Mecánica Computacional Vol. XXI, pp. 2903–2922
S. R. Idelsohn, V. E. Sonzogni and A. Cardona (Eds.)
Santa Fe–Paraná, Argentina, October 2002

COMPUTATIONAL FLUID DYNAMICS THROUGHOUT THE DESIGN PROCESS IN NUCLEAR APPLICATIONS

Gustavo C. Buscaglia*, Enzo A. Dari*, Daniela L. Arnica*, Axel E. Larreteguy†, and Claudio Mazufri**

*Centro Atómico Bariloche and Instituto Balseiro, 8400 Bariloche, Argentina
e-mail: gustavo@cab.cnea.gov.ar, web page: http://cabmec1.cnea.gov.ar/gustavo
†Universidad Argentina de la Empresa, Lima 717, 1073 Buenos Aires, Argentina
e-mail: alarreteguy@uade.edu.ar
**INVAP S.E., P. O. Box 961, 8400 Bariloche, Argentina. e-mail: mazufri@invap.com.ar

Key Words: Computational heat transfer, Nuclear reactors, Mixed convection.

Abstract. *Though Computational Fluid Dynamics (CFD) is recognized as an emerging engineering tool, it is not infrequent that engineering projects make little or no use of it during the early design stages. CFD codes, if used at all, are left for the final stages in which an already finished design is tested to confirm its performance before construction. Scaled mock-ups are also used at these stages, but in many cases they are economically unaffordable. In a nuclear reactor the operational radiation dose results from coupled processes of coolant flow and species radioactive decay. Thermal-hydraulic phenomena such as stratification, stagnation and inappropriate flow pattern should thus be considered from the early design stages. In this article we want to report on an innovative experience in this direction, namely the application of CFD since the first stages of the design of some components of the reactor to be built by INVAP S. E. in Lucas Height, Australia, for the Australian National Science and Technology Organization (ANSTO). The analysis of the aforementioned components leads, sooner or later, to the need for massive three-dimensional simulations which can only be performed with high performance computing techniques. This motivates a brief description of the software (PAR-GPFEP, presented in MECOM'2000) and of the hardware (Linux PC-cluster) used to successfully produce the simulations. The two case studies addressed in this presentation elaborate on the scientific, software-related and strategic aspects of CFD consulting work throughout the design process. Though the application is specific to the nuclear industry, the methodologies are applicable to many other areas of engineering design.*

1 INTRODUCTION

Though Computational Fluid Dynamics (CFD) is recognized as an emerging engineering tool, it is not unfrequent that engineering projects make little or no use of it during the design stages. Heuristic, experience-based approaches are the most popular design tools, complemented by very crude approximations such as input/output balance models. CFD codes, if used at all, are left for the final stages in which an already finished design is tested to confirm its performance before construction. Scaled mock-ups are also used at these stages of the project, but in many cases they are economically unaffordable.

In a nuclear reactor safety is the fundamental variable. The operational radiation dose in its different areas results from the coupled processes of coolant flow and species radioactive decay. Thermal-hydraulic phenomena such as stratification, stagnation and inappropriate flow pattern should thus be considered from the early design stages.

In this article we want to report on an innovative experience in this direction, namely the application of CFD since the first stages of the design of some components of the reactor to be built by INVAP S. E. in Lucas Height, Australia, for the Australian National Science and Technology Organization (ANSTO).

The Replacement Research Reactor Facility (RRRF) at the Lucas Heights Science and Technology Center will replace the currently operating 43-year-old High Flux Australian Reactor. The design, construction and commissioning of the reactor is carried out by INVAP S. E. and its Australian alliance partners, John Holland and Evans Deakin Industries. The Reactor Facility has been designed and will be constructed and operated to meet Australia's current and future needs for a neutron source in a manner that complies with all health, environmental and safety standards. Specifically, it is intended for the following purposes: a) To maintain Australia's technical expertise in nuclear science and technology; b) to maintain and enhance health-care benefits to the Australian community supplying diagnostic and therapeutic radio-pharmaceuticals; c) to provide a neutron beam research facility for national and regional industries and scientific institutions; d) to enhance the educational opportunities available to Australia's students, particularly in science and engineering; and e) to provide industrial radioisotopes and facilities for neutron activation analysis, irradiation of materials and neutron radiography to serve the needs of agriculture and industry.

The Reactor Facility (see Fig. 1) occupies 13,000 square meters, which include the Reactor Building, the Neutron Guide Hall, and associated auxiliary and office buildings. The reactor is of pool-type design with a rated thermal power from the core of 20 MW. It includes operational and safety characteristics consistent with international best practice. The reactor core is located near the bottom of the pool and is cooled by demineralized water. A heavy water reflector is contained in a cylindrical vessel surrounding the core.

Being a leading-edge technological project, RRRF involves several components which are unique, taylor-made for this specific reactor. Experience from INVAP S.E.'s past projects and available know-how from the nuclear industry applies to a great extent to

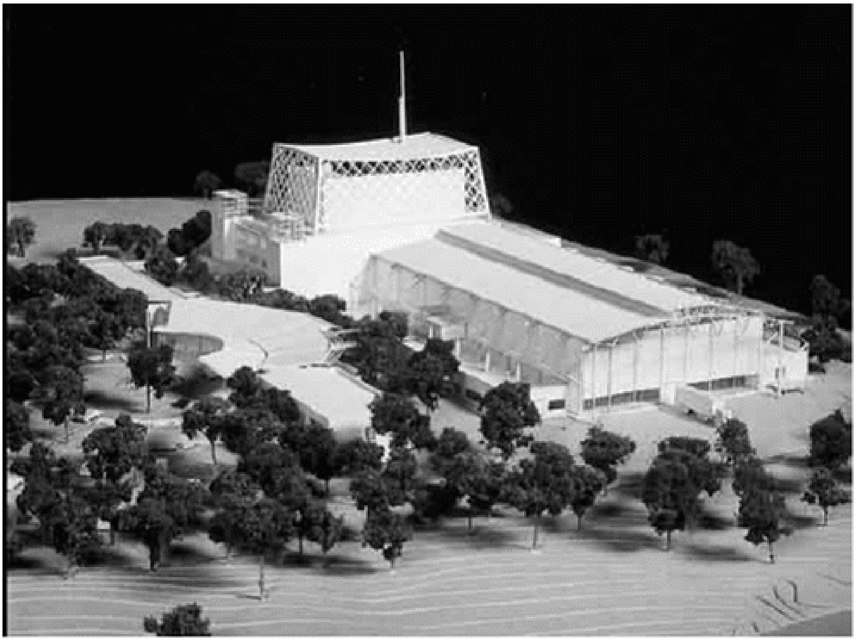

Figure 1: A view of the projected RRRF building.

