CFD SIMULATIONS OF MELT-VESSEL INTERACTION IN CONTEXT OF IVMR PROBLEM

A. Filippov\textsuperscript{1,2}, D. Kamenskay\textsuperscript{1,3}
\textsuperscript{1}: Nuclear Safety Institute of the Russian Academy of Sciences, B. Tulskaya 52, Moscow, Russia
\textsuperscript{2}: phil@ibrae.ac.ru, \textsuperscript{3}: kdd@ibrae.ac.ru

ABSTRACT
Possibilities of CFD to exact modeling melt-structure interaction during IVMR are rather limited. One of the main reasons is in "simulative" character of CFD modeling in this case: correct representation of the convective flow in the melt pool requires turbulence model validated on representative tests, which are rather poor in case of IVMR problem. Melting, melt stratification and other physics bring more problems. At the same time, CFD is able to indicate essential qualitative features and quantitative characteristics of the convective heat transfer. Actually, it is the only accessible tool for studying the melt pool in prototype scale. In the paper the examples are gathered demonstrating that CFD can do good predictions of the experimental results but also may give deflections in cases of too simplified problem statement.

The presented results deal with simulation of natural convection in externally cooled cylinder imitating upper metallic layer of the stratified melt pool in the lower head of pressurized water reactor. The problem is considered in 2D/3D statements with absent/presenting solid/fusible steel wall of the vessel. The two-layer melt pool is also considered for the cases of inverse stratification. The results demonstrate relatively unsteady and non-uniform distribution of the boundary heat flux in case of simplest 2D "rectangle" configuration. The obtained local singularities of heat flux were interpreted as artefacts decreasing after addition of contacting lateral massive wall. The influence of the melting of this contacting wall on the outer heat flux distribution along the external cold boundary was also studied. Wall melting was found quite non-uniform, and, besides, the finite thickness of the wall may essentially decrease the display of focusing effect on the external cooled wall surface. All that shows that ablating vessel wall should be considered as the essential component of the melt pool, affecting the flow and heat exchange. The flow pattern obtained in 2D calculations looks quite smoothed but the integrated heat transfer was more or less same in 2D/3D statements. The estimated mesh size for correct 3D modeling of stratified melt pool contacting with melting boundary was found close to ~100 million cells, and, to accelerate calculations, 2D axisymmetric approach may be used for preliminary analysis.

KEYWORDS
Severe accident, IVMR, stratified melt pool, massive wall, CFD

1. INTRODUCTION
Possibilities of CFD to exact solution of the problem of melt-structure interaction are rather limited. One of the main reasons is in complex and relatively uncertain melt composition and boundary conditions. Another reason is in "simulative" character of CFD modeling in this case: due to intensive turbulence, correct representation of the convective flow in the melt pool requires turbulence model, which needs in its own validation. Thus the role of CFD models as surrogate of the experiment is limited, and in this sense CFD models may be comparable with very crude but quite more effective correlation models.
From the other hand, no doubt that CFD is able to indicate essential qualitative features of the convective flow and melt boundary heat flux distribution. And, it is more or less clear that the experiments with a melt pool being prototype as by melt composition (stratified melt) as by hydrodynamics (high Rayleigh numbers) are practically impossible. Actually, CFD is the only accessible tool for studying the melt pool of prototype size although the obtained results need in attentive interpretation and checking. In this work some numerical examples are presented, which demonstrate some essential qualitative features of convective flow and heat transfer in the melt pool of some typical configurations. Most of the discussed questions are known but they generate the questions about heat transfer in IVMR that need in clarification.

In what follows brief consideration of problem statement is provided first. Then the CFD modeling options are outlined: the ANSYS FLUENT code (version 16.2) is used for simulation of the melt natural convection. Validation on hemispherical pool tests (slice geometry) for moderate Rayleigh numbers is discussed. Then cylindrical metallic melt pool imitating upper metallic layer of the stratified melt pool on the lower head is modeled in different ways including 2D/3D statement with absent/presenting steel wall which may be fusible. The two-layer melt pool is considered for the case of inverse stratification, which is not yet sufficiently investigated to this time.

