Ricerca Sistema Elettrico	Sigla di identificazione	Distrib.	Pag.	di
ENEA	ADPFISS - LP2 - 076	L	1	21

Titolo

Validation of the coupled calculation between RELAP5 STH code and Ansys FLUENT CFD code

Ente emittente **UNIPI (CIRTEN)**

PAGINA DI GUARDIA

Descrittori

Tipologia del documento: **Rapporto Tecnico**

Collocazione contrattuale: Accordo di programma ENEA-MSE su sicurezza nucleare e reattori di IV generazione

Argomenti trattati: Tecnologia dei metalli liquidi
Generation IV reactors
Calcolo Parallelo

Sommario

This report, carried out at the DICI (Dipartimento di Ingegneria Civile e Industriale) of the University of, illustrates the activities performed in the framework of the AdP2013 and dealing with the application of an in-house developed coupling methodology between a modified version of RELAP5/Mod3.3 and Fluent commercial CFD code, to the NACIE (Natural Circulation Experiment) LBE experimental loop (built and located at the ENEA Brasimone research centre). In particular, the first part of this work deals with the application of the developed two way explicit coupling tool to a simulation representative of a natural circulation test with a heating power of about 21.5 kW. The second part of the document deals with the development of an implicit coupling scheme in order to increase the numerical stability of the coupled simulations and its application to simulate an experimental test representative of an isothermal gas enhanced circulation test. Obtained results show a good agreement with the experimental data; moreover, the robustness of the developed implicit scheme allows the use of larger time steps reducing significantly the computational time. Finally, in order to further reduce the computational efforts, the User Defined Function realized for the Fluent code to receive b.c. data from RELAP5 and to send b.c data to RELAP5 for each CFD time step was parallelized allowing to run the CFD code using multiple processor and thus reducing the computational time especially for those simulations involving a 3D-CFD geometrical domain.

Note

Rapporto emesso da UNIPI (CIRTEN).

Autori:

D. Martelli, N. Forgione, G. Barone, W. Ambrosini

Copia n. **In carico a:**

2			NOME			
			FIRMA			
1			NOME			
			FIRMA			
0	EMISSIONE	19/09/2014	NOME	Mariano Tarantino	NA	Mariano Tarantino
			FIRMA			
REV.	DESCRIZIONE	DATA		CONVALIDA	VISTO	APPROVAZIONE

CIRTEN

<u>C</u>onsorzio <u>I</u>nteruniversitario per la <u>R</u>icerca <u>TE</u>cnologica <u>N</u>ucleare

UNIVERSITÀ DI PISA

Validation of the coupled calculation between RELAP5 STH code and Ansys FLUENT CFD code

Autori

D. Martelli, N. Forgione, G. Barone, W. Ambrosini

CERSE-UNIPI RL 1536/2014

PISA, 6 Settembre 2014

Lavoro svolto in esecuzione dell'Attività LP2. C1
AdP MSE-ENEA sulla Ricerca di Sistema Elettrico - Piano Annuale di Realizzazione 2013
Progetto B.3.1 "Sviluppo competenze scientifiche nel campo della sicurezza nucleare e collaborazione ai programmi internazionali per i nucleare di IV generazione".

Index

Summary

This report, carried out at the DICI (Dipartimento di Ingegneria Civile e Industriale) of the University of, illustrates the activities performed in the framework of the AdP2013 and dealing with the application of an in-house developed coupling methodology between a modified version of RELAP5/Mod3.3 and Fluent commercial CFD code, to the NACIE (Natural Circulation Experiment) LBE experimental loop (built and located at the ENEA Brasimone research centre).
In particular, the first part of this work deals with the application of the developed two way explicit coupling tool to a simulation representative of a natural circulation test with a heating power of about 21.5 kW. The second part of the document deals with the development of an implicit coupling scheme in order to increase the numerical stability of the coupled simulations and its application to simulate an experimental test representative of an isothermal gas enhanced circulation test. Obtained results show a good agreement with the experimental data; moreover, the robustness of the developed implicit scheme allows the use of larger time steps reducing significantly the computational time. Finally, in order to further reduce the computational efforts, the User Defined Function realized for the Fluent code to receive b.c. data from RELAP5 and to send b.c data to RELAP5 for each CFD time step was parallelized allowing to run the CFD code using multiple processor and thus reducing the computational time especially for those simulations involving a 3D-CFD geometrical domain.

