INVESTIGATIONS ON THE GAS DISTRIBUTION PHENOMENA INSIDE THE CONTAINMENT SYSTEM OF LWRs

A. Manfredini°, F. Oriolo°, S. Paci*, A. Villotti°
° Dipartimento di Costruzioni Meccaniche e Nucleari, via Diotisalvi, 2 - 56126 Pisa (I)
* Themas srl, via P. Landi, 9 - 56100 Pisa (I)

ABSTRACT

The importance of mixing and distribution phenomena of hydrogen gas in the reactor safety is emphasised in the advanced reactor concepts, that heavily rely upon the passive cooling systems during a typical severe accident sequence. An advanced methodology for evaluating the temporal and spatial distribution of non condensable gases, including the simulation of buoyancy-driven flows and the effects of the various ESFs activation, in a multicompartment containment system of a LWR is reviewed. The methodology employs an analogy technique with electrical networks to determine the convection flows among the containment compartments and evaluates, inside a single node, the profile of the vertical concentrations of steam and non condensable gases. The application of the proposed models to simulate the gas distribution phenomena occurring in the HDR E11.2, in the FIPLOC-F2 and in the NUPEC M-7-1 tests demonstrates the importance of these models providing information about local details and spatial distribution. The main results from the post-test analysis performed to simulate the thermal-hydraulic responses of the above mentioned experiments are presented and demonstrate the improvements and the reduction of the error band with respect to the experimental data. This methodology allows to perform a realistic prediction of severe accident sequence inside the containment system of the actual and advanced passive generation of LWRs.

1. INTRODUCTION

A fundamental item to obtain a realistic description of the overall thermal-hydraulic transient and, in particular, of the gas distribution and aerosol behaviour in a LWR multicompartment containment system is the simulation of buoyancy-driven flows, due to density gradients or to the ESFs activation [1]. The ability to evaluate the mixing and distribution phenomena of hydrogen gas, released in the containment during a severe accident in a LWR, is of primary importance for various aspects of the containment design and safety analysis, particularly during the long term phase of a severe sequence [2]. Besides, natural circulation is the most important way to remove the decay heat and therefore to maintain the integrity of the system in the new advanced nuclear power plants. The modelling of buoyancy-driven flows plays therefore a fundamental role in the source term evaluation and for the design and the operation of mitigative safety features.

A complex theoretical and experimental activity has been performed by the international community in order to investigate the phenomenology that takes play during a severe accident and to obtain a systematic set of data for the assessment of the numerical models developed for the simulation of these transients. Of noticeable interest are the experiments performed on the HDR facility, fundamental for the analyses of the hydrogen stratification inside a multicompartment system [3] and the FIPLOC-F2 test focused on the natural circulation phenomena during a typical severe accident sequence [4]. The study of NUPEC test [5], in comparison with the HDR E11.2 one, gives an opportunity to study the influence of the internal spray activation on the gas distribution against the effects of an external spray system in well-mixed conditions. These tests also highlight the limitations of the present containment codes in analysing the thermal-hydraulic transient inside a complex multicompartment containment system. The usual approach in reactor safety analysis is to develop lumped parameter codes, which divide containment space into control nodes and resolve inside the node the usual equations for the conservation of energy, mass and momentum, but are insufficient to provide information about local details and spatial distribution. As will be shown in the following, the hydrogen mixing and distribution phenomena, including the effects of stratification, can be simulated very effectively using the new version of the FUMO code, that includes new models able to overcome some limitation of lumped parameter approach. In the previous flow models the driving-force was based only on the pressure difference, not taking into account of the gravitational or density effects. To evaluate such buoyancy-driven flows, specific models were developed for the FUMO code.

2. OVERVIEW OF THE FUMO CODE

The FUMO code [1], [6] was developed as a part of an ongoing severe accident research program by the DCMN of the Pisa University. The purpose of the code is to provide a best estimate tool for the analysis of containment system under severe accident conditions. In particular, it is able to predict pressures, temperatures and gas distributions within the containment for assessing loads and associated threats to the system integrity. The major phenomena that are simulated include: intercell flow, heat and mass transfer processes and simulation of emergency safety features, as the internal spray system. Specific models for the simulation of the cooling of the external surface of the containment liner by means of a gravity-driven spray [7] and of the internal "pool management" provide FUMO with the capability to analyse a wide range of both LWRs and advanced plants' accident sequences [8]. The flexibility of the code also allows for the simulation of experimental facilities and other non-standard configurations. An integrated version of the code is able to perform coupled analysis of primary and containment systems [9]; this version includes models related to hydrogen burning, simulation of direct heating of the containment [10] due to the heat debris released during a high pressure severe accident and corium-water interaction.
The FUMO code uses a multicompartment configuration that allows an arbitrary arrangement of control volumes and flow paths. The junction flow-rate is derived from the acceleration flow model, with the option of being quasi-steady, and can be evaluated in three different ways:

1) from the pressure difference across the flow-path, considering the inertia of the mixture as well as the frictional resistance. It can be also calculated by neglecting the inertia along the junction: the flow rate is assumed to come instantly to the steady-state value, appropriate to the pressure quasi-steady flow model;
2) for the flow regime characterised by small pressure differences compared to the pressures in the cells (situation typical of the long term phase) a fast running option, based on a semi-implicit model, has also been developed.

