Production d'énergie (hydraulique, thermique et nucléaire)

CALCULS TRIDIMENSIONNELS D'UN ECOULEMENT DANS UNE CUVE D'UN REACTEUR A EAU PRESSURISEE 900 MWE

THREE DIMENSIONAL CALCULATIONS OF THE PRIMARY COOLANT FLOW IN A 900 MW PWR VESSEL. STEADY STATE AND TRANSIENTS COMPUTATIONS

EDF-- 96NB00126

Direction des Etudes et Recherches

EDF
Electricité de France

Direction des Etudes et Recherches

SERVICE APPLICATIONS DE L'ELECTRICITE ET ENVIRONNEMENT
Département Laboratoire National d'Hydraulique

Mars 1996

MARTIN A.
ALVAREZ D.
CASES F.

CALCULS TRIDIMENSIONNELS D'UN ECOULEMENT DANS UNE CUVE D'UN REACTEUR A EAU PRESSURISEE 900 MWE

THREE DIMENSIONAL CALCULATIONS OF THE PRIMARY COOLANT FLOW IN A 900 MW PWR VESSEL. STEADY STATE AND TRANSIENTS COMPUTATIONS

Pages : 12

96NB00126

Diffusion : J.-M. Lecœuvre
EDF-DER
Service IPN. Département SID
1, avenue du Général-de-Gaulle
92141 Clamart Cedex

ISSN 1161-0611

SYNTHÈSE :

Ce document retrace la chronologie de l'étude et les premiers résultats numériques du programme de R&D sur les mélanges dans les cuves de REP 900 MW. Après une présentation du type de tranche simulé, nous définissons l'outil numérique utilisé, en l'occurrence le code aux éléments finis N3S.

Deux résultats sont présentés avec une comparaison avec les résultats expérimentaux issus de la maquette BORA BORA. Le premier cas relate l'étude en régime permanent de l'écoulement induit par la puissance résiduelle du réacteur après arrêt des pompes primaires. Ce résultat, identifié comme une validation de notre outil numérique, est similaire à celui trouvé expérimentalement. Le deuxième cas traite du problème de la dilution d'une poche d'eau claire dans la cuve lors du redémarrage d'une pompe primaire.

Nous comparons les résultats expérimentaux et numériques donnant la concentration moyenne de bore en entrée cœur pour plusieurs volumes d'eau claire entrant dans la cuve. Les comparaisons montrent encore une bonne similitude.

EXECUTIVE SUMMARY :

The paper explains the chronological account and the first results obtained in the R&D program on the mixing in the 900 MW PWR vessels. After the presentation of the plant type simulated, we define the numerical tool, the FEM N3S code.

Two results are presented with a comparison with the experiment results issued of the BORA BORA mock up. The first case is dealing with the isothermal steady state mixing in the vessel with the three loops mass flow rate balanced. This case identified as a validation of our numericl tool shows a good agreement. The second case is dealing with the transient mixing of a clear plug in the vessel when one primary pump starts-up.

We compare the numerical and experiment results giving the mean boron concentration at the core inlet for several clear water plugs. The results show again a good agreement.

EXECUTIVE SUMMARY :

The paper explains the chronological account and the first results obtained in the R&D program on the mixing in the 900 MW PWR vessels. After the presentation of the plant type simulated, we define the numerical tool, the FEM N3S code.

Two results are presented with a comparison with the experiment results issued of the BORA BORA mock up. The first case is dealing with the isothermal steady state mixing in the vessel with the three loops mass flow rate balanced. This case identified as a validation of our numericl tool shows a good agreement. The second case is dealing with the transient mixing of a clear plug in the vessel when one primary pump starts-up.

We compare the numerical and experiment results giving the mean boron concentration at the core inlet for several clear water plugs. The results show again a good agreement.

96NB00126

(HE-41/96/013/A)

THREE DIMENSIONNAL CALCULATIONS OF THE PRIMARY COOLANT FLOW IN A 900 MW PWR VESSEL. STEADY STATE AND TRANSIENTS COMPUTATIONS.

A. Martin, D. Alvarez, F. Cases

Département Laboratoire National d'Hydraulique
Électricité de France
Chatou
FRANCE

INTRODUCTION

After the Tchernobyl accident, a working group was created to analyse the French PWR Safety with a respect to potentiel risk of reactivity accident. Potentially risky situations are those which can lead to heterogeneous boron concentration or temperature of the primary coolant fluid. They may appear :

- after a normal hot shut-down operation, when the reactor cooling is obtained by natural convection induced by the residual power or by a Residual Heat Removal System,

- when a primary circuit is diluted or cooled by clear water injection,

- after a significant water volume injection, when a primary pump or an auxiliary circuit starts-up.

A Research and Development action has been created and the program aiming at an increased knowledge of vessel thermalhydraulics consists in two complementary approaches based on mock-up experiments and numerical simulations (Fig. 2).

The paper presents the CPY PWR plant, the numerical tool used to study it and the last numerical results available. The previous numerical results presented during the "Fourth International Topical Meeting On Nuclear Thermal Hydraulics Operation And Safety" had been obtained with a mesh not taking into account the obstacles in the lower plenum of the vessel. A second mesh (Fig. 3) has been generated and in this paper, the last results related to the validation schedule and to the transient studies are presented. The first result concerns the primary coolant mixing after a normal hot shut-down operation. Two numerical results (with and without inner structure) and the experimental results are compared. Then, the numerical tool is applied to the study of clear water transients in the vessel during a pump start-up. In this case, the purpose is to determine the transient boron concentration at the core inlet. Numerical and experimental results in term of boron concentration at the core inlet for several clear water plug volume are compared.

