Progress in Nuclear Energy 103 (2018) 243–253

Contents lists available at ScienceDirect

Progress in Nuclear Energy

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

Laminar fluid flow and heat transfer in non-circular sub-channel geometries of nuclear fuel bundle

Annamalai Moorthi[a,c], Anil Kumar Sharma[b,c,*]

[a] Bhabha Atomic Research Centre Facilities, Kalpakkam, India
[b] Safety Engineering Division, Fast Reactor Technology Group, Indira Gandhi Centre for Atomic Research, Kalpakkam, India
[c] Homi Bhabha National Institute (HBNI), Mumbai, India

ARTICLE INFO

Keywords:
Non-circular sub-channel geometries
Laminar flow
Poiseuille number
Average Nusselt number
Nuclear fuel bundle

ABSTRACT

Numerical simulation of fully developed laminar fluid flow and heat transfer in non-circular sub-channel geometries representing a typical nuclear fuel bundle is carried out. The governing equations of fluid flow and heat transfer are reduced in the form of Poisson's equation and implemented in COMSOL multi physics code as classical PDEs. The fully developed laminar flow equations are solved on the non-circular sub-channel and fuel bundle geometries in a two dimensional domain cut in the longitudinal cross section. The domain is solved for the flow velocities in the out of plane direction by applying the pressure gradient as the boundary condition. The study is focused on the detailed analysis of the Poiseuille number (fRe) and average Nusselt number ($\overline{Nu}$) in square and triangular pitched sub-channel geometries of rod bundle. These investigations are carried out for two distinct boundary conditions as axially uniform wall heat flux with uniform wall temperature in the periphery (H1 Boundary condition) and uniform wall heat flux in both axial and circumferential directions (H2 boundary condition) for different p/d ratios varying from 1 to 2. It is found that the fRe increases significantly from 6.5 to 24 for p/d ratio of 1–1.15 for the sub-channel shaped geometry. In case of the sub-channel shaped tube of the same p/d ratio, the fRe increases sharply upto p/d ratio ~1.15 and then starts decreasing slightly till the p/d ratio of 1.5. Analyses are also carried out for square pitched bundles of different sizes varying from 2 × 2 to 10 × 10 to find the variation of fRe with p/d and W/d in the laminar flow regime. The effect of bundle size on the variation of bundle friction factor (fRe) reveals that for an equal p/d and W/d ratios, the change in bundle friction factor (fRe) is within 5% and the effect of bundle sizes vanishes. Correlations are developed to predict the flow and heat transfer characteristics as a function of p/d ratio for non-circular sub-channels by least square regression analysis.

1. Introduction

The convective flow and heat transfer in circular and non-circular channels are of considerable importance in many industrial applications and especially in the nuclear reactor fuel bundles. The knowledge of pressure drop, flow and temperature distribution in these geometries are required, especially for the thermal hydraulic design calculations and safety related studies of reactor core. Extensive numerical simulations are required to estimate the thermal hydraulics safety margins as well as the safe limiting power of operation, under different conditions of nuclear reactor core. Traditionally, the safety margins are estimated by sub-channel analysis codes, which are applied at the bundle level as well as the partial symmetry of the core requiring a very large computational time. In past, the conservative estimates of the coolant and fuel temperatures were carried out by sub-channel analysis using semi-empirical correlations. Experimental and computational fluid dynamics works can reveal the actual velocity and temperature distributions in these sub-channels as compared to the sub-channel averaged coolant temperatures and to predict the hot spot regions. In the sub-channel analysis, the key input parameters like sub-channel loss coefficients and inter-channel loss coefficients are required to calculate the flow distribution within the fuel bundle. These are obtained experimentally on simple geometries. Even today, the application of CFD for thermal hydraulics analysis of full bundle is a challenge due to limitations of computational resources and large number of design calculations for different nuclear reactor operating regimes. However, CFD can be applied to the sub-channel level geometries to get more insight to the flow and temperature distribution within the rod bundle. During the emergency situations like Loss of Coolant Accident (LOCA), there can be a less cooling of fuel pins leading to ballooning, neighboring fuel pins can

