Retraction Notice

Title of retracted article:	Numerical Assessment of Prandtl Number Effect on Transient Heat Flux Distribution Imposed on Nuclear Reactor Pressure Vessel by Application of PECM in a Volumetrically Heated Molten Pool
Author(s):	Mohammad Khan
* Corresponding author' Email:	mmhk@kth.se

Journal:	Engineering (ENG)
Year:	2019
Volume:	9
Number:	8
Pages (from - to):	504-522
DOI (to PDF):	https://doi.org/10.4236/eng.2019.118035
Paper ID at SCIRP:	94558
Article page:	http://www.scirp.org/Journal/paperinformation.aspx?paperid=94558
Retraction date:	2019-09-12

Retraction initiative (multiple responses allowed; mark with **X**):

X All authors
☐ Some of the authors:
☐ Editor with hints from
- O Journal owner (publisher)
- O Institution:
- O Reader:
- O Other:

Date initiative is launched: 2019-09-12

Retraction type (multiple responses allowed):

☐ Unreliable findings
O Lab error O Inconsistent data O Analytical error O Biased interpretation
O Other:
☐ Irreproducible results
☐ Failure to disclose a major competing interest likely to influence interpretations or recommendations
☐ Unethical research

☐ Fraud
O Data fabrication O Fake publication O Other:
☐ Plagiarism ☐ Self plagiarism ☐ Overlap ☐ Redundant publication *
☐ Copyright infringement ☐ Other legal concern:

☐ Editorial reasons
O Handling error O Unreliable review(s) O Decision error O Other:

X Other: there are some errors in the discussion and conclusion part.

Results of publication (only one response allowed):

X are still valid.
☐ were found to be overall invalid.

Author's conduct (only one response allowed):

☐ honest error
☐ academic misconduct
X none (not applicable in this case – e.g. in case of editorial reasons)

* Also called duplicate or repetitive publication. Definition: "Publishing or attempting to publish substantially the same work more than once."

History

Expression of Concern:

☐ yes, date: `yyyy-mm-dd`

X no

Correction:

☐ yes, date: `yyyy-mm-dd`

X no

Comment:

There are some errors in the discussion and conclusion part.

This article has been retracted to straighten the academic record. In making this decision the Editorial Board follows COPE's Retraction Guidelines. Aim is to promote the circulation of scientific research by offering an ideal research publication platform with due consideration of internationally accepted standards on publication ethics. The Editorial Board would like to extend its sincere apologies for any inconvenience this retraction may have caused.

Engineering, 2019, 11, 504-522
http://www.scirp.org/journal/eng
ISSN Online: 1947-394X
ISSN Print: 1947-3931

Numerical Assessment of Prandtl Number Effect on Transient Heat Flux Distribution Imposed on Nuclear Reactor Pressure Vessel by Application of PECM in a Volumetrically Heated Molten Pool

RETRACTED

Mohammad Khan

Division of Nuclear Power Safety, Royal Institute of Technology (KTH), Alba Nova University Center, Stockholm, Sweden

Email: mmhk@kth.se

How to cite this paper: Khan, M. (2019) Numerical Assessment of Prandtl Number Effect on Transient Heat Flux Distribution Imposed on Nuclear Reactor Pressure Vessel by Application of PECM in a Volumetrically Heated Molten Pool. *Engineering*, **11**, 504-522.
https://doi.org/10.4236/eng.2019.118035

Received: July 23, 2019
Accepted: August 20, 2019
Published: August 23, 2019

Abstract

In the framework of this research, the principle focus is to analyze the effects of fluid Prandtl number (Pr) on natural convection heat transfer in a volumetrically heated molten pool. As a part of the work, numerical analysis is performed for hemispherical 3-D vessel slice to investigate the physics of the effect of Pr number on convective heat transfer characteristics in the melt pool. The investigation is based on ANSYS FLUET, where natural convection heat transfer effect is taken into consideration by Phase-change Effective Convectivity Model (PECM), which is implemented with FLUENT CFD as User Defined Function (UDF), programed by the user. The PECM is tested first by a benchmark test against CFD to gain confidence in its applicability as an analysis tool. Different simulant materials are used with their thermo-physical properties representing different Pr number as input for modelling for both single and double layer melt pool configuration. The selected modelling approach is validated against RASPLAV experimental result with respect to the inner temperature distribution that qualifies our model to run in the proceeding calculation. It is ensured that an isothermal boundary condition (343 K) is applied along vessel outer wall throughout the series of simulation cases. The corresponding Rayleigh number (Ra) ranges from 10^{14} - 10^{15} and Prandtl number (Pr) 3 - 5. It is found that the fluid Pr number has small effects on the averaged Nu numbers in the convection-dominated regions. The decrease in the Pr number may cause a decrease in the Nu numbers on the top and sidewalls of cavities. In the conduction dominated regions (stably stratified bottom parts of enclosure), the effect of fluid Pr number on heat transfer is more significant and it grows with increasing Ra number.

Keywords

Phase Change Effective Convectivity Model (PECM), Computational Fluid Dynamics (CFD), User Defined Function (UDF)

1. Introduction

In the hypothetical case of a severe accident, the reactor core could melt and form a mixture, called corium and can relocate to the lower plenum of the reactor pressure vessel (RPV) and form an in-vessel debris bed [1]. If there is no effective cooling, the core debris may heat up by decay heat and evolve to a molten pool, which can threaten the thermal and structural integrity of the reactor vessel [2]. The phenomena associated with melting core and relocation of the molten debris play an important role in the degree of material mixing and interaction in the lower head. When the corium is relocated in the lower head and re-melting occurs, the main heat transfer mechanism is the volumetric heat source (representing decay heat) driven natural convection [3]. Research interest in natural convection has been motivated by its relevance in many applications including geophysical, chemical, and nuclear. In particular, thermal convection driven by internal heat sources plays an important role in the post-accident heat removal problem in the event of a core meltdown accident in a nuclear power reactor.

Therefore, an intensive knowledge of the natural convection heat transfer in a volumetrically heated stratified pool for different configuration is essential for predicting and preventing thermal failure of the RPV during severe accident scenario of an LWR. In order to get insight of the convective heat transfer phenomena, experimental research is essential. Investigations of natural convection phenomena in a fluid with volumetric heat generation began in the early 1970s with Kulacki and collaborators [4] [5] [6], who conducted several experiments using Joule heating as a volumetric heat source. In those experiments, heat transfer through a horizontal fluid layer was assessed for different boundary cooling arrangements. Some of the experimental works done previously with prototypic materials are as follows:

MASCA test [7] utilized prototypical materials to study of melt stratification and distribution of major species (U, Zr) in the melt pool. It is found that some amount of metallic uranium & zirconium may migrate from oxide phase to metal, leading to the change of density and inversion of corium pool configuration.

