HOT FUEL ELEMENT THERMAL-HYDRAULIC MODELING IN THE JULES HOROWITZ REACTOR NOMINAL AND LOFA CONDITIONS

R. Pegonen and H. Anglart
Nuclear Reactor Technology, Department of Physics
KTH Royal Institute of Technology
Roslagstullsbacken 21, SE-10691 Stockholm, Sweden
pegonen@kth.se; henryk@kth.se

S.Bourdon and C. Gonnier
CEA, DEN, DER, SRJH
CEA Cadarache, 13108 Saint-Paul-lez-Durance Cedex, France
serge.bourdon@cea.fr; christian.gonnier@cea.fr

ABSTRACT

The newest European high performance material testing reactor, the Jules Horowitz Reactor, is under construction at CEA Cadarache research center in France. The reactor will support the existing and future nuclear reactor technologies and will start operation at the end of the decade.

The current CEA methodology for simulating the thermal-hydraulic behavior of the reactor gives reliable results. Today the CATHARE2 code simulates the full reactor with a simplified approach for the core and the boundary conditions are transferred into the three-dimensional FLICA4 core simulation. However this procedure needs to be further improved and simplified to shorten the computational time and to give more accurate core level data. Specific CFD calculations will better identify the thermal-hydraulics phenomena and optimize the meshing/model of the improved procedure.

This article presents the current one-coupled thermal-hydraulic modeling of the reactor utilizing the system code CATHARE2 and the core analysis code FLICA4 and describes the more realistic new hot fuel element modeling by using CFD code STAR-CCM+ including conjugate heat transfer. Finally, the results from the both modeling are compared in the hot channel in the nominal condition and in the case of LOFA.

This study has improved the thermal-hydraulic knowledge of the complex hot fuel element and the most prominent finds are presented. In addition, the possible improvements for the more realistic CATHARE2's core model are proposed. In all simulations the safety criteria were satisfied, the reactor stayed in the single-phase regime and overall integrity of the fuel plate was ensured.

KEYWORDS
Jules Horowitz Reactor, CATHARE2, FLICA4, STAR-CCM+, loss of flow accident

1. INTRODUCTION

There is a necessity in Europe for at least one new material testing reactor (MTR) to support the existing and future nuclear reactor technologies. The current aging MTR's fleet is over half a century old. The newest high performance MRT, the Jules Horowitz Reactor (JHR), meets the nuclear industries latest safety standards and is under construction in France. It is expected to start operation at the end of the decade.

The aim of this paper is to present the current CEA one-coupled thermal-hydraulic modeling of the reactor [1] utilizing the system code CATHARE2 and the core analysis code FLICA4, and also to describe the more realistic new hot fuel element modeling by using CFD code STAR-CCM+ including conjugate heat transfer. The results from the both modeling are compared in the hot channel in the nominal condition and in the case of loss of flow accident (LOFA).

This paper is part of an ongoing four year project aimed at the development of an improved JHR CATHARE2 model, where the simplified core model will be replaced with a more complex model, similar to the approach used in FLICA4 core modeling. New single tool model will shorten the computational time and will provide more accurate core level data.

In addition, modeling with the “sub-channel codes” is usually done using “best practice” and code developer recommendations but this approach could lead to non-optimized modeling (number and size of channels, 2D or 3D effects,…). Therefore, the present approach utilizes a more geometrically detailed CFD model (and a comparison with FLICA4 model) in order to find possible improvements for the future CATHARE2 model. It is expected that the CFD approach (STAR-CCM+) will give more reliable results than the sub-channel codes currently utilized (FLICA4).

Furthermore, safety calculations are performed (with CATHARE2/FLICA4 codes) with pessimistic assumptions and it is generally difficult to evaluate the amount of pessimism. This work with the CFD approach proposes to calculate the safety margin under similar thermal-hydraulic conditions (in the hot channel) compared to those performed with the current methodology, in order to evaluate the penalty induced by these pessimistic assumptions.

This paper first gives a short overview of the JHR and describes the modeling done with CATHARE2, FLICA4 and STAR-CCM+. In continuation, the modeling assumptions are introduced. Next an analysis of the core performance during nominal conditions and during LOFA is described. Finally, the most prominent conclusions and the possible future work are presented.

2. THE JULES HOROWITZ REACTOR

The Jules Horowitz Reactor is a new high-performance material-testing reactor currently under construction at the CEA (Alternative and Atomic Energy Commission) in Cadarache, France. The JHR project is an international cooperation project involving several organizations. The maximum core power of this pool-type reactor is 100 MW_{th} and the light water will be utilized for cooling and for moderation [2].

The JHR will be used to investigate the behavior of nuclear materials and fuels under irradiation and to produce medical radioisotopes (e.g. ^{99}Mo). The reactor's flexible high-performance experimental capacity will support existing and future nuclear reactor design. It will provide a high neutron flux- twice as large as the maximum available today in European MTRs [3]. In the core, neutron fluxes can achieve up to 10^{15} $n/cm^2/s$ ($E > 0.1$ MeV) [4].

The reactor block is located at the bottom of the reactor pool and the core is about 9 meters below the water level. The cylindrical core has a diameter of 71 cm and is 60 cm high. The 30 cm thick beryllium reflector surrounds the reactor block. The core and the reflector have separate cooling systems- the core is cooled by the forced convection primary cooling system. The pressure and the flow inside the core are in the order of 1.0 MPa and 7400 m^3/h [5].

