Collection de notes internes de la Direction des Etudes et Recherches

CALCULS TRIDIMENSIONNELS DE L'ECOULEMENT DU FLUIDE PRIMAIRE DANS LA CUVE D'UN REP 900 MW. SIMULATION NUMERIQUE DE LA COURBE REELLE DE DEBIT INDUITE [illegible] DEMARRAGE DU GMPP

THREE DIMENSIONAL CALCULATIONS OF THE PRIMARY COOLANT [illegible] IN A 900 MW PWR VESSEL. NUMERICAL SIMULA[illegible] THE ACCURATE RCP START-UP FLOW RATE

EDF
Electricité de France

DIRECTION DES ÉTUDES ET RECHERCHES
SERVICE APPLICATIONS DE L'ELECTRICITE ET ENVIRONNEMENT
DÉPARTEMENT LABORATOIRE NATIONAL D'HYDRAULIQUE

EDF
Electricité de France

Juin 1997

MARTIN A.
ALVAREZ D.
CASES F.
STELLETTA S.

CALCULS TRIDIMENSIONNELS DE L'ECOULEMENT DU FLUIDE PRIMAIRE DANS LA CUVE D'UN REP 900 MW. SIMULATION NUMERIQUE DE LA COURBE REELLE DE DEBIT INDUITE PAR LE DEMARRAGE DU GMPP

THREE DIMENSIONNAL CALCULATIONS OF THE PRIMARY COOLANT FLOW IN A 900 MW PWR VESSEL. NUMERICAL SIMULATION OF THE ACCURATE RCP START-UP FLOW RATE

Pages : 10

97NB00085

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

ISSN 1161-0611

SYNTHÈSE :

Ce document présente les derniers résultats autour des mélanges dans les cuves de REP 900 MWe. On y met en évidence la représentation du transitoire réel envoyé dans la cuve lors du démarrage d'un GMPP.

Dans un premier temps, nous présentons le code de thermohydraulique aux éléments finis N3S utilisé lors des calculs tridimensionnels.

Puis, nous donnons les résultats obtenus pour un cas de fonctionnement du réacteur. Ce cas concerne le mélange en régime transitoire d'une poche d'eau claire dans la cuve lors du démarrage d'une pompe primaire. Une comparaison est effectuée entre les deux modes d'injection de cette poche ; configuration stationnaire de l'écoulement ou représentation exacte de la rampe de débit provoquée par la mise en route d'un GMPP. Nous comparons les résultats numériques donnant la concentration minimale et l'évolution temporelle de la concentration moyenne en entrée du cœur. Les résultats montrent l'importance des caractéristiques instationnaires de l'écoulement lors du transport de la poche d'eau claire.

EXECUTIVE SUMMARY :

This report explains the last results about the mixing in the 900 MW PWR vessels. The accurate fluid flow transient, induced by the RCP starting-up, is represented.

In a first time, we present the Thermalhydraulic Finite Element Code N3S used for the 3D numerical computations.

After that, ~~we give the~~ results obtained for one reactor operation case are given. This case is dealing with the transient mixing of a clear plug in the vessel when one primary pump starts-up. A comparison made between two injection modes ; a steady state fluid flow conditions or the accurate RCP transient fluid flow conditions. ~~We compare~~ The results giving the local minimum of concentration and the time response of the mean concentration at the core inlet are compared. The results show the real importance of the unsteadyness characteristics of the fluid flow transport of the clear water plug.

97NB00085

THREE DIMENSIONNAL CALCULATIONS OF THE PRIMARY COOLANT FLOW IN A 900 MW PWR VESSEL. NUMERICAL SIMULATION OF THE ACCURATE RCP START-UP TRANSIENT FLOW RATE.

A. Martin, D. Alvarez, F. Cases, S. Stelletta

Département Laboratoire National d'Hydraulique
Électricité de France
Chatou
FRANCE

INTRODUCTION

Pointed out by the 1986 French PWR Safety studies and by a 1995 OCDE/NEA Specialist Meeting, a potential risk of a reactivity accident may exist in association with boron dilution events and a poor degree of mixing of the diluted fluid in a PWR vessel (Figure 1).The diluted slug transport and mixing in the vessel when a Reactor Coolant Pump starts-up is the most significant case.

A Research and Development action has been started in 1991 and the program aiming at an increased knowledge of vessel thermalhydraulics consists in two complementary approaches based on experiments performed on a BORA BORA 0.2 scale mock-up (Fig. 2) and numerical computations (Fig. 3) using the EDF's Thermalhydraulic Finite Element Code N3S (Alvarez, 1991). In the two approaches, the exact geometry of a 900 MW CPY vessel was taken into account as well as possible like the plates and instrumentation columns of the lower part of the vessel.

Up to now we have published results on a validation case consisting of comparisons between experiment results and numerical computations of the isothermal steady state fluid flow mixing in the vessel for a normal reactor operation when the three loops mass flow rate balanced. Then the mock-up and the N3S code were used for the plug mixing study in the vessel taking into account a steady state fluid flow rate in the vessel, only the transient mixing of the plug was investigated (Alvarez, 1994), (Martin, 1996) et (Alvarez, 1995). The effect of the RCP start up transient flow rate on the plug mixing was not taken into account in this study.