1. Introduction

One of the main objectives of coupling computer codes is to model the interaction of different physical phenomena. The coupling of codes, referring to nuclear research and development activity, generally involves the coupling of primary system thermal hydraulics codes (STH) with neutronics, in order to take into account 3D neutron kinetics and fuel temperature distribution or with structural mechanics, in order to take into account vibration induced by the flow or thermal striping (Hannink et all. 2008). Other cases include coupling of STH with fission production chemistry or with computational fluid dynamics (CFD) in order to calculate the system behaviour and the local behaviour simultaneously (IAEA-TECDOC-1539). In this paper attention is focused on the coupling between STH and CFD codes. System Thermal Hydraulic codes have been widely developed by nuclear R&D and nuclear safety organizations with the aim of improve the reliability of results, maintaining low computational costs (Davis et Shieh, 2000, Austregesilo et al, 2006, Geffraye et al., 2011, RELAP5-3D code development Team, 2013, RELAP5/Mod.3.3 Code Manual, 2003, etc.). These codes, are based on partial differential equation for two phase flow and heat transfer (mass, momentum and energy) usually solved by finite-difference methods based on one dimensional approximations. Three dimensional analyses based on approximate formulation of the momentum balance equation are available in some codes (e.g. RELAP5-3D, CATHARE etc.) with limitations on nodalization, field equations, physical models including the lack of turbulence modeling and the use of idealized friction tensors in rod bundles. In Bestion, 2010 the Multi-Scale analysis of reactor thermal Hydraulics is introduced and four scales corresponding to four category of simulation software are illustrated. In particular the system-scale is dedicated to the overall description of the circuit of the reactor. All accidental scenarios including LB-LOCA and SB-LOCA can be simulated with a reasonable CPU time. The component-scale uses CFD in porous medium. This scale is dedicated to the design and safety of reactor cores and heat exchangers; the minimum spatial resolution is fixed by sub-channel size. The meso-scale uses CFD in open medium and the average scale (millimeters or less) allows obtaining a finer description of the flow. This scale includes turbulence models (RANS, URANS, LES etc.). Finally the micro-scale corresponds to DNS approaches with scales in the order of micrometer or less. STH codes are generally inadequate when applied to transient investigating mixing and thermal stratification phenomena in large pool systems. On the other, the exclusive use of CFD codes still remains prohibitive for the requested computational effort. Coupling between two or more scale thus appears to be a promising technique when the small scale phenomena taking place in a limited part of the domain have to be investigated. For this reason, great interest is given by the leading European research centre to the R&D of coupled simulation tools that combines system code and CFD analysis. In particular, at the French atomic energy commission (CEA) a coupled tool between the 3D computational fluid dynamics code TRIO_U with the best estimate thermal hydraulic system code CATHARE, was developed in order to perform one-phase thermal hydraulic analysis for the French SFR Phénix (R. Bavière et al., 2013). This coupling tool has been developed with the aim of supporting the design and address safety issues for SFR ASTRID demonstrator. It is based on a common Application Programming Interface (API), named ICoCo (Interface for Code Coupling) and the overlapping method was selected. In the coupling application presented in R. Bavière et al., 2013, the CFD domain was restricted to the core whereas the STH code domain includes both the core and loops and the components (pumps, heat exchanger etc.). The system-code gives at the CFD boundaries conditions mass flow rates and temperatures, while the CFD domain gives momentum and enthalpy feedback to the system-code. The Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) is deeply involved in the development of a coupling tool between the ATHLET system code (Analysis of THermal-hydraulics of LEaks and Transients) and the Ansys CFX CFD code. In the application presented in Waata and Frank, 2008, the working fluid is water and the coupling strategy is based on an explicit coupling scheme and ATHLET obtains pressure and temperature from the CFD tool, while it provides at the end of the time step mass flow and enthalpy to ANSYS CFX Inlet. The calculation of these parameters is inverted when the coupling interface is at the outlet. As a further improvement (Papukchiev and Lerchl, 2009), the interface code was modified to allow the use of Opening – Opening boundary conditions in ANSYS CFX. ANSYS CFX Opening is used at a boundary where the flow direction can change (into or out of the CFD domain). With the new strategy, ATHLET provides fluid velocity instead of mass flow rate at the ANSYS CFX inlet Opening. The CFX-ATHLET coupling strategy was developed in close collaboration between GRS and ANSYS Germany for this reason the CFD source is available and shared library containing the interface and ATHLET code were extended in the CFX code. The division of Nuclear power Safety of the Kungliga Tekniska Högskolan (KTH) in Sweden is deeply involved in the development and implementation of a domain overlapping methodology for coupling RELAP5/mod3.3 STH code and Star-CCM+ CFD code. This research activity is dedicated to GEN IV LFR nuclear reactor and the considered working fluid is LBE. The adopted numerical scheme is an explicit scheme where the STH provides inlet boundary temperature and mass flow rate to CFD that calculates the 3D test section outlet boundary temperature. STH model is iteratively corrected until its solution is converged with the CFD solution. The coupling algorithm is implemented in a Java macro executed from the Star-CCM+ Application Programming Interface (API). The macro controls the time-marching, execution of Star-CCM+, boundary data export from, and import to, RELAP5/Star-CMM+, STH input model correction, execution of RELAP5 and logging of all necessary variables. This tool is applied to the TALL-3D experimental facility, a thermal hydraulic Lead-Bismuth loop, designed and built in KTH to provide validation data for both stand alone and coupled simulation (Jeltsov et al. 2014). Pre-Test simulations were already performed but no experimental data are still available. In Schultz et al., Fluent and RELAP5-3D©/ATHENA were linked using an Executive Program (PVMEXEC) (Weaver et al., 2002) that monitors the calculation progression in each code, determines when each code has converged, governs the information interchanges between the codes and issues permission to allow each code to progress to the next time step. The Executive Program was interfaced with Fluent and RELAP5-3D©/ATHENA using user-defined functions. Studies on coupling strategies were also carried out at the Paul Scherrer Institut (PSI) in Switzerland (Bertolotto et al., 2009). In this work a coupling tool between TRACE and Ansys CFX is presented. In particular the information exchange is realized by means of the Parallel Virtual Machines (PVM) software. Both explicit sequential scheme and a semi-implicit scheme are developed for time advancement. Verification of the coupling tool has been carried out on a simple test case consisting of a straight pipe filled with water and on an experimental test conducted on a test facility made of two loops connected by a double T-junction. At the research Institute of Nuclear Engineering of the University of Fukui (Japan) the transient behavior of flow instability in Steam Generator U tubes is simulated numerically by performing a coupled STH-CFD simulation (Watanabe et al., 2014). The codes involved are the RELAP5/MOD3.3 for the simulation of the secondary side and FLUENT for the simulation of the primary side. The hot-leg inlet conditions and the secondary-side temperatures are given by RELAP5 as an output file in each time step, and these data are read by FLUENT using the user defined function. The cold-leg outlet conditions calculated by FLUENT are averaged and written in another output file using the user defined function, and given to RELAP5 as the node and junction variables defined in the restart input file. The restart input file for RELAP5 is edited by a conversion program. Moreover, a small-scale experiment was conducted with the test facility consisting of the heating loop and the model SG and obtained numerical results were compared with the obtained experimental data. At the Department of Nuclear, Plasma and Radiological Engineering of the University of Illinois a coupled CFD system code was developed based on FLUENT and RELAP-3D and applied to simulate the primary coolant system in Modular Helium Reactor (GT-MHR) GEN IV VHTR (Y. Yan and R. Uddin,2011). The CFD model of the lower plenum is coupled with the RELAP5-3D model of the reactor core and upper portion of the GT-MHR and set of User Defined Functions (UDFs) are written to perform the interface data exchange during the coupled simulation.

In 2012 at the DICI of the University of Pisa a first application, to a simplified representation of NACIE facility, of an in-house developed coupling tool between the RELAP5/Mod.3.3 thermal-hydraulic system code and the CFD Fluent commercial code was performed and a preliminary comparative analysis among the simulations performed by RELAP5-Fluent coupled codes and by RELAP5 stand-alone code showed good agreement among them, giving confidence to this innovative coupling strategy (Martelli et al.,2012). In 2013 the coupling tool was improved modifying the correlations used by the RELAP5/Mod3.3 system code to obtain the thermodynamic properties of lead, LBE and sodium and are presented for both saturation and single phase conditions in agreement with Sobolev work (Sobolev, 2010, Martelli et al.,2013). Moreover RELAP5 stand alone and RELAP5/Fluent coupled simulations were performed and the experimental results are compared with the experimental data coming from the experimental campaign performed on NACIE facility.

Finally in this work new developments and improvements dealing with the coupling technique are presented. In particular, a new test involving the promotion of natural circulation is investigated and a new implicit numerical scheme is developed, then both the explicit and implicit coupling tool are parallelized in order to perform multi processor CFD calculations inside the coupled calculation (RELAP5 calculations are performed with single processor). The new improvements are developed to enhance the stability of the coupling procedure and to reduce the computational efforts. For the description of the coupling tool in the explicit coupled version and for the description of the NACIE facility see Martelli et al.,2012 and Martelli et al.,2013.

2. Natural Circulation Test

The last experimental activity performed on the NACIE loop (Di Piazza and Martelli, 2013) included a series of 10 test concerning natural circulation, forced circulation and transition from forced to natural condition and vice-versa. Each test has been performed with only one pin activated in the heating section, with a maximum nominal power of 21.5 kW. The experiment chosen as reference test for numerical simulation of natural circulation test is Test 301 characterized by a heating power of 21.5 kW. At the beginning of the experiment the average temperature of the LBE in the loop is about 250-300 °C. The heating power increased linearly in the first 262 s of the transient and then it is maintained constant for the remain transient. After the activation of the fuel bundle, the water secondary system was also activated. Regarding the numerical simulations, both RELAP5 stand alone and coupled simulations using a 2D axis symmetric domain were performed and an explicit coupled scheme was adopted. A time step value of 0.1 s guarantee the convergence and independency of the results from the adopted time step. Transient simulations with fixed time step were carried out for an overall simulated time of 2680 s (about 0.74 hours).