The influence of massive wall on the outer heat flux distribution on the external cooled boundary is considered in more details. Actually the finite thickness of the heat conducting vessel wall may essentially decrease the display of focusing effect for the external HF in cases when the metallic layer thickness is less than the vessel initial thickness. This demonstrates that ablating vessel wall is the essential component of the whole system, affecting all flow and heat exchange.

2. PROBLEM STATEMENT

2.1. Physical phenomena under consideration

The main physical factors having significant effect on steady maximum of side heat flux distribution are: 1° melt separation on two immiscible phases and resulting melt stratification, and, likely – 2° melting/dilution of the vessel steel by hot liquid i.e. vessel ablation. First item is more or less investigated although in the main – numerically. The experimental investigations of the second effect are not known to the authors. Some qualitative numerical results are discussed below.

We will consider the stratified melt in the lower head of PWR (VVER) type reactor or close configuration. If we fix decay heat power, which depends mainly on time and NPP power then boundary heat flux distribution (melt-vessel boundary) will depend mostly on hydrodynamics of melt convection and material properties of melt phases defining melt phase boundaries through the vessel wall melting and crust formation.
The normally stratified melt pool is reduced to two subsystems: heat generating liquid having hemispherical/torospherical lower boundary (oxide layer) and cylindrical volume (metallic phase) heated from below. These two simple systems will be considered in what follows. Effect of vessel wall will be examined from the viewpoint of its diffusing influence on local peaks of heat flux. The scope of physical phenomena to be taken into account is then limited by turbulent natural convection in the melt pool with boundary conditions of different kinds and melting (dissolving) of the wall boundary that is considered using some simple assumption about melting process.

Even such quite simplified problem statement adapted for CFD capabilities is very difficult for complete solution. The results show some irregularities, which are partially the effect of simplifications. This is briefly discussed below along with specification of the used CFD approaches.

2.2. Simplifications of the flow models and used CFD options

Convective flow in the reactor melt pool is highly turbulent ("hardly turbulent") and thus is quite unsteady. This implies: (1) Using of some kind turbulence model is inevitably. (2) The quasisteady state may be considered only through unsteady calculations with time averaging and analysis of perturbations. (3) Only 3D simulations may be considered as reliable problem statement.

Two latest items make calculations quite expensive because of required long relaxation time (~1 hour) and indefinably high spatial resolution required by turbulence models. The first item introduces a large piece of uncertainties since only RANS models may be actually enabled with guarantied applicability. Application of LES type models does not provide guaranteed accuracy despite of better resolution of flow details. In general, the problem statements may be subdivided on two large classes:

- "indefinably inaccurate" but rather effective simulations operating with more or less investigated but known as inexact methods like RANS models in 2D-3D geometry;
- "indefinably accurate" and quite ineffective methods that in the first place may be referred to LES type models in 3D statement with their increasing spatial resolution but not fully clear applicability.

These characterizations may be shortly explained as follows.
RANS models are based on averaging procedure related to the whole flow region (here – the melt layer), without differentiating flow behaviour by spatial/energy scales. Most of these models introduce eddy viscosity (or its equivalent), which does not explicitly depend on spatial scale of the flow and thus does not contain the intrinsic requirements to mesh resolution for representations of definite flow peculiarities. Some refinements in the $k$-$\omega$ SAS model [1] used in this paper are not discussed here. If, working with a RANS model we would take more and more fine meshing, then no new flow structures will appear from some time. In this sense RANS results are close to the laminar regimes (see pictures given below). In short, all turbulent effects in RANS models are enclosed in the eddy viscosity and the involved turbulent equivalents of the Prandtl and Smidt numbers introducing effective heat conduction and diffusion. This is convenient for simulations but requires comprehensive validation, in principle, for every class of problem and every range of flow parameters in this class. The quantitative result may turn out to be unsatisfactory.

LES models are based, by definition, on differentiating of large-scale and small-scale turbulence. The latest one is isotropic and may be modeled in a way analogous to that of RANS – using a kind of eddy viscosity. The "large eddies" are to be modeled directly, without averaging, and the mesh resolution and time step should satisfy corresponding conditions. There are no general criteria of the adequate mesh resolution, and usually this requires the same BPG investigations as that for RANS models.

