



Some progresses in system code modelling validation and application including 3D modelling

Dominique Bestion, Philippe Fillion, Raphaël Préa

► To cite this version:

Dominique Bestion, Philippe Fillion, Raphaël Préa. Some progresses in system code modelling validation and application including 3D modelling. ATH'2020 - Advances in Thermal Hydraulics 2020, Oct 2020, Palaiseau, France. pp.661-674. cea-04220353

HAL Id: cea-04220353

<https://cea.hal.science/cea-04220353>

Submitted on 27 Sep 2023

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

SOME PROGRESSES IN SYSTEM CODE MODELLING VALIDATION AND APPLICATION INCLUDING 3D MODELLING

Bestion D,

dominique.bestion@wanadoo.fr

Fillion P. and Pr  a R.

CEA – Commissariat    l’Energie Atomique et aux Energies Alternatives
DEN/DM2S/STMF – Service de Thermohydraulique et de M  canique des Fluides
Universit   Paris-Saclay, F-91191, Gif-sur-Yvette, France
philippe.fillion@cea.fr; raphael.prea@cea.fr;

ABSTRACT

System thermalhydraulic (SYS-TH) codes are extensively used for Design Basis Accidents (DBA) simulations in the frame of safety analyses and licensing studies. The second generation of SYS-TH codes based on the two-fluid model reached a good maturity for industrial use in the 90s. Then several ways were followed for further improvements of the capabilities, including dynamic interfacial area and turbulence modelling, multi-field modelling and 3D modeling in the porous body approach. This paper present the current status of the R&D performed in the last twenty years with the rationale of each research topic, the current achievements and the perspective for future. The challenges related to system scale simulations are first listed including the issues related to the need to address the complex system geometry with a coarse nodalization. Then the limitations of the 2-fluid model are identified to introduce the advanced 1-D models. The porous 3D pressure vessel (PV) models are now used with a finer nodalization which allows a more complete modelling of small scale geometrical effects. On-going progress is significant in this domain, which is illustrated by the application to loss of coolant accidents (LOCAs).

KEYWORDS

System code, modeling, validation, 3D modeling

1. INTRODUCTION

System thermalhydraulic (SYS-TH) codes are extensively used for Design Basis Accidents (DBA) simulations in the frame of safety analyses and licensing studies. The second generation of SYS-TH codes based on the two-fluid model reached a good maturity for industrial use in the 90s. In the late 90s, several reflections at national and international level identified capabilities and limitations of the SYS-TH codes and identified several ways for further improvements of the capabilities, including dynamic interfacial area and turbulence modelling, multi-field modelling and 3D modeling in the porous body approach. This paper present the current status of the R&D performed in the last twenty years with the rationale of each research topic, the current achievements and the perspective for future.

The challenges related to system scale simulations are first listed including the issues related to the need to address the complex system geometry with a coarse nodalization. Then the limitations of the 2-fluid model are identified to introduce the advanced 1-D models. The porous 3D Pressure Vessel models are now used with a finer nodalization which allows a more complete modelling of small scale geometrical effects. On-going progress is significant in this domain, which is illustrated by the application to loss of coolant accidents (LOCAs).

The Figure 1 illustrates the modelling of a 3 loop PWR modelling by the CATHARE code. Depending on the reactor component phenomena may be treated either with 0D models for Pressurizer, Upper Head (UH), Lower Plenum (LP), Upper Plenum (UP), Steam Generator (SG) dome, Inlet and outlet headers of SGs,...1D models for flow in components which have a privileged flow direction such as pipes, core assemblies, annul or 2D and 3D models for components of the Pressure Vessel (PV). The sub-channel modeling of core assemblies was traditionally treated by specific component codes although some modern system code versions have included this capability. Containment thermalhydraulics may be simulated either as a 0D or Multi 0D model using SYS-TH codes or as a full 3D CFD model

SYS-TH codes are designed to be able to simulate with sufficient confidence all involved phenomena in the DBA domain. System thermalhydraulic (SYS-TH) codes considered here have three main attributes, they are *best-estimate code*, *safety codes* and *industrial codes*.

A **Best-Estimate** (BE) code aims at modeling with sufficient reliability all dominant processes that take place in the domain of simulation. For example BE codes intend to predict the Peak Clad Temperature (PCT) during LOCAs with the best possible accuracy when conservative codes just predict an upper limit of PCT. This constituted a very big scientific challenge since they have much higher level requirements:

- All phenomena of all transients must be modelled at a sufficient level of fidelity
- The 2-fluid 6-equation model was selected to describe the very complex physics of two-phase flows, while remaining accessible to experimental validation.
- 0D and 1D models can describe most flows but a few large scale 3D effects required 3D PV models and specific 2D-3D experimental programs.
- Closure laws for mass momentum and energy transfers at the walls and at steam-water interfaces had to be established in a wide range of parameters, for a large variety of complex component geometry, and for all two-phase flow regimes. This may need 150 to 200 closure laws, often identified as “correlations” or “constitutive laws”. A very extensive separate effect data base was necessary to develop and validate each closure law.

Safety analysis codes used for licensing must have some necessary attributes:

- Being developed with clearly specified QA procedures and being extensively verified and validated (V&V process) and being complemented by uncertainty quantification (UQ) tools and methods
- The code scalability or scale-up capability should be addressed
- Being fully documented including precise User’s Guidelines to reduce the User Effect

Industrial codes are used in standardized industrial processes in a routine way. This requires:

- Being be highly robust and perform all simulations without code failure
- Following precise QA procedures with successive versions and updates being tested with impact analysis.
- Keeping CPU time reasonable and allowing many calculations for uncertainty propagation.
- Allowing real time calculations of transients which require operator actions when implemented in plant simulators for operator training

Addressing the geometrical complexity

Reactors have a very complex geometry which influences flow and heat transfers. Processes are relatively easily modelled in regular geometry like a circular pipe, a rod or tube bundle but this becomes more difficult for components with a complex 3D geometry such a UP, LP, SG headers. In components with a privileged flow direction, the 1D modelling is used but local geometrical perturbations (e.g. spacer grids, perforated plates, abrupt area changes, nozzles, Tee connections, bends, perturbations may need specific models. For most complex components, either 0-D or 3D models in a porous body approach are used. 0-D models cannot model precisely all processes but in many cases no sensitive process occurs in the component; if a sensitive process takes place (LP voiding, de-entrainment in UP) 0D models can only be tuned on prototypical experiments. 2D or 3D in porous body were first used for the downcomer refill in a LBLOCA where important azimuthal heterogeneity plays an important role and in the core for describing radial power profile effects.

