DOI: 10.15669/pnst.8.196

Progress in Nuclear Science and Technology
Volume 8 (2025) pp. 196-201

ARTICLE

Thermal Behavior of a MOX 1300 MWe Assembly While Lifted in Air before being Immersed into a Spent Fuel Pool

Christophe PÉNIGUEL [1*], Jean-Marc BARTHOULOT [2] and Davide COSTA [2]

[1] *EDF R&D – 6 quai Watier, 78401 Chatou Cedex, France*
[2] *EDF DCN – 1 Place Pleyel, Saint Denis 93200 Cedex, France*

The deployment of MOX fuel assemblies in some of the 1300 MWe plants of the EDF nuclear fleet is under preparation as part of the French strategy concerning the closure of the nuclear fuel cycle. The present paper addresses the thermal aspects related to the handling of MOX fuel assembly when extracted from a MX6® transport cask and put into the spent fuel pool prior to its loading in the reactor core. The assembly, suspended vertically in an air atmosphere, is transferred from the preparation pit up to the spent fuel EDF pool where is it finally immerged. A conjugate heat transfer approach based upon the coupling of a finite volume CFD code (*Code_Saturne*) and a finite element thermal code (SYRTHES) takes into account all the heat exchange mechanisms (convection, conduction and radiation) to obtain the assembly temperature distribution during the transfer through the air to assess the mass of vapor generated during the immersion phase simulated using a simplified approach and validated against three experiments.

KEYWORDS: MOX 1300 MWe assembly, fuel handling, thermal management, numerical simulation, water vapor, conjugate heat transfer, Code_Saturne, SYRTHES

I. Introduction

EDF is considering the possibility of using an MX6® (property of Orano NPS) transport cask for the transport of unirradiated 1300 MWe MOX fuel assemblies. Moreover, instead of unloading the MX6® cask under water, the feasibility of a dry process is being studied. The entire operation takes place inside the pool building shown in **Fig. 1**.

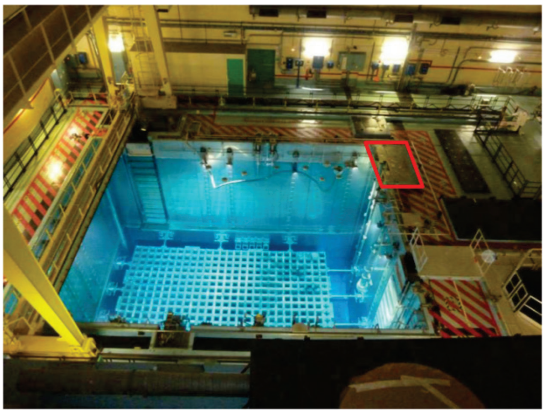


Fig. 1 Spent fuel pool hall where assembly is immersed (at the location highlighted in red)

Once extracted from the MX6®, the hot assembly is lifted vertically and follows the path shown in **Fig. 2**, so that most of the transfer occurs above the spent fuel pool.

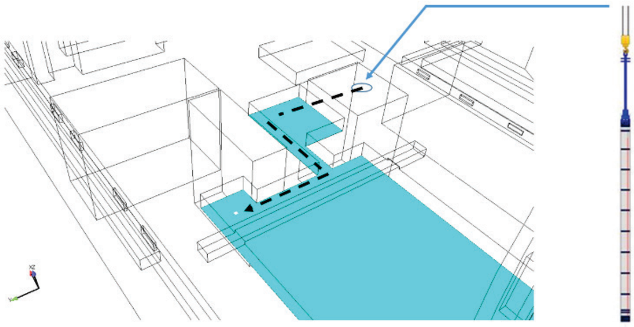


Fig. 2 Path followed by the suspended assembly

The assembly is suspended by its top nozzle in air without any shielding device during approximatively 10 minutes. The aim of the present thermal study is to get a better understanding of the assembly thermal behavior during the handling. The thermal study must also consider the accident scenario where the assembly would be blocked at a given position for a longer period. Then the assembly is immersed into the spent fuel pool at a constant speed specified by the operator as shown on **Fig. 3**. A sliding solid mesh and a conjugate heat transfer approaches are used to determine the mass of vapor emitted both by boiling evaporation from the free surface of the pool.