these components, but in some cases a certain amount of research was needed to properly assess the design. In this article we refer in particular to thermal-hydraulic systems with significant buoyancy forces and complex geometries. Such systems are far from being found in heat-transfer handbooks, and being highly nonlinear in nature it is not evident how to build scaled experiments that capture the basic physics. Buoyancy-driven flow structures and instabilities are shown below to play a significant role in the performance of some of these components, and CFD is one appropriate tool for prediction.

The urging time schedule, the permanently evolving input data, and the continuously changing questions that are characteristic of the early stages of design are however not the normal working conditions for CFD analysts. The relation between INVAP S. E. and the MECOM group of the Centro Atómico Bariloche will be discussed to some extent in this respect. We address two case studies and elaborate on the scientific, software-related and strategic aspects of CFD consulting work as inserted in the basic design process.

It should be remarked, however, that though the simulations reported here consider a few safety issues they are not connected to the Safety Analysis of the reactor. After defining a design using different tools (handbooks, CFD, etc.) one must *prove* that it complies with all safety-related regulations and that the plant withstands severe accidental conditions without risk to the public. To prove this, much more conservative approaches are used than the ones used in the work reported here. We stress that the analyses discussed in this article have the purpose of assessing the performance of different systems under *normal* (not *accidental*) operating conditions.

2 METHODOLOGY

This section discusses three methodological aspects of the studies: The software, the ex-post heuristic analysis, and the approach used to cope with a continuously-evolving design typical of the early stages of a project.

2.1 The software

The CFD software used in this article is code FEMCO 4.1. This code's validation against several analytical solutions and experimental data have been reported elsewhere.[1–5] It is a CFD solver built on the flexible PAR-GPFEP system for generating high performance finite element software.[2,6]

The numerical method employed is based on a finite element formulation. Piecewise linear interpolation is used for all variables. Upwinding is added by means of the SUPG technique in both the momentum and energy equations (see, e.g., Codina[7] and references therein). Thermal coupling is accounted for in a staggered manner, solving for velocity and pressure with frozen temperature and viceversa. The equal-interpolation scheme used for velocity and pressure is stabilized by means of the Pressure Gradient Projection (also called Orthogonal Sub-Scales) method introduced by Codina and Blasco[8] and further discussed by Buscaglia, Basombrío and Codina[9] and Codina *et al.*[1] As turbulent model we adopt in most cases Smagorinsky's subgrid-scale eddy diffusivity concept,[10,11] which allows dynamically-evolving large-scale vortical structures to be modeled if the mesh is fine enough.[12]

FEMCO 4.1 is designed to run on MIMD distributed-computing architectures. In particular, the Linux Cluster of the Instituto Balseiro was used for this work. It currently consists of 22 Single-CPU 1 GHz Athlon nodes connected through Fast Ethernet switches. The Linux operating system is used in all the nodes, with the `mpich` package for message passing. As essential ingredient, PETSc (www.mcs.anl.gov/petsc) handles all distributed objects and parallel linear algebra subproblems. Graph partitioning is performed using METIS (www.cs.umn.edu/karypis), in particular its parallel version PARMETIS. Typical speedup attainable with FEMCO 4.1 is between 70 and 85 percent of the ideal value, depending on the problem and the number of processors used.

Concerning the mesh generation aspects of the work, several in-house codes were used for the different substeps: Surface abstraction, Surface triangulation, Volumetric mesh generation and Mesh optimization. Some aspects of these codes have been discussed in previous presentations[13–15]

2.2 Combining CFD with heuristic reasoning

Numerical models, of course, are not to be taken as the final answer to every problem. In some cases, the best we can expect from them is little more than a rough first approximation. In other cases, the results of our numerical model may well be very accurate, but even in those cases we may not know that they are so, simply because we may be unsure that our physical and mathematical model captures all the relevant phenomena.

In the examples discussed later in this work a crucial contribution of the numerical simulations was their ability to reveal fundamental complexities of the flows that were difficult to predict using simpler analyses. Fluids flowing in seemingly simple domains and constrained by seemingly simple boundary conditions are prone to very complex, fully 3D, and very unstable behaviors. Coupled with strong thermal effects, flows tend to be almost unpredictable for anything less that a full 3D fully coupled numerical simulation.

Though it is true that, losely following J.C. Ferreri's words, "you should never attempt to numerically solve a problem for which you don't even know how the solution should look like...", it is also true that these simulations usually provide an insight into the fundamental physics involved. This sole result of numerical simulations is important enough so as to justify its use in any complex enginnering problem. The fundamental features of the problem at hand and the interplay between the different phenomena, revealed by the simulations, allowed us to perform heuristic analyses that were much more realistic than what could have been done just by relying on past experience and fluid mechanics knowledge.

2.3 Some recollections on interfacing

Discussions between a construction firm and a computational mechanics consultant are not always easy. The firm normally needs "holistic" simulations that consider all possible phenomena, with sensitivity analyses on all existing variables, to determine once and for all how the component is going to work and what values of the design variables are optimal. The consultant, on the other hand, can only provide limited-scope results and under several simplifying assumptions.

This aggravates when the consulting work refers to a project that is just in an incipient stage of design. It is obviously impossible to simulate the flow inside a vessel with unknown geometry, unknown flow rate, and unknown heat input. But, on the other hand, it is useless to design the vessel down to its nuts and bolts to later find out that CFD predicts that it will not work at all.