2.1 Evaluation of Buoyancy Driven Flows

To evaluate buoyancy-driven flows, specific models have been developed for the FUMO code. Using these models it is possible to overcome some limits of lumped parameter approach.
In the first model for the description of a closed loop, a realistic evaluation of the gravitational head developed within two consecutive compartments of a branch is performed. With reference to the situation presented in Fig. 1, the integration of density along the considered path is performed as indicated in Eq. (1).

$$\int_A^B \rho(x)\ dx = \rho_i\,(h_{i1} - h_{i2}) + \rho_j\,(h_{j1} - h_{j2}) + \int_{h_{i2}}^{h_{j1}} \bar{\rho}\ dx \qquad (1)$$

The integration in a cell is therefore between the two ends of the flow path linked to the cell, with the hypothesis of a constant density of the atmosphere, while the integration along the junction that links two consecutive nodes is performed relating to an average value of the densities of the two connected cells. The natural circulation flow-rate in a single closed loop is determined integrating the momentum equation. The value of convective flow Q_{nc} is given by Eq. 2:

$$Q_{nc} = A_m \sqrt{\frac{2\,\rho_m\, g\,(\Delta\xi_1 - \Delta\xi_2)}{1+\lambda\frac{H}{D}+\Sigma\lambda_i}} \tag{2}$$

with $\Delta\xi_i = \int_{\text{branch } i} \rho(x)\, dx$ (2.1)

where:

A_m average flow area of the loop;
D junction hydraulic diameter;
H nodes relative elevation;
g acceleration of gravity;
λ_i concentrated loss coefficients;
λ distributed loss coefficients;
ρ_m density in the closed loop.

The convective flow may be also written as:

$$Q_{nc}\, K_i = \sqrt{\Delta\xi_1 - \Delta\xi_2} \qquad \text{where } K_i = \frac{1}{A_m}\sqrt{\frac{1+\lambda\frac{H}{D}+\lambda_i}{2\,\rho_m\, g}} \tag{3}$$

The momentum balance equation is solved for the whole nodalization using an electrical analogy. First the system is divided into a number of finite sized branches, then a resistance-capacitance network, with a proper generator, is used to represent the region. In this analogy the natural circulation flow Q_{cn} is equivalent to the current in a single network, the factor K_i, that represents the friction losses along the branch, is equivalent to the sum of electrical resistances present in the network, while the "driving force" $\sqrt{\Delta\xi_1-\Delta\xi_2}$, due to the difference of the integral of density along the two branches, is equivalent to a direct current generator. Finally, each node of the system is equivalent to an electrical capacitance. This analogy allows for the evaluation of convective flows also in complex containment geometries.

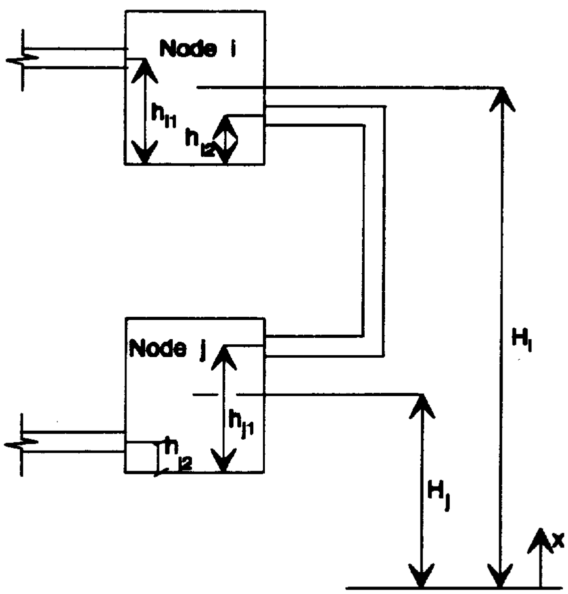

Fig. 1: Branch of a closed loop.

Together with the model for the evaluation of the natural circulation flow, a model for the simulation of buoyancy and diffusive flows due to gas and steam stratification inside a large containment volume has been also implemented in the FUMO code. This model is organised in two temporal phases: in the first one the buoyancy mass flow towards the upper zone of the analysed node is described while, in the second one, the homogenisation of the gas and steam concentrations, due to the diffusion process towards the lower zone of the volume, is simulated. The first phase of this stratification process is analysed by dividing the considered control volume into n vertical zones and resolving, for each zone, a balance equation for the specific flow-rate of considered gas moles. In this way it is possible to obtain a vertical distribution of the gas or steam concentrations in the form of the Eq. 4.

$$C(x) = C_0 e^{\theta x} \quad (4)$$

$$\text{with} \quad \theta = \frac{n}{H} \ln K \quad (4.1)$$

where:
C_0 gas concentration at elevation 0;
$C(x)$ gas concentration at the height x;
x direction along the volume height;
n number of zones;
H volume height;
K function of gas velocity and geometry.