The RASPLAV Program [8], conducted in Russia, using prototypic (UO_2-ZrO_2) materials, in which the thermal loadings imposed by the prototypic melt on a cooled vessel wall are measured. It is found that the homogeneous corium melt behaved comparably to simulant materials in natural circulation. However, using prototypic materials (corium), which are radioactive and have very high melting temperatures, to carry out this type of experiments, is limited because of many technical difficulties and safety requirements. Therefore, many experimental programs were conducted with molten salt to study natural heat transfer in a volumetrically heated molten pool. Some of the experiments done previously with simulant materials (corium salts) are as follows:

Experimental program of SIMECO (Simulation of Melt Coolability) [9] was performed at KTH, Sweden with the objective of investigating effect of boundary crusts and mushy layers on natural convection heat transfer, the effects of melt stratification on natural circulation and so on. The experiment concluded that the upward Nusselt number is close to that determined from the Steinberner-Reineke correlation. The LIVE [10] experimental program was performed with the objective to investigate the late In-Vessel core melt behavior in terms of heat flux along vessel wall, temperature in the melt pool and crust thickness. The main focus of the LIVE experimental program was to address remaining uncertainties in melt pool heat transfer with phase change. The COPRA (Corium Pool Research Apparatus) [11] experimental facility was designed to investigate the in-vessel molten corium pool behavior applying in different melt volumes, heat generation rates. It presented the behavior of a large-scale homogenous melt pool in transient and steady state conditions.

Even though several research works have been performed in this regard, a numerical analysis is far more than complementary due to the complexity and limitations in measurement capabilities. Therefore, an efficient numerical analysis tool is needed to study such phenomena, as until today entire scenario of convective heat transfer is not well understood. ANSYS CFD is a set of numerical methods applied to model fluids in multidimensional space using the Navier-Stokes equations [12]. However, CFD code cannot cover the entire scenario of convective heat transfer using one comprehensive model. Moreover, considering 3D geometry and mesh, CFD method is computationally expensive and not affordable for so many cases in sensitivity analysis of the key parameters. Therefore, a simplified model called Phase-change Effective Convectivity Model (PECM) [13] is used in the present work, discussed in details later in Section 2.3. Furthermore, the PECM uses reduced characteristics velocities as a function of the melt mass fraction to describe the phase change heat transfer and represent the natural convection heat transfer at mushy zones [14]. The characteristic velocities are determined using heat transfer correlations based on Rayleigh number, namely the upward, sideward and downward Steinberner-Reineke correlations [15].

The current paper demonstrates the simulation case utilized for investigating the effect of Pr number on melt pool thermal hydraulics by applying PECM (Section 2) with their thermo-physical properties as input for modelling for both single and double layer melt pool configuration (Section 3). The selected modelling approach is validated against RASPLAV experiment (Section 4) with respect to the inner temperature to qualify our model to run the proceeding calculation using PECM. After successful completion of the benchmark test of PECM (Section 5) against FLUENT CFD, PECM is applied to single-layer & two-layer melt pool configuration. Finally, quantification of thermal load is compared among different simulant materials corresponding different Pr number for single and double layer configuration (Section 6).

2. Modelling & Numerical Treatment

2.1. Geometry of the Model

The preliminary design of the model representing lower head of the reactor pressure vessel is a semicircular slice with a radius of 500 mm and a thickness of 120 mm (**Figure 1**). The domain is considered to have a single and two-layer configuration: top layer and bottom layer to represent the light metal layer and oxide layer, respectively. The LiCl-$CeCl_3$, NaCl-$BaCl_2$, CsCl-KCl-LiCl system are used as oxide layer and Al is used as top metal layer simulants of corium, respectively. A volumetric heat source (Q_v) is implemented in the oxide layer to represents decay heat. A water-cooling system is used to maintain isothermal boundary condition 343 K (~70˚C) surrounding the test section. This type of boundary conditions has been well documented in [16] [17] [18], as it has not required any additional information.

The single & two-layer configuration geometry model was developed by ANSYS ICEM and the corresponding computational mesh has been generated by ICEM CFD using O-grid method which guaranteed the unstructured mesh having hexahedral cells in Section 3.1. The mesh near the boundaries was refined as shown in **Figure 2** & **Figure 3**. It should be noted that for the visualization of crust formation, the quartz glass has been used in the front and backside of the vessel, oxide layer and metal layer.

2.2. Computational Domain

Another important step of the simulation work is drawing meshes for the working model. Mesh cannot be equilibrium distributed since more mesh should be

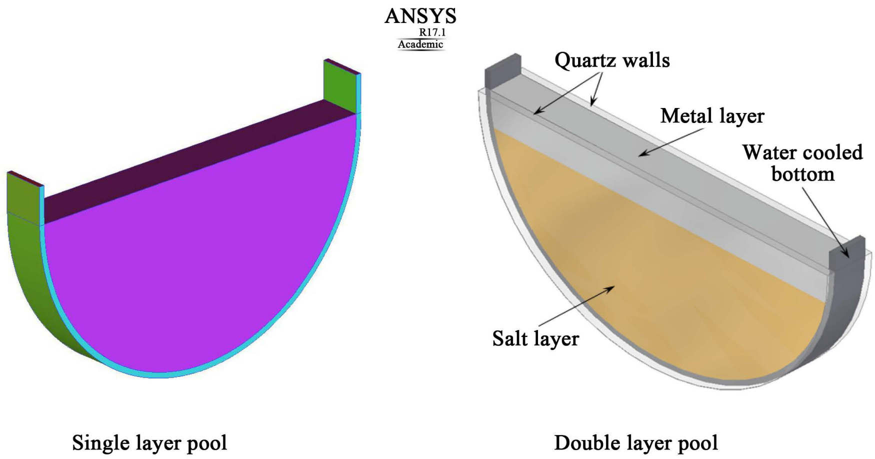

Figure 1. Conceptual design of the model.

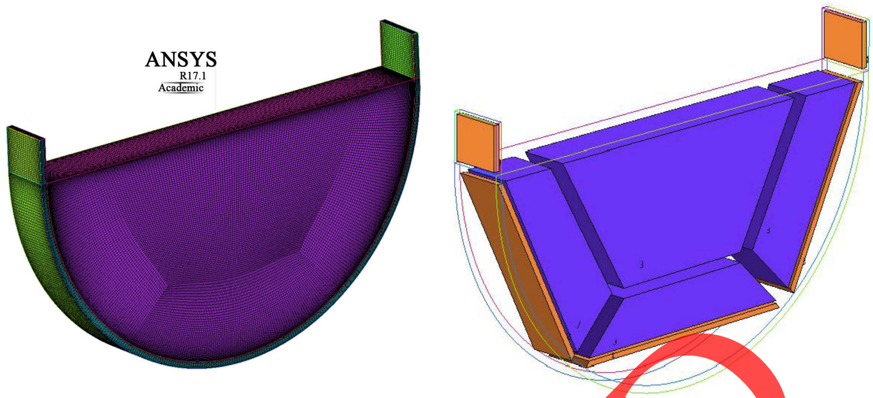

Figure 2. Mesh of single layer melt pool.

