Annals of Nuclear Energy 63 (2014) 51–58

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Numerical simulation of convection of argon gas in fast breeder reactor

Dinesh Nath [a], Mahendra K. Verma [b,*]

[a] Nuclear Engineering and Technology Program, Indian Institute of Technology, Kanpur 208016, India
[b] Department of Physics, Indian Institute of Technology, Kanpur, UP 208016, India

ARTICLE INFO

Article history:
Received 1 September 2012
Received in revised form 15 July 2013
Accepted 16 July 2013
Available online 15 August 2013

Keywords:
Annulus convection
Turbulent convection
Fast breeder reactor

ABSTRACT

In this paper, we present the results of numerical simulations of the turbulent convection in the Argon gas present in the annulus of a fast breeder reactor. We employ RANS scheme with k–ϵ model and solve the equations using an open-source software OpenFOAM. The Rayleigh numbers Ra of our simulations lie in the range of 10^8 to 10^{10}. We observe a pair of rolls with a hot plume rising from one end, and a cold plume descending from the opposite end of the annulus. This feature results because the aspect ratio of the geometry is near unity. We also find that the circumferential temperature difference (CTD) is proportional to Ra.

1. Introduction

In a pool type fast breeder reactors (FBR), schematically represented in Fig. 1, many mechanical components such as pumps, intermediate heat exchanger (IHX), rotating plugs, and control plugs are penetrated from the roof of the reactor vessel (Velusamy et al., 2010). These penetrations create vertical annulus which are closed at the top and open inside the reactor vessel. Argon gas, used as a cover gas over the liquid sodium in the reactor vessel, occupies the annular spaces as shown in Figs. 1 and 2. The liquid sodium is at a temperature of approximately 800 K, and it must be strictly separated from the open air. The sidewall of the cylindrical annulus and the top enclosure at the roof are cooled by an external circuit of air to maintain its temperature at around 400 K (Fig. 2). Convective rolls are created due to the temperature difference between the roof-top and bottom opening. The hot and light gas above the liquid sodium enters the annulus, loses heat to the annulus walls thus becoming heavier, and comes down the annulus at another circumferential location. This phenomenon of natural convection in the annulus results in a non-uniform circumferential temperature distribution in the penetrating component. This uneven temperature distribution causes uneven expansion near the walls of the annulus thus creating stresses which could cause deflection or tilting of components. These deformations would be detrimental to the operation of the reactor. A safe usage of reactors requires the deflections to be within a certain limit, hence, a good knowledge of the circumferential temperature distribution in the annulus is essential. In this paper, we perform numerical simulations of natural convection of Argon in the Fast Breeder reactor, and study the circumferential temperature distribution in the annulus as a function of system parameters. Several experiments related to the turbulent convection in PFBR annulus have been performed. Vijayan et al. (1986) performed experiments in a vertical annulus with water as the working fluid, and studied the variation of the temperature profile and Nusselt number as a function of Rayleigh numbers (Ra). They also studied the effects of baffles on the flow. The exponent of the Nusselt number varied from 0.342 in the absence of baffles to 0.182 with baffles. Hemanath et al. (2007) and Meikandamurthi et al. (1991) performed experiments in a vessel similar to PFBR and studied the flow properties of Ar gas, which was heated by the liquid sodium from below. They observed a logarithmic increase in the CTD with the increase of Ra from 10^8 to 10^{10}. Hemanath et al. (2007) have further shown that cooling the sidewalls markedly reduce CTD, but the use of helium gas instead of Ar does not affect the CTD significantly. Yamakawa and Sakai (1986) arrived at similar conclusions in an experiment on IHX with water as a fluid. Earlier Timo (1954) had studied the heat transfer in an annulus using experiments and theoretical modeling. In all the above experiments, the researchers typically observed a pair of convective rolls.

A large number of numerical simulations have been performed to understand the flow profile in the annulus of PFBR. Yamakawa and Sakai (1986) performed numerical simulations using a three-dimensional computational fluid dynamics code THERVIS III and found consistency with experiments. They observed that the flow patterns inside the annulus depends quite critically on the radiative heat loss from the sidewalls. Hemanath et al. (2007) simulated the system using the PHEONICS code and observed behavior simi-

---

* Corresponding author. Tel.: +91 512 2597396; fax: +91 512 2590914.
E-mail address: mkv@iitk.ac.in (M.K. Verma).
URL: http://home.iitk.ac.in/~mkv (M.K. Verma).