The core is situated in an aluminum rack with 37 possible positions for fuel elements, 34 to 37 occupied by fuel elements. Each fuel element, see Fig. 1, consists of eight circular rings of curved fuel plates assembled with three Al 6061-T6 stiffeners. The fuel assembly has external diameter of 97.7 mm and the hydraulic gap between fuel plates is 1.95 mm wide [6]. Each fuel plate comprises of AlFeNi cladding, U_3Si_2/Al fuel and borated aluminum poison at the top end of the fuel element. The total height of the fuel plate is 700 mm from which 600 mm is the fuel zone. The overall height of the fuel element is 1015 mm. The top end cap and the bottom end cap have the lengths of 115 mm and 145 mm, respectively.

Figure 1. Isometric view of the JHR fuel element.

Fuel element has a Ø40 mm central hole to host: the guide tube filled by the control rod or by the Al rod; the protection tube and the experimental device. In the core, experimental devices can occupy maximum 7 fuel elements central holes and can replace maximum 3 fuel elements. JHR allows about 20 simultaneous experiments placed into the core or into the reflector [3].

3. MODELING OF JHR

The current CEA methodology performs thermal-hydraulic calculations of the reactor by utilizing the system code CATHARE2 and the core analysis code FLICA4. The full reactor with simplified core model is modeled in CATHARE2 that is one-coupled to the FLICA4 more detailed core model with the boundary conditions obtained from the CATHARE2 model.

In purpose for improving the core modeling in CATHARE2 model a new more realistic CFD model of the hot fuel element including conjugate heat transfer is created by using STAR-CCM+. In the following sections CEA current modeling and CFD model of the hot fuel assembly are described under the nominal condition and under the loss of flow accident condition.

3.1. CATHARE2 model

CATHARE2 is fully modular thermal-hydraulic code and modeling is done with five main modules: the 1D module for pipe flows, the 0D module for volume elements (the pressurizer, the accumulator, the steam generator dome or the lower plenum of a PWR), the 3D module for multidimensional effects in vessel, the boundary condition module and the double ended break module. Sub-modules are used for supplementary models (pumps, valves, T-junctions, sources and sinks, breaks, etc.) and junctions for connecting modules [7].

The modeling of JHR primary circuit with a reactor pool in CATHARE2 is presented in Figs. 2 and 3(left). The reactor block model consists of 1D and 0D modules connected by junctions. 1D modules model a mean core channel (with a weight of 36), a core bypass channel, a gap between the vessel and the core, control rods (3 modules representing 27 rods) and test devices (2 modules representing 10 devices). The upper and the lower plenum are modeled by 0D modules.

The modeled cooling system consists of the reactor pool, the pressurization circuit, the reactor cooling system (three loop circuits), the core safety cooling system (RUC), the reactor pool safety cooling system (RUP) and the two additional safety circuits, see Fig. 2. The reactor has two sets of RUC/RUP systems for redundancy, however only one is modeled. All systems are simulated by using 1D modules in combination with proper sub-modules (heat exchanger, pump, T-junction) and junctions. The 0D module is implemented to model the reactor pool and for connecting several systems (DERIV, JONC).

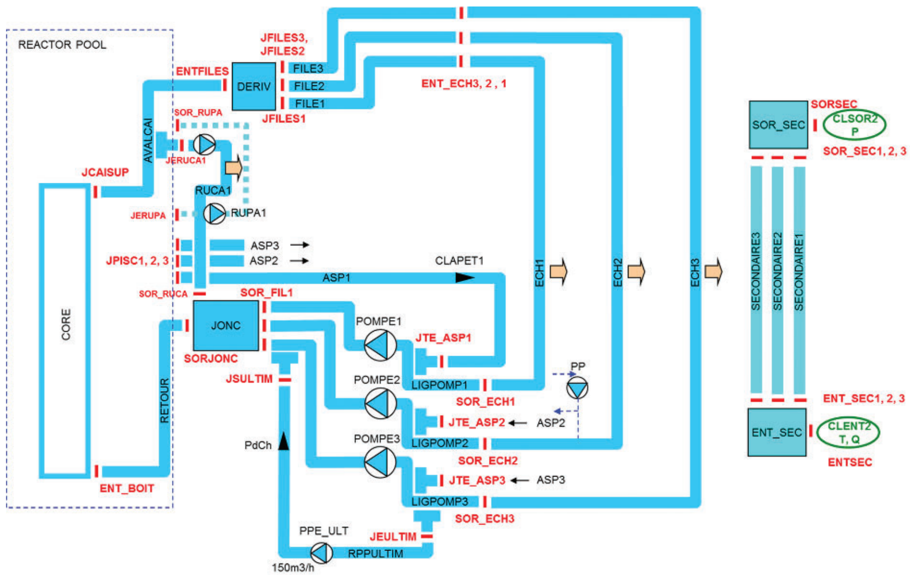

Figure 2. JHR primary and secondary circuits modeled in CATHARE2.

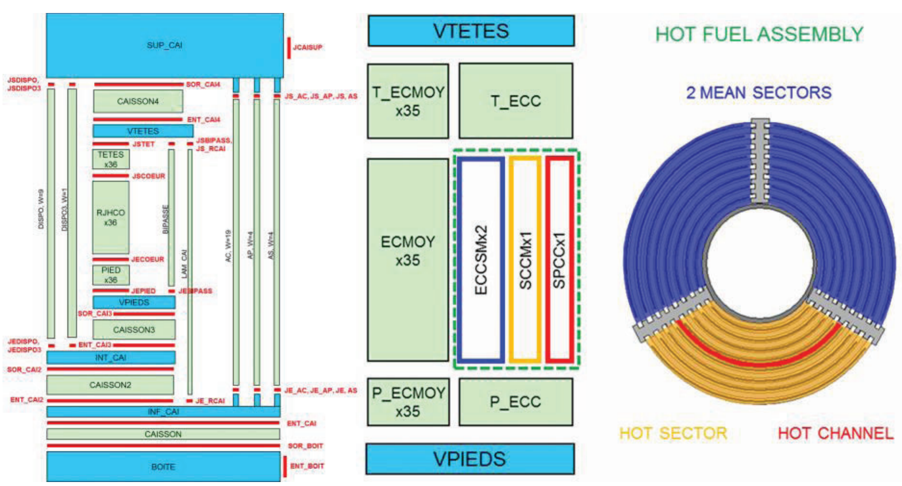

Figure 3. (Left) JHR reactor block modeled in CATHARE2; (middle) JHR core modeled in FLICA4; (right) JHR hot fuel assembly modeled in FLICA4.

