NUMERICAL INVESTIGATIONS OF A SPENT FUEL STORAGE POOL IN ABNORMAL CONDITIONS

J-p Simoneau[1], B. Gaudron[1], Jérôme Laviéville[2], Julien Dumazet[1]
Electricité de France (EDF)
[1]12-14, av. Dutriévoz, 69628 Villeurbanne cedex, France
[2] 6, quai Watier - 78401 Chatou cedex, France
jan-patrice.simoneau@edf.fr; bruno.gaudron@edf.fr; jerome-marcel.lavieville@edf.fr

ABSTRACT

This paper deals with the thermal behavior of spent fuel storage pools, in the situations of loss and restart of the forced cooling system. The objective is to set up methodologies for the modeling of the two phase flow behavior. A first approach is performed with a single phase code (*Code_Saturne*). A simplified boiling model is then implemented considering flashing below the free surface. An agreement with in situ measurements is obtained for the natural convection flow occurring in normal conditions and in the first hours after the loss of cooling. In a second step a more detailed multiphase modeling is set up, thanks to the Neptune_CFD code. The boiling inside the pool and the heat and mass transfers at the free surface are simultaneously calculated. The early times computed are analyzed and compared positively with the single phase simplified model. Finally, further steps are described.

KEYWORDS
Spent fuel pool, CFD, Boiling, Two-phase flow

1. INTRODUCTION

The thermal behavior of spent fuel storage pools is of first importance to ensure the safety of nuclear installations. In this respect, one important safety criterion for spent fuel assemblies' integrity is to maintain a liquid water level over the assemblies. Some previous analyses can be found in [1] and [2]. In nominal conditions, a forced water cooling circuit ensures low temperatures inside the pool and hence an efficient fuel heat removal at constant water level. In abnormal conditions (loss of cooling circuit), boiling may occur locally or globally in the pool to remove spent fuel thermal power. Safety water injection maintains liquid water at normal operating level but the restart of the cooling pumps can be subject to cavitation or steam suction when the saturation margin is low.

The objective of this paper is to better understand the thermal behavior of the pool in that accidental situation thanks to CFD analyses applied on a generic case. The scenario will first be described in section 2, highlighting some specific characteristics of this kind of flow. The first approach, based on a single phase calculation with the CFD software *Code_Saturne* [3], is described in section 3, where boiling is modeled in a simple way as regards to the saturation temperature. In section 4, the multiphase software Neptune_CFD [4] allows a more precise description of the boiling phenomenon. However, its combination with the water / air free surface representation leads to some difficulties and the calculations presented are a first step.

A general analysis and perspectives are then provided in section 5.

2013, pp. 496-504.
2. Boyd, C. F. “Predictions of Spent Fuel Heat up After a Complete Loss of Spent Fuel Pool Coolant”, NUREG-1726, U.S. Nuclear Regulatory Commission, June 2000.
3. Archambeau, F., Mechitoua, N., Sakiz, M., “Code_Saturne: a Finite Volume Code for the Computation of Turbulent Incompressible Flows - Industrial Applications,” *International Journal on Finite Volumes*, vol. **1**, 2004.
4. Douce, A., Mimouni, S., Guingo, M., Morel, C., Laviéville, J., Baudry, C., “Validation of Neptune_CFD 1.0.8 for adiabatic bubbly flow and boiling flow”, *proceedings of the CFD4NRS-3 conference*, Washington DC, USA, 2010.
5. Mimouni, S., Baudry, C., Denèfle, R., Fleau, S., Guingo, M., Hassanaly, M., Laviéville, J., Méchitoua, N., Mérigoux, N., “Multifield Approach and Interface Locating Method for Two-Phase Flows in Nuclear Power Plant”, *proceedings of the CFD4NRS-5 conference*, September 9-11 2014, Zurich, Switzerland.
6. S.V. Patankar, “Numerical heat transfer and fluid flow”, Series in computational methods in mechanics and thermal sciences, McGraw-Hill book company, New-York, 1980, 197 p.
7. M. Ishii and N. Zuber, “Drag coefficient and relative velocity in bubbly, droplet or particulate flows,” *AIChE Journal*, vol. 25.
8. Tullis, J. P., “Modeling Cavitation for closed conduit flow”, *J. Hydraul. Div. ASCE*, **107**(HY11), Proc. Paper 16650, November 1981, pp. 1335-1349.
9. Ishii, M., “Thermo-fluid dynamic, theory of two-phase”, Collection de la Direction des Etudes et Recherches d’Electricité de France, n°**22**, Eyrolles, Paris, 1975, 275 p.
10. Coste, P., “A large interface model for two-phase CFD”, *Nuclear Engineering and Design*, vol. **255**, Feb. 2013, pp. 38-50. (http://dx.doi.org/10.1016/ j.nucengdes.2012.10.008 (2012)).
11. W. Wagner et al., "The IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam," ASME J. Eng. Gas Turbines and Power, 122, 150-182 (2000).
12. W.H. McAdams, “Heat transmission”, 3rd edition, McGraw-Hill publishing company ltd, London, 1954, 532p.
13. Laviéville, J., Mérigoux, N., Guingo, M., Baudry, C., Mimouni, S. “A Generalized Turbulent Dispersion model for bubbly flow numerical simulation in NEPTUNE_CFD”, *Proceedings of NURETH-16*, Chicago, IL, USA, Sept. 2015.
14. Serre G., Bestion D.: Physical Laws of CATHARE Revision 6. Pipe Module. CEA report SMTH/LMDS/EM/98-038, September 1999.