*Corresponding author, E-mail: christophe.peniguel@edf.fr

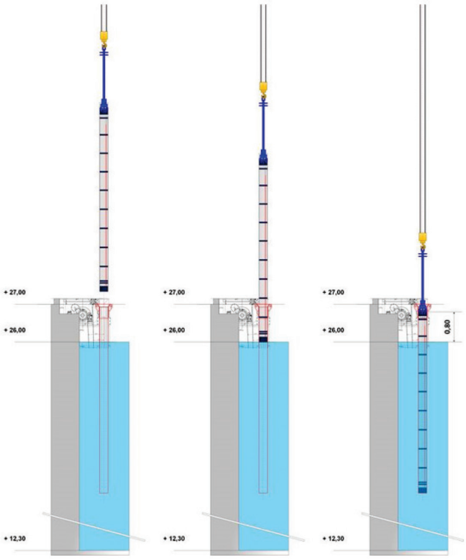


Fig. 3 Immersion phase (speed imposed by operator)

II. Methodology

The temperature of a MOX assembly is determined by the heat generated by the alpha-decay (roughly 1 kW for a MOX assembly) and the thermal properties of the environment that control its dissipation through the fuel assembly material into the surrounding environment. Once extracted from the MX6® cask, the MOX assembly temperature distribution will evolve with the changing environment. As the fuel assembly temperature is a key parameter when assessing the mass of the vapor generated during the immersion in the pool, a better knowledge of the temperature distribution evolution when it is taken out from the cask and transferred through the air of the pool building is paramount. The fuel assemblies are hooked by their head, in a vertical position, without any shielding or any active cooling device. Therefore, the evolution of the temperature distribution in the assembly volume will result from a competition between the heat generated by the alpha-decay and the heat loss by radiation towards the environment and a convective flow induced by the hot rods and the guide tubes in contact with the cooler environment. The heat source is located inside the fuel pellets. Then, the heat is transferred by conduction trough the MOX fuel and its cladding, until it reaches the external surface of the fuel pins where it is removed by convection and radiation.

A conjugate heat transfer approach is used, based on open-source numerical tools developed at EDF R&D: SYRTHES[1] (solid thermal analysis) and *Code_Saturne*[2] (CFD). The fluid domain and the solid assembly meshes are generated by the mesh generator SALOME[3] and presented in **Figure 4**. The grids are not explicitly modelled in the solid mesh, their effects on the flow are accounted for thanks to head losses in the CFD calculation. They seem to have a negligible effect on the air flow. At each time step during the transient, at the interface between solid and fluid, heat is transferred from one domain to the other.

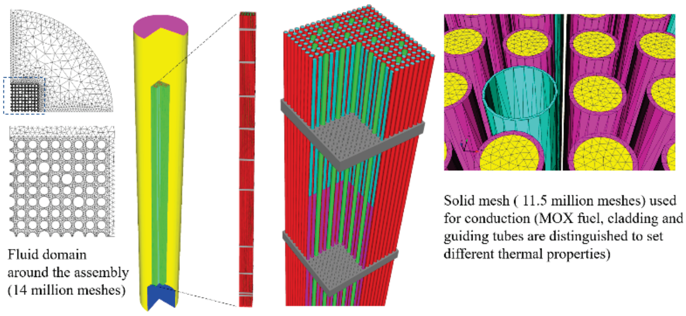


Fig. 4 Fluid (left) and Solid (right) meshes

Air can be considered as a non-participating medium from the radiation point of view, so radiation takes place only from wall to wall. Walls behave in a diffuse and isotropic manner and are considered as grey bodies. The approach used is based on radiositymm.[4] Rods and guide tubes walls radiate against each other with strong shadowing effects. In the present case, modelling the radiation phenomenon is challenging due to the large number of rods and tubes leading to a great number of radiative patches.

