ENEA Ricerca Sistema Elettrico	Sigla di identificazione NNFISS – LP3 - 050	Distrib. L	Pag. 1	di 12

Titolo

Freezing of the LFR primary pool: CFD modeling and preliminary analysis on prototypical configuration

Descrittori

Tipologia del documento: **Rapporto Tecnico**

Collocazione contrattuale: Accordo di programma ENEA-MSE: tema di ricerca "Nuovo nucleare da fissione"

Argomenti trattati: Termoidraulica dei reattori nucleari

Generation IV Reactors

Tecnologia del Piombo

Sommario

In the context of GEN-IV Heavy Liquid Metal safety studies, the lead freezing in the pool is considered one of the main issues to be addressed and a realistic accident for LFR. The present document is a first step towards a detailed analysis of such phenomena, and a CFD model and approach is presented to have a detailed thermo-fluid dynamic picture in the case of freezing. In particular, a numerical methodology is presented to model the freezing of lead and the heat transfer in a proper way. At this stage, examples are given on a very simplified and prototypical geometry representing the primary pool of the LFR DEMO.

Note

Autori: I. Di Piazza

Copia n. **In carico a:**

2			NOME			
			FIRMA			
1			NOME			
			FIRMA			
0	EMISSIONE	17/09/2012	NOME	I. Di Piazza	M. Tarantino	M. Tarantino
			FIRMA			
REV.	DESCRIZIONE	DATA		REDAZIONE	CONVALIDA	APPROVAZIONE

Index

1. Introduction

In the context of GEN-IV Heavy Liquid Metal safety studies, the lead freezing in the pool is considered one of the main issues to be addressed and a realistic accident for a LFR DEMO.

The present document is a first step towards a detailed analysis of such phenomena, and a CFD model and approach is presented to have a detailed thermo-fluid dynamic picture in the case of freezing. In particular, a numerical methodology is presented to model the freezing of lead and the heat transfer in a proper way.

At this stage, examples are given on a very simplified and prototypical geometry representing the primary pool of the LFR DEMO.

With respect to LWR or to sodium-cooled fast reactors, in Lead cooled fast reactors (LFR) [1] the boiling of the coolant is very unlikely due to the high boiling temperature of Lead (1740 °C). This represents a big advantage of the Lead technology with respect to the other ones. On the other hand, lead freezing temperature is T_{freez}~330 °C and lead is frozen at room temperature. Therefore the lead freezing has to be considered as a possibility, and the safety issues related to scenarios with frozen lead must be careful investigated.

The consequences of a LEad FReezing Accident (LEFRA) in some part of the lead pool are probably the full damaging (mechanical braking) of the components involved and a serious damaging of the reactor with a big economic and safety impact and this scenario must be considered as extreme. The first components to focus are of course the cold sinks of the pool, i.e. the Steam Generator (SG) and the Decay Heat Removal heat exchangers (DHR). In the case of accident, for example, if the DHR extract from the system more power than the decay power, the average temperature of the pool goes down and in this case the HX is the first component in which the lead temperature could fall below the freezing temperature.

In this framework, the impact of lead freezing on the system components, on the heat removal and on the global safety of the plant must be addressed very carefully. A program of experiments and numerical studies should be planned in the next future to have a detailed picture of the freezing in LFR pool reactors. This program has not yet planned and both in the literature and in the reports of the European research centers the freezing topic has not been investigated very much.

From the numerical side, the first step towards this investigating program is the definition of the numerical methods and tools to correctly capture the physical phenomena involved. For the Computational Fluid Dynamics (CFD) it is not really convenient to describe the problem as a real two-phase problem with phase changing. In fact, in this case the problem formulation and the numerical discretization would be highly complex without probably adding relevant physical information. The simplified approach proposed here allows capturing the 95% of the physics with a much easier formulation and maintaining the one-fluid, one-phase approach. Considering that other sources of errors are present in the CFD modeling (i.e. turbulence models, numerical discretization, geometry simplifications, etc.) the present proposal is considered by the author by far sufficient to capture the relevant physics.

