

Thermal-Hydraulics in Liquid Metal Fast Reactors

Dr. Antoine Gerschenfeld, CEA, France

Berta Oates

Good morning. Welcome everyone to the next Generation IV International Forum webinar presentation. Today's presentation is on 'Thermal-Hydraulics in Liquid Metal Fast Reactors' presented by Dr. Antoine Gerschenfeld. Before we get started, there are a couple of housekeeping things to kind of go over with you.

The first is in regards to asking a question. Everyone should have a control panel that they can see that has the orange rectangle with a white arrow. When you click that, it will open up a dialog box and in that there is a pane to ask your questions. So, go ahead and type questions into that pane and submit them. We will take all the questions at the end.

The audio is broadcast over your computer speakers. So, if you cannot hear me, please unmute your speakers. You can select the radio button on the right-hand audio pane display to adjust the volume. If you have any technical difficulties, please contact the Go To Webinar's helpdesk at the number shown on your screen. Again, the questions will be taken at the end of the presentation. You can go ahead and type them at any time during the presentation as they occur to you, but we will take those at the end of the presentation as time allows.

Today's broadcast is being recorded, so please feel free to watch it again or share it with others. Give us a few days just to upload the recording to the GEN IV website at www.gen-4.org. On the handouts pane, you should have a copy be able to download right to your workstation or your device, a copy of the presentation slide deck as a PDF. You will also have a handout showing all of the previous GIF webinars and the upcoming webinars through November 2020. So, an information brochure for you.

Last but not least, on the screen is a link to our SurveyMonkey, a brief survey. There is also the QR code for people on mobile devices. It's a little easier perhaps. We do appreciate your feedback and request if you to take just a few minutes following today's presentation, it helps us improve the webinars and get your input. We do take your inputs very graciously.

Doing the introduction today is Dr. Patricia Paviet. Patricia is the Technical Group Manager of the Radiological Materials Group in the Nuclear Sciences Division at Pacific Northwest National Laboratory. She is also the chair of the GEN-IV International Forum Education and Training taskforce. Without further delay, I give you Patricia.

Patricia Paviet

Thank you so much, Berta. Good morning everyone and good morning, Antione. Thank you again for volunteering to give this webinar. Dr. Antoine Gerschenfeld obtained his Ph.D. from Ecole Normale Superieure in France in 2012 and he has been coordinating R&D on the thermal-hydraulics of sodium fast reactor at the CEA Thermal-Hydraulics and Fluid Mechanics Section since 2013. In that capacity, he has led the development of a sub-channel thermal-hydraulics code as well as the development of a tool for coupling coarse and fine models in a single reactor scale simulation. He has also been involved in a number of collaborations, bilateral exchanges with project on liquid metal reactors with DOE, JAEA, IPPE, and EURATOM. And finally, he is very involved in international working group with the GEN-IV International Forum, the Nuclear Energy Agency and the International Automatic Energy Agency. Thank you again, Antoine, for volunteering to give this webinar. And without any delay, I give you the floor. Thank you.

Antoine Gerschenfeld

Thank you, Patricia, for this very kind introduction and good morning everyone and thanks for connecting to this webinar on sodium fast reactor of thermal-hydraulics. I will start immediately.

As many of you probably know, around one-third of our six Generation-IV designs in the Generation-IV forums actually use liquid metal as a coolant. So, one of the two is sodium fast reactor. This is actually a design that has been tested and put into practice at the industrial scale a number of times. So, there have been 20 SFRs in as much as 8 countries and currently we have 2 in commercial use in Russia. And the second Gen-IV design that uses liquid metal is the lead fast reactor, so the LFR. And this one has a GIF project in Russia as well, in Belgium, and in collaboration between Italy and Romania.

Of course, the reason that we have two designs using liquid metal is that they have a number of advantages over, for instance, water. The first one is that they have good neutronic properties for fast reactors, so little neutron moderation, small absorption. The other thing is that they have very large temperature working range at ambient pressure. So usually, for instance, sodium becomes liquid at around 100 Celsius and remains liquid until around 900. So, we have got 800 degrees of temperature range and with that, it's even better. And the next thing is that they have very good thermal conductivity. So, they are very useful as heat transfer tools.

But at the same time, they come with their own challenges. And one of the regions where there are specific challenges is in thermal-hydraulics. So, the behavior of heat and momentum transfer in the field.

First, what is thermal-hydraulics exactly? That's the behavior of the fluid in the reactor, more or less. What it deals with is what's the velocity in the fluid, what's the temperature, and what is the pressure. And here in our case, we are mainly going to be concerned with liquid metal in our liquid metal fast reactor, so maybe we will have sodium, lead, or lead-bismuth eutectic. But also, in the package, you also need to cover the cover gas in the reactor, for instance argon at the top of the primary circuit. Or maybe the power conversion cycle which could use steam or nitrogen or another thing.

And it covers also two very different types of calculations. One is for the normal operating state of the reactor, so it's going to be steady state in normal operation. And the goal in that case is going to assess the loads imposed by the fluid on the structural materials of the reactor. The main issue with that will be that we will need to justify that these materials can last the expected lifetime of the reactor, so for instance 60 years for the structures. And also, we need to deal with accidental scenarios, and in that case, the goal will be a much shorter time, but it will be to assess the transient, and we want to know if the reactor will be safe, for instance, in the event of an unprotected loss-of-flow. And if we see that maybe we've got a problem or two, maybe we need to adapt to design of the reactor as well.

What I will be dealing with in this presentation is first a little overview of the main thermal-hydraulics issues that we encounter in liquid metal fast reactors. And then I will talk about the types of tools that we have at our disposal to analyze these issues. There are many simulation tools and experiments. And then I will give you an example application of this practical case, and it's going to be something very fashionable since the Fukushima accident, which is the study of natural-convection passive decay heat removal in a reactor.

So, the main feature of this liquid metal reactors, as I said, is that first they've got high working temperatures. In sodium fast reactors, we usually have cold temperature of around 400 Celsius, and we go all the way to 550 for the average hot temperature, and the local maximum temperature will be a bit higher even. In lead fast reactors, they end up being around the same. The main reason for this is that the main thing that limits temperature in a liquid metal reactor is related to the materials. Since it's somewhat hard to justify that common steels can resist long-term temperature, long term for instance 4 years or may be 60 years for the structures at above 700 degrees, we usually limit ourselves to something like 650 degrees. And then in order to reach these temperatures, we do not need any pressurization and so we can rely on these pool-type designs. So, we put all the primary circuit in one big pool, and this allows us to minimize the consequences of a pipe break, and this is very useful for a sodium reactor where breaks and leaks could have

chemical consequences. But it's useful in general to minimize the consequences on core cooling. It also gives us very large thermal buffer in case of an accident because we are going to have thousands of cubic meters of liquid metal that we can heat up progressively to absorb the decay heat. And then, at the top, of course, there will usually be a cover gas to allow for thermal expansion of the liquid. So, the fluid in this primary circuit will move the heat from the core in the middle to the heat exchanger, you can see on the right. In sodium fast reactors, usually, we have these exchangers go to intermediate loops that are also in sodium, and this is to prevent a possible sodium-water reaction in the exchanger. In lead fast reactor, you could have direct exchange system.

This is the overall scheme of common sodium fast reactors, and lead fast reactors look about the same. You have the core in the middle with a number of subassemblies all in parallel. The liquid metal goes in first convection from bottom to top, and then it takes it at the top. In this purple hot pool, you can see which is delimited by this inner vessel in purple. It goes straight into what we call an upper core structure, a structure that is above the core and that includes some instrumentation and maybe the drive lines of the control rods, for instance. And the hot fluid hits this UCS, goes to the outside radially, and enters this intermediate exchanger right there.

From there, it gives away its heat, ends up in this cold pool here at the cold temperature of the reactor, so 400 degrees from sodium. And then, it ends up going around and moving all the way into these primary pumps here that are going to pick it up and pressurize it a bit in order to compensate for the pressure drop in the core. Usually, the whole pool is at hydrostatic ambient pressure, and right after the pump, you may have something like three or four bars of added pressure to push the liquid into the core. And then, you end up in this diagrid pressurized structure under the core, and there are some holes at the bottom to allow the liquid to go back into the core, and we do a complete turn like this.

And then, there is a little auxiliary system you can see at the bottom, which is something you need to prevent this main primary vessel on the outside here from heating up at the top.