THE 900MW CPY PWR :

As mentioned before, the purpose of these studies concerns the primary coolant mixing capabilities in French PWR vessels. From an overall point of view, the 900MW CPY plant has been chosen because it is a three loop type reactor which is the more common running plant in our country and because the CPY type offers the most geometric complexities such as the downcomer's four sectorized thermal shields (Fig. 1).

With the benefits of previous works on PWR, especially the CEA/EDF/FRAMATOME/WESTINGHOUSE Joint Research Program PWS 2-9, the exact vessel geometry is taken into account as much as possible. The simulation area of interest starts several cold legs diameters upstream the vessel entrance elbows, up to the lower core plate (See Fig. 1 for the reactor and the domain concerned). The core itself is not simulated and the hot legs are only represented by the space required for them in the downcomer.

THE N3S CODE

The FEM code N3S has been developed by the Research Branch of E.D.F for thermalhydraulics studies in nuclear engineering design (see Pot et al., 1993, Chabard, 1994) taking advantage of our experience on 3D finite difference codes. The developpement of N3S started in 1982. The main feature is the use of unstructured meshes for complex geometries modelling. After intensive testing required by EDF Quality Assurance policy, it is now available for use as a general purpose tool which has been applied successfully to a wide variety of incompressible laminar or turbulent flows (see Chabard et al., 1992) with or without heat transfer (see Delenne and Pot, 1993). For code assessment a wide range of computer program validation are made under a Quality Assurance procedure for every major release of N3S. Code results are compared with analytical solutions when available or with literature experiments (see Chabard, Pot and Martin, 1993). Computations of benchmark exercises for international numerical workshops constitute a code validation (see Leal de Sousa, 1994).

An important work has been done on the choice of efficient algorithms and on their implementation in order to reduce CPU time and memory allocation (see Pot et al., 1993) leading to release 3.1 of N3S. Table 1 gives an illustration of this work, because we begun our computations with release 3.0 and then go on with release 3.1.

N3S release	3.0	3.1	Ratio 3.0/3.1
CPU (ms/Dt/node)	1.61	0.41	3.93
Memory (MWords)			
disk	355	79	4.5
in core storage	55	43	1.28

Table 1 : CPU time and Memory allocation on a CRAY Y-MP for primary coolant flow study in a 900 MW PWR vessel.

The N3S package contains four main steps : the pre-processor, the solver's interface, the solver N3S itself and the post-processor.

For the pre-processing task, the I-DEAS™ software and most exactly Object Modelling (OM) and Finite Element Modelling (FEM) have been used for the geometry definition (mapped meshing) and SIMAIL™ (free meshing) for the unstructured mesh generation.

The solver's interface PREN3S checks the mesh and prescribes the boundary conditions.

The solver N3S solves the Reynolds Averaged Navier-Stokes Equations for an unsteady incompressible or compressible flow with a standard k-ε two equations turbulence model (see Chabard and Pot, 1995). For buoyancy driven flows with small temperature differences, the momentum equation is coupled to the energy equation using Boussinesq approximation. To take into account more important thermal effects this leads to consider Navier-Stokes equations in which density depends on temperature (varying density) : $\rho = \rho(T)$. The time discretization is based on a fractional step method (see Caruso and Mechitoua, 1992). At each time step the code solves successively :

- an advection step, for the non-linear convection terms of the Navier-Stokes equation and the k and ε equations, by using the characteristics method (Boukir, Maday and Metivet, 1992).

- a diffusion step on scalar variables.

- a generalized Stokes problem for the velocity and the pressure, solved either by a Uzawa (projected gradient) or a Chorin-Temam algorithm (preconditioned conjugate gradient [P/G/N]).

Assuming a logarithmic velocity profile, we use wall functions on the boundaries to compute the friction shear stress at each time step. Additionally, a zero flux condition is given for k which leads to the definition of a second velocity scale used to prescribe ε at the wall (see Bidot et al.).

For the post-processing task, the *GRAFN3S* and *ENSIGHT* softwares were used for FE visualisations in fluid dynamics. Obviously, the mesh and all computed variables can be plotted but also 2D isocontours or fluid trajectories with the same interpolation method used in the solver GRAFN3S. ENSIGHT allows the visualisation of 3D fields.

For the application to the CPY PWR, P1-isoP2 tetrahedrons are usedfor efficient turbulence solutions (see Pot and al.,1993). The preprocessing task led to two 3D meshes : the first (201565 nodes and 133815 elements) with the lower plenum free of obstacles and the second (361911 nodes and 237801 elements) with the plates and instrumentation columns (Fig. 3). Results presented in this paper are from computation with both meshes.

STEADY STATE COMPUTATIONS AND EXPERIMENTAL RESULTS

The objective of these comparisons is a validation of the code on the PWR vessel complex geometry.