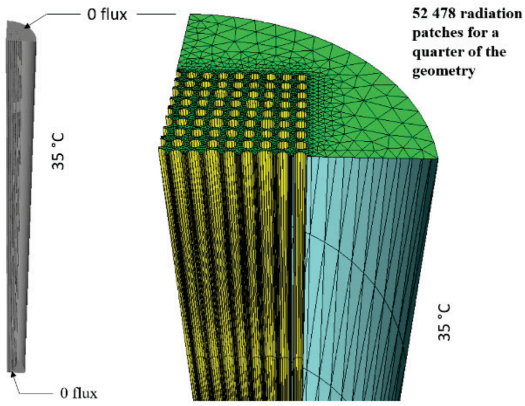
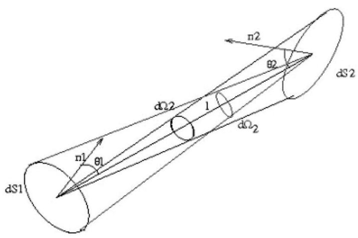


Fig. 5 Radiation mesh and radiative boundary conditions

Within the thermal code SYRTHES, the view factors are purely geometrical entities which need to be calculated only once for a given geometry. To reduce the CPU cost, radiation relies on a specific radiation mesh presented on **Figure 5** counting 52 478 independent patches for a quarter of the geometry. each one corresponding to the expression given on **Fig. 6**.



$$F_{12} = \frac{1}{S_1} \int_{S_1} \int_{S_2} \frac{\cos\theta_1 \ \cos\theta_2 dS_1 \ dS_2}{\pi l^2}$$

Fig. 6 View factor between two independent patches

Such many independent patches lead to many view factors (here $(4{*}52478)^2 = 44$ billion).

III. Numerical Tools

The solid thermal code SYRTHES,[1] developed by EDF, relies on a finite element technique to solve the general heat equation where all properties can be time, space or temperature dependent. Air in hall is a non-participating media, so SYRTHES radiation from wall-to-wall model is well adapted. Surfaces properties are considered grey (only emissivity must be defined) and radiation solver is based on a radiosity approach (see Ref.4)).

Code_Saturne,[2] developed by EDF is used for thermal-hydraulic. The CFD code solves Navier-Stokes equations for steady or transient, single phase, incompressible, laminar or turbulent flows. A k-ε turbulent model has been used.

Conjugate heat transfer is used at the fluid/solid interface. Let T_s be the temperature of an internal solid node, T_w the temperature at a node which belongs to the interface, and T_f the temperature of a fluid point (located generally in the log layer). At time $t^{(n)}$, the CFD tool *Code_Saturne* provides after calculation $h^{(n)}$ (the local heat exchange coefficient at time $t^{(n)}$) and $T_f^{(n)}$ (the local inside fluid temperature at time $t^{(n)}$). Then, with the flux $\varphi_s^{(n+1)} = h^{(n)}\left(T_w^{(n+1)} - T_f^{(n)}\right)$, SYRTHES can solve the heat conduction equation inside the solid. At the same time, a similar procedure is used in the fluid to update the fluid temperature field.

IV. Handling in Air

The initial temperature field is based on thermal studies performed on the MX6® cask showing that the rods located near the centre of the assembly are hotter than those at the periphery. Radiation cools down the assembly, particularly the outer rows, as they are not completely shadowed by surrounding pins. The inner part of the assembly cools down more slowly due to shadowing effect of the outer rows. In parallel, an air flow develops induced by the density effect. Cool air is attracted both from the bottom and from the side of the assembly and forms a plume as shown on **Fig. 7**.

