Nuclear Engineering and Design 326 (2018) 333–343

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

CFD analysis and experimental validation of steady state mixed convection sodium flow

U. Bieder[a,*], G. Ziskind[b], A. Rashkovan[c]

[a] DEN-STMF, CEA, Université Paris-Saclay, F-91191 Gif-sur-Yvette, France
[b] Department of Mechanical Engineering, Ben-Gurion University of the Negev, P.O. Box 653, Beer-Sheva 84105, Israel
[c] Nuclear Research Center Negev (NRCN), P.O. Box 9001, Beer-Sheva 84190, Israel

A R T I C L E I N F O

Keywords:
Low Prandtl number fluid
Thermal stratification
Mixed convection
Wall bounded jet
Sodium

A B S T R A C T

Formation and destruction of thermal stratification can occur under certain flow conditions in the upper plenum of sodium cooled fast breeder reactors (SFR). The flow patterns in the hot sodium pool of SFR upper plenum are very complex and include zones of free and wall-bounded jets, recirculation and stagnation areas. The interaction of sodium flow and thermal stratification has been analyzed experimentally at CEA in the SUPERCAVNA facility. The facility consists of a rectangular cavity with heated side walls. The flow is driven by a wall-bounded cold jet at the bottom of the cavity. Experimental data of the temperature distribution inside the cavity are available for steady-state and transient flow conditions. In the present study, the steady state experiments are analyzed with the CEA CFD reference code TrioCFD and the commercial code FLUENT employing Reynolds Averaged Navier-Stokes (RANS) equations. The SUPERCAVNA modeling is based on preliminary separate effects' studies of wall-bounded jets and free convection benchmark simulations for low Prandtl number fluids.

It was found that a two-dimensional treatment is sufficient to reproduce correctly the measured thermal stratification for steady-state SUPERCAVNA experiments. However, it is necessary to take into account conjugated heat transfer between cavity sodium flow and side-wall heating channel. Using temperature-dependent physical properties was also found to be an important factor in simulating the experiments correctly. Applying the Boussinesq approximation to study the impact of buoyancy on the vertical flow momentum was found to be justified. Turbulence modelling is necessary for a successful simulation of the experiments. For practical reasons, the analysis was restricted to turbulence models of the k-ε family. High Re k-ε models, in either standard, realizable or RNG formulations, do not lead to significant differences in the calculated temperature fields. Nonlinear eddy viscosity modelling might improve the quality of the simulation.

1. Introduction

High molecular thermal conductivity of liquid metals makes them especially attractive for applications where effective heat transfer is of primary concern. The fields of application of liquid metals as heat transfer agents span from steel production by continuous casting and crystal growth in semi-conductor production to applications in liquid metal cooled fission nuclear reactors and target cooling free surface jets employed in the framework of International Fusion Materials Irradiation Facility (IFMIF) (Grotzbach, 2013).

1.1. Liquid metals as coolant

The flow and heat transfer applications of liquid metal cooling are spanned from forced through mixed to free convection regimes. For the case of comparable importance of buoyancy and externally applied momentum source, conditions of mixed convection prevail. Mixed convection is encountered in many industrial applications, including heat exchangers and nuclear reactors to name a few (Marocco et al., 2017). The mixed convection regime can be further subdivided into the buoyancy opposed, assisted and cross[1] flows with the externally applied flow having downward, upward or horizontal component respectively (Gebhart et al., 1991). While for buoyancy assisted turbulent flows wall shear stress is higher than that for the pure forced convection case, it is lower for the buoyancy opposed ones. Further from the wall the situation is reversed: the fluid stress is higher for the buoyancy opposed and lower for the buoyancy added flow. This general description (Gebhart et al., 1991) is expected to hold for the case of liquid metals

* Corresponding author.
E-mail addresses: ulrich.bieder@cea.fr (U. Bieder), gziskind@bgu.ac.il (G. Ziskind).

[1] out of scope of the present study

https://doi.org/10.1016/j.nucengdes.2017.11.028
Received 23 June 2017; Received in revised form 11 November 2017; Accepted 17 November 2017
0029-5493/

flow and heat transfer as well (Jackson, 1983).

Modelling liquid metal flows in all convection regimes for industrial scale applications is mostly performed employing Reynolds Averaged Navier-Stokes (RANS) Equations for momentum and Reynolds heat-momentum analogy with constant turbulent Prandtl number assumptions for solving the energy equation. As more advanced models are constantly being introduced (see Roelofs et al. (2015) for recent review of liquid metals modeling challenges), detailed experimental data are needed for model validation. However, measurements of flow and temperature fields in liquid metal applications are recognized as being much more challenging than those in e.g. water or air due to the optical opaqueness and relatively high temperatures of the former (Schulenberg and Stieglitz, 2010).