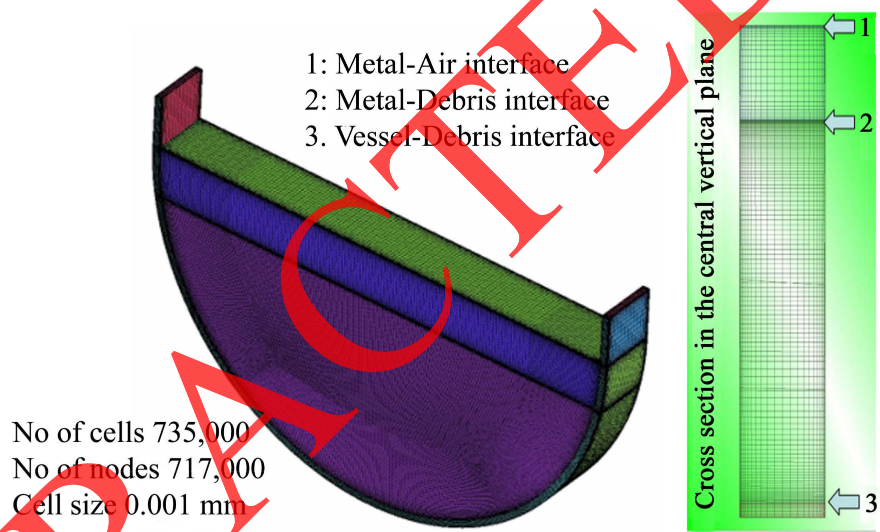

Figure 3. Mesh of double layer melt pool.

placed near the model boundaries. The single layer and two-layer configuration oriented model has been developed by ANSYS ICEM and the corresponding computational mesh is generated by ICEM CFD using O-grid method which split the model into different areas and guaranteed the unstructured mesh having hexahedral cells as shown in Figure 2 & Figure 3.

The mesh near the boundaries is refined as shown in Figure 2 & Figure 3.

Before starting the simulation, mesh sensitivity analysis and mesh-independence test is carried out in the succeeding simulations, which consist of 735,000 cells and 717,714 nodes for two-layer configuration with 0.10 m top layer and 580,260 cells & 591,250 nodes for single layer configuration. The minimum quality is recorded 0.92 according to the FLUENT mesh quality histogram (Figure 4) which is enough to produce precise simulation result.

2.3. Implementation of the PECM Model

Phase Change Effective Convectivity Model (PECM)

Most difficulties are in the mathematical description of free-convection and heat

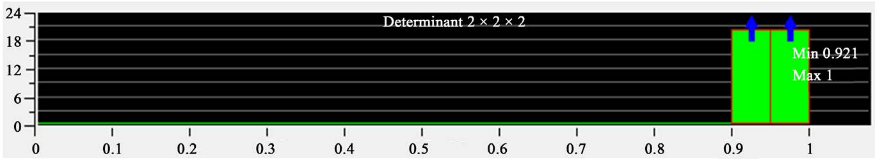

Figure 4. Mesh quality histogram.

transfer of the heat-generating fluid. Using traditional turbulence models, which have been developed for the case of forced convection [19] [20], is not justified and can give considerable errors when calculating heat flux, because these models do not describe turbulence generation due to thermal gravity, stable and unstable stratification, and the like. From the engineering standpoint, the simplest and most efficient way of simulation is to use integral heat balance correlations. In this case, empirical correlations for the heat transfer coefficient are used to close the problem, which has been used in the current PECM tool. In the present study, natural convection heat transfer is accounted for by only the Effective Convectivity Model (ECM) where the heat transport and interactions are represented through an energy-conservation formulation. In order to describe the Phase-Change heat transfer, a temperature-based enthalpy formulation is employed in the ECM (so called Phase-Change ECM or PECM) which is capable to represent possible convection heat transfer in a mushy zone. The simple approach of the PECM method allows implementing different models of the fluid velocity in a mushy zone for a non-eutectic mixture. The developed model is validated by a dual approach, *i.e.*, against the existing experimental data and the CFD simulation results.

In this model, the convective terms of the energy conservation equation are described using directional characteristic heat transfer velocities to transport the heat; therefore, the need of solving Navier Stokes equations are eliminated. This assumption makes this model much more computationally efficient than conventional CFD codes. Furthermore, the PECM uses reduced characteristics velocities as a function of the melt mass fraction to describe the phase change heat transfer and represent the natural convection heat transfer at mushy zones. The characteristic velocities are determined using heat transfer correlations based on Rayleigh number, namely the upward, sideward and downward Steinberner-Reineke correlations. The PECM is implemented in the commercial code FLUENT by using User Defined Functions (UDF) utilizing all advantages of a CFD commercial code solver such as the pre- and post-processing and has been validated against many experiments . In the PECM, the flow velocities u_x, u_y, u_z in the energy equation are replaced by characteristic velocities U_X, U_Y and U_Z respectively. The characteristic velocities are calculated with empirical correlations, such that the necessity of solving the Navier-Stokes equation can be eliminated.

As soon as the characteristic velocities are calculated, the new energy equation can be expressed as follows [21]:

$$\frac{\partial\left(\rho C_p T\right)}{\partial t}+\left(\frac{\partial\left(\rho C_p U_x T\right)}{\partial x}+\frac{\partial\left(\rho C_p U_y T\right)}{\partial y}+\frac{\partial\left(\rho C_p U_z T\right)}{\partial z}\right)=k\nabla^2 T+Q_v \tag{1}$$

where, the first term is the transient term, 2^{nd}, 3^{rd} and 4^{th} terms are the convection terms on the left hand side and on the right hand side, diffusion & volumetric heat source corresponds to the 1^{st} and 2^{nd} terms respectively. The convection term then can be calculated explicitly with use of values in the last time step, and then can be treated together with the volumetric source term. Therefore, the PECM finally simplifies the energy equation to a conduction equation which is easy for solving and as a result, will significantly increase the computational efficiency in simulations.

The characteristic velocities are defined using heat transfer correlations. In the bottom layer which involves a volumetric heat source driven natural convection, the Steinberner-Reineke correlations are employed to get the characteristic velocities shown as follows [22]:

$$\begin{cases} U_{up}=\dfrac{\alpha}{h_{pool}}\times\left(Nu_{up}-\dfrac{h_{pool}}{h_{up}}\right) \\ U_{down}=\dfrac{\alpha}{h_{pool}}\times\left(Nu_{down}-\dfrac{h_{pool}}{h_{down}}\right) \\ U_{side}=\dfrac{\alpha}{h_{pool}}\times\left(Nu_{side}-\dfrac{2\times h_{pool}}{W_{pool}}\right) \end{cases} \tag{2}$$

where, h_{pool} is the melt pool depth (m), h_{up} is the thickness of well mixed layer of the pool (m), h_{down} is the thickness of lower stratified region of melt pool (m), W_{pool} is the width of pool (m); α is the thermal diffusivity (m^2/s). The profile of sideward Nu is described using the Eckert's type correlation for a vertical boundary layer. The PECM employs reduced characteristic velocities to describe mushy zone convection heat transfer.

In the top metal layer, which involves the Rayleigh-Benard natural convection, the Globe-Dropkin correlation is used for the calculation of upward characteristic velocity with the Churchill-Chu correlation used for horizontal characteristic velocity.

$$\begin{cases} U_{up}=\dfrac{2\alpha}{h}\times\left(Nu_{up}-1\right) \\ U_{side}=\dfrac{\alpha}{h}\times\left(Nu_{side}-\dfrac{h}{W}\right) \end{cases} \tag{3}$$

where, the h and w are the liquid metal layer thickness and width, respectively:

The PECM also considers the solidification/melting process and the physical properties vary as a function of temperature.

It is also noted, that though the simplification of the equation can significantly speed up the calculation and gives good heat flux profiles. The method still somehow introduces some minor distortion in the temperature field: the temperature increase from the boundary to the bulk is more rapid than that of CFD method, which means the bulk temperature domain may be a bit larger than

CFD results.

3. Physical Properties of Simulant Materials and Test Matrices

The thermo-physical properties of the simulant materials used in the simulation has been presented here in Table 1.

Test Matrix & Boundary Condition

The tests were performed for different cooling conditions of top and sidewalls of either insulated or isothermal or radiation boundary conditions as shown in Table 2.