At the top, you have hot fluid in the hot pool here, and it's beneficial to have this main vessel here at the coldest temperature possible. And so, what is usually done is that you pick up a little flow that would otherwise go into the core, and you inject it in a narrow space all the way around the pool. So, this is going to cool it down at the cold temperature. So, 400 in sodium, maybe a little less for lead. It's going to go around and come back into the core. So, this is the vertical view.

And then on this sideview, you can see that everything is usually distributed radially. You have got the core in the middle, liquid coming into the heat exchangers on the hot pool side, coming down, and then coming from the heat exchangers to the pumps on the cold side.

So, from this overall view, let's begin a little journey for each element in the reactor. The first thing we encounter starting from the core is the inside of the core subassemblies. Actually, the subassemblies in a liquid metal reactor are usually quite complicated. So, you've got very tightly packed secure pins like this, and this is because we want to minimize moderation of absorption by the coolant. And so, since they are packed so tightly, usually, we use these wire spacers right here. This is common in almost all sodium fast reactors and in around 50% of lead fast reactors. And using these wires means that there will be some non-trivial coolant mixing effects, and they are going to be very important when we want to compute the nominal state of the reactor. So usually, material specialists from irradiating material studies will give us a goal on the maximum temperature of the fluid cladding, and they will tell us that if you manage to stay below a given temperature like 600-650 degrees, then there will be no cladding rupture over the lifetime of the fuel. And so, we need to do the thermal-hydraulics computation to guarantee this point. And then, during an accident, the goal will be much shorter term, and we need to check that the cladding will remain below its melting points, and we will not have a transition to a severe accident scenario, for instance, where the core would be disrupted.

We need to know this at a local scale, and at least, we need to have it for each of the pin in each subassembly in the core. So, as I said, in the nominal state the main thermal-hydraulic issues in this thing are going to be taking into account these mixing effects by the wires or by the grids that you could have in a lead fast reactor. And then, in accidental conditions, so in sodium fast reactors, there is the possibility that the coolant will boil before the fuel melts because the melting temperature is 1200 degrees, and the boiling temperature will be around 900, so you could have sodium boiling before the cladding breaks down. Of course, then you could have cladding, melting, and possibly rupturing before and releasing fission gases.

A common problem both in sodium and lead fast reactors is what could happen if you have a blockage inside the fuel assembly. In this case flow will be disturbed, cooling will be disturbed as well, and the local temperature will increase, and this requires thermal-hydraulic computation.

Moving to the overall core, so first looking at each subassembly separately and then to the big overall view. When you consider the core in its entirety, another thing that happens is that with these fast reactor

subassemblies, usually the subassemblies have hex cans around them. They are isolated from each other from a hydraulic point of view, and you can choose for each subassembly what the flow rate will be in normal operation. Normally, in the core, you would have power distribution with more power in the center and less on the outside. And so, if you want to have an optimal cooling of the core, it's beneficial to have more flowrate going into the middle than in the outside. This is something that's done by allocating different flow zones in the core, and this is actually a rather complex optimization problem. So how to allocate the overall flow between the subassemblies of the core is a thermal-hydraulic optimization problem.

And then, you have also got to account for the mechanical behavior of the core, and this will be mainly determined by the actual temperatures of the hexagonal cans around each subassembly. And so, if you want to know this one, you need to compute the thermal-hydraulics in the whole core, deduce the hex can temperature by accounting for conduction, and then do a calculation of the mechanical equilibrium of the core.

There are also some interesting problems in accidental scenarios. The first one is that on top of the flow inside each subassembly, because we've got these hex cans separating subassemblies from each other, you can have coolant coming down between the subassemblies. This is something called 'inter-wrapper' flow and can have a very strong effect when you are in natural convection decay heat removal mode. In particular, it can completely cool, for instance, if you add subassemblies in internal storage, so use subassemblies that you store on the outside of the core at the end of the cycle, and they can be completely cooled down by this mechanism.

At this scale, you can also have coupled effects with, for instance, the neutronics of the core. So usually, as thermal-hydraulics specialists, we use very simple point kinetics models, but you could also use 3D kinetics for instance.

Also related to this is the thermal mechanics of the fuel because the neutronics is strongly influenced by Doppler effect which depends on fuel temperature. Fuel temperature itself depends on the thermal mechanical properties of the fuel, and at this level, you can have the coupling with thermal-hydraulics.

Moving out of the core, we encounter the hot pool, and here, you can an overall view of the hot pool of a liquid metal fast reactor. This is the temperature, so you've got flow coming out of the core at very high velocity in normal operation, heating this upper core structure, and here, you can see some control rod drive line mechanisms at the top. And you've got the jets of hot liquid coming out almost horizontally, hitting the

first thing it comes into contact with, so in this case the heat exchanger, and coming into this intermediate heat exchangers and then down.

This is the temperature profile, and now, you can see the velocity profile, and you can see similarly this very strong jet coming out. At the top, you've got these nice recirculating flow patterns like this.

Coming back to the temperatures, you can see here that you have some temperature heterogeneities here because, for instance, this is the subassembly position for a control rod, and so, there is no fuel in this position. There is also much less heating, and so, you have a cold jet coming out in parallel to all the hot jets of liquid metal coming out of the subassemblies.

This is actually one of the first thermal-hydraulics issues that we encounter coming out of the core. Talking about this upper core structure, the first thing is that it's a very complex piece of equipment. You've got usually thermocouples coming down above each subassembly to monitor the outlet temperatures. You've got this control rod drive lines here. Here, you can see the outlet thermocouples for the Phenix reactor. And you've got this strong temperature difference between control rods and the [Unclear] fuel subassemblies.

In nominal state, this is actually the source of the first main issue for thermal-hydraulics. If you have cold coming out here and hot liquid coming out right there, in practice, you will have turbulent temperature fluctuations right at this position, and this can be very damaging to the materials of this control rod drive line. So, this jet mixing effect has to be taken into account. And then, when you study incidents or accidents in the reactor, one of the first things you could have is that if, for instance, you scum the reactor, the power will go down very rapidly, and you will have cold shock under UCS. If you have an unprotected transient, you will get a hot shock instead.

Another issue you could have is that because you have a jet coming out of the core and then sideways, it's not actually obvious that the measuring positions of the thermocouples here will give you the actual outlet temperature of the subassembly below. You will see some mix of the actual subassembly temperature and it stables, and this needs to be assessed in order to interpret correctly the measurements from these thermocouples.

And then, coming into the rest of the hot pool, the hot pool is, of course, a very large volume of liquid metal. So, in the ASTRID reactor project in CEA, for instance, it was more than 1000 cubic meters. And so, for this, well, you have got some issues as well. The first is where is this outlet jet actually going? So, in normal power operation with strong pump speed, it

will usually go horizontally like this. On the other hand, if you have slow flow, so for instance natural convection or something like this, instead of going radially, it's going to be moved by buoyancy forces, the fact that hot liquid is a bit lighter, and it's going to go up instead like this. This can be an issue for reactors that want to do partial power regimes, for instance, 50% power, 50% flow rates, all velocities divided by 2. And this jet might not go in the same position as in normal operation.

Another issue is the position of this thermal interface right here. This is an issue for the steel structure, the inner vessel that you get here. If it sees thermal fluctuations or a very strong temperature gradient, it may have mechanical issues. And so, this needs to be assessed from the thermal-hydraulics point of view and checked to ensure that the design will last for its 60-year lifetime.

During accidental scenarios, you can also have – if you have hot shocks coming into here, then they will also be propagated all the way there, so you can get hot and cold shocks on the vessel and on the components in the vessel. And then, during a transient, if the flow gets lower and lower, then this jet will have a very different direction, and stratification might also occur, and this is something we will see in more detail later.

There are also some specific issues at the very top of this pool. As I said before, in these big pool type reactors, we need to have a cover gas with something like argon at the top in order to absorb the volume changes of the liquid below with temperature, for instance. And if we have a free surface like this, one thing you could get is some wave patterns right here, some oscillation of the free surface. And this, of course, can cause thermal fluctuation on the structures at the outside and maybe thermal damage. So, we need to check for this.

Another thing we need to check for is the thermal transfer between the liquid metal and the reactor slab at the top. Because this will give you the amount of cooling you need to add to the slab in order to keep it at the temperature you want in normal operation.