The RUC and RUP systems are connected through a heat exchanger transferring energy from the reactor block while using forced convection. The pressurization circuit ensures with the pump and the check valve reference pressure for the primary system and provides a mean for monitoring water transfer between the reactor pool and the primary system. The first safety circuit is employed in case of a primary pump failure and the second allows injecting water from the pool into the primary circuit in case of depressurization.

The primary circuit consists of 53 hydraulic modules (42 pipes, 10 volumes, 1 boundary condition) and the total amount of scalar meshes is 3561. The simplified JHR secondary circuit is modeled by using 7 hydraulic modules (3 pipes, 2 volumes, 2 boundary conditions) with 197 total accounted scalar meshes. In the secondary circuit model each 1D module is equipped with a heat exchanger sub-module.

From the CATHARE2 results, core power, inlet void fraction, inlet mass flow, inlet enthalpy and outlet pressure from each time step are imposed as boundary conditions in the FLICA4 3D thermal-hydraulic code simulation, where only the reactor core is modeled.

3.2. FLICA4 model

The FLICA4 core simulations are using two-level method described in [8]. In the first level the whole core with 36 fuel assemblies is calculated and boundary conditions from the hot fuel assembly are deduced for the second level, where only the hot fuel assembly is simulated.

In the first level simulation the core is divided radially into 36 elements (35 mean and a hot fuel assembly) and axially into five zones: inlet, bottom zone, fuel zone, top zone and outlet. The fuel zone has axially 35 mesh elements, the rest have 8 elements per zone. Each fuel element is radially divided into three sectors containing 9 water channels and 8 fuel plates. Each sector is further radially divided into four equal parts (36 water cells in total per sector). Three central axial zones (bottom zone, fuel zone, top zone) form the fuel element.

In the second level, the hot fuel assembly is divided into three sectors: two mean sectors and the hot sector, see Figure 3(middle-right). Thereafter the hot sector is split into the hot channel and into one or two mean channels, depending on where the hot channel is situated. Finally the hot channel is divided into four sub-channels. In this step the fuel zone has 35 axial mesh elements, but in other zones it is reduced by the factor of two.

3.3. STAR-CCM+ model

The design of the geometry used in this work is based on the 2011 technical drawing of the JHR fuel element [6]. In order to maintain a reasonable mesh cell count extremely small irrelevant details/gaps were neglected. For this research two CAD models were created: the water geometry of the fuel element inside the rack, and the hot channel's assembly (water + metal parts). The first geometry broadens our understanding of the conditions in the hot fuel assembly and deduces the boundary conditions for the second geometry utilized to evaluate the conditions in the hot channel.

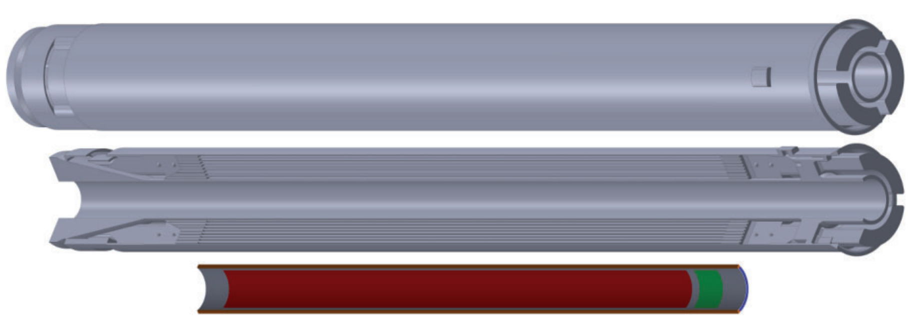

Figure 4. Isometric views of the following CFD geometries: (top) water geometry between the rack and the fuel assembly and its 2/3 view (middle); (bottom) the hot channels assembly for conjugate heat transfer calculation.

The hot channel's geometry assembly, see Fig. 4(bottom), represents the second water channel from the center of fuel assembly with its surrounding structural materials and consists of nine parts: the water, the inner and the outer cladding/fuel meat/poison and two stiffeners. In order to simplify the model, symmetry of the heat transfer has been used. Therefore the modeled geometry consists of one water channel; surrounded by structural materials, of which only half is modeled, see Fig. 4(bottom). The overall height of the assembly is 700 mm. In the case of the first geometry, see Fig. 4(top-middle), the geometries inlet was prolonged by 10 mm annulus for the numerical reason and the total height of the geometry is 1078 mm. It is impossible to model only one part of the geometry due to the asymmetry of the fuel assembly and of the thermal-hydraulic conditions.

The fluid region is assumed to be three dimensional, steady and turbulent. Segregated flow, segregated fluid temperature, realizable k-ε turbulence model, and gravity physical models as well as two-layer all y+ wall treatment are used. The last is designed to give accurate results regardless of the sub-layer of the turbulent boundary layer in which the near-wall centroid is located in. In the hot channel's simulation each metal part has a separate region and they are assumed to have constant density and to be three dimensional and steady, furthermore segregated energy model is utilized.