The computation takes into account the case of the primary coolant fluid flow when the reactor cooling is assured by free convection induced by the residual power. The total core flow rate is 1050 kg s^{-1} (350 kg s^{-1} in each cold legs). The pressure is 155 bars and the fluid is isothermal at 300°C. A passive tracer (an advection-diffusion equation for a passive scalar is solved with the turbulent fluid flow quantities) is used to make sure that a converged state has been reached. At the beginning of the computation the tracer value is zero, except at the three cold legs entrance plane boundaries where the prescribed value is one. The convergence state will be reached when a value of one is obtained everywhere in the domain. A second passive tracer, prescribed only on cold leg 2, gives data about the mass mixing between the cold legs in the whole computed domain and particularly at the core inlet. Predicted fields of this tracer give boron or temperature mixing (without buoyancy effects), by homothetic considerations.

Two meshes have been used; one with the instrumentation in the lower plenum and one without this instrumentation.

About 60 CPU hours on Cray C90 have been necessary for these computations.

Figure n°4 shows the mixing of the fluid injected in cold leg 2. The isolines are plotted on the developed downcomer cylinder.

Figure n°5 gives the maps of iso value of the tracers at the core inlet for the computation (with and without instrumentation in the lower plenum) and experimental (with instrumentation) results. A little gyration effect is observed at the core inlet and in the downcomer. For these features, this figure shows a good agreement between the numerical and the experiment tools.

Figure n°6 shows the fluid flow pattern in the lower plenum for the two meshes.

BORON TRANSIENT EXPERIMENTS AND COMPUTATIONS

The case of a dilution transient studied here is related to the mixing of a clear water plug when one RCP starts up, with a zero mass flow rate in the two others cold legs. The plug has a zero boron concentration when entering the vessel . The remaining part of the primary coolant fluid has an initial value of 2000 ppm boron concentration. When the plug driven back by the RCP reaches the vessel inlet, the fluid flow has well established turbulent mixing caracteristics. So in study, the transient mixing of the plug is only considered between the vessel inlet and the core inlet, at a constant mass flow rate of 1450 kg s^{-1}. This mass flow value is determined by the transit time of the plug, between the pump and the vessel, and the RCP start-up flow-rate data (i.e. from 0 to 4400 kg s^{-1} in 20 s).

In the experiments, several clear water reactor volumes between 5 to 100 m^3 have been tested in order to obtain a wide range of variation giving us some knowlegde on the plug mixing.

In the computations two clear water volumes have been studied and reported in this paper (3 and 8m^3). Three steps have to be reached: firstly the steady state fluid flow pattern, secondly the clear plug simulation in the cold leg and finally, the mixing and the total disappearance of the plug in the vessel. The steady state fluid flow pattern is obtained as in the previous computational case.

We use again the passive tracer method to simulate the plug. In cold leg, the tracer number two is prescribed for a time corresponding to the entrance of the clear water plug. The plug of 8m^3 has been computed with the two meshes and the plug of 3m^3 only with the full mesh. A comparison has been done between full computation results.

The computation results for the two clear water volumes has been compared with the different experiment tests using several water reactor volumes. About 55 CPU hours have been necessary for these computations.

Figure n°7 gives time history of this tracer for the two clear water volumes, first to reach the steady state fluid flow and then during the transient. Figure n°8 shows the mixing of the plug in the downcomer.

Figure n°9 gives comparison between computations and experiments in terms of mean concentrations at the core inlet. For experimental results, several clear water plug volume have been reported and for the numerical results, two clear plug volumes (3 and 8m^3) have been reported.

The time scale is dimensionless, taking into account the global transit time in the vessel (i.e. ratio of the volume of the vessel with the flow rate). This figure shows a good agreement on the general temporal trends between the experiments themselves and between computational results and experiments.

Figure n°10 gives the mean of concentration at the core inlet for the complete numerical transient studies.

CONCLUSIONS

In this paper, the last results available on the french Research and Development action on PWRs' primary coolant capabilities are presented. Firstly, as a validation step, mock-up and computational results obtained for a steady state flow are, on the whole, in good agreement. The inner structures have an important role in the orientation of the flow. When the fluid upwards to the core inlet, the plates and the instrumentation columns concentrate the flow to the geometry center.

After that, different studies have been initiated on a clear water plug mixing in the vessel. The transient results show a good trend in term of mean concentration at the core inlet.

In general, the different levels of concentration at the core inlet computed and measured show a good agreement. However, an important work still must be done on the localisation of the minimum or the maximum concentration at the core inlet.

REFERENCES

D. Alvarez, A. Martin, J.P. Schneider, "Boron mixing transients in a 900 MW PWR vessel for a reactor start up operation." NUTHOS 4, Taïwan, 5-8 April 1994.

D. Alvarez, O. Daubert, L. Janvier, J.P. Schneider, "Three-dimensional calculations and experimental investigation of the primary coolant flow in a 900 MW PWR vessel." Proc of the NURETH 5 meeting, pp 586-592, Salt Lake City, USA 1991.

J.P. Chabard, "Modelling of turbulent complex flows on unstructured meshes using a finite element method", ECCOMAS : Second European computational fluid dynamics conference, Proc pp. 210-219, Stuttgart, 5-8 septembre 1994.

T. Bidot, S. Delaroff, J.-M. Vanel, G. Monville, G. Pot, "Application of the N3S finite element code to numerical simulation around a Peugeot car 405", presented at the Basel World User Days C.F.D. 1992.

K. Boukir, Y. Maday, B. Metivet, "A high order characteristics method for incompressible Navier-Stockes equations", Proc. of. Int. Conf. on Spectral and High Order Methods, Montpellier, France, 1992.

