Nuclear Engineering and Design 308 (2016) 182–193

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

A computationally efficient method for full-core conjugate heat transfer modeling of sodium fast reactors

Rui Hu *, Yiqi Yu

Nuclear Engineering Division, Argonne National Laboratory, 9700 S. Cass Avenue, Argonne, IL 60439, USA

HIGHLIGHTS

- Developed a computationally efficient method for full-core conjugate heat transfer modeling of sodium fast reactors.
- Applied fully-coupled JFNK solution scheme to avoid the operator-splitting errors.
- The accuracy and efficiency of the method is confirmed with a 7-assembly test problem.
- The effects of different spatial discretization schemes are investigated and compared to the RANS-based CFD simulations.

ARTICLE INFO

Article history:
Received 3 June 2016
Received in revised form 15 July 2016
Accepted 18 August 2016
Available online 8 September 2016

JEL classification:
K. Thermal Hydraulics

ABSTRACT

For efficient and accurate temperature predictions of sodium fast reactor structures, a 3-D full-core conjugate heat transfer modeling capability is developed for an advanced system analysis tool, SAM. The hexagon lattice core is modeled with 1-D parallel channels representing the subassembly flow, and 2-D duct walls and inter-assembly gaps. The six sides of the hexagon duct wall and near-wall coolant region are modeled separately to account for different temperatures and heat transfer between coolant flow and each side of the duct wall. The Jacobian Free Newton Krylov (JFNK) solution method is applied to solve the fluid and solid field simultaneously in a fully coupled fashion. The 3-D full-core conjugate heat transfer modeling capability in SAM has been demonstrated by a verification test problem with 7 fuel assemblies in a hexagon lattice layout. Additionally, the SAM simulation results are compared with RANS-based CFD simulations. Very good agreements have been achieved between the results of the two approaches.

1. Introduction

Detailed core-wide temperature predictions of sodium-cooled fast reactors (SFR) need to be performed in order to ensure that all the design criteria are satisfied, such as fuel, cladding, and sodium outlet temperatures under various conditions. Additionally, one important design requirement for SFR is the knowledge of the temperature on the hexagonal ducts for a thermo-mechanical analysis. This is particular important to ensure the passive safety of the reactor under the unprotected accident conditions if the reactor control system fails to function and the reactivity feedback from structural deformation such as core radial expansion is significant (Chang et al., 2006). This information requires a good evaluation of the inter-assembly flow and heat transfer in this region. The physical phenomena are particularly complicated and require a reliable modeling of the whole core including the inter-assembly region. For repeated detailed core-wide temperature calculations, a computationally efficient modeling approach is necessary.

Multiple levels of modeling are possible for the analysis of the whole core thermal hydraulic behavior, depending on the purpose of the calculation and the accuracy required. Generally, the 3D full core modeling using the computational fluid dynamics (CFD) approach is prohibitively expensive. In order to reduce the amount of computational effort but to maintain a reasonable accuracy, a low resolution RANS approach is developed by Hu and Fanning (2013). In this approach a momentum source model was developed to mimic the effects of the wire-wraps without explicitly modeling the geometrical representation of the wire. Similar low grid resolution approaches of rod bundle simulation have been reported by Roelofs et al. (2012) and Viellieber and Class (2015). Such low-resolution approaches are required to simulate the overall behavior of the coolant flow in a complete fuel assembly or even

* Corresponding author.
E-mail address: rhu@anl.gov (R. Hu).

http://dx.doi.org/10.1016/j.nucengdes.2016.08.018
0029-5493/

multiple fuel assemblies interacting with each other. The low-resolution approaches can provide more local details than typically can be deducted from subchannel simulations, while maintaining a reasonable accuracy compared to detailed CFD analyses for a complete fuel assembly that are too costly for engineering purposes. The low-resolution approach can be very useful to study the effects of partial flow blockage and local deformations, but still too expensive for repeated full-core simulations.

Over the past several decades, much effort has been made to develop the sub-channel analysis codes to predict the detailed coolant temperatures for all the sub-channels in a fuel assembly or reactor core. More recent efforts include the development of SE2-ANL (Yang and Yacout, 1995), MATRA-LMR (Kim et al., 2002) and TRIO_U subchannel module (Tenchine et al., 2012). Yu et al. (2015) developed a porous medium modeling approach to predict SFR duct wall temperatures, which is much less computationally expensive than conventional CFD simulations that explicitly represent the wire-wrap and fuel pin geometry. However, even the porous medium or subchannel description of each subassembly and the inter-assembly space is still too computational expensive for quotidian analyses.

System analysis is favorable to describe the global behavior of the whole core during transient situations. In traditional reactor safety analysis, the reactor core is modeled with several 1D parallel channels to take into account the main different core regions, with a possible by-pass channel to simulate amassed flow rate in the inter-assembly zone (Tenchine, 2010). However, it is very cumbersome to obtain the detailed temperature predictions of inter-assembly gaps and duct walls, which requires a full-core one-to-one representation of fuel assemblies and inter-assembly gaps.

