Dear reviewers, thank you for your time and for the accurate revisions.

The paper was now updated following your instructions, we are sure the current version better describes the performed work and we hope it can be accepted for the Technical Meeting. Please find further explanations in the following: after each reviewer’s comment, our answer is reported in red.

Reviewer 1: The article is well written but clearly shows first steps in a longer process. Since the authors point this out clearly, this is OK to me. Some minor comments:

Thank you for this comment, it is indeed a long process and we are happy we will soon have the chance to compare our results with the incoming experimental results.

Figure 9: What does P4 mean in the legend? It would be more clear if this was called ANSYS FLUENT.

Thank you. Caption is now correct.

Section 4.1: Figure 10 shows large difference between STH and CFD. Could a potential reason be that the CFD code employs a turbulence model and fine mesh resulting in a shear layer which diffuses radially and dampens the jet exiting the heat exchanger, while in the STH code, there is hardly any friction and damping, and the jet protrudes much lower. I think a similar reasoning is provided later by the authors in section 4.2, but it might be good to also explain this here.

Thank you. The following explanation was added in the updated version.   
“This difference is probably due to the lack, in the STH code, of shear forces that enhance the radial diffusion of the fluid.”

Figure 11: REAP5-3D should be RELAP5-3D

Thank you. Caption is now correct.

Reviewer 2: The paper presents simulation results of the CIRCE facility incorporating the new THETIS heat exchanger obtained using an STH code (RELAP5-3D) and a CFD code (ANSYS Fluent), and then proceeds to compare the results, both for a steady-state and a transient. Though for the transient, CFD results are only obtained for the final steady-state at the end of the transient. The topic of itself is interesting for a paper and useful for future reference work, but the quality of the paper must be significantly improved in order to accept it as a full paper. The major improvements needed are:

My main concern is the validity of the results. STH and CFD simulations are performed for a steady-state. I understand these are pre-test studies, but no sensitivity studies on the used parameters or nodalization is presented. With the results being inadequate (at least the STH results, according to the author himself), and not matching, this is definitely needed. Especially because assumptions were made that later on turned out may be incorrect, such as no axial heat conduction and environmental heat loss in the STH code, or the used boundary conditions in the CFD codes. Without results matching, or without being able to compare to experimental results, what then is the merit of these simulations. It is said they are done to determine where to place the measurement devices, but can any conclusions be drawn based on this? Or what is the plan with the obtained results. Because in the last paragraph of the conclusion, it is said that the obtained results will help the experimentalist with positioning of instrumentation and the definition of the experimental campaign. How is that?

Thank you for this comment, the goal of these studies is now better explained. As reported in the paper, this is a preliminary study having the main goal of investigating the thermal-hydraulics behaviour of the CIRCE-pool and evaluating the code’s capabilities.

These preliminary results suggest that RELAP5-3D reporting very limited capabilities in dealing with viscous forces occurring in the pool, almost neglecting LBE mixing. CFD was thus selected to evaluate the 3D behaviour of the pool. As added in the article, this preliminary study, underline the absence of strong thermal stratification as in the CIRCE-HERO configuration and point out the need to change the fitting volume design, from single wall to double isolated wall, to limit thermal losses from the loop towards the pool. The CFD model was recently updated, eliminating the boundary conditions on the loop side, simulating the internal part of the loop providing similar results. This step will be done even for the heat losses towards the external environment, we are confident the results will not change relevantly, supporting the conclusions reported in the present paper.

The introduction needs more references to related research, not just to those of CIRCE. For example, in the first paragraph some statements are made of more work needed on modeling liquid-metal TH. Why? Where is that based on. Also, it’s said many EU projects have started. Which one? And for what?

Thank you. Additional References referring to experimental facilities and EU projects were included.

•The abstract reads like an introduction, no results or conclusions are given. So please add some statements on what was achieved.

Thank you for the comment. Some information about the conclusions is now reported.

“The obtained results provide interesting suggestions for the experimentalists and represent a valuable support in better setting the experimental conditions and measurements tools layout, e.g. the need to isolate the fitting volume preventing excessive heat losses from the loop towards the pool. In the frame of future works, further analyses will be performed also trying to develop coupled STH/CFD application. “

For the CFD simulation, the main reason to use the settings and mesh that is used is that it worked previously for other CIRCE cases. However, now clearly the results are inconclusive. So then one should go back questioning if the approach is actually good. Or justify better why the used approach is valid or show some sensitivity studies.