Another thing that can happen, and this is mainly a problem for sodium fast reactor, is that if the liquid velocity is high in this region, then it's possible that vortices could form like in a sink in your bathtub, for instance. And this can lead to vortices going down into the hot pool and possibly entraining some bubbles of gas into the complete primary circuit. And this is something that can have consequences later.

If you have a load-following reactor, usually, when you change the power of the reactor and the flow rate at the same time, this free surface will change its level, and this can lead to thermal stresses on the vessel right here.

Finally, here, we've got an accidental scenario where you can have consequences in lead fast reactor. So, if you have, for instance, an earthquake, then the earthquake can cause sloshing in this pool. You will, again, have waves. And if you have lead fast reactor, lead has a very high density right around 10, and so, it can lead to mechanical constraints on all the structures right here. This is something that is not too important in sodium fast reactors but becomes more important if you have lead.

So, moving into the heat exchangers. The heat exchangers are actually some very complex pieces of machinery that have got thousands of tubes. Usually, they are straight tubes like this. The first thing that needs to be assessed in normal operation is what is the actual performance of this heat exchanger. So, what is the pressure drop on both the primary side going around the tubes and the secondary side going inside the little tubes. What's the pressure drop? What is the actual heat transfer? And also, what thermal loads are going to be experienced on the tubes during a normal startup and shutdown. And this actually was the source of many problems in the Phenix reactor in France. Because of that differential thermal dilation of the tubes and the rest of the exchanger during move from stop state to normal operation, these exchangers had to be replaced many times. And then, in accidental scenarios, you can also have additional thing. One is that at very low flow rate, you can actually have some complicated flow patterns inside the exchanger, for instance, re-circulation between the inside and the outside. The shape of the jet coming out of the exchanger is also of particular interest, and this is something we'll see just later in the cold pool. And finally, if you have hot and cold shocks, for instance, if you are scrumming the reactor or doing an unprotected transient, then you could have some tube rupture problems.

So, going into the cold pool. Usually, the cold pool will be even larger than the hot pool, and it's going to have some very different temperature profiles in normal operation and in, for instance, low flow natural convection. In normal operation, the cold pool will usually be very homogeneous and completely at the cold temperature with some profiles at the top where you have the vessel cooling system on the outside and coming back down. You will have these outlet jets from the heat exchangers right here.

In normal operation, what we want to know will be the shape of these jets. For instance, you could have some – so as you can see, this jet will usually not be at homogeneous temperature. There is a little hotter part at the outside and a colder part at the inside. And this little temperature extension into the cold pool can actually hit some structures right in front

of it and potentially cause thermal damage. So, we need to know where this heat exchanger or the jet is going.

One thing that can happen is that if the flow gets a lot lower, then this jet will go straight up like this and contribute to stratification in the cold pool and maybe thermal loads and some other structures. And so, that's actually what we need to compute for in case of an accident, so where the jet is going and what thermal stratification will be. And this can be important both during loss-of-flow transient, so when we lose the pumps. This is what is actually pictured on this right figure.

Another thing of interest is what we call 'dissymmetric event.' For instance, if the reactor is operating and then one of the intermediate loops trip, then cooling will be removed from this exchanger and suddenly you will have hot liquid coming down on one side of the cold pool only. So, you will have hot shock in the cold pool on one side, and then it's going to propagate in the cold pool, and this can be actually quite interesting to model.

So then coming out of the cold pool, you got the pumps. So, in the pumps, the main issue is in the nominal state, so they are going to pick up liquid from the cold pool and inject it back into the core. And the main issue is going to be their performance in normal operation, so their full characteristic, so this is the ΔP , the pressure difference provided by the pump for a given rotation speed and a given flow rate of liquid metal. In normal operation, we will only be interested in the normal operating point of the pump. But then, for accidental scenarios, we will be interested in a lot of other values of this function. So, for instance, if we are modeling loss of flow scenarios, then the pumps will be stopped, and we will need to know the pressure drop of the pumps when it is stopped.

If you've got an event where one of these pipes at the outlet of the pump and before the diaphragm breaks down, then the pressure in the diaphragm will drop, and the other pumps may go into overspeed, and this can lead to cavitation inside the pump.

In some cases, for instance, if one of the pumps stops but not the other, you may have reverse flow in one of the pumps and some other complicated problems.

Another thing that can happen. If you have transient loss of heat sink, then you may have the pumps operating, but the temperature in the whole pool will go up little by little, and at some point, the temperature may be so high that the thermal dilation of this rotating part may make it contact the outside, and then you will have seizure of the pump at high temperatures, and you will go into a loss of flow scenario on top of your

loss of heating. So, this is also something that needs to be watched out for.

Then, coming out of the pumps, we've got the diagrid. This is actually a picture of the EBR-II diagrid on the reactor and the same name in INL. There are two things that we need to model in this diagrid. First, it's actually a quite complicated structure. So, you've got lots of tubes going around because they are going into the subassemblies and so on. And so, the pressure drop of this component is actually not obvious. You can also have some local flow effects. For instance, at the part of the diagrid that are closest to the inlet pipes, for instance here, the flow will not be homogeneous. And so typically, this is something that needs to be computed to know if you may not have what you expect in the subassemblies that are closest to these inlet pumps.

In sodium fast reactor, we also have the potential for gas accumulation in this diagrid to watch out for, and this is something I will elaborate a bit on later.

And then, we've got the accidental issues. The first thing is in all liquid metal reactors when you have an unprotected accident, the thermal dilation of the diagrid is actually the source of one of the strongest neutronic feedbacks of the reactor. If the diagrid heats up, then all the subassemblies will be blown apart, the complete core will dilate, and the neutron reaction will slow down more and more. So, this is something that is actually very important when you want to model an unprotected transient.

Another thing of importance is for the dissymmetric transients. For instance, trip of a single pump or the breakdown of a pipe like this, then you will have usually uneven flow at the core inlet. If this pipe breaks down and the only pump contributing is this one, then the subassemblies on this side will have more flow rate compared to the average. This is something that can be an important to model.

Same thing for these intermediate loop transients. If you have one of the heat exchangers losing its heat removal power, then hot liquid will come out into one of the pipes before the other, and one part of the core will be seeing hotter liquid before the other, and this is also something that you may need to model.

Finally, the only thing we still need to look at is this little vessel cooling system here. This is sodium that you pick up from below the core, and you inject on the outside of the main vessel to cool it down. It's not so little, actually usually it's 10% to 20% of the flow rate provided by the pumps. And so, for this one, what we need to model is the actual thermal transfer from this hot liquid to this liquid in the cold pool which may be a

bit colder, to the liquid from the vessel cooling system. And so, we need to model it in order to know if our vessel cooling system has the right size.

And then, if you use a system here where you have liquid metal falling down like a fountain from the vessel cooling into the cold pool, then you may have bubble entrainments at the bottom of this weir-type design, and so there is a potential for gas accumulation right there.

The final thing that can happen is that in an accidental scenario where you have what we call 'reactor vessel cooling system' so you've got a cold source here, then you could have cooling of this vessel cooling system, and you can have liquid metal going this way, cooling down, and going straight below the core and contributing to the cooling of the core inside. This can actually be a very good source of cooling in liquid metal fast reactor. It's the flow reversal in this kind of cooling system.

Before we move on to the modeling, I took about two issues that involve the complete primary circuit. The first one is this gas entrainment problem in sodium fast reactors. So, what can happen with this gas? You can have vortices forming at the top of the hot pool here and entraining bubble into the primary, and then, the flow of sodium will touch these bubbles and move them down into the rest of the circuit. If you have a kind of weir at the top of the vessel cooling system, you can have gas coming down from here. And also, here, you could have dissolved gas nucleating in the coolest part of the reactor. So here, you will have hot liquid metal and so higher solubility threshold for gas. And here, because the temperature is lower, this dissolved gas may nucleate and give you additional bubbles. All this can be moved around and potentially accumulate at the bottom of the diagrid right here. So, because usually the inlets of the subassembly will be in the middle, there is some kind of dead zone at the top, and you can have a pocket of gas accumulating in this somewhat dead top part of the diagrid. And then, the worrisome consequence would be that if you change the speed of the pumps, for instance, you could have these pockets that had accumulated in the pipes moving all at once into the core, transferring into the core and causing positive reactivity event. So, we need to assess that no pocket large enough to disrupt the core can accumulate. And in order to do that, we need both to control for all of these sources and then use specific system to ensure that no large pocket can form below the core.

