Applied Thermal Engineering 61 (2013) 198–205

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Heat transfer and pressure drop characteristics of molten fluoride salt in circular pipe

A.K. Srivastava*, A.M. Vaidya, N.K. Maheshwari, P.K. Vijayan

Reactor Engineering Division, Bhabha Atomic Research Centre, Mumbai 400085, India

HIGHLIGHTS

- Thermal-hydraulic study of FLiNaK salt has been performed using CFD code.
- Laminar and turbulent regimes of the flow have been analyzed.
- Analysis has been performed for various Reynolds number.
- Results were compared with the correlations of heat transfer and pressure drop.

ARTICLE INFO

Article history:
Received 1 March 2013
Accepted 25 July 2013
Available online 8 August 2013

Keywords:
FLiNaK
Nusselt number
Friction factor
Heat transfer
Pressure drop

ABSTRACT

FLiNaK salt, eutectic mixture of fluorides of lithium, sodium and potassium, is a potential candidate coolant for high temperature reactors (HTRs). In this paper, a CFD analysis has been performed to study the heat transfer and pressure drop characteristics of FLiNaK salt flowing in a circular pipe for various regimes of flow. Simulation is performed with the help of in-house developed CFD code, NAFA. The calculated local Nusselt number (*Nu*) and friction factor along the pipe have been compared with the available correlations. The dependence of Nusselt number on Reynolds number (*Re*) has also been predicted and compared. Pressure drop along the pipe has been compared with various correlations. This study shows that the considered correlations for heat transfer and pressure drop are suitable to predict the heat transfer and pressure drop behavior of FLiNaK salt.

1. Introduction

Next generation high temperature reactors needed for hydrogen production and similar other applications, will need to incorporate innovative approaches to further enhance their reliability and safety as needed for large scale deployment in different regions of the world. An important feature of these reactors will be the use of coolants at temperatures much higher than that being used in current generation reactors. Molten salts, liquid metals and supercritical fluids are being considered as potential coolants for core cooling of HTR's. In order to design reactor, it is necessary to study the various coolants and characterize their thermal hydraulic performance in terms of heat transfer coefficients, pressure drops, etc. Molten Salts, which can be used for high temperature (~1000 °C) at near atmospheric pressure, proved as potential candidate coolant for high temperature operation. FLiNaK is deemed as one of the most suitable molten salts, for their good nuclear properties, chemical stability and favorable thermo-physical properties.

Most of the previous studies related to molten salts are based on the experimental works. These experiments essentially measured the physical properties of molten salts [1–3] and their heat transfer characteristics [4–7]. Several analytical studies have also been carried out [8,9]. Mandin et al. [10] used Navier–Stokes and energy conservation equations to estimate thermal gradient in a molten salt. Ferri et al. [8] introduced the property definitions for molten salts in the RELAP5 code to perform transient simulations of the Prova Collettori Solari (PCS) test facility. Kearney et al. [11] investigated the feasibility of utilizing a molten salt as the heat transfer fluid and for thermal storage in a parabolic trough solar field. Bradshaw and Siegel [12] identified a range of quaternary molten salt compositions, based on solar nitrate salt. Yu-ting el al [13] studied experimentally the forced convective heat transfer behavior of molten salt ($LiNO_3$) in laminar–turbulent transition region and generated a correlation based on experimental data. Liu

* Corresponding author. Reactor Engineering Division, Bhabha Atomic Research Centre, Room No. 101, Hall No. 7,Mumbai 400085, India. Tel.: +91 22 2559 1597; fax: +91 22 2559 5151.
E-mail addresses: aksri@barc.gov.in, aks0071986@gmail.com (A.K. Srivastava).

1359-4311/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.applthermaleng.2013.07.051

Table 1
Governing equations used in NAFA code.

Equation	ϕ	Γ_r	Γ_z	S_ϕ
Radial momentum equation	V_r	$4/3(\mu+\mu_T)$	$(\mu+\mu_T)$	$-r\frac{\partial p}{\partial r}-V_r\frac{\partial}{\partial r}\left(\frac{2}{3}\mu\right)-r\frac{\partial}{\partial r}\left(\frac{2}{3}\mu\frac{\partial V_z}{\partial z}\right)+$ $\frac{\partial}{\partial z}\left[r(\mu+\mu_T)\frac{\partial V_z}{\partial r}\right]-\frac{4}{3}\mu\frac{V_r}{r}-\frac{\partial}{\partial r}\left(\frac{2}{3}r\rho k\right)$
Axial momentum equation	V_z	$(\mu+\mu_T)$	$4/3(\mu+\mu_T)$	$-r\frac{\partial p}{\partial z}-\frac{\partial}{\partial z}\left(\frac{2}{3}\mu\frac{\partial rV_r}{\partial r}\right)+$ $\frac{\partial}{\partial r}\left[r(\mu+\mu_T)\frac{\partial V_r}{\partial z}\right]-\frac{\partial}{\partial z}\left(\frac{2}{3}r\rho k\right)+r\rho g$
Thermal energy equation	h	$(k/c_p+\mu_T/Prt)$	$(k/c_p+\mu_T/Prt)$	$q'''+rV_r\frac{\partial p}{\partial r}+rV_z\frac{\partial p}{\partial z}$
Turbulent kinetic energy	K	(μ_T/σ_k)	(μ_T/σ_k)	$r(G-\rho\varepsilon)$ where $G=\mu_T\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$
Rate of dissipation of turbulent kinetic energy	ε	$(\mu_T/\sigma_\varepsilon)$	$(\mu_T/\sigma_\varepsilon)$	$r\left(c_1\frac{\varepsilon}{K}G-c_2\rho\frac{\varepsilon^2}{K}\right)$