This paper presents the CPY PWR plant and the N3S code developed under EDF Quality Assurance Policy. Then the scenario of the diluted plug is discribed. The clear water plug was previously made up in the primary coolant intermediate loop and then shot into the vessel when the RCP starts up. The accurate RCP transient flow chart and the time history of the diluted plug is also given. The N3S numerical simulation of this scenario and the results are given in details. We also compare these new results with the previous one's performed without the transient behaviour of the RCP start up flow rate. This comparison given in terms of the plug mixing in the downcomer and boron dilution maps at the core inlet shows the importance of the RCP transient flow rate behaviour. The mixing is better with a RCP steady state flow rate than with the accurate RCP transient flow rate.

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 Figure 1 for the reactor and the domain concerned). The core itself is not simulated and the hot legs are only represented by their outer diameter in the downcomer.

The BORA-BORA mock-up

The BORA-BORA mock-up was designed in 1991 and has been running since march 1992. The three cold legs, the outer vessel itself, the lower plenum instrumentation columns and plates are made of Perspex for flow fields visualisations and for further Laser Doppler Velocimetry measurements (Fig 2).

For the hydraulic experiments, the primary flow mixing characteristics are studied at the core inlet in the mock-up close to the lower core plate by means of temperature tracking. For the steady state fluid flow mixing tests the temperature of one of the cold legs is warmed 10°C hotter than the others. For the transient discharge study of a clear plug in the vessel, the plug is simulated by a warmed water plug.

The vessel mixing characteristics are determined between the cold legs and the core inlet plate. A temperature rake gives the reference temperature several diameters before the cold leg elbow and 60 sensors regularly spaced measure the temperature at the core inlet. The sensors used in the experiments are 0.25 mm thermocouples with a 7 ms measured time response. The calibration of the sensors has been performed with an oil bath leading to a 0.1°C accuracy. By computing the total energy balance between the inlet and the outlet of the vessel gives the global accuracy of a run. A run is taken into account only when this global accuracy does not exceed 10%.

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 taking advantage of our experience acquired on 3D finite difference codes. N3S's development started in 1982 with as main feature 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 (Chabard, 1992) with or without heat transfer (Delenne, 1993). For code assessment, a wide range of computer program validation are made under a Quality Assurance procedure for every main release of N3S. Code results are compared with analytical solutions when available or with literature experiments (Chabard, 1993). Computations of benchmark exercises for international numerical workshops constitute a other kind of code validation (Leal De Sousa, 1994).

For solving our study, we used the N3S package which 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.

N3S solves the Reynolds averaged Navier-Stokes equations for an unsteady incompressible or compressible flow, with a constant time step or a space or time variable time step. Different turbulence models can be used but in our computation, we use the standard k-ε two equations turbulence model beeing given the high level of turbulence in our study. For buoyancy driven flows with small temperature differences, the momentum equation is coupled with the energy equation using Boussinesq approximation. To take into account more important thermal effects we consider the Navier-Stokes equations in which density depends on temperature : $\rho = \rho(T)$. The time discretization is based on a fractional step method (Caruso, 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, 1992),
- a diffusion step on scalar variables,
- a generalized Stokes problem for the velocity and the pressure, solved either by a Chorin-Teman 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 (Bidot, 1992).

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. All specific functionalities were developed according to EDF specification.

For the application to the CPY PWR, P1-isoP2 tetrahedrons are used for efficient turbulence solutions (Pot, 1993). The preprocessing task led to one mesh : 361911 P2 nodes and 237801 elements with the plates and instrumentation columns (Figure 3).

BORON TRANSIENT EXPERIMENT AND COMPUTATION

The case of 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 when entering the vessel has a zero boron concentration. The remaining of the primary coolant fluid has an initial value of 2000 ppm boron concentration. The primary fluid flow characteristics involved by the RCP start-up is shown in Figure 4. This flow chart was accurately measured in a EDF's full scale qualification mock-up specialy designed for the characterisation of the first off Reactor Coolant Pumps. On Figure 4, V1 represents the fluid volume between the pump succion zone and the vessel inlet and V2 represents a 8 m^3 clear plug volume.

In our firsts approaches (Alvarez, 1994), (Martin, 1996) et (Alvarez, 1995), we have made the assumption that when the plug driven back by the RCP reaches the vessel inlet, the fluid flow has well established turbulent mixing characteristics. So in our previous studies we have considered only the transient mixing of the plug, between the vessel inlet and the core inlet, at a constant mass flow rate of 1450 kg s^{-1} (see Figure 4 for caption). In this new approach, we have drawn back the assumption taking into account that the accelerating fluid flow may change the turbulent mixing process. So we have simulated the real transient fluid flow and the results are presented.