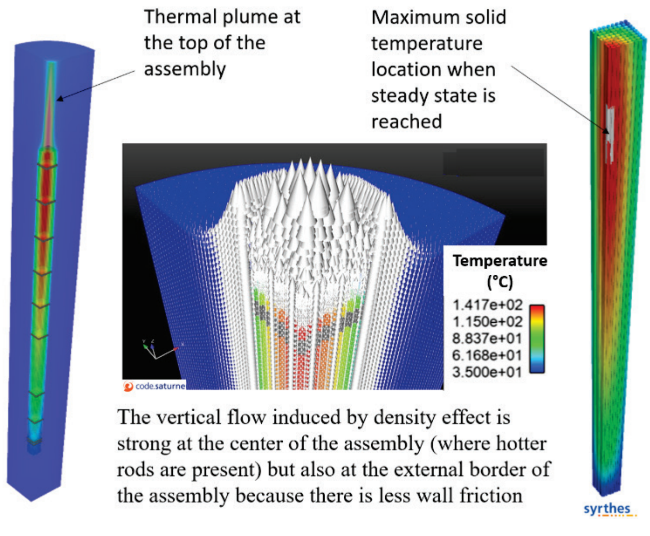


Fig. 7 Fluid (left) and solid (right) temperature fields

After some time, this flow induces two main effects as shown on **Fig. 8** and **Fig. 9**: the first one is to cool down globally the assembly and the second one is to induce a vertical temperature gradient.

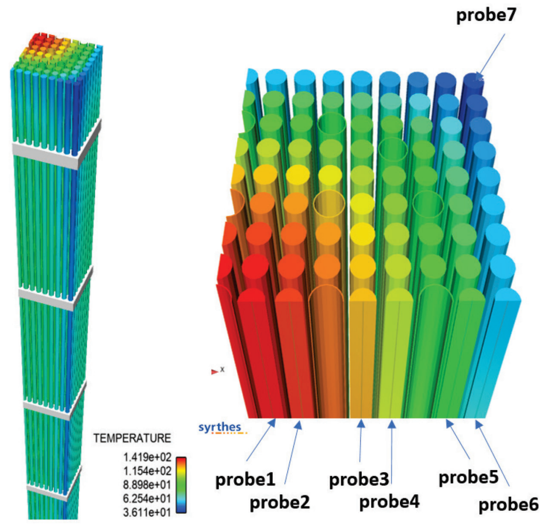


Fig. 8 Fluid (left) and solid (right) temperature fields

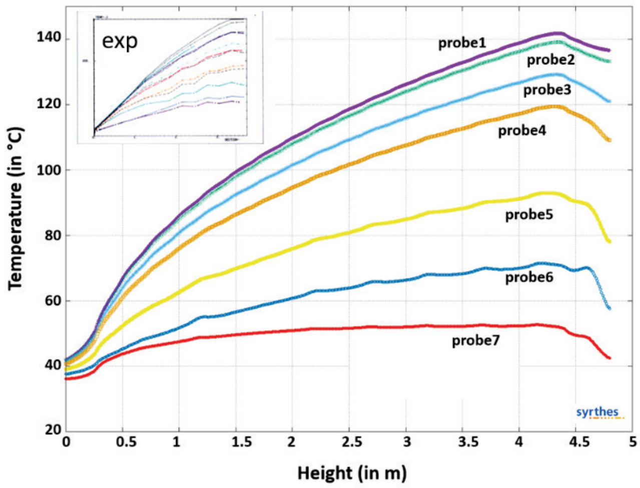


Fig. 9 Vertical solid temperature profiles (for the 7 probes location given on Fig. 8)

The thermal power generated by the alpha-decay (1 kW) doesn't induce any temperature increase even if no active cooling device has been introduced. From the temperature point of view, the most penalizing situation is therefore the one existing just at the time of the extraction of the fuel assembly from the MX6® cask. That temperature distribution is therefore retained to perform the immersion calculation.

V. Immersion`Phase

Once the assembly temperature distribution obtained, the immersion phase can be addressed. The assembly rods and guide tubes being hotter than 100°C, boiling and evaporation at the surface occur.