2.2 Two Phase Phenomena and the Limitations of the Two-fluid Model

Many local processes occur in the various single-phase and two-phase flow regimes including turbulence and interface deformations. The interactions between interfaces and turbulence include shear layers along interfaces and wakes with turbulence production and dissipation, interface instabilities, bubble or drop break up and/or coalescence. Various internal scales for eddies and for bubble and drop sizes cover a wide spectrum from macroscopic to microscopic scales. Each phase may be present in the form of a continuous

field and one or several dispersed fields for bubble or drops. Continuous and dispersed fields may have a very different behavior including countercurrent velocities, which is not described in two-fluid model. This complexity is illustrated in Figure 2 with two of the most complex flow regimes in a horizontal and a vertical pipe.

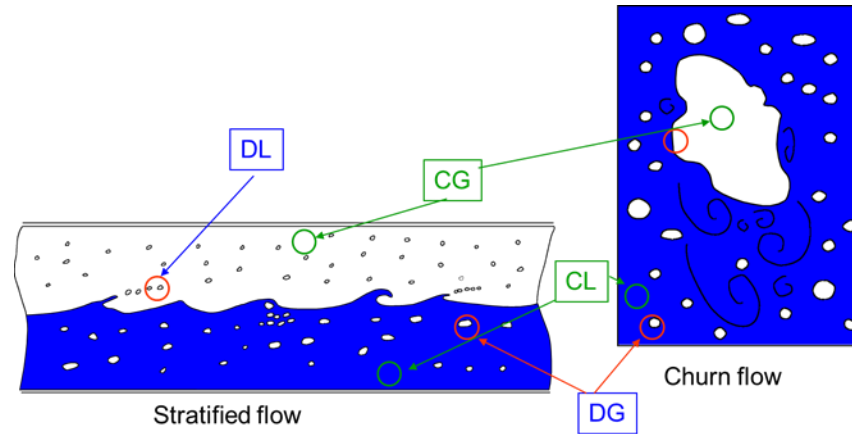


Figure 2: Examples of complex two-phase flow with dispersed gas (DG) dispersed liquid (DL) fields and continuous gas (CG) and continuous liquid (CL) fields

A 1-D two-fluid model needs to model all wall and interfacial transfers which depend on all the scales of the various two-phase flow regimes without having any transport equation for predicting such scales. This is in principle possible when some kind of equilibrium exists between two processes which control a scale resulting in an algebraic expression of the scale. For example a droplet size may correspond to a maximum size allowed by surface tension and inertial forces which gives a maximum Weber number. However this corresponds only to ideal situations which are not so frequent. Considering these limitations of the two-fluid model, advanced modeling have been investigated including dynamic interfacial area and turbulence modelling and multi-field models.

3. ADVANCED 1D MODELING

3.1 Dynamic Interfacial Area and Turbulence Modelling

The transport of interfacial area (TIA) equations may be used for a better modeling of non-established flow regimes, a better prediction of flow regime transitions, a more accurate modeling of interfacial transfers. Dispersed bubbly flow and droplet flows have a mean bubble and drop size which depends on several mechanisms such as break up, coalescence, nucleation, collapse and interfacial mass transfer. Some of these phenomena have relatively long relaxation times so that the “average size” is not an “equilibrium size” which can be expressed based only on the six principal variable of the 2-fluid 6-equation model. TIA intends to take relaxation times into account provided that source and sink terms are modelled and validated. Many flow regime transitions are corresponding to a change of IA: bubbly-to-slug flow or bubbly-to-churn flow result from a coalescence process which decreases IA, bubbly-to-stratified flow transition also results from bubble sedimentation and coalescence of bubbles. Therefore TIA may be used for a prediction of some flow regime transition and may also model intermediate states between two flow regimes. Other flow regime transitions such as the onset of droplet entrainment increases IA dramatically but the governing phenomenon is an interfacial instability. One should distinguish transitions which result from a change on IA from transitions which induce a change of IA. Interfacial transfers of mass, momentum and energy depend on IA (or on drop/bubble size for example). Therefore a dynamic prediction of IA would also improve the prediction of interfacial friction and interfacial heat transfers which control void fraction and slip between phase and also phase temperatures and mass transfers by vaporization and condensation.

Very valuable research work has been done particularly at the Purdue University. Data were produced in adiabatic bubbly-slug-churn flow in vertical pipes and some adiabatic air-water data also in 8X8 rod bundles. Some data also exist in condensing and flashing flow and in boiling flow (see Ishii and Hibiki [1]). Other data were used in the NURESIM and NURISP projects to model bubbly flow [2], boiling bubbly flows [3] including condensing bubbly flow. However after two decades of R&D on TIA data are still missing to cover the whole domain of system code simulations. Today there are almost no data or too few data and no validated IA model for important flow situations such as boiling bubbly-churn flow in core geometry, condensing bubbly-slug-churn flows by wall cooling in Steam Generator (SG) tubes, annular-mist flow in heated core or in cooled SG tubes, droplet flow during core reflooding of a LBLOCA, stratified-mist flow in Upper Plenum and HL during Reflooding of a LBLOCA and bubbly-slug flow and bubbly-to-stratified flow transition in HL, IL and CL as it occurs in LOCAs. Therefore TIA is still very far from being able to improve predictions of current system codes since the most important flow situations in the most important reactor components have no representative IA data for modelling and validation.

