International Journal of Thermal Sciences 125 (2018) 256–272

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Melting and multi-phase flow modelling of nuclear fuel in fast reactor fuel rod

Anuj Dubey[a], Anil Kumar Sharma[b],*

[a] Homi Bhabha National Institute, Mumbai, Kalpakkam Centre, Tamil Nadu, India
[b] Fast Reactor Technology Group, Indira Gandhi Centre for Atomic Research, HBNI, Kalpakkam, India

ARTICLE INFO

Keywords:
Melting
Enthalpy method
Multi-phase flow
In-pin fuel motion
Nuclear safety

ABSTRACT

Melting of nuclear fuel in a fast breeder reactor is accompanied by simultaneous release of gaseous fission products. These gases are trapped inside porosities present in the nuclear fuel and escape upon melting. The simultaneous release of fission gas and molten fuel results in a transient multi-phase flow. Initially, the fission gases are highly pressurized and exert hydrodynamic forces on the molten fuel. This in turn causes the molten fuel to undergo displacement. This displacement of molten fuel influences the reactor kinetics and the overall accident outcome. The present work is aimed at developing a mathematical model which simulates the melting and multi-phase flow phenomenon in a fast reactor fuel rod under a slow transient over-power accident scenario. The developed model is validated with the experimental data of the CABRI-E9 test. Thereafter, the behaviour of molten fuel is analysed with a nominal case study and parametric changes. Finally, the influence of molten fuel motion over reactor kinetics is analysed. The results show that under slow transients, molten fuel primarily relocates towards the lower portion of the fuel inner cavity. This behaviour of molten fuel can positively assist in the mitigation of a slow transient over-power accident.

1. Introduction

For the safety analyses of nuclear reactors, accurate estimation of heat transfer and melting in the fuel elements of the reactor core during a severe accident condition is essential. The thermal and mechanical behaviour of the fuel, coolant and structural steel during the transient influences the probable energy excursion. Another important aspect to reactor accident analysis is the geometric positioning of the fuel elements. Any displacement of nuclear fuel may increase or decrease the overall neutron density and influence the power excursion.

During nuclear fission, gaseous fission products are liberated within the microstructure of the fuel element and get stored in its micro porosities. These porosities are initially under high pressure due to continuous influx of fission gases. Upon melting of fuel, these trapped fission gases escape from the pores and travel within the molten fuel, thereby exchanging momentum with the molten fuel. This interaction along with other hydrodynamic forces results in a multi-phase flow of molten fuel and fission gases in the fuel rod geometry. The direction of flow determines the change in neutron density.

Most of the work reported in literature on the thermal analysis of melting inside nuclear fuel rods is under the assumption that the liquid fuel phase remains either stationary or natural convective within the confines of the heat conduction domain during the accident. This assumption remains justified in the case of melting in solid fuel geometries. Approximate solutions to the heat conduction equation are reported with different numerical methods. Chen et al. [1] have developed a mathematical model based on enthalpy formulation to solve similar problem. Crepeau and Siahpush [2] have compared quasi static analytical solutions to the Stefan problem with numerical results and have studied the existence of natural convection cells within the liquid domain. An analytical model has been developed by Kalaiselvam et al. [3] for the investigation of solidification and melting characteristics of phase change materials in cylindrical encapsulation. Tabassum et al. [4] have investigated melting problems with mushy region using the boundary-fitted coordinate method. Typical approaches available for numerical study of phase change in nuclear fuel rods are the finite difference method with enthalpy formulation [5], heat balance integral method [6], boundary-fitted coordinate method [4], and lumped parameter method [7]. Detailed information on the available methods for analysis of fuel behaviour under accident conditions and some practical guidance on their use have been provided by IAEA [8].

Under most probable accident scenario, the flow behaviour of liquid phase is heavily influenced by its inter-phase interaction with the released fission gases and the available flow space. A striking example is

---

* Corresponding author.
E-mail address: aksharma@igcar.gov.in (A.K. Sharma).

https://doi.org/10.1016/j.ijthermalsci.2017.10.009
Received 4 April 2017; Received in revised form 4 October 2017; Accepted 5 October 2017

