

Postdoctoral position : CFD analysis of flow deviation induced by ballooned nuclear fuel rods

Workplace

Cadarache Center – Provence-Alpes-Côte d’Azur – France

Main scientific field

Mécanique des Fluides

Keywords

CFD, multiphase phase flow, heat transfer

Employer

IRSN (Institut de Radioprotection et de Sûreté Nucléaire) is the french public expert in nuclear and radiation risks. Placed under the joint authority of the French Ministries of Defense, Environment, Industry, Research, and Health, it is the technical support organization of the French regulator for nuclear safety. For more information, see the website : <http://www.irsn.fr/EN>.

Job & Mission

The candidate will work for the Incident and Accident Management Service (SEMIA). The activities conducted in this department covers especially the assessment of design basis accidents in nuclear power plants. This includes LOss of Coolant Accidents (LOCA) that correspond to transients induced by a pipe break in the primary loop of the plant.

1) Context & Objectives

During a hypothetical Loss of Coolant Accident, the fuel rods grouped into assemblies may balloon due to the high temperatures in the core and the differential pressure across the clad of the rods. Consequently, partial obstruction of the flow channel can jeopardize the cooling capability of the core during the following reflooding phase. During this phase, water is injected into the core leading to the progression of a quench front. Above this quench front, are located the ballooned hot rods exchanging heat with a steam-droplets flow. To study such configuration, IRSN has developed the DRACCAR software [1,2] allowing to simulate the thermal-mechanical behavior of a rod assembly during LOCA with a 3D multi-rod description. At the same time, simulations using the CFD code SATURNE-NEPTUNE have been carried out at a CFD scale to evaluate the wall heat transfer above the quench front [3,4,5]. Due to the lack of experimental data at such scale, tests were performed in the mock-up MASCARA using Reynolds analogy and MRI (magnetic resonance imaging) with the aims to get a fine mapping of the flow velocity [6].

The main objective of the proposed work is the development of adequate closure laws for the thermal-hydraulics of DRACCAR software through CFD simulations validated on MASCARA results.

2) Program of work

The study can be divided in the main following tasks :

1/ CFD Simulation of flow deviation with the SATURNE-NEPTUNE code

CFD calculation of the flow deviation induced by different geometrical and flow-rate configurations will be compared to experimental results in order to better understand the flow just downstream deformed rods, where the deterioration of heat transfer may be critical. The calculations will be performed under adiabatic conditions.

2/ Upscaling model of flow deviation

Cross-flows deduced from CFD in the region downstream deformed rods will be analyzed for prescribing a model for a sub-channel scale analysis in the DRACCAR code. The implementation of the model will be carried out with the help of the DRACCAR developing team located in the SEMIA department.

3/ CFD simulation of convective heat transfer

Thermal heat transfer between rods and fluid will be studied in order to identify the hot spots within the geometry and to relate their position to the previous hydrodynamics study. The fluid considered will be steam taking into account its thermal expansion that may induce an additional phenomenology.

4/ Upscaling model of single-phase heat transfer

Cross-flows and heat transfer intensity deduced from CFD in the region downstream deformed rods will be analyzed for prescribing models for a sub-channel scale analysis (see task n°2).

5/ Study of other geometries

The hereinabove studies are limited to a set of configurations of an array of 4x4 rods deformed within a 7x7 rods array. The ability of modeling the phenomena on a more generic fuel assembly geometry is required to study core geometry. Based on the previous analyzes, recommendation for the design of new experiments or for the performance of additional computations will be made. This work will be performed in interaction with both the experimental IRSN department SEREX [7] at Cadarache and the LEMTA lab at Lorraine University [8] where the MASCARA experiments have been performed.

6/ Two-phase flow heat transfer

In more realistic conditions, droplets flow within superheated steam act as an additional cooling source of the rods. Previous studies will be complemented by modeling their flow within the geometry based on a Euler-Euler two-phase model with heat and mass transfer thanks to the Neptune-CFD software.

3) Candidate profile

Applicant should possess a Ph.D. degree in CFD and have a good knowledge of turbulence modelling, heat transfer processes and multiphase flow simulation. Knowledge of the C (or FORTRAN) programming language as well as the Unix environment will also be an asset. For consideration, applicants should submit a resume, a publication list along with the name of referees and a cover letter outlining research interest to M. Herve MUTELLE at the following address : herve.mutelle@irsn.fr.

4) References

- [1] Glantz, T., Taurines, T., Belon, S., De Luze, O., Guillard, G., Jacq, F. "DRACCAR: A multi-physics code for computational analysis of multi-rod ballooning, coolability and fuel relocation during LOCA transients. Part Two: Overview of modeling capabilities for LOCA." Nucl Eng Des. 2018 Dec;339:202–14. <https://doi.org/10.1016/j.nucengdes.2018.08.031>
- [2] "DRACCAR computer software." <https://www.irsn.fr/EN/Research/Scientific-tools/Computer-codes/Pages/DRACCAR-software.aspx>
- [3] Ruyer, P., Seiler, N., Biton, B., Lelong, F., Secondi, F., Baalbaki, D., Gradeck, M. "Twophase flow across a partially damaged core during the reflood phase of a LOCA." Nucl Eng Des. 2013;264:187–94. <https://doi.org/10.1016/j.nucengdes.2013.02.026>

- [4] Guelfi, A., Bestion, D., Boucker, M., Boudier, P., Fillion, P., Grandotto, M., Hérard, J.- M., Hervieu, É., Péturaud, P. “NEPTUNE: A new software platform for advanced nuclear thermal hydraulics.” Nucl Sci Eng. 2007;156:281–324.
- [5] “Code_Saturne is the free, open-source software developed and released by EDF to solve computational fluid dynamics (CFD) applications.” <https://www.codesaturne.org/cms/>
- [6] Oliveira, A.V.D et al. “Velocity field and flow redistribution in a ballooned 7×7 fuel bundle measured by magnetic resonance velocimetry.” Nucl Eng Des. 2020. <https://doi.org/10.1016/j.nucengdes.2020.110828>
- [7] “IRSN SEREX and corresponding experimental research lab.” <https://www.irsn.fr/EN/Research/Research-organisation/Research-units/nuclearsafety-unit/LE2M/Pages/mechanical-material-experiment-laboratory.aspx>
- [8] “LEMTA, Laboratoire Énergies & Mécanique Théorique et Appliquée.” <http://lemta.univ-lorraine.fr/>