One of possible problem of modeling hardly turbulent natural convection may be in the likely existence of secondary structures i.e. a kind of "middle-scale turbulence", which is discussed in particular for the Rayleigh-Benard type convection [2]. This may violate applicability of LES ideology at all. Another very serious problem of LES is in high requirements to computer resources. The mesh size for LES is estimated as 5% of that for DNS (all mentioned estimates are taken from [3] and [4]). Taking into account the known estimate for mesh size for DNS calculations: N~Re$^{9/4}$ and evaluating effective Reynolds number for the Rayleigh-Benard convection according to [2] one can obtain that correct LES calculations for the reactor melt pool would require the grids of N~$10^9$ cells or more. And - correspondingly small time step of the unsteady relaxation calculations to (quasi)steady state.

And, to finish this short overview, we mention that near-wall treatment being of primary importance in modeling of natural convection in closed volume, will require same mesh size for boundary layer only [3].

Of course, the LES type models are often used with not very fine meshing. But their integrated predictions may be even worse than that of RANS. Some examples in case of hydrogen safety CFD benchmark with 19 participants may be found in [5]. All that defines the RANS models as the only "realizable" class for CFD analysis of the reactor melt pool natural convection. The calculation examples considered in this work follow this thesis: they use RANS and were performed in 2D statement except some 3D calculations showing some limitations of 2D approaches. Large simplifications of pool configuration and some other were introduced for the simple configuration of Fig. 1. The reasons of presenting such simplified investigations are following:

- In cases of moderate Rayleigh numbers (Ra<10$^{14}$) 2D RANS calculations show quite satisfactory quantitative agreement with experimental data. For high Ra (Ra~$10^{15}$-$10^{16}$) the agreement is only qualitative that may show some limitations in the area of high Rayleigh convection;
- Some observed features may be considered as artefacts of simplified formulation, which eliminates after some improvement of the problem statement;
- There are some large and quite understandable deflections of obtained CFD results from predictions simple correlation models that requires examination.

The used CFD modeling approaches are based on those implemented in the ANSYS FLUENT code [6]. The optimal set of models/options includes following:

- coupled solver (here – coupling of flow equations);
- SAS turbulence model [1] or $k$-$\varepsilon$ model in case of large $y^+$ (>20-30).
- second order spatial discretization;
- built-in FLUENT model of solidification/melting (porous media approach for representation of mushy-zone).

Note that there is some uncertainty in near-wall treatment in case of melting wall: current position of the flow boundary changes that is not taken into account in wall functions of turbulence models. In that sense modeling of solidifying flow near the cold wall will be "laminar" anyway. But, possibly, it is not so far from reality. In FLUENT, phase transfer is modeled through temperature depended sinks of heat and momentum acting within the mushy zone temperature range. It is treated as a kind of latent heat and effective viscosity existing in this temperature interval characterizing near-wall region.

3. HOMOGENEOUS SPHERICAL POOL (SLICE) WITH VOLUMETRIC HEAT SOURCE: VALIDATION ON RASPLAV–SALT EXPERIMENTS

RASPLAV–SALT experiments were performed within an OECD RASPLAV Project [7] in a course of choice of type of heater in large-scale AW-200 series with ~200 kg of molten corium. The SALT tests were performed in a long series, with different salt compositions, in different regimes including crust formation, two types of heaters, varied boundary conditions. The obtained results take independent interest. The series of more than 40 instants was carried out for $R_a$ range $2.7 \cdot 10^{14} \leq R_a \leq 1.6 \cdot 10^{15}$.

In the considered cases the liquid salt NaF-NaBF₄ was a corium stimulant. The test facility had a "slice" geometry (Fig.2). The melt cavity was bounded by flat insulating walls and roundish $R = 0.2$ m steel wall of 0.02 m thickness which was cooled externally. The melt was heated by two lateral wall heaters near the flat boundaries (SWH series) or by Ohmic heat sources (DEH series).

![Figure 2. Slice geometry of RASPLAV Salt tests](image)

The two presented tests differed by absence/presence of solid crust on the cooled wall. They were modeled in 2D flat geometry, in two configurations (Fig. 3), in which steel wall subregion was modeled either as boundary condition ("No vessel") or explicitly ("With vessel"). Experimentally measured external sidewall temperature was used as the outer temperature boundary condition. In all simulations the heaters were modeled as an equivalent volumetric heat source. Here the used turbulence model was $k-\varepsilon$. 

