream region, a porous region, a downstream region, and two interfaces between the fluid and the porous regions. The upstream and downstream regions representing the lower and upper parts of the setup are defined as fluid regions while the region containing the particles is defined as the porous region. An in-place interface is required in between two adjacent regions of different property types; this is to allow for appropriate transfer of quantities of mass and energy calculated during the simulation. The thermal specification for the heated plate located at the bottom of the enclosure is set to a constant heat source corresponding to the experimental value. The external surface of the bottom wall (Y 0) and walls of other regions are considered to be adiabatic (this is only for experimental purposes) while the internal surfaces of the solid walls are assumed to be impermeable to mass transfer. The porous region considered here is homogeneous and isotropic, saturated with helium fluid, and the flow is steady and laminar. Reference values for the fluid thermophysical properties remain constant at initial conditions except density in the buoyancy term, which varies linearly with both temperature and concentration on the assumption that the Boussinesq approximation is implicitly valid. Viscous dissipation in the medium is assumed to be negligible.

Energy transport across the porous region is dealt with by specifying the thermal properties of the region in addition to the thermal properties of the fluid passing through it. Computation of flow is achieved by mathematical models based on conservation principles, namely, the conservation of mass, momentum, and energy governed by the 3-D Navier-Stokes equations representing the fluid as a continuum for an arbitrary control volume

Partial Preview