This is the Author's Generated Preprint of an article published in ANNALS OF NUCLEAR ENERGY and available online at the following link:
https://doi.org/10.1016/j.anucene.2016.11.041

Coupled simulations of the NACIE facility using RELAP5 and Ansys Fluent codes

Daniele Martelli[1]*, Nicola Forgione[1], Gianluca Barone[1], Ivan di Piazza[2]

1. Dipartimento di Ingegneria Civile e Industriale, University of Pisa, Largo Lucio Lazzarino, 1-56100 Pisa Italy.
2. Italian National Agency for New Technologies, Energy and Sustainable Economic Development, C.R. ENEA Brasimone, Italy.

*Corresponding author. E-mail: daniele.martelli@ing.unipi.it

HIGHLIGHTS

- A STH/CFD coupling tool is developed for thermal-hydraulics analyses.
- Explicit numerical coupling scheme are implemented.
- A preliminary coupling tool assesment is presented.
- Natural, assisted circulation and ULOF tests are simulated.

Abstract

This work is carried out at the DICI (Dipartimento di Ingegneria Civile e Industriale) of the University of Pisa in collaboration with ENEA Brasimone RC. It involves the development and preliminarily assessment of a coupling methodology between a modified version of RELAP5/Mod3.3 STH code and Fluent commercial CFD code, applied to the NACIE (natural circulation experiment) LBE experimental loop (built and located at the ENEA Brasimone research centre).
In the first part of the work, the coupling procedure is described and the NACIE experimental facility, together with its RELAP5 model is presented. Model modifications are introduced to perform the coupled simulations (coupled-nodalization) with the 2D CFD geometrical domain and the implemented explicit coupling scheme is discussed. In the second part of the paper, the coupling methodology is applied to simulate experiments representative of both natural circulation conditions and isothermal gas enhanced (assisted) circulation. Furthermore, an accidental test reproducing an Unprotected Loss of Flow (ULOF) scenario is also simulated with the coupling procedure and the outcomes are presented.
A preliminary sensitivity analysis has shown that, to guarantee a suitable numerical convergence, the assisted circulation tests require a time step one order of magnitude lower compared to natural circulation ones. The comparison between the RELAP5 stand-alone simulations and RELAP5/FLUENT coupled simulations proved the capability to simulate the thermal-hydraulic behaviour of a loop experimental facility for all the examined conditions.

1. Introduction

The development of GEN IV nuclear reactors has benefited from the constant increase of today's computational power. Best-Estimate (BE) codes, also named System Thermal-Hydraulic codes (STH) such as RELAP5, CATHARE, ATHLET, TRACE, GOTHIC etc., are widely used for safety analysis aiming to licence, assess and improve the safety of existing and new nuclear power plants (NPP). They are commonly used to investigate NPP response to a wide range of accidental scenarios including the design basis accident (DBA) or beyond DBA [1]. STH codes are generally based on one-dimensional (1D) form of mass, momentum and energy balance equations, for two-phase flow, solved in Eulerian coordinates, including models based on empirical correlations (e.g. heat transfer, frictional pressure losses, etc.). Their development started in the early 70s requiring over the years an extensive validation activity that made them actually very well established with a high degree of maturity [2]. The use of validated models ensures remarkably accurate predictions of NPP behaviours within reasonable computational time. Likewise, in the last decades, the use of computational fluid dynamics (CFD) has increased its extension in nuclear reactor safety (NRS) field. In 2002, the Committee on the Safety of Nuclear Installations (CSNI) defined an action plan to provide a set of guidelines for the application of CFD codes for NRS to monitor the status of CFD applicability and to identify its restrictions [3]. System codes are extensively validated for two-phase flow phenomena, while two-phase CFD codes are less mature and not yet ready for extended applications. On the other hand, single-phase CFD codes attained a satisfactory degree of maturity such as to justify and promote their use, especially for the study of complex three-dimensional (3D) phenomena where 1D codes are not suitable (e.g. STH codes are generally inadequate when applied to transient investigating mixing and thermal stratification phenomena in large pool systems). In this context, the procedure of a synergic coupling of the two codes has been developed to model the interaction of specific and distinct physical phenomena. Codes coupling applied to nuclear R&D activity generally involve thermal hydraulic analysis of the nuclear primary system performed by a STH code, associated with neutronics or structural mechanics 3D codes. Other cases include coupling of STH with fission products chemistry or with CFD in order to calculate the system and the local behaviour simultaneously [4].

In the present work, the CFD/STH coupling technique between RELAP5 and Ansys Fluent is proposed, assessing the methodology on the LBE NACIE experimental loop. The NACIE system is totally modelled using Realp5, except for the heating zone (FPS) simulated using Ansys Fluent code. Through the simulation, the two domains, Relap5 and Fluent, exchange data (as temperature, pressure, mass flow, etc.) at their respective boundary region providing an overall solution of the complete system. Such an approach, once validated, would permit an accurate thermal-hydraulic characterization of the region of interest operating within a generic system without the need of a complete CFD simulation.

Symbols

β	isobaric thermal expansion [K^{-1}]
Δp	pressure head [Pa]
ΔT	temperature difference [°C]
g	acceleration of gravity [m/s^2]
H	elevation [m]
$\dot{m}_1$	mass flow rate at inlet section [kg/s]
$\dot{m}_2$	mass flow rate at outlet section [kg/s]
p_1	pressure at inlet section [Pa]
p_2	pressure at outlet section [Pa]
T_1	temperature at inlet section [°C]
T_2	temperature at outlet section [°C]

Acronyms

AC	Assisted Circulation
AISI	American Iron Steel Institute
ATHLET	Analysis of thermal-hydraulics of leaks and transients
CATHARE	Code for analysis of thermal hydraulics during an accident of reactor and safety evaluation
CFD	Computational fluid dynamic
CPU	Central process unit
CSNI	Committee on the Safety of Nuclear Installations
DBA	design basis accident
DICI	Dipartimento di Ingegneria Civile e Industriale
GEN IV	generation four
ENEA	Italian National Agency for New Technologies, Energy and Sustainable Economic Development
FPS	fuel pin simulator
HLM	heavy liquid metal
HX	heat exchanger
IAEA	International Atomic Energy Agency
LBE	lead bismuth eutectic
NACIE	natural circulation experiment
NC	natural circulation
NPP	nuclear power plant
NRS	nuclear reactor safety
RELAP	reactor loss of coolant analysis program
STH	system thermal hydraulic
TECDOC	technical document
TRACE	TRAC/RELAP advanced computational engine
UDF	user defined function

2. Coupling procedure

The developed coupled approach can be classified as “non-overlapping”, “two-way coupling scheme”. The geometry or domain to be analysed is subdivided into regions that are modelled using CFD approach and regions that can reasonably be well-simulated using system code (non-overlapping method). This partition identifies the interfaces where thermo-fluid-dynamics data are transferred from the system code fluid portion to the CFD code portion and vice-versa (two-way coupling).

