Series of J NED Special Issues to streamline the future of Nuclear Fission Technology (NFT)

From Past to Future: The Role of Computational Fluid Dynamics in advancing Nuclear Safety in Spain and Portugal

Y. Rivera[1], A. Escrivá[1], C. Berna[1], E. Vela[2], J.M. Martín-Valdepeñas[2], G. Jiménez[3], C. Vázquez-Rodríguez[3,4], K. Fernández-Cosials[3,5], F.J.S. Velasco[6], R.A. Otón[7], I. Esteban[8], R. Matías[8], D. Lázaro[9], M. Lázaro[9], E. Morgado[2], J. Benavides[10]

[1] Universitat Politécnica de Valencia, Instituto de Ingeniería Energética, 46021 Valencia, Spain.
[2] Consejo de Seguridad Nuclear, 28040, Madrid, Spain
[3] Universidad Politécnica de Madrid (UPM), 28040, Madrid, Spain
[4] Forschungszentrum Jülich, 52428, Jülich, Germany
[5] Texas A&M University, 77843, College Station, USA
[6] Universidad Politécnica de Cartagena, Departamento de Ingeniería Térmica y Fluidos, 30202, Cartagena, Spain
[7] Centro Universitario de la Defensa, Departamento de Ingeniería y Técnicas Aplicadas, 30720, San Javier, Spain
[8] ENUSA Industrias Avanzadas, SA, 28040, Madrid, Spain
[9] Universidad de Cantabria, GIDAI - Seguridad Contra Incendios- Investigación y Tecnología, 39005, Santander, Spain
[10] Empresarios Agrupados, 28015, Madrid, Spain

Correspondence should be addressed to Yago Rivera; yaridu@upv.edu.es

Abstract

This document explores the role of CFD in nuclear safety studies, with a particular focus on advancements in Spain and Portugal. The field of nuclear safety has increasingly included CFD models to address complex safety-critical phenomena, given their ability to capture three-dimensional behavior with high resolution. While traditional one-dimensional codes remain very valuable, they often rely on conservative approximations, making 3D CFD approaches indispensable for accurate predictions across various nuclear scenarios. In the context of Spain and Portugal, this document shows the contribution to CFD applications for nuclear safety research. The topics covered include code development and validation, combustion studies, simulations of nuclear spent casks, fuel assembly analyses, and investigations into containment safety issues. These advances have mainly contributed to improving the predicting capabilities of CFD models for applications in various safety-critical domains. In addition to detailing the achievements and contributions in the field of CFD in Spain and Portugal, readers will find a comprehensive overview of the current challenges and future perspectives of CFD within the domain of nuclear fission technology.

Keywords: Computational Fluid Dynamics, Nuclear Safety, CFD Validation, Combustion, Nuclear Spent Casks, Fuel Assemblies, Containment, Nuclear Fission Technology.

40 Acronyms

ATF	Artificially Thickening Flame
AUSM	Advection Upstream Splitting Method
BPG	Best Practice Guidelines
BSC	Barcelona Supercomputer Center
BWR	Boiling Water Reactor
CEA	Commissariat à l'énergie atomique et aux énergies alternatives
CFD	Computational Fluid Dynamics
CNRS	Centre National de la Recherche Scientifique
CSN	Consejo de Seguridad Nuclear de España
CSNI	Committee on the Safety of Nuclear Installations
DCS	Dry Cask Simulator
DNS	Direct Numerical Simulation
ENUSA	Empresa Nacional de Uranio, S.A.
EPRI	Electric Power Research Institute
ETSON	European Technical Safety Organisation Network
EURATOM	European Atomic Energy Community
FADA	Fuel Assembly Drop Accident
FDS	Fire Dynamic Simulation
FEA	Finite Element Analysis
FWP	Framework Programs
FZK	Forschungszentrum Karlsruhe
GOTHIC	Generation Of Thermal-Hydraulic Information for Containments
GTRF	Grid-To-Rod-Fretting
HI-STORM	Holtec International Storage Module
iLES	implicit LES
IRSN	Institut de Radioprotection et de Sûreté Nucléaire
ISFSI	Spanish Independent Spent Fuel Storage
KAERI	Korea Atomic Energy Research Institute
KWU	Kraftwerk Union
LES	Large Eddy Simulation
MTES	Modelling of Thermal and Energy Systems Research Group
NEA	Nuclear Energy Agency
NFPA	National Fire Protection Association
NIST	National Institute of Standards and Technology
NPP	Nuclear Power Plants
NRC	Nuclear Regulatory Commission
NS	Nuclear Safety
NUGENIA	Nuclear Generation II and III Association
OECD	Organization for Economic Cooperation and Development
OKBM	Опытное конструкторское бюро машиностроения (Experimental Design Office)
PAR	Passive Autocatalytic Recombiners
PRA	Probabilistic Risk Analysis
PRISME	Propagation d'un incendie pour des scenarios multi-locaux elementaires
PSI	Paul Scherrer Institute
PSR	Partially Stirred Reactor
PWR	Pressurized Water Reactor
RANS	Reynolds-Averaged Navier-Stokes
SAS	Scale Adaptive Simulation
SMR	Small Modular Reactor
UC	University of Cantabria
UPCT	Universidad Politécnica de Cartagena
UPM	Universidad Politécnica de Madrid
UPV	Universitat Politècnica de València
UQ	Uncertainty Quantification
V&V	Verification and Validation
WGAMA	Working Group on Analysis and Management of Accidents

1. Preface

CFD codes are a powerful tool for simulating fluid behavior and analyzing its interactions with the boundaries. Therefore, for almost half a century, they have been used as a complement to system codes to evaluate the safety of nuclear systems where thermal-hydraulic analyses are relevant. Their greatest advantage in this field is the ability to characterize the phenomenology on a much smaller scale than system codes, which allows for a much more in-depth analysis in many scenarios. However, there is an open debate regarding the credibility of these codes in safety studies, and their use in this sense is still limited to certain applications. CFD codes increasingly use first principles models capable of reproducing the physics of thermal-hydraulic behavior in three dimensions. The growth they have experienced over the last few decades has been significant, in part, thanks to the increase in computing and parallelization capabilities, bringing their use closer to companies, universities, and research centers. However, the requirements for performing accurate high-resolution simulations are still large, especially in multiphase scenarios with mass and energy transfer. CFD represents a truly promising option, but a great effort will be required by the scientific community, industry, and regulatory bodies to fully extend its use in the nuclear field.

In the field of NS, CFD codes represent a major advancement mainly due to the three-dimensional behavior of many of the processes that occur during events of interest in this area (IAEA-TECDOC-1370, 2003). For example, fluid mixing, thermal stratification or more complex phenomena such as liquid-gas or liquid-solid interactions represent mechanisms that can be reproduced using CFD. Additionally, these codes can numerically simulate flows with complex geometries without the limitations inherent to system codes or empirical correlations. In fact, the equations governing fluid motion are independent of the reference scales and thus are theoretically representative for all scales. However, CFD codes still rely in many cases on closure equations that implicitly include dependencies on flow characteristics or length scales. As a result of these advantages, the use of CFD to assess certain phenomena related to safety such as those related to reactor cooling systems, natural circulation, containment or spent fuel cooling has increased in recent years (OECD/NEA, Smith et al., 2014). In these and other scenarios, pseudo 1-D calculations could lead to erroneous conclusions due to the simplifications they implicitly assume. CFD codes are also a valuable tool that provides information throughout the entire domain, making them highly effective for complementing experimental measurements. Currently, it is common to observe publications on experimental facilities where certain parameters are preliminary foreseen based on validated CFD models (Rivera et al., 2022).

Safety studies require highly reliable CFD approaches to assess their effectiveness. The validation process plays a crucial role in evaluating reliability by comparing CFD results with experimental data, identifying any discrepancies and verifying their accuracy (IAEA NR-T-1.20, 2022). Additionally, the validation process entails comparing predicted and measured data from separate and integral effect tests to understand any deviations. CFD's Verification and Validation process, along with Uncertainty Quantification (commonly known as V&V+UQ), involves selecting models for physical phenomena and developing numerical models for the physical domain and fluid properties. While the scalability of governing equations is not an issue, closure models and the validity range of models can affect the accuracy and uncertainty of results. Additionally, the validity range of the models used may extend beyond the explored domain of the database, which can impact the justification of certain fittings or parameter choices, as well as introduce uncertainty into the results. To make matters worse, adapting UQ methods to CFD remains limited regardless of the efforts done for

88   system-scale analyses. The primary sources of uncertainty are diverse and more complex to
89   analyze than in simplified modeling approaches. Turbulence models, proper representation of
90   boundary conditions, fluid properties, and numerical methods used to solve the governing
91   equations are major sources of uncertainty that affect the confidence of CFD results. The
92   difficulties in performing a holistic uncertainty quantification question the application of CFD
93   simulations to address NS predictions, especially from a regulatory perspective. Significant
94   efforts have been made by the scientific community to develop BPG in order to establish a
95   common and well-thought-out methodology for the use of CFD in NS (OECD/NEA, Mahaffy
96   et al., 2014; NUREG 1934, 2012). This way, errors can be minimized by establishing rigorous
97   steps to follow to obtain better reliability in results.

98   The inherent complex phenomenology related to NS analyses has motivated the development
99   of nuclear-field codes with 3D capabilities, such as GOTHIC or FDS. Some of these codes are
100   less demanding of computational resources while accommodating the unique characteristics of
101   their respective fields of application. In particular, GOTHIC is a thermal-hydraulic code based
102   on a two-phase, multi-fluid formulation, which solves separate conservation equations for
103   mass, momentum, and energy for three fields: vapor, continuous liquid, and droplets (Harvill
104   et al., 2022). GOTHIC is mostly used for containment safety analysis of light water reactors,
105   but it has also been used for calculating the thermal behavior of waste fuel casks, and it has
106   been extended to tackle liquid-metals phenomena (Lane et al., 2020). In the framework of 3D
107   application-oriented analysis, GOTHIC is seen as a hybrid between CFD and System codes
108   because it does not have the capability of resolving the boundary layer explicitly. The code
109   includes a set of built-in correlations—performing a similar role to wall functions in
110   commercial CFD— cell faces and cell volume porosity factors to model friction, heat, and
111   mass transfer close to the walls. It significantly decreases the computational cost of the
112   simulations, a cost-benefit balance that enables the use of 3D codes in the large (more than
113   60.000 m^3) and demanding computational domain to obtain variables of pressure, temperature,
114   or hydrogen concentration.

115   In the same way, as mentioned above, the development of CFD codes has also enabled the
116   integration of multiple physics in problem-solving, allowing the incorporation of phenomena
117   of interest that affect fluid performance. For instance, it is possible to obtain information about
118   the evolution of different species subjected to chemical reactions or predict fire propagation.
119   Since the 1970s, a large amount of computational fire models have appeared to predict fire
120   behavior and try to increase the safety of nuclear power plants. A major milestone in nuclear
121   safety was the publication of the standards NFPA 805 Performance-Based Standard for Fire
122   Protection for Light Water Reactor Electric Generating Plants in 2001. This standard allows
123   the application of computational fire models to support performance-based fire protection and
124   risk analyses, but they must be verified and validated for the specific phenomena analyzed.
125   Although there are three main types of fire models, algebraic models, zone models and field
126   models (CFD based), the last type is more detailed and accurate, and they are applicable to a
127   large number of fire scenarios in nuclear power plants. Inside field models (CFD), FDS is the
128   most widely employed and validated (K. McGrattan et al., 2023) (NUREG-1824, 2016). This
129   model has been developed by the NIST, and it is a CFD model of fire-driven fluid flow. FDS
130   solves numerically a form of the Navier-Stokes equations, appropriate for low-speed,
131   thermally-driven flows emphasizing smoke and heat transport from fires. This model is useful
132   to analyse all fire phenomena (combustion, fire spread, species generated, smoke, etc.) and the
133   interaction with the fire protection systems (sprinklers, water mist, exhausts, compartmentation
134   systems, etc.). Nevertheless, its application to certain specific phenomena related with fire
135   safety is still to be validated, showing different discrepancies in the results of international

136 benchmarks that remark the necessity to continue working in the validation and the definition
137 of guidelines to model those phenomena (Plumecocq et al., 2023).

138 This paper outlines Spain and Portugal's contributions to CFD for Nuclear Safety. Section 2
139 presents a synthesis of the different projects conducted, categorizing the research topics into
140 code development, combustion, nuclear-spent casks, fuel assemblies, containment, and other
141 areas. Following the same structure, Section 3 describes the main outcomes for each research
142 domain. In Section 4, the authors discuss current and upcoming challenges and planned
143 activities for their respective organizations. Finally, the conclusions offer a holistic summary
144 of the paper's main contributions.

2. Research synthesis

2.1. Code Development

147 Since 2002 the **UPCT**, has worked in the development of numerical CFD codes and its
148 validation for nuclear safety applications through different international research
149 collaborations with European institutions. During 2002 and 2003, the Modelling of Thermal
150 and Energy Systems Research Group (MTES) of UPCT worked in collaboration with
151 Commissariat à l'énergie atomique et aux énergies alternatives (CEA) at Saclay, in the
152 development and validation of AUSM+ numerical schemes and other Riemann Solvers under
153 unstructured grids and on the bases of a one-pressure two-fluid model denominated CATHARE
154 Model, in the context of the European Project: Advanced 3D two-phase flow simulation tool
155 for application to reactor safety, from the 5th EURATOM Framework Programme (FP-5)
156 (Paillère and García, 20031-2). This collaboration was extended up to 2006 and included not
157 only CEA (France) but also FZK (Germany) and other institutions (Staedtke et al., 2005)
158 (García and Paillère, 2007). Among the specific developments, members of the group have
159 extended existing Riemann Solvers such as AUSM-type to compressible two-phase mixtures
160 and to low Mach number flows. They successfully applied them to transient two-phase (six-
161 equation models) with non-hyperbolic systems that raise in nuclear safety and other fluid
162 dynamic problems. In this case, the system provides complex eigenvalues in some parts of the
163 solution yielding to ill-posed problems in finite volume formulation for multi-dimensional
164 applications. Both solid and gas phases where resolved using a Eulerian approach. Examples
165 of these problems are 3D shock tube tests, which are important for aerosol mobilization and
166 dispersion, 2D boiling channel test (key in BWRs) or the problem of predicting the dispersion
167 of two-phase (liquid-gas) flow in a convergent-divergent nozzle. This last problem is key as it
168 includes strong mechanical and thermal non-equilibrium conditions typical of the fast
169 depressurization of nuclear reactor systems during loss-of-coolant accident conditions (so
170 called "blowdown" events). Furthermore, the presence of combustible dust particles in a
171 hydrogen detonation sequence presents several challenges in physical modeling and numerical
172 resolution. In particular, the source terms coupling the gas and solid phases generate a strong
173 stiffness of the system of equations, so a splitting method that uses AUSM+ and Rusanov
174 schemes and the linearization of source terms was developed (García-Cascales, 2014) and
175 validated in order to achieve a numerical solution to the problem. The validation was performed
176 through experimental data from a spherical bomb filled in with $H2$–$O2$–$N2$ mixtures in the
177 absence and presence of solid particles (Velasco et al., 2016 and García-Cascales et al., 2015),
178 giving very promising results, despite the simplicity of the combustion model used at that time.
179 These models can be used for the analysis of accident sequences with $H2$ combustion, both in
180 fusion and fission reactors.