The simulation is complicated by the vertical motion of solid assembly which must be considered at each time step. As seen on Fig. 8, the temperature distribution is complex.

Heat from the assembly is transferred to the water of the pool which cools down the part of the rod progressing downwards. To prevent very costly conjugate heat transfer calculations using a coupling between SYRTHES and a two-phase flow CFD code, a more pragmatic approach has been used. It uses several approximations:

- a sliding solid mesh shown on **Fig. 10** is used, at the beginning of each time step, we translate vertically the thermal field,
- water is not pushed away by the incoming solid,
- when the water temperature (heated by the contact with the hot rods) goes above 100°C, it is immediately transformed into vapor (with no possibility to condensate again before arriving to the surface of the pool),
- evaporation at the surface is using a correlation.

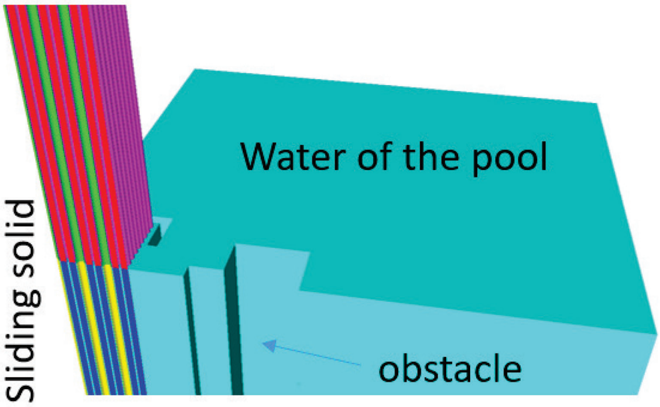


Fig. 10 Geometry used for the immersion phase

Using the procedure illustrated by the sketch on **Fig. 11**, the solid mesh stays still, just the temperature field is moved downwards at the speed imposed by the operator. For that phase, the same conjugate heat transfer approach can be used.

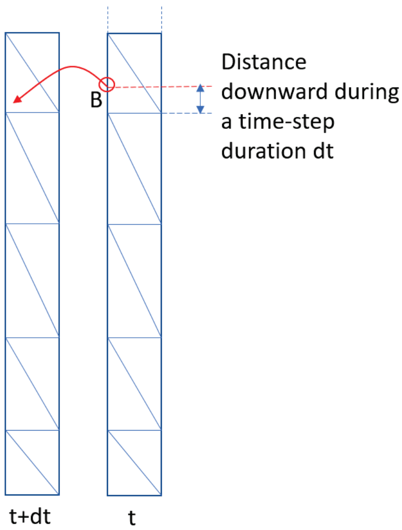


Fig. 11 Pragmatic procedure used to simulate the translation of the solid assembly

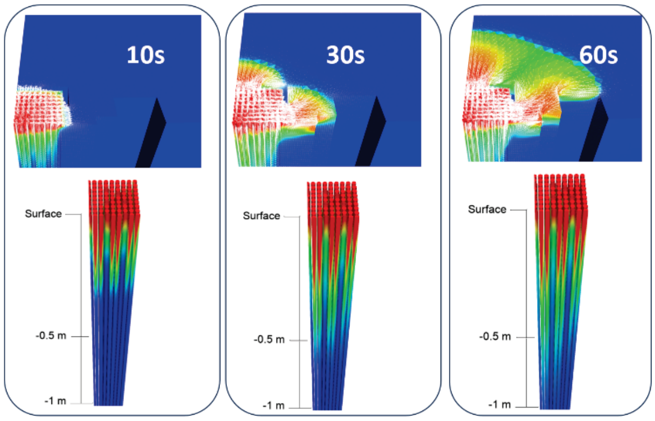


Fig. 12 Fluid temperature (top) and solid rod temperature at different time during the transient