Depending on the specific physical conditions in question, the needed acceptable turbulence models range from locally applicable turbulent Prandtl number formulations, over algebraic flux models, to full second order models in RANS as well as in LES, all with special liquid metal extensions (Grotzbach, 2013). The geometry of the real life applications of liquid metal cooled equipment is usually complicated and pose a challenging problem for employing high fidelity turbulence models. It is therefore important to acquire more experimental data and to better assess the limits of validity of computationally affordable turbulence models for their applications in predicting of flow and temperature fields in industrial scale applications of liquid metals cooling systems.

Of the three convection regimes, predicting liquid metal flow and heat transfer in the mixed convection regime was identified as particularly challenging (Roelofs et al., 2015). Experiment and modelling of mixed convection regime in liquid metal cooled equipment will be presented in the present study. The geometry and boundary conditions of the experiment chosen were set to mimic the flow conditions expected during certain operating conditions in a general pool type Sodium Fast Reactor (SFR).

1.2. Hot pool flow field in sodium cooled fast breeder reactors

SFRs were operated in a number of countries including USA, Russia, China, France, Germany, Japan and India. Three sodium cooled fast breeder reactors, Rapsodie, Phénix and Superphénix, have been built in France in the last 50 years. Many thermal-hydraulic studies were performed to support design and safety analysis for Superphénix and for the European Fast Reactor project (Tenchine, 2010). Initially based on analytical and experimental methods, thermal-hydraulic studies also progressively included numerical simulation.

The study presented here concerns a temperature-stratified liquid sodium flow which may be established under certain operating conditions in the upper plenum (also called hot pool) of pool type SFRs. The experimental work on the subject was initiated in the early 1970s with reference to Phénix and Superphénix plants (Vidil et al., 1988), where liquid sodium coolant is circulating in the upper plenum as shown in Fig. 1.

The core delivers hot sodium beneath the core head plug. The sodium flux is deflected vertically and transported to the intermediate heat exchangers (IHX) at the other end of the upper plenum, where it is driven by natural convection through the IHX to the lower plenum. The flow conditions in the plenum are highly complex, involving jet- and recirculating zones, where the sodium flows at velocities which are significantly lower than its main-stream velocities near the core outlet.

During certain operating transients, the core outlet temperature may change rapidly with time. For example, an emergency shutdown system activation results in a sharp temperature reduction. These temperature variations result in density changes which may affect flow conditions, particularly in regions of low velocity. Buoyancy effects may cause thermal stratification characterized by fluid distribution into layers with temperature increasing from the bottom up – case of stable stratification. Once established, stratification conditions may last for a long time (Vidil et al., 1988).

In order to understand the physical phenomena in the upper plenum and develop the modelling capability, thermal-hydraulic studies use two complementary approaches to predict realistically temperature distribution and flow fields:

- Numerical models capable of simulating multi-dimensional flow with complex geometries and multiple physical effects;
- Reduced-scale experimental models, using liquid sodium or other fluids, as a means for models' validation.

The numerical and experimental model approaches are both approximate. The accuracy of computer codes depends on the mathematical modelling, with the correct modelling of turbulence in stratified flows of liquid metals being particularly complex. The physical modelling should respect all non-dimensional numbers, being potentially impossible for sodium flow due to its low Prandtl number. Thus, it is necessary to perform experiments with sodium to develop capability to numerically predict the real SFR plant conditions in an experimentally validated manner.

A simplified experimental representation of the upper plenum of a pool type SFR is shown on the right of Fig. 1. The SUPERCAVNA facility geometry resembles simplified rectangular, temperature-controlled cavity. This set-up involves the two most essential phenomena occurring in the plenum:

- Thermal stratification imposed by the heated wall,
- Recirculation zones, imposed by the wall jet.

The objective of the presented study is to develop and validate a modelling and calculation strategy by analyzing the SUPERCAVNA experiment. This strategy should be applicable in the future to full scale SFR simulations what includes robustness and fast running performance. As a consequence, turbulence modelling was restricted to wall modelled methods of the k-ε family (high Re modelling).

2. The Supercavna facility and experimental conditions

2.1. Geometry

The SUPERCAVNA facility consisted of a rectangular cavity as shown in Fig. 2, connected at the bottom to rectangular inlet and outlet channels. The facility dimensions are given in Table 1.

All the cavity walls and channels are thermally insulated. Two distinct sodium test loops are operated to control the temperature and flow conditions independently in the inlet channel and the side wall heating channel – see Vidil et al. (1981, 1988) for further details. In the first flow loop, the sodium flow forced into the inlet channel induces a recirculating flow in the cavity.

In the second flow loop, for the vertical wall heating, the inlet of the side-wall heating channel is located at mid-height of the cavity. Sodium is forced downward to the bottom of the heating element, then turns upwards, and finally exits at the top of the facility. At the bottom, a 20 mm gas jacket limits heat transfer to the channel – denoted "insulation" in Fig. 2 (right).

2.2. Physical properties

The physical properties of liquid sodium are given in Table 2 for the relevant temperature range of 500 K to 600 K (Fink and Leibowitz, 1995). Physical properties of the INOX wall separating heating channel and cavity have been given by the experimentalists and are added to Table 2.