2.1 Obtained results

The LBE mass flow rate time trend obtained from Test 301 simulated by the coupled methodology is reported in Figure 1, where the results are compared with those obtained by stand-alone RELAP5 simulation and with experimental results as well. The inductive flow meter installed on the NACIE facility is accurate for high mass flow rate (8-20 kg/s) and the uncertainty of the measurements at low mass flow rate is due to the diameter of the pipe in which the inductive effect is measured, (i.e. 2.5″ diameter pipe). At low flow rate, the LBE velocity is not high enough to allow a good flow measurement, therefore the experimental mass flow rate is evaluated by an energy balance equation. When the natural circulation starts the difference in temperature between the inlet and outlet section of the heater is small resulting in the high peak value obtained for the mass flow rate. The different behaviour observed at t = 0.1 h can be related to the fact that the difference in temperature between the heater inlet and outlet sections has not reached a steady state condition, therefore the balance equation results, obtained for steady state, are not reliable for the first 0.1 h of the transient. Results obtained from the RELAP5 stand alone simulation and from the coupled simulation are practically overlapped. The mass flow rate predicted by the simulations at steady state condition is about 5.03 kg/s overestimating the mass flow rate obtained from the energy balance equation by less than 2%

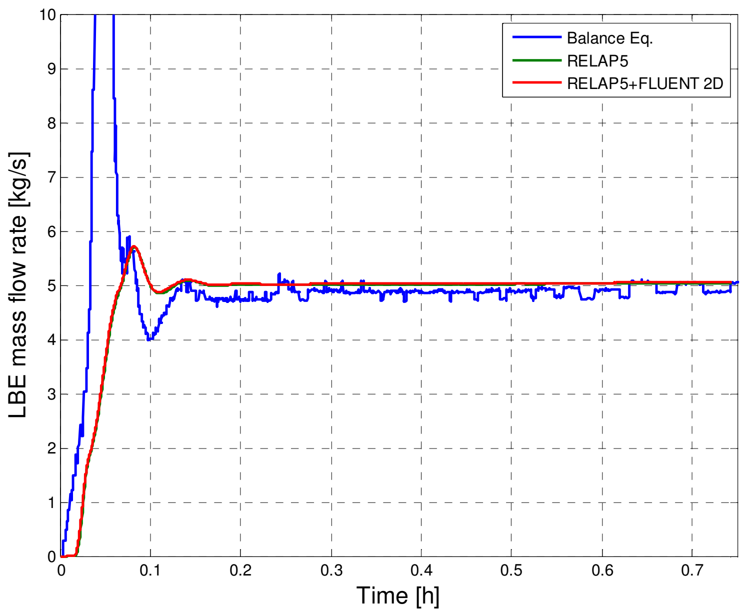

Figure 1: LBE mass flow rate results for Test 301

Temperature measurements evaluated at the heater inlet and outlet section are showed in Figure 2. Good agreement is found with the experimental data. In particular, at the heater outlet section the peak temperature observed at t = 0.09 h (324 s) is well predicted by RELAP5 and coupled simulations. The temperature at the inlet of the heater, predicted by RELAP5 stand alone and coupled simulations, starts to increase about 180 s earlier than the experimental data. This behaviour is due to the simplified temperature distribution imposed at the beginning of the simulation as loop boundary condition of the RELAP5 nodalization; in particular, the initial temperature trend in the REALP5 model is approximated according to the local experimental data along the loop. At the inlet section of the fuel pin bundle, numerical results tend generally to overestimate the experimental temperature by less than 2%.

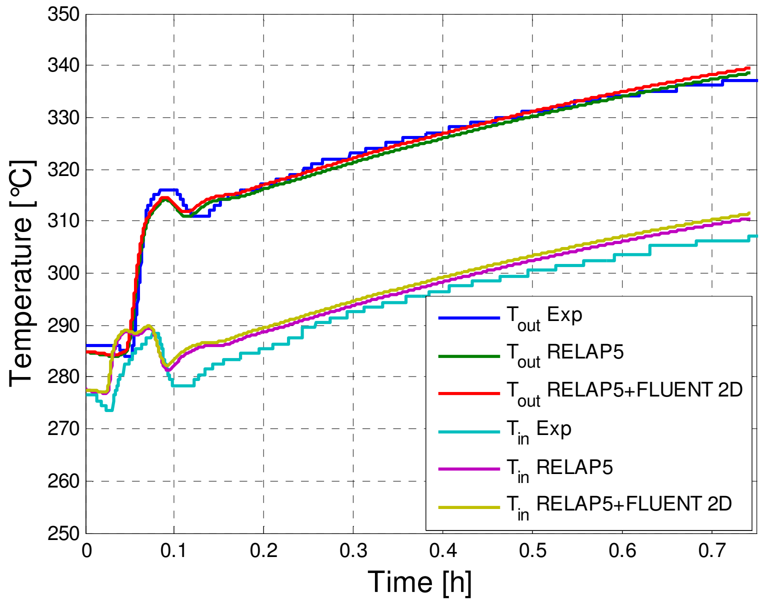

Figure 2: FPS inlet and outlet sections temperature for Test 301

Figure 3 shows the temperature trends at the inlet and outlet sections of the heat exchanger for the primary LBE side. Again, as mentioned for the FPS inlet and outlet temperature differences at the beginning of the transient (see Figure 3) are due to the simplified temperature distribution imposed at the beginning of the simulation as boundary condition in the RELAP5 nodalization.

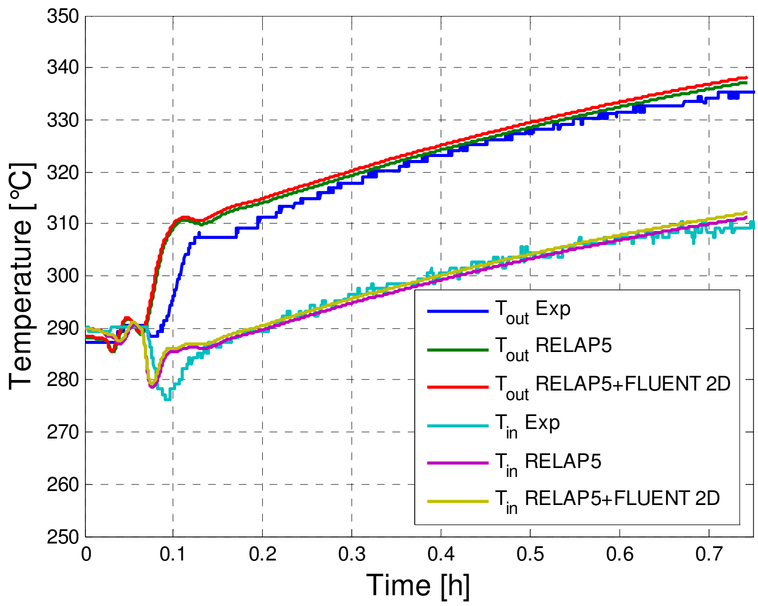

Figure 3 *HX* inlet and outlet section temperatures for Test 301

In Figure 4 temperature distribution inside the 2D domain (Fluent domain for the FPS section) is reported at t = 0.74 h (2680 s) from the beginning of the transient. This instant corresponds to the maximum average temperature time reached at the outlet section of the FPS domain (see Fig. 12). The maximum temperature reached near the heated wall is in the order of 384°C.