Boundary Conditions Applied

Volumetric power in the molten salt layer (MW/m^3)	1.5
Temperature of water cooled outer surface of the steel vessel (K)	343
Radiation from the	
– top metal layer	0.15
– Top oxide layer	0.5
– front/back quartz wall	0.5
– front/back vessel wall	0.5
Ambient temperature (K)	298
Initial temperature of the pool (K)	1073

In present work, PECM is implemented in a version of ANSYS FLUENT 17.1 [24]. The simulations were run on a LENOVO D20 workstation with two Xeon (R) E5620 @ 2.4 GHz CPUs and 20 GB RAM. Each calculation typically costs about 9 hours for two-layer configuration cases to reach a steady state.

Table 1. Thermo-physical properties of different simulant materials [23].

Properties	$NaCl\text{-}BaCl_2$	$LiCl\text{-}CeCl_3$	CsCl-KCl-LiCl
Solidus temperature (K)	923	768	533
Liquidus temperature (K)	983	1023	778
Density, Kg/m^3	2323	3007	1825
Viscosity, mPa.S	2.4	2.72	1.45
Specific heat (KJ/Kg*K)	663	682.9	898.7
Thermal conductivity (W/m*K)	0.37	0.662	0.41
Latent heat of fusion (J/Kg)	203,526	229,152	298,369
Thermal expansion coefficient	0.000447	0.00029	0.0003517

Table 2. Simulant materials corresponding different fluid Pr number.

Material used	Pr	Ra, Single layer (Pool height 500 mm)	Ra, Double layer (Pool height 400 mm)
$NaCl\text{-}BaCl_2$	5.42	1.39×10^{15}	4.53×10^{14}
$LiCl\text{-}CeCl_3$	3.80	4.65×10^{14}	1.52×10^{14}
CsCl-KCl-LiCl	3.10	1.32×10^{15}	4.33×10^{14}

4. Benchmark Test of PECM against ANSYS CFD

First, we present a comparison between PECM and CFD simulations of as a reference case. The reference case is a single layer configuration of oxide layer (NaCl-$BaCl_2$) with a radius of 0.5 m. The vessel outer surface (Figure 1) is cooled by water with constant temperature 343 K.

B. C applied Side and top walls: Adiabatic, Vessel outer surface: Isothermal cooling (343 K).

The CFD includes a viscous heating model with SST k-omega turbulence model and a pressure-based solver. Solidification and melting model is also included. Figure 5 shows heat flux distributions along the vessel in both simulations. Both heat flux profiles of PECM method and CFD method increase along the polar angle. The increase trend is slow when polar angle is small while it goes steep as the angle become larger and close to 90˚. This trend can be explained by the fact that in pool natural convection, the lower part region of the pool is laminar which is dominated by heat conduction and the upper part region is turbulent and dominated by heat convection. Both profiles have quite close minimum value and peak value. It is also observed that in small polar angle region (0˚ - 60˚), PECM gives higher value than CFD value. The reason is that the temperature field of PECM may have larger bulk domain and a relatively higher temperature in the lower part of the pool, which will result in a relatively higher heat flux. Meanwhile, as the total removed energy should be equal to each other in steady state, the heat flux value in high polar angle domain of PECM is lower than that of CFD. Overall, results of the PECM and CFD show good agreement to each other in the local heat flux distribution. As CFD method is a more mechanistic way and may be computational expensive, the agreement between CFD and PECM methods may suggest that PECM method was properly used in the simulations and can give acceptable results.

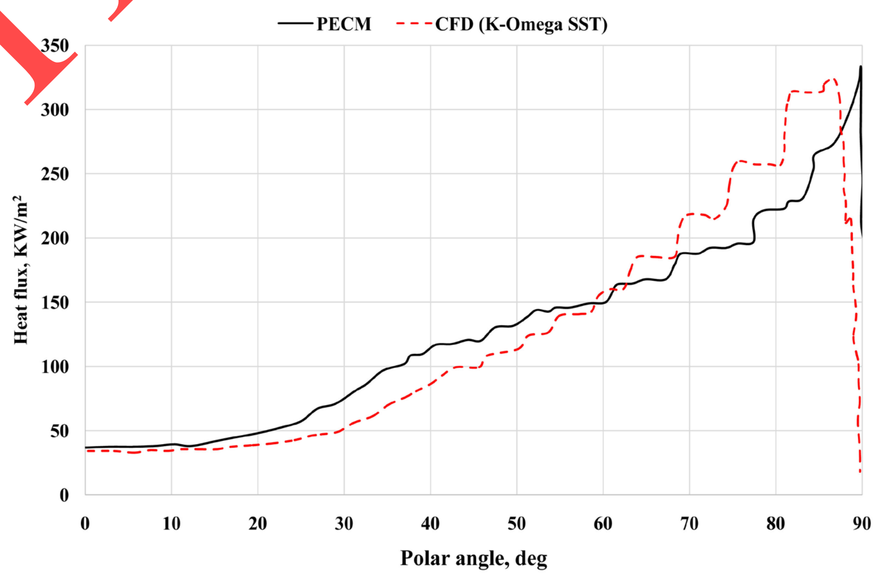

Figure 5. Comparison between PECM and CFD simulation results.

5. Validation

The first part of this work is to simulate the previous experimental work applying PECM to qualify the current modelling approach so called validation process. As a part of validation procedure, RASPLAV Experimental facility (Figure 6) is selected, as our geometry model and working temperature are similar to RASPLAV experiment. As a part of validation procedure, a simulation case is run using RASPLAV experimental data (test wall outer surface temperature):

Boundary conditions applied

Test wall outer surface temperature: Non-isothermal boundary condition, applied by UDF

Front & Back of the vessel: nearly adiabatic

Top of the vessel: nearly adiabatic

Volumetric heat source 333,000 W/m^3

Properties of Salt, Used in the RASPLAV Experiment [8]

Parameter	8NaF-92NaBF$_4$
Melting temperature, T_{sol}/T_{liq} (˚C)	384/385
Liquid phase density, ρ (kg/m^3)	1968 (400˚C)/1825 (600˚C)
Heat capacity, Cp (J/kg·K)	1507
Heat conductivity, λ (W/m·K)	0.45 (400˚C)/0.398 (600˚C)
Volumetric expansion coefficient, β (1/K)	3.61×10^{-4} (400˚C)/3.90×10^{-4} (600˚C)
Dynamic viscosity, μ (Pa·s)	2.45×10^{-3} (400˚C)/1.14×10^{-3} (600˚C)
Kinematic viscosity, ν (m^2/s)	1.25×10^{-6} (400˚C)/6.25×10^{-7} (600˚C)

Regarding deviation between the simulation and experimental results, one potential reason could be the placement of the thermocouples. The placement of the thermocouples may have been changed by the volume of the liquid salt. After the vessel is full of liquid salt, it may not be possible to check the placement of the thermocouples.