The second phase simulates the gas molecular diffusion due to the gradient of the concentration inside a single control volume. The Fick law for the diffusion (Eq. 5) is resolved at each time-step, in a one-dimensional geometry, using a completely implicit method of numerical solution.

$$\frac{\partial C(t,x)}{\partial t} = D \frac{\partial^2 C(t,x)}{\partial x^2} \quad (5)$$

where:
C (t, x) gas concentration;
t time;
x vertical abscissa;
t_0 beginning of the time step.

3. RESULTS OF THE NEW MODELS

The new models for the evaluation of the gas distribution and mixing have been applied to the analyses of three international experiences: the large scale HDR test E11.2 [11], the FIPLOC-F2 test [4] and the NUPEC M-7-1 test [5].

3.1 FIPLOC F2 Test

The major objective of this test was to investigate the thermal-hydraulic long term phenomena, with special emphasis on natural convection phenomena in a containment with a loop-type geometry. The natural circulation flow is affected by variations of steam and air injections at different locations as well as dry energy supply into the various compartments. The loop-type geometry used in FIPLOC F2 test has some analogies with the large loop in a PWR containment formed by the two steam generator compartments, with connecting flow paths at their lower ends, and with the large dome compartment at the top. It consists of the central injection zone at a low level, a dome and, in

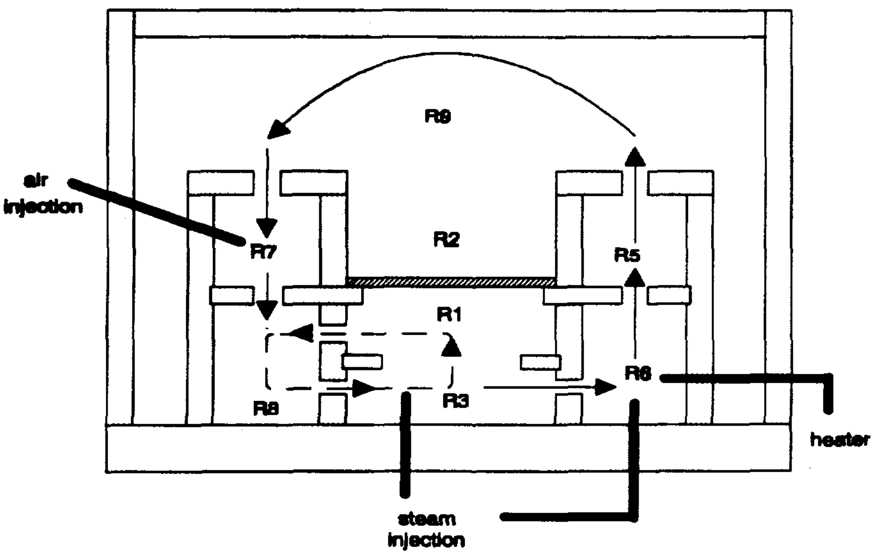

Fig. 2: Natural circulation flows during FIPLOC F2 test.

into n vertical zones and resolving, for each zone, a balance equation for the specific flow-rate of considered gas moles. In this way it is possible to obtain a vertical distribution of the gas or steam concentrations in the form of the Eq. 4.

$$C(x) = C_0 e^{\theta x} \quad (4)$$

$$\text{with} \quad \theta = \frac{n}{H} \ln K \quad (4.1)$$

where:

C_0	gas concentration at elevation 0;
$C(x)$	gas concentration at the height x;
x	direction along the volume height;
n	number of zones;
H	volume height;
K	function of gas velocity and geometry.

The second phase simulates the gas molecular diffusion due to the gradient of the concentration inside a single control volume. The Fick law for the diffusion (Eq. 5) is resolved at each time-step, in a one-dimensional geometry, using a completely implicit method of numerical solution.

$$\frac{\partial C(t,x)}{\partial t} = D \frac{\partial^2 C(t,x)}{\partial x^2} \quad (5)$$

where:

C (t, x)	gas concentration;
t	time;
x	vertical abscissa;
t_0	beginning of the time step.

3. RESULTS OF THE NEW MODELS

The new models for the evaluation of the gas distribution and mixing have been applied to the analyses of three international experiences: the large scale HDR test E11.2 [11], the FIPLOC-F2 test [4] and the NUPEC M-7-1 test [5].