A. Caruso, N. Mechitoua, "Finite element computations of turbulent diffusion flame", Proc. of the 2nd Int. Conf in Advanced Computational Methods in Heat Transfer, Milan, Italy, july 1992.

J.P. Chabard, B. Métivet, G. Pot, B. Thomas, "An Efficient Finite Element Method for the Computation of 3D Turbulent Incompressible Flows", Finite Element in Fluids, vol 8 Hemisphere Publishing Corporation Ed, 1992.

J.P. Chabard, G. Pot, A. Martin, "Industrial applications of N3S finite element code", FED-Vol 171, pp 1-9, Advances in Finite Element Analysis in Fluid Dynamics ASME 1993.

J.P. Chabard, G. Pot, "Turbulence modelling in finite element industrial applications", FEMIF 9, Venise, octobre 1995.

L. Leal De Sousa, "Extension of a incompressible algorithm for compressible flow calculations : validation on a transsonic flow in a bump.", Workshop ETMA : Turbulence modeling for compressible flow arising in aeronautics, 1994.

G. Pot, J.P. Gregoire, L. Leal De Sousa, Y. Souffez, FEMIF 8, Proc : pp 527-536. Barcelone 20-23 septembre 1993.

G. Pot, O. Bonnin, V. Moulin, B. Thomas, "Improvement of finite element algorithms implemented in CFD code N3S for industrial turbulent applications", Proc of 8th International Conference on Numerical Methods in Laminar and Turbulent Flow, Swansea, U. K. 1993.

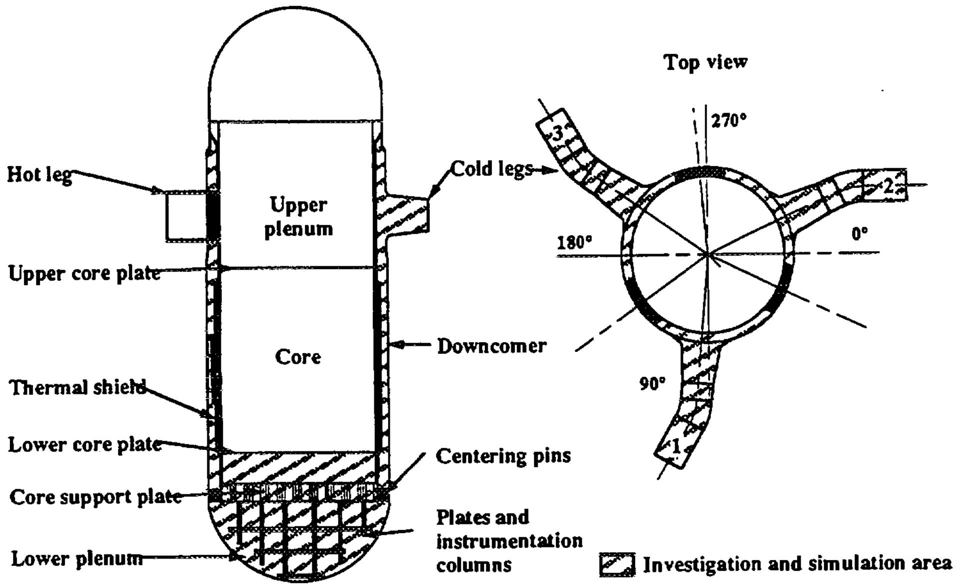


Figure 1 - 900 MW CPY PWR geometry.

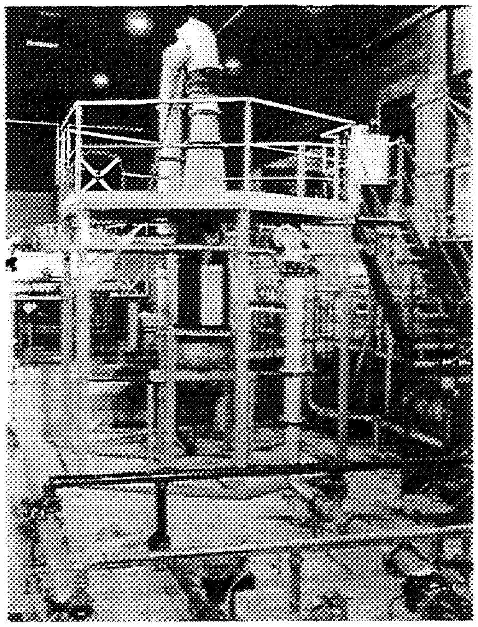
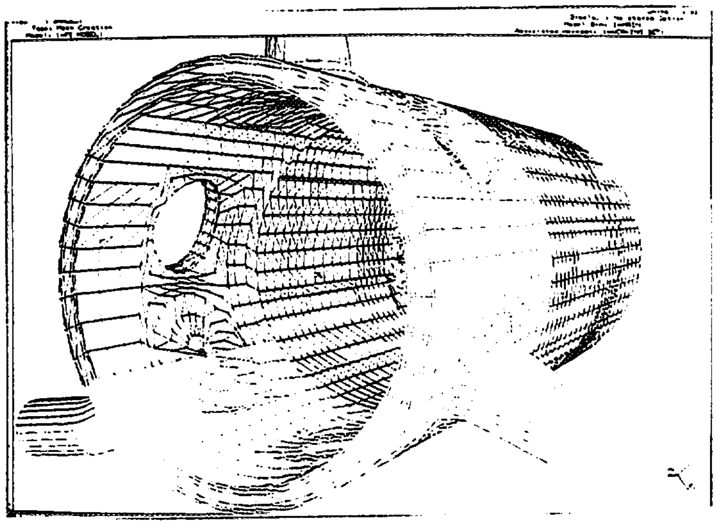