The sets of CFD and STH equations have to be solved in order to obtain the solution of the coupled domain. Implementing a “monolithic solution”, the two sets of equations are unified into a single system to be solved with a monolithic solution procedure. This method requires major modifications to the source code of both the software. Alternatively, adopting a “partitioned solution” the two set of equations are independently solved using for each code the specific solver algorithms and requiring solely minor codes variation. Nevertheless, the latter method necessitate a coupling interface to handle the synchronization between the two codes and the exchanged information. In the present work, the “partitioned solution” approach is adopted (Ansys Fluent source code not available) and its scheme is reported in Fig. 1. The CFD domain is solved from the CFD solver and the obtained solutions at the interfaces are averaged and written into a file in order to be exchanged with the STH code and set as boundary conditions in time dependent volume (TDV, pressure and temperature data) and time dependent junction (TDJ, mass flow rate data). Similarly, the STH solver supplies data computed in TDJ and TDV (written in dedicated files) at the interfaces in order to be transferred to the CFD domain where they are set as constant profiles of temperature and velocity.

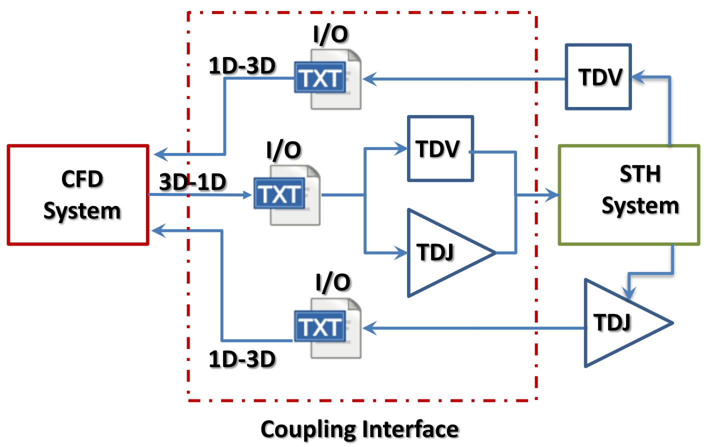


Fig. 1: Partitioned scheme

A third external software acts as coupling interface. In particular in this work the execution of the RELAP5 and Fluent codes is operated by an appropriate MATLAB script, where a processing algorithm is implemented allowing to receive boundary conditions (b.c.) data from Fluent, at the beginning of the RELAP5 time step, and to send b.c. data to Fluent code, at the end of the RELAP5 time step. In addition, a special User Defined Function (UDF) is developed for the Fluent code to receive b.c. data from RELAP5 and to send b.c. data to RELAP5 for each CFD time step.

An initial RELAP5 stand alone transient of 1000 s is executed to reach steady state conditions with a uniform temperature (depending on the simulated test) and with fluid at rest. The end of this initial transient is considered time zero from which the coupled simulation starts. After that, a sequential coupling calculation is activated, where Fluent (master code) advances firstly by one time step and then

RELAP5 advances for the same time step period, using data received from the master code (in-line coupling). After both the codes terminate the current time step, RELAP5 data needed as Fluent b.c. are exchanged and the procedure for a new time step advancement is repeated according to the explicit coupling scheme reported in Fig.2. For the developed explicit coupling scheme, the solution at time step *i*+1 is evaluated in terms of known quantities at the previous time step *i*. Explicit numerical methods are conditionally stable and, in order to guarantee the method convergence, the time step duration is limited by the Courant-Friedrich-Levy (CFL) limit (necessary but not sufficient condition).

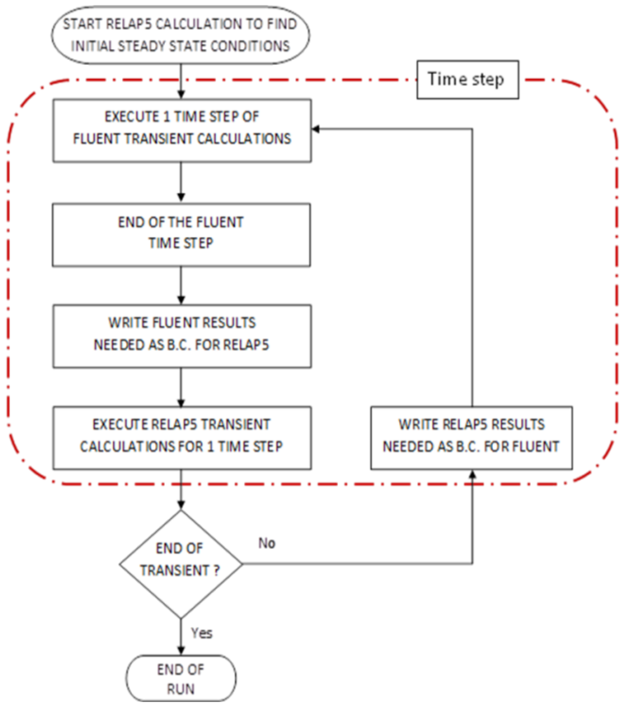
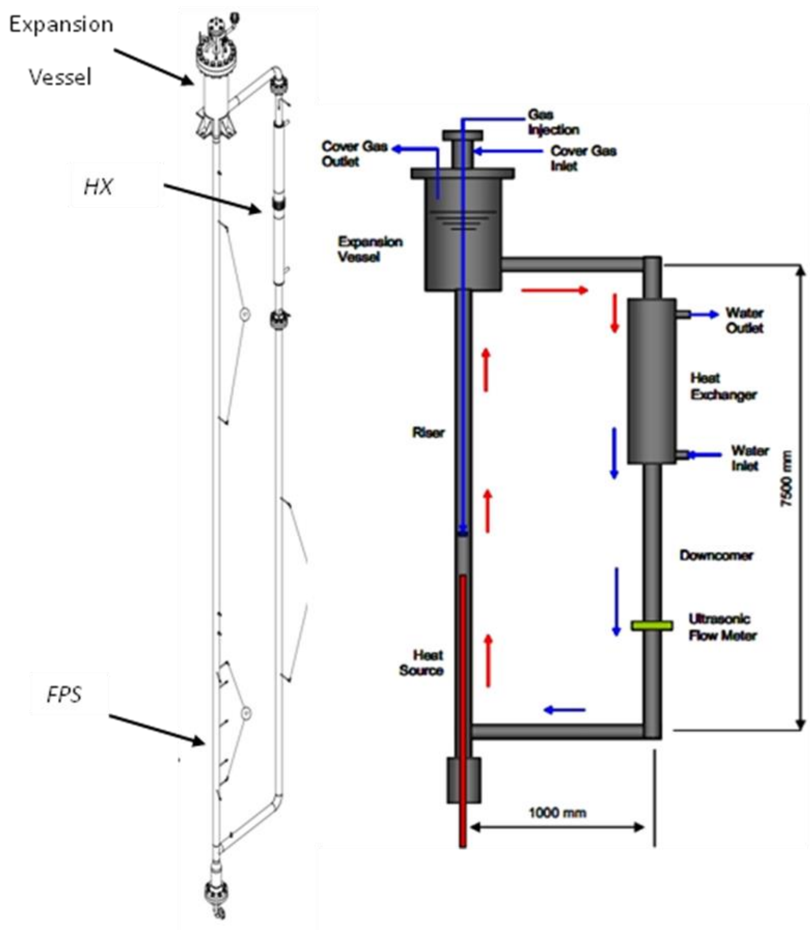