3.1 FIPLOC F2 Test

The major objective of this test was to investigate the thermal-hydraulic long term phenomena, with special emphasis on natural convection phenomena in a containment with a loop-type geometry. The natural circulation flow is affected by variations of steam and air injections at different locations as well as dry energy supply into the various compartments. The loop-type geometry used in FIPLOC F2 test has some analogies with the large loop in a PWR containment formed by the two steam generator compartments, with connecting flow paths at their lower ends, and with the large dome compartment at the top. It consists of the central injection zone at a low level, a dome and, in

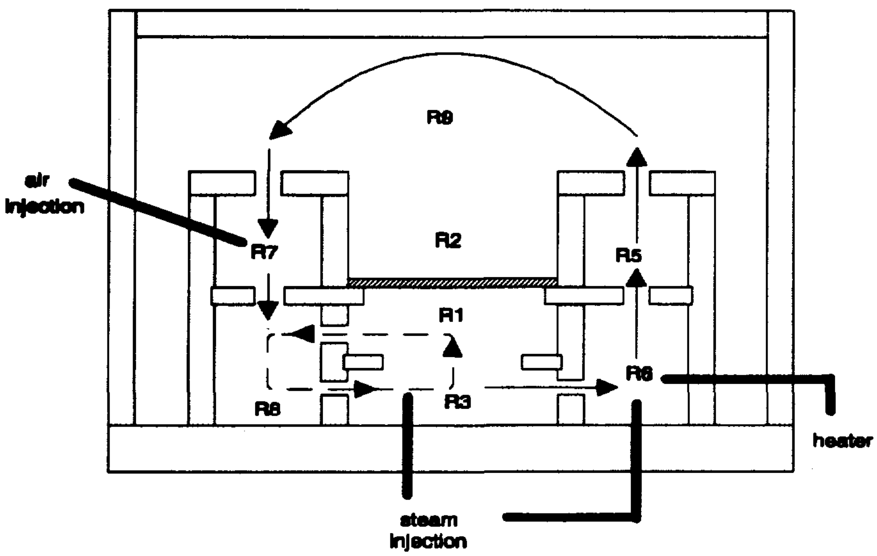

Fig. 2: Natural circulation flows during FIPLOC F2 test.

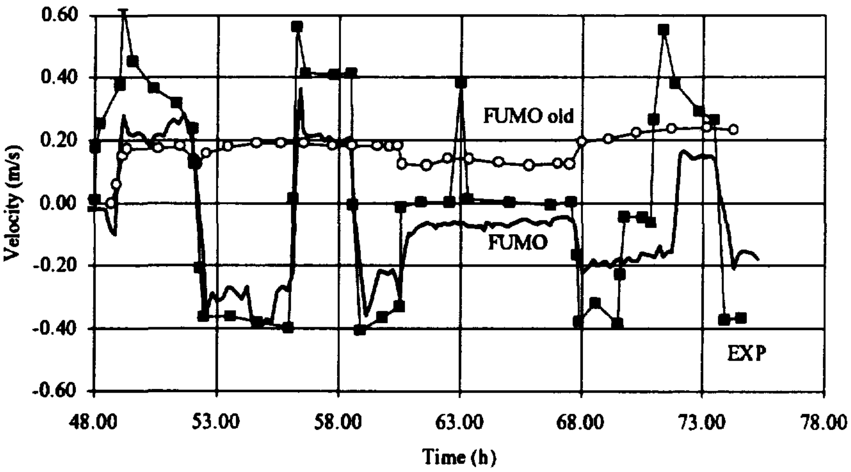

Fig. 3: Natural circulation velocity through junction from R6 to R3.

between, two connecting compartments. The convective loop flow that results from atmosphere density differentials between the right and left branches of this geometry, has been simulated either by heat release or by steam and air injections in different locations. Size and location of the connecting openings have been chosen somewhat asymmetric to effect a preferential flow direction in case of a steam release in the central zone.

The description of natural circulation through both closed loops present in the configuration of the BMC (Fig. 2) for the F2 test, was correctly performed and also the prediction of the overall pressure transient was good [13], because the phenomenology was well described; some limitations were nevertheless highlighted for phenomena influenced by containment local conditions. The results obtained (FUMO), with respect to natural convection flow patterns and changes in the main flow direction, are compared with the experimental data (EXP) and the previous FUMO calculation (FUMO old) without the new models in Fig 3. The distribution of the flow velocity is characterised by the main global convection loop, which changes its direction under the influence of the different injections. The F2 test matrix is formed by four similar sequences, each consisting of four comparable tests at different levels of containment pressure and atmosphere composition. In particular, when the steam was injected into the central compartment, as it has a lower density than the existing air-steam mixture, a buoyancy supported natural convection flow is induced which, due to the asymmetric arrangement of the vent openings of the facility, ascends on the left branch of the main circulation loop, formed by R9, R7, R8, R3, R6 and R5 compartments, and descend on its right branch.