2.2. Combustion and Fire Safety

Between 2008 and 2014, the **UPCT** worked through specific research contracts financed by IRSN, in the development and validation of numerical codes for the 3D modelling of particle mobilization, combustion of two-phase mixtures, and detonation of hydrogen-air mixtures with CAST3M CFD code (García et al., 2014). In the last eight years, the research work in CFD modelling has focused on the collaboration with IRSN for the development of new models for turbulent hydrogen combustion to predict flame acceleration and deflagration to detonation transition with progress variable models and thickening flame models. These codes were developed to be used in OpenFOAM and can be combined with other available multi-physics libraries. In this context, UPCT has represented Spain in the ETSON-CFD-Benchmark organized by the European Technical Safety Organisation Network within the SAM-HY-CO net and sponsored by NUGENIA and MITHYGENE programs. This benchmark aims to identify the current level of accuracy of the available CFD tools in the area of hydrogen combustion simulation, under conditions that are representative of safety considerations in a Nuclear Power Plant, and testing CFD models' capacity to predict flame acceleration. Besides, additional efforts were focused on the development and validation of CFD models able to predict hydrogen flame dynamics of lean mixtures under isotropic turbulent fields in collaboration with CNRS at Orleans and IRSN at Fontenay-aux-Roses where different experiments were performed.

The **UC** has been actively collaborating with the CSN in the analysis, validation and application of CFD fire codes to enhance safety in NPP. Since 2014 both institutions have been working on the analysis in depth of the background of main fire models applied to nuclear power plants, in the development of a guide with the steps and input parameters necessaries to perform a simulation, in the creation of a software to easily create fire scenarios in nuclear power plants, in the development of simplified models to afford some specific fire related phenomena, as cable fire propagation in NPP (Lázaro D. , Lázaro, Jiménez, & Alvear, New approach to cable tray fire modelling through a FLASH-CAT modification, 22-27 May 2022), in the definition of new methodologies to obtain the input parameters to improve FDS model prediction (Lázaro, Alonso, Lázaro, Jiménez, & Alvear, Experimental and analytical study of the influence of the incident heat flux in cables heat release, 2023) (Lázaro D. , Lázaro, Alvear, & Jiménez, Methodology to estimate the fds chemical reaction equation in fire scenarios, 2022), in the applicability of FDS to model real NPP scenarios (Lázaro D. , Lázaro, Peco, & Alvear, 2018), in the validation of CFD codes through the implementation of large scale experimental tests, etc. A summary of the different projects in which both entities have participated is shown below.

- Collaborative project with the CSN entitled "Simulación de Incendios en Centrales Nucleares" ("Simulation of Nuclear Power Plant Fires") from 2014 to 2018. The main objectives of this project were the definition of methodologies to obtain the input parameters to define material reactions in FDS model, the analysis of the applicability of FDS to model complex fire sources, like cabinets or cable trays, and the comparison of different methodologies to represent their behaviour in FDS, the simulation of real scenarios of nuclear power plants and the definition of a methodology to define fire scenarios in FDS and simulation and comparison of results versus the experimental results of the project OECD/NEA-PRISME (Propagation d'un incendie pour des scenarios multi.locaux elementares) (NEA/CSNI/R(2017)14 Investigating Heat and

226   Smoke Propagation Mechanisms in Multi-Compartment Fire Scenarios. Final Report
227   of the PRISME Project, 2018). Finally, a freeware was developed to facilitate the FDS
228   computational modelling of nuclear power plant scenarios.

229   • Collaborative project with the CSN entitle "Metodologías avanzadas de análisis y
230   simulación de incendios en centrales nucleares" ("Advanced methodologies for
231   analysis and simulation of nuclear power plant fires") from 2020 to 2024. Following
232   the steps marked by the previous project, this project also adds the study of machine
233   learning algorithms to improve the definition of the FDS input parameters and to
234   decrease the computational cost. It also included the modelling of the detection and
235   extinction systems, the development of a database and the improvement of the freeware
236   to define FDS fire scenarios in nuclear power plants.

237   • Research project "NUCLEVS - Validación, calibración y aplicación de modelos de
238   propagación de incendios en escenarios reales de Centrales Nucleares" ("NUCLEVS -
239   Validation, calibration and application of fire models in real Nuclear Power Plant
240   scenarios"). In this project, the validation of different computational fire models is
241   analyzed. Based on this, an experimental test campaign will be performed to validate
242   some phenomena like extinction due to lack of oxygen. Other phenomena like cable
243   fire propagation will be validated by the use of literature available from experimental
244   tests. Finally, it is proposed the development of a methodology applicable to real plant
245   scenarios that facilitates Probabilistic Fire Safety Analysts to correctly define the
246   maximum expected fire scenario.

247   These projects have allowed the participation of the UC jointly with the CSN in the common
248   OECD PRISME 3 and FIRE Benchmark Exercise. The following three-step methodology for
249   conducting this Benchmark Exercise was employed: (1) a calibration phase, (2) a blind
250   simulation of a PRISME cable fire experiment, and (3) the real fire event simulation. The
251   importance of this type of benchmark lies in the still need of validating fire computer models
252   under certain fire related phenomena, as cable fire propagation (Plumecocq et al., 2023).

253   The application of CFD to predict fire propagation in NPP has been incremented since the
254   publication of the NFPA 805 that allows the performance-based design in NPP (NFPA 805
255   Performance-Based Standard for Fire Protection for Light Water Reactor Electric Generating
256   Plants, 2020). Nevertheless, the NFPA 805 indicates that the CFD has to be verified and
257   validated for the specific phenomena explored in the study. For this reason, although fire CFD
258   models, such as FDS, have been widely validated (NUREG-1824 Verification and Validation
259   of Selected Fire Models for Nuclear Power Plant Applications Supplement 1 FINAL REPORT.
260   U.S. NRC, EPRI, 2016), more works and assessments as those developed in the projects in
261   which the UC and the CSN collaborate are required to extend its applicability and increase
262   their reliability.

2.3. Nuclear Spent Casks

264   **ENUSA** has developed a strong research portfolio in the investigation of nuclear spent casks.
265   Their main goal is to obtain a better understanding of the different phenomena and to study the
266   capabilities for specific designs. The greatest efforts within the topic of spent fuel storage are
267   specifically the dry storage systems. Due to the increasing need for medium-term spent fuel
268   management, the realistic and accurate calculation of the peak cladding temperature during its

269 dry storage is a key parameter. There are different tools to perform simulations, being one of
270 the most advanced the use of CFD.

271 Since 2012, ENUSA has been participating in different projects for the thermal characterization
272 of spent fuel casks using different tools (CFD and COBRA-SFS, that is a code specific for
273 spent fuel package thermal analysis). Regarding the CFD codes that are the object of interest
274 of this paper, in particular the following projects have been carried out. In 2012, a simulation
275 of a ventilated dry storage cask was carried out under normal, off-normal and accident
276 conditions and the results were compared with the experimental test performed by KAERI
277 (Barrera et al., 2012). In 2017, a three-year project in collaboration with the UPM has been
278 conducted to analyse the thermal behaviour of both the TN-24P spent fuel cask and the fuel
279 inside it, and benchmarking against experimental measurements available (Benavides et al,
280 2018). In 2018, ENUSA collaborated with UPM to study the thermohydraulic response of a
281 single BWR fuel assembly in the DCS (Dry Cask Simulator) experimental cask, obtaining
282 results consistent with the experiments carried out in the Sandia National Laboratories
283 (Benavides et al., 2019 and 2020). Based on these studies, it has been possible to develop a
284 validated methodology for the thermal calculation of spent nuclear fuel containers. Currently,
285 ENUSA is working on a project in collaboration with the CSN to thermally characterize the
286 HI-STORM100 container considering the PWR fuel elements in detail, without simplifying the
287 geometry by using porous media and validated with the subchannel code COBRA-SFS
288 (Esteban and Matías, 2022).

289 In this way, the main objective of these projects has been to verify that realistic calculations
290 can be made using CFD code, reducing the uncertainties compared to conventional
291 approximations that are not capable of observing certain phenomena such as the flow path
292 between the fuel rods and the grids. Consequently, the management of spent fuel is improved
293 by calculating the efficient heat dissipation that determines the temperature distribution in the
294 cask/fuel.

295 Additional details concerning activities associated with spent fuel can be found in the parallel
296 paper that will be featured in the same special issue (Feria et al., 2023), which also includes
297 contributions from ENUSA and UPM focusing on CFX simulations in spent fuel containers.

2.4. Fuel assemblies

299 Related to the flow over fuel assemblies, different projects were carried out by **ENUSA**. This
300 includes the participation in the phase 1 of the NESTOR-Round Robin benchmark (EPRI
301 Technical Report 2014). The objective of this benchmark was to assess CFD temperature,
302 velocity and pressure loss predictions of the flow over a 5x5 fuel rod bundle with Simple
303 Support Grid against experimental results.

304 In collaboration with **Westinghouse** and **UPM**, hydraulic forces were calculated to study fuel
305 assembly distortion. ENUSA studied single and multiple fuel assembly spans to calculate
306 lateral forces as a function of fuel rod bow and explore the effect of local hydraulic forces on
307 fuel assembly distortion. Pressure drop over the grids was benchmarked against test results.
308 The lateral hydraulic forces were then used in mechanical codes to assess the impact during
309 operation (Petrarca et al., 2015) (Corpa Masa et al., 2015). A characterization of the flow
310 around the fuel assembly bottom nozzle was carried out to obtain a 3D velocity profile for the
311 water downstream of it. Also, different bottom nozzle designs were studied to verify that

312 changes in the flow distribution and fluid pressure drop due to each design has no safety impact
313 (Corpa Masa et al., 2014).

314 In collaboration with UPM, two projects regarding fuel assembly integrity were carried out
315 with computational codes. One was to develop a methodology to study GTRF. The
316 methodology was based on the calculation with CFD codes of the forces subjected to
317 turbulence-induced fluctuation that cause small amplitude vibrations in the rod. These forces
318 are then used as inputs for a FEA code that calculate fuel rod motion (Matías et al., 2019). The
319 other study was related to fuel assembly drop accident into the spent fuel pool. The impact
320 velocity as a function of the drop height was calculated as a realistic input to calculate the
321 mechanical damage produced by the accident. The objective of the projects related to fuel
322 assembly is to have a better understanding of fuel assembly performance to ensure integrity
323 during operation and handling. This allows to go to increased cycle length and burnup and
324 more challenging operational scenarios.

325 In the last years, ENUSA has also been developing activities related with radiological
326 characterization robots and drones, for which CFD simulations were carried out during the
327 design of these.

2.5. Containment

329 The **UPM** has participated in several projects related to CFD codes in plant application for
330 containment analysis, especially hydrogen behavior during a severe accident. Since 1995, UPM
331 has participated in CFD simulation activities in the FWP of the EURATOM and the European
332 Commission (EC) and projects for CSN and OECD/NEA. A summary of the different projects
333 in which the UPM has participated, along with the main objective of each one is shown below.

334 Containment Analysis with CFX (1995 – 2004):

335 • CEC-SCA-VOASM: 1997 – 1999: Validation of a simulation methodology for
336 hydrogen mixing and catalytic recombination. Some steady states of Zx experiments in
337 BMC Containment with PAR were simulated with a structured CFD code. These results
338 were compared with similar simulations of other organizations and the experimental
339 results. Finally, the results were applied to representative accidental conditions scaled
340 to the experimental facility (Martín-Valdepeñas et al., 1999).
341 • CSN/UPM: 1995 – 2004: Several projects were done in collaboration with CSN related
342 to hydrogen behavior within containment and application to Level 2 Probabilistic
343 Safety Analysis. CFD plant applications were done during these projects for a
344 restorative analysis in a Steam Generator room to a simplified full-scale PWR plant
345 (Martín-Valdepeñas and Jiménez, MA, 2003).
346 • CEC-EXPRO: 2001 – 2004: Experimental and numerical study of reactive flows in
347 complex geometries with relevance to industrial safety for explosion protection. A
348 methodology combining integral codes and a CFD code was developed during this
349 project.

350 Containment analysis with GOTHIC (2013 - present):

351 • Iberdrola Engineering/UPM (2013-2017): several projects were done in collaboration
352 with Iberdrola Engineering, including the PAR location and sizing for the Cofrentes

Nuclear Power Plant containment building with a 3D GOTHIC model (Jimenez et al., 2015; Serrano et al., 2016; Díez Álvarez-Buylla et al., 2021) and the estimation of the hydrogen combustion risk in the Filtered Containment Venting System of Cofrentes NPP (Fernández-Cosials et al., 2018).

- CNAT/UPM (2013-): since 2013, Almaraz and Trillo NPPs have supported the UPM with a continuous line of research with GOTHIC from 2013. In those years, several 3D models have been created for both Almaraz NPP (Vázquez-Rodríguez et al., 2019) and Trillo NPP (Fernández-Cosials et al., 2019). Extensive work regarding Best Estimate Plus Uncertainty (BEPU) applied to a design basis accident for a PWR-W containment building was also done in the framework of this collaboration (Bocanegra and Jimenez, 2018).

- Ministry of Economy and Competitiveness/UPM (2016-2019): in the framework of the PYrolysis Gas Analysis and Safety project (ENE2015-67638R), it was possible to develop an AP1000 3D full containment model with GOTHIC (Estévez-Albuja et al., 2021a), validating the model both with experimental data from the Technical University of Munich (Estévez-Albuja et al., 2018) and from the North China Electric Power University (Estévez-Albuja et al., 2021b).

- EC-HORIZON2020/UPM (2020-2024): the UPM coordinates the AMHYCO European project (Herranz et al., 2022; Jiménez et al., 2022), in which several works related to 3D containment modelling are in progress (Serra et al., 2021), as well as PAR coupling with GOTHIC (Domínguez-Bugarín et al., 2022).

- Cardiff University/University of Liverpool (2020-2024): The UPM has worked in collaboration with the mentioned universities in helping to develop a GOTHIC model of a VVER-1000 nuclear power plant, (Kanik et al., 2024), (Kanik et al. 2022).