Fig.2: Explicit coupling scheme

3. The NACIE loop facility

NACIE [5], is a lead bismuth eutectic (LBE) loop type facility conceived to qualify and characterize components, systems and procedures relevant for Gen IV lead cooled nuclear technologies. In particular, experimental campaigns are executed to investigate the thermo-hydraulic aspects and the heat transfer phenomenology (to assess empirical correlations) in prototypical fuel bundle simulators. Besides, an extensive numerical simulation is performed for the qualification and development of CFD and STH codes. The facility consists of a rectangular loop made of two, 7.5 m, vertical stainless steel pipes (2½", Sch. 40), acting as riser and downcomer, connected with two, 1 m, horizontal pipes. Two electrical heated rods acting as fuel pin simulator (FPS) are installed at the bottom part of the riser, while a water/LBE heat exchanger (HX) is placed on the upper part of the downcomer (Fig.3). The loop (LBE inventory ~ 800-900 kg) is designed to work with temperatures and pressures up to 550°C and 10 bar respectively, in both assisted (gas lift) and natural circulation.



(*a*) (*b*)

Fig.3. Isometric view (a) and layout (b) of NACIE primary loop

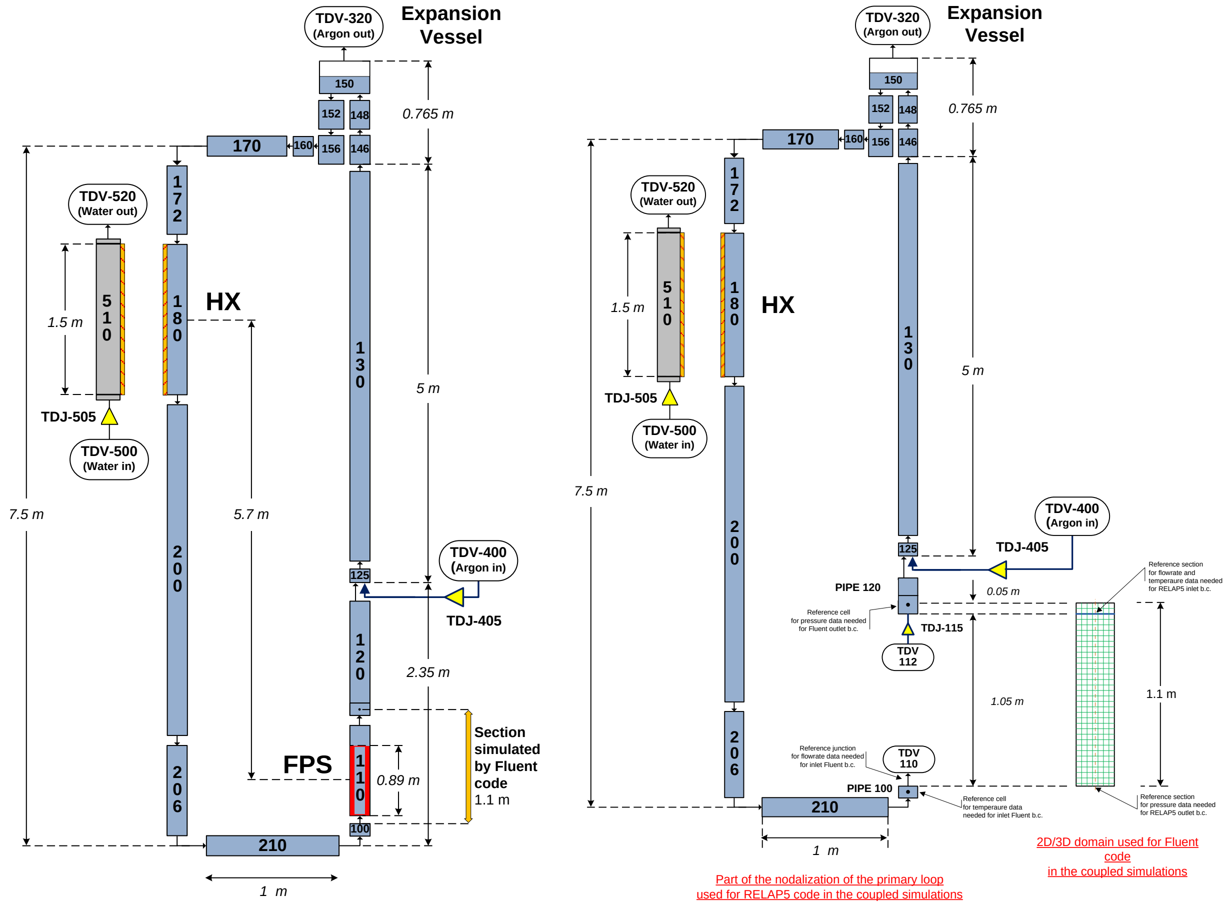
Assisted circulation (AC) conditions are achieved adopting a gas lift technique with a dedicated system to inject argon (1-20 Nl/min, up to 5.5 bar) into the riser lower section (downstream the FPS) that flows up towards the expansion vessel, promoting the LBE circulation. Furthermore, the loop is designed with a thermal elevation difference, H, between the heat source FPS and the heat sink (heat exchanger, HX) of about 5.7 m, that provides the driving pressure ($\Delta p \sim g\beta\Delta T \cdot H$) to guarantee a suitable LBE flow in natural circulation (NC) conditions. The maximum LBE mass flow rate is around 20 kg/s in gas-lift AC and 5 kg/s in NC conditions.