Another possible reason for the deviation may be in the material properties. It is assumed that the density, heat conductivity and viscosity of the salt are

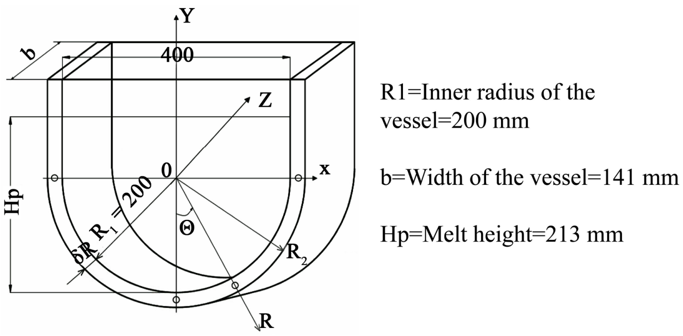

Figure 6. Coordinate system of RASPLAV-A-Salt Facility, Asmolov *et al.* [8].

constant. If the temperature dependent material properties were known, the numerical methods would have given more precise results for the inner temperature distribution. Thus, the temperature dependent properties of the salt should be well known in order to obtain sufficiently accurate results with the numerical methods.

The PECM model employed in the RASPLAV simulation case against RASPLAV experiment shows that there is a good agreement between PECM and RASPLAV experimental result with some minor deviation shown in Figure 7. It means that the PECM model applied in this work is capable of predicting natural convection heat transfer in a semicircular cavity with volumetric inner heat generation.

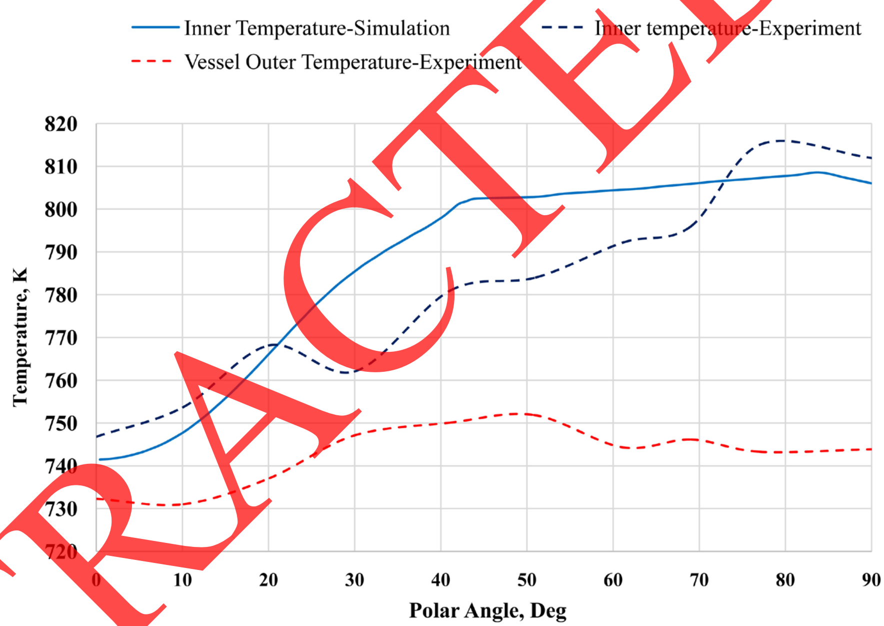

Figure 7. Inner temperature distribution along curved wall (Regime N 47-Crust free).

6. Results & Discussion

6.1. Single Layer Configuration

Generally, the external cooling forms the natural convective flows through which internally heated fluid gets cold and run down along isothermal boundaries (curved surface) and merge at the bottom & then move upward and disperse towards the edge at the top plate as shown in Figure 8.

To investigate the influence of Pr on convective heat transfer phenomena, three simulations were performed with three different materials using the BC: radiation with an emissivity 0.5 applied on the side walls, 0.15 applied on the top wall. Constant cooling temperature 343 K at vessel outer surface and internal heat source $Q_V = 1.5$ MW/m^3.

Figure 9-11 shows heat flux distribution along the vessel, temperature distribution along the central vertical line of the domain, and temperature field of

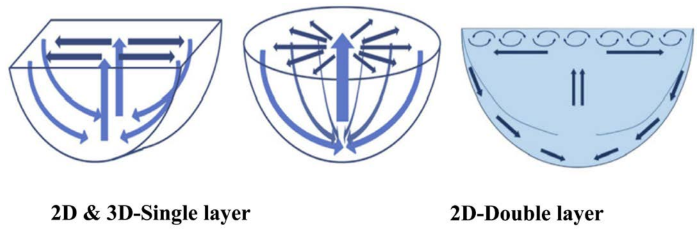

Figure 8. Traditional flow pattern by natural convection, Su-Hyeon Kim *et al.* [25].

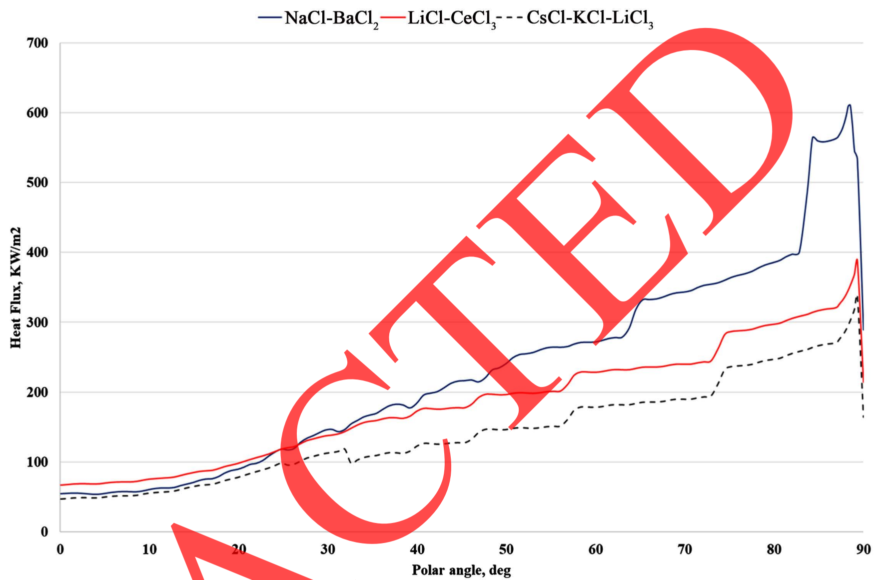

Figure 9. Angular heat flux distribution along the vessel.

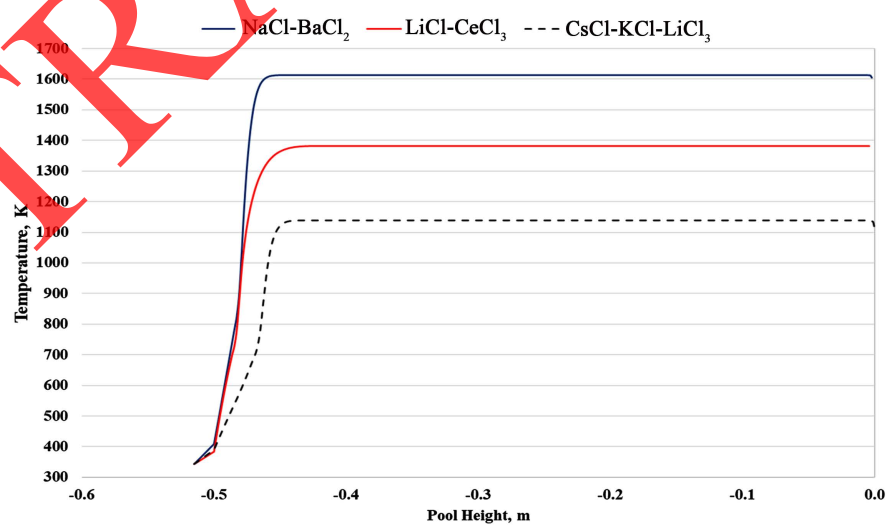

Figure 10. Temperature distribution along central vertical line in the pool.

central cross section respectively. As described above, the heat flux increases along the polar angle and the slope also increases, as the angle is large. Large bulk region is seen in **Figure 11** and central vertical temperature distribution shows quite flat profile in the bulk region in **Figure 10**. Due to the cooling effect of outer surface of the test wall, crust is formed adhere to the vessel, marked with black in the temperature field in Figure 13.