2. Numerical Models and methods

2.1 General Framework

The general purpose code ANSYS CFX 13 [2] was used for all the numerical simulations presented in this paper. The code employs a coupled technique, which simultaneously solves all the transport equations in the whole domain through a false time-step algorithm. The linearized system of equations is preconditioned in order to reduce all the eigenvalues to the same order of magnitude. The multi-grid approach reduces the low frequency error, converting it to a high frequency error at the finest grid level; this results in a great acceleration of convergence. Although, with this method, a single iteration is slower than a single iteration in the classical decoupled (segregated) SIMPLE approach, the number of iterations necessary for a full convergence to a steady state is generally of the order of 10^2, against typical values of 10^3 for decoupled algorithms.

The SST (Shear Stress Transport) k-ω model by Menter [3] is extensively used for the simulations presented here. It is formulated to solve the viscous sub-layer explicitly, and requires several computational grid points inside this latter. The model applies the k-ω model close to the wall, and the k-ε model (in a k-ω formulation) in the core region, with a blending function in between. It was originally designed to provide accurate predictions of flow separation under adverse pressure gradients, but has since been applied to a large variety of turbulent flows and is now the default and most commonly used model in CFX-13 and other CFD codes. The turbulent Prandtl number in the case of lead has been fixed to 1.1, according to the suggestion of the literature [4] and to the author's experience [5].

2.2 Formulation of the Method

To simulate the phase change from the liquid to the solid phase several modifications have been introduced in the code. The modifications have been implemented by the CFX Expression Language (CEL), but the user subroutines could be used as well with similar results.

As already stated in section 1, the single-fluid single-phase approach has been chosen to simulate the freezing phenomena. The basic idea and the main approximation is to consider the freezing process occurring in a small temperature range ΔT_{melt}~1-2 °C (*freezing Temperature window*) instead that at a constant temperature T=T_{freez}=327 °C, i.e. the process is simulated between T_{freez} and T_{freez}-ΔT_{melt}. This issue is addressed by setting the specific heat c_p in the freezing window much higher that c_p in the liquid and in the solid phase. A practical numerical optimizing procedure gave the following values for the specific heat:

$$c_p = c_{p0} = 146.7\ [J/kgK] \qquad for \quad T > T_{freez} \tag{1}$$

$$c_p = c_{p0} = 146.7\ [J/kgK] \qquad for \quad T < T_{freez} - \Delta T_{melt} \tag{2}$$

$$c_p = 100 \cdot c_{p0} = 14600.7\ [J/kgK] \qquad for \quad T_{freez} - \Delta T_{melt} < T < T_{freez} \tag{3}$$

With the above choices the freezing temperature window ΔT_{melt} can be computed in a physically consistent way on the base of the freezing latent heat for lead ΔH_{melt}:

$$\Delta H_{melt} \approx 23800\ [J/kg] \tag{4}$$

$$\Delta T_{melt} = \frac{\Delta H_{melt}}{100 \cdot c_{p0}} \approx 1.63\,[°C] \tag{5}$$

ΔT_{melt} is in the correct range stated above and would lead to an acceptable approximation of the real physical behaviour. These modifications will introduce an 'energetic' behaviour similar to the real fluid because when the fluid temperature reaches the freezing temperature T_{freez}, the unit mass of fluid will be in the freezing temperature windows until it will provide to the neighbours exactly the energy ΔH_{melt}.

The second step towards a full formulation of the problem is to implement the *mechanical* behaviour of the fluid during the phase change and in the solid phase. The idea is to introduce a force which simulates the growing resistance of the fluid during solidification and which stops completely the fluid motion in the solid phase. This is done by the use of proper source terms in the momentum equation. In each direction (x,y,z) , a drag negative source term is chosen proportional to the velocity in that direction (u,v,w), according to the expressions:

$$S_u = -C(T) \cdot u \tag{6}$$

$$S_v = -C(T) \cdot v \tag{7}$$

$$S_v = -C(T) \cdot w \tag{8}$$

The source term must be expressed in N/m^3, i.e. in kg/m^2s^2, and therefore the function $C(T)$ is expressed in kg/m^3. The function $C(T)$ is chosen in a proper way to be 0 in the liquid range and very high in the solid range, with a linear behaviour in the freezing window.