4. Geometrical domains of RELAP5 and Fluent

A numerical model of the entire NACIE facility has been implemented with RELAP5/Mod.3.3 as shown in Fig.4 (*a*). The working fluid (835 kg of LBE) is initialized to be at rest with a uniform temperature. Argon upper plenum pressure in the expansion vessel is set to $1.2{\cdot}10^5$ Pa (*TDV-320*). The LBE circulates anticlockwise flowing through the FPS (*Pipe-110*) simulating a single heating pin (active length of 0.89 m), placed in the bottom section of the riser. Gas lift circulation has been modelled using a time dependent volume *TDV-400* (containing argon) connected to a time dependent junction (*TDJ-405)* injecting the argon flow into the riser (*Branch-125*) and thereby promoting LBE circulation along the loop. Inside the expansion vessel, argon is separated from the liquid metal and exits in *TmdpVol-320*, while the LBE goes through the upper horizontal pipe (*Pipe-160* and *Pipe-170*) to the downcomer where it flows downwards through the heat exchanger (HX) primary side section (*Pipe-180*, located in the downcomer upper zone). Here, the water secondary side thermally coupled with the descending LBE, removes the thermal power. The secondary side water system is modelled by means of *TmdpVol-500*, (where the inlet water properties are set) connected to *TmdpJun-505* that defines the inlet water mass flow rate feeding the HX secondary side annular zone (*Annulus-510*); water flows upwards and exits in *TmdpVol-520*. Primary to secondary heat transfer involves the 1.5 m HX active length and simulates the tube in tube counter flow heat exchanger configuration, taking into account the presence of stainless steel powder filling the gap created by the internal and middle pipe (5.95 mm width). Thermal conductivity of the powder is chosen to be 12.5% of AISI 304 theoretical value [6]. External heat losses are considered taking into account the facility thermal insulation. Fig.4 (*a*) illustrates the described full RELAP5 nodalization ("closed" RELAP5 model). In order to perform the coupling method, the "closed" domain is partitioned into two complementary regions: a CFD (Fluent) computational domain and a STH (RELAP5) computational domain (non-overlapping domains technique). Fluent domain reproduces the FPS for a total length of 1.1 m consisting of 0.89 m active length plus a non-active length of 0.21 m (needed to reduce outlet backflow occurrence).

The domain simulated by Fluent corresponds to RELAP5 components *pipe-110* (1.05 m) plus the first volume of *pipe-120* (0.05 m). Thereby the RELAP5 computational domain is generated from the "closed" model subtracting the components simulated by Fluent ("open" RELAP5 model). In Fig.4 (*b*), the RELAP5 nodalization used for the coupled simulations is reported. The "open" model is integrated with a *TDJ-115* and a *TDV-112* connected to *Pipe-120* inlet and a *TDV-110* to *Pipe-100* outlet.

These additional components are introduced to impose to the "open" RELAP5 model the boundary conditions (b.c) obtained from the CFD. More specifically, in *TDJ-115* and *TDV-112* are set the mass flow, $\dot{m}_2$, and temperature T_2, obtained from an inner reference section of the Fluent domain (in RELAP5 temperature and mass flow rate are computed at the faces of the pipe-cells) , while in *TDV-110* is set the pressure, p_1 obtained from the inlet section of the CFD domain. Similarly, the CFD inlet surface b.c, mass flow $\dot{m}_1$ and temperature T_1, are obtained from the values computed in the last volume of *Pipe-100*, while the outlet surface b.c. pressure p_2 is obtained from the value computed in the first volume of *Pipe-120*. The data flow between the two domains is reported in the scheme of Fig.5. Mass flow rate and LBE temperature required as inlet boundary condition (b.c.) for the CFD geometrical domain are evaluated at *Pipe-110*.



(*a*) (*b*)

Fig.4. RELAP5 nodalization of NACIE loop for stand-alone (*a*) and coupled (*b*) simulations

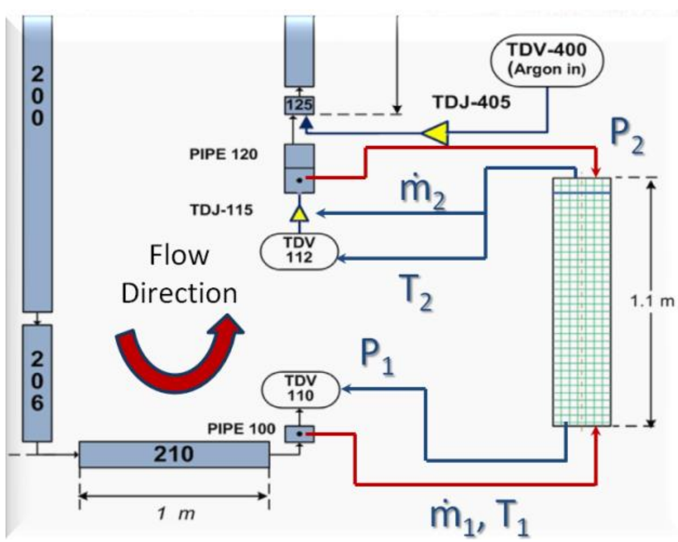


Fig.5. RELAP5-Fluent data exchange

The portion of the loop simulated by the Fluent code is modelled as a simplified 2D axial-symmetric domain in order to reduce computational costs and focus the attention on the coupling methodology. The geometrical model is discretized by a structured mesh composed by 7668 rectangular cells uniformly distributed both in the axial and radial coordinates (Fig.6).

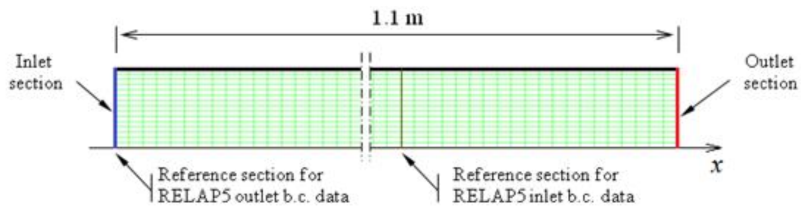


Fig.6. Axial-symmetric domain used in Fluent code for coupled simulations

The power generated by the electrical pin is simulated in the CFD domain as applied to the external wall of the FPS and it is implemented in the code trough the UDF.

To model the FPS form loss coefficient (mainly spacer grids) in the 2D domain, a constant value of 3.5 is assumed. For this purpose, five distinct interior faces are set as “porous-jump”, each characterized by an equivalent constant local pressure drop coefficient of 0.7.

5. Simulation

In order to assess the coupling methodology a series of simulations have been performed:

- Natural circulation conditions (NC);
- Assisted circulation conditions (AC)
- Transient representative of an Unprotected Loss of Flow (ULOF) scenario;

A 2D-CFD computational domain is adopted to limit the computational time. For NC tests the heating power is linearly increased in the first 30 s and then is kept constant, while for AC tests the argon injection in the riser is linearly increased in the first 30 s then is maintained constant. In AC tests both the FPS and the HX are deactivated (zero power) with an isothermal loop temperature of 290°C
The ULOF simulation is performed, with the shutdown of the gas injection into the riser, while the FPS (heat source) and HX (heat exchanger) remain in operation. A first parametric analysis has shown that AC tests require a time step one order of magnitude lower than for NC tests to guarantee the simulation convergence. The test matrix of the performed coupled simulations is shown in Table 1.