Since crust thickness is closely connected to the heat flux distribution, the bottom part forms the thickest crust and the thickness slowly decreases with the increasing of the polar angle, shown in Figure 12 and Figure 13.

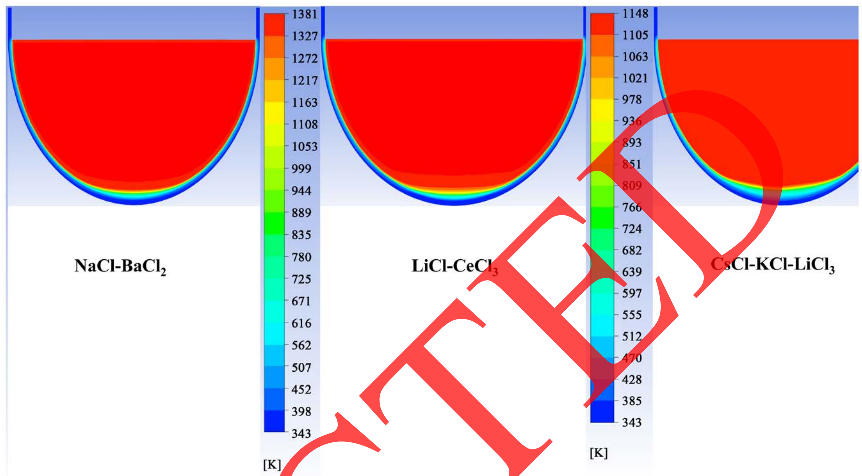

Figure 11. Temperature field.

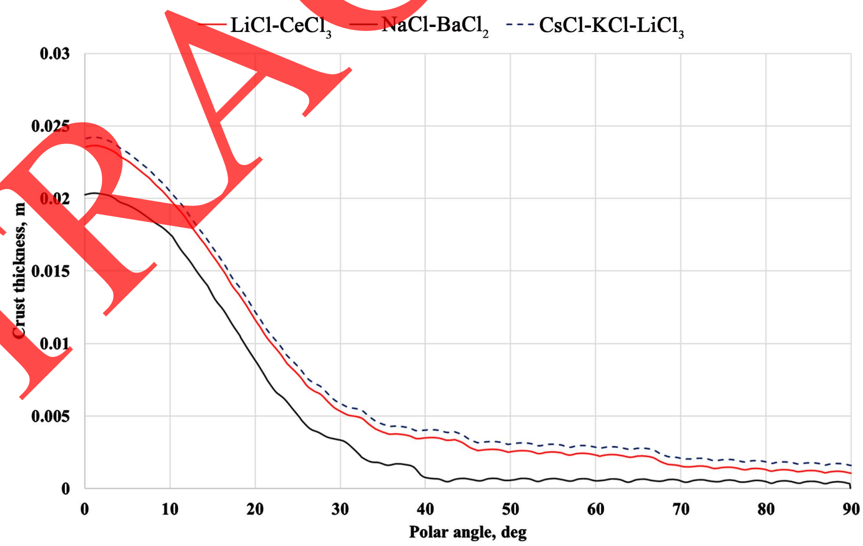

Figure 12. Crust layer thickness along vessel-debris interface.

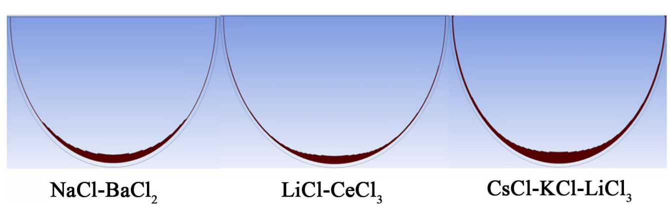

Figure 13. Crust formation due to the isothermal cooling effect along vessel outer surface.

After simulation is done, it is necessary to check the balance between the heat generation and heat loss through all the surfaces in the working domain to ensure that all the boundary conditions are properly set up. Here, the heat balance is shown for one case, as the others are similar.

Case: Single layer (Pool height 500 mm)

Total Heat generation Q = 1.5 MW/m^3*0.0351785 m^3 (volume of the working domain) = 52.76 KW

Heat loss (KW) though the surfaces = Heat Flux*Unit area (m^2)

Q_1 = Vessel outer = 189.4*0.19411544 = 36.765 KW, Q_2 = Vessel front = 0.7055*0.0267231 KW = 0.018853, Q_3 = Vessel back = 0.7055*0.0267231 = 0.018853 KW, Q_4 = Debris front = 19.15*0.293255 = 5.615833 KW, Q_5 = Debris back = 19.15*0.293255 KW = 5.615833, Q_6 = Metal front = 14.26*0.0993184 = 1.41628 KW, Q_7 = Metal back = 14.26*0.0993184 = 1.41628 KW, Q_8 = Metal top = 7.783*0.12 = 0.93396 KW.

Total heat loss through the surfaces = 52.72 KW = Total Heat Generation (52.76 KW)

6.2. Double Layer Configuration

Based on the benchmark test (comparison between PECM & FLUENT CFD) and application of PECM in single layer configuration, we have gained confidence in the application of PECM tool in the application of two-layer configuration. Same boundary conditions are applied in side, top and vessel outer wall (Figure 14) as single layer configuration.

It can be seen that both single and double layer configuration shows the similar trends of angular heat flux and temperature distribution. As described above, the angular heat flux increases along the polar angle, peaked about 80 degree (Figure 15). It is also found that the slope also increases as the angle becomes large. The

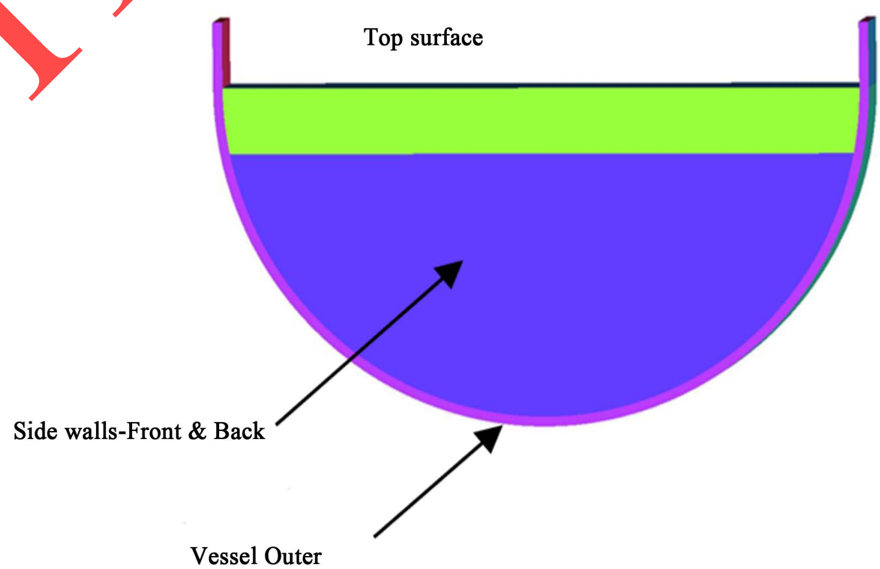

Figure 14. Side & top walls of two layer melt pool configuration.

non-uniform heat flux distribution is the result of natural convection due to volumetric heat generation in the fluid. The central vertical temperature distribution also shows quite flat profile in the bulk region in Figure 16. In the lower part of the cavity, the fluid is thermally stratified and flow field is almost steady (Figure 17).