2.6. Other applications

The **UPV** has collaborated with the CSN de España in the validation of CFD codes through the implementation of benchmark activities featuring a variety of topical issues related to nuclear safety. Since 2008, UPV has participated in the CFD simulation activities organized by the WGAMA CFD Task Group, which is sponsored by the OECD, the NEA, and the CSNI. The main objectives pursued by these benchmarks are to develop BPGs for the use of CFD, define calculation strategies for predicting uncertainty, and establish guidelines for enhancing CFD-grade experiments for verification and validation purposes. Different organizations and entities worldwide are granted access to the measurements of a CFD-grade experiment and use them to validate their computational approaches. Moreover, WGAMA group activities involve assessing the state of the art and maintaining several reference documents, such as the aforementioned BPGs for the use of CFD in reactor safety (OECD/NEA/CSNI, Mahaffy J. et al., 2014). A summary of the different benchmarks in which the UPV has participated, along with the main objective of each one is shown below.

- OECD/NEA-Vattenfall CFD Benchmark Exercise 2008-2011: Validation of a CFX model for the experiment at the Älvkarleby laboratory facilities on fluid mixing at different temperatures in a T-junction to predict the important parameters affecting high-cycle thermal fatigue (OECD/NEA/CSNI, Smith, B.L. et al., 2011).
- OECD/NEA-KAERI CFD Benchmark Exercise 2010-2012: Development of a model for the turbulence parameters' prediction downstream of a generic design of spacer grid

398   in a rod-bundle geometry (MATiS-H experiment) (OECD/NEA/CSNI, Smith, B.L. et
399   al., 2013).
400   • OECD/NEA-PSI CFD Benchmark Exercise 2012-2014: Validation of the buoyancy
401   prediction capabilities of CFX for He stratification to address the erosion by a buoyant
402   jet in one of the PSI PANDA vessels (Andreani et al., 2015).
403   • OECD/NEA-PSI CFD Benchmark Exercise 2014-2016: PCE UQ methodology
404   development for CFD simulations through the PSI GEMIX database on the mixing of
405   two fluids with different (OECD/NEA/CSNI, Fokken, J. et al., 2017).
406   • OECD/NEA-Texas A&M University Benchmark Exercise 2017-2019. Extension of
407   PCE methodology for the normal uncertain variables within the frame of the Cold-Leg
408   Mixing experiments at Texas A&M University about fluid mixing in presence of
409   buoyancy effects (OECD/NEA/CSNI, Vaghetto, R. et al., 2022).
410   • OECD/NEA-OKBM Afrikantov Benchmark Exercise 2019-2021. Validation of a one-
411   way coupled CFD-FEA model for the fluid-structure interaction during vortex shedding
412   appearance in a channel with two bottom-fix cylinders.

413   So far, all the benchmarks organized by WGAMA CFD group correspond to single-phase
414   applications. UPV has employed ANSYS commercial code, especially CFX, for this series of
415   exercises focusing on the proper use of available models for each scenario. Since 2014, the
416   different activities carried out have focused not only on obtaining predictions from CFD code,
417   but also on extending the analysis to uncertainty quantification. Analyzing the uncertainty of
418   the results is highly relevant if this tool is to be used in the future to make informed decisions
419   that may affect public safety. Accurately calculating the uncertainty associated with
420   simulations enables a greater degree of understanding of the studied phenomena and will allow
421   for informed decision-making in nuclear technology in the coming years. UPV has mainly
422   focused its efforts on developing and applying the Polynomial Chaos Expansion propagation
423   method for uncertainty quantification in CFD calculations (Rivera Y. et al., 2021).

424   In the framework of the projects started at the **UPM** in the 90s—more information on the
425   section dedicated to containment analysis—a multipurpose commercial CFD code was tailored
426   and validated for hydrogen safety analyses. The most important items are listed below, with
427   more information available in (Martín-Valdepeñas, et al., 2007):

428   • Several condensation models with non-condensable gases were implemented into the
429   commercial software using external functions coded in Fortran at the UPM.
430   • Hydrogen flame acceleration and deflagration to detonation transition criteria (Sigma
431   and Lambda criteria) were implemented as online post-processing of the CFD code
432   results.
433   • Several benchmark and validation exercises were done for different integral
434   experiments: Zx-BMC, NUPEC and MISTRA.
435   • OECD/NEA-PSI CFD benchmark exercise (2014-2016): in the framework of the
436   international benchmark exercise 3, validation exercises were made in the group for a
437   helium stratification test made in PANDA within HYMERES project (Fernández-
438   Cosials et al., 2016).
439   • CSN/UPM (2019-2023): in the framework of the GO-MERES project, several
440   collaborations with PSI within HYMERES2 and PANDA projects from OECD/NEA
441   took place. The main topics were BWR suppression pool test and PWR spray test
442   (Vázquez-Rodríguez et al., 2023). In this project, plant application cases were done for

443 PWR and BWR reactors, learning from the PANDA experimental test simulations with
444 GOTHIC.

3. Research key outcomes

446 The long trajectory followed by the different Spanish research groups, described in the previous
447 section, has led to the achievement of a series of major milestones. In particular, the different
448 groups have contributed to development and validation work in the various fields of 3D codes.
449 Their major contributions are described in the following subsections taking into account the
450 different CFD research topics.

3.1. Code Development

452 During the period 2003-2006, the **UPCT**'s research in collaboration with CEA Saclay (France)
453 led to extend some Riemann Solvers and flux schemes classically used in supersonic flows to
454 multidimensional, transient two-phase flows problems under compressible flow conditions.
455 Specifically, AUSM schemes were extended for both two-fluid mixtures of water and air and
456 phase change processes between water and steam and they were benchmarked with reference
457 problems with satisfactory results.

458 The implementation of international benchmarks in nuclear safety has enabled the scientific
459 community to gather valuable information regarding the most suitable CFD methods for each
460 scenario, based on the underlying physics of the phenomena. Potential issues have been
461 identified where the proper utilization of CFD allows for a more detailed analysis, thereby
462 enhancing the precision with which safety assessments are evaluated. Additionally, the Best-
463 Practice Guidelines have been continuously updated, along with relevant documentation for
464 uncertainty quantification (Bentaib et al., 2022).

3.2. Combustion and Fire Safety

466 The collaboration among **UPCT**, IRSN at Fontenay-aux-Roses and CNSR at Orleans (France)
467 permitted testing of different turbulent combustion models coupled with LES for simulating
468 H_2 combustion with spherical bomb experiments. Simulations and combustion experiments
469 performed in a spherical bomb equipped with fans permitted to assess the effect of turbulence
470 in the H_2 combustion sequences. Results showed that under high turbulent conditions LES
471 models are more effective in predicting turbulent combustion characteristics than RANS
472 models under "blind" conditions. In this context, an ATF model , also called thickening flame
473 model (Collin et al., 2000) and a flame progress variable model were tested with LES and
474 validated for predicting key combustion parameters under different H_2-air conditions. Results
475 also showed that, for a fixed mesh, the ATF model was more robust than the flame progress
476 variable model and provided a similar level of accuracy but with the advantage of providing a
477 detailed description of the chemical composition of the flow, which is key in some sequences
478 involving ignition, flame acceleration or quenching phenomena.

479 In the last years, the research group of Thermal and Energy System Modelling at UPCT has
480 focused on the validation of turbulent combustion models able to predict flame acceleration in
481 hydrogen-air mixtures. In this context, UPCT represented Spain in the ETSON Benchmark
482 within the SAM-HY-CO project. UPCT showed that ATF turbulent combustion models
483 coupled with detailed chemical kinetic schemes and LES were able to predict flame

acceleration experiments of lean mixtures of H_2 in air performed in ENACEFF-2 facility at CNRS Orleans (France). The use of different sub-grid scale (SGS) turbulent models in LES (i.e. Smagorinsky-Lilly, WALE, k-equation model, dynamic k-equation model) did not significantly influence the results of this type of unsteady sequences. Besides, strategies based on Dynamic Adaptative Chemistry, In-situ Adaptive Tabulation and Adaptative Mesh Refinement were very effective to overcome the limitations due to the requirements of mesh resolution imposed by the use of LES and partially reduce the computational costs of the simulations. Other numerical strategies like the PSR model implemented in iLES (Berglund, 2010) and detailed chemistry models were also able to predict flame acceleration but required higher grid resolution. Specifically, this strategy was very effective to predict vortex-flame interaction and its contribution to flame acceleration. However, it was found that a minimum spatial resolution of 1/8 of the laminar flame thickness was required to predict correctly flame acceleration in the initial steps of the combustion sequence (Nicolas et al., 2021a). This increases the computational costs and limits the use of iLES to local regions in specific time lapses within scenarios such as the initial stages of flame growth and initial acceleration due to flame-structure interactions. Regarding numerical schemes, PISO, AUSMup and AUSM+ showed better prediction capacities than other more diffusive schemes like Rusanov's. AUSM was confirmed to be very efficient in transonic conditions, and its variations AUSM+, and AUSMup were also adequate for a wide range of Mach values.

All in all, the use of detailed chemistry models is key to predict not only flame acceleration but also other phenomena like deflagration to detonation transition or quenching (Figure 1). The use of different chemical models tested (detailed with 27 equations, detailed with 21 equations or reduced with 12 equations) did not significantly influence the results as long as the species H, O, OH, H_2O_2 and HO_2 were considered in the scheme. Moreover, recently, H_2-CO-O_2 chemical kinetic models were also tested with the LES approach to simulate combustion experiments of H_2 and graphite in a spherical bomb (Nicolas et al., 2021b). Results showed that the H_2-CO-O_2 combustion model of Yetter et al. (1991) that included C, CO, CO_2, HCO, H, O2, O, OH, H_2O, H_2O_2 and HO species coupled with LES was able to predict pressure evolution with time during the sequence with results in accordance with experiments.
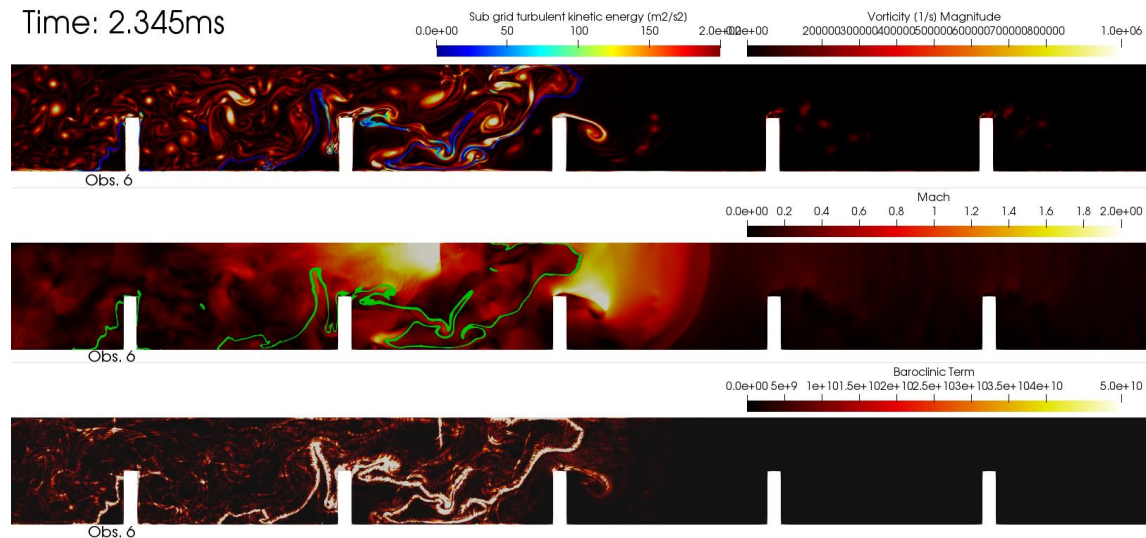

Figure 1: Simulation of H_2 combustion in a channel with LES and Williams detailed chemisty scheme. The reaction flame front is identified by a green line. The figure shows the flame acceleration mechanism due to flame interaction with the obstacles. Both, the obstacles and the magnitude of the baroclinic term ($\nabla \varrho \times \nabla p$) induced by the flame, contribute to the production of vorticity what enhances mixing and flame acceleration. Numerical benchmark (Case B) (Nicolás et al. 2021).

519 The activities conducted by **UC** in collaboration with **CSN** in the different projects exposed in
520 the research synthesis section can be divided into several topics connected to achieving the
521 final goal of increasing fire safety in nuclear power plants through the use of CFD. The topics
522 studied included thermal decomposition and combustion analyses of materials that could be
523 present in a nuclear power plant. Furthermore, different methodologies and experimental tests
524 have been developed to define different materials in the various fire scenarios defined with
525 CFD. Additionally, different real fire scenarios have been analysed with the CFD model FDS
526 (McGrattan, y otros, 2023), examining the influence of input parameters and comparing the
527 results with large-scale experimental tests. Moreover, participation in the benchmark organized
528 under the support of the PRISME program has allowed for the evaluation of user effects during
529 the modelling of cable tray fire propagation. In addition, a specific experimental test campaign
530 has been conducted by UC-CSN to analyse cable fire propagation and improve data for
531 validating computational models. This has also led to the definition of a specific model to
532 address phenomena such as cable tray fire propagation. Finally, standards have been evaluated
533 to enhance safety in nuclear power plants through the use of CFD.

534 According to the definition of the material's thermal behaviour in the CFD model FDS, it is
535 necessary to conduct prior experimental characterization. To do so, thermal analysis apparatus,
536 such as a simultaneous thermal analyser, can be utilized to study thermal decomposition
537 reactions. Nevertheless, these techniques are very sensitive to the selected boundary conditions,
538 which need to be carefully defined by the user based on the material and desired results. In this
539 regard, UC-CSN has conducted important research on the influence of boundary conditions in
540 the simultaneous thermal analysis on the thermal decomposition of thermoplastic polymers
541 (Lázaro D. , Lázaro, Alonso, Lázaro, & Alvear, 2019). Additionally, it has been analysed how
542 the energy release by certain polymers in this apparatus can affect the results (Alonso A. ,
543 Lázaro, Lázaro, & Alvear, Self-heating evaluation on thermal analysis of polymethyl
544 methacrylate (PMMA) and linear low-density polyethylene (LLDPE), 2022). For complex
545 components present in nuclear power plants, such as cables composed of multiple materials, it
546 is common to perform bench-scale experimental tests, such as cone calorimeter tests, to analyse
547 the entire cables and obtain the heat release rate per unit area, which can be directly used in
548 FDS to define cable behaviour during combustion (Lázaro, Alonso, Lázaro, Jiménez, & Alvear,
549 Experimental and analytical study of the influence of the incident heat flux in cables heat
550 release, 2023).