Table 1: Test matrix

Simulation	Test Name	Thermal Power [kW]	Argon Flow [Nl/min]	Time step [s]	Monitored variables
Natural circulation	***A***	10	-	0.1	- LBE flow rate - T_{in} and T_{out} in the FPS - T_{in} and T_{out} in the HX
	B	20	-	0.1	
	C	20	-	0.2	- Time step independence
Assisted circulation (gas lift)	***D***	-	5	0.01	- LBE flow rate
	E	-	10	0.01	
	F	-	20	0.01	
	G	-	20	0.02	- Time step independence
	H	-	20	0.005	
Unprotected loss of flow accident	***I***	20	20	0.02	- LBE flow rate - T_{in} and T_{out} in the FPS - T_{in} and T_{out} in the HX

6. Results and discussion

6.1 Natural circulation tests

The LBE mass flow rate outcomes from the coupled NC Tests A and B are reported in Fig.7 (*a*) and are compared with the ones obtained by RELAP5 stand-alone. LBE mass flow rate steady state conditions are reached before 4000 s, obtaining an asymptotic value of about 1.5 kg/s for Test *A* (thermal power of 10 kW) and 1.9 kg/s for Test *B* (thermal power of 20 kW). A good agreement is found with RELAP5 stand-alone model with a discrepancy of about 2-3% essentially due to differences between RELAP5 and Ansys Fluent in evaluating FPS pressure losses. In fact, RELAP5 is a one-dimensional lumped parameter code requiring user input coefficient for concentrated pressure losses modelling, while Darcy-Weisbach equation is used for the friction losses. The Fluent code is instead, a mechanistic CFD code and concentrated losses are directly computed by the code with the exception of those parts (e.g. FPS spacer grids) not geometrically simulated and whose effect has been accounted for using the porous jump model. Moreover, if the enhanced wall treatment option is used for the CFD domain (as the Near-Wall Treatment) then the wall roughness parameters are not applicable and smooth walls are considered. When the mass flow rate computed by RELAP5 is set as b.c., an area averaged constant temperature and velocity profiles are imposed in Fluent. This uniform inlet velocity produces a developing boundary layer that is responsible for the so-called entrance effects, not considered in RELAP5 stand-alone calculations. In Fig.7 (*b*) the time trends of the FPS pressure difference are reported compared to the RELAP5 stand-alone outcomes. The pressure drops calculated by the coupled codes are higher than those obtained from the RELAP5 stand-alone simulations of about 600 Pa.

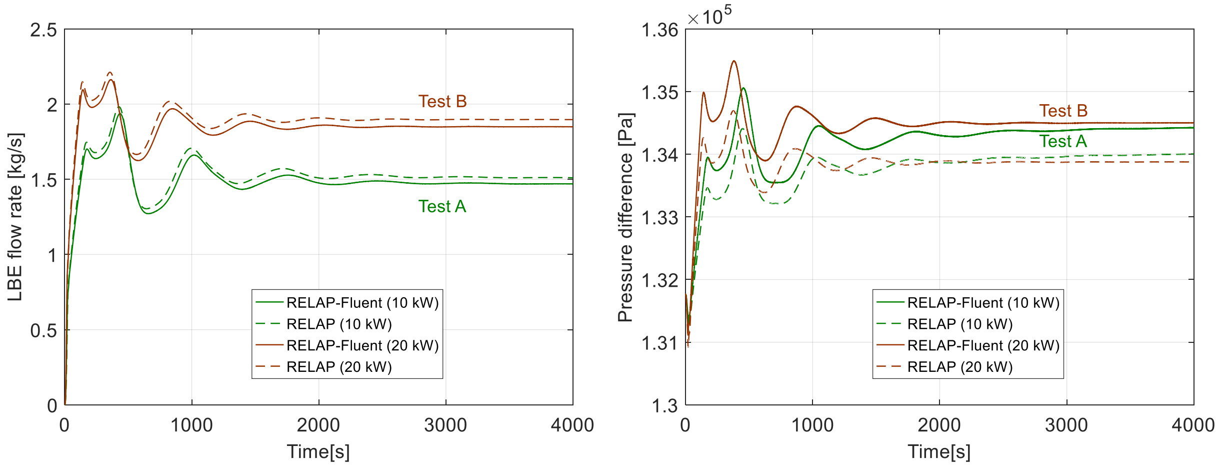


Fig.7. LBE mass flow rate (*a*) and pressure drop through the *FPS* (*b*); Tests *A* and *B*

Concerning the FPS and HX inlet and outlet temperature, an excellent agreement is found between RELAP5 stand-alone and the coupled simulations for both tests A and B, as depicted in Fig. 8 and Fig. 9. In 4000 s of simulation, the steady state conditions are not reached for test A (10 kW) while for test B (20 kW), with higher thermal power, a steady state is achieved. The first temperature peak of 370°C for test A (Fig. 8 (*a*)) and of 414°C for test B (Fig. 9(*a*)) is due to the mechanical inertia that opposes the flow onset while the FPS heat flux begins. In fact the fluid requires a sufficient driving force to overcome the buoyancy effect and this creates, in the first instant of the transient, a heating of LBE at rest inside the FPS section.