Analyzing the bubbly to slug flow transition in the DEDALE air-water flow in a vertical pipe has shown that the turbulence could be strongly affected by the presence of bubbles particularly when the bubble size is rather large. Since turbulence characteristic scales influence source and sink terms of TIA equation, such as break-up and coalescence, a dynamic modelling of IA and of turbulence was developed with additional terms for production in the wake of bubbles and in singularities. It was first adapted for 1D single phase flows in pipes and in rod bundles with spacer grids. The decay of turbulence downstream of a spacer grid could be well predicted by a $k-\varepsilon$ model [4, 5]. Then this $k-\varepsilon$ model was coupled with the TIA equation. Figure 3 shows [5] that a reasonable agreement could be obtained in these non-established flow conditions where turbulence can be multiplied by 4 when the interfacial area is divided by up to a factor 10. One cannot estimate turbulent scales from single phase models and transition from bubbly to slug or to churn flow can actually be better described by a dynamic modelling of both IA turbulence. Relaxation time scales associated to break-up, coalescence, turbulence production and dissipation are relatively long so that “nearly all two-phase flows are non-established flows”.

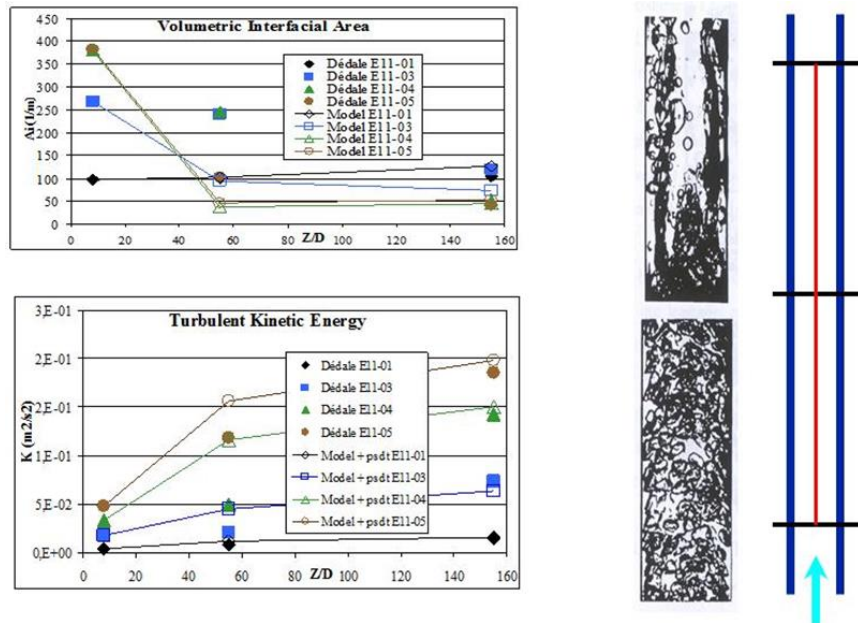


Figure 3: Evolution of turbulent kinetic energy and interfacial area evolution in 4 air-water tests of the DEDALE experiment. Data are measured at $Z=8$ D, 55 D and 155 D.

Significant advances were made in two-phase CFD modelling of bubbly flow [2], and boiling bubbly flow [3]. In such flow, the lift force plays an important role on the void radial distribution since it accumulates small bubbles close to the wall and larger bubbles to the bulk flow with possible influence on the departure from nucleate boiling (DNB) occurrence. This important process requires to model not only an average bubble diameter –as provided by TIA- but a bubble size repartition and several “polydispersion” models were developed and tested. The MUSIG model, which divides the bubbles into a number of groups of different diameters, could capture the different behaviour of small and large bubbles [2]. The method of statistical moments [3, 6] is an alternative to the MUSIG model for taking the poly-dispersion into account. It predicts the bubble diameter distribution by selecting a law of distribution and writing transport equations for some moments. This was tested with log-normal and quadratic laws [6] which are defined by only two statistical moments. Using the bubble size distribution one can express the interfacial heat transfers (e.g. for a subcooled boiling) for each group of bubble instead of only for a Sauter-mean-diameter, which can in principle be more precise. However, attempts to use poly-dispersed models could not yet provide much better results than TIA due to difficulties to model coalescence, break up, nucleation and collapse.

Bubbly-to-stratified flow transition was investigated in the METERO horizontal test facility [7, 8]. Bubbles tend to accumulate at top of the pipe by gravity but turbulence tends to mix them more efficiently for small bubbles than for large bubbles. Here again a good prediction of stratification occurrence would need a prediction of both the bubble size and the turbulence. A coupled TIA and turbulence dynamic modelling, were used where coalescence and break up were modelled together as a relaxation term to an equilibrium IA value $A_{i\infty}$. Models for the relaxation time and for $A_{i\infty}$ were proposed. This showed that it is possible to predict the trends but when approaching the transition to stratified flow a multi-field model with two gas fields would be necessary to take full benefit from TIA in a prediction of bubbly-to-stratified flow transition.

The experience gained in TIA modeling so far was summarized in [9]:

- Almost all flow are non-established flows and an algebraic IA model has many limitations. A TIA equation brings an added value and a poly-dispersion modelling would provide more significant progress but still need a very long effort of modeling and validation and many new data
- Available TIA models cover only a very small fraction of the flow situations of interest for SYS-TH codes. However in a medium term perspective TIA may be used to improve system code predictions provided that the modeling is simplified to use also SET data without IA measurement.

3.2 3-field Modelling

For flow conditions where droplets and liquid film coexist the two fluid model in SYS-TH codes (e.g. TRACE, RELAP5, CATHARE 2) has some limitations and must use a correlation for the entrainment fraction, i.e. the ratio of the droplet flowrate to the total liquid flowrate,. Nevertheless, for example in annular-mist flow, this method cannot take into account the different behavior of the droplets and the liquid film on the wall, which may have very different velocities including counter-current velocities. This is also the case for stratified-mist flow encountered in hot legs, or for drops impinging the structures in the upper plenum. In such cases, a 3-field model in which the liquid phase is split into a continuous field (film) and a dispersed field (droplets), each having its own volume fraction, velocity and temperature is a better approach.