Figure 2 - Complementary Tools
Left : Bora Bora (scale 0.2)
Right : Finite Element Confinement

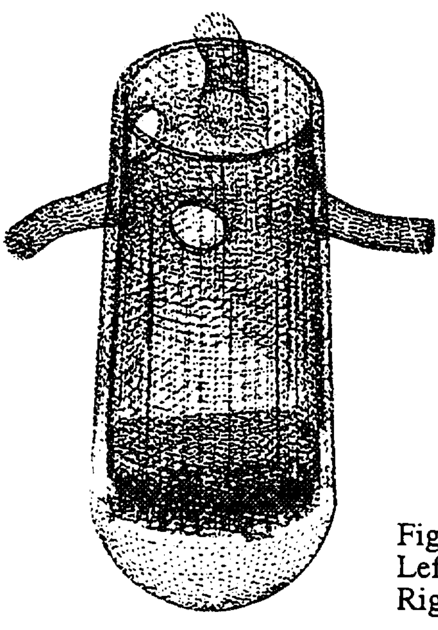
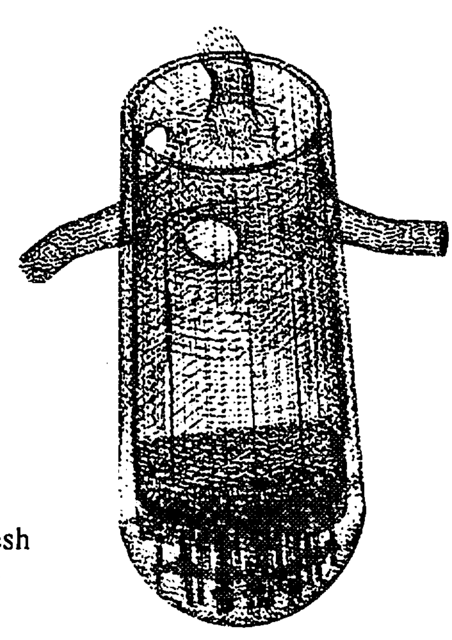



Figure 3 - Finite Element mesh
Left : without inner structure
Right : with inner structures

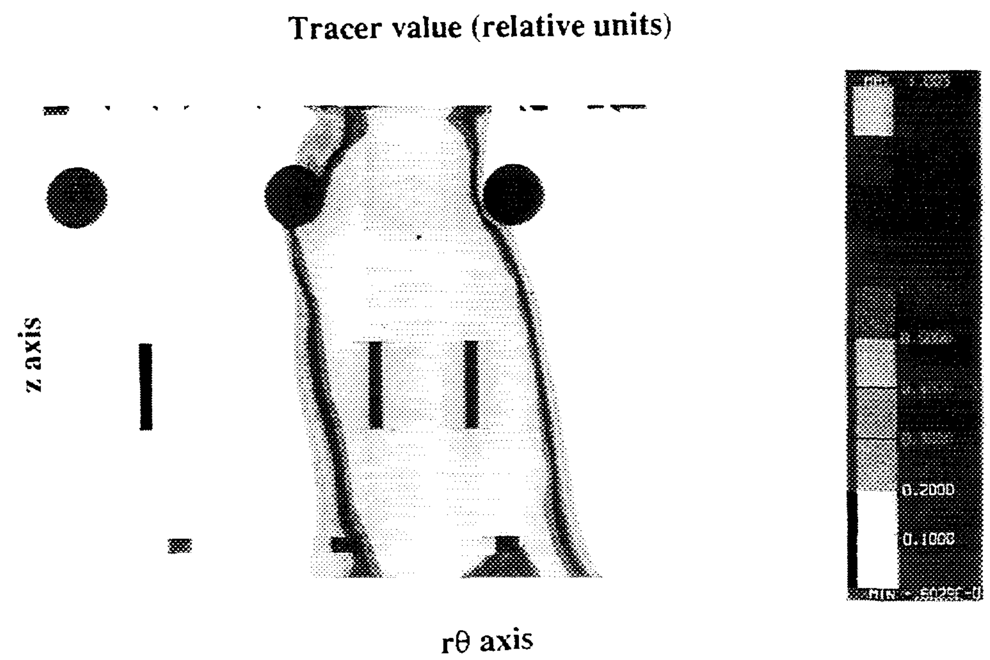


Figure 4 - Steady state mixing - N3S computed field of the passive tracer showing the mixing of the fluid coming from cold leg #2 in the downcomer.