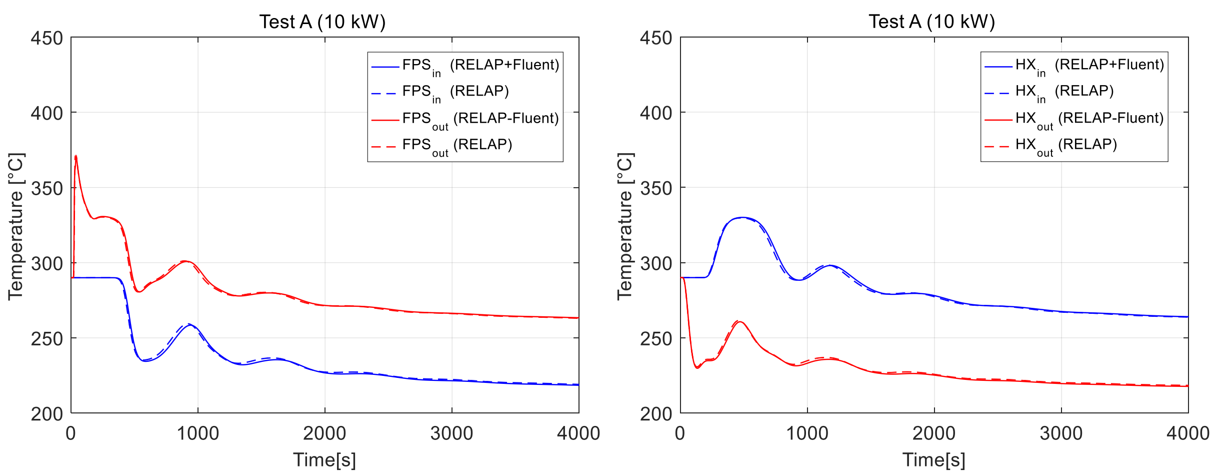
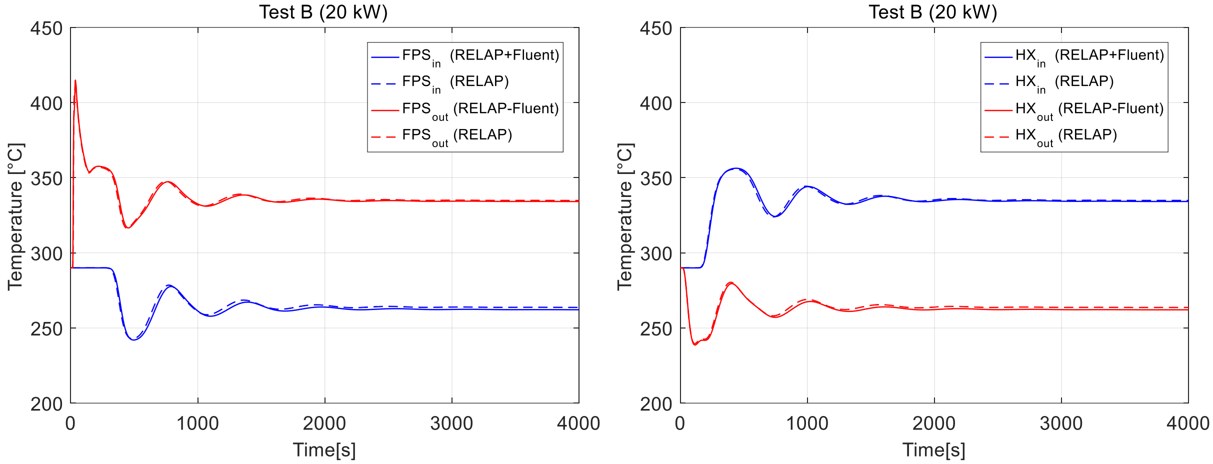


Fig. 8: Inlet and outlet temperature in the *FPS* (*a*) and in the *HX* (*b*); Test *A* (10 kW)



(*a*) (*b*)

Fig. 9 Inlet and outlet temperature in the *FPS* (*a*) and in the *HX* (*b*); Test *B* (20 kW)

The 2D CFD detailed temperature spatial distribution is illustrated in Fig. 10 for test B at simulation time t=40 s (FPS outlet temperature peak).

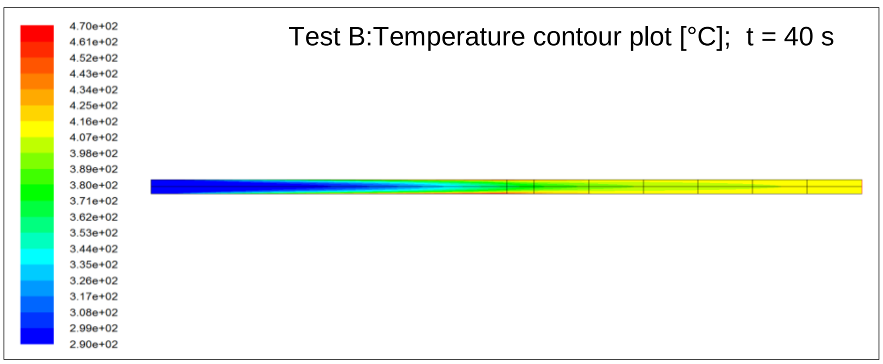
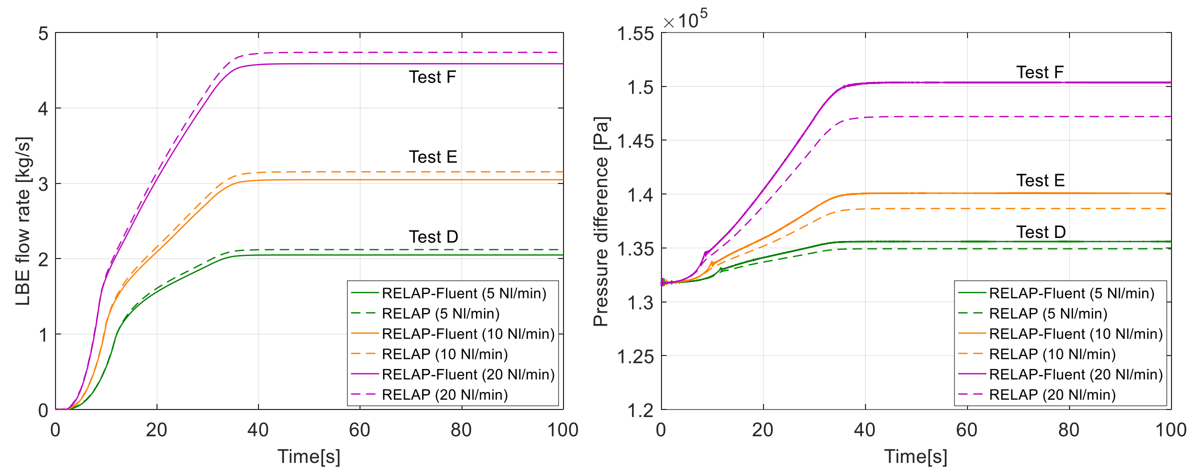


Fig. 10: Temperature contour plot at 40 s of transient; Test B (20 kW)

In test C the time step is increased from 0.1 s (test B) to 0.2 s showing a complete agreement with test B outcomes. For higher time steps, the method stability was no longer guaranteed.