The first application where a 3-field model could improve the simulation is the core dry-out resulting from a progressive loss of liquid film along the fuel rods, which is an operating limit for BWR. Common approaches in SYS-TH codes for the estimation the dry-out location are using look-up tables or empirical CHF correlations. However, such approaches strongly depend on the validity domain of the correlation, or in the case of loop-up table, suffer from a lack of validation in rod-bundle geometries despite of the use of correcting factors to take into account this type of configuration or the effect of the spacer grids. Other approaches are based on the knowledge of mechanisms involved in the liquid film disappearance to quantitatively estimate the dry-out location. These mechanisms are entrainment and deposition of the droplets as well as evaporation of the liquid. Such of models in annular-mist flow have been introduced in

several thermalhydraulic codes, such as FIDAS [10] or COBRA-TF [11]. For this application, and for LOCAs simulations, this type of models have been also developed in SYS-TH codes such as SPACE [12] or CATHARE 3 [13]. In the latter code, a three-field model is being developed, based on an extension of the two-fluid model. Three sets of mass, momentum and energy conservation equations are solved for the, gas or vapor, the continuous liquid, and the dispersed liquid flowing in the form of entrained droplets. Specific correlations for the entrainment, deposition and for the estimation of the droplet size have been developed or introduced during the 2000's, with validation in tubes and rod bundles [14, 15]. More recently, new models of the initial entrained fraction, i.e the entrainment at the onset of annular two-phase flow, which has a significant impact on the prediction of the dry-out, were developed in CATHARE 3 [16, 17].

Another application of 3-field model is the simulation of the PWR Large Break Loss of Coolant Accident (LBLOCA). During the reflooding phase of the LBLOCA, small droplets are produced in the core at the quench front. The steam flow carries out these droplets towards the upper plenum the hot legs and the steam generators, where they vaporize, creating the "steam binding" effect which has a big influence on the reflooding rate. 3-field models can be used for the prediction of the behavior of the droplet in the core. However, the correlations developed for the dry-out prediction involving an annular-mist flow regime usually at high pressure, need to be adapted to conditions that may be encountered in reflooding, namely the inverted annular or inverted slug flow regimes downstream the quench front, and a lower pressure. In CATHARE 3, the entrainment model has been adapted to these conditions and assessed against PERICLES and RBHT reflooding tests around 3 bars, as well as against a LBLOCA reflooding test simulated in BETHSY facility [18] under gravity driven conditions. In such test, oscillatory reflooding is observed at the beginning of the process and a top-down quenching is rewetting the upper part of the core creating a situation with possible falling films and upward drop flow which can be better predicted by the 3-field model.

Deposition and entrainment models are also needed in the upper plenum and the hot legs. For the upper plenum, SEROPS 2 experiments were devoted to the droplets behavior with the influence of a transverse flow in a mockup of a upper plenum, and simple models have been implemented into CATHARE 3 3D module [18] for the de-entrainment process due the droplets hitting onto the thimbles, guide tubes and support columns and the deposition of the droplets due the gravity. In the same way, one-dimensional models have been recently introduced in SYS-TH codes for the entrainment and deposition of the droplets in the hot legs, including both stratified and annular flow regimes. In the SPACE and CATHARE 3 codes, correlations for the entrainment and deposition processes, as well as for the prediction of the droplet size, allowing calculations in horizontal pipes have been developed, taking into account the strong stratification of the droplet field due to the gravity [19,20]. In the latter code, the deposition model takes into account the two mechanisms acting on the droplet deposition in such geometries: gravity and turbulent diffusion. Both codes use dedicated experiments including a large diameter test section, where the vertical stratification of the droplets flow due to the gravity becomes significant. REGARD, featuring a 24 cm diameter, 4 m long horizontal test section, where air and water at near atmospheric pressure and injected at the inlet, is part of these facilities. Tests performed in stratified-mist conditions showed a strong vertical stratification of the droplet field within the pipe. The experimental procedure and the measurement of the droplet field characteristics (velocity and concentration) along the test section allow to study the disequilibrium between the entrainment and deposition processes (see Figure 4), which can also occur in PWR hot legs. To complete these 3-field models in the hot legs, data from dedicated experiments, including the elbow and the inclined part of the hot leg have to be considered (e.g. MHYRESA).

Some progress are also expected by the use of the three-field model in SYS-TH codes. For example, during the safety injection in case of LOCAs, droplets can be generated near the injection of subcooled water from the emergency core cooling system (ECCS) in the cold leg, in presence of a stratified flow. Direct contact condensation occurs onto the jet, on the free surface near the ECCS impact and far from the jet influence. Some correlations have been proposed to model these phenomena, e.g. [21, 22], but, for example in CATHARE 2, the presence of possible droplets is treated with an entrainment fraction correlation, which

does not take into account the different thermal disequilibrium of the droplets and the stratified liquid flowing in the bottom of the pipe. The 3-field model, including separate energy equations for the continuous liquid and the droplets, seems more likely to simulate this part of the primary circuit in these conditions.

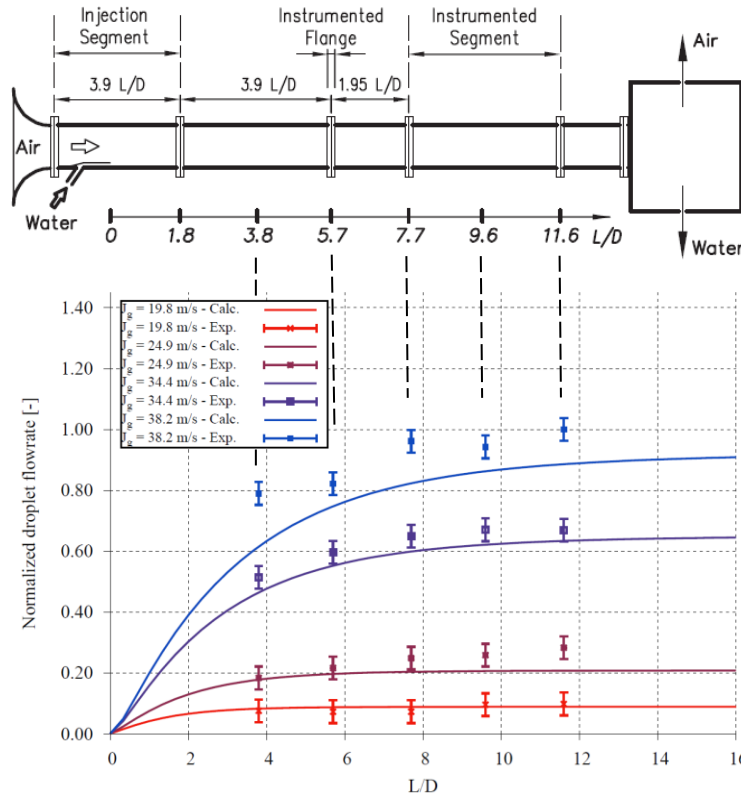


Figure 4: Comparisons of the REGARD experimental droplet flowrates profiles along the test section and the CATHARE 3 simulations using the 3-field model (from [20]).