When dry heat is introduced into the right branch, first the additional heat effects a significant decrease in the flow velocity and in the same time the atmosphere temperature within the loop region increases of about 5 K. This affects the temperature of the structures and, after a delay of about 1.5 hours, the convective flow loop develops in counter clockwise direction. During the experiment, the steam is injected into the right branch of the loop to maintain the actual flow direction and the new transition to a central steam injection soon results in a second flow reversion. The importance of a

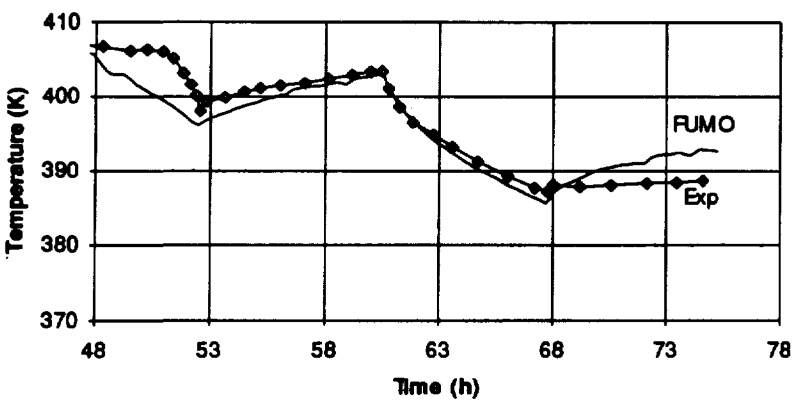

Fig. 4: Temperature trends in the dome.

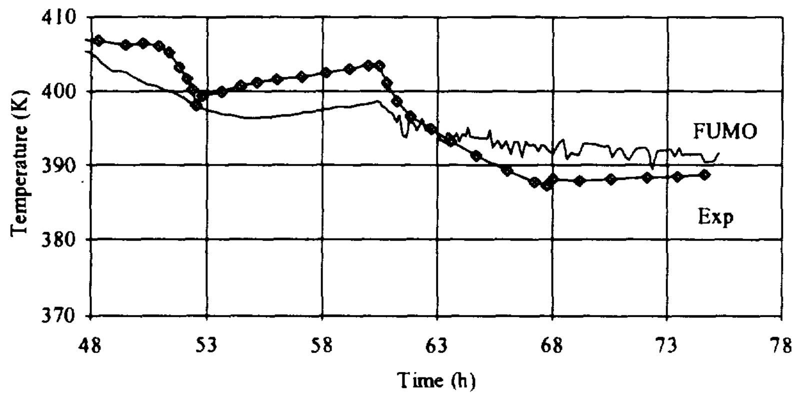

Fig. 5: Temperature trends in the annulus.

realistic simulation of natural circulation phenomena is evident from the analysis of the atmosphere temperature trends. The values of this thermalhydraulic variable calculated with the new models (Figs. 4 and 5), are plotted together with the experimental data. In particular, the natural circulation models are able to describe the local phenomenology and the timing of flow reversals for the main circulation loop, leading to a good prediction of steam and air distributions and consequently of the temperature trends in each one of these nodes. In the following the comparisons for the upper dome zone (Fig. 4) and the annulus compartment (Fig. 5) trends are presented. In particular, the good prediction of the overall natural circulation field allows to describe the increase in the temperature trend, at about 53 hours, due to the warmer steam injected in a lower compartment and carried by buoyancy in the upper dome zone.

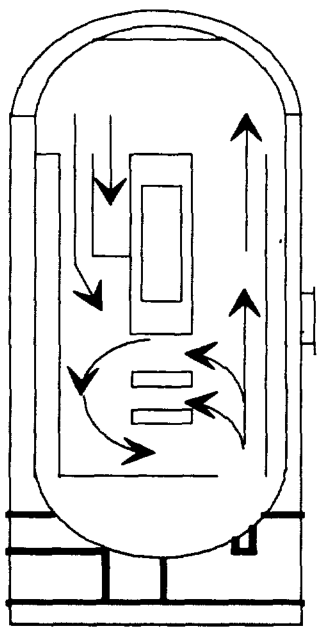

Fig. 6: Natural circulation in E11.2.

3.2 HDR E11.2 Test

The experimental conditions of the HDR E11.2 test simulate, in the first phase of the transient, a SBLOCA within a multi-compartment full pressure containment (Fig. 6). This first phase is followed by a steam and hydrogen mixture injection. The foremost objective of the test was to study the distribution of hydrogen inside a PWR containment. The comparison between the measured and the predicted total pressure trend is good for all the test and is indicative for the quality of the FUMO code to simulate the overall transient.

More insight into the problem of a correct prediction of the long term containment conditions is obtained from the analysis of the

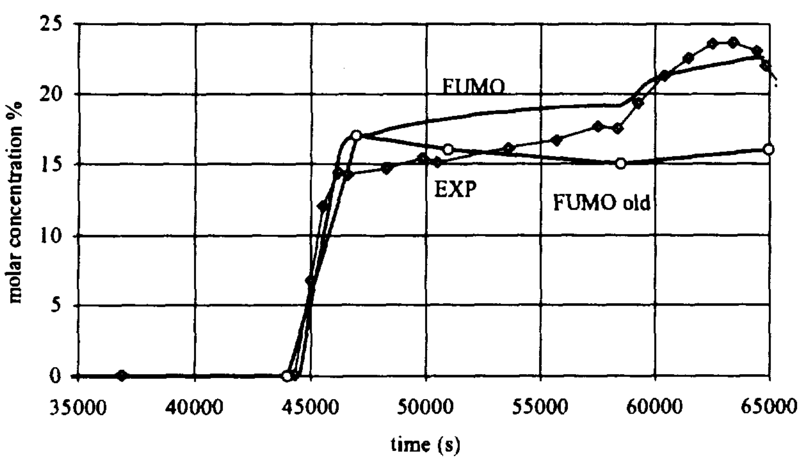

Fig. 7: Hydrogen concentration in the dome region.