6.2 Assisted circulation tests

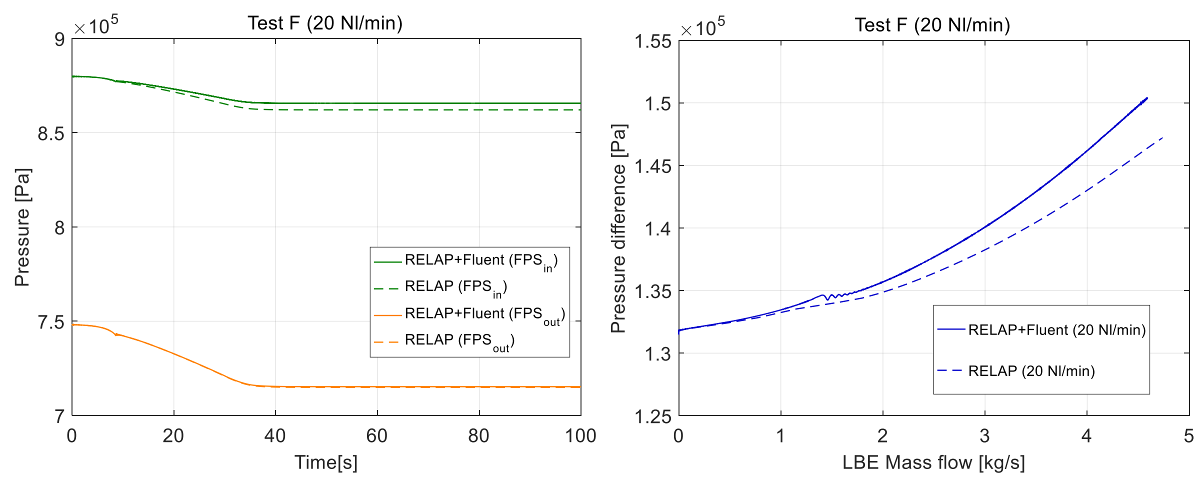
The LBE mass flow and FPS pressure difference obtained from assisted circulation (zero power) tests (D, E and F) are compared with the RELAP5 stand-alone results in Fig. 11.



(*a*) (*b*)

Fig. 11: LBE mass flow rate time trend (*a*) and pressure difference through the FPS (*b*)

Differences in the LBE mass flow rate between RELAP5/Fluent and RELAP5 stand-alone are lower than 5% (test F). As for NC simulation, this is due to differences between RELAP5 and Ansys Fluent in evaluating pressure losses. In particular, is due to the higher pressure drop computed by the CFD code for the *FPS* 2D domain as shown, for test F, in Fig. 12 (*a*) where the absolute pressure time trend at the inlet and outlet sections of the CFD domain are reported. This discrepancy can be visualized in Fig. 12 (*b*) (that combines the data of Fig. 11) plotting the *FPS* pressure difference as a function of the LBE mass flow rate. As for the NC tests, it can be seen that the FPS pressure drop calculated by Fluent in the coupled simulation is higher compared to the correspondent one evaluated by the RELAP5 stand-alone calculation.



(*a*) (*b*)

Fig. 12: Inlet-outlet pressure time trend for the *FPS* (*a*) and *FPS* pressure difference vs. LBE flow rate (*b*); Test F

The Relap5/Fluent coupled methodology provides a detailed characterization of the thermo-hydraulic parameters related to the CFD domain throughout the simulation. As an example, Fig. 13 and Fig. 14 illustrate respectively the FPS velocity vector and the turbulent kinetic energy at the end of test F. As previously stated, the CFD inlet boundary conditions, $\dot{m}_1$ and T_1, are transmitted from Relap5 to Fluent as scalar values that are afterwards converted as a uniform profiles on the CFD inlet surface (see the inlet flat velocity profile in Fig. 13). This approximation affects the CFD simulation to a lesser extent the higher is the distance between the inlet surface and the zone to investigate in order to reach a fully developed thermal-hydraulic profile as for the outlet surface in Fig. 13.

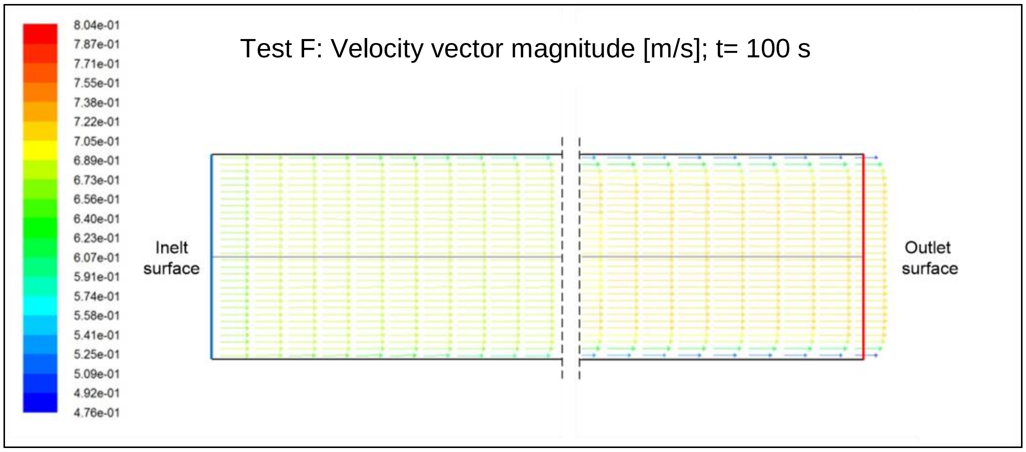


Fig. 13: Velocity vector distribution at the end of the transient; Test F (20 Nl/min)

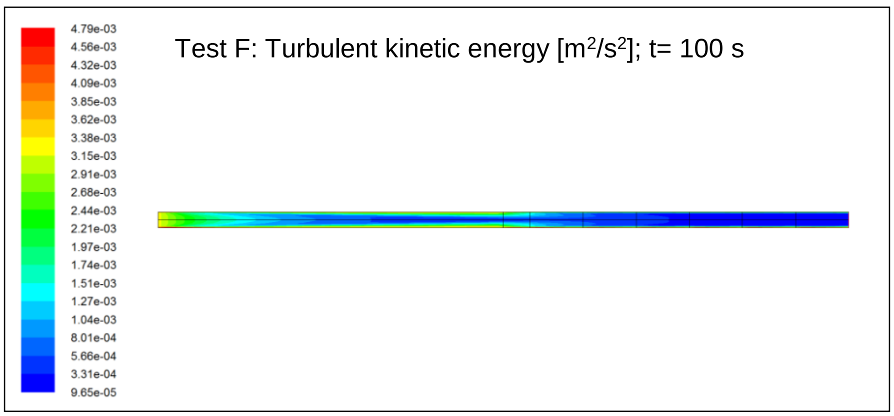


Fig. 14: Turbulent kinetic energy at the end of the transient; Test F (20 Nl/min)

In tests G and H the time step value is brought respectively to 0.005 s and 0.02 s, leaving unchanged the test F boundary conditions. Outcome showed a complete overlapping compared to test F results.