The step-wise approach was used during the steady-state calculations. First the constant density simulation was carried out, then the IAPWS-IF97 water properties were added and in the last step power table/functions were included to perform simulation with the desired (nominal or certain time spot during the LOFA) operating conditions.

The current best practice is to utilize a conformal mesh for conjugate heat transfer and it can be only created by polyhedral mesher [9]. Furthermore the meshing procedure for the complex geometry (fuel assembly) should be fully automatic to produce optimal mesh and to save time. Therefore in this study unstructured polyhedral mesh with surface remesher and prism layers (except from the solids) was used.

To study the influence of the mesh size several meshes of the first geometry were automatically created by using the same topology by just reducing the base size from 6 mm to 3 mm and by creating meshes from 8,516,491 cells to 23,211,863 cells. The prism layers physics were left unaltered by specifying its parameters in absolute rather than relative terms. As a convergence criterion the overall pressure drop was monitored with different meshes. It was observed that by utilizing the mesh larger than 16.5 million cells the total pressure drop varies less than 0.3% compared to the finest mesh. Although mesh with 16.5 million cells would have been sufficient, the finest mesh was chosen for the simulation to have the most cells computationally available (limited by the RAM) for the best possible resolution. The second geometry is modeling only its channel including the solid parts. Therefore it is valid to use identical meshing parameters for meshing geometries two and three. As in the case of the first geometry the finest mesh possible was generated.

Table I. Mesh details

	Mesh 1 (full fuel element water)		Mesh 2 (hot channel + structural mat.)	
	Cells	**Faces**	**Cells**	**Faces**
Water	23,211,863	96,899,148	11,110,954	51,853,736
Cladding 1/2	-	-	2,084,863/2,219,125	10,871,080/11,566,052
Fuel 1/2	-	-	1,427,151/1,585,793	7,423,714/8,249,086
Poison 1/2	-	-	71,808/79,780	372,594/413,264
Stiffener 1/2	-	-	325,485/327,565	1,805,268/1,818,713

The total number of cells is presented in Table I. Two sets of meshing reference values were used: coarser values for the mesh 1 (for modeling the water geometry of the whole full fuel element inside the rack) and finer values for mesh 2 (for modeling the hot channel). There were five prism layers along the walls in the fluid domain to resolve the boundary layer and it is considered sufficient when wall functions are included.

Prerequisite of a good volume mesh is an error free surface mesh, containing valid elements. Mesh quality can be described by assessing the face validity and the volume change of the cell. The first is an area-weighted measure of the correctness of the face normals relative to their attached cell centroid and the value 1.0 means that all face normals are correctly pointing away from the cell centroid [10]. Values lower than 0.5 identify a negative volume cell. At the same time the volume change metric describes the ratio of the volume of a cell to that of its largest neighbor and the cells with a value of 10^{-5} or below should be investigated further [10]. In all three grids generated the face validity overall value was 1.0 and the volume change had values above 0.001 and most of the cells above 0.1.

3.4. FLICA4 and STAR-CCM+ modeling assumptions

The following are the assumptions made in both models:

- The core reference power is 100 MW·1.065·1.03·0.9916= 108.77 MW, where the first factor in multiplication is the core nominal power, the second and the third take into account power measurement uncertainties and operating range. The final factor describes 0.84% power loss due to experimental devices and internal structures of the core.
- The power is distributed among 36 elements in a way that the hot fuel element and each of the 35 mean fuel elements have the normalized factors of 1.705 and 0.980, respectively [11]. The hot channel power density peaking factor is: $F_{3D}=F_{element}\cdot F_{axial}\cdot F_{radial}\cdot F_{azimuthal}= 1.705\cdot 1.295\cdot 1.243\cdot 1.093=3.001$, where Felement is the factor representing power distribution between fuel elements, F_{axial}, F_{radial} and $F_{azimuthal}$ are the axial, radial and azimuthal hot channel factors. Notice that the calculation presented in this paper uses the more realistic power distribution than those utilized in the JHR safety analyses.
- Taking into account the power distribution, the hot channel is the second channel from the center of fuel assembly in the hot sector.
- In case of reactor scram the power evolution profile corresponds to the situation where three emergency stop control rods are inserted and it is assumed that the mechanism of the 4th, the most effective rod, has malfunctioned.

The following are the additional assumptions made in the CEA's FLICA4 model of the 36 assembly JHR core:

- Turbulence is not modeled.
- The hot channel's hydraulic gap is reduced from 1.95 mm to 1.64 mm to take into account manufacturing tolerances and the operational effect on the fuel plates (swelling, oxidation, thermal expansion, etc.) In addition its mass flow is further reduced by 4% by increasing the channel's inlet local pressure loss coefficient. It is calculated in cold conditions under nominal pumping and is due to inhomogeneity of the flow in the lower plenum and the channel's input velocities.

The following are the additional assumptions made in the CFD simulations:

- Turbulence is modeled.
- The total power of the hot fuel element is 4.900 MW and 0.251 MW is the power in the water, in the hot channel, originating from neutron moderation and gamma heating. In addition it is assumed that

only the fuel meat and the water are the source of power. In the hot channel calculation it is assumed that both fuel plates contribute 1/2 of the power into the hot channel: 74.5 kW from the first and 80.9 kW from the second fuel plate. In addition 6.0 kW is the power in water due to neutrons and gamma rays.

- All parts are considered geometrically new and no manufacturing tolerances or operational effects (oxidation, swelling, etc.) on the fuel plate are taken into account.
- The total mass flow rate in the core is 1546.4 kg/s, from which 42.955 kg/s (1/36th) is assumed to flow through the hot fuel element. In the future this value will be corrected according to the results of the JHR lower plenum hydraulic CFD calculation.