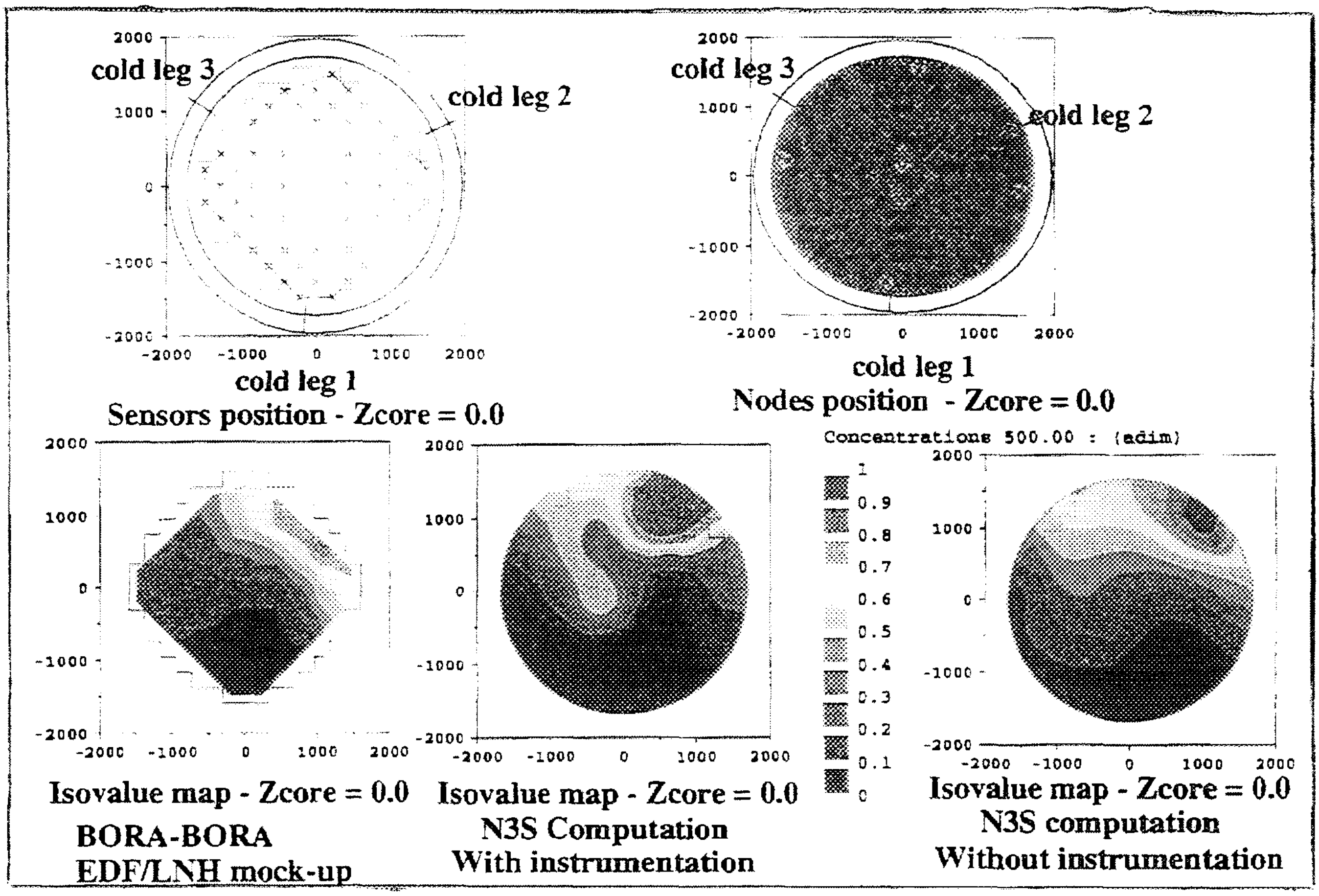
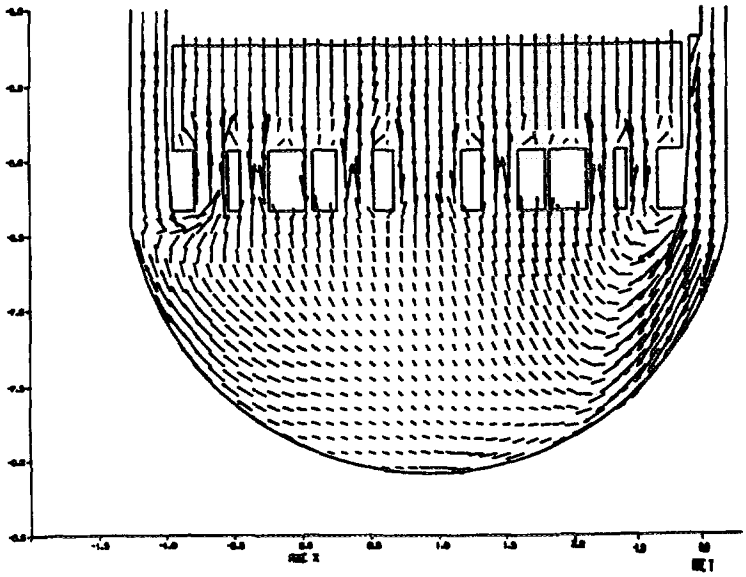
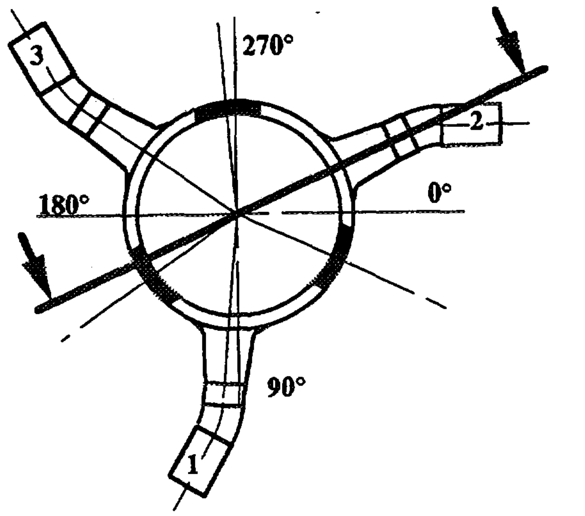


Figure 5 - Steady state mixing - Comparisons between N3S computations (two meshes) and BORA-BORA experiments on the mixing of the fluid coming from cold leg #2 in the vessel Concentrations maps at the core inlet.