The reviewer is correct. We lack comparisons with experimental data but, as a starting point, adopting “the same philosophy” of previous works which provided good results seems the best way. Independency of the mesh was also performed but not reported. At the present stage of the work, we were asked to mainly focus on the postulated operating conditions in order to achieve a preliminary estimation of the temperature distribution. No sensitivity analyses were thus performed for the CFD calculations, on the other hand we investigated several different possible operating conditions, which should allow for a sufficient understanding of the involved phenomena. Concerning the validation, the only available (estimated) experimental result is the heat which is transferred towards the environment when the pool is at an average temperature of 400°C. The CFD prediction matches the experimentalists estimations. We hope that with the beginning of the experimental campaign we will be able to further validate our approach.

STH results are used as input for the CFD simulations. However, the STH results are presented as being inadequate. How then can you use these still as input for your CFD simulation? Please justify this approach?

As reported above, these are preliminary studies. This had to be done as a first step, lacking real data coming from the experimental campaign to obtain a preliminary guess of the pool behaviour. According to our previous experiences in this facility, the adopted approach provides sufficiently good results. The ongoing calculations also including the internal loop in the CFD domain seem qualitatively confirming the results reported in the present work

Paragraph 3.2 on boundary conditions used in the CFD simulations is lacking in explanations. Numbers are presented, but hardly explained or shown why they are correct and where used precisely. Please explain better where the 3500 W/m^K comes from, and where it is used. And why a convective boundary condition is used for the FPS and FV walls. Also, what about the LBE entering the pool, at what temperature is it and why. Finally, what is the heated region alluded to? Where in particular is this?

Thank you, justification is now added in the paper. The temperature of the LBE entering is in accordance with what ENEA has planned for the experimental campaign. The heated region refers to the FPS region, where the electrical heater is placed.

Some other more minor comments:

Please capitalize Martelli in the author list, similar to the other names.  
In abstract: Gen IV reactor technologies, not reactorS.

In abstract: “recent activation of the new EU” reads kind of weird. Just write “The recently started EU..” or something similar. And also, it’s “provides” and not “provided”

In abstract: In the third line, it’s referred to “involved phenomena”. What phenomena are meant here, please specify. Also in the first sentence of the introduction.

In abstract, fifth line: please delete “well-known”.

In abstract: Please split the sentence on “The analyses mainly…PLOFA scenario”. It’s two separate sentences, and shouldn’t include a ;.

Thank you, all the above comments were accepted. The paper was updated as suggested.

In abstract: How can a coupled code overcome “intrinsic limits” of a CFD code? If the authors are referring to computational costs, then that’s not an intrinsic limit of the CFD code, but of the computing facilities that are used. So please clarify what is meant or reword it.

The intrinsic limit of the CFD is referred to the ability of the CFD codes to deal with two-phase flow on the secondary side. coupling CFD with STH code, this could be overcome simulating the secondary side with the STH code. Since this is not the main subject of this paper, the sentence was rephrased omitting this topic.

When opening the file, many links of the references are broken. Please check if all went well.  
On page 2, 2nd paragraph. It’s written “validation of the modelling tools”. The tools aren’t being validated here, but the models themselves are.

Page 2, 3rd paragraph: the reference [9] already refers to simulations of CIRCE-THESIS. How is

that used as a validation process?

Last paragraph of introduction already presents results and hence shouldn’t be included in the introduction.  
Page 2, bottom paragraph: “made by” ◊ “made of”, “withstand” ◊ “hold”, “which flow” ◊ “flowing”  
•Page 3, top paragraph: “composed by” ◊ “composed of” (2x), “and from a” ◊ “and based on a”, “it is foreseen” ◊ “THESIS is foreseen”, “in the foreseen” ◊ “in the planned”

Page 4, top paragraph: “with the surface ratios….of the most relevant pool sections” ◊ with the horizontal surface ratios as depicted in Fig 2, showing the most relevant pool sections”  
Page 4, top paragraph: add a “being” before 1.5m, and also “resumed” ◊ “shown”  
Page 4, bottom paragraph: Start new sentence after simulated.“is present on” ◊ “is present in”  
“in the FIG 4, it must” ◊ “in FIG 4. It must”

Thank you, all the above comments were accepted. The paper was updated as suggested.