4. RESULTS

The following subsections present the comparison between the STAR-CCM+ and FLICA4 calculations under the same flow conditions in the hot channel. Thereafter transient results of the LOFA are presented. Notice that the calculation presented in this paper uses the more realistic power distribution than those utilized in the JHR safety analyses and the CEA 36 assembly configuration.

4.1. STAR-CCM+ and FLICA4 comparison in the hot channel

The first step of the analysis is to compare the STAR-CCM+ results in the hot channel to those obtained using the current CEA methodology FLICA4 model, without considering the reduction of the hydraulic gap from 1.95 mm to 1.64 mm. Both simulations have been performed with similar mass flow rate in the hot channel. The results are summarized in Table II.

Table II. STAR-CCM+ and FLICA4 results in the hot channel

	STAR-CCM+	FLICA4	
Max wall temperature	394.2	412.5	K
Max cladding temperature	496.6	535.3	K
Max fuel temperature	570.0	604.3	K
Min safety margin	M= 71.8	M= 56.1	K
Max heat flux	4.11	4.52	MW/m^2
Hot channel's mass flow rate	0.924	0.925	kg/s
Surface area	88.7	88.8	mm^2

* For safety margin M, see Appendix A.

Comparison of the result data reveals that the minimum safety margin is larger when calculated with CFD. This is because of: i) the heat transfer exchange is enhanced in the CFD calculation compared to the FLICA4 calculation utilizing the Dittus-Boelter correlation, and ii) a smaller maximum heat flux value (-9%), due to the more accurate 3D thermal conduction in CFD.

Figure 5 shows the hot channel outlet temperature distribution in the CFD simulations. In the CEA current methodology the full hot channel is divided into 4 sub-channels. From Fig. 5 it can be observed that the temperature could be considered piece wisely constant in azimuthal direction and the channel splitting into sub-channels is justified and the similar method could be used in the future CATHARE2 modeling of the JHR with more accurate core model. At the both edges, temperatures are changing more rapidly; therefore it would be advisable to either increase the channel subdivisions in the full channel or only in the edges.

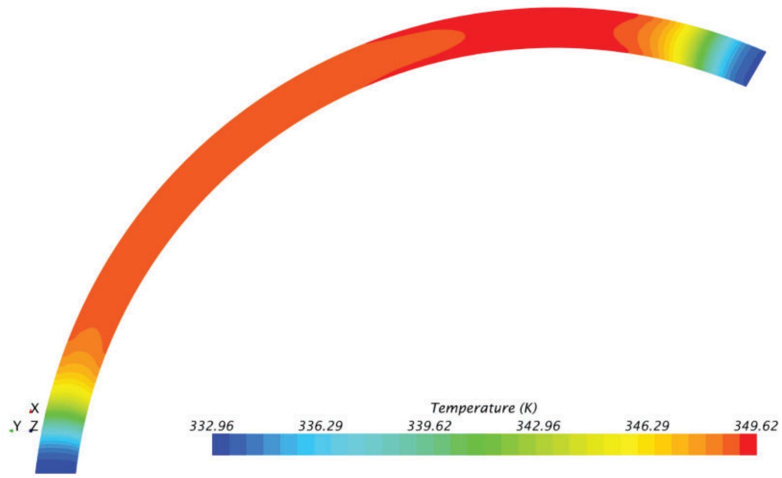

Figure 5. Hot channel outlet (z= 0.7 m) temperature distribution in the CFD simulation.

There is a difference in the temperatures after the vertical height equal to the upper edge of the fuel meat. In FLICA4 simulation the temperatures in all 4 sub-channels stay at the constant temperature, while in the CFD simulation turbulent mixing causes the water temperatures to smooth the temperatures. Therefore the constant maximum water temperature in FLICA4 is located vertically within an interval before the outlet and in the CFD model at vertical height equal to the upper edge of the fuel meat. Although the current modeling provides reliable safety analyses results, in future CATHARE2 modeling of the JHR, with more accurate core model, this difference could be taken into account.

4.2. Loss of flow accident

In this subsection a loss of flow accident scenario is simulated. At first the transient is calculated in CATHARE2 and afterwards it is one-coupled with FLICA4 model. Thereafter specific moments in time are simulated in STAR-CCM+ core model in steady-state mode. Input data for CFD calculation is retrieved from CATHARE2 simulation. The following moment are picked: time spot that correspond to the maximum fuel and cladding temperatures and the moment where the minimum safety margin is obtained, both obtained in FLICA4 simulation.

4.2.1. Scenario

The simulation starts with immediate loss of external power that leads to the rotational speed decrease of the three primary pumps and the pressurization pump. Pumps are totally stopped in 600 s leading to the loss of the heat exchangers. Once the pressure drop in the core reaches the first threshold (Δpcore= 0.290 MPa) the reactor is scrammed with 0.6 s delay. After the second threshold (Δpcore= 0.200 MPa), the signal to activate the safety circuit RUC/RUP (run on batteries) is given with a delay of 20 s.

4.2.2. CATHARE2/FLICA4 transient

Before emergency stop:

In the FLICA4 model the hot channel is modeled by 4 sub-channels and at the outlet at t= 0 s, each channel has the following temperatures: sub-channel 1- 358.0 K, sub-channel 2- 356.6 K, sub-channel 3- 356.2 K, sub-channel 4- 353.0 K. The average temperature is 355.95 K giving with inlet temperature of 305.25 K the hot channel temperature increase of 50.7 K. For comparison, the average hot channel outlet temperature and the temperature raise in the CFD simulation are 346.9 K and 41.7 K.