4. A POROUS 3D MODELLING FOR THE PRESSURE VESSEL

4.1 The Porous 3D Approach

3D PV modelling in SYS-TH codes use the 3D in porous medium approach where equations are time averaged for each phase and space averaged over a volume containing both fluid and solids (see Chandesris et al., [23]). Double (time and space) averaging of the local convection terms of momentum and energy results in macroscopic convection terms, turbulent diffusion terms, and dispersion terms. Depending on the geometry of the component, the porosity, the hydraulic diameter, and on the flow conditions, the relative impact of diffusion and dispersion terms may be negligible or not.

4.2 The Recent 3D PV Modelling

3D PV modelling in SYS-TH codes can now be used with a finer and more flexible nodalization. CATHARE-3 is the new version of the French thermal-hydraulic code for safety analysis of nuclear reactors developed since 2006 as part of the NEPTUNE project launched by the CEA, EDF, FRAMATOME and IRSN in 2001 [24]. A first industrial version of the code validated for the PWRs, V2.1 [25], was delivered in December 2019, in the continuity of the reference multi-concept CATHARE-2 [26]. The 3D module of CATHARE-2 has been validated to model either the whole reactor PV [27] or some subcomponents of the PV like the core [28] or the downcomer [29] for all LOCAs. A global overview of the CATHARE-2 3D

module is available in [30]. CATHARE-2 uses only one cylindrical 3D grid for the whole PV (Figure 4) and it has been proved to be sufficient for simulating any kind of LOCA transient.

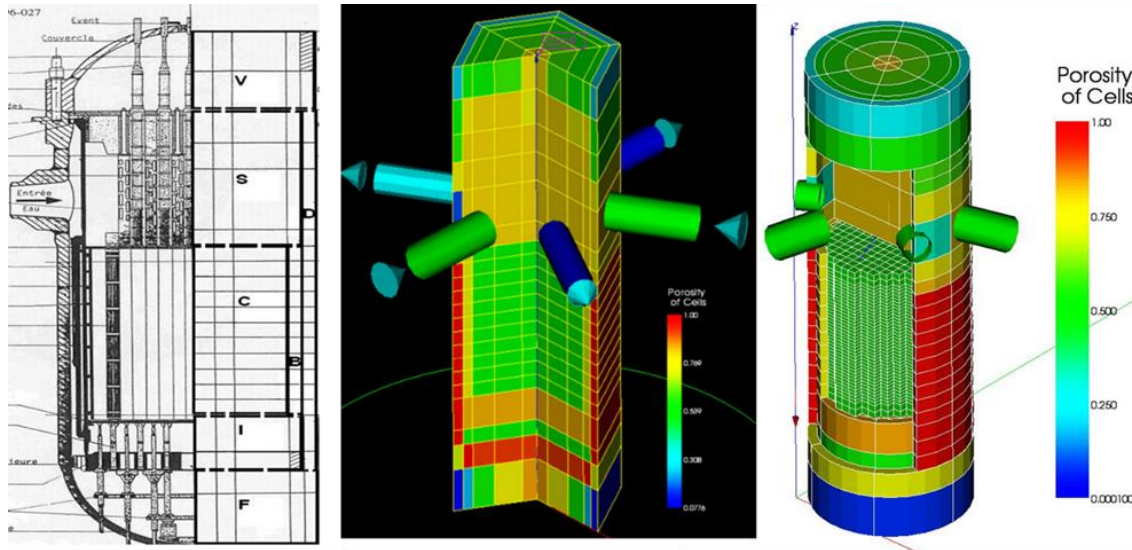


Figure 5: Reactor vessel modeled by CATHARE-2 (middle) and CATHARE-3 (right)

One of the main novelty of CATHARE-3 is to allow a 3D PV modeling using fine and non-conformal structured meshes [13]. However, for a better flexibility of the modeling, for transients with significant transverse flow in the core, for neutronic-coupled simulations, or any other 3D flow investigations, a Cartesian 3D meshing of the core and an adapted meshing of the plena further improves the simulation capabilities. CATHARE-3 can couple different kinds of 3D modules for core, downcomer, plena. A typical set of grids (Figure 5) uses cylindrical 3D grids for all the vessel components except the core, which uses a Cartesian 3D grid (one cell per assembly in a horizontal section and 40 axial meshes).

New optimized resolution methods have been developed in CATHARE-3 to couple 3D modules [13, 31]. The GUITHARE graphical user interface helps to create Cartesian-cylindrical junctions; the standard way to define the junction between two 3D modules is to let the user define the geometry and let the code compute the connectivity tables between meshes. A LOCA computation using these new 3D functionalities of CATHARE-3 is available in [32]. The development of spherical grids for the UH and LP is on-going in order to use the most adapted grid for each component geometry in the PV [13]. In order to use the same type of meshing for validation and for reactor, non-conformal junctions will also be used for the CATHARE-3 validation on the UPTF-10c experiment [30], which investigates the core upper plate CCFL (Counter-Current Flow Limitation).

The main objective of these modelling refinements is to reduce code uncertainty by a better description of all small scale phenomena such as gravity driven or friction driven crossflows in a core due to radial power profile or azimuthal mixing in the annular downcomer which may be encountered in many transients such as LOCAs, main steam line breaks (MSLB) or boron dilution transients. Also diffusion and dispersion terms may have some effects in some situations.