Introducing the solid fraction γ, which represents the frozen fraction of the unit mass in the freezing window:

$$\gamma = \frac{T_{melt} - T}{\Delta T_{freez}} \tag{9}$$

The specific value set to ensure numerical convergence to the code and a suitable mechanical behaviour of the fluid in the melting window are expressed by the following formulas:

$$C(T) = 0\,[kg/m^3] \quad for \quad T > T_{freez} \tag{10}$$

$$C(T) = C_1 = 10^5\,[kg/m^3] \qquad for \quad T < T_{freez} - \Delta T_{melt} \tag{11}$$

$$C(T) = C_1 \cdot \gamma \qquad for \quad T_{freez} - \Delta T_{melt} < T < T_{freez} \tag{12}$$

This formulation ensures a realistic behaviour, because the resistance to the flow circulations grows with the solid fraction, although the physical consistent form of the constant in the freezing window (Eq. 12) should be assessed by proper experimental data. The high value of the constant in the solid region (Eq. 11) ensures that the velocity is practically 0 for $T<T_{freez}-\Delta T_{melt}$.

The whole formulation is completely auto-consistent and guarantees that the correct heat transfer mechanisms can take place. In the liquid range, for $T>T_{freez}$, the set of the equations to solve is that of an ordinary single-phase flow; in the solid range, for $T<T_{freez}-\Delta T_{melt}$, the convective terms are negligible and the heat transfer is governed by conduction; in the freezing windows, $T_{freez}-\Delta T_{melt} <T<T_{freez}$, the convective terms decreases while the solid fraction increases.

The global temperature distribution and the energy balance is completely physically-consistent because the freezing window is tied to the freezing latent heat ΔH_{melt}.

3. Application to a prototypical case

3.1 Model

The general method illustrated in section 2.2 is now applied to a prototypical case of lead pool to assess the robustness and the global feasibility of the formulation. The overall dimensions of the lead cylindrical pool are close to those of the LFR DEMO ALFRED with a radius of about 4 *m*. A section of the computational domain with the mesh is shown in Figure 1.

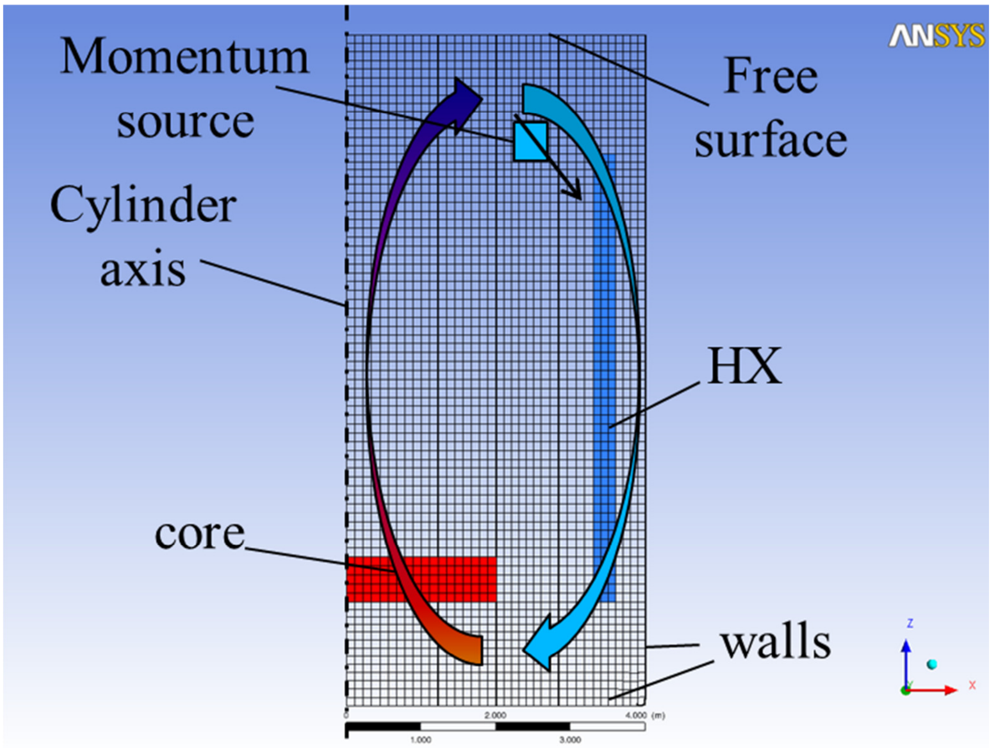

FIGURE 1 CFD COMPUTATIONAL DOMAIN USED TO TEST THE FREEZING MODELING.