et al. [14] performed experiments with molten salt ($LiNO_3$) in the Reynolds number range of 19,800–46,000. The Prandtl number of $LiNO_3$ is in the range 12.7–14.7. They used the well known convective heat transfer correlations by Dittus–Boelter, Sieder–Tate, Hausen, and Gnielinski for comparison purpose.

Although the low melting point and high boiling point is the benefit of these heat transfer mediums, it also causes difficulties in performing experiments. These difficulties include salt solidification and container corrosion during experiment. Computer simulations are therefore of particular assistance in evaluating the thermal hydraulic characteristics of molten salt. In the present study, these characteristics, including the friction factor and Nusselt number distributions for flow in a circular pipe are investigated using CFD methodology. The investigation has been performed for both laminar and turbulent flow regimes.

2. Model description

2.1. Governing equations

The analysis is performed using in-house developed 2D code NAFA, acronym for Numerical Analysis of Flows in Axi-symmetric geometries. It can handle laminar/turbulent flows with/without heat transfer with constant or temperature dependent thermophysical properties [15,16]. The code numerically solves the conservation equations of mass, momentum and energy [17]. The axisymmetric forms of these equations have been used which is appropriate for modeling a pipe/annulus. These equations are represented by following scalar transport equation.

$$\frac{\partial\left[r\left(\rho V_r\phi-\Gamma_r\frac{\partial\phi}{\partial r}\right)\right]}{\partial r}+\frac{\partial\left[r\left(\rho V_z\phi-\Gamma_z\frac{\partial\phi}{\partial z}\right)\right]}{\partial z}=S_\phi \quad (1)$$

Exact equations are obtained by defining the scalar (ϕ), diffusion coefficient (Γ) and source term (S_ϕ). These definitions for each equation are given in Table 1. In the table, the Reynolds averaged forms are given which are applicable to turbulent flows. The laminar forms can be obtained from these equations by setting turbulent viscosity (μ_T) and turbulent kinetic energy (k) to zero. In incompressible flow, there is no direct equation for pressure. Hence pressure correction equation is obtained (from mass and momentum equations) and solved to obtain pressure corrections [18]. On a computational grid, the pressure and velocity locations are staggered to avoid pressure–velocity decoupling [18]. For modeling turbulence, the high Reynolds number standard k-ε model is implemented with standard wall functions [19,20].

2.2. Numerical schemes

The governing equations are discretized in accordance with the finite volume method. For modeling convection terms of all these equations (except energy equation), power law scheme is used. For energy equation, 1st order upwind scheme is implemented [18]. The equations are non-linear and coupled. To solve these equations, iterative procedure as per the SIMPLE algorithm is implemented [18]. Alternating direction TDMA solver is developed for solving the resulting implicit equations [18]. Residual of each equation is checked for convergence. In each case, the iterations were continued till mass residue was reduced to 1E-6.

2.3. Boundary conditions

In this work, flow through circular pipe is modeled in an axisymmetric manner. The computations are performed in RZ (radial–axial) plane. The schematic diagram of the computational domain is shown in Fig. 1. As shown in the figure, four boundary conditions, namely, inlet, wall, axis and outlet, are encountered. At inlet boundary, uniform axial velocity and temperature (500 °C) are specified. For turbulent cases, it is assumed that the turbulent intensity (TI) is 10% and length scale is equal to diameter of pipe. Then the inlet turbulent kinetic energy and rate of dissipation of turbulent kinetic energy are obtained from following two expressions.

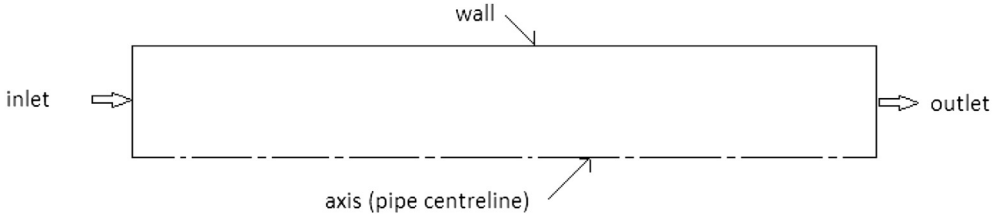

Fig. 1. Schematic diagram of computational domain.