The impact of mesh size in such porous 3D models requires a special attention. Fine mesh calculations may have a higher accuracy than coarse-mesh simulations only if the physical model -i.e. the system of equations with all closure laws – is very good. Fine mesh means the numerical solution is close to the exact solution. It may happen that coarse mesh solutions have a higher accuracy than fine-mesh simulations when there are compensating errors between physics and numerics. This is clearly possible when the physics is not well understood, e.g. in the case of a downcomer refill during a LBLOCA where the flow structure is not

well known and where the physical models were first extrapolated from tube models and then tuned on UPTF scale 1 tests. So several strategies about mesh size effect may be adopted:

- when the physics of the flow is rather well known and presumably well validated, the finer mesh will be the better and an estimation of the numerical error due to mesh non-convergence is recommended
- when the physics of the flow is rather well known and if dispersion terms play a significant role one must keep in mind that these terms may depend on the space filter scale related to the mesh size and the validation of these terms should be done with the same mesh as used in the reactor application.
- when the physics is not well identified (e.g. Downcomer refill in a LBLOCA) , the closure laws can only be tuned on scale one tests using rather coarse nodalization. Since this tuning may imply compensation of the numerical error, which depends on the mesh size, and the physical error, the validation and the reactor application should be made with the same reference meshing for keeping the error compensation.

4.3 Multi-Scale 3D Core Validation for PWR IB-LOCA

An advanced 3D validation is presented focusing on the dry zone of the LSTF core behavior during ROSA2 Test 3 and using a multi-scale approach.

ROSA-2 Test 3 description

The OECD/NEA ROSA-2 Project [33] was performed to resolve key safety issues of PWR Thermal-hydraulics by means of LSTF experiments at the Japan Atomic Energy Agency (JAEA). LSTF is a full-pressure and full-height integral test facility using a full-height core (3.66m of heating length) composed of 1008 simulated fuel rods with 10 MW electrical power. The rods are regrouped in 24 assemblies: 45 heating rods and 4 non-heating rods for 16 square 7×7 bundles and 8 semi-crescent shaped bundles. The radial power profile is devised in three zones (high, mean and low) with radial peaking factor of 1.51 for the high power assemblies, 1.0 for the mean power assemblies and 0.66 for the low power assemblies. Figure 5 presents the radial power distribution. All electrical fuel rod have the same cosine-shaped axial power profile. Test 3 is a Hot-Leg SB-LOCA transient during which two core uncover phases took place, one around 7MPa and one around 4MPa.

For both phases, the swell level drop down from the top of the core to half height with a cladding temperature excursion in the dry zone. Experimental data package [34] provide us the evolution of:

- Some heating rods cladding temperatures (localized at the center of the assemblies),
- Some fluid temperatures (localized at the corner of the assemblies on non-heating rods),
- Fluid temperature at the core inlet, which is at the saturation temperature (or just few °C lower) all during both phases,
- The system's pressure (measured in the upper plenum)
- The collapsed level (obtained with ΔP measurement)

For both phases, the cladding temperatures reach about 800K in the hot assembly at elevation 7 of the core. Using the symmetrical radial power distribution only a few well-instrumented assemblies are modelled for comparison with data.

CATHARE 3D models

The 252 (1008/4) heating rods and all non-heating structures (dummy rods, tie rods, core barrel ...) are modelled. To calculate the core uncover transients, the experimental core power and outlet pressure are imposed. At core inlet a saturated water is injected with a flowrate regulated so that the collapsed level evolution follows the experimental one obtained by ΔP measurement.

Two radial meshes are used: the sub-channel model and an assembly-scale model with one mesh per assembly. Diffusion and dispersion terms are used in the sub-channel model.

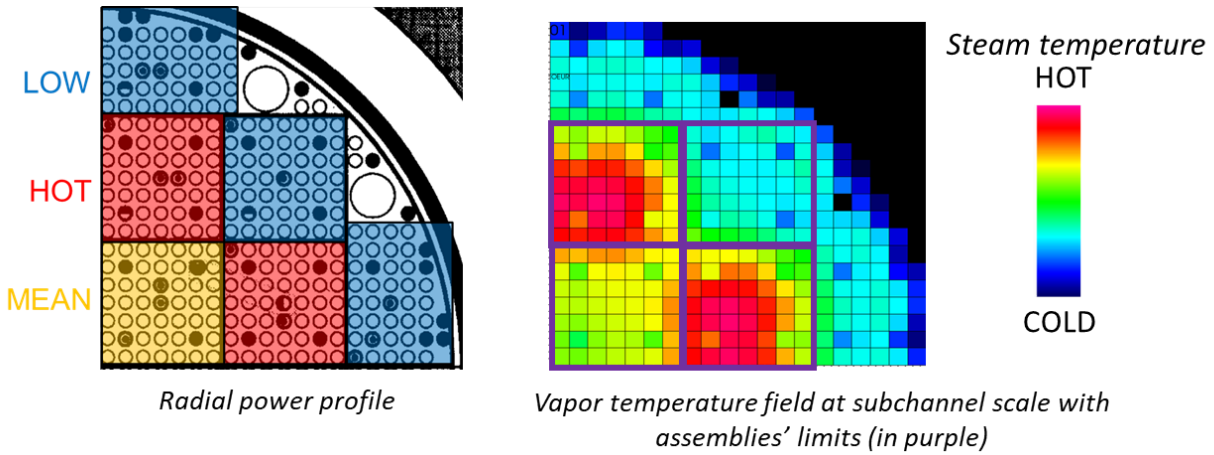


Figure 6: Radial power profile (left) and vapor temperature field (right)

Comparisons between CATHARE calculations and the experiment

As explained in [35, 36], cross-flows from cold to hot assembly at 4 MPa and 7 MPa (“Chimney effect”) are expected and are predicted. They mainly affect some boundary layers between assemblies as shown by the vapor temperature field presented in Figure 6. Comparison between CATHARE and the experimental temperature in the hot assembly are shown in Figure 7. Assembly results are assembly averaged values and sub-channel results are localized at experimental positions. The cladding temperature thermocouple is at the center of the assembly and its experimental evolution is well predicted by both assembly scale and sub-channel scale models. The fluid temperature thermocouple is at the corner of the hot assembly, next to the cold assemblies. Thus, it is localized in the boundary mixing zone between assemblies. It is over-predicted by the assembly scale model and it is better predicted by the sub-channel scale model.