The pool is cylindrical with the cylinder axis indicated in Figure 1: a small slice of cylinder has been simulated with periodic boundary conditions between the sides. The free surface on the top is simulated by a free-slip boundary condition, while at the side and bottom walls the ordinary no-slip boundary conditions have been imposed. Although the focus is not on the accuracy of the simulation but in the method feasibility, a 10 *mm* thickness well resolved layer at the walls allows to fully capturing the hydrodynamic boundary layer in these regions. A core region provides to the system the power density and the HX represents the heat sink of the system. All the dimensions are representative of the DEMO LFR. A momentum source is located in a region of the domain on the top to test a mixed convection case.

The goal is to obtain a stationary converged solution in natural and mixed circulation with partial freezing.

The choice of correct boundary conditions is crucial for the global stability of the procedure. As first test cases, the following choices have been selected:

- ✓ The core region has a total power source corresponding to the heat decay, i.e. 15 MW in the case of the LFR DEMO.
- ✓ The vertical temperature profile in the HX has been imposed to vary from 326 °C to 330 °C by the use of appropriate source terms; this condition ensures the partial freezing in the HX.
- ✓ The wall temperature has been fixed to ~350 °C, i.e. 20 °C above the freezing point.
- ✓ The initial temperature was fixed to 350 °C.

3.2 Results

3.2.1 Natural Convection Case

If the momentum source term is set to 0, the flow is driven by the buoyancy forces. This represents a prototypical picture in the LOFA conditions, with the coast-down of the main circulation pump and the decay heat to be removed from the system.

Figure 2 shows the equation residuals against the iteration number for the test case. Residuals are 10^{-5} or lower for all the equations in about 6000 iterations, and thus the solution can be considered converged.