3.1. Heat transfer under no crust conditions

Some results for Salt test 29 (without crust) are shown in Fig. 4. The measured/calculated temperature profiles on the wall boundaries are plotted vs. the angle $\theta R$ normalized on inner radius $R$ of steel wall. Temperature distribution was close to that in the test 30 (with crust formation) considered below. The difference in calculation domains (without/with steel wall) resulted in the difference in temperature profiles. For "No vessel" configuration the calculated temperature profile has a maximum in the top of the wall (black line in Fig. 4). Addition of the steel wall makes temperature distribution more realistic (right line in Fig. 4). This feature will appear below also in another configuration.

The integrated heat transfer through the cooled steel wall was studied in serial computations using "no vessel" grid. The upper boundary of the pool was taken adiabatic that approximately corresponds to the estimated heat losses from the upper boundary at experiments (not more than 15%). The Rayleigh number
was varied by changing the gravitation acceleration \( g \). In the cases of small \( Ra \), the turbulence model was not used. The obtained dependencies \( Nu(Ra) \) for laminar (Laminar) and turbulent (Turbulent) simulations are depicted in Fig. 5 as "1" and "2". These two curves were approximated by power low correlations (curves 3 and 5). The experimentally obtained correlation is plotted as line 4.

![Figure 5: Integrated heat transfer in the RASPLAV SALT tests](image)

3.2. Heat transfer with crust conditions

In calculations with crust formation the same effect of steel wall on the upper part of temperature profile was observed (Fig. 6-left). 2D temperature field given in Fig. 6-right shows picture of thermal stratification typical for laminar flow that is smoothing effect of turbulence model. Modeling of crust formation is in qualitative and reasonable quantitative agreement with the integrated results given in Fig. 7.

![Figure 6. Test 30 with crust. Temperature profiles and general view of steady temperature](image)
The given results demonstrate applicability of CFD to simulation convective heat transfer in heat generating liquid including the cases of crust formation. Note that in simulations of tests with higher Rayleigh numbers close to realistic values, discrepancies of CFD predictions were higher [8].

4. STRATIFIED MELT POOL

In what follows, calculation of focusing effect in different treatments is considered. After that the flow in inverse stratified pool with the cylindrical oxide layer atop the metal layer is briefly discussed.

4.1. "Classical" focusing effect

As it was mentioned above, heat transfer in the upper metallic layer of stratified melt pool may be modeled separately from the lower oxide layer if the appropriate input heat flux is specified on its lower boundary. Additional simplification is in treatment of side vessel wall as an isothermal solid boundary.

The problem statement in such a very simplified form is depicted in Fig. 8-left. The cylindrical volume of height $h$ and radius $R$ corresponds to upper part of stratified melt depicted in Fig.1. The dependence of steady sidewall heat flux on the aspect ratio for fixed value of input heat flux $F$ is to be determined. One of the first analytical models was described in [9].

Figure 7: Crust profile along steel wall boundary in SALT test 30

Figure 8: Metallic layer problem statement (left) and sidewall heat flux vs. aspect ratio $h/R$
The CFD calculation and comparison with analytical model was performed in [10] in conjunction with the problem of modeling normal stratification of the core melt in the VVER-1000 vessel. Calculation domain was steel cylinder of $R=2.07$ m and height $h=(0.1-0.5)R$. The thermal boundary conditions were set as in [9]: no volumetric sources, constant flux BC on the lower boundary – $F=400$ kW/m$^2$, temperature BC with constant temperature $T_{LQ}$ on lateral (sidewall) boundary, radiation BC on upper boundary:

$$F_{rad} = \varepsilon \sigma (T^4 - T_b^4)$$

(1)

where: $T_b=0$ and $\varepsilon=0.45$. The velocity BC are: "no slip" – on lower and lateral boundary, "slip" – on upper boundary (that didn't play essential role). 2D CFD simulations were carried out using orthogonal grid of 20,000 cells with the $k$-$\varepsilon$ turbulence model.