In the RRRF project the heat input is calculated by the *neutronics group*, the pumping flow rates throughout the reactor are the concern of the *process design group*, while the structural stability and mechanical feasibility of solid parts is addressed by the *mechanical design group* (who actually provides the CAD data for mesh generation). All the design aspects are tightly coupled and each group advances rather independently, so that trying to simulate any component is really aiming at a moving target. Under such uncertainties any specific system presents "as many reasons for investigating additional phenomena as excuses for making further simplifying assumptions".

In our case (firm: INVAP S.E., in particular its thermal-hydraulics group, THI for short; computational mechanics analysts: MECOM group) things went quite satisfactorily, which we *a posteriori* attribute to several factors:

- The decision group was small, one or two persons on each side, and consisted of people with technical background.
- All reports and recommendations were to be audited by an external organization, namely ANSTO, in the framework of a formal contract.
- Each specific analysis was *not* agreed as one determined set of simulations, but as an assessment on some specific problem. In other words, the output requested from MECOM was not some numerical result but the answer to a technological question.
- The simplifying assumptions were agreed upon at the meeting table, and written down. If MECOM made further assumptions, it was "at their own risk", implying that it was their responsibility to ensure that the additional assumptions did not invalidate the results as concerning the problem agreed upon with THI.
- The different design variables coming from other groups (neutronics, mechanical design, etc.) were *frozen* at given values. In this way, MECOM answered questions that did not change with time. It was THI's responsibility then to apply it to the final design, if it was different (and it was) from the one at the time the question was formulated. Of course THI could always ask for another analysis with the new design variables.

Some of these points will become clearer from the examples below.

3 FIRST CASE STUDY: THE CORE RISER OR CHIMNEY

3.1 Description and basic design

The Primary Cooling System removes fission heat from the core by forced upward circulation of light water and transfers the heat to the Secondary Cooling System via heat exchangers. The pump discharge line has an interconnection with the Reactor Pool that diverts a fraction of the core cooling flow to produce a closure flow in the chimney. The purpose of this flow is to provide a hydraulic plug precluding hot water from the core to reach the top of the Reactor Pool. Pool water enters the chimney, flows downwards, joins the upward stream from the core and the total flow is driven along the outlet pipe (see Fig. 2). Relevant design data considered as input are:

- The chimney is a prismatic vertical duct with a square cross section of 0.35 m $\times$ 0.35 m, with an oblique branch on one side (see Fig. 2).

- The core cooling flow rate is 1900 m^3/h.
- The difference ΔT between the average core outlet temperature and the (colder) pool temperature is 15°C but may in some situations reach 30°C.

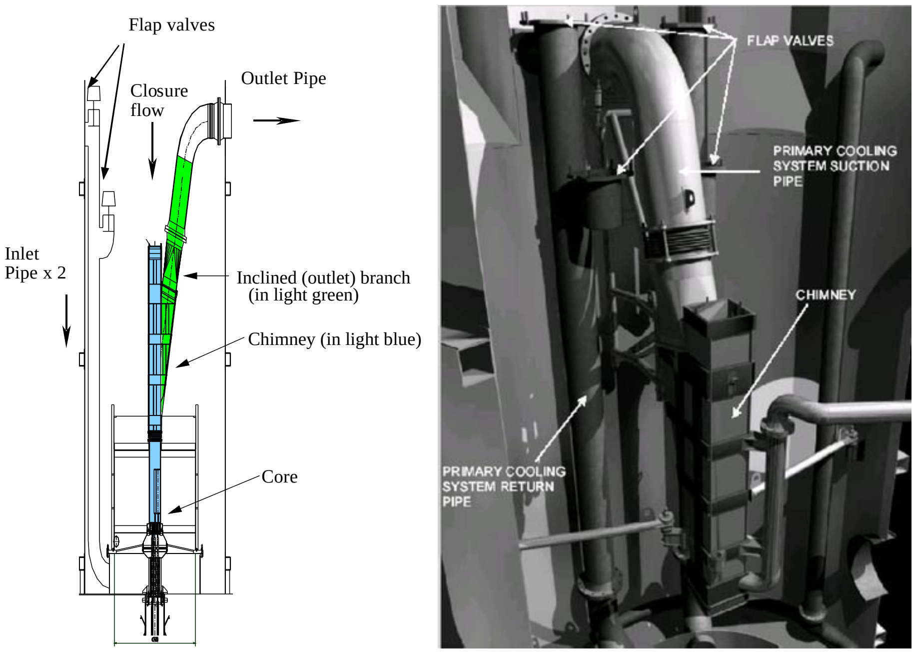

Figure 2: Scheme of the chimney's geometry.

3.2 The questions

The permanent closure downflow in the chimney acts as detention mechanism of the hot water coming from the core, which is activated mainly with ^{16}N. It is extremely important to avoid the presence of activated water in the reactor pool, since otherwise the dose of the personnel working in the Reactor Hall would not comply with strict design conditions on this issue. It is clear, on the other hand, that this closure flow does not participate in the cooling of any components and from the operational viewpoint is a parasitic flow which should be minimized. An overestimation would imply unnecessary pump capacity, oversizing of some equipment and the concurrent economic penalization.

It must be remarked that it is not at all standard practice to prevent an upward leak of hot water by imposing a downward mean flow of cold water. It was evident from the beginning that, for a wide and short chimney and a low enough closure flow there would be release of hot water towards the pool. The *physical mechanism* of the release, on the other hand, was not obvious to MECOM and THI. This was the first question to answer. The second question was to estimate the minimum closure flow that *prevents* such releases.

Though there exist several structures above the core, it was *agreed* to disregard them in the analysis and focus on the basic, large scale phenomena.

3.3 Preliminary analysis

The problem at hand can be classified as a mixed (natural+forced) convection of a liquid in a vertical chimney with a side bifurcation under conditions of inverse stratification. The *hot* core flow is intended to be sucked up the bifurcation along with some additional *cold* flow, called closure flow, taken from the pool at the top of the chimney. The purpose of the closure flow is to prevent the core flow from reaching the top of the chimney and consequently the main pool.

Under isothermal or nearly isothermal conditions, like those found at null or very low power levels, there is no physical mechanism that could amplify large scale perturbations to a level enough to destroy the overall two-dimensionality of the flow. Furthermore, velocity gradients are low except for the zones near the walls. No source of large scale vorticity is present and thus the flow is almost vorticity free everywhere. Therefore, the flow pattern closely resembles that of potential flow, and the interface between the core flow and the closure flow is stable and does not reach the top of the chimney.