The simulation results show that the core outlet temperature is slightly increasing due to the reduction of flow and the pressure is significantly decreasing due to the pumps stopping, see Fig. 6(left). The core inlet temperature remains uninfluenced. After 3.9 s from the start the first pressure drop threshold is reached and the mass flow rate at the hot channel's inlet has fallen to 0.706 kg/s. The signal to scram the reactor is given at t= 4.5 s (the hot channel's inlet mass flow rate 0.697 kg/s). At the same time the maximum cladding and fuel temperatures ($T_{cladding,max}$= 543.3 K (570.3 K with uncertainties), $T_{fuel,max}$= 612.1 K (644.0 K with uncertainties)) are obtained. The criterion for avoiding buckling ($T_{cladding}$< 673 K [12]) as well as melting ($T_{cladding,melt}$= 889 K, $T_{fuel,melt}$= 918 K [12]) was fulfilled. Safety margins have values M= 43.8 K and M_I= 27.6 K.

After emergency stop:

The core outlet temperature decreases sharply due to sharp power reduction. The pressure continues to decrease until the final state is reached (core outlet pressure $p_{core,out}$= 0.175 MPa). Safety margins M and M_I reach their local minimum values (M= 41.4 K, M_I= 25.7 K) at t= 4.7 s, see Fig. 6(right). The DNBR (departure from nucleate boiling ratio) has minimum value of 1.529 at t= 4.65 s. After the reactor scram the maximum heat flux drops within a second sharply from 4.53 MW/m^2 to 1.18 MW/m^2 and continues to decrease, see Figure 6(right).

The second pressure drop threshold is reached at t=14.3 s (the hot channel's inlet mass flow rate 0.530 kg/s) and the safety circuit RUC/RUP is executed at 34.3 s, see Figure 6(left). At first it causes the core inlet temperature to rise due to the initial temperature in RUC. The inlet and the outlet temperatures continue to rise until t= 1297 s ($T_{in,max}$= 333.7 K) and t= 1176 s ($T_{out,max}$= 340.4 K), respectively, and start to decrease thereafter. The rise is due to the residual power, the thermal inertia of the RUC circuit and the heat removal capacity in the RUC/RUP heat exchanger. There is a short period of about 35 s when the core outlet (or RUC) temperature is still below the reactor pool (or RUP) temperature and the RUC/RUP heat exchanger is heating the primary circuit, see Fig. 6(left).

The core flow rate changes from 1546.4 kg/s to 74.8 kg/s (the hot channel's inlet mass flow rate 0.038 kg/s) in 147 s. At the same time safety margins obtain their global minimum values (M= 23.2 K, M_I= 12.7 K) and the maximum cladding and fuel temperatures are $T_{cladding,max147s}$= 382.6 K (396.3 K with uncertainties) and $T_{fuel,max147s}$= 385.0 K (398.8 K with uncertainties). Afterwards the flow rate decreases only by 0.8 kg/s in 453 s before the total stop of the pumps. Fig. 6(left) illustrates that the heat exchangers capacity is high enough to remove the heat generated in the reactor until t= 111 s. At t= 178 s the heat removal capacity of the safety circuit RUC/RUP becomes dominant and from t= 1274 s it surpasses the heat produced by the reactor. It can be concluded that there is a time interval of 1163 s when the reactor is under-cooled and the temperature in the primary circuit rises.

It has been observed that the pressure drop decreases continuously except at two points, i) at the time of the reactor scram the pressure drop spikes by 6 kPa, and ii) the pressure drop remains nearly constant when the RUC/RUP safety circuit initiation affects the core. Both of these events last for less than a second.

The obtained results indicate that the main safety criterion is satisfied and the reactor stayed in the single-phase regime with the minimum safety margin of 12.7 K including the uncertainties. Moreover the thermo-mechanical criterion demonstrates the absence of buckling as well as the overall integrity of the fuel plates and cladding.