The dependencies of sidewall heat flux $F_{id}$ vs. aspect ratio $h/R$ are depicted in Fig. 8-right. Obtained good agreement with the semi-analytical solution likely originates from high effective heat conductivity in the bulk of fluid, which is assumed in the analytical models of integrated heat transfer and introduced in the turbulence model.

4.2. Perturbations and smoothing of HF distribution: 3D and contacting wall

The reliable CFD simulations in case of high Rayleigh numbers may be performed only as unsteady. The flow and boundary heat fluxes will oscillate that corresponds to reality. Thus, along with averaged quasi-steady values of HF, its fluctuations should also be considered. In what follows unsteady distribution of HF is discussed by the previous example which was slightly modified: in Eq. (1) $T_b=300$ K and $\varepsilon=0.4$. Size of 2D mesh was about 46k cells. The plots are related to the cases of the layer heights $h=0.2$ m and $h=0.4$ m ($h/R \approx 0.1$ and $h/R \approx 0.2$). In such statement the problem was also considered in 3D: ¼ of whole circle was taken for CPU saving. The mesh had 4,420k cells. Cases with $h=0.2$ m and $h=0.4$ m were calculated for the same input heat flux $F=400$ kW/m$^2$ ($=5.54$ MW of total heat transfer from oxide to metallic layer).

General views of flow velocity for 2D cases with $h=0.2$, 0.4 m are given in Figs. 9-10. One may see rather smoothed flow pattern that is effect of turbulence model (here – $k$-$\omega$ SAS).

![Figure 9. Flow velocity in 2D: h=0.2 m](image1)

![Figure 10. Flow velocity in 2D: h=0.4 m](image2)

Heat flux profiles are plotted along the cooled sidewall, bottom-up. There are two ones: time averaged flux (labeled as "A") and instantaneous value for definite time (labeled as "I"). The significant difference between them appears in the case $h=0.4$ m (see Figs.11-12). It was also obtained for the case $h=0.6$ m in 2D. This may be treated as partial "defrosting" of convective flow structure smoothed by eddy viscosity as it is illustrated by Figs. 9 and 10.
The obtained results demonstrate non-uniform distribution of the boundary heat flux, humps and singularities on the ends of the curves. This may be interpreted as effect of simplified configuration – there is no contact with vessel wall which plays a role of dampener. Close picture was obtained in above in SALT calculations although with smaller peaks. To see the possible effect, the infusible cooled steel wall contacting with the melt was joined to the melt region as depicted in Fig. 13-left. Temperature distribution and unsteady heat flux profiles taken on the outer (labeled as "O") and inner wall sides are given in Figs.13-14. The middle part of HF profile became flat that is the consequence of non-melting wall. But the humps on the ends became smaller although the point singularities in the corners remained. They are considered as the result of simplified near-wall treatment. Heat flux in the outer boundary is quite smooth.

Another line of making calculation more realistic is transfer to 3D geometry. The general view of solution may be seen in Figs. 15-17 with surface distributions – on down, symmetry and lateral (sidewall) boundaries. The vertical heat flux profiles along the cooled boundary were taken for three equidistant angular directions within the considered 90° angle: 22.5, 45, and 67.5 degrees (see Fig. 18; the lines may be seen in Figs. 15 and 17).
Figure 15. Instantaneous flow velocity in 3D calculation

Figure 16. Instantaneous temperature in 3D calculation

Figure 17. Instantaneous side heat flux in 3D calculation
These results may be commented as follows:

- The obtained surface distributions (the values were taken in the centres of boundary cells) are irregular and fluctuating, and look quite "realistic" although it would be rather difficult to obtain such fluctuating picture in experiments;
- Averaged (and integrated) values of side heat flux are close in 2D and 3D calculations;
- Angular nonuniformity of heat flux distribution may be seen in 3D: there is a difference between time averaged HF profiles – those taken for angles 22.5° and 67.5 degrees are the same and differ from 45 degrees profile. This likely is an effect of ¼ formulation instead of taking full circle.

The obtained small difference between 2D and 3D (Fig. 11) allows consider 2D as a "draft" tool. To check the angular effect, the 3D calculations should be performed for full circle and with joined vessel wall.