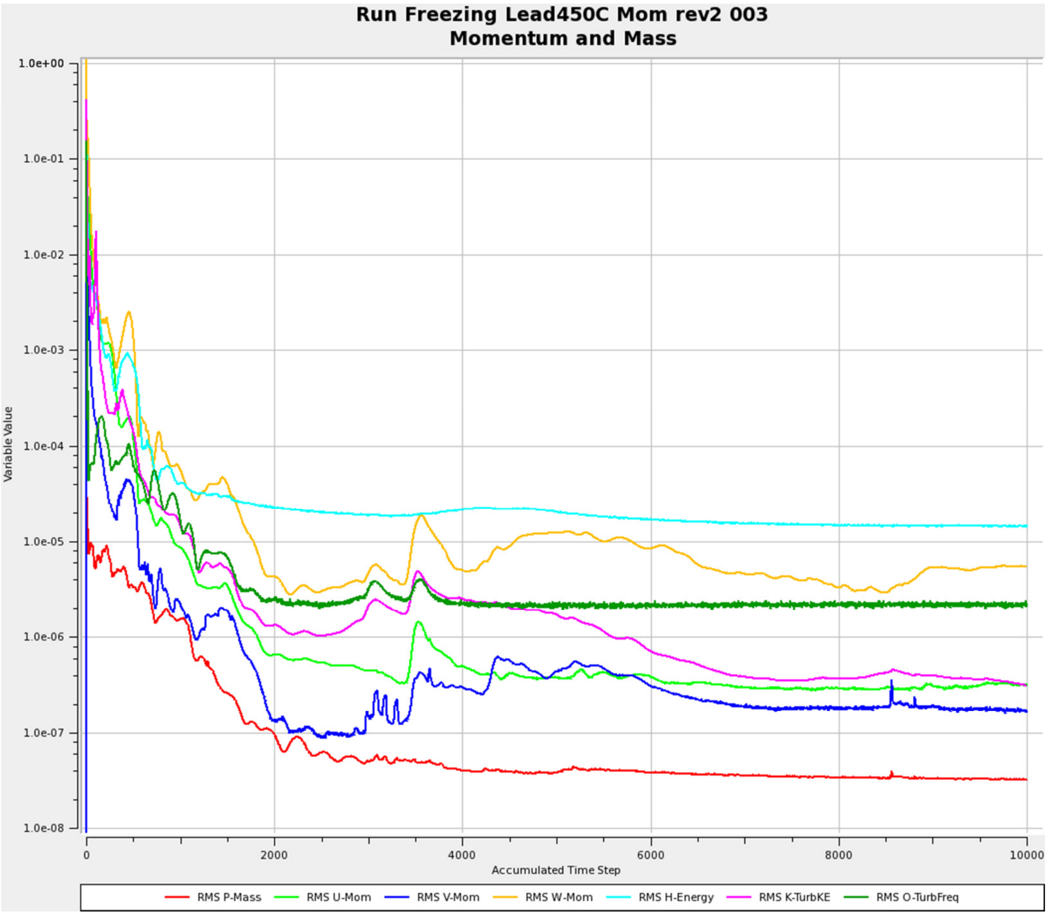

FIGURE 2 RESIDUALS AGAINST THE ITERATION NUMBER IN THE FREEZING TEST CASE.

Figure 3 shows the maximum, minimum and monitoring point temperatures as a function of the iteration number: the quantities reach asymptotic constant values. Figure 4 shows vertical velocities in two monitoring points as a function of the iteration number: the quantities reach asymptotic constant values.

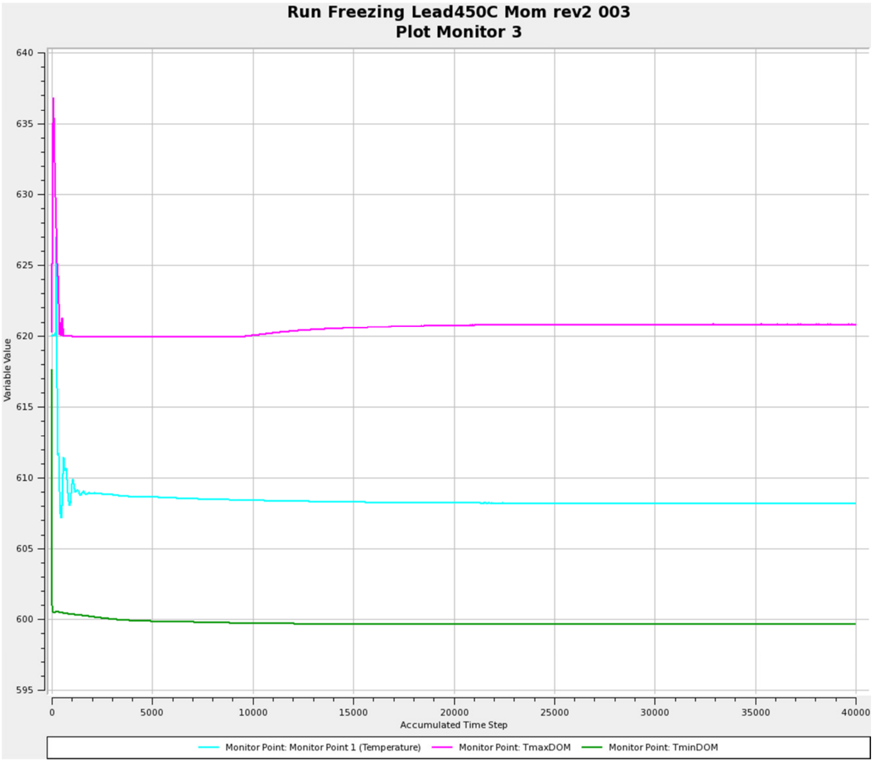

FIGURE 3 MAX, MIN AND MONITORING POINT TEMPERATURE AGAINST THE ITERATION NUMBER IN THE FREEZING NATURAL CONVECTION TEST CASE.