Due to the heat losses through top surface, a drastic drop of temperature at around 0 m depth is observed in Figure 16 in cases with radiation. These results indicate that radiation may be the dominant heat loss for the top wall. As large amount of energy losses through side walls, it is necessary to consider some methods (like coating) to reduce the heat losses. The comparative data table between single and double layer is shown in Table 3.

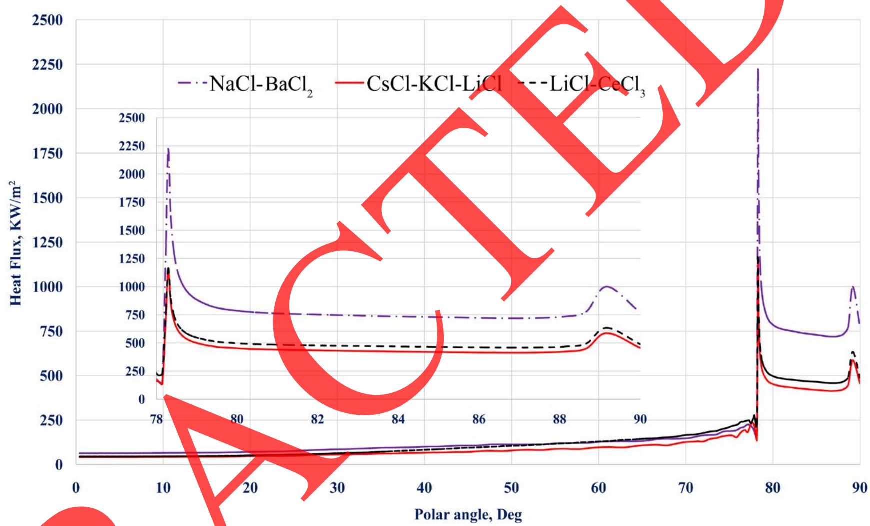

Figure 15. Angular heat flux distribution along the vessel.

Table 3. Comparative data table of 1-layer & 2-layer melt pool.

Parameters		Single layer (pool height 500 mm)			Double layer (pool height 400 mm)		
		$NaCl-BaCl_2$	$LiCl-CeCl_3$	CSCl-KCl-LiCl	$NaCl-BaCl_2$	$LiCl-CeCl_3$	CSCl-KCl-LiCl
Volumetric Power Q_V: MW/m^3		1	1	1	1	1	1
Oxide layer	$T_{average}$, K (molten part)	1585	1360	1345	1266	1154	1108
	T_{max}, K (molten part)	1614	1383	1365	1310	1178	1132
	$T_{top\ surface}$, K	1515	1350	1328	1014	1003	925
	$q_{average\_top}$, KW/m^2	45	29	26	195	174	145
	$q_{average\_side}$, KW/m^2	240	228	185	187	**162**	**129**
Metal layer	$T_{average}$, K (molten part)	-	-		1010	983	906
	T_{max}, K (molten part)	-	-		1022	991	925
	T_{top}, K	-	-		986	978	903
	$q_{average\_top}$, KW/m^2	-	-		8.07	7.78	5.6
	$q_{average\_side}$, KW/m^2	-	-		796	684	579

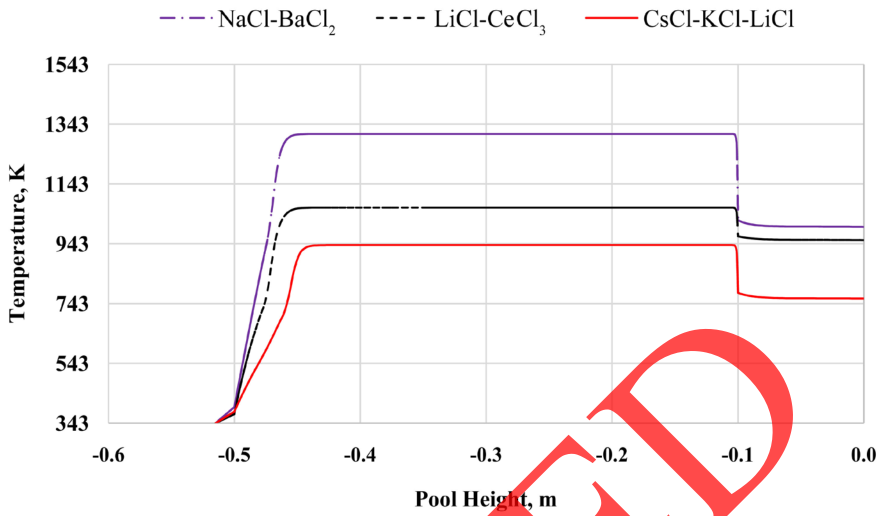

Figure 16. Temperature distribution along central vertical line in the pool.

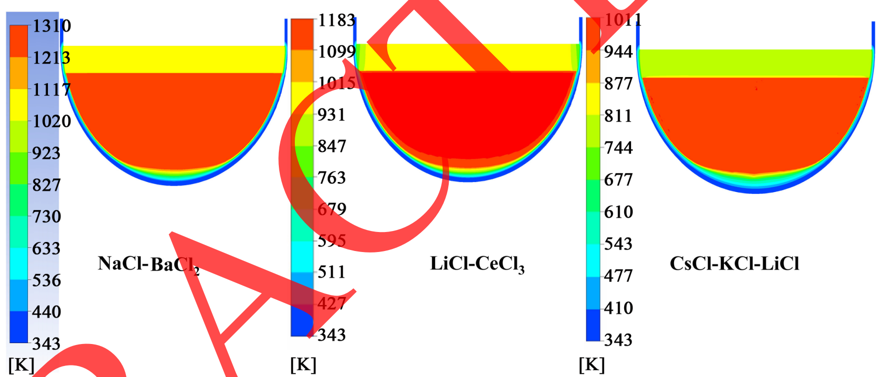

Figure 17. Temperature field of central cross section.

7. Conclusions

The aim of this paper was to analyze the physics of natural convection in internally heated fluid pools with different Pr numbers in isothermally bounded hemispherical 3-D vessel slice by the application of PECM model. Based on the numerical investigation, it was found that:

- The results of transient calculations reveal stable and thermally stratified lower part of the flow and much more dynamic upper layer that is dominated by time dependent transient phenomena.
- The fluid Pr number has small effects on the averaged Nu numbers in the convection-dominated regions. The decrease in the Pr number may cause a decrease in the Nu numbers on the top and sidewalls of cavities.
- In the conduction dominated regions (stably stratified bottom parts of enclosure), the effect of fluid Pr number on heat transfer is more significant and it grows with increasing Ra number.
- The heat flux is found lowest at the stagnation point and increases along the semicircular segment. The maximum heat flux is found at about 89° of the molten pool (one layer melt pool) and 80° degree (two-layer pool).