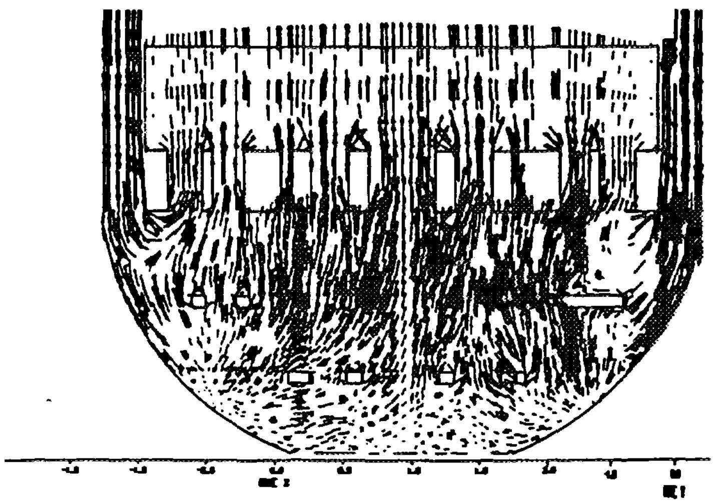
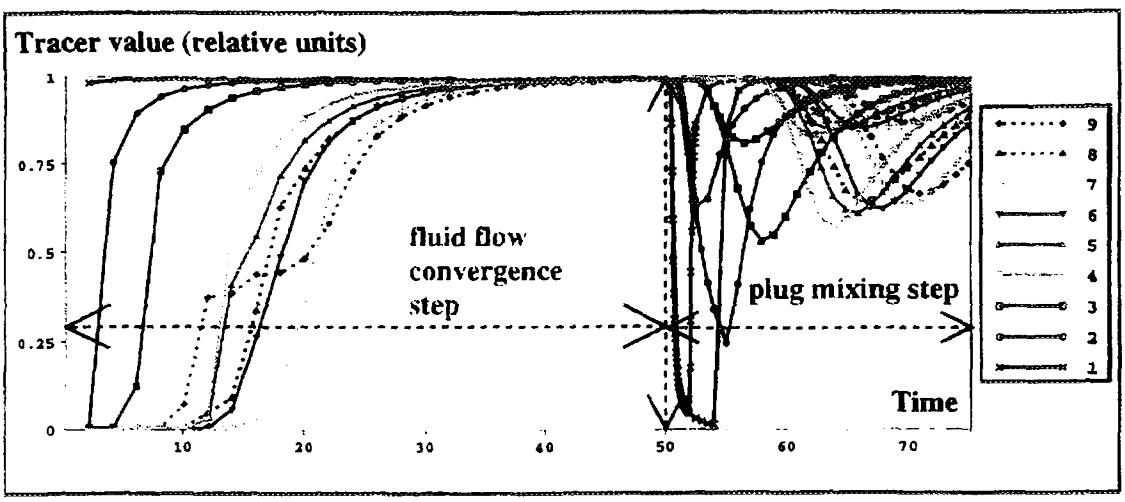
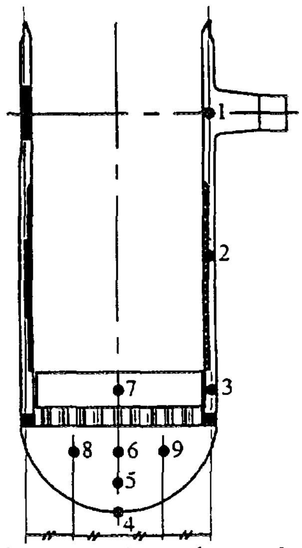


Figure 6 - Steady state mixing - N3S computed velocity plots in a vertical cut plane of the lower part of the vessel.
upper : without inner structure
lower : with inner structures





Nodes position in the vessel

Figure 7 - Clear plug transient mixing - Tracer concentration versus time computed by N3S for several spatial locations - Illustration for two volumes