---

[*] Corresponding author. Indira Gandhi Centre for Atomic Research, Kalpakkam, India.
E-mail address: aksharma@igcar.gov.in (A.K. Sharma).

https://doi.org/10.1016/j.pnucene.2017.10.009
Received 12 January 2017; Received in revised form 25 June 2017; Accepted 26 October 2017

touch each other and can form the cusped sub-channel (Duck and Turner, 1987). Gunn and Darling (1963) measured the friction factors in cusped channels of square pitched four pin bundles. An experimental investigation of peripheral temperature variation on the wall of a non-circular duct was carried out by Barrow et al. (1984). An experimental investigation of the turbulent flow and heat transfer characteristics in a four cusped channel with uniform wall temperature was studied in detail by Dutra et al. (1991). Mixed convection in 4-cusped duct was carried out for the estimation of fRe and average Nusselt number by Dong and Ebadian (1994). Hassan and Hunaehn (2008) numerically studied the incompressible flow and heat transfer in 2-cusped and 3-cusped ducts of wall channels. Rehme (1973) has proposed a simple method of predicting friction factor of turbulent flow in non-circular channels on the basis of geometry factor (fRe) for laminar flow. The experimental and analytical data of laminar friction factor and forced convection heat transfer in non-circular ducts were given by Shah and London (1971, 1978). CFD analysis of a non-circular duct similar to rod bundle sub-channel, requires a distinct wall boundary conditions like, axially uniform wall heat flux with uniform wall temperature in the periphery (H1 Boundary condition), uniform wall heat flux in both axial and circumferential direction (H2 boundary condition) or uniform wall temperature (T1 boundary conditions). Numerical analysis of this kind of heat flux boundary conditions on the rectangular geometry were carried out by Spiga and Morini (1996) and Barletta et al. (2003). Also, the forced turbulent heat convection in square duct with non-uniform wall temperature was carried out by Rivas et al. (2011).

In recent times, due to advancement of computational power and CFD techniques, the fluid flow and heat transfer characteristics of the rod bundle are studied using CFD (Kaiser and Zeggel, 1987; Lee and Jang, 1997; Baglietto and Ninokata, 2005, 2006; Cheng and Tak, 2006; Guo and Oka, 2015). Even today the application of CFD for thermal hydraulics analysis of full bundle is very difficult due to the computational resources and the design time (Hu and Fanning, 2011). In recent past rod bundle has been simulated using different turbulent models. It was shown that the k–ε model could not reproduce the basic characteristics of such flows rather than Reynolds-stress model, which can predict accurate modeling of rod bundle flows (Házi, 2005).

From the above literature survey, it is clear that, the functional forms of fRe and average Nusselt number ($\overline{Nu}$) are available only for conventional geometries like circular and non-circular shapes. However, the application of these functional forms for analysis of nuclear fuel bundles does not hold good as the bundles possess complex geometry and boundaries. Hence, applying these functional forms to nuclear fuel bundles lead to large uncertainties in fRe and average Nusselt number ($\overline{Nu}$). This in turn leads to inaccurate prediction of fluid flow and heat transfer characteristics of the fuel bundle as well as sub-channel geometries. Therefore, the present study is focused to apply the CFD model on 2D geometry representing the non-circular rod bundle sub-channels by solving the out-of-plane flow velocities rather than a conventional 3D CFD analysis. Analyses are carried for different pitch to diameter (p/d) ratios to estimate the laminar fiction factor and average Nusselt number. The influence of pitch to diameter (p/d) and width to diameter (W/d) ratio on fluid friction in square and hexagonal fuel bundle geometries are also investigated in detail for different bundle sizes ranging from 2 × 2 to 10 × 10.

2. Numerical simulation

2.1. Geometry of rod bundle sub-channels

The geometry of the computational domain considered in the present study is depicted in Fig. 1(a and b). The regular non-circular geometries are modified at the corners, by considering the fuel pins, to represent a square/triangular pitched sub-channel as shown in Fig. 1(b). The p/d equal to 1 represents a cusped channel and p/d equal to infinity represents a square/triangular channel. The four pin square pitched and seven pin triangular pitched fuel bundles are also considered for analysis. These pin bundles are selected as it represents of all types of sub-channels in rod bundles, i.e. interior, wall and corner sub-channels. The geometries of square pitched pin bundle of size 2 × 2 to 10 × 10 are also generated and investigated the effect of bundle size on flow and heat transfer characteristics.