At normal power levels, however, the vorticity-free flow field is no longer a reasonable approximation due to the presence of net sources of vorticity in the bulk of the flow, more precisely at the interface between the hot and cold streams. Therefore, potential flow theory is no longer applicable, at least in the region of the interface.

One mechanism of instability is vorticity amplification due to negative stratification. The vorticity transport equation under Boussinesq approximation reads

$$\frac{\partial \zeta}{\partial t} + \nabla \times (\zeta \times \mathbf{V}) + \beta\, \nabla T \times \mathbf{g} = 0.$$

where ζ is the vorticity vector, $\mathbf{V}$ the velocity, T the temperature, and β the coefficient of thermal expansion. The term $\beta\, \nabla T \times \mathbf{g}$ stands for the source of vorticity due to stratification. The temperature gradient at the cold-hot interface (located slightly above the upper limit of the suction window) is large, so that even small tilts of the interface provide strong, concentrated sources of vorticity. Assume a steady situation in which the flow is two-dimensional and close to a potential flow. If due to a perturbation the flow reaches the interface zone with even a small vorticity parallel to the local direction of movement (which is almost horizontal there) the interface would start to tilt to one side or the other, depending on the direction of the vorticity. This would cause the gradient of temperature to develop a component in the direction normal to the symmetry plane of the chimney, thus causing the term $\beta\, \nabla T \times \mathbf{g}$ to have a component in the same direction of the original vorticity and amplify exponentially. It the temperature gradient is stable (∇T pointing downwards) the contrary happens, with buoyancy effects adding stability against interface tilting.

It was numerically verified that indeed a 2D Navier-Stokes simulation predicts a stable interface in isothermal conditions and no large-scale interface instability at normal power levels. Remarkably, in the latter case the interface was not completely steady but exhibited a weak wavy motion, though it had no consequence concerning upwards release of hot fluid.

3.4 Simulation

The simulation was aimed at analyzing the motion of the interface between the hot and cold fluids, which as discussed before is prone to 3D instabilities. It was also necessary to analyze the interaction of these instabilities with the wavy motion identified by the 2D simulation. To capture the dynamics Reynolds-averaged formulations (such as $k - \epsilon$) were discarded, and a simple Smagorinsky subgrid-scale eddy diffusivity was used. The counterpart was, obviously, that a rather refined and necessarily 3D mesh was to be used, together with a time-accurate simulation which had to be long enough to collect representative dynamic information. The adopted mesh can be seen in Fig. 3. It consists of 72,080 nodes and is refined at the mean interface position. The time step was chosen as 0.02 sec. Typical large-eddy turnover times are L/U, with L a representative eddy size and U a representative velocity. Taking L equal to 0.35 m (approx. the chimney's width), and U equal to 0.45 m/s (10% of the main flow velocity) we get an upper bound for the eddy turnover time of 0.78 sec. Simulations were then carried out for about 60 seconds of simulated time, much more than 10 times L/U as usually recommended. This implies that about 3000 time steps were performed, each one requiring about 40 seconds of walltime when running on 10 processors of the Linux cluster.

The first simulation corresponded to a ΔT of 15 °C and a closure flow rate of 190 m^3/h (10% of the through-core flow rate). It was observed that the *mean* hot-cold interface coincided in fact with that obtained in the 2D simulations. But, contrary to the isothermal predictions and as discussed in the previous paragraph, the interface was unsteady, with a wavy pattern. Also, as predicted by the vorticity analysis, the interface tended to tilt spontaneously, which made the aforementioned waves to impact on the wall making a part of its crest to detach from the main stream. This is shown in Fig. 4, in which a fluid element from the crest of one of these waves is followed in time. It is observed that the wave-impact mechanism indeed releases parcels of hot fluid into the upper part of the chimney, but though they are buoyant the closure flow is strong enough to convey them downwards towards the outlet.

Further simulations were conducted with different closure flow rates and temperature differences, all of them followed the pattern described above and predicted that the proposed closure flow meets the design criteria.

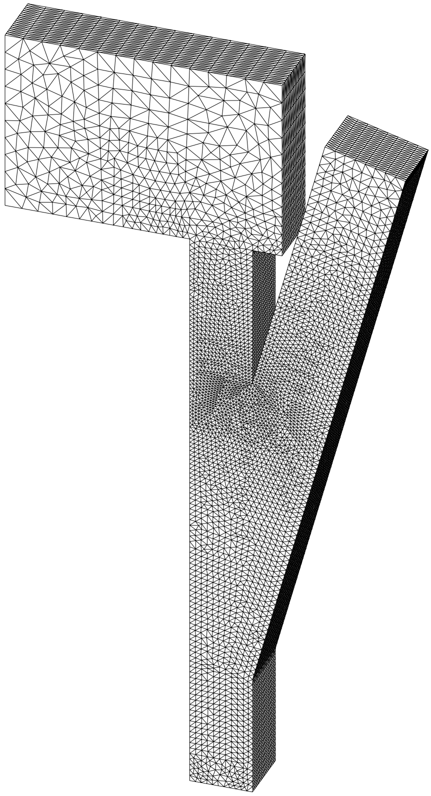

Figure 3: Mesh used to model the flow in the chimney.

3.5 Discussion. Heuristic interpretation

Careful observation of the 3D full simulation results, post-processed to produce a digital movie, shows that the hot fluid parcels are released from the hot/cold interface intermittently. It was thus decided to conduct a heuristic modeling work of this phenomenon so as to complement the numerical predictions with simpler, physically-based reasoning. Two approaches were followed:

- The first approach was to consider the time-average of the release parcels as a buoyant plume. From well-known plume correlations[16] the velocity at the top of the chimney in the absence of closure flow was estimated. Intuitively, if the velocity of the closure flow (which has a lower bound of 22.5 cm/s) is much larger than this estimated velocity (a few centimeters per second) the plume cannot develop.