As shown on **Fig. 12**, calculation suggests that the water temperature increase is mainly located within a layer near the surface. Regarding the immerged portion of the rods and tubes, the temperature decreases quickly. Since the assembly moves down at a speed specified by the operators, hot parts of the assembly keep arriving until the entire assembly is immerged thus leading the water temperature to reach 100 °C. The additional energy brought by the incoming hot rods induces a phase change from liquid to vapor. Likewise, due to density effect, a thermal stratification arises at the surface of the pool and induces an evaporation phenomenon modelled thanks to a correlation.[5)]

The approach uses some approximations to be able to handle the problem in a reasonable time (several hours of CPU times instead of months or years when using a two-phase flow approach). So, the strategy followed has been to apply this "approximated approach" to existing mock-up for which experimental results were at our disposal. Three of them have been used to validate our approximated approach.

- A very small mock-up studied at EDF R&D using a 5x5 bundle made of steel rods, only 50 cm heigh has been heated in an oven at 200°C and immersed in a cylindrical water volume with a possibility to precisely measure the weight and consequently the mass of vapor emitted (**Fig. 13)**.
- An old immersion mock (MOCKA) performed at CEA in the early eighties which was giving an estimation of the water lost (and therefore the vapor emitted). This experimental setup had been used to investigate the effect of the velocity on the mass of vapor emitted (**Fig. 14)**.
- Visual experiment carried out in a French nuclear power plant (at TRICASTIN). During this real case performed on a 900MW assembly, operators had noted phenomena occurring during the immersion inside a pool.

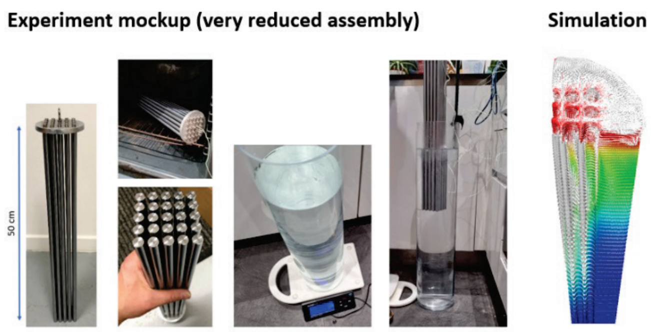


Fig. 13 Very small experimental mock-up PVAP

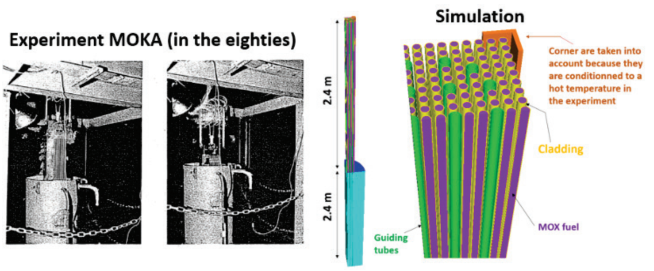


Fig. 14 Mockup MOKA studied by CEA

When performing calculations using the procedure described earlier on these three configurations, good agreement is found regarding both the physical phenomena observed and an estimation of the mass of vapor produced. **Table 1** gives a synthetic comparison of the mass of vapor measured and the one calculated with the procedure relying on the conjugate heat transfer between SYRTHES and the one phase flow CFD code *Code_Saturne*. The uncertainty associated with the PVAP experiment is of the order of ±1g it comes from the scale precision, and the immersion time (of the order of a second). Regarding, the MOKA experiment, several experiments have been performed which explains the range (1.57kg to 1.67kg) indicated in Table 1. Regarding simulations, changing the time step or grids used doesn't change significantly the mass found. One underlines that the main aim of these comparisons is make sure that the right order of magnitude is found and that the same trends are observed. We do not expect to find exactly the same values for the mass of vapor produced.