- The study suggests that the PECM is an adequate and effective tool to compute the effect of natural convection on melt pool thermal hydraulics.
- This work can be further extended to three-layer configuration;
- PECM could be applied to OPENFOAM, STAR CCM+ or CFD++ through some modification in the current PECM model and compared with the current result from ANSYS Fluent coupled with PECM.

Conflicts of Interest

The author declares no conflicts of interest regarding the publication of this paper.

References

[1] Theofanous, T.G., Liu, C., Angelini, S. and Salmassi, T. (1997) In-Vessel Coolability & Retention of a Core Melt. *International Journal of Nuclear Engineering & Design*, , 169, 1-48.

[2] Sehgal, B.R. (2012) Nuclear Safety in Light Water Reactors, Severe Accident Phenomenology. Academic Press, New York, 119.

[3] Nourgaliev, R.R., Dinh, T.N. and Sehgal, B.R. (1997) Effect of Fluid Prandtl Number on Heat Transfer Characteristics in Internally Heated Liquid Pools with Rayleigh Numbers up to 10^{12}. *Nuclear Engineering and Design*, **169**, 165-184. https://doi.org/10.1016/S0029-5493(96)01282-4

[4] Kulacki, F.A. and Goldstein, R.J. (1972) Thermal Convection in a Horizontal Fluid Layer with Uniform Volumetric Energy Sources. *Journal of Fluid Mechanics*, **55**, 271-287. https://doi.org/10.1017/S0022112072001855

[5] Kulacki, F.A. and Nagle, M.E. (1975) Natural Convection in a Horizontal Fluid Layer with Volumetric Energy Sources. *Journal of Heat Transfer*, **97**, 204-211. https://doi.org/10.1115/1.3450342

[6] Kulacki, F.A. and Emara, A.A. (1975) High Reynolds Number Convection in Enclosed Fluid Layers with Internal Heat Sources. US NRC Report NUREG-75/065.

[7] Asmolov, V. and Tsurikov, D. (2014) MASCA Project: Major Activities and Results. RRC Kurchatov Institute, Moscow.

[8] Asmolov, V. (1997) Latest Findings of RASPLAV Project. NSI Russian Research Center Kurchatov Institute, Moscow.

[9] Sehgal, B.R., Bui, V.A., Dinh, T.N., Green, J.A. and Kolb, G. (1998) SIMECO Experiments on In-Vessel Melt Pool Formation and Heat Transfer with and without a Metallic Layer. *Proceedings of In-Vessel Core Debris Retention and Coolability Workshop*, Garching, 3-6 March 1998, 205-213.

[10] Gaus-Liu, X., Miassoedov, A., Cron, T. and Wenz, T. (2010) In-Vessel Melt Pool Coolability Test-Description and Results of LIVE Experiments. *Nuclear Engineering & Design*, **240**, 3898-3903. https://doi.org/10.1016/j.nucengdes.2010.09.001

[11] Zhang, L., Zhang, Y.P., Zhao, B., Ma, W.M. and Zhou, Y.K. (2016) COPRA: A Large Scale Experiment on Natural Convection Heat Transfer in Corium Pools with Internal Heating. *Progress in Nuclear Energy*, **86**, 132-116. https://doi.org/10.1016/j.pnucene.2015.10.006

[12] ANSYS, Inc. (2016) ANSYS FLUENT Theory Guide. Release 16.2, Canonsburg.

[13] Tran, C.T. and Dinh, T.N. (2008) Application of the Phase-Change Effective Con-

vectivity Model to Analysis of Core Melt Pool Formation and Heat Transfer in a BWR Lower Head. *Proceedings of the Annual Meeting of the American Nuclear Society*, Anaheim, June 2008, 617-618.

[14] Tran, C.T. (2009) The Effective Convectivity Model for Simulation and Analysis of Melt Pool Heat Transfer in a Light Water Reactor Pressure Vessel Lower Head. PhD Thesis, Royal Institute of Technology, Stockholm.

[15] Steinberner, U. and Reineke, H.H. (1978) Turbulent Buoyance Convection Heat Transfer with Internal Heat Sources. *Proceedings of the Sixth Internal Heat Transfer Conference*, Vol. 2, Toronto, 305-310. https://doi.org/10.1615/IHTC6.3350

[16] Asfia, F.J., Frantz, B. and Dhir, V.K. (1996) Experimental Investigation of Natural Convection in Volumetrically Heated Spherical Segments. *Journal of Heat Transfer*, **118**, 31-37. https://doi.org/10.1115/1.2824063

[17] Asfia, F.J. and Dhir, V.K. (1996) An Experimental Study of Natural Convection in a Volumetrically Heated Spherical Pool Bounded on Top with a Rigid Wall. *Nuclear Engineering and Design*, **163**, 333-348. https://doi.org/10.1016/0029-5493(96)01215-0

[18] Asfia, F.J. (1995) Experimental Investigation of Natural Convection Heat Transfer in Volumetrically Heated Spherical Segments. PhD Thesis, University of California, Los Angeles.

[19] Dinh, T.N., Nigmatulin, B.I., Nourgaliev, R.R. and Rassokhin, N.G. (1996) Assessment of the Thermal Load When Molten Debris Interacts with a VVER Reactor Vessel during a Severe Accident. *Teploenergetika* (*Moscow*), No. 3, 9-17.

[20] Rassokhin, N.G., Mukhtarov, E.S., Granovskii, V.S., *et al.* (1996) On Simulating Physical Processes during the Interaction of the Core Melt with the Reactor Vessel. *Vestnik MEI* (*Moscow*), No. 3, 27-40.

[21] Greene, G.A., Hartnett, J.P., Irvine Jr., T.F. and Cho, Y.I. (2001) Heat Transfer in Nuclear Reactor Saf. Volume 29.

[22] Tran, C.T. and Dinh, T.N. (2009) The Effective Convectivity Model for Simulation of Melt Pool Heat Transfer in a Light Water Reactor Lower Head. Part II. Model Assessment and Application. *Progress in Nuclear Energy*, **51**, 860-871.

[23] Komlev, A., Konovalenko, A., Ma, W., Villanueva, W., Yu, P., Laato, T., Karbojian, A., Sköld, P. and Bechta, S. (2015) IVMR Living Document, Internal Material. KTH, Stockholm.

[24] ANSYS, Inc. (2017) ANSYS FLUENT Theory Guide. Release 17.1, Canonsburg.

[25] Kim, S.-H., Prk, H.-K. and Chung, B.-J. (2016) Mass Transfer Experiments for the Heat Load during In-Vessel Retention of the Core Melt. *Nuclear Engineering & Technology*, **48**, 906-914. https://doi.org/10.1016/j.net.2016.02.015

Nomenclature

Cp: Heat capacity [$J \cdot kg^{-1} \cdot K^{-1}$]

H: Height of the melt pool

Pr: Prandtl number (υ/α)

Ra: Rayleigh number ($g\,\beta H5Q/k\ \alpha\ \upsilon$)

Greek symbols

β: Thermal expansion coefficient

ρ: Density [$kg \cdot m^{-3}$]

α: Thermal diffusivity [$m^2 \cdot S^{-1}$]

υ: Kinetic viscosity [$m \cdot S^{-1}$]

g: Acceleration due to gravity force [$m \cdot s^{-2}$]

k: Heat conductivity [$Wm \cdot K^{-1}$]

Subscripts

B.C: Boundary condition

Ref: Reference