For efficient and accurate full-core heat transfer calculations in SFR, especially for structure temperature predictions, a 3-D full-core conjugate heat transfer modeling capability has been developed for an advanced system analysis tool, SAM, being developed at Argonne National Laboratory. The hexagon lattice core can be modeled with 1-D parallel channels representing the subassembly flow and 2-D duct walls and intra-assembly gaps. Note that each fuel assembly can be modeled as multiple channels and the 6 sides of the hexagon duct wall are modeled separately to account for different temperatures and heat transfer between inner assembly flow and each side of the duct wall. A core lattice model has been developed in SAM so that it can, based on very simple input descriptions, generate all the core channels and inter-assembly gaps, and build the connections among them.

Hu et al. (2015) provided an overview and update of the SAM code development. The in-depth discussion of the underlying physics models and numerical methods were report by Hu (2015). This paper presents a computationally efficient conjugate heat transfer modeling method in SAM for radial heat transfer in the reactor core. The 3-D full-core conjugate heat transfer modeling capability is demonstrated by a verification test problem with 7 fuel assemblies in a hexagon lattice layout. The simulation results are compared with the RANS-based CFD simulation using the commercial CFD code STAR-CCM+ (CD-adapco, 2014). Good agreements are achieved between the results of the two codes.

2. SAM conjugate heat transfer modeling

2.1. SAM overview

SAM is an advanced system analysis tool being developed at Argonne National Laboratory under the U.S. DOE's Nuclear Energy Advanced Modeling and Simulation (NEAMS) program. The code is aimed to solve the tightly-coupled physical phenomena including fission reaction, heat transfer, fluid dynamics, and thermal–mechanical response in the SFR structures, systems and components in a fully-coupled fashion but with reduced-order modeling approaches to facilitate rapid turn-around for design and safety optimization studies. As a new code development, the initial effort focused on developing modeling and simulation capabilities of the heat transfer and single-phase fluid dynamics responses in the SFR systems.

SAM utilizes a modern application framework MOOSE (Gaston et al., 2009), the underlying meshing and finite-element library libMesh (Kirk et al., 2006), and linear and non-linear solvers PETSc (Balay et al., 2016) to leverage the available advanced software environments and numerical methods. The physics modeling and mesh generation of individual reactor components are encapsulated as component classes in SAM along with some component specific models. A set of components has been developed based on the finite element models (for fluid flow and heat transfer), and the system simulation capabilities of general thermal-hydraulics systems have been demonstrated and benchmarked with the EBR-II tests (Hu and Summer, 2016).

Fluid dynamics is the main physical model of the SAM code. SAM employs a one-dimensional transient model for single-phase incompressible but thermally expandable flow. The transport equations for one-dimensional, single-phase flow can be described by a set of partial differential equations. The mass, momentum, and energy conservation equations are closed by the equation of state for the fluid. After some simplifications, the conservative form of the governing equations can be written in Eq. (1), and it is solved in SAM using the primitive variable based approach. The details of the finite-element model for single-phase incompressible but thermally expandable flow and the associated stabilization scheme in SAM can be found in Hu (2015).

$$\begin{aligned}
&\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial z}=0\\
&\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u u+p)}{\partial z}=-\rho g-\frac{f}{D_e}\frac{\rho u|u|}{2}\\
&\frac{\partial(\rho H)}{\partial t}+\frac{\partial(\rho u H)}{\partial z}=q'''\\
&\rho=\rho(p,T)
\end{aligned} \tag{1}$$

In which f: the friction coefficient; D_e: equivalent hydraulic diameter. When considering the convection heat flux from solid surface q''_s, $q'''=q''_s P_h/A_c$, where P_h and A_c respectively denote heated perimeter and cross-sectional area of the coolant channel.

Heat structures model the heat conduction inside the solids and permit the modeling of heat transfer at the interfaces between solid and fluid components. Heat structures are represented by one-dimensional or two-dimensional heat conduction in Cartesian or cylindrical coordinates. Temperature-dependent thermal conductivities and volumetric heat capacities can be provided in tabular or functional form either from built-in or user-supplied data. The modeling capabilities of heat structures can be used to predict the temperature distributions in solid components such as fuel pins or plates, heat exchanger tubes, and pipe and vessel walls, as well as to calculate the heat flux conditions for fluid components.

The thermal conduction inside the solid structures is governed by a diffusion equation:

$$\rho C_p \frac{\partial T}{\partial t}-\nabla(k\nabla T)-Q'''=0 \tag{2}$$

where k is the solid thermal conductivity, and Q''' is the volumetric internal heat source in the solid. It can be discretized in both Cartesian and cylindrical coordinates. The types of boundary conditions (BC) include: (1) Dirichlet BC, $T=T_0$; (2) Neumann BC, $k\nabla T=q''_0$; or (3) convective BC: $-k\nabla T=h\cdot(T-T_{fluid})$.