Another issue that I will cover in more detail later is the decay heat removal mode in liquid metal cooled type reactor. A big advantage of these liquid metal reactors is that we can have completely passive decay heat removal. Usually, what we do is that we insert a specific heat exchanger in the hot pool at the top, and we connect this exchanger to a liquid metal circuit that can operate in natural convection. We connect the other end to an exchanger in a chimney like this that will actually be

cooled down by air, and so, air will be your final heat removal source, and this never goes away. And so, you will have cooling here, natural convection in this circuit, and cooling will be provided at the top of the pool like this. And then, the question becomes how does heat go from the core right here to this exchanger right there?

The answer is that in a cooled-type reactor, it's quite complicated. You can have the normal flow path like this. So, path number 1 is liquid metal coming down into the cold pool into the pumps, below the diagrid, and moving up in the core doing the normal path. But you can also have cool liquid metal coming down at the outside of the hot pool, coming down in the cooler subassemblies in the core, doing a U-turn below and coming back up. This is the circulation loop between subassemblies.

A final thing you can have is that this same cooled liquid metal can actually go down on the outside of the core and then move around between the subassemblies in this inter-wrapper region like this. And then, it can cool subassemblies from the outside, heat up, come back up and contribute to core cooling.

So, the first challenge is to model all these flow paths correctly to know what's going to happen at the level of the core itself.

Another possible issue is in this circuit right here, because usually this circuit will be stopped in normal operation and then you will have to start it when the loss of heating happens. And so, the first issue is that you need to ensure that during startup, the temperature will not go down in this part too fast and so you will not have freezing in this part which would completely stop the overall flow. And then, another thing you need to ensure is that the natural convection like this will start up on time and will not have problem.

Also, of potential interest is that the intermediate loops connected to these exchangers may contribute to the removal of decay heat, at least at the beginning.

Now that we've seen all these interesting thermal-hydraulics problems, one thing what we want to know how do we describe them, how do we model them. And there is both good news and bad news. The good news is that for thermal-hydraulics, especially in single phase, so if we don't have to deal with liquid metal vapor, then we've got the Navier-Stokes equations, which can describe what is going to happen ab initio, from first principles. The only problem is that these equations are not linear, and they induce turbulence, they induce a lot of small fluctuations. So typically, if you want to do an ab initio simulation like this, you will need to go all the way from the small turbulent fluctuation in the reactors, which will be of a scale around 1 micrometer and 1 microsecond, all the

way to the complete reactor itself, which will be of around 10 meters big and maybe you will have to simulate something like 100,000 seconds, a few days. And this simulating from this scale to this scale may be possible before I retire, but it's not very likely.

And so, because this ab initio modeling is very heavy on the computational side and not practical in a real situation, we need to do something more. And what we do is that we introduce what we call a cutoff scale. So, we fix ourselves a given scale and all phenomena above that scale, we would aim to simulate directly by simulating the equation with computer programs.

All the phenomena below that scale, so everything that is smaller, then we will not be able to simulate directly, and so, we will need to introduce what we call physical models or correlation to describe this phenomenon and introduce them into what we can actually simulate.

So, when we go into actual particle application, what thermal-hydraulic specialists did is that they chose different values for this cut-off scale. The smallest scale on the right is where you have a very small or low cut-off. If your cut-off is so small that you can describe the exact geometry of the reactor, so for instance, the little wires between all of these pins in the subassembly, then you will be doing what we call computational fluid dynamics or CFD. And in this case, the simulation scale would be very small. You will have a few physical models, not too much, but the computational cost will be very high.

On the other hand, here, this is what happens when you choose a very large cutoff scale. So here, we have what we call the system scale or STH, system thermal-hydraulics. This is a scale where, for instance, for the complete hot pool like this, you will have one big average value. In a model like this for an exchanger, you will have one average value in a 1D profile. So, the computational cost will be very low, but on the other hand, you will need lot of physical models actually for every phenomenon you need to model.

And then sometime, you get something in between like this. This is what happens when you want, for instance, the local temperatures around each fuel pin, and so, you end up with something that is 3D, and this is what we call, in this case, the subchannel scale. So, you will have one temperature value between each trio of pins. And here, you will then model for all the fine geometry that you cannot have. For instance, at this scale, you cannot see the wires and you will need physical models for the wires. And here, just for information, I have given you the names of the codes we use at CEA for liquid metals.

So, going into a bit more detail. We've got, so first going from the finest to the biggest. First, we encounter CFD. As I said, CFD is when you model all the geometry of the reactor or of a local part directly. The geometry is above your cut-off scale. What is not above your cut-off scale is all these small turbulent fluctuations. And there, you have three choices. The first one is you simulate everything. So, this is what we call direct numerical simulation or DNS, and it requires some very small computational meshes, 1 micrometer. You can choose to model, so to not directly simulate the smallest turbulent fluctuations. And this is what we call large eddy simulation, and this can allow you to have somewhat bigger meshes. So, you gain usually a factor of 10-100, and this decreases the computational cost and the cost of a physical model for the small fluctuation.

Finally, you've got the option of trying to have all turbulent fluctuations below your cut-off scale. So, you need to model everything from turbulence with a physical model, and this is what we call Reynolds-averaged Navier-Stokes. If you do this, normally, you can use a mesh size that is only given by geometrical features. So, this usually allows for meshes that are from 0.1 millimeter to 1 millimeter in size.

So, this RANS lowers numerical cost, but you have to introduce physical models, and this is mainly what we call turbulence models, and they will need to be validated. It's against final simulation that would directly stimulate the turbulence or using an experiment. And of course, if you do a simulation where the turbulence is described by your model, then you will lose information on all the turbulent fluctuations like, for instance, in this picture, a Japanese experiment. This is actually an oscillatory behavior between hot and cold jet. And if you do Reynolds-averaged Navier-Stokes, these fluctuations will not be available to the simulation.

So, here are two examples of what we typically can do. If you want to do a direct numerical simulation, usually, the best we can do is, for instance, a few, like 10 centimeters in height in a single subchannel between three pins in the subassembly. This is very expensive, and we can only do a small region.

On the other hand, if you use the RANS, the Reynolds-averaged Navier-Stokes, then usually, the cost is so much lower and you can get to model a few subassemblies in the entire region, so you can see the gain between direct simulation and the RANS.

Then, moving up in cut-off scale to the subchannel. So, these are codes where you have one computational mesh per fluid intersized [ph] between two pins. So, this is something where what you cannot simulate might need the effect of these wire wrappers a bit. And so, what you need for models are the frictions and the mixing effects caused by these helical

wires. They have a tendency to increase friction in the bender, but they also induce mixing between neighboring channels.

The next thing you need to model is what the actual heat transfer is between the pins. This is something that needs to be calibrated on experiments or what's more, the most fashionable thing today is to do a fine simulation and use this to calibrate the coarsest model. This is what is called 'Hi2Low.' And so, going to this coarsest scale has a very big advantage over CFD in terms of computational cost. For instance, if you are only interested in a steady state, you can usually use these codes to get the full steady state of your core in less than one second, very fast. If what you want to compute is a transient state, it will take a little more effort, but it's perfectly doable.

And then moving all the way up into the system codes. First, you should know that these are actually the original thermal-hydraulics calculation tools. The first one, RELAP-1 is from 1966. They use the coarsest mesh possible. So, you've got pipes like the heat exchanger here, you do them in 1D and big volumes. You do them with one big average value like the OD or may be sometimes in coarse 3D. You need physical models for everything, so friction, heat transfer, and so on.

And also, what's interesting in this code is that usually they include additional models for simulating a complete reactor transient. For instance, they tend to have a model for the core neutronics with point kinetics usually, but some codes have some 3D kinetics, so you can compute the evolution of the power. They have modules for pumps, for heat exchangers. And another interesting thing with this code is that they can simulate several circuits, so you can have, for instance, the primary circuit, the intermediate circuit, and the power conversion system in the same simulation.

They do all this with a very low numerical cost. It's very rare to have a system scale simulation that takes longer than 15 minutes on your own computer.