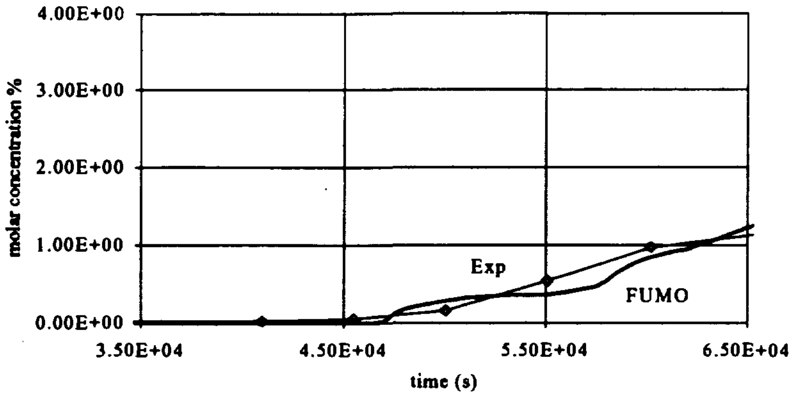

Fig. 8: Hydrogen concentration in the lower containment region.

temperature and hydrogen distributions. The steam leaves the input location near the staircase and, being lighter than the air already present in the containment, rises in the direction of the dome. The air already in the dome is thus displaced over the spiral stair into the lower region of the HDR containment like in a plug flow. A very large amount of the air is displaced in the first 100. minutes, so that at this time 50% of the air in the dome has been expelled. The injected steam strongly enriches volumes above the break location but it does not return down in a substantial quantity. Local gas concentrations of up to 16% by volume were measured near the point of injection and concentrations between 12 and 14% were observed in the dome region (Fig. 7). The gas was only able to penetrate in small amounts into the lower regions through diffusion processes (Fig. 8). The hydrogen gas distribution, which was according to the stratification in the temperature, occurred only in the "hot" regions of the containment. This phenomenon of low natural circulation flow towards the lower regions has been difficulty simulated by the lumped parameter codes participating to the ISP 29 [12]. Immediately after ending the gas injection, an additional steam injection in the lower part was begun, in order to verify whether this would cause changes in the stratification. Up to the end of the additional steam injection in the lower region, the gas was enriched in small steps up to 16% by volume in the dome region; below the point of gas injection, convective or diffusive flow process leads to an enrichment of the atmosphere of the lower nodes of about 1% by volume.

3.3 NUPEC M-7-1 Test

The M-7-1 test on the HMDTF facility of NUPEC (J) is a part of the project "Proving Test on the Reliability for Reactor Containment Vessel" sponsored by MITI [5]. The aim of this project is to evaluate the integrity of a containment under various severe accident scenarios. The tests were carried out in a model of a full pressure containment, 1/4 linear scaled, in simplified conditions in order to employ experimental data for codes validation. The dominant gas flow inside the containment was generated by the gas mixture injected in

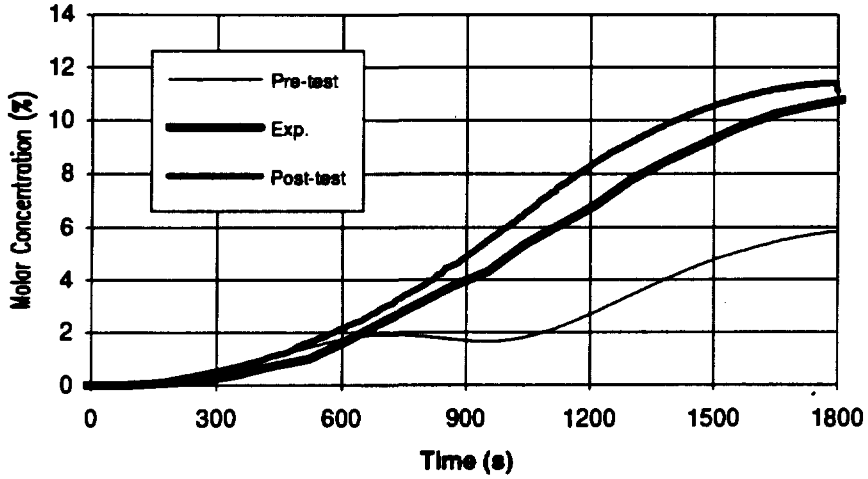

Fig. 9: Hydrogen concentration in the lower containment region.

