Nuclear Engineering and Design 295 (2015) 162–172

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Thermal–hydraulic numerical simulation of fuel sub-assembly using a dedicated meshing tool

Thierry Cadiou*, Aakanksha Saxena[1]

CEA DEN/DER/SESI, 13108 Saint-Paul-Lez-Durance, France

ARTICLE INFO

Article history:
Received 28 January 2015
Received in revised form 24 August 2015
Accepted 2 October 2015
Available online 13 November 2015

ABSTRACT

As the CEA is involved in the pre-conceptual design phase of a Sodium-cooled Fast Reactor (SFR), the thermal–hydraulics modeling of sodium flow in the reactor core is a key scientific subject, not only due to the innovations proposed for the core design but also for the cost and the difficulties encountered to carry out experiments with sodium. Taking advantage of the progress made in numerical simulation and associated computational time, the sodium flow in a fuel pin sub-assembly is characterized in this work.

The fuel pin sub-assembly is composed of 217 fuel pins, each wrapped by spacer wire, and surrounded by a hexagonal tube. In order to overcome the main limitation on the required number of mesh cells for modeling such a complex geometry, an original meshing tool, developed for this purpose, was necessary and is presented in this paper.

This approach reveals to be of great help for modeling the sodium flow. Indeed, the pressure drop in the rod bundle is firstly evaluated. In addition, the local effects (at the scale of fuel pin) and global effects (at the scale of fuel pin bundle) for sodium velocity and temperature gradients for the alleged homogenization made by the spacer wire on the sodium flow are examined and clarified. Taking into account these results, the optimization of the fuel bundle geometry can be considered in order to homogenize the outlet temperature distribution in nominal condition. Lastly, the definition and the analysis of experiments on sub-assembly to be carried out in future experimental CEA platform, will also take advantage of this approach.

1. Introduction

In France, Fast Reactor strategy was approved in May 2008 at Ministry level and in September 2010, an agreement was published between CEA and French Government in order to conduct design studies of Generation IV (GEN IV) Sodium-cooled Fast Reactors (SFRs). An innovative core design, the CFV (French acronym for Low sodium Void effect), was first suggested by the CEA (Sciora et al., 2011) with the objective of reducing the probability of core meltdown and/or limiting energy release accident potentialities. Characterized by axial and radial heterogeneous geometry design, this particular core features a negative sodium void worth and allows improving the natural behavior during unprotected thermal–hydraulics transients.

A thorough understanding of the thermal–hydraulic core behavior and the characterization of the sub-assembly cooling is one of the key parameters for any reactor. The main parameters to be evaluated in the thermal–hydraulic studies are the pressure, temperature and velocity distributions in the sub-assembly that helps to determine the following quantities:

- The total sub-assembly pressure drop.
- The clad temperature and especially the clad maximum temperature.
- The hexagonal can temperature for thermal–mechanical analysis.
- The temperature gradients and the maximum temperature of the coolant liquid sodium also referred as hot-spot.

The method used hitherto for this evaluation was based on the use of correlations called mixing laws (Cheng and Todreas, 1986), which provided the main thermal–hydraulic characteristics of the main flow, useful for the sub-assembly design, including optimization of flow areas. However, this approach was limited to the knowledge of the macroscopic flow. On the contrary, the numerical simulation provides a precise and local knowledge of the temperature, pressure and velocity fields throughout the fluid domain of the sub-assembly.

* Corresponding author. Tel.: +33 4 42 25 66 17.
E-mail address: thierry.cadiou@cea.fr (T. Cadiou).
[1] Former PhD student at the CEA, DEN, DER/SESI/LE2S.

http://dx.doi.org/10.1016/j.nucengdes.2015.10.001
0029-5493/

Nomenclature

D_h	hydraulic diameter (M)
D_p	diameter of fuel pin in SFR sub-assembly (M)
D_w	diameter of the spacer wire in SFR sub-assembly (M)
f	friction coefficient in the pressure losses (–)
Gap	distance between outer pin and the wrapper tube (M)
H	helical pitch of the spacer wire (M)
ε	energy dissipation rate (m^2/s^3)
k	kinematic energy (m^2/s^2)
L_z	total height of the fuel bundle (M)
Nu	Nusselt number (–)
P	pitch of the fuel pin (M)
$\triangle P$	pressure loss (Pa)
Pr	Prandtl number (–)
Pr_t	Prandtl turbulent (–)
r	coordinate in radial direction (M)
Re	Reynolds number (–)
T	Temperature (K)
u	axial velocity (m/s)
y^+	wall normal dimensionless distance (–)
z	coordinate in axial direction (M)
$\rho(T)$	fluid density (kg/m^3)
c_p	specific heat capacity (J/kg K)
$\lambda(T)$	thermal conductivity (W/m K)
$\mu(T)$	dynamic viscosity (kg/m s)
$\varepsilon(x, y)$	X coordinate in the conformal space
$\eta(x, y)$	Y coordinate in the conformal space