### 4.3. "Melting" focusing effect: fusible wall

Damping effect of steel wall on external heat flux distribution was investigated in conjunction with its melting. The results (2D) were reported in [11].
Fig. 19 shows calculation domain. It includes metallic melt joined with the wall. Boundary conditions applied on wall inner and outer surfaces model effect of oxide melt and external cooling. To demonstrate the effect of transient "inertial" wall melting and spatial nonuniformity on outside heat flux, the problem was solved as unsteady, with small initial temperature of the wall. The layer thickness was \( d = h = 0.1 \) m or \( d = h = 0.4 \) m, the value of input heat flux was the same, \( F_{in} = 400 \text{ W/m}^2 \).

**Figure 21. Steady and transient (max) heat flux profiles in calculations with \( h = 0.1 \) m and \( h = 0.4 \) m**

In Fig. 20 molten fraction distribution may be seen for the two times and two layer thicknesses. The heat flux profiles along the cooled side are plotted in Fig. 21 with positive sign, and the show the following:

- As transient as steady heat flux profiles decrease up-bottom that is the effect of increasing wall thickness formed by melting. The effect of cooling the flow by the wall which may be seen in Figs. 11-12 is amplified here;
- Rather small display of focusing effect displayed on the external boundary compared with Fig. 8;
- Large value of peak transient HF caused by initial heat supply of the melt. This is essentially the effect of nonuniform and transient wall melting.

### 4.4. Some features of melt convection in case of inverse stratification

Inverse stratification of the melt during IVMR is possible in case of large amount of uranium in the metallic phase resulted from small oxidation of zirconium and relatively small amount of steel [12]. More or less stable inverse stratification is less probable than normal stratification, and this configuration was investigated in small extent. Actually the authors do not know experiments imitating lower metallic layer. The experiments with separate cylindrical heat generating pool were mainly dedicated to other problems. The pool properties discussed here were obtained numerically, in CFD calculations of model problems for in-vessel and ex-vessel (core catcher) melt retention.

Calculation example for VVER RPV taken from [13] demonstrates the flow pattern given in left side of Fig.22. Weak convection in the metallic layer results in the local peak of side heat flux which may be seen in Fig.22-right, in which two HF profiles corresponds to different models of oxide-metal boundary.
Other numerical investigations show a qualitative picture of heat and mass transfer presented in Fig. 23. It arises as follows. 85-90% of decay heat generation is concentrated in the upper oxide layer. Oxide melt temperature reaches its maximum in some middle level. The flow above the level of this maximum is of Rayleigh-Benard type. The flow below the maximum should be close to laminar because of stable temperature stratification: its height gradient is positive. The same should be in the upper part of the lower metal layer, which is heated from above by the oxide layer through the crust.

Inverse stratification is one of the key features of melt retention in crucible type VVER core catcher, and it was considered as a model example. Melt retention and stabilization in the crucible type core catcher (CC) for new VVERs is analogous to the in-vessel strategy. The core melt from a damaged reactor runs into the CC vessel of larger diameter supplied with a large amount of sacrificial material [15]. As a result, melt pool is formed within the CC vessel. One of the main differences from the case of IVMR is in the size and shape of the vessel, which is close to the cylinder. Another difference is in the melt layers compositions and relative positions – it is guaranteed to be inverse, with the light oxides layer atop the steel layer. The thicknesses of the oxide and metallic layers are essentially larger than the thickness of the oxide layer alone, which altogether results in large CHF margin.
The main parameter of successful melt retention in the crucible type CC and reactor vessel is the same – it is the side heat flux transferred to the external coolant. In case of core catcher the flux maximum is smaller due to dilution of the melt by sacrificial material, which also mitigates thermal shock, possible in case of immediate heat contact of the melt with the CC vessel. CC structure practically excludes this, and to the time of contact with vessel, the melt pool in the core catcher should be almost steady. CFD model of the CC melt pool is considered below as it demonstrates some typical features of convective heat transfer in inversely stratified molten pool.

The 2D problem statement is clear from Fig. 24. CC vessel is schematized by steel cylinder of 50 mm thickness. The role of lower head is played by the lowest layer of solid crust. The metallic layer is above this crust, and oxide layer lies atop the metal. Decay heat power (20 MW) is distributed in oxide-metallic melts as 9:1. Third kind boundary condition modeling external cooling is applied on lateral vessel boundary. Radiation boundary condition (1) was specified on the upper side of the oxide layer. Vessel melting was not introduced because of its insufficiently high temperature.

