ARTICLE IN PRESS

Nuclear Engineering and Design xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Validation of ANSYS CFX for gas and liquid metal flows with conjugate heat transfer within the European project THINS

A. Papukchiev *, S. Buchholz

Gesellschaft fuer Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstr. 14, 85748 Garching n. Munich, Germany

HIGHLIGHTS

- ANSYS CFX is validated for gas and liquid metal flows.
- L-STAR and TALL-3D experiments are simulated.
- Complex flow and heat transfer phenomena are modelled.
- Conjugate heat transfer has to be considered in CFD analyses.

ARTICLE INFO

Article history:
Received 30 December 2015
Received in revised form 21 July 2016
Accepted 25 July 2016
Available online xxxx

ABSTRACT

Within the FP7 European project THINS (Thermal Hydraulics of Innovative Nuclear Systems), numerical tools for the simulation of the thermal-hydraulics of next generation rector systems were developed, applied and validated for innovative coolants. The Gesellschaft fuer Anlagen- und Reaktorsicherheit (GRS) gGmbH participated in THINS with activities related to the development and validation of computational fluid dynamics (CFD) and coupled System Thermal Hydraulics (STH) – CFD codes. High quality measurements from the L-STAR and TALL-3D experiments were used to assess the numerical results. Two-equation eddy viscosity and scale resolving turbulence models were used in the validation process of ANSYS CFX for gas and liquid metal flows with conjugate heat transfer. This paper provides a brief overview on the main results achieved at GRS within the project.

1. Introduction

For the long-term development of nuclear power, innovative nuclear systems such as Generation IV reactors and transmutation systems need to be developed for meeting future energy needs. The design of such facilities and the evaluation of their safety properties is a challenging and interdisciplinary process. This process is supported by the experience gained with the design and operation of current reactors as well as by the scientific and technological progress in the last decades. The increasing computer power allows the application of more sophisticated numerical approaches in the technical analyses, providing in-depth view and improved understanding of the investigated phenomena.

Thermal-hydraulics is recognized as one of the key scientific subjects in the development of innovative reactor systems (THINS, 2009). The European THINS project was devoted to important crosscutting thermal-hydraulic issues encountered in various innovative nuclear systems, such as advanced reactor core thermal-hydraulics, single phase mixed convection and turbulence, specific multiphase flow, and code coupling and qualification. The main scientific objectives of the THINS project were:

- Generation of a data base for the development and validation of new models and codes describing the selected crosscutting thermal-hydraulic phenomena. This data base contains both experimental data and data from direct numerical simulations (DNS).

Abbreviations: BP, bottom plate; CAD, computer aided design; CEL, CFX Expression Language; CFD, computational fluid dynamics; CHT, conjugate heat transfer; CIP, circular inner plate; DNS, direct numerical simulation; GRS, Gesellschaft fuer Anlagen und Reaktorsicherheit (GRS) gGmbH; ILW, inner lateral wall; IPT, inner pipe thermocouples; KIT, Karlsruhe Institute of Technology; LBE, lead-bismuth eutectic; LDA, Laser Doppler Anemometry; LES, Large Eddy Simulation; L-STAR, Luft – Stab, Abstandshalter, Rauheiten; OECD/NEA, Organization for Economic Cooperation and Development/Nuclear Energy Agency; OLW, outer lateral wall; Pr, molecular Prandtl number; Prt, turbulent Prandtl number; RANS, Reynolds Averaged Navier–Stokes; URANS, Unsteady Reynolds Averaged Navier–Stokes; SAS, Scale Adaptive Simulation; SST, shear stress transport; STH, System Thermal Hydraulics (Code); TC, thermocouple; TH, thermal-hydraulic; THINS, Thermal Hydraulics of Innovative Nuclear Systems; ZLES, Zonal Large Eddy Simulation.

* Corresponding author. Fax: +49 893200410553.
E-mail address: angel.papukchiev@grs.de (A. Papukchiev).

http://dx.doi.org/10.1016/j.nucengdes.2016.07.028
0029-5493/

Please cite this article in press as: Papukchiev, A., Buchholz, S. Validation of ANSYS CFX for gas and liquid metal flows with conjugate heat transfer within the European project THINS. Nucl. Eng. Des. (2016), http://dx.doi.org/10.1016/j.nucengdes.2016.07.028

- Development of new physical models and modeling approaches for more accurate description of the crosscutting thermal-hydraulic phenomena such as heat transfer and flow mixing, turbulent flow modeling for a wide range of Prandtl numbers, and modeling of flows under strong influence of buoyancy.
- Improvement of the numerical engineering tools and establishment of a numerical platform for the design analysis of the innovative nuclear systems. This platform contains numerical codes of various classes of spatial scales, i.e. system analysis, sub-channel analysis and computational fluid dynamics (CFD) codes, their coupling and the guidelines for their applications.

Over the last decades, interest has grown amongst the nuclear community for the use of CFD programs for the evaluation of different nuclear reactor safety issues, where traditional analysis tools show deficiencies. Within the FP7 European THINS project, system thermal-hydraulics (STH), CFD and coupled 1D-3D STH-CFD thermal-hydraulic simulations were carried out for Generation IV nuclear systems. These are dedicated to the analysis of the thermal-hydraulics of gas, liquid metal and supercritical water cooled reactors. Such innovative fluids represent challenge for the engineers, since these cooling media have different properties and behavior from the ones used in the current nuclear facilities.