Nomenclature

Latin

A	area of fuel pellet, m^2
Bu	burnup
C	specific heat, J/kg·K
e	enthalpy of source term, J/kg
g	acceleration due to gravity, m/s^2
h	heat transfer coefficient, W/m^{2}K
H	enthalpy, J/kg
J	phase boundary mass flux, kg/m·s
K	conductivity, W/m·K
L	number of axial nodes
Nu	Nusselt number
M	inter-phase momentum interaction, N/m
N	number of radial nodes
P	pressure, N/m^2
Pr	Prandtl number
q	heat flux per unit axial distance, W/m
Q	heat generation per unit mass, W/kg
$\dot{Q}$	volumetric heat generation, W/m^3
r	radial coordinate, m
R	radius, m
S	source term, kg/m·s
St	Stefan number
t	time, s
T	temperature, °C
V	velocity, m/s
X	linear power, kW/m
z	axial coordinate, m
Z	elevation, m

Greek letters

α	phase volume fraction
ρ	density, kg/m^3
φ	reactivity feedback

Subscripts

b	bottom most node of cavity
bl	blanket
blb	bottom of lower blanket
bs	bottom surface of cavity
bu	bubble
c	clad; continuous fluid phase
cav	fuel column inner cavity
$clnt$	coolant
d	disperse phase
D	drag
dr	droplet
f	fuel
g	gas
gap	fuel-clad gap
i	phase i, radial node index
$i-j$	gas-liquid inter-phase transfer
int	solid-liquid fuel interface
I	inner periphery
k	axial node index
l	liquid fuel
O	outer periphery
pel	pellet
pl	plenum
pool	liquid pool
s	solid fuel
si	source term of phase i
t	top most node of cavity
ts	top surface of cavity
tub	top of upper blanket
vm	virtual mass
w	cavity wall (fuel inner surface)

the phenomenon known as fuel squirting which was first observed experimentally in the C3C test performed by Hanson and Field [9]. It was found that upon melting, the liquid fuel was ejected through an internal stainless steel capillary tube. This extreme mobility of fuel implied that an over-power accident could potentially be mitigated due to removal of fuel from high flux location. The PINEX series of experiments performed in the TREAT facility demonstrated the feasibility of fuel squirting as a negative reactivity feedback mechanism [10]. In the experiments, molten fuel was squirted through the central hole of the fuel rod (or rod) with a velocity of 0.6 m/s. The sudden release of high pressure fission gases along with the molten fuel was determined to be the main reason behind this ejection. These experiments were carried out under a very high reactivity insertion rate of the order of 3 $/s. Due to sudden increase in power generation, the melting process was instantaneous and the pressurization of the fuel inner cavity was very high. The experimental data helped in understanding fuel squirting and established it as a negative reactivity feedback mechanism which would curtail an over-power accident. It is notable that the fuel under testing so far comprised of annular fuel, annular blanket and annular reflector pellets. Smith reported a comparison of internal fuel motion in different axial blanket designs [11]. All the designs had at least one annular blanket, in which the molten fuel solidified upon squirting and choked the cavity. The TS-1 and TS-2 experiments performed on solid fuel established fuel squirting as an accident mitigating mechanism for solid fuel under slow transients as well [12]. Similar results were reported in the HUT 5-2A experiment [13]. It was established that the squirting action was possible due to the formation of a highly pressurized cavity in the centre of the fuel, which allowed for fuel squirting once the cavity evolved in length up to the blanket pellets.

To understand the fuel squirting phenomenon in annular fuel rods (or pins) mathematically, Smith et al. [14] and Martin and Smith [15] developed FUMO-E, a two-phase fluid flow model. The influence of flow regime on the movement of molten fuel was found to be insignificant. The transport of fuel to the plenum in PINEX-2 experiment was predicted using PINEX-AR numerical code [16]. In-pin motion module, known as PINACLE, was developed for implementation in the SAS-4A safety analysis code system [17] [18]. This module assumed solid fuel geometry and utilized a homogeneous multi-phase fluid flow model. In case of complete solid fuel geometry, the assumption of homogeneous flow is adequate as the fission gases and molten fuel are locked inside a small cavity volume and their movement is intricately coupled.

Fast breeder reactor fuel design has undergone changes through the decades. The modern fuel rod design differs from the test rods used in the fuel squirting experiments. The fast reactor fuel rod design uses solid blanket pellets instead of annular blanket pellets [19]. The fuel pellets are given an annular geometry (1–2 mm diameter). This forms a closed cylindrical cavity in which the molten fuel motion may occur. Also, a change in nature of the rate of power increase in case of accident has been predicted by the reports on safety analysis. It is postulated that a slow over-power transient (few cents/s, 1 to 3% power ramp) shall be experienced by the reactor in case of a localized control rod withdrawal accident [20] whereas in the PINEX experiments, which established