551 Once the materials and components have been experimentally characterized, the data needs to
552 be processed. Some material properties, like thermal conductivity, can be directly obtained
553 from experiments, while other properties, such as kinetic parameters used to define reactions,
554 require different methodologies. Throughout the projects, UC-CSN has developed new
555 methodologies to obtain thermokinetic parameters for describing the thermal behaviour of
556 materials in the CFD model. The methodology developed has specific characteristics, including
557 a simple direct method defined in (Lázaro, Alonso, Lázaro, & Alvear, A Simple Direct Method
558 to Obtain Kinetic Parameters for Polymer Thermal Decomposition, 2021) for obtaining kinetic
559 parameters for polymer thermal decomposition in a few seconds. More complex methodologies
560 using artificial intelligence techniques, such as genetic algorithms, have been defined in
561 (Alonso A. , Lázaro, Lázaro, Lázaro, & Alvear, 2019) and (Alonso A. , Lázaro, Lázaro, &
562 Alvear, Numerical Prediction of Cables Fire Behaviour Using Non-Metallic Components in
563 Cone Calorimeter, 2023). These methodologies use a CFD model of an experimental test, like
564 cone calorimeter (Figure 2a), to obtain the thermokinetic parameters that best fit the employed
565 CFD model. This is important because, due to simplifications in some equations of the CFD
566 models, directly using properties from experimental tests does not always yield the best CFD

results. This is also why a study has been conducted on the influence of input parameter definition on decomposition prediction performed by CFD (Alonso & Alvear, 2020). The definition of cable combustion can be performed in FDS by directly imposing the heat release rate per unit area, but the incident heat flux on the cables can greatly affect this parameter, as evaluated by UC-CSN with the cone calorimeter tests (Lázaro, Alonso, Lázaro, Jiménez, & Alvear, Experimental and analytical study of the influence of the incident heat flux in cables heat release, 2023). UC-CSN propose a new methodology to better consider the effect of the incident heat flux on the heat release by the cables. Finally, it is also crucial to define the gases released during a fire in nuclear power plant simulations. UC-CSN has developed a methodology to estimate the FDS chemical reaction equation for modelling toxicity in fire scenarios (Lázaro D. , Lázaro, Alvear, & Jiménez, Methodology to estimate the FDS chemical reaction equation in fire scenarios, 2022).

In addition, fire modelling in nuclear power plants involves predicting a large number of phenomena, whereas it is necessary to evaluate and validate the CFD model against real scenarios by comparing simulations with large-scale experimental tests (Figure 2b). Some related achievements by UC-CSN include evaluating the influence of mesh size in real scenarios, as described in (Lázaro et al., 2016; 2017a; 2017c; 2018). The works performed on validating the species generated during a compartment fire (Lázaro et al., 2017b) and analysing the effect of ventilation on fire propagation in nuclear power plants (Lázaro et al., 2021) have also been significant. To improve the application of fire PRA, UC-CSN has evaluated the cable failure probability caused by a fire originating in a cabinet (Lázaro et al., 2020). All of these efforts have contributed to the definition of best practice guidelines for fire modelling in nuclear power plants.

With the goal of CFD validation and defining best practices for simulations, the UC-CSN team has participated in the OECD PRISME 3 and FIRE Benchmark Exercise (Plumecocq et al., 2023) (Figure 2d). During the different phases of this benchmark, UC-CSN has developed a modification of the FLASH-CAT model (NUREG/CR-7010, 2012) to predict fire propagation in cable trays and has subsequently used the results of the propagation and heat release by the cables as input in FDS to predict fire manifestations in the nuclear power plant (Lázaro et al., 2022). This developed model has also been validated with large-scale experimental tests conducted in the Room Corner Test Apparatus by UC-CSN to analyse fire propagation in cable trays (Lázaro et al., 2022) (Figure 2c).

Another specific model being developed by UC-CSN to improve CFD application in NPPs is the neural network-based model described in (Lázaro et al., 2023). This model predicts tenability conditions during a fire in a nuclear power plant in real time by training it with validated fire computational models.

To provide users with a database that contains typical fire sources and material properties, as well as to facilitate the definition of CFD simulations, a tool called NuclearFire has been developed (NuclearFire 1.0). This tool is being improved for a second version, which will include a large database and more options for scenario definition.

The advances developed by UC-CSN also have a regulatory scope. To this end, analyses and comparisons of standards to enhance nuclear power plant safety have been carried out by the use of CFD. It allows performing detailed calculations in order to reduce the conservatism inherent in deterministic requirements. Additionally, in complex scenarios where this regulation may be difficult to satisfy, the use of CFD to support risk-informed regulation can

612 be applied to demonstrate that fire damage to structures and systems important to safety does
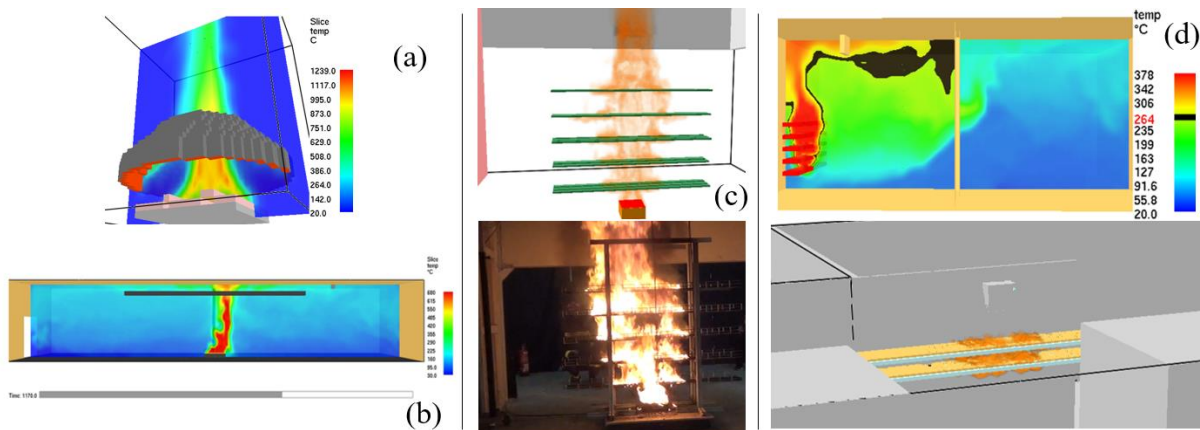613 not affect their integrity.

614

Figure 2: Summary of some CFD fire safety results: (a) characterization and validation of thermal behaviour of typical components of NPP, (b) Modelling of real NPP fire scenarios, (c) Validation of CFD models by comparing with real scale experimental tests, and (d) Common OECD/NEA FIRE and PRISME Cable Benchmark Exercise.

3.3. Nuclear Spent Casks

The advancement of CFD techniques in the nuclear industry over the years has made possible to obtain verified and validated CFD models for successful simulations and reliable results, as well as experience in dealing with specific thermal analysis problems. Within the expertise for CFD modelling, the solution algorithm must be defined, an appropriate turbulence model that gives reliable results for the characteristic flow of the problem must be chosen, and numerical parameters such as differentiation schemes, relaxation factors, convergence criteria, etc. must be correctly established.

In projects based on spent fuel containers carried out by **ENUSA**, for the thermal characterization of the cask/fuel through CFD codes, a two-step coupling methodology is followed. The first step consists of simulating the outside of the cask, modelling the external part of the container on the environment and its support slab; and the second step consists of simulating the inside of the cask that contains coolant gas surrounding the fuel elements, applying the heat transfer boundary conditions that are extracted from the first step. In this way, the precision of the calculations is improved employing numerical simulations with specific turbulence models, with respect to the use of empirical or semi-empirical correlations for the heat transfer mechanisms of the outer part of the cask (Benavides et al., 2018). This iterative two-phase methodology allows knowing the distribution of temperatures as well as the mechanisms of heat transmission inside the container. As a result, we can know one of the most important parameters which is the maximum rod temperature, and which must not exceed a specific value in accordance with current regulations.

This methodology has been correctly implemented with their respective sensitivity analysis of the mesh and of turbulence models, and validated through experimental results in TN-24P and DCS experimental cask. Also, for the DCS cask an approach of spacers in the fuel assembly modelled with porous media has been conducted. It has allowed it to be a reliable and useful methodology for its implementation in other containers such as HI-STORM 100. The modelling of the inside of the HI-STORM 100 with a high level of detail has resulted, despite

16

its simplification in 1/8 of its cross section, in a computationally expensive model due to the large number of cells generated in the meshing of the fluid volume of helium (Esteban and Matías, 2022). The simulation, performed with a Realizable K-Epsilon Two Layer Buoyancy Driven turbulence model, has allowed to realistically predict the behavior of the fluid and the phenomena inside the canister with a very detailed geometry in addition to adequately capturing the heat transfer by the elements such as grids, nozzles and rods (Esteban and Matías, 2022) (Figure 3a).

For the TN-24P cask, the turbulence model selected for the exterior cask simulations has been the Realizable K-Epsilon Shear Driven and the Realizable K-Epsilon Buoyancy Driven for the inner cask simulations due to the good agreement against experimental data obtained, and faster time to convergence (Benavides et al., 2019). Also, a study of the gaps between the canister and the inner cylinder has been carried out. The temperature results obtained on the outer surface can be seen on (Figure 3b).

For the DCS container, the model used for the CFD simulations was the Realizable K-Epsilon. Several sensitivity studies have been performed under variety of heat loads and internal vessel pressures (Benavides et al., 2020). The case with the most accurate results is obtained, in addition to a better understanding of the different heat transfer mechanisms interactions under different conditions.
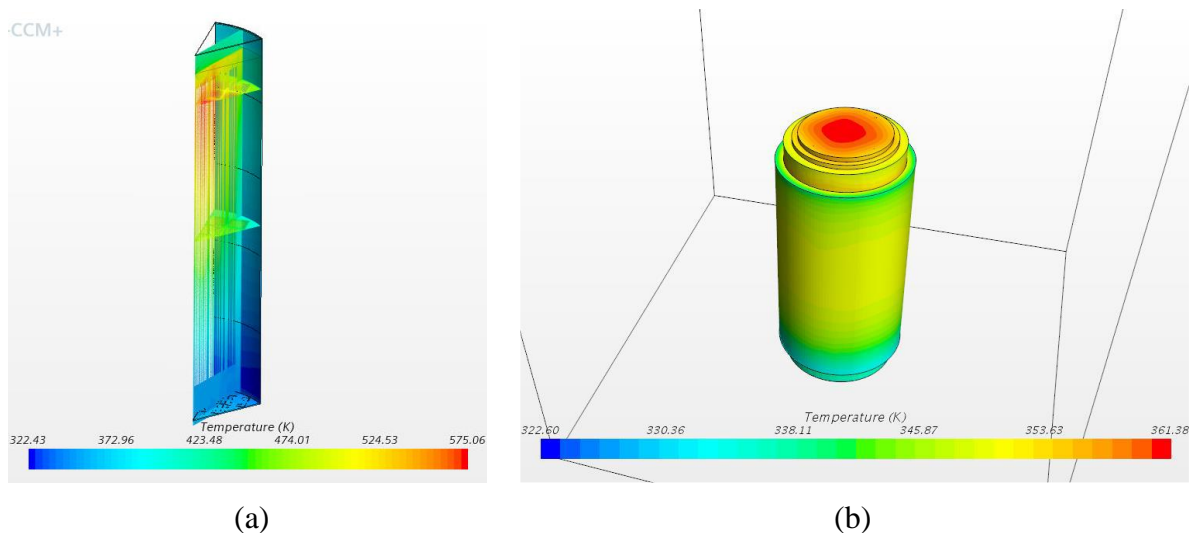

| (a) | (b) |

Figure 3: Temperature distribution of a) helium inside the canister in the radial and axial direction (HI-STORM 100), b) shell of the TN-24P cask.

3.4. Fuel assemblies

Related to fuel assembly projects carried out by **ENUSA**, the main objective is to have a better understanding of fuel assembly performance to assure the integrity during operation and handling. This allows to go to increased cycle length and burnup and more challenging operational scenarios. The calculation of hydraulic forces in operating conditions using porous media and hydraulic resistance force in the FADA accident, were used as inputs to mechanical code to study the integrity of the fuel assembly in that situation. For these projects and others such as the study of different bottom nozzles designs, standard k-epsilon models were used, as the averaged values of forces and velocities are enough to characterize the flow. Other problems like GTRF (Figure 4a), for which pressure or velocity fluctuations need to be properly characterized, have to used more complex models such as LES turbulence models.

683 One of the conclusions of all these projects is that simplifying geometries is a good way to
684 model properly some problems, for example by replacing zones with less influence on the flow
685 with porous media, applying symmetries or analysing only a part of the geometry.

686 Also, CFD codes could be used to design other elements that are also relevant for the future of
687 the nuclear fission energy. In the case of ENUSA, some simulations were done to help in the
688 design of robots and drones for radiological characterization (Figure 4b).

689

(a)   (b)
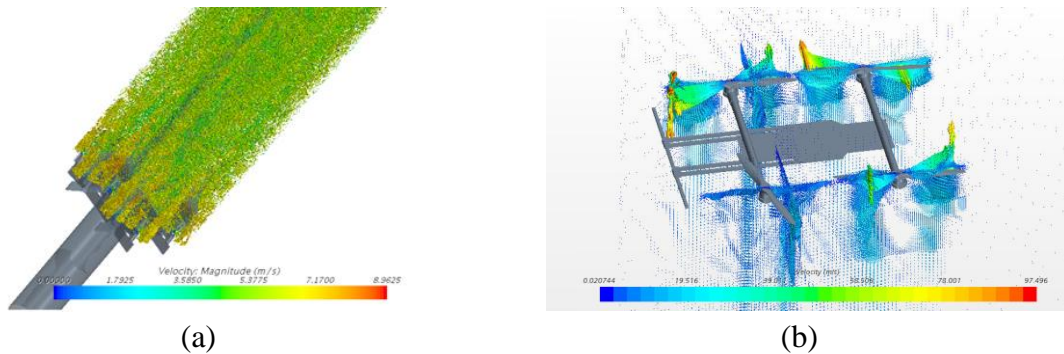
690 Figure 4: (a) Q-criterion isosurfaces around the rod fuel for GTRF analysis, and (b) Velocity vector around a
691 drone.

3.5. Containment

693 As results of the projects carried out by the **UPM** indicated in the Research Synthesis and
694 extending their outcomes, a PhD thesis was developed (Martín-Valdepeñas et al., 2007) in this
695 work CFX commercial CFD code was adapted for hydrogen containment analyses:

696 • Two examples of full plant applications were completed: a 3-loop large-dry PWR
697 Westinghouse design and a 3-loop PWR KWU design. This analysis was done only for
698 short temporal windows during the hydrogen release phase of the in-vessel accident for
699 a limited number of sequences due to computational limitations.