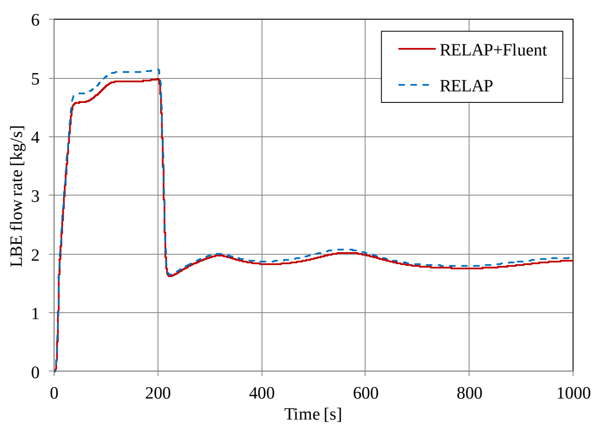
6.3 ULOF test

The ULOF accident transient (Test I) is a safety relevant test in HLM reactors consisting in the transition from forced to natural circulation conditions without heating power reduction. In Table 2, the boundary conditions set for Test I are specified. The time step used for the coupled codes simulation is 0.02 s.

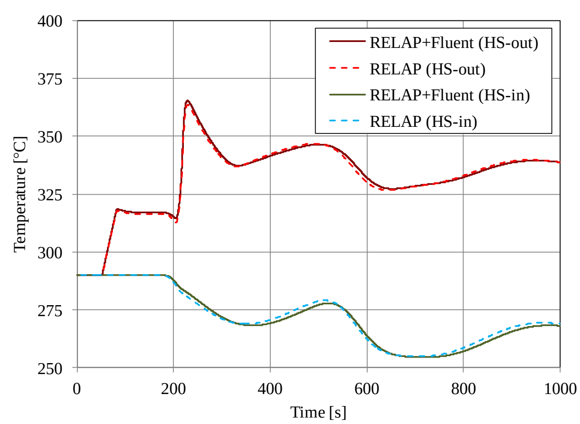
Table 2: ULOF transient (Test I)

Time [s]	Event	Description
0-30	Argon gas flow rate increase linearly from zero to 20 Nl/min; after 30 s its value is kept constant up to ULOF event.	***Starting phase****: achieving of the reference conditions*
50-80	Thermal power supplied through the FPS increases linearly from zero to 20 kW; in the same interval, the water flow rate injected in the secondary side of the HX increases linearly. From 80 s to the end of the analysed transient, the value of the FPS thermal power and of the HX water flow rate remains constant.	
200-210	Gas flow injection system switched off decreasing linearly its value in 10 s.	***ULOF event***
210-1000	The FPS power remains constant (20 kW) and the HX continues to operate.	***ULOF evolution***

As shown in Fig. 15 (*a*), the LBE mass flow reaches a value of about 4.6 kg/s in the initial gas injection period and a value of about 5 kg/s in the phase with both gas injection and heating/cooling. After the argon injection shutdown, the LBE mass flow rate decreases to a value of about 2 kg/s. The trend agrees sufficiently well with the RELAP5 stand-alone code, the discrepancies being attributed to the different method in evaluating the FPS pressure losses. The LBE temperature for the FPS shows an adequate agreement with those obtained by the RELAP5 stand-alone code (see Fig. 15 (*b*)), validating the suitability of the coupled numerical scheme also for a transient simulation.



(*a*)



(*b*)

Fig. 15: LBE mass flow rate time trend (*a*) and inlet and outlet FPS temperature time trends for Test I (*b*)

7. Conclusions

A 2D axial-symmetric Fluent domain of the FPS was generated to be interfaced with its complementary region of NACIE facility modelled with Relap5. Essentially, the method allows a detailed CFD simulation of a specific component integrated within the system of which it is part. The developed Relap5/Fluent explicit method was numerically assessed simulating the coupled code for three different operating conditions. The main outcomes have been compared with Relap5 stand-alone results showing an appropriate consistency between the involved physical parameters. Pressure, temperature and mass flow rate properly followed the Relap5 stand-alone trends even for the fast transient ULOF scenario. Differences are found for the FPS inlet and outlet pressure due to the different approach adopted by the two codes for pressure losses computation. In fact, Fluent compute the pressure losses in a more precise mechanistic manner, while Relap5 uses correlation requiring user estimated coefficients. These discrepancies affect also the LBE mass flow rate while the FPS inlet and outlet temperatures are almost coincident with the Relap5 stand-alone results. Accordingly, the Fluent model produced an accurate fluid-dynamic characterization of the FPS component incorporated in the entire NACIE system with considerable advantages in terms of computing time and modelling efforts (no need of a CFD complete system model). A sensitivity analysis of the used time step was carried out assessing the maximum value for each simulation without compromising results accuracy. For NC and AC tests, time step values of 0.1 s and 0.01 s respectively, have shown to be adequate. Therefore, the proposed method has shown to be a promising tool for coupled codes development and may represent the starting point for future improvements in terms of numerical scheme (implicit), parallel computation and simulation of complex CFD 3D phenomenology (e.g. wire spaced fuel bundle).

References

[1] D'Auria, F., Salah A.B., Petruzzi, A., Del Nevo, A., “State of the art in using best estimate calculation tools in nuclear technology”. Nucl. Eng. Tech. 38, pp.11-32, 2006.

[2] Petruzzi, A., D' Auria, F., “Thermal-hydraulic system codes in nulcear reactor safety and qualification procedures”. Science and Technology of Nuclear Installations, vol. 2008, Article ID 460795, 16 pages, 2008. doi:10.1155/2008/460795.

[3] Bestion, D., et al., “Reccomandation on use of CFD codes for nuclear reactor safety analysis”, Evaluation of computational fluid dynamic methods for reactor safety analysis (ECORA), European Commision 5th EURATOM Framework Program, 2004.

[4] IAEA-TECDOC-1539, “Use and development of coupled computer codes for the analysis of accidents at nuclear power plants”, proceedings of technical meeting held in Vienna 26-28 November 2003.

[5] Tarantino, M., Agostini, P., Benamati, G., Coccoluto, G., Gaggini, P., Labanti, V., Venturi, G., Class, K., Liftin, A., Forgione, N., Moreau, V., “Integral circulation experiment: thermal-hydraulic simulator of heavy liquid metal reactor”. Nucl. Mater., 415, 433-448, 2011.

[6] Coccoluto G., Gaggini P., Labanti V., Tarantino, M., Ambrosini W., Forgione N., Napoli A., Oriolo F., “Heavy liquid metal natural circulation in a one-dimensional loop”, Nuclear Engineering and Design 24, 1301-1309, 2011.

Acknowledgements

This work is performed in the frame of and supported by the Euratom Seventh Framework Program Collaborative Project: Thermal-Hydraulics of Innovative Nuclear Systems (no. 249337) and by the Programmatic Agreement (AdP) between the Italian Ministry of the Economic Development (MSE) and ENEA. The authors gratefully acknowledge the technical staff of the experimental engineering technical unit of Brasimone (UTIS-TCI).