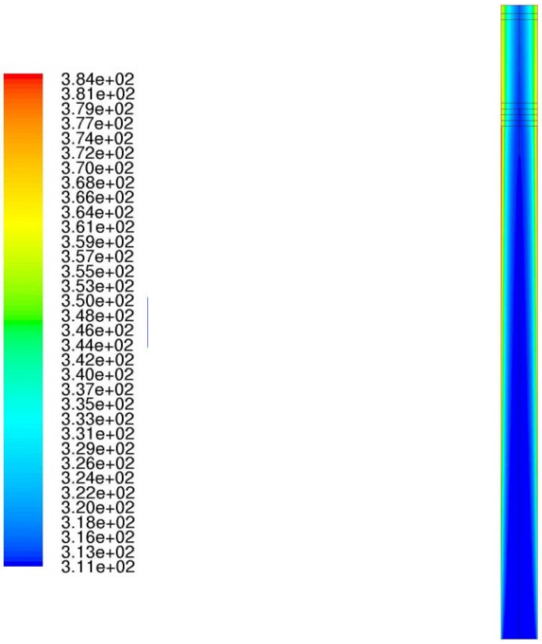

Figure 4: Temperature contour plot [°C]

3. Coupling procedure improvements

In the previous works performed in the framework of PAR2011 and PAR2012 (Martelli et al., 2011 and Martelli et al., 2012) the developed coupling tool was an Explicit coupled scheme in which the b.c. evaluated at the previous time step were exchanged at the interface and used as b.c. for the next time step. The advantage of this method lies in its simplicity of implementation on the other hand exchanging data only after the closure of the time step can be penalizing for the simulation stability, hence the need to use lower time step values.

In this work performed in the framework of the new PAR2013, improvements in the coupling procedure were developed in order to improve the stability of the method and to reduce computational efforts.

In particular, an implicit coupling scheme was developed and, moreover, the User Defined Function (UDF) wrote for the Fluent CFD code to ménage the data exchange at boundaries was parallelized giving in this way the possibility to work with multiple processor, with both explicit and implicit coupling scheme. Another important improvement developed for both the explicit and implicit schemes is the way how the MATLAB code obtains the data to be exchanged from the RELAP5 code. In the new versions of the coupling scheme MATLAB can access directly into the restart RELAP5 file (name.rst file) and save data to be passed to the Fluent code. In the previous version of the coupling scheme the RELAP5 data to be passed to the Fluent code were read in the output file (name.o file). We noted that the data wrote in the output file were rounded with respect to data directly read into the restart file, improving in that way the accuracy of the solution.

3.1 Implicit Coupling scheme

The basic idea behind the implicit scheme is to repeat each time step several times with update b.c at each "inner -cycle" until specified convergence criteria are satisfied; after that both codes proceed to compute b.c. for the next time step. Variables exchanged at each inner cycle and at each time step are pressure, temperature and mass flow rate according to the scheme showed in Figure 5.
The implicit method shown in Figure 6 is more difficult to implement on the other and any disturbances is limited by the feedback at each sub cycle leading to a stronger numerical stability and allowing the use of relatively larger time step with respect to the explicit coupling scheme.

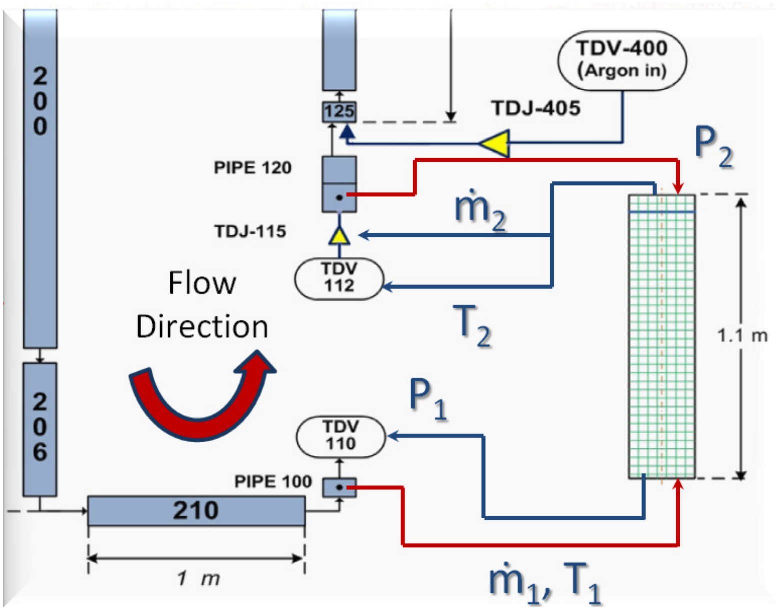

Figure 5: RELAP5-FLUENT data exchange

Each inner iteration can be repeated until specified convergence criteria are satisfied or, for a simplified programming, setting a fixed number of inner iterations for each time step. For the performed simulations a fixed number of inner iteration was imposed and from a first sensitivity analysis a total of 3 inner iteration per each time step was chosen as a good compromise between CPU time and accuracy of results.
The Fluent code (master code) advances firstly by one time step and then the RELAP5 code (slave code) advances for the same time step period, using data received from the master code. In particular, for each of the three RELAP5 boundary condition data, a linear interpolation inside the time step period between the initial value (final value of the previous time step) and the final value of the current time step (obtained by the Fluent code calculation) is considered for RELAP5. In the Fluent code, instead, b.c. are considered fixed in the time step, hence, for each inner iteration the b.c. imposed in the Fluent code are averaged between the previous and at the current iteration.