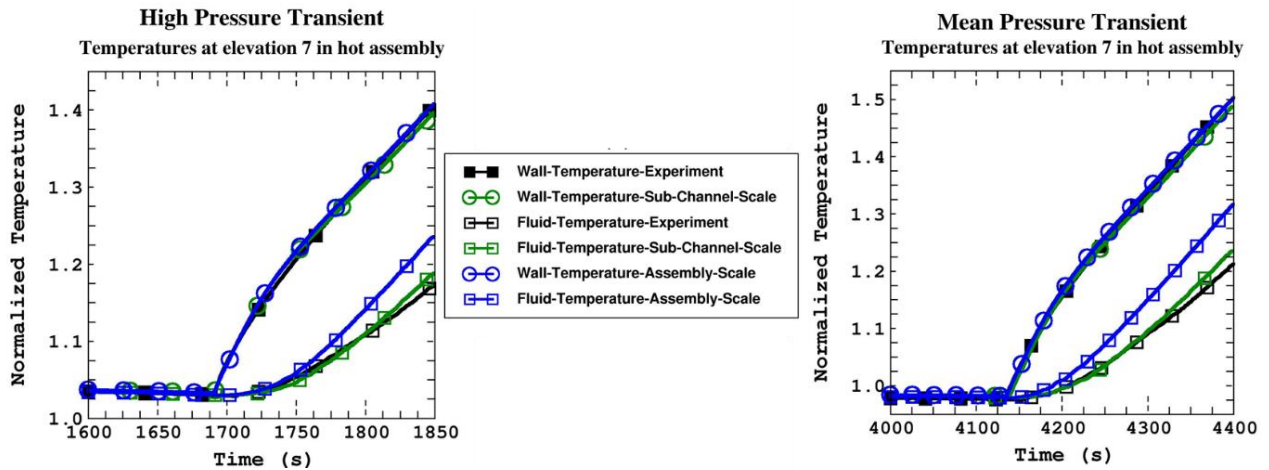


Figure 7: Temperatures in the hot assembly for the two core uncover phases

These results show that sub-channel scale can describe local effects of the cross-flows in the dry zone of a PWR during an IB-LOCA. However, a 3D assembly scale modelling well predicts the high temperature in the center part of the hot assembly.

More details on simulations at sub-channel scale are available in [37] and on assembly scale simulations in [38]. Moreover, singular pressure drops in the radial direction have an impact on the predictions and there is an important uncertainty on the coefficient. The experimental program METERO-V [0] that will

be made at CEA, will provide more information on transversal head loss in rod bundle geometries. The test section is composed by two half PWR-type assemblies with the objective of improving and validating the models involved in 3D simulations of the core behavior during PWR LOCA.

5. CONCLUSIONS

The R&D work performed for improving SYS-TH codes in the past twenty years followed several new ways. An in depth analysis of the basic physics of two-phase flow identified the need to go beyond the limitations of the 2-fluid 6-equation model. Adding transport equations for predicting turbulent length and velocity scales and two-phase scales (bubble and drop sizes, wavelengths,...) may provide more mechanistic mass momentum and energy transfers within the phases and between phases. Progress was obtained in 1D 2-fluid models by developing such advanced modeling using TIA and $k-\epsilon$ models but these models cannot cover all flow regimes in all geometrical situations and did not reach the industrial level. Three field models with two liquid fields is a more mature improvement and may reach the industrial level when sufficient data base is available for validation. It may bring a real benefit for some high quality flows for dry-out prediction and LBLOCAs. The progress of porous 3D modeling of PV was made possible by the increased computer power which allows finer space discretization. A core modelling at assembly scale is now possible and may provide a more complete description of all mixing processes including local geometrical effects. Local mesh refinements up to sub-channel modeling for one or a few assemblies seem a reasonable extension of SYS-TH 3D models for medium term future, providing even more accuracy in the predictions of complex transients such as SBLOCAs, IBLOCAs, SLB, or boron dilution transients.

ACKNOWLEDGMENTS

This paper summarizes the work performed by many contributors to the CATHARE code development and validation, which is a joint project of CEA, EDF, FRAMATOME and IRSN. They are not listed here but our thanks are due to all of them. Also all the contributors to European projects such as EUROFASTNET, ASTAR, ECORA, NURESIM, NURISP and NURESAFE are thanked for their contributions to the progress of SYS-TH codes reported here.

REFERENCES

1. M. Ishii and T. Hibiki, "Modeling and measurement of interfacial area concentration in two-phase flow", *XCFD4NRS*, Grenoble, France, 2008
2. E. Krepper, C. Morel, B. Niceno, P. Ruyer, "CFD modeling of adiabatic bubbly flow", *Multiphase Science and Technology*, **23**(2-4), pp. 129-164 (2011)
3. B. Končar, C. Morel, S. Mimouni, L. Vyskocil, G. Hazi, M. C. Galassi, "CFD modeling of boiling bubbly flow for DNB investigations", *Multiphase Science and Technology*, **23**(2-4), pp. 165-222 (2011)
4. M. Chandesris, G. Serre, "One dimensional averaged ($k-\epsilon$) turbulence model applied to channel, pipe and rod bundle flows", *NURETH-11*, Avignon, France, 2005
5. G. Serre, D. Bestion, "Progress in improving two-fluid model in system code using turbulence and interfacial area equations", *NURETH-11*, Avignon, France, 2005.
6. D. Zaepffel, C. Morel, D. Lhuillier, "A multi-size model for boiling bubbly flows", *Multiphase Science and Technology*, **24**(2), pp. 105-179 (2012)
7. D. Bestion, G. Serre, "On the modelling of two-phase Flow in horizontal legs of a PWR", *Invited paper in Nuclear Engineering and Technology*, **44**(8), pp. 871-888 (2012)
8. M. Marchand, M. Bottin, J.P. Berlandis, G. Serre and E. Hervieu, "Experimental Investigation and Physical Modeling of a Two-Phase Bubbly Flow in Horizontal Pipe", *ICMF-7*, Tampa, FL, USA, 2010.
9. D. Bestion, G. Serre, "Transport of interfacial area in system codes: status and perspectives", *NURETH-15*, Pisa, Italy, 2013