700 The main outcomes after ten years of research modelling in 3D with GOTHIC are:

701 • The continuous evolution of the 3D model development methodologies with GOTHIC
702 has found an optimal balance between computational cost and thermal-hydraulic
703 resolution of the models. The relative coarse-mesh approach used by GOTHIC allows
704 addressing the long transients (several hours) of hypothetical accidents driven by
705 phenomena with small characteristic scales (below the millimeter) within large
706 computational domains (buildings 60 m in height and 40 m in diameter). Depending on
707 the containment design, different strategies of geometrical simplification and mesh
708 adaptation are needed, varying from a "Nesting Dolls" approach (Estévez-Albuja et al.,
709 2021a; Jimenez et al., 2015) to an *a priori* adaptation of the geometry to the mesh,
710 called Preventive Methodology (Arfinengo-del-Carpio et al., 2021), as shown in Figure 5.
711

Mesh-fitted geometry

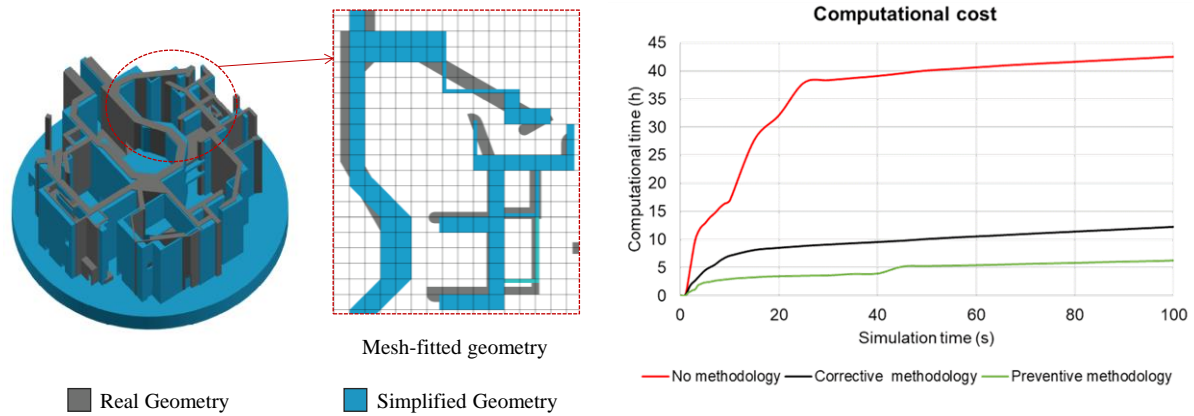■ Real Geometry   ■ Simplified Geometry

Figure 5: Computational cost optimisation via an a priori adaptation of the geometry to the mesh.

- The code validation has proven its ability to reproduce the main drivers of the containment thermal hydraulics such as wall condensation with large concentrations of non-condensable gases, buoyancy-driven flows, conjugate heat transfer, etc. (Vázquez-Rodríguez et al., 2022). Furthermore, under the correct set of hypotheses, it is also possible to replicate experiments on the main containment safety systems of PWR-W, such as the spray operation (Vázquez-Rodríguez et al., 2023) and the PAR actuation.
- From the 3D modelling work done previously in the research group (Bocanegra et al., 2016), it was also possible to evaluate the equipment qualification and instrumentation surveillance criteria for a PWR-W containment (Jimenez et al., 2017), developing a methodology for estimating the hydrogen combustion risk in a severe accident (Fernández-Cosials et al., 2017), an evaluation of the information on the hydrogen concentrations that may be available for the operators in the control room to use the FCVS as a hydrogen mitigation measure (Díez Álvarez-Buylla et al., 2021), and a detailed characterization of the 3D flow patterns within the containment of the AP1000 (Estévez-Albuja et al., 2021a).
- For the ex-vessel phase of a severe accident, further developments are needed regarding the mass and energy releases and the corium modelling.

The most recent research line is integrating all the information discussed above from some of the UPM in-house software whose development started in the 90s, such as PARUPM (Domínguez-Bugarín et al., 2022; Jimenez et al., 2007) and the sigma criterion for the hydrogen risk evaluation (Martín-Valdepeñas et al., 2007), to the *a priori* modification of the geometry and the implementation of the spray safety system. The combination of all the former research lines enables a comprehensive evaluation of the impact of different safety system actuation on the hydrogen risk in severe accident conditions. Figure 6 shows the impact of the spray actuation on the hydrogen risk based on the sigma criterion. The main outcome expected from this on-going work is to provide clear guidelines for severe accident management to minimize the risk of combustion inside the containment.
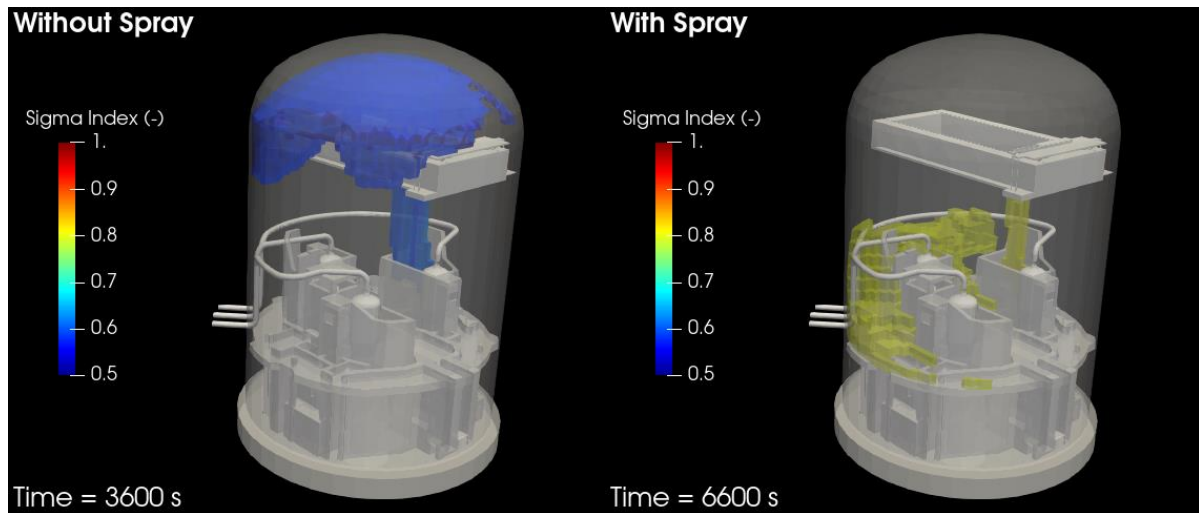
Figure 6: Spray actuation on the hydrogen risk based on the sigma criterion.

3.6. Other applications

The various activities conducted by **UPV** have predominantly focused on single-phase problems. Throughout the collaboration between UPV and CSN, multiple thematic areas have been covered, including thermal cycles and pressure thermal shock, the effects of spacer grids on turbulent parameter predictions, disruption of stratification layers through jets, UQ methodologies assessment, and calculations of buoyancy forces in different geometries. The research synthesis section provided an overview of each topic, offering additional details regarding the primary objectives and conclusions associated with each of them. Further information about geometry, mesh, models and data treatment employed by the UPV can be found in the respective references. Additionally, Figure 7 provides a summary of qualitative results from the benchmarks conducted by the UPV, aiming to facilitate readability and comprehension for each activity.

- In relation to the OECD/NEA-Vattenfall CFD Benchmark Exercise 2008-2011, the main goal was to understand and predict the physics behind the T-Junction problem. One of the key findings highlighted the significant importance of accurately representing turbulence in achieving satisfactory performance across the various codes employed. It was strongly recommended to utilize LES models as opposed to SAS or RANS models. LES models demonstrated superior capabilities in capturing the turbulent flow characteristics, resulting in improved simulation accuracy. UPV-LES simulations demonstrated a good agreement on the velocity profiles and turbulent parameters employing a relatively small computational power (OECD/NEA/CSNI, Smith, B.L. et al., 2011).

- Moving on to the OECD/NEA-KAERI CFD Benchmark Exercise 2010-2012, the activities in this benchmark exercise primarily focused on obtaining turbulent parameters following a generic spacer-grid configuration in a rod-bundle structure (Figure 7a). It was observed that incorporating second-order accurate discretization algorithms was crucial for obtaining reliable outcomes. The UPV employed a second-order discretization scheme for both time and space with a good agreement with the experimental data. Regarding the turbulence model, while RANS models performed well in calculating average velocity profiles, LES models offered superior predictive capabilities for the overall parameters studied. Nonetheless, computational cost again

play a role for some participants as the UPV which were limited to RANS or hybrid models (OECD/NEA/CSNI, Smith, B.L. et al., 2013).

- In the OECD/NEA-PSI CFD Benchmark Exercise 2012-2014, the focus shifted to containment calculations using CFD. This exercise provided insights into the dynamics of transient events within the containment structure (Figure 7b). As expected, user's choice of models emerged as a critical factor in accurately capturing the transients and ensuring the correct representation of the system behavior. Surprisingly, the selection of the turbulence model did not demonstrate a direct correlation with the correct prediction of transients. In fact, simulations employing LES models failed to meet expectations due to unfulfilled mesh requirements (Andreani et al., 2015).

- The most significant conclusions drawn from the OECD/NEA-PSI CFD GEMIX Benchmark Exercise 2014-2016 primarily focused on the development of uncertainty quantification methodologies in CFD codes. Traditional methods such as Wilks-type approaches are often computationally prohibitive in CFD simulations. In contrast, both metamodeling techniques (propagation) and extrapolation methods (comparing uncertainty in open and blind tests) achieved good results, even with a relatively small number of simulations. UPV started developing the PCE method with satisfactory results when using uncertain variables with uniform probability density functions (OECD/NEA/CSNI, Fokken, J. et al., 2017).

- The OECD/NEA-Texas A&M University Benchmark Exercise 2017-2019 was designed with a more complex geometry to push the boundaries further in single-phase applications with buoyancy effects and the quantification of uncertainty in CFD models (Figure 7c). In fact, the activity involved conducting simulations for approximately 42 seconds (real time), leading to a significant increase in computational cost. The results once again highlighted the importance of utilizing LES models to solve problems where the turbulent component heavily influences the fluid behavior. Regarding uncertainty quantification, UPV continued developing the PCE method, extending its use to uncertain variables with a normal probability density function (Rivera et al, 2021).

- The most recently completed activity corresponds to the OECD/NEA-OKBM Afrikantov Benchmark Exercise 2019-2021, and its results are not publicly available. The focus of this activity was to validate different approaches for calculating fluid-induced vibrations in a tube bundle (Figure 7d). Preliminarily, it can be mentioned that the choice of LES turbulence model and the level of detail in resolving vortices were crucial for accurately predicting the experiment as well as the coupling method between CFD and mechanical code. The one-way coupling approach applied by the UPV delivered good results for the cases where vortex shedding and the natural vibration frequency of the rods were not synchronized.
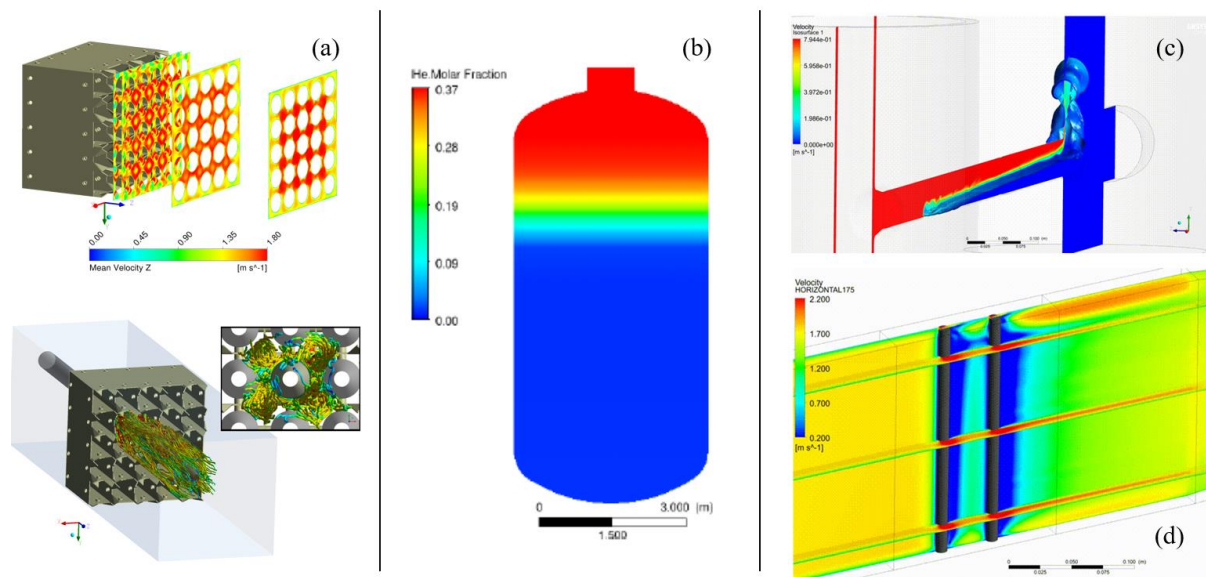
Figure 7: Summary of different benchmarking results: (a) Velocity profile and streamlines for the OECD/NEA-KAERI CFD Benchmark Exercise 2010-2012, and (b) Helium molar fraction OECD/NEA-PSI CFD Benchmark Exercise 2012-2014, (c) Velocity isosurface for the OECD/NEA-Texas A&M University Benchmark Exercise 2017-2019, (d) Velocity contours for the OECD/NEA-OKBM Afrikantov Benchmark Exercise 2019-2021.

4. Challenges and expected developments

4.1. Identified challenges and general roadmap

As has emerged in the course of the present review, over the recent decades there has been a notable increase in the use of CFD models for the prediction of flow and transport phenomena within nuclear safety issues. This shift towards 3D CFD methodologies can be attributed to the recognition that numerous safety-critical phenomena within these systems, such as boron dilution, pressurized thermal shocks, main steam line breaks (with inherent flow asymmetry), containment atmosphere mixing and stratification, and hydrogen combustion processes, inherently exhibit three-dimensional characteristics. The incorporation of CFD methodologies into formal NS studies represents a complex task characterized by several significant challenges. One prominent challenge arises from the absence of well-established and universally accepted methodologies. For instance, in cases requiring coupling with other physics and thermal hydraulics at different scales, a common strategy should be adopted by the community in order to enhance its capabilities. In addition, the high-requirements expected in safety studies within the nuclear field arise the need for UQ analysis with solid methodologies, proven for different CFD studies in the field. UQ methodologies, though powerful, often lack definitive proof of applicability due to the substantial computational demands and associated costs they entail. According to the NEA/CSNI/R(2016)4 report, UQ includes two distinct contributors: stochastic uncertainty, stemming from factors like the propagation of measurement uncertainties in boundary conditions, leading to a scatter around the 'real' value; and epistemic uncertainty, resulting from gaps in knowledge such as unknown boundary conditions, model geometry, or assumptions in physical modeling. Epistemic uncertainty cannot be quantified through statistical evaluations of parametric studies and frequently introduces biases into analysis results. While methodologies for UQ have been developed or adapted from system-scale analyses, their widespread adoption remains limited and a greater effort should be made by the CFD community, including Spain and Portugal.