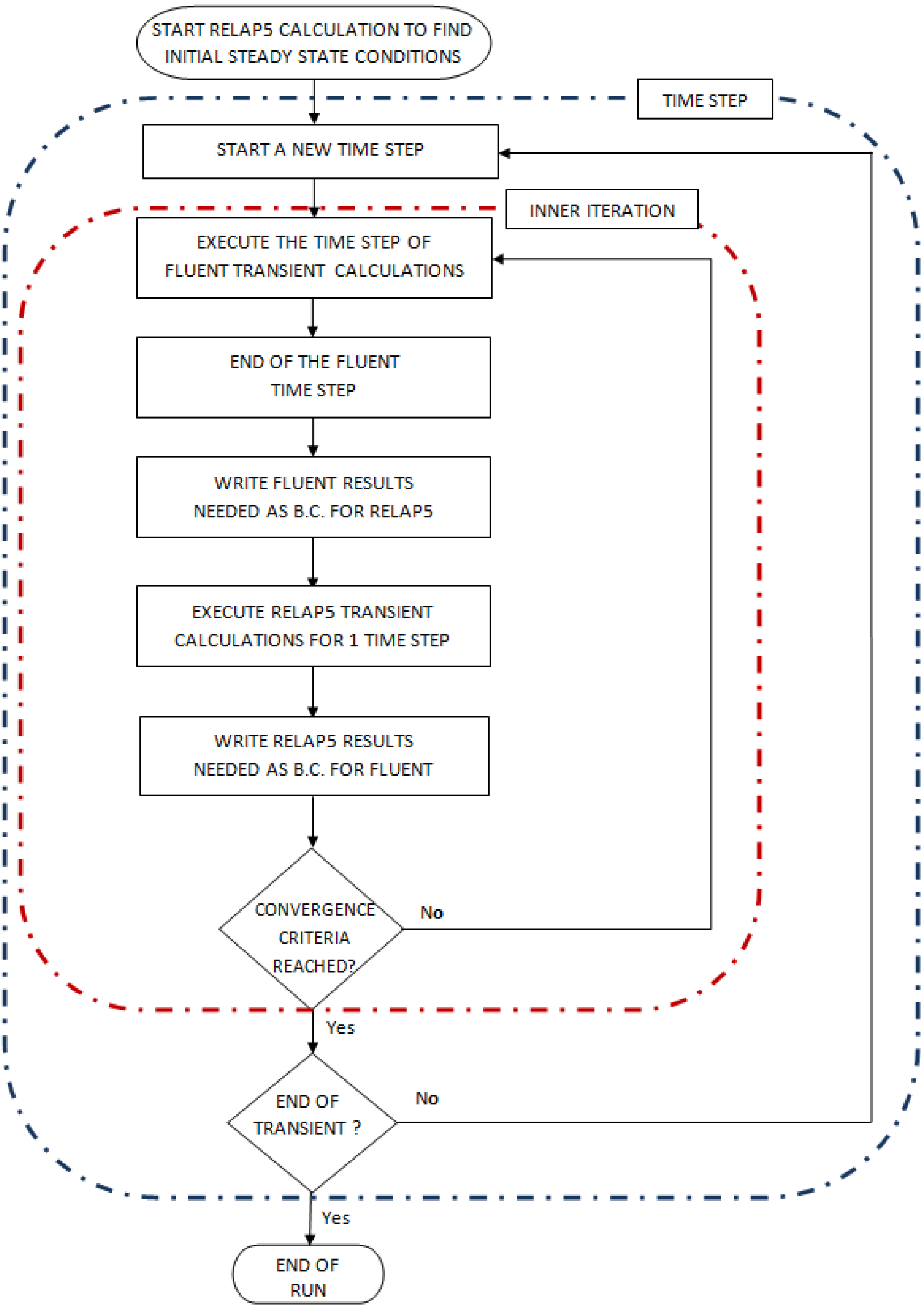

Figure 6: Implicit coupling scheme

3.2 Parallelization of the UDF

The Fluent serial solver is essentially composed by a Cortex and a single Fluent process. The Cortex is the Ansys Fluent process responsible for user-interface and graphics related functions. The Fluent parallel solver, instead computes the solution using simultaneously multiple processor splitting up the computational domain into multiple partitions and assigning each data partition to a different compute process (compute node). The fluent parallel architecture is composed by the Cortex a Host a Compute node-0 and *n* Compute node-*n*.

The Host primary purpose is to interpret commands from Cortex and to pass those commands to Compute node-0 which then distributes them to the other computer nodes (see Figure 7). The Cortex and the Host do not have any data.

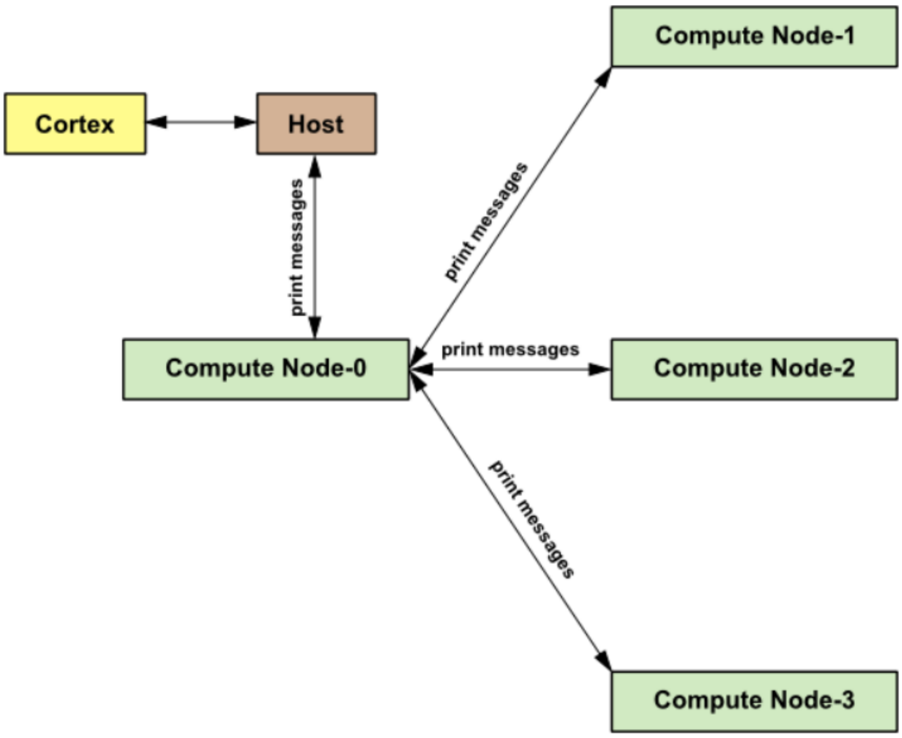

Figure 7: Example of Fluent parallel architecture

An UDF needed to be parallelized when it performs operations that require information located on different compute nodes, such type of operations are operations involving summation or addition (integration) commonly performed in general purpose define macros such as DEFINE_ADJUST, DEFINE_EXECUTE_AT_THE_END, etc.

When a UDF is converted to run in parallel, some part of the script may need to be done by the host and some other by the compute nodes. As an example, since the host does not contain mesh data, it has to be not included in any calculation that otherwise will result in NaN value. At the same time when writing files in parallel, the file must be opened by the Host, then Compute nodes must send their data to Compute node-0 which collect the data and sends them to the host which write it to the file and then close the file.

4. Forced Circulation Test

The new developed implicit coupling scheme was adopted to simulate the experimental test named Test 206 (Di Piazza and Martelli, 2013, Martelli et al., 2013) representative of a gas enhanced circulation test and reported in Table 1.

Table 1: Forced circulation test

Name	T_{AV} [°C]	**FPS Power**	**Glift [Nl/min]**	**Monitored variables**
Test 206	200-250	0	2,4,5,6,8,10,6,5,4,2	LBE flow rate P_{in} and P_{out} in the *HS*

The geometrical domains adopted for the 2D and 3D simulations and their spatial discretizations are described in AdP2012. A total of five simulations were performed, involving both 2D and 3D geometrical CFD domains (Martelli et al., 2013). The test matrix of the performed simulations are reported in Table 2.