And so, when you look at the type of issues we have, usually, what will happen is that if you want to model a safety transient, an accidental transient where some pumps stop or the reactor is scrammed and so on, you need to model the complete reactor, and so only the system scale, that's all you need. They've got all the models, and they can describe the complete reactor at a low cost. This is what you would typically use for a transient.

If you want to do something like cold design, the maximum pin temperature above the complete core, then typical tool will be this subchannel scale because it's going to give you the per pin temperature

maximum for all of the core at a low cost. And sometimes maybe you will want to know some more about local effects, and in this case, you can use CFD of some specific places. And then, if you've got some more complicated geometry-dependent phenomena, then the typical tool of choice will be CFD. So, if you want to model big 3D evolution in the big pools for instance, RANS, so CFD, that model's turbulence would be the tool of choice. And if you want to model something that will depend on thermal fluctuations that are related to turbulence, then the typical tool would be LES scale CFD.

And then, you've got some cases where nothing works. So, this will be actually my example application for this, and it's going to be the modeling of natural convection in liquid metal reactors. So, natural convection by a sense that is global phenomenon. It involves the complete primary circuit and maybe some other parts of the reactor. So, the natural choice is the system scale.

But the problem is that with the reactor designs, with big pools of liquid metal all over the primary circuit, this is something that is hard to describe if you only have 0D or 1D components that you network together.

One phenomenon of interest, for instance, is the behavior of the outlet jet in the hot pool and if you will have stratification. Of maybe the possibility of inter-wrapper flow cooling between the hot pool and the core. You also have this radial heterogeneity between subassemblies that could have an effect. And you could try to introduce physical models in the system core, but they are going to be both dependent on geometry and transients. And so, it's complicated. It's very hard to introduce physical models as well.

The first thing you can do, and this was done in the past, the first thing is if you are studying a transient where you may have local effects that are hard to simulate but they have no consequences on the overall behavior of the transient, then you can just do your system scale simulation, core scale, and then do some kind of post processing with the finer core.

If there is an effect but you don't want to model it or describe it in detail, you can take conservative hypothesis. For instance, you can say I don't really know what the inlet temperature in this pipe will be, but I will take the worst possible value, for instance the one coming out of the core area, and I take that coming into the core, and I will get a conservative model. But maybe with more recent design, it would be interesting to reduce this conservative hypothesis to have some more margins and optimize the reactor design. So, it's a big motivation. There is a big motivation to move beyond these two simple approaches.

When you look at the actual situation, this diagram comes up again. When you look at these three competing paths for moving heat from the core to this decay heat removal exchanger, you have path 1, normal flow; path 2, recirculation loops between subassemblies; and path number 3, recirculation loops in the inter-wrapper gaps. If you have the standard system scale approach, the only thing you can model, more or less correctly, is this path number 1 because, for instance, trying to model path number 2, you need to know what the temperatures are inside the hot pool. If you have one big average value between this part and this part, then computing the flow here will be completely inaccurate. This path going between the subassemblies, it requires 3D temperature inside the subassemblies and the heat conduction through the inter-wrapper gap.

Same thing, it's very hard to model in STH. So usually with the system scale approach, you compute this, but it's not accurate. And this one number 3, the inter-wrapper flow is neglected.

So, what's going to actually happen is that because you have removed some mechanism for decay heat removal, the core temperature will tend to be higher in the simulation than in the real case, which is actually okay. It's conservative. But you will also overestimate the amount of work done by this path number 1. This means that you will overestimate the primary flow rate, and this will lead you to increasing problems for the average temperature on this side, average temperature for the second one, and can potentially be bad.

And so, you need an approach where you can somehow model these other two paths. The first thing you can do is that you can say, okay, let's look at all the tools at my disposal, and I will choose the one that can model all this phenomenon, and the one that you can use is CFD. This is what is required in the hot pool, in the cold pool, and in the core. And what people have done is that they have done models of complete reactor primary circuits using the CFD code.

And here, you can see an example model of this for a lead fast reactor, and here, you can see the same for a sodium fast reactor. This means that you can use the code you already have. But at the same time, it also means that you will need to re-implement the models that you already had in the system scale in order to model reactor transient. For instance, the useful models are the point kinetics in the core, the pumps in the primary circuit and so on, and you will need to re-implement that in the transient simulation.

Another potential cost is that if you do everything in CFD, you will have fine modeling in the regions that are of interest to you. For instance, the outlet of the core, the cold pool, but you will also have a 3D modeling of this diagrid here, for instance. This is a region where there are no 3D

effects, but you will still have CFD mesh, and so you will pay for computational cost even where you do not need it.

Another thing that you can do, and this is what we actually did in CEA, is that you can take the codes you already have, so system subchannel CFD, and try to use them together. So, for instance, you would have the system scale and then inside the subassemblies, you would use the subchannel scales. Inside the hot and cold pool, you would use CFD because you want to know the 3D behavior. Then, you can have maximum re-use of what you have already.

But on the other hand, you will need to make all these codes work together, and you will need an algorithm to ensure that when all these codes work together, they are actually giving you three aspects of a consistent global simulation.

This is something you can do in a number of ways. And here, I've given you an example the way we use at CEA. What we will use is that initially we got the system code here, so the coarsest code is actually present in the complete simulation. And then, we decide that part of the simulation, which for instance could be the hot pool, is going to be overlaid with a finer CFD simulation like right here. If you want to ensure that you have a consistent simulation where the system code is computing this part and the CFD is computing this part, what you can do is that, for instance, you take the temperature here from the system code, you impose it as a boundary condition to the CFD. And on the other hand, for instance here, you take the average outlet temperature on the CFD, and you impose it right on the inside of the system core. And with this, you can have global energy conservation between the cold system scale and the fine CFD scale. It's a bit more complicated when you consider the velocities and the pressures. So usually, what you do is that you got the system model, you measure the flow rate at these points, and you impose it as a boundary condition to the finer scale of CFD.

Because liquid metals are almost incompressible, then you will have the same flow coming out, and you will not need to do anything. It's also going to come out on that end in the system side.

The interesting thing is that in a simulation like this, if you want to have a consistent picture between this side and this side, you need to ensure that the pressure difference between those two points in the system side is the same as the one predicted by the CFD code. So actually, you need to do something extra. You need to change what the system code calculates for this in order to match the delta P on the CFD side. This can be done by introducing little what we call a 'momentum source term,' an artificial force here. And this force, we adjust it in order to change the pressure difference on the system side until we've got the same delta P

on the system side as on the CFD side. And this is enough to ensure a consistent global simulation. On the other hand, it needs to be as an iterative process. You need to try different values and take the right one.

Once you do this, you can do some complicated coupled simulation of sodium fast reactor going into natural convection. So here, you have an example from the ASTRID project at CEA. This is the reactor in normal state. You've got the core and the subchannel scale. Starting from this part, this is CFD, so here you can see the outer jet of the core, for instance, goes into the primary side of the heat exchangers and into the cold pool with this outer jet here. And finally, into the pumps on this side and below the core. So, starting from the pumps, we are at the system scale with our system code CATHARE, and we stay at the system scale until we go back into the core.

In natural convection, the picture is completely different. At the outlet of the core, instead of being a strong outer jet like this, you have a small chimney flow like this. You've got a chimney effect going all the way up from the core. The hot pool itself is stratified. There is some hot sodium at the top, an interface in the middle, and some cooler sodium at the bottom. And this is because you have this decay heat exchanger cooling down the sodium in the hot pool and injecting cooler flow here.

On the other side, on the cold pool, you also have stratification. The jet from the heat exchanger is no longer present. Instead, you have this kind of completely stratified flow. The cold pool is not uniform anymore. You've got hotter sodium at the top and cooler sodium at the bottom with a very strong interface. And in general, the situation is completely different.

So, when you look at the actual results from this, this is the predicted flow rate in a number of simulations corresponding to this case. If you do an initial system scale computation, you will have a flow prediction like this. It goes down to around 1% of the initial flow rate – the nominal one, and then, at some point, it raises up and goes to around 2% of the nominal flow rate. And then starting from this system scale approach, we did our first coupled simulation with CFD, and we obtained the blue curve. So that was a bit worrisome because the first coupled simulation predicts a lower flow rate, and it's almost minus 30%-40% at the end of the transient.

This meant that if you only have the system scale, you will have problems predicting the actual flow rate in a conservative way. One thing you can do is that if – the main source of this overestimation is that in the system scale, you have usually a thermal exchange computed between the hot pool and the cold pool. And this heat exchange is computed with the

average temperatures. So, you would have a heat exchange coefficient time difference between this average and this average.