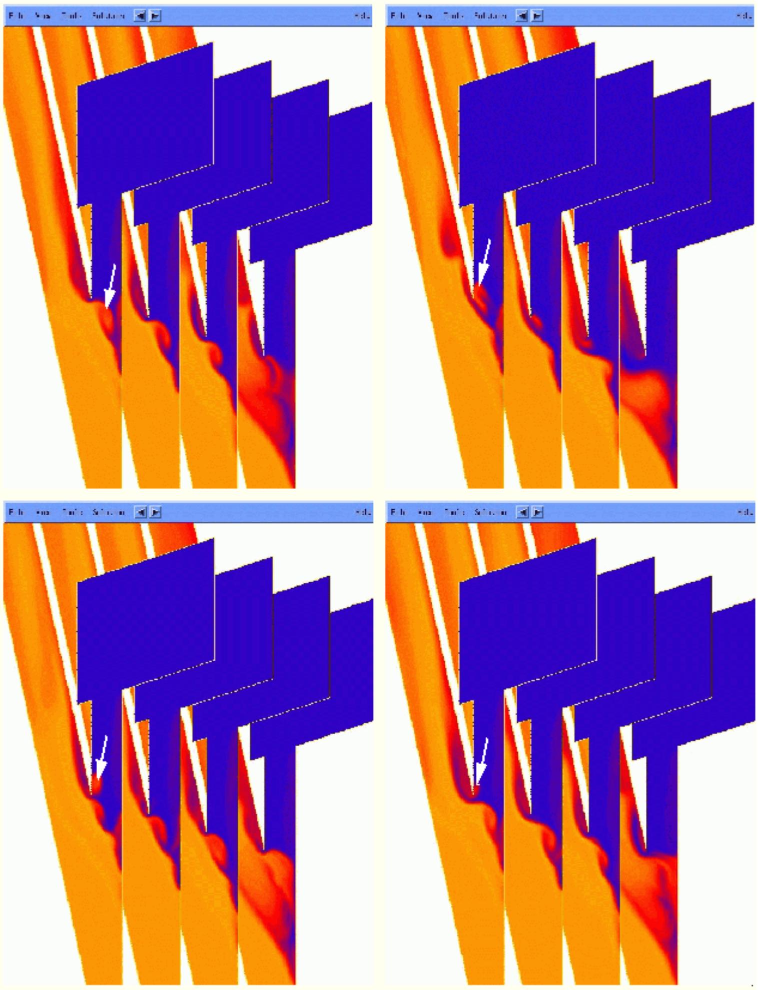

Figure 4: Vertical sections of the chimney showing the temperature field at different instants, with about 0.3 seconds between snapshots. The arrows indicate the successive positions of a hot fluid element that starts in the crest of a wave, collides with the wall, detaches from the main stream, and is later sucked downwards by the flow.

- The second approach was to consider each parcel *per se*, in which case direct balance of buoyancy with drag against the closure flow allows to calculate the velocity and from it to integrate the trajectory. This approach is very conservative since one considers no mixing, which would reduce the buoyancy of the parcel. Notwithstandingly, this analysis again showed that any realistic parcel of hot fluid released from the interface cannot reach the top of the chimney against the closure flow and will thus end up sucked by the outlet.

3.6 Recommendations

The study suggested that a closure flow higher than 5 % of the core flow was sufficient to fulfill the design criteria. The THI group of INVAP adopted a closure flow of 10 % after incorporating engineering factors. This flow rate was the input data for the final design of the layout, piping, pump and equipment specifications of the Primary System.

4 SECOND CASE STUDY: THE NEUTRON REFLECTOR VESSEL

4.1 Description and basic design

The Reactor Core is surrounded by a Reflector Vessel containing heavy water, which provides adequate neutron reflection and a large zone with high neutron flux. It is cylindrical in shape with flat top and bottom, and is made of Zircaloy-4.

The Reflector Vessel is traversed in the axial direction by tubes of various diameters that house irradiation rigs and targets. It also contains a Cold-Neutron Source, two cold neutron beam assemblies and two thermal neutron beam assemblies. An additional beam is available to serve a possible future hot-neutron source. All beam tubes are tangential to the core. The overall flow rate of heavy water through the Reflector Vessel is 137 m^3/h, and the inlet temperature is 40 °C. Heat is volumetrically deposited in the heavy water and in the structural components. The total heat deposited is about 1 MW. Figure 5 shows two cross sections and a view from above.

4.2 The questions

This problem was significantly more open than the previous one. A thermo-fluid-mechanics based proposal for the location of the inlet and outlet piping of the cooling system was needed, accounting for the following requirements: a) The neutronic aspects require homogeneous water temperature in the region of the neutron irradiation channels, intermittent flow patterns energized by buoyancy are to be avoided; b) the concentration of ^{16}N in the outlet water needed quantification, so as to make the design of the shielding of pipes, pumps and heat exchangers feasible; c) appropriate heat removal from the few components that are not refrigerated from inside must be guaranteed by the flow pattern in the Reflector Vessel.