Table 2: Matrix of performed simulations

Name	***Time Step***	**CFD Geometrical Domain**	**Serial/Parallel**
Test 206-0	0.005 s	2D	serial
Test 206-1	0.025 s	2D	"
Test 206-2	0.025s	2D	Parallel
Test 206-3	0.025 s	3D	Serial
Test 206-4	0.025 s	3D	Parallel

The LBE mass flow rate time trends are reported in Figure 8, were the experimental data are compared with calculated results obtained using for the Explicit (developed in the previous AdP2012) and Implicit coupled scheme and with RELAP5 stand alone calculation. Both simulations adopted the same time step (0.005 s) in order to verify the behaviour of the new Implicit coupling version. Differences in the LBE mass flow rate obtained using the Explicit and the Implicit coupling tool are lower than 1%, while results of the coupled simulations overestimate results of RELAP5 stand alone calculation by less than 5%.

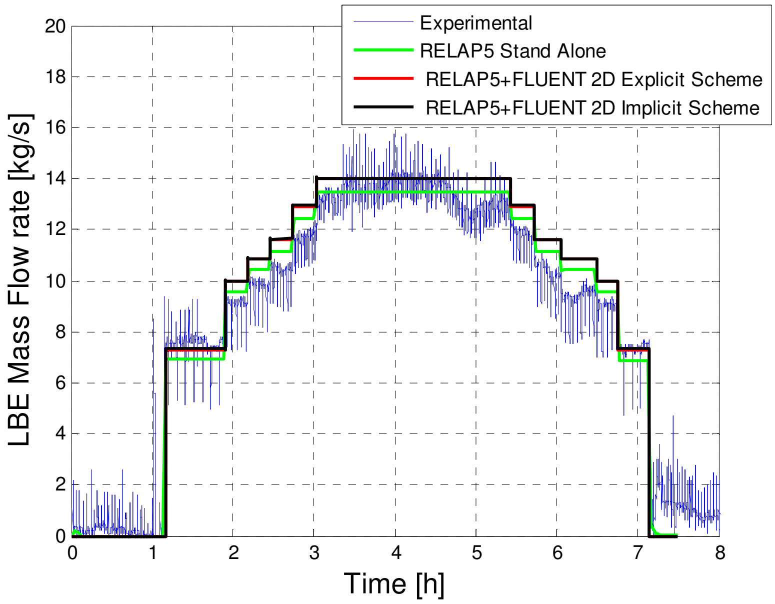

Figure 8:LBE Mass Flow Rate, Explicit Vs Implicit Coupling scheme

A first sensitivity analysis showed that using the Implicit coupling scheme, the time step adopted can be increased up to 0.025 s (five time greater than those adopted for the Explicit scheme) without losing in accuracy of the results. Instead, using a time step of 0.025 s with the Explicit coupling scheme, calculation showed stability issues.

Figure 9 show results obtained for the forced circulation, Test 206, applying the new Implicit coupling tool. Good agreement was found among the performed simulation with 2D axis symmetric and 3D symmetric CFD geometrical domains and with serial and parallel CFD solver. Obtained LBE mass flow rate time trends are practically overlapped for all the performed simulations with differences with the stand-alone RELAP5 by less than 5% and by less than 12% with respect to the experimental flow rate, according to results obtained for the simulations with the Explicit coupling scheme.

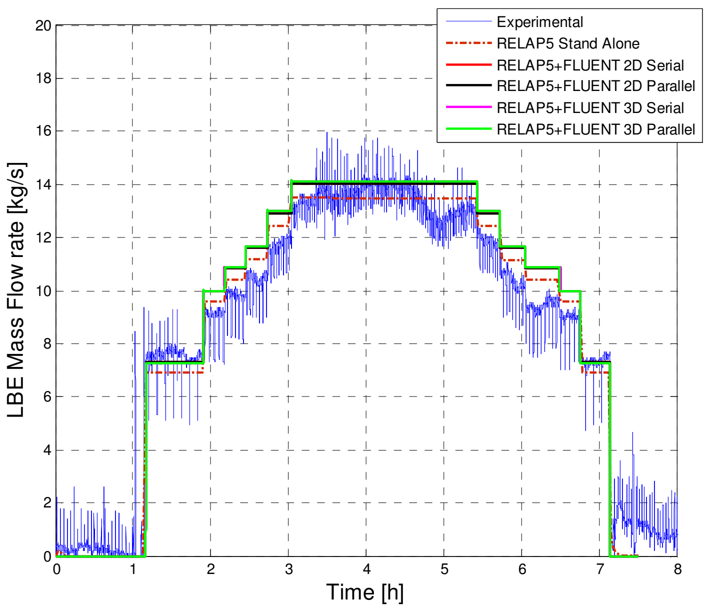

Figure 9: LBE mass flow rate results for Test 206

In Figure 10 the pressure difference between the inlet and outlet section of the FPS is showed. The maximum differences between the RELAP5 stand alone data and the coupled simulations is lower than 1%.

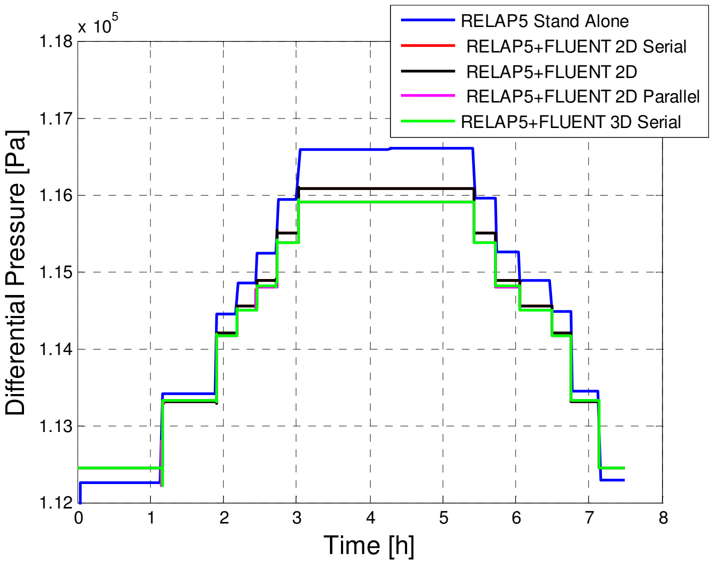

Figure 10: FPS pressure difference for Test 206

Figure 11 summarizes results obtained for the simulations carried out in the frame of the AdP2012 and 2014. In particular, calculated LBE mass flow rate is plotted versus the experimental mass flow rate for RELAP5 stand-alone calculations and RELAP5/Fluent coupled simulations performed both with 2D and 3D CFD computational domains and with the Explicit and Implicit coupling scheme. Test 306 (see Martelli et al. 2013) is a forced circulation test analogous to Test 206 but with different initial temperature conditions, which implies different initial pressures (due to different buoyancy forces). As it can be noted from the plot, calculated results satisfactory predict the experimental data (most of the obtained results lies in a range between +10% and -10%) with a trend that generally tend to overestimate the experimental LBE mass flow rate.