For the point on axial conduction, specify that it’s heat conduction. Also, is no heat conduction included in the structures either? And what about radial or azimuthal heat conduction?

The paper was updated answering this comment. In RELAP5-3D no axial heat conduction is present (also azimuthal is absent) in the hydraulics components, heat is transferred only if mass is transferred, too. In heat structures, the code only considers radial heat conduction, the axial contribution is neglected.

Mass transfer ◊ convection

Page 5, 1st paragraph of 3.2: Are structural walls included in the CFD setup?

No structural walls are implemented in the CFD domain. Potential heat transfer phenomena are modelled assuming wall temperature distributions provided, as a first guess, by the RELAP5 calculation. Ongoing CFD calculations taking into account structural walls seem confirming the suitability of the assumptions considered in the frame of the present work.

Page 5, bottom paragraph: STH provides one-off boundary conditions to CFD. I wouldn’t go as far as call this a “one-way coupling”. STH is more a precursor simulation here.  
Page 6, top paragraph: add references for the used Pr\_t used.  
Page 6, bottom paragraph:  
“section reports on the results provided for” ◊ “section reports results obtained for ” “by” ◊ “from”  
o “With respect the” ◊ “with respect to the”  
o “instead, it is reported the…” ◊ instead, the temperature….is reported”  
o “predicts a … CFD results” ◊ predicts a much lower mean temperature compared to the CFD results” o “derived by” ◊ “obtained from”

Thank you, all the above comments were accepted. The paper was updated as suggested.

• Page 7, Figure 8: very hard to compare due to different legends used. Can you use one legend for both?

Would that still be clear?

Unfortunately, it is not possible without impairing the quality of the pictures. The two codes predict different temperature range and, adopting the same scale one of the two pictures would become meaningless.

Page 7, figure 9: please use Kelvin in all figures

• Page 7, last sentence: “the positions of…3D environments”. This is a very bold statement to make at this point, when no sensitivities studies on results have been done, as well as on the used simplifications and assumptions. Better not say that here yet.

Thank you, all the above comments were accepted. The paper was updated as suggested.

Page 8, top paragraph:

Please specify what the decay heat curve looks like Please specify when the transient starts. I assume at t = 50.000 s, but make it clear so no confusion exists

A table showing the decay heat curve and the time of the beginning of the transient is now included providing a better description of the postulated transient.

Figure 11: it says “FPS-otlet” ◊ “FPS-outlet”  
Figure 11: why no peak at the FPS outlet? I would expect that when the pump is turned off, LBE heats up a lot due to still heat being added.

No peaks are present because MCP stops with one seconds of delay with respect to the power decrease. This is a safety criteria envisafed by ENEA in order to avoid possible large peaks in temperature.

o Bottom paragraph: “the jet exiting THESIS effects less the mixing phenomena”. The jet drives the mixing, not affects it. So please rephrase. Also, now the jet is weak, hence mixing is suppressed and hence you get stagnant LBE in bottom o Bottom paragraph: replace “particles” by “fluid”

This sentence was rephrased following your suggestion.

• Page 9, fig 12 and 13: please specify at what time in the transient these figures are

• Page 9, top paragraph: for what other conditions has RELAP5-3D shown success? Just thermal conduction? Please clarify that here, because the sentence is quite impactful and needs to be referenced properly.

The sentence is referred to the old CIRCE-HERO configuration for which RELAP5-3D showed good accordance with experimental data. We better explained the concept in the updated version of the paper.

Last sentences on cold LBE accumulating at bottom. This is not necessarily true, it really depends on the strength of the circulating flow. As long as the flow circulates strongly enough, LBE will keep mixing. So please rephrase this sentence to make it more accurate.

In principle this is true, but the RELAP calculations show that the LBE jet, in a certain sense, “bypasses” the mixing inside the pool and goes directly to the FPS inlet. Consequently, the LBE inside the pool is less affected by any mixing and the colder LBE tends to be collected in the bottom region. Once there, without any interference connected with the jet exiting the steam generator, only buoyancy forces may play a relevant role, resulting in the accumulation of cold LBE in those volumes.

The authors with to thank the reviewers for their valuable comments, we feel the paper was improved after this first round of revisions. We hope the achieved quality is sufficient for the paper to be accepted for the Technical Meeting.

The Authors