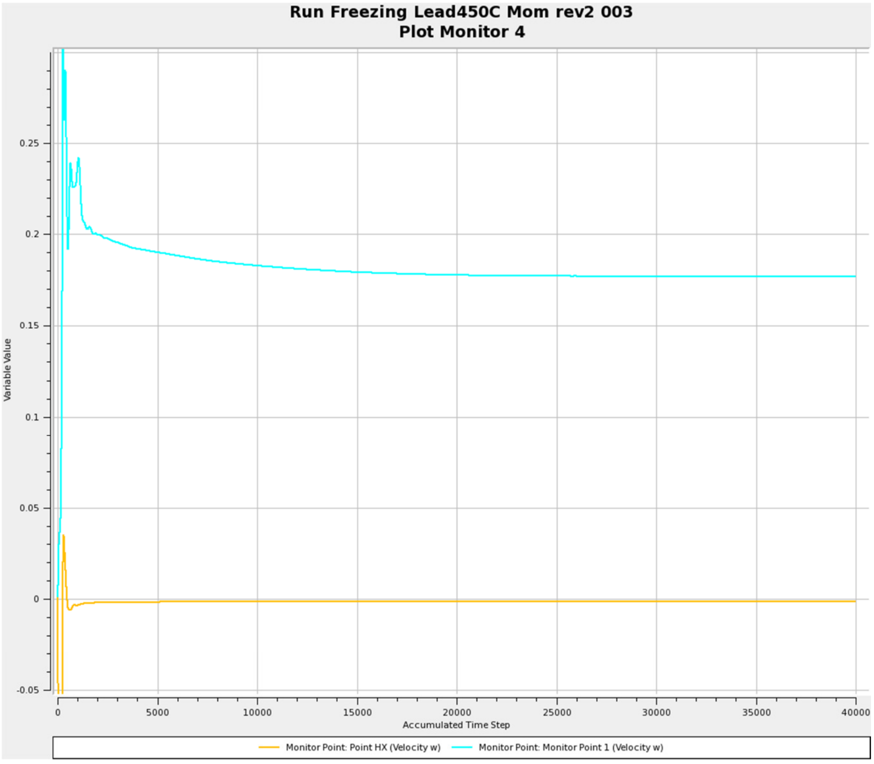

FIGURE 4 VELOCITIES IN TWO MONITORING POINTS AGAINST THE ITERATION NUMBER IN THE FREEZING NATURAL CONVECTION TEST CASE.