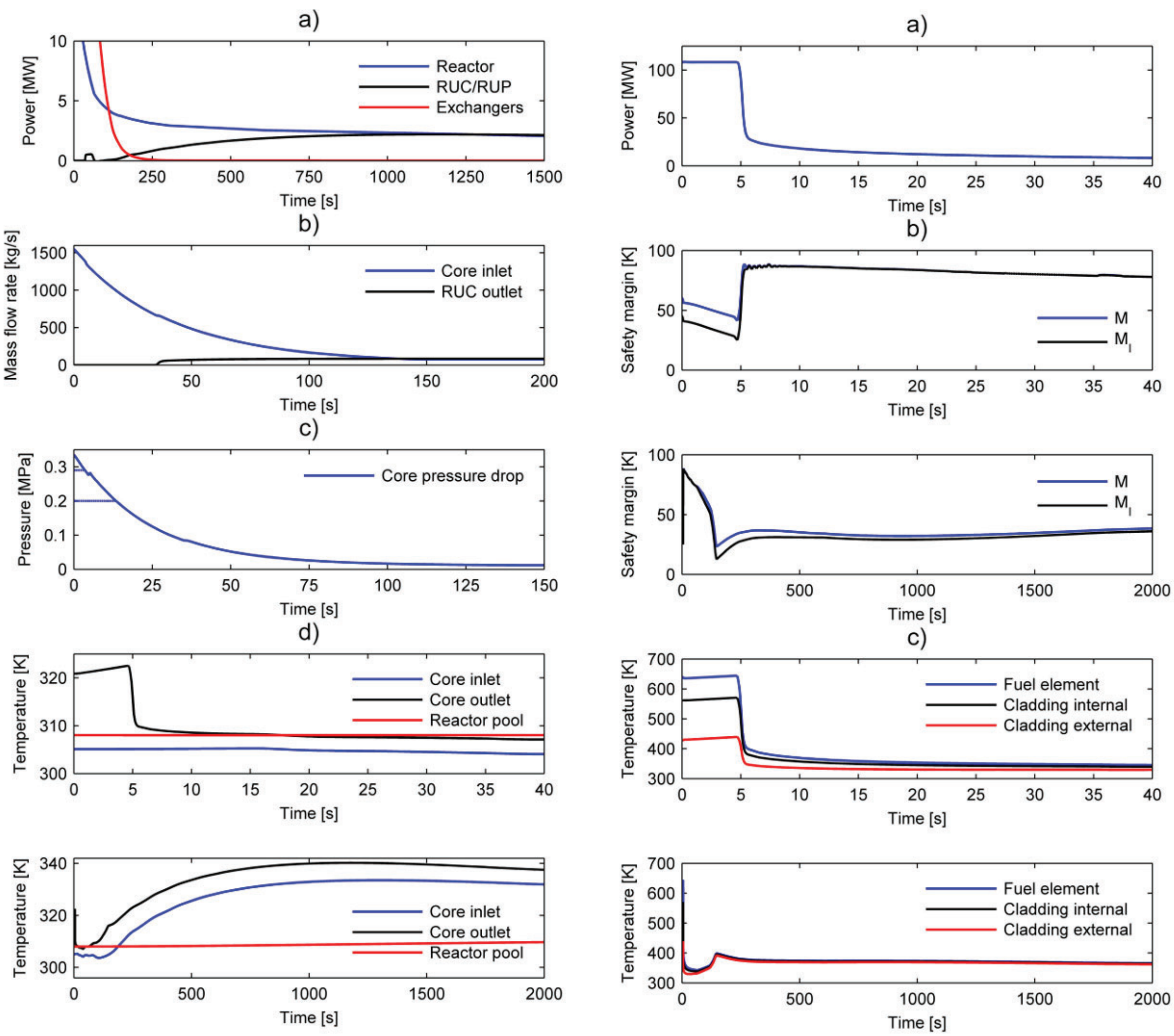

Figure 6. (Left) CATHARE2 simulation time series of: (a) power, (b) mass flow rate, (c) pressure drop and (d) temperature; (right) FLICA4 core simulation time series of: (a) power, (b) safety margin, (c) fuel element and cladding maximum temperatures.

4.2.3. STAR-CCM+ specific time moments

In addition to the nominal conditions steady-state simulation with both geometries was carried out at the time moments of 4.5 s (the maximum cladding and fuel temperatures) and at the moment of 147.0 s (the minimum safety margin). This is a first approach using STAR-CCM+ with nominal geometry of the fuel assembly. Therefore this is not a conservative approach. In the future, this analysis should be completed by CFD calculations introducing the operational effect of the fuel plates.

These steady-state simulations may introduce some distortions by ignoring some transient parameters (e.g. stored energy in the structure/water, water inertia) compared to the results that could be obtained directly from transient calculations.

The results are summarized in Table III. At t= 4.5 s the maximum cladding temperature $T_{CFDcladding,max}$= 503.3 K and the maximum fuel temperature $T_{CFDfuel,max}$= 576.7 K are as expected below the value obtained in FLICA4 core simulation by 40.0 K and 35.4 K, respectively. Correspondingly the buckling as well as the melting of the cladding and the fuel is avoided. The minimum safety margin 59.7 K is 15.9 K higher than M and 32.1 K higher than M_I obtained in FLICA4 core calculation. The time moment t= 4.5 s corresponds in the both core simulations to the case of the maximum temperatures due to the decreased mass flow and signal given to scram the reactor.

The minimum safety margin calculated at t= 147.0 s is 53.8 K. An additional transient CFD simulation would be required for locating the exact time where the minimum safety margin exists with this CFD approach. Although a transient CFD would give the minimum safety margin time moment instantly, it is prohibitively “expensive” in computational time, to simulate 140-175 s in a model with more than 20 million elements, which could not be afforded in this work.

At t= 147.0 s the maximum cladding and fuel temperatures are 34.1 K and 34.2 K below FLICA4 core simulation values. The mass flow rate in the hot channel has fallen in 147 s from 0.924 kg/s to 0.046 kg/s.

Table III. Simulation results

	STAR-CCM+		FLICA4		
	t= 4.5 s	**t= 147.0 s**	**t= 4.5 s**	**t= 147.0 s**	
Max wall temperature	401.3	345.6	420.9	378.4	K
Max cladding temperature	503.3	348.5	543.3	382.6	K
Max fuel temperature	576.7	350.8	612.1	385.0	K
Min safety margin (M)	59.7	53.8	43.8	23.2	K
Max heat flux	4.11	0.14	4.53	0.16	MW/m^2
Hot channel’s mass flow rate	0.834	0.046	0.697	0.038	kg/s

* For safety margin M, see Appendix A.

5. CONCLUSIONS

From the research that has been conducted, it is possible to conclude that:

- The use of CFD provides larger safety margin under similar thermal-hydraulic conditions in the hot channel compared to the FLICA4 calculation.

- Under the proposed scenario the JHR safety margin criteria are satisfied and the reactor stays in the single-phase regime.
- The thermo-mechanical criterion demonstrates the absence of buckling as well as the overall integrity of the fuel plates and cladding.
- The fluid temperature increase in the hot channel is 50.7 K in the case of FLICA4 simulation and 41.7 K in the case of CFD calculation.
- It has been found that during 1163 s the reactor is under-cooled.

The current CEA methodology for calculating the thermal-hydraulic behavior of the reactor during accidental transients provides reliable safety analysis results, however the current CATHARE2's core model could be improved, to utilized a single tool giving more realistic, but still conservative results.