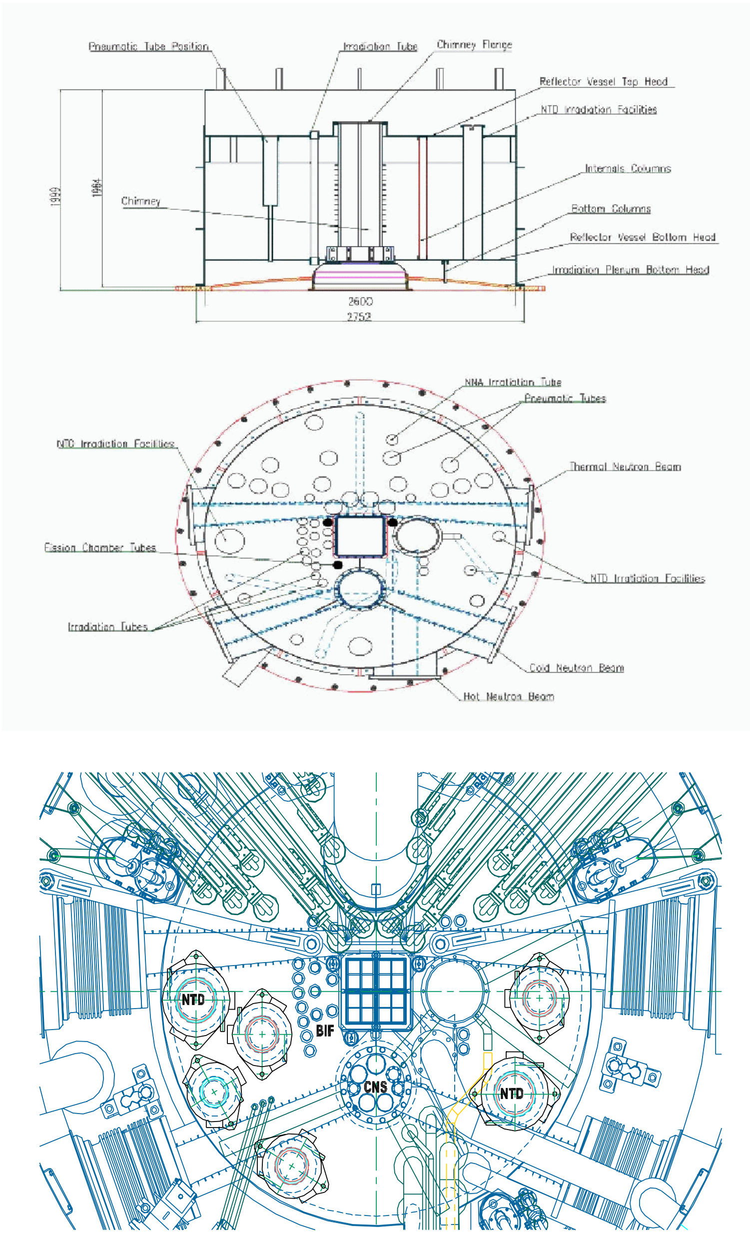

Figure 5: Two sections and a view from above of the Reflector Vessel.

4.3 Preliminary analysis

This is a typical case of a problem with unknown domain and boundary conditions. Moreover, flows with internal heat generation are not abundant in the literature. The problem of a closed cavity with internal heat generation, cooled through the walls, was studied in the sixties by Smith and Hammitt.[17] More recently, interest reappeared in connection with heat removal from a flooded reactor lower plenum in accidental conditions[18,19] (and references therein).

In the Reflector Vessel case, however, heat is removed by simply circulating fluid through the vessel at a suitable flow rate. The injection/extraction of fluid has significant effects on the flow dynamics, since the energy exchange is accompanied by mass and momentum transfer. This prompted a *basic* research effort so as to determine what the basic physics of such problems are.[20] It was found that two basic regimes exist, the inertia-dominated regime when the inflow enters as a highly energetic jet, and the stratified regime when the inertia of the inflow is weak. In particular, it was shown that significant streaming of cold fluid towards the outlet and trapping of hot fluid in recirculating regions may take place in the inertia-dominated regime. The actual amounts of streaming and trapping depend strongly on the geometry of the vessel and on the location and orientation of inlet and outlet. The average and maximum temperatures in the vessel may largely exceed the mean outlet temperature.

Both the possibility of high temperatures and the sensitivity to geometry were undesirable features of a high-inertia design (notice that the exact size, shape and location of the inserts shown in Fig. 5 were not available at the time of defining the flow injection and extraction systems). Numerous exploratory simulations were however performed in a model toroidal domain of square cross-section assuming revolution symmetry. Different alternatives for the position of the inlet, such as close to the core or far from it, at the top of the vessel or at its bottom, with higher or lower velocity (for the same flow rate), were considered. The same was done with the outlet's position. From this preliminary analysis it became clear that a design that favored as much as possible the establishment of a stratified thermal structure in the vessel is preferrable. For this, a toroidal distributor with orifices at the bottom of the vessel and with the maximum possible radius (as permitted by the radius of the external vertical wall) was selected. Another, similar toroid collects the outflow near the top of the vessel. As shown in Fig. 6, oversimplified 2D simulations indeed predict that in this design a stratified thermal structure prevails. The THI group recommended the toroidal-distributors alternative based on the preliminary analysis, but of course a more thorough, 3D analysis was pending. It had to wait until the mechanical design of the different components that are housed by the reflector vessel had been defined, together with their positions. This is reported next.

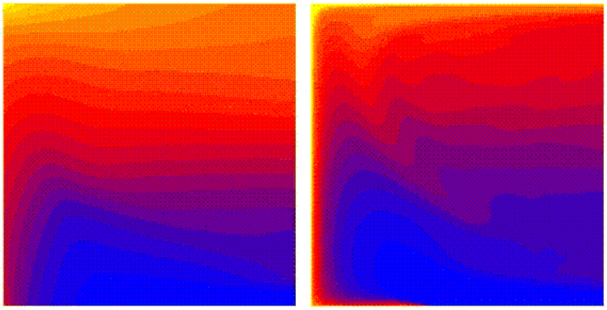

Figure 6: Temperature field in an idealized model of the reflector vessel. Water enters from the bottom right corner and leaves by the top right one. The left boundary models the wall between the reflector and the core. Revolution symmetry around a vertical axis 17 cm to the left of the left boundary is assumed. Run1 assumes a RANS turbulent model, Run2 laminar flow.

4.4 Three-dimensional simulations

The layout of the computational model can be seen in Fig. 7. A detail of the surface mesh is also shown, together with the inflow and outflow areas. The inflow is uniformly distributed over a ring, with an imposed velocity of 0.0288 m/s. The outflow, which takes place through a perforated toroidal pipe, was modelled as a region of high mixing and abrupt pressure-drop at the outer part of the top of the vessel.

The volume mesh contains 377471 nodes and 2101544 tetrahedral elements. Fig. 8 shows this volume mesh, with some elements removed, note the mostly regular structure in the bulk and a single layer of stretched elements along the surface.

The heat deposition data at the walls of the beams, the chimney and the vacuum container were provided by the *neutronics group* of INVAP S. E. The volumetric heat load was assigned a spatial dependance of the kind: $q''' = c_1 e^{-\frac{z^2}{c_2+c_3 r}} e^{-r/c_4}$ where z is the vertical distance from the centre of the core in meters, and r is the radius from the core centre. The constants c_i were obtained fitting MCNP simulation results.