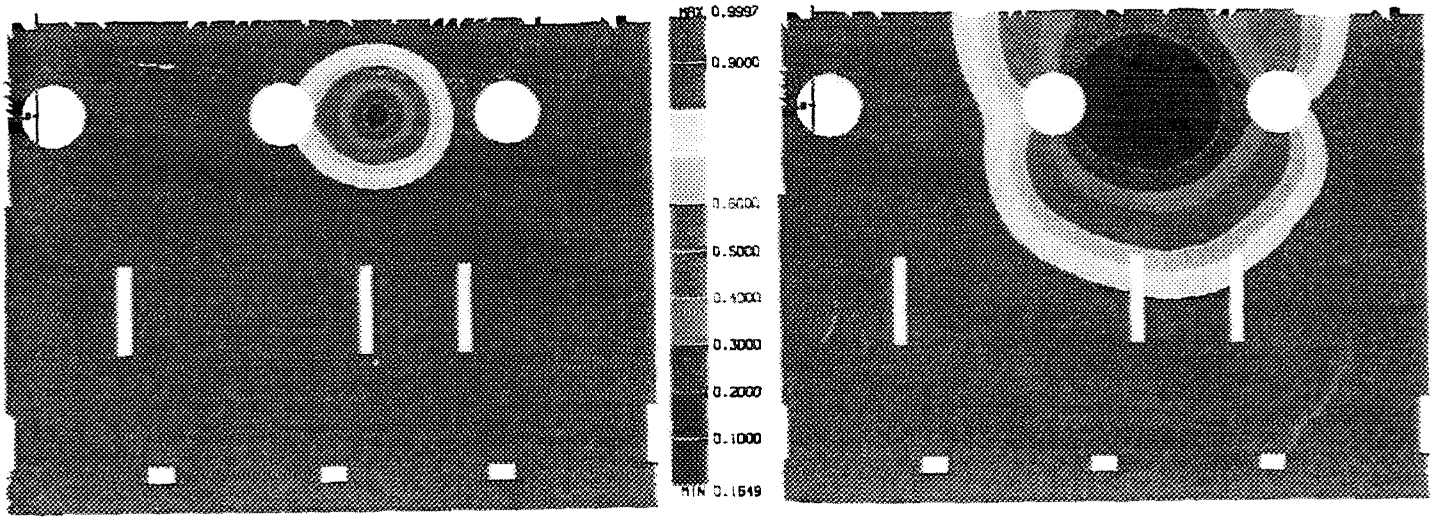


Figure 8 - Clear plug transient mixing - Concentration maps of the tracer in the downcomer computed by N3S for two different time steps.

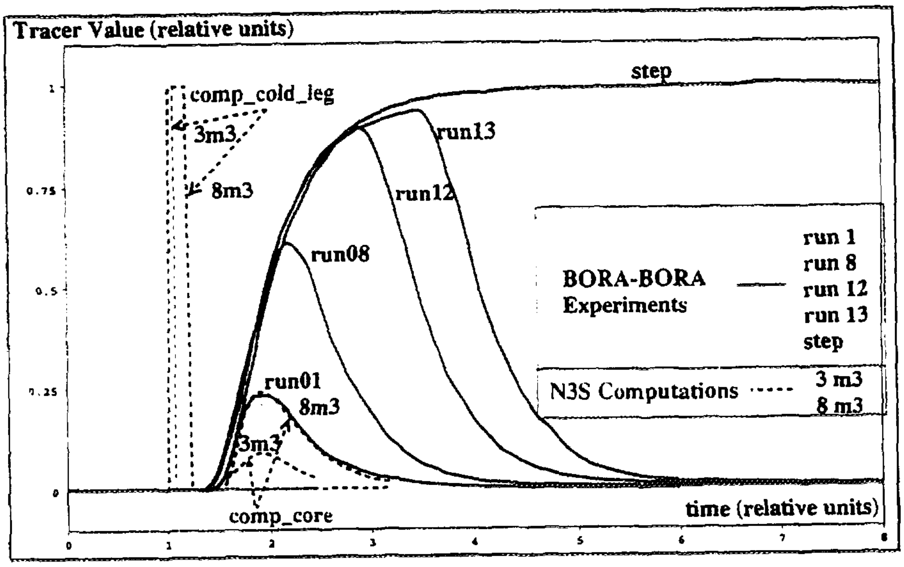


Figure 9 - Clear water plug transient mixing - Mean concentration measured and computed at the core inlet.

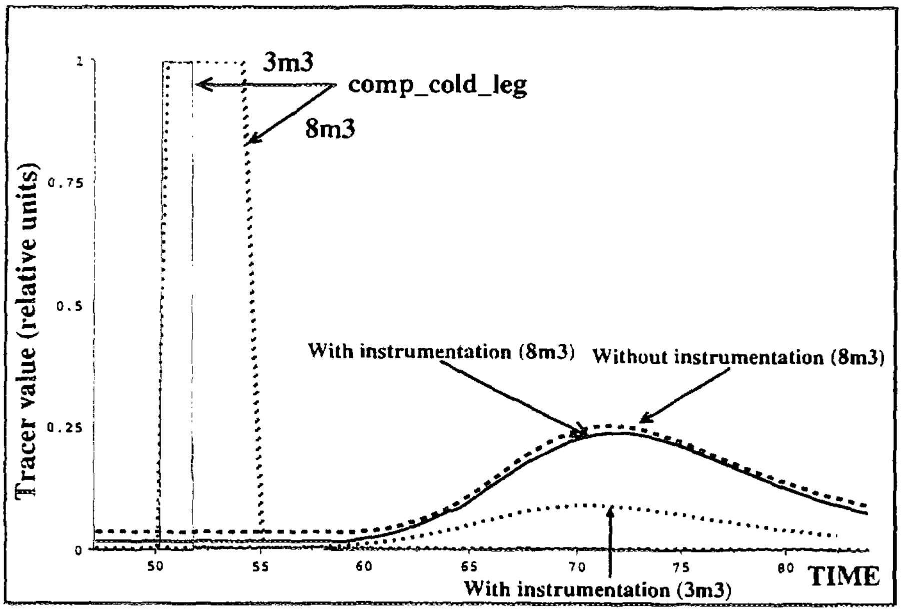


Figure 10 - Clear water plug transient mixing - Mean concentration computed at the core inlet. Sum up on the complete numerical transient studies.