In nominal state, this is a very good approximation because this and this are homogenous. In natural convection, if you take the average of this temperature minus the average of this temperature, it will be very far from the actual reality. And if you have the system scale, one thing you can do is that you can say, okay, I won't be able to compute this correctly, so I'm going to remove this coefficient and produce a conservative estimate.

This is the green curve. The green curve is system scale only again but with the conservative hypothesis, and you can say, okay, I am back in business again, because now I've a conservative prediction of the flow rate compared to the first couple calculation.

The next step is how about when you do a simulation where you have system scale and CFD in the pools and the subchannel scale between the subassemblies. This is what you need to do to predict inter-wrapper flow, and you end up with the purple curve. And the purple curve is even lower than the most conservative system scale calculation we can make. This means that, in this case, it's very hard to get a conservative prediction of the flow rate without an advanced approach at the model local effects. If you compare the initial prediction here to the final one with the coupled simulation here, the difference is more than 100%. Actually, for the design we had at the time at CEA, we really needed to have these advanced simulation tools with subchannel in CFD in order to get a correct prediction of our flow rates.

Then, of course, you've got the next issue is that once you've done your simulation you need, you will have introduced a lot of physical models. For instance, in CFD, you could have turbulence models. In the system scale, you could have friction and heat transfer models, and so on. And all these models must be established from experiments. If you have DNS CFD, you do not need anything, and if you have a system scale simulation, you need a model for every physical phenomenon. And this is what we call 'validation.'

But because thermal-hydraulics is non-linear, one important thing is that if you have, for instance, phenomenon like this one, and it's a local phenomenon like heat transfer in a subassembly, and you validate it using your local experiments on this full range of parameters, the same as in the reactor. When you combine it with another phenomenon, you could have interactions between them and new effects. In this case, what you need to do is that you need to not only have experiments over the whole range of parameters from one phenomenon, but you also need experiments that combine these parameters to have some idea of this

behavior. But because, of course, you cannot do something as big as the reactor, before you do the reactor you will have a size limit on this experiment, you will not be able to go all the way to reactor scale. And you also need what we call system or industrial tests, integral validation – tests that are done on existing reactors that you predict with the modeling tool of choice, and you check that you are consistent with what was actually measured. And then you will actually try to predict your own reactor case right here. So, this is what we call hierarchy of experiments. You've got separate effect experiments, one phenomenon but in detail, combined effect experiments – several phenomena all together inside your large experiment, and then integral validation reactor cases.

If you look at some examples of experiments you need, when you want to model this natural convection issue, the first thing that you need to model is natural convection and its interaction with 3D effects in different parts of the reactor. And for this, there is a range of lead and sodium experiments, especially a lot of them have been performed in European projects with names like THINS or SESAME. For instance, in KTH in Sweden, they have an experiment where you have a loop in lead-bismuth eutectic, and in the middle, there is what we call a 3D section, a little cylinder of hot lead, and this cylinder will stratify like this, and there may be a jet at the bottom that can either go all the way to the top or get stuck on the stratification. This is something you can use to check if you are modeling correctly the interaction between 3D effects and overall network circulation.

At ENEA in Italy, there is an integrated experiment called CIRCE on the same issue. And there is scaled down experiment of the MYRRHA reactor project called E-SCAPE being done at SCK/CEN.

There is also the issue of the thermal-hydraulic coupling between the inside of each subassembly and the inter-wrapper flow. For this one, you can do some small-scale analytical experiments like this one in KIT in Germany where they have three subassemblies with seven pins each, the thermal-hydraulic transfer with the inter-wrapper flow in the middle and a lot of thermocouples to check that the code is completed correctly.

And then, at a larger scale, you have Japanese experiments PLANDTL at JAEA, which can model both the core itself with as much as 55 subassemblies and its coupling to the overall hot pool. These experiments are actually very important to show how much heat inter-wrapper flow can remove from the core to the hot pool and then to the decay heat exchangers.

And finally, there is a possibility that in the near future, we could have large experiments where you have both the core with inter-wrapper flow and the rest of the pool. And this is something that, for instance, could

be experimented on in the CLEAR-S facility for lead bismuth fast reactors in China.

And then going at the larger scale, we have integral tests, and so for this one, here I've put a list of interesting liquid metal tests for validating liquid metal from hydraulics codes. All of these tests are on sodium fast reactors because we've got the integral, the reactor scale experience of this one. For instance, we've got tests that were performed at the end-of-life of the Phenix reactor right here. For instance, at the end-of-life of Phenix in 2009-2010, we did natural convection tests. First, we stopped the heat sink, actually we started with what we call a loss of heat sink, and then we shut down the pumps and we did protected loss of flow, PLOF, natural convection test. This is something on which we did both, we did public benchmarks in the IAEA CRP and in European public framework. And here you can already see the importance of international organization in sharing this reactor scale data. More recently, we also shared data on dissymmetric test, a test where we shut off one of these two intermediate loops and not the other. And this allowed us to measure 3D effect in the cold pool. We shared data on this test for new EU benchmark in the SESAME project, and it is currently being shared in the GIF SFR safety and operation working group. I think this is a really big incentive to join this SFR as a new working group.

And then, we've got test on EBR-II from DOE. In the past, there was an IAEA CRP on two loss of flow tests, so natural convection, 17 which was protected loss of flow and 45R which was unprotected. So, to date, two public tests through IAEA. And then more recently DOE has proposed two loss of heating tests, so the pumps keep operating but the heating is lost in the same GIF SFR safety and operation framework. These ones are called BOP-301 and 302R.

Finally, since September 2018, we have had the benchmark on the FFTF loop type sodium fast reactor at DOE as well. This was an unprotected loss of flow test called the LOFWOS. This one is actually in progress, and this is an IAEA CRP. The first meeting with the presentation of the blind result and so on will be next April actually, and this is an ongoing benchmark.

So, as you can see, if you want to obtain integral validation, check and share reactor data, IAEA and the GIF play a very important role.

And so, moving on from this application to natural convection, I got a little conclusion right here. The first one is that in liquid metal reactors, as I showed you, there are a lot of interesting thermal-hydraulics phenomena, and we need to describe them. Some are related to normal operation. Some are related to accidental scenarios, but we usually end up needing to model them.

In order to do that, we have developed a range of codes with different modeling scale. We've got CFD computational fluid dynamics, both DNS – direct numerical simulation where you simulate everything, and you need no physical models. Then to LES, some fluctuations, and then RANS, all turbulent fluctuations are modeled with a physical model. Then do subchannel scale, a bit coarser, and finally, the system scale where you need physical models for everything.

And then in many cases, given the problem of choice, one of these cases here will be what you need. For instance, for local phenomenon if you need to account for fluctuations, you need to use LES. If you don't care about fluctuations, RANS would be all you need. If you need subassembly thermal-hydraulics at the level of a single subassembly and locally, you can use CFD. If you need to do it over the whole core, then the subchannel scale will be the one of interest. And if you want to model reactor transient, you will need the system scale, the coarsest one. And in some cases, this may not be enough, and you may need to do something more complicated. For instance, coupling between different codes. In our cases, a lot of work and actually the majority of the work will be in an aspect that we don't think about at the beginning. It's the experimental validation of all these physical models, and for this, we need experimental data at small scales all the way to a real actual reactor, and this is where international collaboration is extremely important.

That's all from me. Thank you for your attention.

Berta Oates

Thank you, Antoine, very much for your presentation. There have been several questions posed. We will give people just another second to type in questions that they may have following your conclusion. In that time, we'll just take a peek at the upcoming webinar presentations. In February, there is a presentation on 'SFR Safety Design Criteria and Safety Design Guidelines.' I'd like to draw your attention to the shift in time scheduling for that presentation. Rather than in the morning US Eastern Standard Time, we're going to shift that to 8 p.m. US Eastern Standard Time to align better with the Japanese presenter.

In March, we have a presentation on 'MicroReactors: A Technology Option for Accelerated Innovation,' and in April, the 'GIF VHTR Hydrogen Production Project Management Board.'

So, the first question on the list is shouldn't the LMR Reactor Group include solid fueled but clean molten salt cooled reactors as well? It's just another high temperature, low-pressure coolant.