Turning now to the results, we observe that except for the regions near the core, and the associated sheet of hot water at the top of the vessel, the bulk of the liquid is strongly stratified. This is in agreement with theory, since the Richardson number is approximately 3, large enough for stratified conditions to appear.

The buoyant ascending jet that develops against the chimney's wall spreads out radially after reaching the top of the vessel. In its way towards the outlet, this hot liquid passes several objects (irradiation tubes, vacuum container, etc.) that obstruct and deflect the flow. This is most evident in Fig. 9 that shows the contours of the velocity modulus at the surface of the vessel.

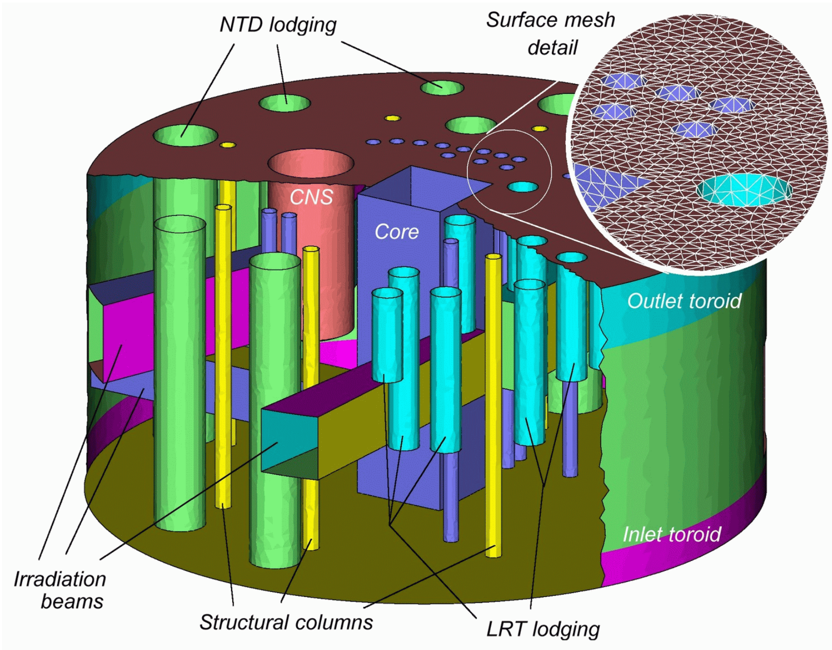

Figure 7: Computational domain and a detail giving an idea of the mesh size.

Values of the temperatures at the interior solid surfaces were plotted in Fig. 10, Though the flow is unsteady, snapshots at other times reveal no significant changes in the variables of interest. The stratified temperature profile in the liquid can also be appreciated on the solid surfaces. The flowfield and temperatures were considered appropriate for a correct neutronic behavior of the Reflector Vessel, which is now in a stage of detailed design.

5 CONCLUDING REMARKS

Large engineering facilities involve complex, coupled technological problems which challenge both the current knowledge of fluid mechanics and the current capabilities of CFD software and hardware. Although all of them (knowledge, software, hardware) are growing rapidly these days, the magic canned code that exactly (and instantly) predicts the behavior of any technologically relevant system is still in the distant future. This means that the engineer in charge of an analysis must abstract a model from the real data, translate it into a mathematical-numerical form, and solve it with reasonable speed and accuracy. When the analysis refers to a project that is in its initial design stages, CFD tools are, as shown above, useful for identifying global trends and behaviors. In

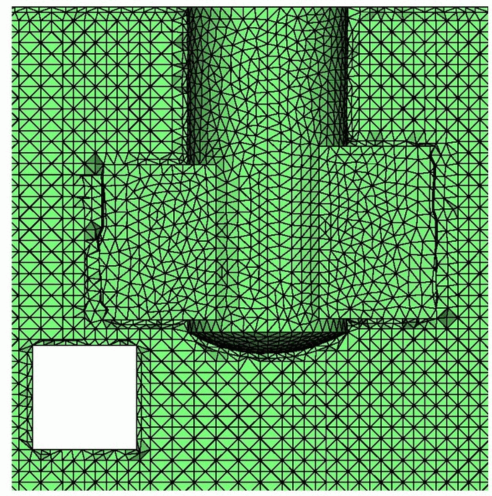

Figure 8: Detail of the volume mesh.

the case of the chimney of the RRRF, for example, revealing the mechanism of hot-water release towards the pool proved to be conceptually most important. Once this mechanism was understood, hand calculations lead to the same conclusion as CFD results: Closure flows with vertical velocities exceeding about 15 cm/s satisfactorily “seal” the chimney. The same applies to the Reflector Vessel study: Once the problem of trapping hot fluid by inertia-dominated flow structures was identified and a stratified regime was selected, the appropriate design followed.

Concerning software development, the first thing to notice is that during this project practically no truly steady flow was encountered. Some dynamical aspects are present in almost all the systems, and understanding them was most important for the assessments. When a steady state exists a steady code finds it in optimal time, but only a time-accurate code can in fact predict spontaneous bifurcations to unsteady motions. The same can be said about the two-dimensionality assumption, which saves lots of CPU time but, as shown in the chimney’s case, can predict stable behavior simply because the unstable mode is intrinsically three-dimensional. When both time-accuracy and three-dimensionality are put together, the need for high-performance computing appears. Our experience with the Linux Cluster of the Instituto Balseiro, which started with 11 Dual-Pentium nodes at 300 MHz and has recently been upgraded to 22 Athlon nodes at 1 GHz has been most satisfactory in what concerns implementing and running CFD unstructured solvers. Portability to other Linux Clusters (Pentium-based and Itanium-based) also proved to be good.