Table 1 Experimental and calculations comparisons regarding the vapor produced during the immersion of a hot assembly

Experiment	Experiment	Simulation
PVAP	About 10g ± 1g	De 7g to 13g according immersion velocity
MOKA 5m/min	1.57 kg to 1.67 kg after 15 min 2.25 kg to 2.36 kg next morning	1.54 kg after 15 min
MOKA 0.4m/min	3.14 kg after 15min 3.74 kg the next morning	3.42 kg after 15min
TRICASTIN	Vapor visible after 5ème grid according operators	Emission peak between grid 5 and grid 6

When applied on the case of MOX 1300 MWe assembly calculated above, for an immersion taking place at a speed of 1.5 m/min, half a kilogram of vapor emitted is found, which turns out to be less than the values retained so far.

Numerically, it is possible to separate the contribution originating from the boiling phenomenon and the part coming from the evaporation taking place at the free surface of the pool. **Figure 15** shows the evolution at which vapor is produced. It turns out that vapor produced by boiling takes place only during the immersion and the vapor emission stops immediately when the top of the assembly goes below the surface. The speed at which the immersion is performed has a significant influence on the mass of vapor produced as it had been noted both on MOKA experiment and more recently on PVAP.

On the other hand, evaporation does not start immediately but plays a part a much longer time.

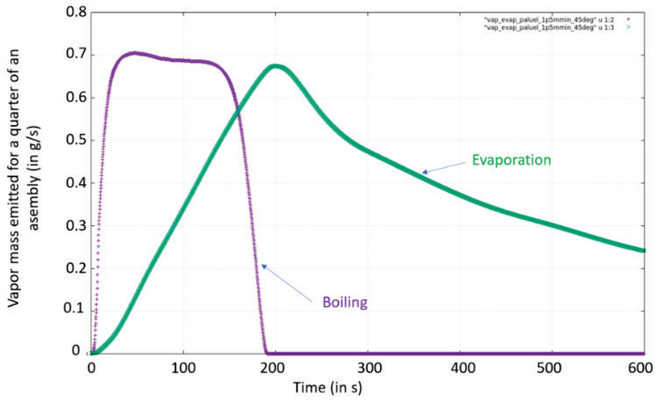


Fig. 15 Emission of vapor: boiling and evaporation during the immersion process

Temperature of the pool has some influence on the quantity of additional vapor emitted but its effect seems to be limited as suggested by **Table 2**.

Table 2: Influence of the pool temperature on the mass of vapor emitted (Paluel pool case for a new MOX 1300 Mwe assembly).

Pool temperature	Vapor mass emitted by boiling (kg)	Vapor mass emitted by evaporation (kg)	Vapor total mass (kg)
20°C	0.316	0.650	0.967
35°C	0.397	0.892	1.29
45°C	0.440	0.908	1.348

On the other hand, calculations suggest that the velocity at which the assembly is immerged has a much bigger impact. The faster the downward velocity, the smaller the mass of vapor emitted, as also confirmed by experimental measurements.

VI. Conclusions

The present thermal study provides an accurate description of the assembly temperature field at any time during the fuel handling in an air atmosphere. This field is then used to estimate the mass of water vapor likely to be produced through boiling and evaporation phenomena.

To handle the complexity of the phenomena taking place, an approach using some approximations has been used. Much effort has been spent to make sure that the right order of magnitude of the mass of vapor emitted can be found with that simplified approach. Comparisons with three experimental mockups makes us quite confident that simulations are able to capture both the phenomena taking place but also provide a reasonnable estimation of the mass of vapor emitted during the manutention in air of a new MOX 1300 Mwe assembly.

This study will contribute to better dimension a special device designed to capture most of the vapor emitted.

References

1) C. Péniguel, I. Rupp. “SYRTHES : open-source thermal code.” http://www.edf.fr/recherche/code-syrthes
2) *Code_Saturne* open-source CFD code http://www.code-saturne.org
3) SALOME open-source CAD and meshing platform, http://www.salome-platform.org
4) R. Siegel, J.R. Howell, “Thermal Radiation Heat Transfer”, 2nd ed, Hemisphere, Publishing Corporation, New York, NY, (1981).
5) M. M. Shah, “Methods for calculation of Evaporation from swimming Pools and other Water Surfaces”, ASHRAE, transaction 2014.