6. IMPROVEMENTS AND FUTURE WORK

The proposed improvements for the future CATHARE2 model with more accurate core modelling could be:

- Replacing the mean fuel assembly with a similar approach used in the FLICA4 core modeling. In the future more detailed core model will be implemented.
- Channel could be split into sub-channels as it is current done in FLICA4 modeling and it is advised to increase the number of sub-channels for increasing the modeling accuracy.
- A more realistic heat transfer coefficient model should be introduced.
- Taking into account 3D effects on the heat conduction in the plates that result in local reduction of the heat flux
- The flow temperature mixing after passing the fuel meat zone should be more accurately modeled, for avoiding the maximum fluid temperatures at the outlet of the hot channel.
- Improving the water power modeling due to neutrons and gamma rays. Currently all the power is located within the fuel meat zone. In improved model, the power should have more realistic distribution within the entire fuel element.

The future work should concentrate on:

- CFD modeling of the upper plenum and the lower plenum of the JHR, in order to assess the initial modeling assumptions and to detect possible necessity for more accurate modeling.
- Simulating the hot fuel element and the hot channel in CFD in transient mode.
- Adding the operational effects into CFD model for simulating operational fuel assembly.

A journal article describing the thermal-hydraulics of the hot fuel element in the Jules Horowitz Reactor, more in details based on CFD, is planned to follow up this research.

ACKNOWLEDGMENTS

The financial support obtained from the Swedish Research Council through the DEMO-JHR project is gratefully acknowledged by the first author (R. Pegonen).

REFERENCES

1. R. Pegonen, S. Bourdon, C. Gonnier and H. Anglart, "A review of the current thermal-hydraulic modeling of the Jules Horowitz Reactor: A loss of flow accident analysis," *Nuclear Engineering and Design*, **280**, pp. 294-304 (2014).
2. "The Jules Horowitz Reactor (JHR)," http://www.cad.cea.fr/rjh, (2013).

3. G. Bignan, P.Lemoine and X. Bravo, "Jules Horowitz Reactor: A new European MTR (Material Testing Reactor) open to international collaboration: description and status," *Proceedings of European Research Reactor Conference 2011(RRFM 2011)*, Rome, Italy, March 20–24, (2011).
4. C. Gonnier, "The Jules Horowitz research reactor. A new high performance MTR in Europe," *Proceedings of Meeting in IVA Seminar*, Stockholm, Sweden, February 26, (2013).
5. A. Ballagny, Y. Bergamashi, Y. Bouilloux, X. Bravo, B. Guidon, M. Rommens and P. Trémodeux, "Main technical options of the Jules Horowitz Reactor project to achieve high flux performances and high safety level," *Proceedings of the 9th Meeting of the International Group of Research Reactors (IGORR-9)*, Sydney, Australia, March 24–27, (2003).
6. AREVA TA, "Project RJH. Fuel element technical drawings," (2011).
7. CEA, "CATRHARE 2 User Guide – General Description of the CATHARE 2 V2.5_2mod8.1 System Code," (2012).
8. E. Royer and I. Toumi, "CATHARE-CRONOS-FLICA, coupling with ISAS: a powerful tool for nuclear studies," *Proceedings of 6th International Conference on Nuclear Engineering (INCONE 6)*, San Diego, USA, May 10–15, (1998).
9. CD-adapco, "Best Practice Workshop: Heat Transfer," *Workshop presentation of STAR South East Asian Conference*, Singapore, (2012).
10. CD-adapco, "STAR-CCM+ Version 8.02 User Guide," (2013).
11. B. Pouchin, Private communications, (2011).
12. M. Anselmet, G. Bignan, D. Iracane, P. Lemoine, B. Maugard and P. Tremodeux, "Jules Horowitz Reactor project: status and fuel licencing process," *Proceedings of European Research Reactor Conference 2009(RRFM 2009)*, Vienna, Austria, March 22–25, (2009).
13. K. Engelberg-Forster and R. Greif, "Heat Transfer to a Boiling Liquid: Mechanism and Correlation," *Transactions of the ASME, Journal of Heat Transfer*, **81**, pp. 43-53 (1959).

APPENDIX A

For calculating safety margins for JHR a conservative approach is used. The wall temperature must stay below the wall onset of fully developed subcooled nucleate boiling (which precedes onset of significant void) temperature. The safety margin (M) criterion without uncertainties is:

$$M = T_{lSAT} + \Delta T_{SAT} - (T_l + \phi/h), \tag{1}$$

where T_{lSAT} is the liquid saturation temperature, $(T_l+\phi/h)$ is the wall temperature (T_w) given by the liquid temperature (T_l), by the wall heat flux (ϕ) and by the heat transfer coefficient (h). The wall superheat (ΔT_{SAT}) at which the fully developed subcooled boiling begins is given by Engelberg-Forster & Greif correlation [13] most suitable for JHR conditions:

$$\Delta T_{SAT} = 4.44(\phi/10^4)^{0.385} \cdot (p/10^5)^{-0.23}, \tag{2}$$

where p is the pressure. For calculating the safety margin during the FLICA4 core calculations, 15% uncertainty is assumed to the Engelberg-Forster & Greif correlation and 15-20% uncertanity to the heat transfer coefficient. The heat transfer coefficient uncertainty of 15% is used when Re>3000 otherwise the uncertainty 20%. We get the worst possible case of the safety margin when the heat transfer coefficient and the Engelberg-Forster & Greif correlation are reduced by the uncertainties:

$$M_I = T_{lSAT} + 0.85 \cdot \Delta T_{SAT} - (T_l + \phi/(c \cdot h)), \tag{3}$$

where c= 0.80 if Re<3000, otherwise c= 0.85.