Abbreviations

CAD	computer aided-design
CEA	Alternative Energies and Atomic Energy Commission (Commissariat à l'Energie Atomique et aux Energies Alternatives in French)
CFD	Computational Fluid Dynamics
CFV	French acronym for Low sodium void effect core
DNS	Direct Numerical Simulation
GEN-IV	Generation IV
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier–Stokes
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
SFR	Sodium-cooled Fast Reactor
k,ε	the k,ε turbulence model used in RANS
2D	2 dimensional
3D	3 dimensional

The difficulty of the numerical simulation arises from the complex geometry of the sub-assembly, which requires a large number of mesh cells for the fluid domain and hence long computational domain. However, the interest of this approach is evident since it will gives access to the design features such as pressure loss or the maximum temperature gradients in the sub-assembly. In this regard, a special mesh tool was developed to reduce the number of mesh cells and the computational time without compromising the quality of results.

In the first part, a review of Reynolds-Averaged Navier–Stokes (RANS) is presented. Then, the main geometrical characteristics of the SFR sub-assembly are recalled. The concepts of numerical CFD (Computational Fluid Dynamics) simulation and the models used in the industrial STAR-CCM+ software are then explained. The meshing tool is discussed in the third section as well as the modeling choices and the boundary conditions selected for the simulation data set. In the fourth and last part, the main results obtained on the SFR sub-assembly are presented.

2. Review of RANS and LES studies

This section presents the numerical studies using RANS approach.

Gajapathy et al. (2015), Natesan et al. (2010), Péniguel et al. (2010), Kim and Ahmad (2005) and Ahmad and Kim (2006) successfully analyzed the three-dimensional turbulent flow and heat transfer in wire-wrapped assembly based on RANS calculations. Raza and Kim (2008) performed similar studies and concentrated on the wire-spacer shape. Taking into account the difficulty to properly mesh the complex geometry of a fuel sub-assembly, these results were encouraging, since it appears that RANS computations can reproduce the correct flow description.

RANS modeling approaches for tight lattice sub-assembly (pitch to diameter ratio, $P/D_p < 1.3$) without spacer wires were assessed by Chang and Tavoularis (2007) and Baglietto (2007). These studies showed that anisotropic effects are important and they cannot be accounted for in RANS approach. Pointer et al. (2008) focused on the benchmarking of flow field predictions on in 7 fuel pin by comparing the LES (Large Eddy Simulation) and RANS calculations showing a good comparison between LES and RANS results. Pointer et al. (2009a) extended these studies to 217 fuel pin assemblies in support of initial efforts to benchmark heat transfer predictions using the RANS models against conventional sub-channel models.

Rolfo et al. (2012) examined the effect of the turbulent Prandtl number. Since the low value of the Peclet number, higher turbulent Prandtl number, as suggested in Cheng and Tak (2006), has also been tested in some configurations (7-pins and 19-pins geometries at $Re = 25000$), but without noticing a large variation of results (the percentage variation of the Nusselt number for the Dirichlet boundary condition is below 1% and below 3% in the case of a Neumann boundary condition).

Recently, Saxena (2014) focuses on the numerical simulation of sodium flow in wire wrapped subassembly of SFR using RANS approach, but also more accurate CFD methods like LES and DNS (Direct Numerical Simulation) to study the thermal field of sodium in sub-assembly. This work helps in understanding the flow temperature distribution and thermal fluctuations on structures in presence of sodium.

Using experimental results based on 19 pin sub-assemblies, Fricano concludes the predictive capabilities of the CFD approach (Fricano and Baglietto, 2014).

Apart from the nominal conditions, the flow within the sub-assembly was also analyzed in natural circulation in order to assess the influence of its geometrical characteristics (Doda et al., 2010) in transient states.

Relatively to the sodium flow specificities, the consequences of a sodium low Prandtl number on the flow were largely studied by Grötzbach (2013).

These approaches are synthesized in Table 1.

3. SFR sub-assembly

The fuel sub-assembly consists of a hexagonal duct wall with fuel pins (see Fig. 1). The cylindrical fuel pins are arranged in a triangular lattice. The fuel pins are separated from each other by a helical wire wrap called a spacer wire. The role of the spacer wire is:

- to avoid pin-to-pin contact,
- to guard the pin bundle against the flow induced vibrations, and
- to avoid the trapping of the coolant.