Antoine Gerschenfeld

Of course, molten salt is a very interesting option. Actually, even in CEA in France, as you may know, our big sodium fast reactor project, ASTRID, came to a stop at the end of last year. Since the beginning of this year, we have started to look into molten salt as well, mainly as a liquid fuel, so with the fuel inside the coolant. There is also the option of having solid fuel but molten salt as a liquid.

In this case, the main issue will be both the material problem like corrosion, corrosion and the [Unclear] and things like that. The main advantage of liquid metal in this case, for sodium for instance, there is almost no corrosion issue if you control for oxygen content. You can use standard stainless steel and be convinced that this material will last the lifetime of your reactor. Of course, if we do the material R&D, we may find a material that does the same thing for molten salt. But if you ask me to design high temperature molten salt reactor right now, I will have big issue with the materials of choice. But this may change in the near-to-medium future, and this is something that we should look into.

For lead, it's more or less the same. There are a few more issues with corruptions seen in lead. You need some oxygen content in order to have an oxide layer of the steel to prevent lead from somewhat corroding it. But still, it's always your materials problem than molten salt problem.

So, I think as an option, we need to keep investigating the two liquid metal reactor concepts because these ones more or less, we know how to make them. Even, for instance, we know the choice of material, we know all the issues and so on. But if we are considering a project for a bit of a longer term, then everything is interesting, of course.

Berta Oates

Thank you. The next question. Please describe the function of core catcher and the flow patterns from the core catcher catching the melted core.

Antoine Gerschenfeld

This is something that we included in the ASTRID project, and we also see in a fuel sodium fast reactor design. Of course, if you have a severe accident in a sodium fast reactor, in this case, the fuel is going to melt in the core, and at some point, it may come down. Both, for instance, going into the cold [Unclear] subassemblies and all the way down. And of course, you've got the main vessel at the bottom and so you can choose two things, either to keep the corium inside the primary vessel at all costs, or you let the corium go down to the main vessel, close the lid, and catch the corium below.

In ASTRID, our choice was to say that we will catch all the corium inside the main vessel all the time. The advantage with that is that there will

not be a leak. And once the corium is on this core catcher, the thing that will catch the corium going down and stop it finally, then we will be in a position where we can cool it more easily because we will be in sodium, we can use one of these decay heat exchangers to remove heat from the top of the pool, and there will be natural convection flow from the core catcher at the bottom to the heat exchanger at the top.

Of course, the big question we had was what material can we include for this core catcher, and we were considering things like zirconia or other materials like that. The big issue is that can you find the material that will both be able to catch and dissipate the corium and will also last for 60 years of lifetime of the reactor. Will it work if there is a core disruption accident? And if we don't need to use it, can we at least check that it will not cause problems during the 60 years' lifetime of the reactor. So, the function of the core catcher is that if corium manages to come down from the core, we need to have a material that will not be ablated by the corium. So usually, ceramics or zirconia.

Berta Oates

Thank you. There is a question about the IHX design, specifically why the aspect ratio of IHX is so high. If more heat transfer tubes are used, the height for the IHX can be reduced. Is there any restriction about the IHX design regarding to its height or aspect ratio?

Antoine Gerschenfeld

This is actually something that we did in the ASTRID project. We used the overall diameter of the primary pool. In the French SFR design, and I think is also valid for the Russian SFR designs, we have the core in the middle and then we have the heat exchangers and the pumps on the outside. And so, if we want to have the smallest vessel we can, we need to make our heat exchangers as small as possible in the original direction, and because we're trying to make them thinner, if we want the same heat transfer, we tended to make them longer and so they ended up going all the way down as you can see in the picture.

This is actually something that is related to our complicated flow behavior in natural convection. It's because our heat exchangers are so long, that we have low network convection flow rate, as you can see on one of the slides. It's slower than it would be if the heat exchanger was short and at the top of the reactor. Nevertheless, it's a thing you can do if you want to optimize your reactor design as much as possible. And even with long heat exchangers like this, you can still have good overall core cooling if you model all the possible decay heat removal parts between the core and the final heat sink.

Berta Oates

Thank you. We also have a question about extending the code use to molten salt reactors, and I am not sure that we answered that question. Do you have any more thoughts on that?

Antoine Gerschenfeld

Normally, if you go from liquid metal to molten salts, from the thermal-hydraulics point of view, the big difference will be the heat transfer modeling in, for instance, CFD code or system codes. So, in liquid metals, usually, they have very high thermal conductivity. And so, you are in a case where the turbulence has an effect on momentum, but it does not affect the heat transfer itself very much because the heat transfer is being done by the very high thermal conductivity. And in molten salts on the other hand, you have a very low thermal conductivity actually, lower than water for instance. And so, we need different turbulence models, for instance, to correctly predict the behavior in the molten salt reactor.

Another thing that needs to be done is that if you have liquid fuel and you know you will have these delayed neutron emitters, the radioactive elements that emit the delayed neutron fraction, will be moved on by the fluid. So, in this case, we really need both to predict where they are going with the thermal-hydraulics, so with the CFD code, and we need to couple it with 3D neutronics to get the actual power distribution in the core even.

Currently, for molten salt reactors, we are working on the coupling between our CFD code and a 3D deterministic neutronics code in order to be able to correctly model molten salts reactor, for instance.

Berta Oates

Thank you. Do you liaison with the university mathematics departments, for example the application of group theory and geometrical methods to the solution of differential and difference equations, the emphasis on combination of analytical or numerical methods and also the use of symbolic computation?

Antoine Gerschenfeld

On this part, there is a range actually of computational methods that have been tried in each of the scales of modeling we have. For instance, one of the system codes, so the one with 0D and 1D models, the German version of this, they use direct methods of characteristics to solve their differential equation. Usually, the main thing that we see used in these codes are finite volumes because physicists have made them and so they are interested in conservation relations.

And myself, I've worked on numerical scheme of this sort, and for people that really have – they are going to have a very good conversion rate on the differential equations from the DNS direct numerical simulation. The

DOE uses a code with spectral elements. So finite element methods where you increase the degree inside each element as you refine. And so, you get exponential convergence to the analytical solution.

But I should say that the main thing you see is finite volume methods with first order discretization, both in time and in space. And this is because usually you see physicists behind these calculations and so they're interested in some very down-to-earth [ph] conservation properties, even on the discrete solution.

Berta Oates

Those is a couple of similar questions. I'm going to go ahead skip through a little bit. The non-dimensional sodium [Unclear] operating temperature is very low, what is the significance of this? And then there is one more on how do you explore the parameter space? Do you conduct probabilistic such as Monte Carlo analysis?

Antoine Gerschenfeld

First for the Prandtl number, the practical consequence in sodium is that if we, for instance, look at how you can perform a CFD simulation of sodium reactor, then we will use a turbulence model for the momentum, for the velocity. But on the thermal side, because of the [Unclear] Prandtl number, on the thermal side, we can just choose the naïve laminar conduction rates, and this actually accounts for all the conduction you need. Same thing at the system scale. The non-dimensional number for heat transfer is what we call the Nusselt number, and it's around 4 in laminar flow and then it goes up with turbulence. And in sodium, you can take constant Nusselt number equal to 5, and it will predict the whole interesting range of parameters. So, this very low Prandtl actually in sodium is so low that it simplifies things a lot. You can model heat transfer as if you were in laminar flow almost.

In lead, it's a bit less guaranteed because the Prandtl is a bit higher, but still, you have this type of phenomenon.

For the second question, for the non-dimensional parameter range, when we do validation and experiments, we try to cover as much of the actual parameter range as possible when we are doing our experimental tests. And then there is an interesting point when we perform reactor simulation. In these simulations, one thing that is very common today is that we may want to do uncertainty propagation or uncertainty analysis of reactor scale calculations. And so, we will have a range of input parameters, for instance what was the power of the reactor at the beginning.

We may have some uncertainties on the physical models as well. So, for instance, we know the friction coefficient somewhere but not with 100%

accuracy. We have some uncertainty on that. And then, we will propagate these uncertainties and assess their effect on the final output result. This is something we do sometime usually in a naïve way with the LRIC method where you do a 207 calculation and do some statistics. But what's also really common is to use Monte Carlo sampling of these parameters, build a meta model of the simulation on top, and then do our uncertainty analysis on the meta model. Actually, it's rather common to use Monte Carlo propagation at some point.