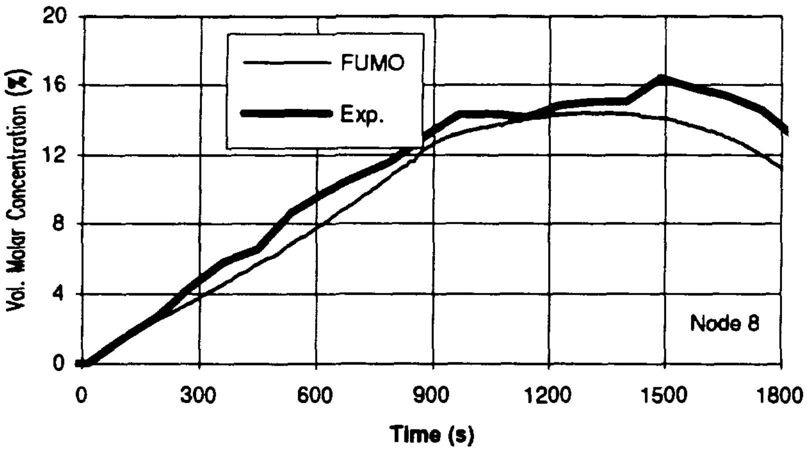

Fig. 10: Hydrogen concentration in the blow-down node.

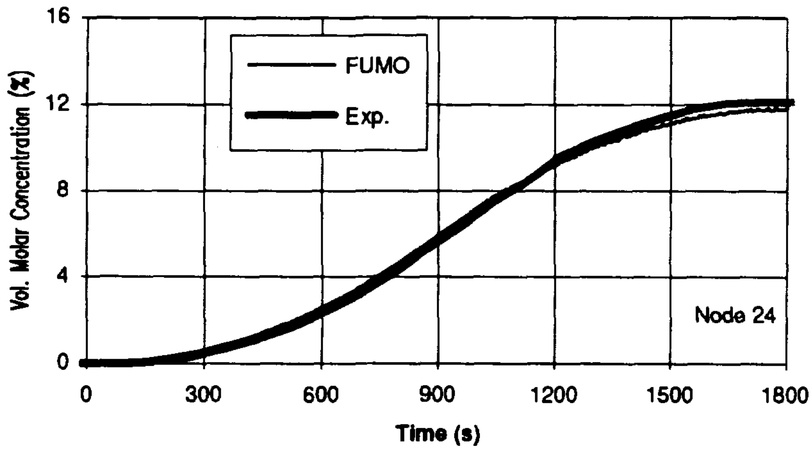

Fig. 11: Hydrogen concentration in the upper containment region.

the SG compartment (Node 8) that flows toward the dome and splits itself into two main upward currents. Downward flows are observed in the other two SGs flow paths. The overall containment behaviour and the macroscopic flow pattern are well predicted by FUMO calculation [14]. In fact the helium distribution (Figs. from 9 to 11), directly related to the natural convection flows, shows a good agreement a with the experimental data. In the compartments affected by the blow-down (Fig. 10), the helium concentration increases until 900. s (maximum value for the helium flow-rate) and then stabilises itself during the second phase of the transient. A different trend of the helium concentration is present in the volumes (Fig. 11) not directly linked to the blow-down compartment but involved in the natural convection cycles: the concentration increase in a more gradual way and the blow-down characteristics influence only the curve shape at about 900. s. Lower gas concentrations are revealed on the contrary in the lower compartments of the containment (Fig. 9) and in the nodes not involved in the natural circulation cycles. Relevant for these nodes is the improvement in the results using the new hydrogen distribution models (Post-test) against the standard FUMO modes (Pre-test).

The NUPEC test is an excellent basis to prove the code ability in modelling the main phenomena for hydrogen distribution and mixing in *well-mixed* conditions, complementary for the *high stratified* conditions present in the previous HDR E11.2 test. The FUMO results agree well with the experimental results and the ability of the code to predict the main containment thermal-hydraulic phenomena has been confirmed. In particular, this is due to improvements in the user capability in understanding and describing the main physical phenomena as:

- Increase of the effective heat transfer coefficient respect to the standard containment correlations due to the internal spray activation (resulting conditions of forced convection against the natural convection field normally present inside a compartment in the long term phase of the accident).

- Good description of the water falling from the upper pools (simulation of the sump formation and its falling down towards the lower compartments).
- Flow tracking of the spray mass from the injection zone towards the atmosphere of the lower nodes (cooling action not limited in the spray node).
- Increase in atmosphere homogenisation of the dead rooms due to the mixing action of the internal spray.