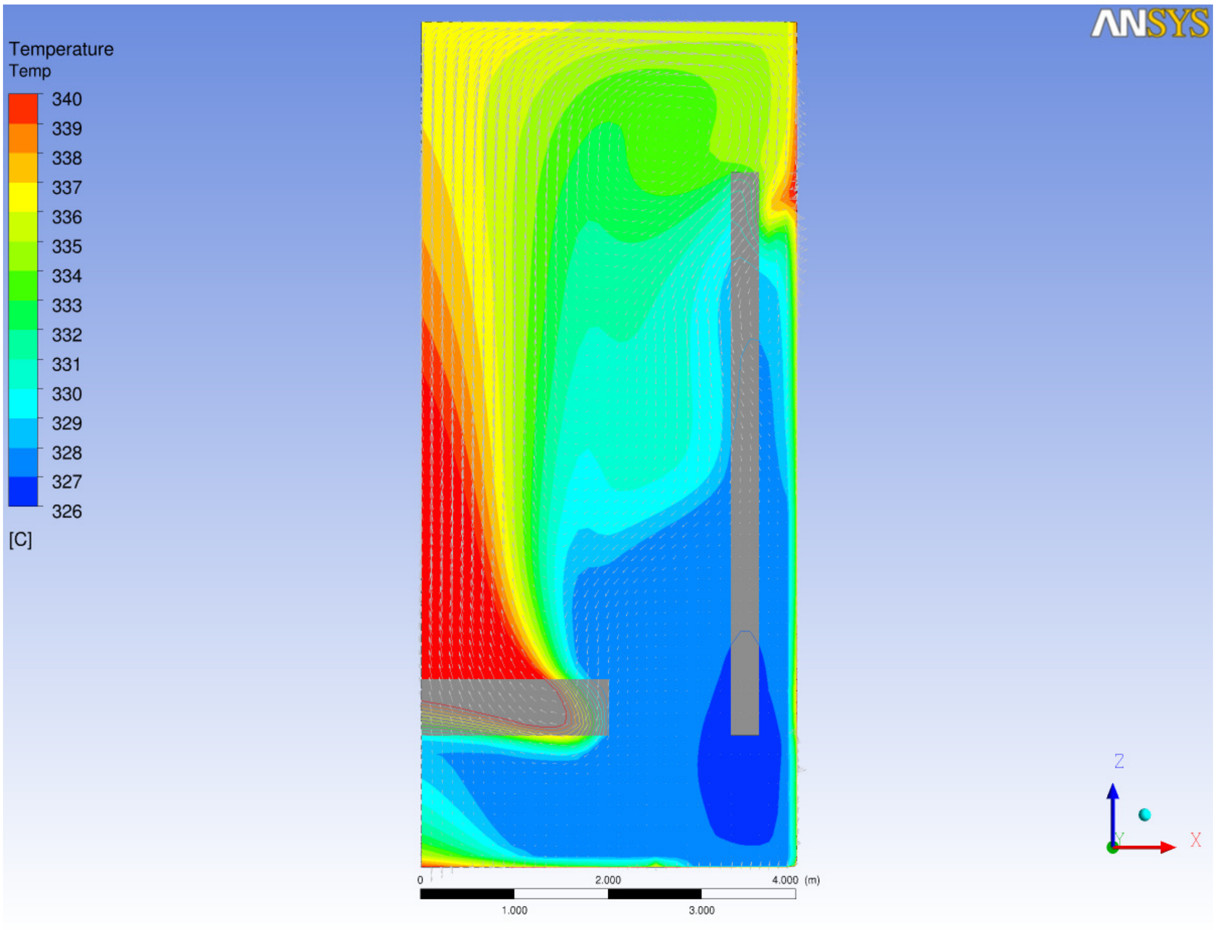

FIGURE 5 TEMPERATURE CONTOURS IN THE FREEZING NATURAL CONVECTION TEST CASE.

Figure 5 shows the temperature field of the converged solution; the velocity vectors are superimposed. A natural circulation cell establishes to drive hot fluid from the core to the HX. The lead in the HX is frozen in the lower part where the temperature falls below the freezing temperature. A large region (in **blue**) attached to the HX is completely frozen with *0* velocity. The velocity vector field reflects the structure of the negative source term, Eqs. 6 to 12.

3.2.2 Mixed Convection Case

The momentum source term in the region indicated in Figure 1 simulates the presence of a pump. If this term is switched on, the flow in the pool is driven both by buoyancy forces and by the pressure head provided by the pumping system. A pressure head of 0.25 *bar* has been imposed in the transversal direction indicated in Figure 1.

The convergence in this case is similar to the natural convection case with the quantities reaching asymptotic values.