2.2. Hydrodynamic model of non-circular geometry

The steady state fully developed Newtonian fluid flow and heat transfer are simulated by solving the following governing equations,

Continuity:

$$\frac{\partial \rho \boldsymbol{u}}{\partial \boldsymbol{x}} + \frac{\partial \rho \boldsymbol{v}}{\partial \boldsymbol{y}} + \frac{\partial \rho \boldsymbol{w}}{\partial \boldsymbol{z}} = \boldsymbol{0} \tag{1}$$

Momentum Equation in axial direction:

$$\rho\left(\boldsymbol{u}\frac{\partial \boldsymbol{w}}{\partial \boldsymbol{x}} + \boldsymbol{v}\frac{\partial \boldsymbol{w}}{\partial \boldsymbol{y}} + \boldsymbol{w}\frac{\partial \boldsymbol{w}}{\partial \boldsymbol{z}}\right) = -\frac{\partial \boldsymbol{P}}{\partial \boldsymbol{z}} - \left(\frac{\partial \tau_{xz}}{\partial \boldsymbol{x}} + \frac{\partial \tau_{yz}}{\partial \boldsymbol{y}} + \frac{\partial \tau_{zz}}{\partial \boldsymbol{z}}\right) \tag{2}$$

where,

$$\tau_{ij} = \mu\left(\frac{\partial \boldsymbol{U_j}}{\partial \boldsymbol{x_i}} + \frac{\partial \boldsymbol{U_i}}{\partial \boldsymbol{x_j}}\right) \tag{3}$$

For the simulation of steady state fully developed laminar Newtonian fluid flow through an arbitrary shaped cross-section of constant flow area straight ducts, the continuity and momentum equations are reduced to the Poisson's equation. The source term in the Poisson's equation is the constant pressure gradient along the length of the duct. The resulting governing equations for fully developed laminar flow in constant cross sectional area duct are (Bahrami et al., 2005)

$$\frac{\partial^2 \boldsymbol{w}}{\partial \boldsymbol{x}^2} + \frac{\partial^2 \boldsymbol{w}}{\partial \boldsymbol{y}^2} = \frac{\boldsymbol{1}}{\boldsymbol{\mu}}\frac{\partial \boldsymbol{P}}{\partial \boldsymbol{z}} \quad \text{and } \boldsymbol{w}=\boldsymbol{0} \text{ on domain boundary } (\boldsymbol{\Gamma}) \tag{4}$$

The fully developed flow in circular ducts is generally simulated in reduced dimensions using 1D or 2D axi-symmetric model. In the case of non-circular ducts, such flow can be simulated in 2D cross-section normal to the flow direction. Solution of the Poisson's equation with modified pressure gradient as the source term and no-slip boundary condition on domain walls leads to the longitudinal velocity distribution $\boldsymbol{w}(\boldsymbol{x}, \boldsymbol{y})$.

Integrating the velocity over the domain gives the average velocity as follows.

$$w_a = \frac{1}{A}\int_{\Gamma} w \, dA \tag{5}$$

Similarly integrating the wall shear stress on the boundaries of the domain gives the average wall shear stress as

$$\overline{\tau_w} = \frac{\int_{\Gamma} \tau \, ds}{\int_{\Gamma} ds} \tag{6}$$

The skin friction coefficient is estimated as

$$f = \frac{2\overline{\tau_w}}{\rho w_a^2} \tag{7}$$

and the frictional pressure drop is calculated from the wall shear stress as

$$\Delta P = \overline{\tau_w}\frac{SL}{A} \tag{8}$$

From the frictional pressure drop relationship, the product of friction factor (f) and Reynolds number (Re) gives a constant value and is called the Poiseuille number (f Re) in laminar flow regime. It is the Eigen value of Poisson's equation also called as geometry factor. This factor is solely determined from the geometrical cross section of the