Some results are given in Figs. 24-25. Temperature distribution in Fig. 24 is qualitatively close to that in Fig. 23: stable stratification in lower part of the oxide pool, unstable flow in the upper part, and some
circulation in the metallic layer. Heat flux profiles in Fig.25 demonstrate the damping, which is analogous to that discussed above for melt layer calculations. Due to large thickness of the oxide layer and unfused cooled wall, HF is small enough and HF profile is relatively uniform. Note that relatively large part of decay heat runs into the metal layer but heat flux decreases with height very quickly to small value.

5. MESH SIZE AND COMPUTER CAPABILITIES

The pool parametres in the cases of convection in metallic layer considered above are close to typical ones corresponding to Rayleigh numbers Ra~10^9. Mesh sizes in 3D and 2D cases were 4420k and 44k cells. Number of cells in vertical section of 3D domain was about 40k i.e. close to that in 2D. One of the flow parametres defining application area of definite turbulence model to definite meshing is wall y^+ in near-wall cells. Its profile along cooled side in case of metal layer of h =0.2 m without wall is plotted in Fig.26.

![Figure 26. Wall y^+ in 2D metal layer: lateral boundary](image)

Typical quantity in the presented 2D calculations was y^+ =10-20 that is intermediate value between its "viscous" (0-10) and "turbulent" (>20) ranges. For SAS, which is k-ω type model, recommended near-wall meshing should be 3-5 times finer. Corresponding mesh size for correct application of this turbulence model in 3D calculations, even for ¼ geometry is estimated as some tens of millions - for metallic layer only. Typical time step in presented calculations was 0.1-0.2 s, and it would be smaller for finer meshing. Time interval for relaxation to quasi-steady state is one hour. Addition of wall melting may essentially increase this time. In whole, such CFD calculation may be performed in modern multicore computers during several weeks. From the viewpoint of BPG requirements, such problem statement would be correct but rather expensive in case of multivariate analysis.

6. CONCLUSIONS

Convective heat transfer in the single-phase melt pool is modeled more or less well for moderate Rayleigh numbers. But in cases of realistic Ra and Ra_i numbers related to stratified melt pool the errors cannot be fully controlled because of a lack of experimental data. The sources of uncertainties are mainly in turbulence modeling and in large simplifications usually done in CFD analyses. The presented examples show the effects of nonstationarity of the flow, of simplified boundaries (corners of calculation domain), same effect of two melt layers contacting with the cooled wall, and effects of vessel wall melting.

More or less realistic set of conditions for CFD modeling heat transfer in stratified melt pool would:
- be unsteady 3D with he mesh satisfying conditions of turbulent models applicability;
- base on calculation domain including two melt layers plus RPV of 360° circle;
- take into account melting and solidification.
Further complications may include free boundary of the melt phases using the models like volume of fluid (VOF), modeling of dilution, careful analysis of wall unsteady melting, and possibilities and ways of its solidification. Simplifications may be introduced for model problems demonstrating possible effects (as in this paper) or, for economy, for surrogate modeling (2D axisymmetric) used in multivariate calculations.

Considered CFD examples show that some idealized treatments of analytical results like focusing effect may appear in essentially smaller extent than it is often supposed in correlation models and their consequences.

Extreme physical conditions and large sizes of the melt pool on the RPV bottom make its experimental investigation in fully prototype conditions practically impossible. From the viewpoint of obtaining qualitative picture and estimations of possible quantitative effects, CFD analysis enabled with appropriate physics and experimental data may be rather effective tool of investigations of severe accident phenomena. But this requires careful examination of prototypity of the analysis.

NOMENCLATURE

BC – Boundary conditions
BPG – Best Practice Guides
DNS – Direct Numerical Simulation
HF – Heat flux
LES – Large Eddy Simulation
RANS – Reynolds Averaged Navier-Stokes
RPV – Reactor Pressure Vessel

REFERENCES

11. Strizhov V.F., Filippov A.S. Progress in IVMR modeling since RASPLAV and MASCA: lessons and challenges. IVR seminar 2016, June 6-7, Aix-en-Provence (France)