The Gesellschaft fuer Anlagen- und Reaktorsicherheit (GRS) gGmbH participated with activities related to the development and validation of CFD and coupled STH-CFD codes. Emphasis was given to the calculation and analysis of gas and liquid metal experiments. The TALL-3D facility, operated by the KTH Royal Institute of Technology in Stockholm, is designed for thermal-hydraulic experiments with lead-bismuth eutectic (LBE) coolant at natural and forced circulation conditions. In the experiment T01 the primary pump is stopped, which leads to a complex transient with local 3D flow and heat transfer phenomena. The thermal-hydraulics in gas cooled reactors has been addressed with the analysis of the L-STAR experiment. The L-STAR test section consists of hexagonal flow channel with an air cooled cylindrical heated rod in its center. Since both experimental facilities are well instrumented, high quality measurements were used to support the validation of the commercial code ANSYS CFX for advanced reactor coolants. Different turbulence approaches based on RANS (Reynolds-Averaged Navier–Stokes), SAS (Scale Adaptive Simulation) and ZLES (Zonal Large Eddy Simulation) were utilized in the simulations. The objective of this overview paper is to briefly summarize the numerical effort and to highlight only the most significant achievements.

2. L-STAR experiment and ANSYS CFX model development

2.1. L-STAR/SL facility

The L-STAR facility (the acronym stands for the German **L**uft – **St**ab, **A**bstandshalter, **R**auheiten; which means air-rod, spacer, roughness) has been planned and constructed at Karlsruhe institute of Technology (KIT) for gas-cooling experiments for single-rod geometries (Gomez et al., 2013). It consists of two separate gas loops. The L-STAR/SL loop is driven by a single side channel compressor, while the L-STAR/LL loop includes two parallel strands with 3 compressors in series for each strand. In the present study, data only from the L-STAR/SL facility is used. A 3D view of the facility, showing the main components and the test section, is shown in Fig. 1. The coolant in this loop is air at low pressure, but other gases such as N_2 or CO_2 might be considered in the future. With this construction the following range of process variables can be achieved:

- Mass flow: up to 0.33 kg/s
- Gas pressure: 0.1–0.15 MPa (in the future: up to 0.27 MPa)
- Inlet temperature to test section: ambient (7–40 °C)
- Outlet temperature from test section: up to 200 °C allowed.

The test section in this loop consists of a hexagonal flow channel (length between upper and lower flange surfaces 3398 mm, inner width-across-flats approximately 67 mm), which contains a cylindrical rod in central position.

2.2. Instrumentation in the L-STAR/SL facility

The test section flow channel is equipped with two optical windows, which allow access for optical flow measurement methods, such as Laser Doppler Anemometry (LDA). With the LDA system the flow velocity distribution in the hexagonal channel is measured. Among the flow structure measurement instruments, the Laser-Doppler Profile sensor, which was specially developed for the L-STAR facility, deserves attention. This instrument was developed at TU Dresden. Its optical setup enables the simultaneous measurement of particle velocities (as known from classical LDA systems) and relative position of the individual particle trajectories within the measurement volume. The configuration used at L-STAR features a measurement volume length >1 mm, while the spatial resolution is less than 20 μm. It is well suited for flow velocity measurements in the vicinity of rod surface structures (ribs, etc.) (Arbeiter et al., 2011).

The cylindrical rod has a diameter of 35 mm and is equipped internally with an electrical heater. Its power can be adjusted with the power supply. The rod is instrumented with 48 thermocouples (TCs) positioned under its surface. Additional experiments with a ribbed rod surface were carried out in order to investigate the heat transfer enhancement. In this paper results from the smooth rod surface analyses are presented.

2.3. ANSYS CFX model of the L-STAR/SL facility

2.3.1. Mesh generation

Several high quality structured meshes were generated using ICEM CFD. In a next step mesh studies according to the OECD/NEA Best Practice Guidelines (OECD, 2007a) were performed. For the two different rod designs for both, the smooth and ribbed case, a coarse, medium and fine mesh was generated to obtain mesh independent solutions. The refinement was in radial and axial directions. With respect to the resolution of the boundary layer, the distance of the first nodes at the wall was reduced in each refinement step. The final hexahedral mesh selected for the RANS simulations with the smooth rod had 1.5 M elements (min. orthogonality angle 21°, max. aspect ratio 91, max. expansion factor 14). Later, for the scale resolving simulations (SRS) this final mesh was locally refined in order to lower the value of the dimensionless wall distance y+ down to 0.7.

Fig. 2 represents the ICEM CFD meshes of both rods as well as the inlet and outlet regions of the L-STAR/SL test section model.

2.3.2. ANSYS CFX setup

All steady state calculations were performed with the SST model (Menter, 1994). Comparative simulations with advanced turbulence models, requiring transient solution were carried out as well: a ZLES and the scale SAS turbulence approaches were selected. Both belong to the so called SRS (Scale Resolving Simulation) models, which resolve the larger eddies, while the small ones are modelled with sub-grid scale models (ANSYS, 2012). Since ANSYS CFX Version 14.0 a ZLES model can be used in order to solve a specific part of the CFD domain with complex flow pattern using LES, while the rest of the domain is treated with a RANS turbulence