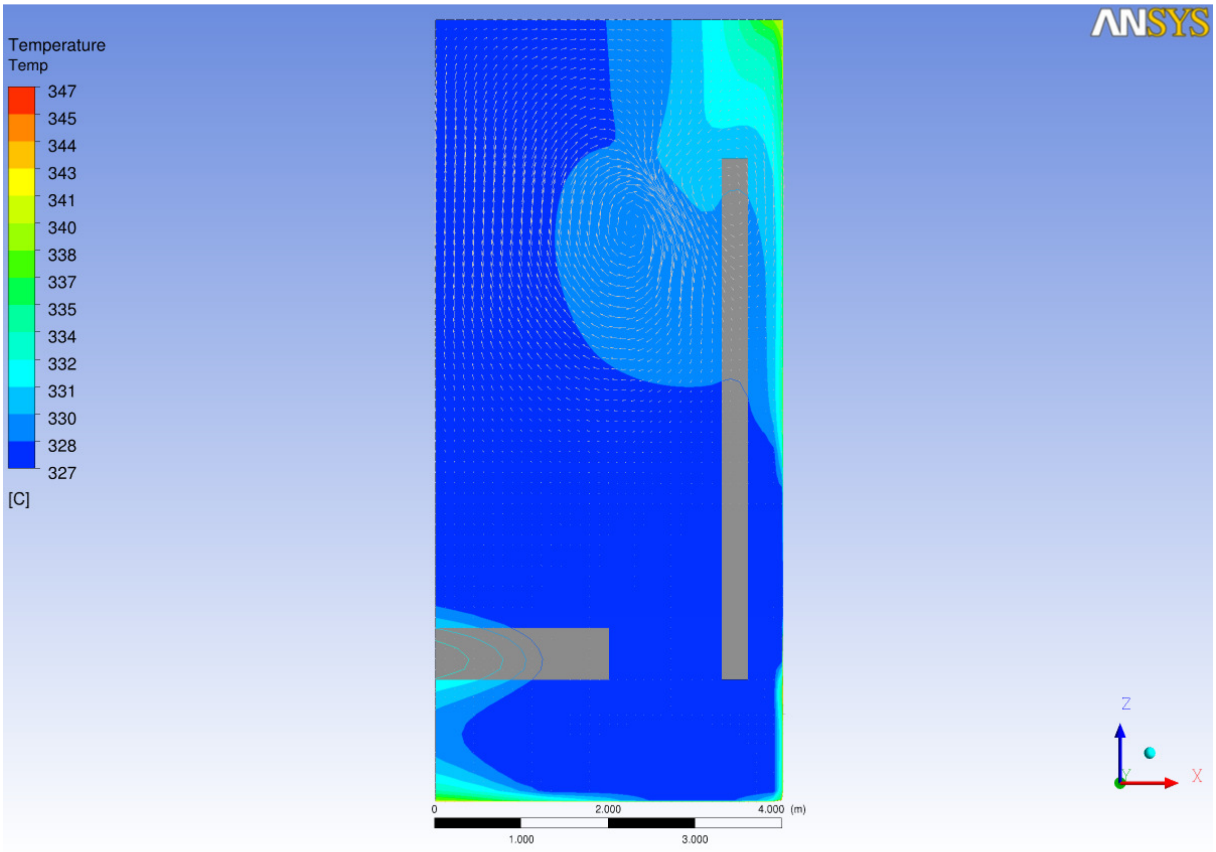

FIGURE 6 TEMPERATURE CONTOURS IN THE FREEZING MIXED CONVECTION TEST CASE.

Figure 6shows the temperature field of the converged solution; the velocity vectors are superimposed. A circulation cell establishes to drive hot fluid from the core to the HX. A large region (in **blue**) attached to the HX towards the core is completely frozen with *0* velocity.

Although the case is not probably very realistic for the pump position, it is presented only to show the possibility to introduce and model pumps in the CFD modeling of the pool circulation.

4. Conclusions

A CFD method is presented to simulate fluid flow and heat transfer during solidification of lead in pools. The method allows formally maintaining a single-phase approach introducing momentum source terms depending on local velocity and temperature dependent specific heat. The global physical consistence of the method is guaranteed by the fact that the melting latent heat is correctly kept into account in the formulation. In the fully frozen regions the velocities are 0 and the transport of heat is governed by the thermal conductivity. Where the solid fraction is between 0 and 1, the convective terms are dumped according to the specific form of the momentum source term.

The method has been applied successfully to prototypical cylindrical pool with core and HX described by energy source terms. The method provided fully converged results both in natural and mixed circulation. Although the cases presented are not very realistic, they are provided to show the global consistency and robustness of the method.

The full feasibility of the method is proved. The next step towards a full assessment of the method is to find a more accurate form of the source terms in the melting regions by comparison with experimental data. Afterword, the method can be applied to more realistic systems.

6. References

[1] A. Alemberti, J. Carlsson, E. Malambu, A. Orden, D. Struwe, P. Agostinif, S. Monti, European lead fast reactor—ELSY, *Nucl Eng Design*, **241**, pp.3470-3480, 2011.

[2] ANSYS CFX Release 13 User Manual.

[3] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA J*, **32**, pp.269-289, 1994.

[4] X. Cheng, N.I. Tak, CFD analysis of thermal-hydraulic behavior of heavy liquid metals in sub-channels, *Nucl Eng Design*, **236**, pp.1874-1885, 2006.

[5] I. Di Piazza, M. Scarpa, Rassegna di Letteratura sulla Termoidraulica dei Bundle Refrigerati a Metallo Liquido Pesante, ENEA Report LM-FR-001, 2012.