4. CONCLUSIONS

The comparison between the experimental data and the new models predictions carried out in the framework of the post test analysis of the main international exercises in this field showed that an appreciable improvement in the modelling of the natural circulation phenomena exists, phenomena that determine the spatial and temporal distribution of non condensable, burnable gas and aerosol depletion processes in complex containment geometries.
These analyses have been performed in order to validate the various models in their applicability domain. The simulation of these tests allowed to get relevant information relating to the knowledge of the influence of the buoyancy driven flows and ESFs activation in the containment system of the actual and of the advanced generation of LWRs. The errors on the temperature and gas distributions using these models are reduced in a substantially way respect to the experimental data, necessary condition to perform a realistic prediction of a severe accident sequence inside the containment system of a LWR and in particular of the source term. The comparison of the results obtained in the analysis performed used the new models and the previous results obtained with a pure lumped parameter approach shows the validity of the proposed methodology to overcame the limitations due to the lumped parameter approach in obtaining information of local type, as the hydrogen concentration.

ACKNOWLEDGEMENTS

The present work was partially funded by the "Ministero della Univerità e della Ricerca Scientifica e Tecnologica" (MURST) - Fondi 40% & 60%.

ABBREVIATIONS

ANS	American Nuclear Society
BMC	Battelle Model Containment
CEC	Commission of the European Communities
CSNI	Committe on Safety of Nuclear Installation
DCMN	Dipartimento di Costruzioni Meccaniche e Nucleari
ENS	European Nuclear Society
ESF	Emergency Safety Features
IAEA	International Atomic Energy Agency
ISP	International Standard Problem
LWR	Light Water Reactor
MITI	Ministry of International Trade and Industry
MURST	Ministero dell'Univerità e della Ricerca Scientifica e Tecnologica
PHDR	Project HDR
OECD	Organization for Economic Co-operation and Development
PWR	Pressurised Water Reactor
SG	Steam Generator
SBLOCA	Small Break Loss Of Coolant Accident
UIT	Unione Italiana di Termofluodinamica

BIBLIOGRAPHY

[1] A. Manfredini, F. Oriolo, S. Paci *"Modelling of Buoyancy-driven Flows in LWRs Containment System"*, X National Congress UIT on Heat Transfer, Genova (I), June 1992.

[2] H. Karwat, *"Containment Requirements for Future Reactor"*, International Conference on New Trends in Nuclear System Thermohydraulics, Pisa (I), May 1994.

[3] L. Valencia et Alii, *"Hydrogen Distribution Experiment E11.1-E11.5: Preliminary Design Report"*, PHDR Work Report Nr. 10.000/89, March 1989.

[4] K. Fischer, *"CEC Benchmark Exercise on FIPLOC Verification Experiment F2 in BMC. Specifications for Phase II"*, Battelle Institute e.V., Frankfurt (D), September 1989.

[5] K. Takumi, A. Nonaka, T. Hirose, J. Ogata *"Proposal of International Standard Problem Exercise for NUPEC's Hydrogen Mixing and Distribution Test, Test M-4-3 and Test M-7-1"*, June 1992, ISP 35-027

[6] A. Manfredini, M. Mazzini, F. Oriolo, S. Paci, *"FUMO Code Manual Volume 1: System Models"*, DCMN Internal Report RL 533 (92).

[7] A. Manfredini, M. Mazzini, F. Oriolo, S. Paci, *"Validation of the External Spray System for Advanced Nuclear Power Plants" (in Italian)*, VIII National Congress UIT on Heat Transfer, Ancona (I), June 1990.

[8] P. Barbucci, A. Manfredini, G. Mariotti, F. Oriolo, S. Paci, *"Development and Qualification of the FUMO Code for the Containment System Simulation of Advanced LWRs"*, IAEA Meeting on Progress in Development and Design Aspects of Advanced Water Cooled Reactors, Roma (I), September 1991.

[9] P. Barbucci, G. Mariotti, A. Manfredini, F. Oriolo, W. Ambrosini, G. Fruttuoso, *"An Integrated Model for Evaluating the Thermalhydraulic Behaviour of Primary and Containment Systems in Innovative Reactors"*, Advanced Nuclear Plant Conference, Tokyo (J), October 1992.

[10] G. Sandrelli, M. Valisi, G. Fruttuoso, A. Manfredini, F. Oriolo, *"A Model for the Analysis of Containment Behaviour During High-Pressure Melt Ejection Scenarios"*, ANS/ENS 1992 International Conference, Chicago (USA), November 1992.

[11] L. Valencia, L. Wolf, *"Overview of First Results on H_2 Distribution Test at the Large Scale HDR Facility"*, II International Conference on Containment Design and Operation, Toronto (C), October 1990.

[12] L. Valencia, *"HDR Workshop on E11.2 Pre-test Calculations"*, KfK Karlsruhe (FRG), November 1990.

[13] K. Fischer M. Schall, L. Wolf, *"CEC Thermal-hydraulic Benchmark Exercise on FIPLOC Verification Experiment F2 in BMC. Experimental Phases II, III and IV. Results of Comparison"*, Battelle Institute e.V., Frankfurt (FRG), December 1991.

[14] A. Manfredini, F. Oriolo, S. Paci, A. Villotti, *"OECD-CSNI ISP 35: Pre-Test Analysis Using The FUMO Computer Code"*, Pisa University, DCMN Report RL 637 (94)