844 Access to pertinent information constitutes another substantial challenge within the field of
845 CFD in NS studies. According to the OECD/Nuclear Energy Agency WGAMA CFD Task
846 Group ongoing studies, this challenge manifests in two dimensions: firstly, for researchers
847 employing CFD, it involves access to comprehensive databases housing experimental results,
848 including "CFD-grade" or Direct Numerical Simulation (DNS) data, crucial for code
849 validation. Secondly, for decision-makers and project managers, it entails access to impartial,
850 non-commercial information concerning CFD's capabilities, potential benefits, and inherent
851 limitations, enabling informed decision-making processes. The CFD community involved in
852 nuclear safety studies should continue pushing in this direction, joining efforts together to
853 spread both their needs from other researchers and the capabilities of current CFD codes.

854 The human factor aspect of CFD implementation in the nuclear safety framework presents one
855 of the most complex challenges. The constitution of a proficient "CFD team" that possesses
856 expertise in both CFD methodologies and deep knowledge of nuclear safety within a research
857 team or a company is itself a challenging task. Moreover, fostering efficient collaboration
858 between this CFD team and other teams engaged in different aspects of physics and thermal
859 hydraulics across varying scales poses an additional challenge. In accordance with numerous
860 benchmarking activities aimed at CFD model validation, it becomes evident that discrepancies
861 not only persist among results obtained from different computational codes but also in the CFD
862 user decisions (Rein et al., 2009). Consequently, there exists a need to establish standardized
863 methodologies that systematically demonstrate their consistency. Within the field of CFD,
864 extensive efforts have been undertaken by the scientific community to formulate best-practice
865 guidelines, with the primary objective of empowering researchers and engineers with a well-
866 established framework to mitigate the aforementioned issues (OECD/NEA/CSNI, Mahaffy J.
867 et al., 2015). However, it is necessary to acknowledge that these actions have not provided a
868 clear solution, primarily due to the inherent variability of scenarios encountered. Additionally,
869 in fields like fire safety, the different fire related phenomena can impose different length and
870 time scales for the computational model resolutions, being the extensive range and
871 incompatibility of scales is an unavoidable complexity of explicit fire modelling (Torero et al.,
872 2013). Although fire models like FDS are widely validated, there is still the necessity to
873 validate it for specific purposed in order to improve the predictions.

874 Regulatory authorities, aware of the role played by CFD in the domain of nuclear safety studies,
875 have shown keen interest in its applications while concurrently expressing concerns regarding
876 the reliability and trustworthiness of the results generated through this methodology. During
877 the recent *9th Computational Fluid Dynamics for Nuclear Reactor Safety* conference hosted in
878 2023 by Texas A&M University in College Station, the United States NRC has raised its
879 pursuit of a "minimum standards" approach tailored specifically for single-phase flows. This
880 initiative aims to facilitate the acceptance and integration of CFD-based validation studies
881 within the regulatory framework for nuclear applications. The Spanish nuclear regulatory
882 authority, CSN, has also shown interest in this initiative and further engagements are expected
883 to happen in the next years. CSN has recently received topical reports for licensing
884 methodologies in which some of the thermo-hydraulics issues are supported with CFD
885 simulations. The need to establish this kind of framework to evaluate such studies is there. In
886 the meantime, a case-by-case approach should be used from the regulatory perspective.

887 Cost considerations are also another main challenge in the utilization of CFD for NS studies.
888 The computational cost associated with CFD calculations remains a substantial limiting factor,
889 although advancements in computing power, continue to mitigate this challenge. Furthermore,
890 the cost associated with geometry design and mesh generation prior to initiating CFD

simulations also needs to be taken into account. This cost can vary depending on the chosen meshing technique and can be especially expensive for very highly refined meshes. While automatic meshing tools offer rapid solutions, the construction of fully conformal hexahedral meshes, in some instances, remains one of the most time-consuming phases of a study. The CFD research community is actively engaged in advancing techniques adapted for high-performance computing systems. However, experts in nuclear safety CFD must need to make deliberate efforts to integrate these promising new solutions into the framework outlined within this paper. A noteworthy example of such innovation is the ALYA code developed by the BSC for the MareNostrum Supercomputer. The ALYA code's team is developing a high-performance computational tool explicitly designed for massively parallel supercomputing architectures. Exploring potential collaborations between the nuclear safety CFD community and the BSC holds the promise of substantial enhancements in simulation performance.

In broad terms, to address the various challenges posed to CFD codes, it is expected that the scientific communities in Spain and Portugal will explore possible approaches. Promoting knowledge sharing, exemplified by the continuous and commendable efforts collaborating with working groups such as the WGAMA CFD task group and international workshops like CFD4NRS, constitutes an effective strategy for advancing the application of CFD in the context of NS analyses. An stride towards enhancing the reliability and consistency of CFD models within the NS field involves the promotion of benchmarking exercises. In Spain and Portugal, the collaboration in these tasks will be continued and nuclear engineering groups are encouraged to join the activities.

Another notable avenue for progress lies in the establishment of databanks covering safety-related configurations. Such repositories would facilitate the identification and accessibility of both experimental data and numerical solutions. Some research groups in Europe, such as the Thermo-Fluid Dynamics and System Analysis' group at the Institute of Energy and Climate Research at Forschungszentrum Jülich GmbH in Germany is fostering this activity through international associations (Kelm, et al., 2021). The UPV has committed to support the initiative, but a bigger effort needs to be made by the community. In addition to the potential benefits of such database, there is potential value in exploring the adaptability of existing ones, originally constructed for the development and validation of system-scale analyses, to meet the specific requirements of CFD applications in the nuclear safety domain.

The ongoing efforts to enhance and commercialize novel SMR designs, including III and IV generation reactors, can find a formidable ally in the CFD. The versatile potential of CFD simulations can play a key role in advancing these unique SMR designs, offering a robust tool for their optimization and validation. Examples of the use of CFD in the context of water-cooled SMR designs have been documented in the existing body of literature, with a notable expansion in such studies in recent years. These investigations have even explored highly detailed resolutions, simulating the entire geometry of the reactor for steady-state simulations, as exemplified by the work of Böttcher et al. (2023). In the domain of liquid metal-cooled SMRs, significant advances were established by Roelofs et al. (2019), laying a robust foundation for CFD studies. It is noteworthy, however, that current CFD research in this domain predominantly affects single-phase flows, despite the CFD potential for two-phase flow investigations in the context of liquid metal applications. Furthermore, the application of CFD with alternative coolants, distinct from water, is currently at a relatively early stage of development and lacks comprehensive studies including both CFD-grade experimental measurements and CFD validation. Regrettably, the research status of future SMRs technology within Spain and Portugal remains somewhat limited. Nonetheless, a promising trajectory is

emerging, as collaborative initiatives with other European organizations in this sphere begin to take shape. For instance, promising collaborative efforts with PANDA facility and UPM are currently under development, setting the stage for forthcoming CFD investigations in the context of system testing for SMRs, as detailed in the work of Paladino et al. (2023).

4.2. Future lines in Spain and Portugal

This last section includes the different future activities expected for the following years within the organization in Spain and Portugal researching on CFD for NS. The main areas of future research for **UPCT** are focus on the application of turbulent combustion models to H_2-CO mixture employing RANS approaches. Reduced-order H_2-CO will be validated through reference databases, including experiments with flame acceleration. As a last step, the future research lines will focus on the application this type of models to PWR containment scenarios (OPENHYCOMB project with the CSN).

Within the following years, **UC** and CSN will jointly collaborate in the development of specific CFD models to improve prediction of various fire phenomenon that have not been completely validated yet as cable trays fire propagation, or combustion under vitiated atmospheres, and will increase the validation of the existing CFD models by the comparison to the large-scale experimental tests to be performed in the UC installations, and in international programs like Fire Risk Assessment through Innovative Research (FAIR) Project. Additionally, artificial intelligence-based methods will be developed to reduce CFD computational cost, allowing real time simulations to be used in emergency management at nuclear facilities.

The main effort that **ENUSA** expect to do in the coming years on CFD for NS is to continue working on the thermal characterization of the different container Spent Nuclear Containers designs that will be used in the Spanish ISFSI, applying the previously acquired knowledge regarding models and detailed geometry. Additionally, ENUSA will seek opportunities to perform thermal and CFD calculations in SMRs

At **UPM**, the 3D GOTHIC model development methodologies have proven to be ready for technical-scale applications. The priority in future years will be to use these consolidated methodologies to produce an extensive database of containment sequences simulated with 3D resolution. The detailed implementation of the most relevant containment safety systems for Spanish nuclear power plants is being consolidated in ongoing projects: fan coolers (INTERCON3D Project with the CSN), spray safety system (GOMERES-2 Project with the CSN), and PARs (GOMERES-2 and EC-HORIZON AMHYCO Project). The 3D GOTHIC Models will be used to continue investigating design basis accident and in-vessel severe accident sequences, and also to advance on the state-of-the-art by producing the first 3D investigations of the ex-vessel phase of severe accidents. The UPM is expected to come with a methodology to study SMRs containments during transients using GOTHIC and commercial CFD software in the coming years. Additionally, more research on the different types of VVER reactor containments will continue in pair with their future build up. Finally, the use of GOTHIC and CFD to assess micro-reactors safety will be a part of the future studies.

UPV will continue developing methods for efficient UQ in CFD, specifically tackling the extension of PCE for all types of uncertain variables and the development of reduced polynomials order techniques such as sparse-grids. The coupling between CFD with FEA codes will also be explored for certain relevant scenarios in nuclear safety such as pressure thermal shock. Finally, as part of the future portfolio, comparatives between CFD commercial

982 codes and new open sources codes with HPC capabilities will be done in collaboration with
983 the BSC.

984 **CSN**, Spanish regulator, will continue supporting and collaborating with the Spanish
985 organizations in the different activities for CFD within the NS framework. The main goal of
986 CSN is applying the lessons learned, for instance throughout WGAMA benchmark
987 participations or projects with UC, UPCT, UPM and UPV, to the evaluation of some utilities
988 submittals about licensing methods modifications that make use of CFD calculations as
989 hypothesis justification within methodologies.

5. Conclusions

991 The use of CFD models has become increasingly important in the field of nuclear reactor safety
992 studies. This shift is primarily attributed to the rise in computational capacity together with
993 enhancements in experimental methodologies, accumulated experience, and the refinement of
994 complementary tools, including meshing and post-processing. In this context, CFD is well-
995 suited to analyze various safety-critical aspects of nuclear reactors, such as cooling systems,
996 combustion processes, and containment structures, which inherently possess three-dimensional
997 characteristics. Traditional one-dimensional codes often require conservative approximations
998 to account for these complexities. In contrast, 3D CFD approaches provide the precision and
999 spatial resolution needed for accurate predictions across a wide range of scenarios in nuclear
1000 safety. The global efforts dedicated to verifying and validating these models are crucial, as they
1001 ensure that CFD tools can consistently deliver reliable and, when necessary, conservative
1002 results for specific flow configurations. In Spain, significant advancements have been made in
1003 the field of CFD for nuclear safety research, particularly in the context of code development
1004 and validation, combustion studies, nuclear spent-cask simulations, fuel assembly analyses,
1005 and containment investigations.

1006 Regarding CFD code development, UPCT have done research, collaborating with international
1007 partners to refine numerical CFD codes. In the domain of combustion and fire safety research,
1008 UPCT and the UC have actively cooperated with the Spanish regulator, CSN, to apply CFD
1009 fire codes in nuclear power plants. Their projects include various subareas such as fire
1010 modeling methodologies, software development for fire scenario creation, and validation of
1011 CFD codes through large-scale experimental tests. Furthermore, ENUSA has dedicated
1012 significant resources to researching nuclear spent casks, focusing on dry storage systems.
1013 Through CFD modelling, they aim to accurately calculate, among other parameters, the peak
1014 cladding temperature during dry storage, a critical parameter for spent fuel management.
1015 Collaborations with research institutions like UPM have resulted in validated methodologies
1016 for thermal calculations of spent nuclear fuel containers. In terms of fuel assembly studies,
1017 ENUSA has participated in benchmark exercises to assess CFD predictions of temperature,
1018 velocity, and pressure loss over fuel rod bundles. These studies involve the analysis of
1019 hydraulic forces, fuel assembly distortion, pressure drop over grids, and fuel assembly
1020 integrity, contributing to the understanding and improvement of fuel assembly performance
1021 and safety. UPM has done extensive research in containment analysis using CFD-like tools,
1022 particularly GOTHIC. Projects span different topics, such as hydrogen behavior during severe
1023 accidents and combustion risk assessment in filtered containment venting systems. These
1024 initiatives have led to the development of 3D containment models, validation through
1025 experimental data, and ongoing collaborations under European projects like AMHYCO. UPV
1026 has focus on benchmarking activities within the WGAMA CFD Task Group tackling a wide

variety of topics for nuclear safety such as grid spacers, buoyancy effects, uncertainty quantification stratification and fluid-structure interaction.

Several significant challenges were identified in the incorporation of CFD methodologies into formal nuclear safety studies. One key challenge is the lack of universally accepted methodologies, especially when coupling CFD with other physics and thermal hydraulics at different scales. Addressing this issue requires a common strategy adopted by the community to enhance the capabilities of CFD. Uncertainty quantification also poses challenges, as it demands solid methodologies proven for different CFD studies but involves substantial computational demands and costs. Access to databases with experimental results and impartial information about CFD capabilities is essential for researchers and decision-makers. As a result of the CFD features, regulatory authorities are showing keen interest in CFD applications but also express concerns about the reliability of results. The addition of CFD calculation for licensing purposes would be a cornerstone for CFD's continued development. To address all these challenges, the scientific community in Spain and Portugal will prioritize knowledge sharing and collaboration between both research institutions and regulators. Cooperative efforts with working groups and participation in benchmarking exercises will help advance CFD applications in nuclear safety studies and improve the reliability and consistency of CFD models.

Acknowledgments

References

[1] Alonso, A., Lázaro, M., Lázaro, P., Lázaro, D. & Alvear, D. (2020). Assessing the influence of the input variables employed by fire dynamics simulator (FDS) software to model numerically solid-phase pyrolysis of cardboard. J Therm Anal Calorim, 140, 263–273. doi:10.1007/s10973-019-08804-6

[2] Alonso, A., Lázaro, D., Lázaro, M., & Alvear, D. (2022). Self-heating evaluation on thermal analysis of polymethyl methacrylate (PMMA) and linear low-density polyethylene (LLDPE). J Therm Anal Calorim, 147, 10067–10081. doi:10.1007/s10973-022-11364-x

[3] Alonso, A., Lázaro, D., Lázaro, M., & Alvear, D. (2023). Numerical Prediction of Cables Fire Behaviour Using Non-Metallic Components in Cone Calorimeter. Combustion Science and Technology, 195(7), 1509-1525. doi:10.1080/00102202.2023.2182198

[4] Alonso, A., Lázaro, M., Lázaro, P., Lázaro, D., & Alvear, D. (2019). LLDPE kinetic properties estimation combining thermogravimetry and differential scanning calorimetry as optimization targets. J Therm Anal Calorim, 138, 2703–2713. doi:10.1007/s10973-019-08199-4

1071   [5]   Andreani, M., A. Badillo, R. Kapulla, and B. Smith. "Synthesis of the OECD/NEA-PSI CFD Benchmark
1072   Exercise." International Topical Meeting on Nuclear Reactor Thermal Hydraulics 2015, NURETH 2015
1073   3 (2015): 1787–1801.