0306-4549/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.anucene.2013.07.021

Nomenclature

$\bar{x}$	resolved position (m)
$\bar{u}$	resolved velocity (m s^{-1})
$\bar{p}$	resolved pressure (kg m^{-1} s^{-2})
T	resolved temperature (K)
T_0	reference temperature (K)
U	mean velocity (m s^{-1})
P	mean pressure (kg m^{-1} s^{-2})
ρ	density (kg m^{-3})
β	thermal expansion coefficient (K^{-1})
ν_0	viscosity (m^2 s^{-1})
ν_t	turbulent viscosity (m^2 s^{-1})
α	thermal diffusivity (m^2 s^{-1} K)
α_t	turbulent diffusivity (m^2 s^{-1} K)
g	gravity (m s^{-2})
k	turbulent kinetic energy (m^2 s^{-2})
ε	turbulent dissipation rate (m^2 s^{-3})
l	turbulent length scale (m)
D	diameter of the annulus (m)
δ	annulus gap (m)
L	length of annulus (m)
H	height of the whole geometry (m)
ΔT	axial temperature difference
Q	heat flux (J)
Pr	Prandtl Number
Pr$_t$	turbulent Prandtl Number
Ra	Rayleigh Number
ϵ_1	$= \frac{L}{\pi D}$, $\epsilon_2 = \frac{L}{\delta}$
Λ_1	$= 1/\epsilon_1$, $\Lambda_2 = 1/\epsilon_2$: aspect ratios
Ra	$= \frac{g\beta\Delta T L^3}{\nu^2}$ Pr

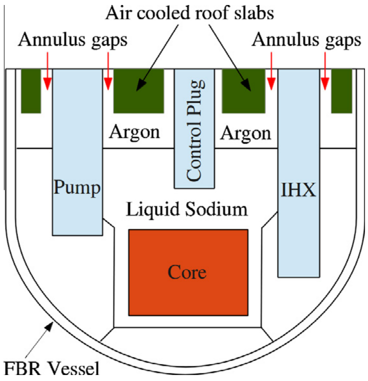

Fig. 1. A schematic view of the prototype fast breeder reactor (PFBR).

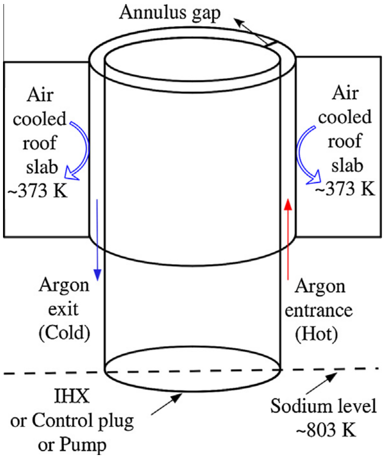

Fig. 2. A zoomed view of the annulus of a PFBR. The air gap between the wall and the component is around 5–10 mm.

lar to their experiments. Velusamy et al. (1998) applied computer codes THYC-2D and COND-2D to simulate the flow and studied the effects of air inlet temperature and gap width on the turbulent flow. In a recent numerical work Paliwal et al. (2012) studied the flow pattern for much larger gap-width (50 mm) and diameters, and observed very different results. For example, they obtain two to ten convective cells for different aspect ratios. Baldassari et al. (1984) and Goldstein and Joly (1979) have also simulated the flow in the annulus and observed results similar to those described above.

Numerical simulation of convective turbulence is very challenging since it involves a wide range of length and time scales. We require very high resolution direct numerical simulations (DNS) with small time steps to resolve the large range of length- and time scales. These direct simulations require large supercomputers, hence they are not always practical for many complex engineering problems. An alternate approach is to perform simulations on a coarser grid with an appropriate modeling of the small-scale turbulence. Two major methods in this class are the Reynolds-averaged Navier Stokes (RANS) and the Large Eddy Simulations (LES). In the RANS approach, we time average the Navier–Stokes equation which produces an additional term, generally referred to as Reynolds stress. This term is simplified using appropriate turbulence models. In the LES schemes, an spatial filtering is performed, and enhanced diffusive parameters are employed to take care of the turbulence at smaller scales. In the present paper we employ a RANS scheme for studying the turbulent convection in PFBR, with the Reynolds stress modeled using the k–ϵ model.

In this paper we simulate the convective turbulence in the annulus of PFBR using OpenFOAM (Open Source Field Operation and Manipulation). Using these simulations we study the circumferential temperature difference (CTD), and the velocity and temperature profiles for two different geometries, including the geometry used in the COBA test facility at the Indira Gandhi Centre for Atomic Research (IGCAR), Kalpakkam, India. The largest Rayleigh number used for our simulation is 1.52×10^{10}. The numerical method and equations are discussed in Section 2, while the numerical results are described in Section 3. We conclude in Section 4.

2. Numerical method and geometrical configuration

As discussed in the introduction, a direct numerical simulation of convective flows for large Rayleigh numbers require extremely fine grid and a large amount of computer time. Therefore we resort to RANS type simulation with a coarser grid, and the Reynolds stress prescribed using k–ϵ model. We solve the following RANS-averaged incompressible fluid equations:

$$\frac{\partial \overline{u_j}}{\partial x_j} = 0, \tag{1}$$