10. S. Sugawara, "Analytical prediction of CHF by FIDAS code based on three-fluid film-dryout model", *Nuclear Science and Technology*, **27**(1), pp. 12-29 (1990)
11. C. Frepoli, L.E. Hochreiter, A.J. Ireland, K. Ivanov, "Modelling of annular film dryout with COBRA-TF", *ICONE-9*, Lyon, France, 2001
12. S. J. Ha, C. E. Park, K. D. Kim, C. H. Ban, "Development of the SPACE code for nuclear power plants", *Nuclear Engineering and Technology*, **43**(1), pp. 45-62 (2011)
13. P. Emonot, A. Souyri, J.L. Gandrille, F. Barré, "CATHARE-3: A new system code for thermal-hydraulics in the context of the NEPTUNE project", *Nuclear Engineering and Design*, **241**(11), pp. 4476-4481 (2011)
14. S. Jayanti, M. Valette, "Prediction of dryout and post-dryout heat transfer at high pressures using a one-dimensional three-fluid model", *Heat and Mass Transfer*, **47**(22), pp. 4895-4910 (2004)
15. S. Jayanti, M. Valette, "Calculation of dry out and post-dry out heat transfer in rod bundles using a three field model", *Heat and Mass Transfer*, **48**(9), pp.1825-1839 (2005)
16. M. Spirzewski, P. Fillion, M. Valette, "Prediction of annular flows in vertical pipes with new correlations for the CATHARE-3 three-field model", *NURETH-17*, Xi'an, China, 2017
17. M. Spirzewski, H. Anglart, "An improved phenomenological model of annular two-phase flow with high accuracy dryout prediction capability", *Nuclear Engineering and Design*, **331**, pp. 176-185 (2018)
18. M. Valette, J. Pouvreau, D. Bestion and P. Emonot, "Revisiting large break LOCA with the CATHARE-3 three-field model", *Nuclear Engineering and Design*, **241**(11), pp. 4487-4496 (2011)
19. J. K. Schimpf, K.D. Kim, J. Heo, B.J. Kim, "Development of droplet entrainment and deposition models for horizontal flow, *Nuclear Engineering and Technology*, **50**(3), pp. 379-388 (2018)
20. P. Fillion, "Development of the CATHARE 3 three-field model for simulations in large diameter horizontal pipes", *NURETH-18*, Portland, OR, USA, 2019
21. A. Janicot, D. Bestion, "Condensation modeling for ECC injection", *Nuclear Engineering and Design*, **145**(1-2), pp. 37-45 (1993)
22. J. Liao, C. Frepoli, K. Ohkawa, "Cold leg condensation model for analyzing loss-of-coolant accident in PWR", *Nuclear Engineering and Design*, **285**, pp. 171-187 (2015)
23. M. Chandesris, M. Mazoyer, G. Serre and M. Valette, "Rod bundle thermalhydraulics mixing phenomena: 3D analysis with CATHARE 3 of various experiments", *NURETH-15*, Pisa, Italy, May 2013
24. A. Guelfi and al., "NEPTUNE: A New Software Platform for Advanced Nuclear Thermal Hydraulics", *Nuclear Science and Engineering*, **156**(3), pp. 281-324 (2007)
25. R. Pr  a, P. Fillion, L. Matteo, G. Mauger, A. Mekkas, "CATHARE-3 V2.1: The new industrial version of the CATHARE code", *ATH'2020*, Paris-Saclay, France, 2020
26. G. Geffraye and al., "CATHARE 2 V2.5_2: A single version for various applications", *Nuclear Engineering and Design*, **241**(11), pp. 4456-4463 (2011)
27. I. Dor, C. Morel, P. Bazin, P. Boudier, "Assessment of the CATHARE 3D module for LBLOCA simulation", *NURETH-11*, Avignon, France, 2005
28. C. Morel and D. Bestion, "Validation of the CATHARE code against PERICLES 2D BOIL-UP Tests", *NURETH-9*, San Francisco, CA, USA, 1999
29. J. Dufr  che and I. Dor, "Assessment of the CATHARE 3D module with UPTF downcomer refill tests", *40th European Two-Phase Flow Group Meeting*, Norway, 2003.
30. I. Dor, M. Chandesris, P. Germain, A. Ruby, "CATHARE 3D Module from CATHARE-2 V2.5_3 to CATHARE-3 V1.0", *NURETH-15*, Pisa, Italy, 2013
31. A. Ruby and al., "CATHARE-3: First computations of a 3-inch break Loss-Of-Coolant Accident using a modular multi-3D modeling of a PWR vessel", *NURETH-15*, Pisa, Italy, 2013.

32. R. Pr  a, V. Figerou, A. Mekkas, A. Ruby, "CATHARE-3: a first computation of a 3-inch break loss-of-coolant accident using both Cartesian and cylindrical 3D meshes modelling of a PWR vessel", *NURETH-17*, Xi'an, China, 2017.
33. OECD/NEA, "Final Integration Report of the Rig-of-Safety Assessment (ROSA-2) Project – 2009-2012", NEA/CSNI/R(2016)10
34. Japan Atomic Energy Agency (JAEA), "Final Data Report of ROSA-2/LSTF Test 3", Private Communication, 2012
35. D. Bestion, P. Fillion, P. Gaillard, M. Valette, "3D core thermalhydraulics phenomena in PWR SB-LOCAs and IB-LOCAs", *NURETH-17*, Xian, China, 2017
36. C. Rabe and R. Pr  a, "Chimney and diverging effects in core PWR: analysis and experimental characterization for predictive behavior during loss of coolant accident", *NURETH-18*, Portland, OR, USA, 2019
37. R. Pr  a and A. Mekkas, "Rod bundle thermalhydraulics mixing phenomena: 3D analysis with CATHARE-3 of ROSA-2/LSTF experiment", *Proceedings of the "Turbulence and Interaction" Conference (TI2018)*, Les Trois Ilets, France, 2018
38. R. Pr  a, "Validation of CATHARE 3D module on LSTF Core", *NURETH-18*, Portland, OR, USA, 2019
- D. Bestion, P. Fillion, R. Pr  a, G. Bernard-Michel, "Improved PWR LOCA Simulations Through Refined Core 3D Simulations – An Advanced 3D Modelling and the Associated METERO Validation Program", *NUTHOS-12*, Qingdao, China, 2018