1074   [6]   Arfinengo-del-Carpio, S., Vázquez-Rodríguez, C., Jiménez, G., 2021. Preventive methodology:
1075   geometric simplifications a priori for containment models optimization with the GOTHIC code, in:
1076   Proceedings of the European Nuclear Young Generation Forum ENYGF'21. Tarragona, Spain.

1077   [7]   Barrera J.E., Hernández J., Riverola J., "Simulación de la transferencia de calor de un contenedor de
1078   almacenamiento de combustible irradiado con el código CFD STAR-CCM+", 38º Annual Meeting of the
1079   SNE, Cáceres, 2012

1080   [8]   Bascou S., Suard S., and Audouin L., "Benchmark Activity of the OECD/NEA PRISME3 and FIRE
1081   projects," in 25th International Conference on Structural Mechanics in Reactor Technology (SMiRT 25),
1082   16th International Post-Conference Seminar on "FIRE SAFETY IN NUCLEAR POWER PLANTS AND
1083   INSTALLATIONS", SMIRT, OTTAWA, Canada, 2019.

1084   [9]   Benavides J., Jimenez G., Galbán M., Lloret M., "Estudio de los mecanismos de refrigeración del exterior
1085   del contenedor de combustible gastado TN-24P con STAR-CCM+", 44º Annual Meeting of the SNE,
1086   Ávila, 2018

1087   [10]   Benavides J., Jimenez G., Galbán M., Lloret M., "Methodology for thermal analysis of spent nuclear fuel
1088   dry cask using CFD codes", Annals of Nuclear Energy, vol 133 (2019), pp. 257-274, may. 2019.

1089   [11]   Benavides J., Jimenez G., Galbán M., Lloret M., "Modeling and results of the dry cask simulator (DCS)
1090   with STAR-CCM+", Nuclear Engineering and Design, vol 369 (2020), pp. 110820, august. 2020.

1091   [12]   Bentaib, A., Chaumeix, N., Nyrenstedt, G., Bleyer, A., Maas, L., Gastaldo, L., Kljenak, I., Dovizio, D.,
1092   Kudriakov, S., Schramm, B., Motegi, K., Velasco, F.J.S., Otón-Martínez, R.A., Danilin, A., Gavrikov,
1093   A., Jaseliunaite, J., Povilaitis, M. (2022). "ETSON-SAMHYCO-NET Benchmark on Simulations of
1094   Upward Flame Propagation Experiments in representative Hydrogen-Air-Steam mixtures of severe
1095   accidents containments atmosphere. The 19th International Topical Meeting on Nuclear Reactor Thermal
1096   Hydraulics (NURETH-19) Log nr.: 35766, 2022.

1097   [13]   Berglund, M., Fedina, E., Tegnér, J., Fureby, C., & Sabelnikov,V. (2010). Finite rate chemistry LES of
1098   self-ignition in asupersonic combustion ramjet. AIAA Journal,48(3), 540.https://doi.org/10.2514/1.43746

1099   [14]   Bestion, D. et al., "Review of Uncertainty Methods for Computational Fluid Dynamics Application to
1100   Nuclear Reactor Thermal Hydraulics". OECD, NEA/CSNI/R(2016)4, Feb 2016. [Online]. Available:
1101   https://www.oecd-nea.org/jcms/pl_19700/review-of-uncertainty-methods-for-computational-fluid-
1102   dynamics-application-to-nuclear-reactor-thermal-hydraulics?details=true

1103   [15]   Bocanegra, R., Jimenez, G., 2018. Proposal of a BEPU Methodology for Containment Safety Analysis
1104   (BEPU2018-165), in: ANS Best Estimate Plus Uncertainty International Conference (BEPU 2018).
1105   Lucca, Italy.

1106   [16]   Bocanegra, R., Jimenez, G., Fernández-Cosials, M.K., 2016. Development of a PWR-W GOTHIC 3D
1107   model for containment accident analysis. Ann. Nucl. Energy 87, 547–560.
1108   https://doi.org/10.1016/j.anucene.2015.10.022

1109   [17]   Colin, O., Ducros, F., Veynante, D., & Poinsot, T. (2000). A thickened flame model for Large Eddy
1110   Simulations of turbulent premixed combustion. Physics of Fluids, 12(7), pp.1843–1863.
1111   https://doi.org/10.1063/1.870436

1112   [18]   Corpa Masa R., Jiménez G., Montero B., "Cálculo de fuerzas laterales hidráulicas en elementos
1113   combustibles tipo PWR con códigos de dinámica de fluidos computacional", 41º Annual Meeting of the
1114   SNE A Coruña, 2015.

1115   [19]   Corpa Masa R., Montero B., "Fuel assembly drop accident simulation with CFD" International Youth
1116   Nuclear Congress Burgos, 2014.

1117   [20]   Díez Álvarez-Buylla, P., Estévez-Albuja, S., Jiménez, G., Gavilán, C., 2021. Analysis of venting
1118   strategies and hydrogen concentration evolution during a station blackout in a BWR-6 containment using
1119   GOTHIC 8.3. Prog. Nucl. Energy 141, 103930. https://doi.org/10.1016/j.pnucene.2021.103930

1120   [21]   Domínguez-Bugarín, A., Jiménez, M.-Á., Reinecke, E.-A., Jiménez, G., 2022. PARUPM: A simulation
1121   code for passive auto-catalytic recombiners. EPJ Nucl. Sci. Technol. 8, 32.
1122   https://doi.org/10.1051/epjn/2022046

[22] EPRI 2014, "Computational Fluid Dynamics Benchmark of High Fidelity Rod Bundle Experiments: Industry round Robin Phase 1 – Rod Bundle with Simple Support Grids", EPRI, Palo Alto, CA: 2014. 3002000504.

[23] Esteban I., Matías R., "Análisis térmico mediante CFD del combustible nuclear gastado en contenedores", 47° Annual Meeting of the SNE, Cartagena 2022

[24] Estévez-Albuja, S., Fernández-Cosials, K., Vázquez-Rodríguez, C., Goñi-Velilla, Z., Jiménez, G., 2021a. AP1000 Passive Containment Cooling System study under LBLOCA conditions using the GOTHIC code. Nucl. Eng. Des. 384, 111442. https://doi.org/10.1016/j.nucengdes.2021.111442

[25] Estévez-Albuja, S., Jimenez, G., Al Issa, S., Macián-Juan, R., Fernández-Cosials, K., Queral, C., 2018. Steam condensation simulation in a scaled IRWST-ADS simulator with GOTHIC 8.1. Nucl. Eng. Des. 334, 96–109. https://doi.org/10.1016/j.nucengdes.2018.04.020

[26] Estévez-Albuja, S., Jiménez, G., Vázquez-Rodríguez, C., 2021b. AP1000 IRWST numerical analysis with GOTHIC. Nucl. Eng. Des. 372, 110991. https://doi.org/10.1016/j.nucengdes.2020.110991

[27] Feria F., et al., 2023, "Spanish R&D in dry storage of spent fuel: outcomes and outlook", Nuclear Engineering and Design. Preprint subjected to review.

[28] Fernández-Cosials, K., Estévez-Albuja, S., Jiménez, G., Bocanegra, R., Vázquez-Rodríguez, C., Rey, L., Martínez-Murillo, J.C., 2019. 3D containment modeling of PWR-KWU Trillo NPP with the GOTHIC code. Ann. Nucl. Energy 133, 387–399.

[29] Fernández-Cosials, K., Jimenez, G., Bocanegra, R., Queral, C., 2017. Study of hydrogen risk in a PWR-W containment during a SBO scenario; Tau parameter definition and application on venting strategy analysis. Nucl. Eng. Des. 325, 164–177. https://doi.org/10.1016/j.nucengdes.2017.10.012

[30] Fernández-Cosials, K., Jiménez, G., Serrano, C., Ibáñez, L., Peinado, Á., 2018. Analysis of Inertization Strategies for the Filtered Containment Venting System in Cofrentes Nuclear Power Plant. J. Nucl. Eng. Radiat. Sci. 4, 031016. https://doi.org/10.1115/1.4038595

[31] Fernández-Cosials, M.K., Jimenez, G., Lopez-Alonso, E., 2016. Analysis of a gas stratification break-up by a vertical jet using the GOTHIC code. Nucl. Eng. Des. 297, 123–135. https://doi.org/10.1016/j.nucengdes.2015.11.035

[32] Garcia-Cascales, José R., & Henri Paillère. "Application of AUSM schemes to multi-dimensional two-phase flow problems". In Proceedings of the ASME/JSME Joint Fluids Engineering Conference, 1 B:1445-50. American Society of Mechanical Engineers, 2003. https://doi.org/10.1115/FEDSM2003-45020

[33] García-Cascales, Mulas-Pérez, & Paillère. "Advances in the characterization of transient two-phase flow". AIAA Journal 45, n.° 10 (2007): 2579-84. https://doi.org/10.2514/1.25644

[34] García-Cascales, Otón-Martínez, Velasco, Vera-García, Bentaib, y Meynet. "Advances in the characterisation of reactive gas and solid mixtures under low pressure conditions". Computers and Fluids 101 (2014): 64-87. https://doi.org/10.1016/J.COMPFLUID.2014.05.017

[35] García-Cascales, José R., Velasco, F.J.S., Otón-Martínez, Ramón A., Espín-Tolosa, S., Bentaib, A., Meynet, N., Bleyer, A., 2015. "Characterisation of metal combustion with DUST code", Fusion Engineering and Design, Volumes 98–99, Pages 2142-2146, ISSN 0920-3796, https://doi.org/10.1016/j.fusengdes.2015.03.012.

[36] Harvill, R.C., Lane, J.W., George, T.L., 2022. Hybrid System Level and Coarse Grid CFD Tool for Three-Dimensional Natural Circulation, Mixing, and Stratification Modeling. Nuclear Technology 208, 1–26. https://doi.org/10.1080/00295450.2020.1870371

[37] Herranz, L.E., Jiménez, G., Nitti, F.S., 2022. Towards an optimized management of accidents. EPJ Nucl. Sci. Technol. 8, 43. https://doi.org/10.1051/epjn/2022019

[38] IAEA Nuclear Energy Series "Summary Review on the Application of Computational Fluid Dynamics in Nuclear Power Plant Design", International Atomic Energy Agency No. NR-T-1.20, 2022. Publicly available at: https://www-pub.iaea.org/MTCD/Publications/PDF/PUB1932_web.pdf

[39] IAEA-TECDOC-1379 "Use of computational fluid dynamics codes for safety analysis of nuclear reactor systems", summary report of technical meeting jointly organized by the IAEA and the NEA of the OECD, 2002. Publicly available at: https://www-pub.iaea.org/MTCD/Publications/PDF/te_1379_web.pdf

1174 [40] IRSN, Contract: Modélisation de la mobilisation et de l'explosion de poussières (DA
1175   3000011012/CA32000911 IRSN SIRET: 440 546 018 00019 Code APE: 7219 Z)

1176 [41] Jimenez, G., Fernández-Cosials, M.K., Bocanegra, R., Queral, C., 2017. Analysis of the equipment and
1177   instrumentation qualification criteria using 3D containment models. Nucl. Eng. Des. 323, 28–38.
1178   https://doi.org/10.1016/j.nucengdes.2017.07.038

1179 [42] Jiménez, G., Herranz, L.E., Bentaib, A., Chaumeix, N., Reinecke, E.-A., Kelm, S., Loeffler, M., Braun,
1180   M., Bratfisch, C., Kljenak, I., Sevbo, O., 2022. AMHYCO PROJECT – Towards An Enhanced Accident
1181   Management Of The H2/CO Combustion Risk. Presented at the The 19th International Topical Meeting
1182   on Nuclear Reactor Thermal Hydraulics (NURETH-19), Brussels, Belgium.

1183 [43] Jimenez, G., Serrano, C., Lopez-Alonso, E., Molina, M. del C., Calvo, D., García, J., Queral, C., Zuriaga,
1184   J.V., González, M., 2015. BWR Mark III containment analyses using a GOTHIC 8.0 3D model. Ann.
1185   Nucl. Energy 85, 687–703. https://doi.org/10.1016/j.anucene.2015.06.025

1186 [44] Jimenez, M.A., Martin-Valdepeñas, J.M., Martin-Fuertes, F., Fernandez, J.A., Jiménez, M.A., Martín-
1187   Valdepeñas, J.M., Martín-Fuertes, F., Fernández, J.A., 2007. A detailed chemistry model for transient
1188   hydrogen and carbon monoxide catalytic recombination on parallel flat Pt surfaces implemented in an
1189   integral code. Nucl. Eng. Des. 237, 460–472. https://doi.org/10.1016/j.nucengdes.2006.08.002

1190 [45] Kanik, M.E., Noori-kalkhoran, O., Fernández-Cosials, K., Gei, M., 2022. "Full scope 3D analysis of a
1191   VVER-1000 containment pressurization during a LB-LOCA by employing AutoCAD and GOTHIC
1192   code", Progress in Nuclear Energy, Volume 152, 104376. https://doi.org/10.1016/j.pnucene.2022.104376

1193 [46] Kanik, M.E., Noori-kalkhoran, O., Gei, M., Fernández-Cosials, K., 2024. "3D analysis of spray effect on
1194   long-term depressurization of VVER-1000 containment during LB-LOCA", Progress in Nuclear Energy,
1195   Volume 170, 105127, https://doi.org/10.1016/j.pnucene.2024.105127

1196 [47] Kelm, S., Kampili, M., Liu, X., George, A., Schumacher, D., Druska, C., Struth, S., Kuhr, A., Ramacher,
1197   L., Allelein, H.-J., Prakash, K.A., Kumar, G.V., Cammiade, L.M.F., Ji, R., 2021. The Tailored CFD
1198   Package 'containmentFOAM' for Analysis of Containment Atmosphere Mixing, H2/CO Mitigation and
1199   Aerosol Transport. Fluids 6, 100. https://doi.org/10.3390/fluids6030100

1200 [48] Lane, J.W., Link, J.M., King, J.M., George, T.L., Claybrook, S.W., 2020. Benchmark of GOTHIC to
1201   EBR-II   SHRT-17   and   SHRT-45R   Tests.   Nuclear   Technology   206,   1019–1035.
1202   https://doi.org/10.1080/00295450.2019.1698896

1203 [49] Lázaro D., Lázaro M., Alvear D. and Jiménez M., "Methodology to estimate the fds chemical reaction
1204   equation in fire scenarios," in Actas del 6º CILASCI - Congreso Ibero-Latino-Americano en Seguridad
1205   Contra Incendio, Belo Horizonte - Brasil, 2022.