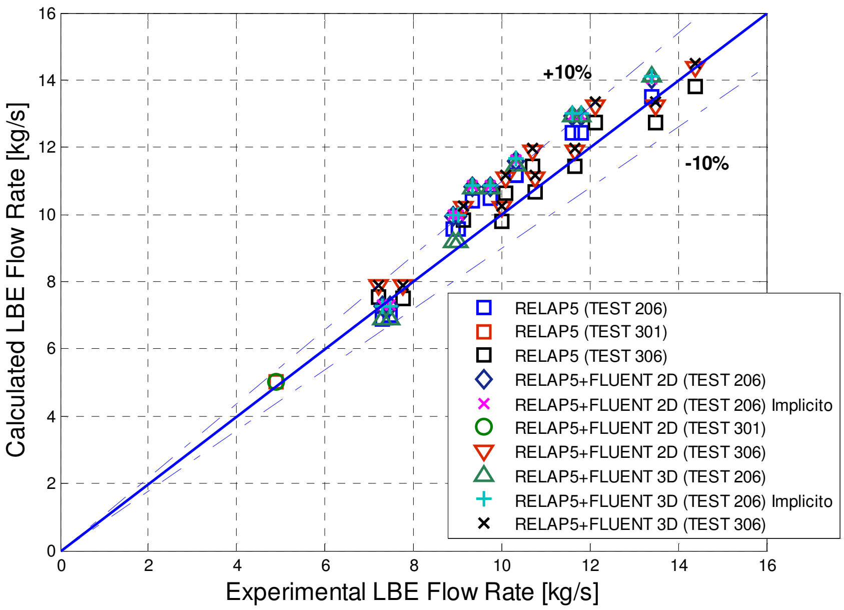

Figure 11: Experimental LBE mass flow rate Vs calculated LBE mass flow rate

5. Conclusions

In this work, carried out at the DICI (Dipartimento di Ingegneria Civile e Industriale) of Pisa University, is presented an in house developed coupling method between the RELAP5/Mod3.3 thermal-hydraulic system code, modified to allow the use of HLM as coolant, and the CFD Fluent commercial code. In particular, in the first part of the work Explicit coupling scheme developed in the previous AdP2011 & 2012 is applied to the simulation of an experimental test carried out on the NACIE loop type facility (hosted at the ENEA Brasimone research center). The performed simulation is related to Test 301, representative of natural circulation test characterized by a heating power of 21.5 kW increased linearly in the first 262 s of the transient and then maintained constant. Both RELAP5 stand-alone and RELAP5/Fluent coupled calculations are presented. The geometrical domain adopted for the portion of the loop simulated by the Fluent CFD code is a 2D axis symmetric domain. Results obtained from the RELAP5 stand alone simulation and from the coupled simulation are practically overlapped while the mass flow rate predicted by the simulations at steady state condition is about 5.03 kg/s, overestimating the mass flow rate obtained from the energy balance equation by less than 2%. Also temperature al the inlet and outlet section of the heater and of the heat exchanger are investigated. In particular at the inlet section of the fuel pin bundle, numerical results tend generally to overestimate the experimental temperature by less than 2%.

In the second part of this work improvements in the coupling procedure previously developed (AdP2011 & 2012) are reported. In particular, in order to improve the stability of the method and to reduce computational efforts, an Implicit coupling scheme is developed and implemented. Moreover, the User Defined Function (UDF) wrote for the Fluent CFD code to ménage the data exchanged at boundaries is parallelized giving in that way the possibility to the CFD code to work with multiple processor both for the Explicit and Implicit coupling scheme. Finally in the new versions of the coupling tool, MATLAB access directly into the restart RELAP5 file (name.rst file) and save data to be passed to the Fluent code avoiding in that way round-off problem found reading in the output file (name.o file). Going into details of the Implicit scheme, each inner iteration can be repeated until specified convergence criteria are satisfied or, for a simplified programming, setting a fixed number of inner iterations for each time step. In the performed simulations a fixed number of inner iteration is imposed and from a first sensitivity analysis a total of 3 inner iterations per each time step is chosen as a good compromise between CPU time and accuracy of results.

The new developed Implicit tool was then adopted to simulate the experimental test named Test 206 representative of a forced circulation test. A simulation with the same time step of those adopted for the Explicit scheme (0.005 s) is performed and used to check the behaviour of the Implicit scheme. Obtained results indicate that the Implicit coupling scheme was correctly implemented. After that a sensitivity analysis was carried out showing that, tanks to the increased stability of the Implicit scheme, the time step used for calculations could be increased up to 0.025 s without losing in accuracy of the results and therefore significantly reducing the computational time. Finally simulations of Test 206 are performed adopting both a 2D axis symmetric and 3D symmetric CFD domains and with serial and parallel solver. Obtained results for LBE mass flow rate, show differences with the stand-alone RELAP5 by less than 5% and by less than 12% with respect to the experimental flow rate, according to results obtained for the simulations with the Explicit coupling scheme.

References

H. Austregesilo, B. Bals, A. Hora, G. Lerchl, P. Romstedt, "*ATHLET Mod 2.1 Cycle A Models and Methods*", code documentation, July 2006.

R. Bavière, N. Tauveron, F. Perdu , E. Garré, "*System-CFD Coupled Simulation of the PHENIX Reactor Natural Circulation Test*", proceedings of NURETH 15, Pisa, Italy, May 12-17,2013.

D. Bertolotto, A. Manera, S. Frey, H.M. Prasser, R. Chawla, "*Single-phase mixing studies by means of a directly coupled CFD/system-code tool*", Annals of Nuclear Energy, 36 pp. 310-316, 2009.

D. Bestion, "*Extension of CFD codes application of two-phase flow safety problems*", Nucl. Eng. Tech., Vol.42 pp 365-376, 2010.

C.B. Davis, A.S. Shieh, "*Overview of the use of ATHENA for Thermal-Hidraulic Analysis of System with Lead-Bismuth Coolant*", proceedings of ICONE 8, Baltimore MD USA April 2-6, 2000.

I. Di Piazza, D. Martelli, "*Experimental circulation tests on a prototypical inductive flow meter with the NACIE loop*", Report ENEA NA-T-R-029, 2013.

G. Geffraye, O. Antoni, M. Farvacque, D. Kadri, G. Lavialle, B. Rameau, A. Ruby, "*CATHARE 2 V2.5_2: A single version for various applications*", Nucl. Eng. and Des. 241, 4456-4463, 2011.

M. Jeltsov, K. Kööp, W. Villanueva, P. Kudinov, "*Multi-scale Coupling of STH and CFD Codes for Pre-Test Analysis of TALL-3D Experiments*", THINS International Workshop

M.H.C. Hannink, A.K. Kuczaj, F.J. Blom, J.M. Church, E.M.J. Komen, "*A coupled CFD-FEM strategy to predict thermal fatigue in mixing tees of nuclear reactors*" EUROSAFE Forum, 2008.

IAEA-TECDOC-1539, "*Use and Development of Coupled Computer Codes for the Analysis of Accidents at Nuclear Power Plants*", proceedings of a technical meeting held in Vienna 26-28 November 2003.

D. Martelli, N. Forgione, G. Barone, A. Del Nevo, "*Pre-test analysis of thermal-hidraulic behaviour of the NACIE facility for the characterization of a fuel pin bundle*", Adp MSE-ENEA LP3-C4, 2012.

