FLOW DEVIATION INDUCED BY BALLOONED FUEL RODS, HYDRODYNAMICS AND THERMAL STUDY

Duration: 18 months

Employer: IRSN

Specificity: reserved for foreign (non-french) applicants

Location: Cadarache research center, France

Profile: Post-doctoral position with a PhD in fluid mechanics/heat transfer

Keywords: CFD, convective heat transfer, upscaling

Context:

IRSN, Institut de Radioprotection et de Sûreté Nucléaire, is the technical support organization of the French regulator ASN for nuclear safety. The activities of the IRSN SEMIA department covers especially the assessment of safety studies of industrials and the corresponding research in the field of design basis accidents in nuclear power plants. This concerns the Loss of Coolant Accident scenario that corresponds to the consequences of a hypothetical pipe break in the primary loop of the plant.

During a hypothetical Loss of Coolant Accident within a nuclear power plant, the fuel rods within the core may balloon by creep and partially obstruct the coolant flow, potentially impeding their cooling. The flow deviation induced by the rods deformation has been experimentally studied in the mock-up MASCARA [1] (see the illustration at the end of the document) using Reynolds analogy and MRI (magnetic resonance imaging), allowing to get a fine mapping of the flow velocity.

Work:

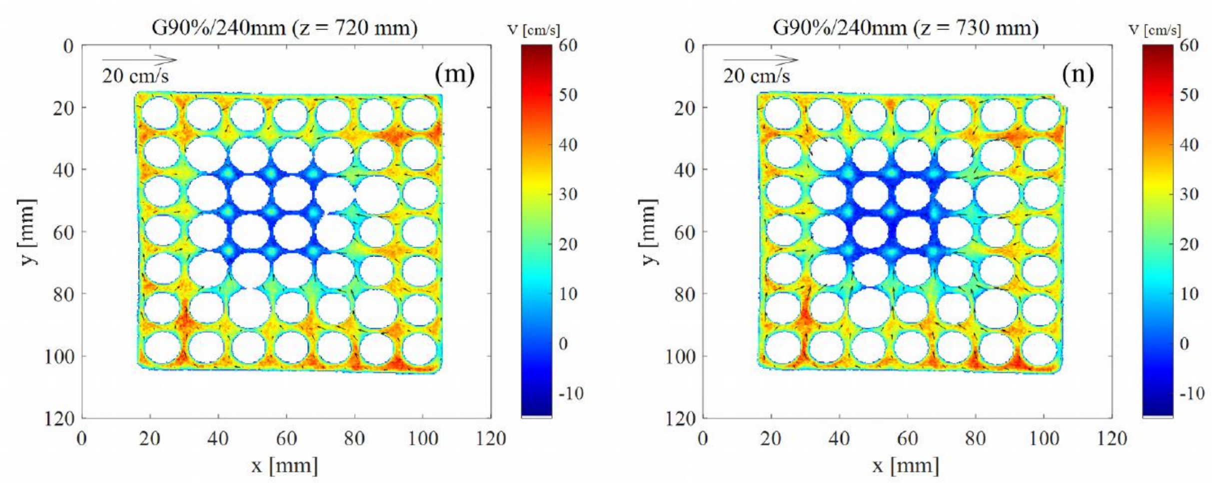
The main objective of the work is to study the consequences of the flow deviation on the cooling of the fuel rods at the assembly scale. Based on the validation of the Code-Saturne CFD computational tool [2] on the MASCARA experimental results, models will be defined to describe the impact of ballooning on a simplified flow model used in the DRACCAR software [3], [4] that describes the hydrodynamics at a so-called sub-channel scale[1].

The study can be divided in different tasks:

1/ CFD simulation of flow deviation

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. Typical experimental results of the flow field through deformed geometry are shown in the figure below.

[1] a single fluid node for the flow cross section between 4 neighbors fuel rods



2/ Upscaling model of flow deviation

Cross-flows deduced from CFD in the region downstream deformed rods will be analyzed for prescribing models for a sub-channel scale analysis.

3/ CFD simulation of convective heat transfer

Thermal heat transfer between rods and flow will be studied thanks to CFD 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.

5/ Toward other geometries

The hereinabove studies are limited to a set of configuration 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 [5] at Cadarache and the LEMTA lab at Lorraine University [6] where the MASCARA experiments have been performed.

6/ Two-phase flow heat transfer [optional]

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 an Euler-Euler two-phase model with heat and mass transfer thanks to the Neptune-CFD software [7], [8].

References

[1] A. V. S. Oliveira *et al.*, “Velocity field and flow redistribution in a ballooned 7×7 fuel bundle measured by magnetic resonance velocimetry,” *Nucl. Eng. Des.*, vol. 369, p. 110828, Dec. 2020, doi: 10.1016/j.nucengdes.2020.110828.
[2] “Code_Saturne is the free, open-source software developed and released by EDF to solve computational fluid dynamics (CFD) applications.” [Online]. Available: https://www.code-saturne.org/cms/
[3] T. Glantz, T. Taurines, S. Belon, O. De Luze, G. Guillard, and F. Jacq, “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.*, vol. 339, pp. 202–214, Dec. 2018, doi: 10.1016/j.nucengdes.2018.08.031.
[4] “DRACCAR computer software.” [Online]. Available: https://www.irsn.fr/EN/Research/Scientific-tools/Computer-codes/Pages/DRACCAR-software.aspx
[5] “IRSN SEREX and corresponding experimental research lab.” [Online]. Available: https://www.irsn.fr/EN/Research/Research-organisation/Research-units/nuclear-safety-unit/LE2M/Pages/mechanical-material-experiment-laboratory.aspx
[6] “LEMTA, Laboratoire Énergies & Mécanique Théorique et Appliquée.” [Online]. Available: http://lemta.univ-lorraine.fr/
[7] A. Guelfi *et al.*, “NEPTUNE: A new software platform for advanced nuclear thermal hydraulics,” *Nucl Sci Eng*, vol. 156, pp. 281–324, 2007.
[8] P. Ruyer *et al.*, “Two-phase flow across a partially damaged core during the reflood phase of a LOCA,” *Nucl. Eng. Des.*, vol. 264, pp. 187–194, 2013, doi: 10.1016/j.nucengdes.2013.02.026.

Contact: H. Mutelle herve.mutelle@irsn.fr

Please send CV and motivation letter.

The MASCARA experiment:

Diagram of the device and the typical geometry for a ballooned 7x7 array of rods