Berta Oates

Thank you. How would you couple STH with CFD when there is local or temporary boiling and voiding in the core? In this case, the core exit flow rate can be different from the inlet flow rate and can be evenly reversed for as the voids collapse.

Antoine Gerschenfeld

Yes. The method I showed in my earlier slide actually only works with single phase flow. We actually tried it in a situation where you have boiling on one of the side and not on the other, and of course, it doesn't work at all. So, what we are working on right now is we want to move from this overlapping coupling you saw here to what we call decomposition coupling where you take out the gray part on the system side. You have the CFD. And then, we want to use what we want to use. We want to use what we call a modality called complete Newton-coupling. So, what we want to do is that on the CFD side, we compute both what the flow would be at the next step and we compute the derivative of this flow versus the input and output pressure, and then, we put both into the system code, we input both the result itself and its derivatives, and so then, we can do the solution on the system side and come back.

The answer is that the type of coupling I showed does not work in two-phase flows. The thing we are working on is a coupling where you do not have an overlapping between codes like this, and at the same time, you need more monolithic tighter procedure to couple between the codes. You need to exchange more data.

Berta Oates

Thank you. There was another question similar. We have the simulation issues when there is two-phase flow due to metal boiling, but I believe you've just addressed that. Do you have any additional thoughts?

Antoine Gerschenfeld

These were the issues related to cold coupling for two-phase flow, but of course, 2-phase flow also needs to be modeled on the thermal-hydraulics side, and this is very complicated. So, the first thing is that usually the models we have initially come from our physical understanding of water flows. And so, we need to change them completely in order to do boiling

sodium. For instance, in sodium, there is almost no critical efflux. It's very rare to encounter critical efflux. At the same time, because the liquid to vapor density ratio is very high, around 2000, it means that almost as soon as there is boiling somewhere, immediately you will have some very large pockets of gas taking all the space in the subassembly. And so, all the physical models for this, so two-phase friction, two-phase heat transfer, the flow map and so on, need to be redone. Actually, in this part, we are collaborating as much as we can with IPPE in Russia on our understanding of two-phase flow and this is because they have an ongoing experiment on two-phase flow in IPPE.

Another important point is that if you have boiling sodium in the core, then there will be a very large effect on the neutronics. Normally, if you have an old type sodium fast reactor like Phenix or Superphenix in France, these reactors have positive reactivity effect. As soon as the coolant starts to boil, the reactivity increases, the boiling accelerates, and you have power restoration in less than 1 second.

On the other hand, in some more recent reactor design like BN-1200 or ASTRID was also of this type, we used devices like sodium plenum above the core. In this case, when the sodium burns, this plenum will be replaced with vapor, and this will increase neutron leakage. With this, you can have instead of a positive global void effect, you can get to a negative global void effect. This means that as soon as the boiling starts, the power will go down. But it will not go down the same way everywhere. It will go down faster in the areas where the sodium is boiling. So, it will be localized in a few subassemblies, for instance.

So, we need to account for very complicated coupled effects between neutronics and thermal-hydraulics, usually in coding 3D effects.

The final one is that the fuel mechanics will also be impacted but the fuel mechanics determines the fuel temperature and then the Doppler effects. You need to know what is going to happen to the fuel thermomechanical properties during the transient, and this is also a very big issue for correctly modeling a transient where the sodium burns.

Berta Oates

Thank you. As for the load-following rate, what are the conclusions of your studies for SFRs? Are material issues different than those with LWRs?

Antoine Gerschenfeld

Our main experience with SFRs is that usually it's simpler than an LWR. For instance, when you look at the physical model, the complexity in them, and so on, they are actually simpler for SFRs or LWRs. The only complexity comes when because of the pool type design, you have

complicated 3D phenomenon, and then, you need to do something more. But for instance, when we were considering the old safety related transients for the ASTRID project, you have all these transients where you still have the pumps operating. These are at nominal or on diesel power. And for all these situations, we could use our system code and get good results and have validated physical models and so on. A big issue was that the models are simpler, but you still need validation experiments. So, we needed to do a lot of literature reviews to get the best model we could. And also, we sometimes end up with the idea that we needed to commission a new experiment for one part or another.

But overall, coming from LWR to SFRs on the system side, the big issue is this pool-type design. This is actually something you see as well in water-type small and modular reactors. Like today, we have designs for water cooled pool type SMRs, so the same areas in SFR actually, except you have water in the middle. And if you use the pool-type design in water as well, you encounter the same problems with complicated 3D flows. So, it's been an interesting journey.

Berta Oates

Thank you. What is the current status of validation of physical models?

Antoine Gerschenfeld

In our current status, I would say that we have had satisfactory like what we call 'separate effect validation,' so validation of each physical model individually. We managed to get good coverage in the system code and so on. The big exception from that was if you are at the system scale, if you introduce a new component design, then we would have needed related experiment, a scaled-down experiment of this particular geometry. So, there was the idea that we have good general radiation, but maybe we will need some design specific validation if there was some geometry change.

For the validation of CFD and coupling, I think using all these different experiments and reactor validation and so on, we had reached a good level of validation for a reactor scale application of this couple simulation. Actually, this type of couple situations were a reference choice for the safety report of ASTRID, for instance.

We would have needed more in case of reactor changes maybe, but I don't think it would have been much. And then, of course, as we receive more reactor validation, it's always good to add it as well.

The big points where we need to do predictions and we do not have enough validation at all is 2-phase flow. On two-phase flow, we need more experimental data for the physical models. We have done as much as we could. We need more. We need experimental validation for the

coupled effects between thermal-hydraulics and neutronics and fluid mechanics. We need this to be able to justify that in a future sodium fast reactor, if the coolant boils, it will not lead to a severe accident.

Actually, for this the FFTF benchmark is very useful, because in FFTF, you have the outside core like ASTRID, for instance, an outside core of a very unfavorable Doppler effect in case of a loss of flow. In FFTF, they did an unprotected transient anyway and [Unclear]. The way they did it is that they added some extra devices on the outside of the reactor and so these devices could mitigate what would have been otherwise power escalation and instead the reactor power went down all the way. And so, with this type of reactor scale multi-physics validation, we may reach a situation where we can have validated two-phase flow simulations. But this is the big work we have for the next 2 years.

Berta Oates

Thank you. Antione, I have forwarded you some discussion that's been posted. I don't see a question in there. Is that something you can just read and share your thoughts?

Antoine Gerschenfeld

I see a question on the – I see a remark from someone on the question side, and it shows that if you are looking at molten salt as just a coolant and not a fuel, the corrosion problem will be much easier.

Berta Oates

It looks like the last question I see is I have heard of sodium heat pipes and NASA reactors, is there any opportunity to remove decay heat using heat pipes?

Antoine Gerschenfeld

In our case, I would say that actually if you – so for this decay heat removal circuit, actually just using liquid sodium, you already get a very high performance heat exchanger. So, I guess if you have like sodium cooled SMR, so something smaller, then there may be an advantage in using a single pipe between the primary pool and the heat sink. So, this is something you could do with the heat pipe instead of a full circuit with two legs.

On the other hand, for instance, even if you have a very large sodium fast reactor, if you have like the ESFR or SMART project in Europe, that's a project for 1.5 gigawatt sodium fast reactor, and you only need I think – yeah, you can remove something like 6 or 7 megawatts per heat exchanger in the chimney. And in this it is easier to just do 2 legs and keep liquid sodium. A thing that may be complicated with the heat pipe is that if you want liquid sodium on the primary side and you put a heat pipe

in, you will need to put the sodium at less than ambient pressure and you may have issues about ingress of air inside the pipe, something like that.

So, I think it's easier to just have a liquid circuit and its own little purification and sodium. But for a small reactor it could be interesting.

Berta Oates

We have received lots of accolades and thank you for the wonderful presentation today. Thank you again for showing your expertise with us. It's always good to see so many engaged participants with the number of questions. Very interesting topic. Thank you.

Patricia Paviet

Thank you, again, Antoine. It was really good. And what is really interesting is this Q&A session. A strong momentum, good dynamics. Thank you, Antoine.

Antoine Gerschenfeld

Thank you.

Berta Oates

With that, we will conclude this presentation. Thank you all for joining.

Patricia Paviet

Thank you. Bye-bye

Antoine Gerschenfeld

Good-bye.

END