D. Martelli, N. Forgione, G. Barone, W. Ambrosini, "*System codes and a CFD codes applied to loop- and pool-type experimental facilities*", Adp MSE-ENEA LP2-C1, 2013.

A. Papukchiev, G. Lerchl, "*Extension of the Simulation Capabilities of the 1D System Code ATHLET by Coupling with the 3D Software Package ANSYS CFX*", proceedings of NURETH13, Kanazawa City, Ishikawa Prefecture, Japan September 27 - October 2, 2009

Modena, Italy, January 20-22, 2014

RELAP5-3D code development Team,"*RELAP5-3D© Code Manual*", INEEL-EXT-98-00834, Revision 4.1, September 2013.

RELAP5/Mod.3.3 Code Manual, Volume II. Appendix A: Input Requirements, Nuclear Safety Analysis Division, January 2003.

R. Schultz, W.L. Weaver, D. Schowalter, N. Basu, "*Evaluating Fluid Behavior in Advanced Reactor Systems using Coupled Computational Fluid Dynamics and Systems Analysis Tools*", proceedings of ICONE 13, Beijing China, 2005.

V. Sobolev, "*Database of thermophysical properties of liquid metal coolants for GEN-IV*", November 2010 (rev. Dec. 2011), SCK-CEN, Belgium.

Ch. L. Waata, Th. Frank, "*Coupling of ANSYS CFX with 1D System Code ATHLET*", Final Report, German Federal Ministry for Economy and Technology, Reactor Safety Research Project 1501328, Germany, 2008.

T. Watanabe, Y. Anoda, M. Takano, "System-CFD coupled simulations of flow instability in steam generator U tubes", Annals of Nuclear Energy, 70 pp. 141-146, 2014.

W. L. Weaver, E. T. Tomlinson, D. L. Aumiller, "*A generic Semi-Implicit Coupling Methodology for use in RELAP5-3D©*", Nucl. Eng. Des., 211 pp. 13-16, 2002.

Y. Yan, Rizwan-uddin, "*Coupled CFD-System-code Simulation of a Gas Cooled Reactor*", International conference on Mathematics and Computational Methods Applied to Nuclear Science and Engineering, Rio de Janeiro, Brazil, May 8-12, 2011.

Nomenclature

Roman letters

$\dot{m}_1$	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]

Abbreviations and acronyms

AdP	Accordo di Programma
API	Application Programming Interface
ASTRID	Advanced Sodium Technological Reactor for Industrial Demonstration
ATHLET	Analysis of THermal-hydraulics of LEaks and Transients
CATHARE	**C**ode for **A**nalysis of **TH**ermal hydraulics during an **A**ccident of **R**eactor and safety **E**valuation
CEA	Commisariat à l'Energie Atomique et aux Energies Alternatives
CFD	Computational Fluid Dynamic
CPU	Central Process Unit
DICI	Dipartimento di Ingegneria Civile e Industriale
Glift	Gas Lift mass flow rate
GEN IV	Generation four
ENEA	Agenzia nazionale per le nuove tecnologie, l'energia e lo sviluppo sostenibile
FPS	Fuel Pin Simulator
GRS	Gesellschaft für Anlagen- und Reaktorsicherheit
GT-MHR	Gas Turbine Medium Helium Reactor
HLM	Heavy liquid metal
HTC	Heat transfer coefficient
HS	Heat Source
HX	Heat Exchanger
KTH	Kungliga Tekniska Högskolan
IAEA	International Atomic Energy Agency
ICoCo	Interface for Code Coupling
LB-LOCA	Large Break Loss Of Coolant Accident
LBE	Lead bismuth eutectic
LES	Large Eddy Simulation
NACIE	Natural Circulation Experiment
NaN	Not a Number
PAR	Piani Annuali di Realizzazione
PSI	Paul Scherrer Institut
RANS	Reynolds Averaged Navier Stokes equations
RELAP	Reactor Loss of Coolant Analysis Program
SB-LOCA	Small Break Loss Of Coolant Accident
SFR	Sodium Fast Reactor
STH	System Thermal Hydraulic
T_{AV}	Average Temperature
TECDOC	TEchnical DOCument
TRACE	TRAC/RELAP Advanced Computational Engine
TRIO_U	TRIO Unitaire
UDF	User Defined Function
URANS	Unsteady Reynolds Averaged Navier Stokes equations

Breve CV del gruppo di lavoro

Ambrosini Walter

Professore associato per l'SSD ING-IND/19 (Impianti nucleari) presso l'università di Pisa. Autore di oltre 150 pubblicazioni su riviste internazionali, conferenze nazionali ed internazionali. Presidente dell'associazione *European Nuclear Education Network;* presidente del Corso di dottorato In Ingegneria Nucleare e della sicurezza industriale presso il DICI dell'Università di Pisa; presidente del corso di laurea magistrale in Ingegneria Nucleare presso l'Università di Pisa.
(http://www.dimnp.unipi.it/walter-ambrosini/index1.htm)

Nicola Forgione

Ricercatore in Impianti Nucleari presso il Dipartimento di ingegneria Meccanica, Nucleare e della Produzione (DIMNP) dell'Università di Pisa dal 20 dicembre 2007. Laureato in Ingegneria Nucleare nel 1996 presso l'Università di Pisa ed in possesso del titolo di Dottore di Ricerca in Sicurezza degli Impianti Nucleari conseguito all'Università di Pisa nel 2000. La sua attività di ricerca è incentrata principalmente sulla termofluidodinamica degli impianti nucleari innovativi, con particolare riguardo ai reattori nucleari di quarta generazione. Autore di oltre 20 articoli su rivista internazionale e di numerosi articoli a conferenze internazionali.
(http://www.dimnp.unipi.it/forgione-n/)

Gianluca Barone

Si è laureato in Ingegnere Nucleare all'Università di Roma nel 2004 ed ha conseguito il Master di II livello in "Tecnologie degli Impianti Nucleari" presso l'Università di Pisa nel 2011. Attualmente borsista presso il DICI dell'Università di Pisa. Esperto di codici di sistema, con particolare riguardo al codice di termoidraulica nucleare RELAP5.

Daniele Martelli

Ha conseguito la laurea in Ingegneria Aerospaziale presso l'Università di Pisa nel 2009. A partire dallo stesso anno ha iniziato a collaborare, attraverso la società spin-off ACTA, con il DIMNP per analisi di fluidodinamica computazionale nell'ambito dell'ingegneria nucleare. Nel biennio 2010-2011 ha usufruito di una borsa di ricerca presso il DIMNP riguardante l'analisi del comportamento termoidraulico dei sistemi di refrigerazione dei reattori nucleari refrigerati a metallo liquido. Da gennaio 2012 è inscritto al corso di dottorato in "Ingegneria Nucleare e Sicurezza Industriale" in svolgimento presso il DICI dell'Università di Pisa. La sua attività di ricerca è principalmente incentrata sullo studio dei fenomeni di scambio termico in regime di convezione naturale e mista per reattori nucleari refrigerati a metallo liquido pesante.