1206 [50] Lázaro, D., Alonso, A., Lázaro, M., & Alvear, D. (2021). A Simple Direct Method to Obtain Kinetic
1207   Parameters for Polymer Thermal Decomposition. Appl. Sci., 11. doi:10.3390/app112311300

1208 [51] Lázaro, D., Alonso, A., Lázaro, M., Jiménez, M., & Alvear, D. (2023). Experimental and analytical study
1209   of the influence of the incident heat flux in cables heat release. J Therm Anal Calorim.
1210   doi:10.1007/s10973-023-12139-8

1211 [52] Lázaro, D., Alvear, D., Lázaro, M., & Jiménez, M. Á. (16-19 noviembre 2020). Análisis de simulaciones
1212   numéricas de la probabilidad de fallo por incendios en cables eléctricos. Reunión Virtual de la SNE.

1213 [53] Lázaro, D., Lázaro, M., & Jiménez, M. (4 – 6 octubre 2023). Predicción de las condiciones de
1214   sostenibilidad durante un incendio en centrales nucleares mediante el uso de redes neuronales. 48ª
1215   Reunión Anual de la Sociedad Nuclear Española. Toledo, Spain.

1216 [54] Lázaro, D., Lázaro, M., Alonso, A., Lázaro, P., & Alvear, D. (2019). Influence of the STA boundary
1217   conditions on thermal decomposition of thermoplastic polymers. J Therm Anal Calorim, 138, 2457–2468.
1218   doi:10.1007/s10973-019-08787-4

1219 [55] Lázaro, D. Lázaro, , M., Peco, J. & Alvear, D. "Influencia del mallado en el modelado computacional de
1220   incendios en centrales nucleares," Informes De La Construcción, vol. 70, no. 549, 2018.

1221 [56] Lázaro, D., Lázaro, M., Jiménez, M. & Alvear, D. "New approach to cable tray fire modelling through a
1222   FLASH-CAT modification," in Proceedings of the Tenth International Seminar on Fire and Explosion
1223   Hazards (ISFE10), Oslo, Norway, 22-27 May 2022.

1224   [57]   Martín-Valdepeñas, J., Jimenez, M.A., Martín-Fuertes, F., Arribas, M., Hernández, E., Pontejo, A., 1999.
1225   Safety Analyses Concerning Hydrogen Phenomena within the Containment Building during Severe
1226   Accidents. Computational Approaches. Presented at the International Conference on Mathematics and
1227   Computation, Madrid, p. 13.

1228   [58]   Martín-Valdepeñas, J., Jiménez, MA, M.A., 2003. CFD analyses of hydrogen risk within PWR
1229   containments, in: IAEA-TECDOC-1379, TECDOC Series. Presented at the Use of computational fluid
1230   dynamics codes for safety analysis of nuclear reactor systems, International Atomic Energy Agency,
1231   Vienna.

1232   [59]   Martín-Valdepeñas, J.M., Jiménez, M.A., Martín-Fuertes, F., Benítez, J.A.Fernández., 2005. Comparison
1233   of film condensation models in presence of non-condensable gases implemented in a CFD Code. Heat
1234   Mass Transf. 41, 961–976. https://doi.org/10.1007/s00231-004-0606-5

1235   [60]   Martín-Valdepeñas, J.M., Jiménez, M.A., Martín-Fuertes, F., Fernández, J.A., 2007. Improvements in a
1236   CFD code for analysis of hydrogen behaviour within containments. Nucl. Eng. Des. 237, 627–647.
1237   https://doi.org/10.1016/j.nucengdes.2006.09.002

1238   [61]   Matías R., Cantón R., Jiménez G., "PWR fuel rod vibration simulation analysis for estimating grid-to-
1239   rod-fretting (GTRF)", Topfuel Prague, 2019.

1240   [62]   McGrattan K., Hostikka, S., Floyd J., McDermott R. and Vanella M., NIST Special Publication 1018-3
1241   Sixth Edition Fire Dynamics Simulator Technical Reference Guide Volume 3: Validation," VTT, NIST,
1242   2023

1243   [63]   NFPA 805 Performance-Based Standard for Fire Protection for Light Water Reactor Electric Generating
1244   Plants 2020.

1245   [64]   Nicolás-Pérez, F.; Velasco, F.; Otón-Martínez, R.A.; García-Cascales, J.R.; Bentaib, A.; Chaumeix, N.
1246   Capabilities and limitations of Large Eddy Simulation with perfectly stirred reactor assumption for
1247   engineering applications of unsteady, hydrogen combustion sequences. Engineering Applications of
1248   Computational Fluid Mechanics 2021, Vol. 15, No. 1, 1452–1472. DOI:
1249   10.1080/19942060.2021.1974092

1250   [65]   Nicolás-Pérez, F.; Velasco, F.; Otón-Martínez, R.A.; García-Cascales, J.R.; Bentaib, A.; Chaumeix, N.
1251   Mathematical Modelling of Turbulent Combustion of Two-Phase Mixtures of Gas and Solid Particles
1252   with a Eulerian–Eulerian Approach: The Case of Hydrogen Combustion in the Presence of Graphite
1253   Particles. Mathematics 2021, 9, 2017. https:// doi.org/10.3390/math9172017

1254   [66]   NUREG 1934 Nuclear Power Plant Fire Modeling Analysis Guidelines Final Report, U.S. Nuclear
1255   Regulatory Commission, 2012.

1256   [67]   NUREG-1824 Supplement 1 Verification and Validation of Selected Fire Models for Nuclear Power Plant
1257   Applications – Final Report, 2016

1258   [68]   NUREG-1824, "Verification and Validation of Selected Fire Models for Nuclear Power Plant
1259   Applications Supplement 1", FINAL REPORT. U.S. NRC, EPRI," 2016.

1260   [69]   OECD/NEA/CSNI, B.L. Smith, J.H. Mahaffy, K. Angele, and J. Westin. "Report of the OECD/NEA-
1261   Vattenfall T-Junction Benchmark Exercise," 2011. www.oecd.org/publishing/corrigenda

1262   [70]   OECD/NEA/CSNI, Fokken, J., B. Krohn, R. Kapulla, B. Niceno, H.-M Prasser, and A. Badillo. "NEA
1263   Benchmark Exercise: Computational Fluid Dynamic Prediction and Uncertainty Quantification of a
1264   GEMIX Mixing Layer Test." Oecd/Nea/Csni, no. February (2019).

1265   [71]   OECD/NEA/CSNI, Rodolfo Vaghetto, Yassin Hassan, Satoshi Abe, Dominique Bestion, Romain Camy,
1266   Adrey Deulin, Fabien Duval, and Yago Rivera. "Cold Leg Mixing Computational Fluid Dynamics –
1267   Uncertainty Quantification (CLM-UQ) Benchmark Results." OECD/NEA/CSNI/R(2022)7, October
1268   (2022).

1269   [72]   OECD/NEA/CSNI, Smith, B.L., C.H. Song, S.K. Chang, J.R. Lee, and J.W. Kim. "Report of the
1270   OECD/NEA KAERI Rod Bundle CFD Benchmark Exercise," 2013. https://www.oecd-
1271   nea.org/jcms/pl_19286.

1272   [73]   OECD/NEA/CSNI, "Investigating Heat and Smoke Propagation Mechanisms in Multi-Compartment Fire
1273   Scenarios". Final Report of the PRISME Project," OECD/NEA/CSNI/R(2017)14, 2018.

1274 [74]   OECD/NEA/CSNI, Mahaffy J. et al., "Best Practice Guidelines for the Use of CFD in Nuclear Reactor
1275   Safety Applications- Revision," OECD, NEA/CSNI/R(2014)11, Feb. 2015. [Online]. Available:
1276   https://www.oecd-nea.org/nsd/docs/2014/csni-r2014-11.pdf

1277 [75]   Paillère, Corre, & García Cascales. "On the extension of the AUSM+ scheme to compressible two-fluid
1278   models". Computers and Fluids 32, n.º 6 (2003): 891-916. DOI: 10.1016/S0045-7930(02)00021-X

1279 [76]   Petrarca A. J., Aleshin Y., Corpa Masa R., Gómez Palomino J. M., "Effect of Lateral Hydraulic Forces
1280   on Fuel Assembly Bow", Topfuel, Zurich, 2015.

1281 [77]   Plumecocq W., Bascou S., Rawekamp M., Hamburger K., "Common OECD/NEA FIRE and PRISME
1282   Cable Benchmark Exercise", Fire Safety Journal, Volume 139, 2023. DOI: 10.1016/j.firesaf.2023.103815

1283 [78]   Rein G., Torero J.L., Jahn W., Stern-Gottfried J., Ryder N.L., Desanghere S., Lázaro M., Mowrer F.,
1284   Coles A., Joyeux D., Alvear D., Capote J.A., Jowsey A., Abecassis-Empis C., Reszka P., "Round-robin
1285   study of a priori modelling predictions of the Dalmarnock Fire Test One", Fire Safety Journal, Volume
1286   44, Issue 4, 2009, Pages 590-602, https://doi.org/10.1016/j.firesaf.2008.12.008

1287 [79]   Rivera, Y., Muñoz-Cobo J. L., Berna C., Escrivá A., and Vela E., "CFD Simulation plus Uncertainty
1288   Quantification of the Mixing of Two Fluid with Different Density for the Cold-Leg Mixing Benchmark."
1289   Nuclear Engineering and Design 383, no. September (2021). DOI: 10.1016/j.nucengdes.2021.111449.

1290 [80]   Rivera Y., Berna C., Muñoz-Cobo J.L., Escrivá A., Córdova Y., Experiments in free falling and
1291   downward cocurrent annular flows – Characterization of liquid films and interfacial waves, Nuclear
1292   Engineering and Design, Volume 392, 2022, 111769, DOI: 10.1016/j.nucengdes.2022.111769.

1293 [81]   Roelofs F., "Thermal Hydraulics Aspects of Liquid Metal Cooled Nuclear Reactors". Elsevier, 2019.
1294   DOI: 10.1016/C2016-0-01216-0.

1295 [82]   Serra, L., Domínguez-Bugarín, A., Estévez-Albuja, S., Vázquez-Rodríguez, C., Jiménez, G., Kelm, S.,
1296   Herranz, L.E., 2021. Development of a detailed 3D CAD model of a generic PWR-KWU containment as
1297   a basis for a better assessment of H2/CO combustion risk, in: Proceedings of the European Nuclear Young
1298   Generation Forum ENYGF'21. Tarragona, Spain.

1299 [83]   Serrano, C., Jimenez, G., Molina, M. del C., López-Alonso, E., Justo, D., Zuriaga, J.V., González, M.,
1300   2016. Proposed methodology for Passive Autocatalytic Recombiner sizing and location for a BWR Mark-
1301   III   reactor   containment   building.   Ann.   Nucl.   Energy   94,   589–602.
1302   https://doi.org/10.1016/j.anucene.2016.03.022

1303 [84]   Smith, B.L. (PSI), Andreani, M. (PSI), Bieder, U. (CEA) et al., "Assessment of CFD for Nuclear Reactor
1304   Safety Problems". OCDE, NEA/CSNI/R(2014)12 Jan. 2015. [Online]. Available: https://www.oecd-
1305   nea.org/nsd/docs/2014/csni-r2014-12.pdf

1306 [85]   Smith, B.L. (PSI), Andreani, M. (PSI), Bieder, U. (CEA) et al., "Assessment of CFD for Nuclear Reactor
1307   Safety Problems". OCDE, NEA/CSNI/R(2014)12 Jan. 2015. [Online]. Available: https://www.oecd-
1308   nea.org/nsd/docs/2014/csni-r2014-12.pdf

1309 [86]   Staedtke H., Franchello G., Worth B., Graf U., Romstedt P., Kumbaro A., García-Cascales J., Paillère H.,
1310   Deconinck H., Ricchiuto M., Smith B., De Cachard F., Toro E.F., Romenski E., Mimouni S. "Advanced
1311   three-dimensional two-phase flow simulation tools for application to reactor safety (ASTAR)". Nuclear
1312   Engineering and Design, 235:379-400, 2005. https://doi.org/10.1016/J.NUCENGDES.2004.08.052

1313 [87]   Torero J.L., "Scaling-Up fire", Proceedings of the Combustion Institute, Volume 34, Issue 1, 2013, Pages
1314   99-124, https://doi.org/10.1016/j.proci.2012.09.007

1315 [88]   Vázquez-Rodríguez, C., Andreani, M., Jiménez, G., Kapulla, R., Paranjape, S., Manuel Martín-
1316   Valdepeñas, J., Paladino, D., 2023. Assessment of the depressurisation and mixing effects of two spray
1317   systems in the PANDA facility, part II – Simulations using GOTHIC8.3(QA). Nucl. Eng. Des. 403,
1318   112120. https://doi.org/10.1016/j.nucengdes.2022.112120

1319 [89]   Vázquez-Rodríguez, C., Bocanegra, R., Jimenez, G., Rey, L., Cadenas, A., Posada, J.M., Martínez-
1320   Murillo, J.C., 2019. "Toward Conservatism in Containment Design Basis Accident Analyses. Lumped
1321   parameters and 3D Approaches". Presented at the ICAPP 2019 – International Congress on Advances in
1322   Nuclear Power Plants, Juan-les-pins, France.

1323 [90]   Vázquez-Rodríguez, C., Jiménez, G., Andreani, M., Paladino, D., Paranjape, S., Kapulla, R., Martín-
1324   Valdepeñas, J.M., 2022. "Simulations of spray tests performed in PANDA thermal-hydraulics facility
1325   using GOTHIC8.3(QA)", in: NURETH-19.

1326 [91] Velasco, F.J.S., Otón-Martínez, R.A., García-Cascales, J.R., Espín, S., Meynet, N., Bentaib, A., 2016.
1327   "Modelling detonation of H2–O2–N2 mixtures in presence of solid particles in 3D scenarios",
1328   International Journal of Hydrogen Energy, Volume 41, Issue 38, Pages 17154-17168, ISSN 0360-3199,
1329   https://doi.org/10.1016/j.ijhydene.2016.06.096

1330 [92] Watson J.D. and Crick F. H. C., "A structure for deoxyribose nucleic acid," *Nature*, vol. 171, no. 4356,
1331   pp. 737–738, 1953.

1332 [93] Yetter, R.A., Dryer, F.L., Rabitz, H. A., "Comprehensive Reaction Mechanism For Carbon
1333   Monoxide/Hydrogen/Oxygen Kinetics". Combust. Sci. Technol. 1991, 79, 97–128.