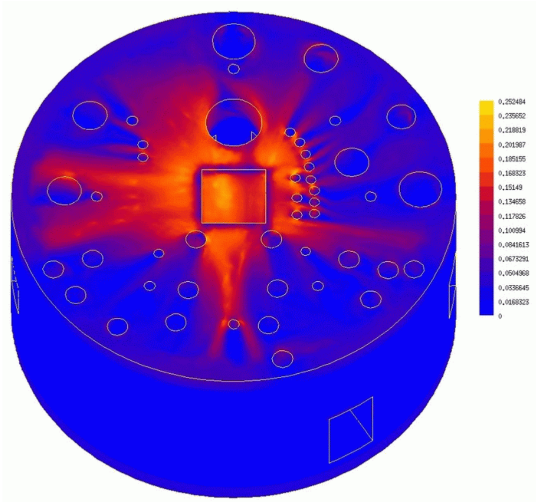

Figure 9: Velocity modulus at the surface of the computational domain.

Finally, the division of responsibilities that, perhaps fortituously, took place between THI and MECOM, was important for the successful completion of this project. The aspects discussed in Section 2.3 proved to work well for this case, allowing the quick identification and distribution of problem-oriented tasks between the two groups.

ACKNOWLEDGMENTS: This work received partial support from Agencia Nacional de Investigaciones Científicas y Tecnológicas through grant PICT'99-6337. Fruitful discussions with P. Abbate, P. Carrica, A. Doval, O. Lovotti, N. Masriera, C. Soga and O. Zamonsky are gratefully acknowledged.

REFERENCES

[1] R. Codina, J. Blasco, G. Buscaglia, and A. Huerta. Implementation of a stabilized finite element formulation for the incompressible navier-stokes equations based on a pressure gradient projection. *Int. J. Numer. Meth. in Fluids*, 37:410–444, 2001.

[2] G. C. Buscaglia, H. Ferrari, P. M. Carrica, and E. A. Dari. An application of distributed computing to the finite element investigation of lift force on a freely-rotating sphere in simple shear flow. In D. Stock, editor, *Proceedings of the ASME Fluids Engineering Division – 1999*, volume FED–Vol.250, pages 265–271. The American Society of Mechanical Engineers, 1999.

[3] A. J. Lew. Master's thesis, Instituto Balseiro, 1998.

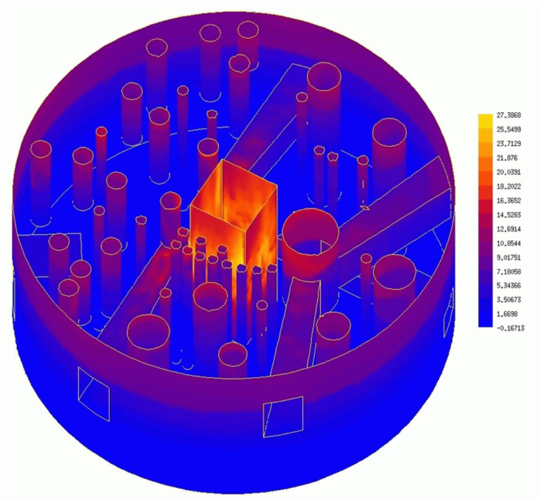

Figure 10: Temperature at solid surfaces (units are degrees Celsius above the inlet temperature).

[4] M. I. Cantero. Master's thesis, Instituto Balseiro, 2000.

[5] J. E. Martín. Master's thesis, Instituto Balseiro, 2001.

[6] G. C. Buscaglia, E. A. Dari, A. J. Lew, and M. Raschi. Un programa general de elementos finitos en paralelo. In C. García Garino et al, editor, *Proceedings VI Congreso Argentino de Mecánica Computacional (Mendoza)*, pages CD–ROM. Ex-Libris, 1999.

[7] R. Codina. Comparison of some finite element methods for solving the diffusion–convection–reaction equation. *Comp. Meth. in Appl. Mech. and Engrg.*, 156:185–210, 1997.

[8] R. Codina and J. Blasco. Stabilized finite element method for the transient navier-stokes equations based on a pressure gradient projection. *Comp. Meth. in Appl. Mech. and Engrg.*, 182:277–300, 2000.

[9] G. Buscaglia, F. Basombrío, and R. Codina. Fourier analysis of an equal-order incompressible flow solver stabilized by pressure-gradient projection. *Int. J. Numer. Meth. in Fluids*, 34:65–92, 2000.

[10] J. Ferziger. Subgrid-scale modeling. In B. Galperin and S. Orszag, editors, *Large eddy simulation of complex engineering and geophysical flows*. Cambridge Univ. Press, 1993.

[11] M. Ciofalo. Large eddy simulation: A critical survey of models and aplications. In J. Hartnett and T. Irvine, editor, *Advances in HEAT TRANSFER, vol. 25*. Academic Press, 1994.

[12] G. C. Buscaglia, E. A. Dari, J. E. Martín, D. L Arnica, and F. J. Bonetto. Finite element modeling of liquid deuterium flow and heat transfer in a cold-neutron source. *International Journal of Computational Fluid Dynamics*, to appear, 2002.

[13] E. A. Dari. PhD thesis, Instituto Balseiro, 1994.

[14] G. C. Buscaglia, E. A. Dari, and P. D. Zavattieri. Mesh optimization: Some results in 3d elasticity. *European Series in Applied and Industrial Mathematics Proceedings*, 2:1–16, 1997.

[15] P. D. Zavattieri, E. A. Dari, and G. C. Buscaglia. Optimization strategies in unstructured mesh generation. *International Journal for Numerical Methods in Engineering*, 39:2055–2071, 1996.

[16] H. Fischer, E. List, R. Koh, J. Imberger, and N. Brooks. *Mixing in inland and coastal waters*. Academic Press, 1979.

[17] W. Smith and F. Hammitt. Natural convection in a rectangular cavity with internal heat generation. *Nuclear Sci. and Engn.*, 25:328–342, 1966.

[18] F. Asfia, B. Frantz, and V. Dhir. Experimental investigation of natural convection heat transfer in volumetrically heated spherical segments. *Trans. ASME, J. Heat Transfer*, 118:31–37, 1996.

[19] C. Tzanos and D. Cho. Numerical predictions of natural convection in a uniformly heated pool. *Trans. Amer. Nucl. Soc.*, 68:496–498, 1993.

[20] G. C. Buscaglia and E. A. Dari. Numerical investigation of flow through a cavity with internal heat generation. *Numerical Heat Transfer*, to appear, 2002.