Figure 5 shows the comparison between the BORA-BORA mock-up results and the N3S computations for the steady state flow rate mixing (Martin, 1996) et (Alvarez, 1995). In the experiments the clear plug mixing is investigated by a temperature tracking method between the cold leg number two and the core inlet. In the computations a passive tracer method is used. In the cold leg, a tracer is prescribed for a time corresponding to the entrance of the clear water plug. In both cases the plug transport is obtained with a steady state flow in the vessel. Figure 5 shows a good agreement between experiments and computations (Alvarez, 1995).

The following Figures concern the computational results on the simulation of the accurate RCP start-up fluid flow that we compare with our previous results. Figure 6 shows the mixing of the plug in the downcomer for two time steps, the first one when the plug reaches the downcomer and the second one when the tail of the plug appears. Figure 7 shows the concentration of the unborated water when reaching the core inlet plate at his minimum value. Related with this figure, Figure 8 shows the concentrations time histories of the reference signal (the plug in cold leg), the core inlet spatial average signal and the minimum local one. The localisation of the minimum concentration is deduced from Figure 7 and the time dependant signal shown on Figure 8 is related to this point. The graphs on Figure 8 are synchronized with respect to the reference signals. Comparing solely the concentrations levels between both cases, we show the importance of the unsteady characteristics of the fluid flow transport of the clear plug.

CONCLUSIONS

In this paper, the last results available on boron dilution reactivity transients in PWR's are presented. These new computational results show that we have to simulate the accurate Reactor Coolant Pump fluid flow characteristics when studying the mixing of an unborated plug shot in the reactor vessel when the RCP starts-up.

However an important work still must be done to qualify the minimum concentration levels obtained in the computational simulation. For this purpose, the BORA-BORA mock-up is being transformed to simulate the time flow chart of the RCP. In an other way, modelling the fluid flow motion starting from the rest with a standard k-ε turbulence model is insufficiently validated. So, a computational validation task is also scheduled for this purpose.

REFERENCES

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.

D. Alvarez, A. Martin, J-P Schneider, "Boron mixing transients in a 900 MW PWR vessel for a Reactor start-up operation." Proc of the NUTHOS-4 meeting, Taïpei, Taïwan 1994.

A. Martin, D. Alvarez, F. Cases, "Three dimensional calculations of the primary coolant flow in a 900 MW PWR vessel. Steady state ans transients computations.", Proc. of the ICONE-4 meeting, New Orleans, USA 1996.

D. Alvarez, F. Cases, A. Martin, S. Stelleta, "Boron mixing transient in a PWR vessel. Physical studies." Proc. of OCDE/NEA specialist meeting, to appear in a NUREG report, Penn State College, USA 1995.

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.

B. Delenne, G. Pot, "Modelling of the turbulent thermal boundary layer implemented in CFD code N3S and applications to flows in heated rooms", 8th Int. Conf. on Numerical Methods for Thermal Problems, Swansea, U. K.,12-16 July 1993.

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.

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.

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.

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.

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.

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

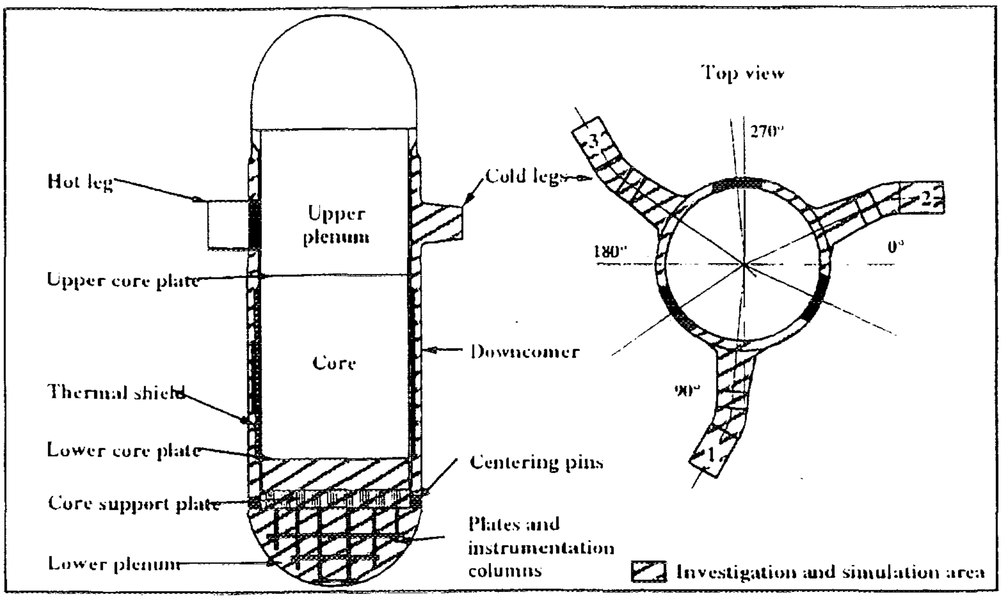

Figure 1 - 900 MW CPY PWR geometry.

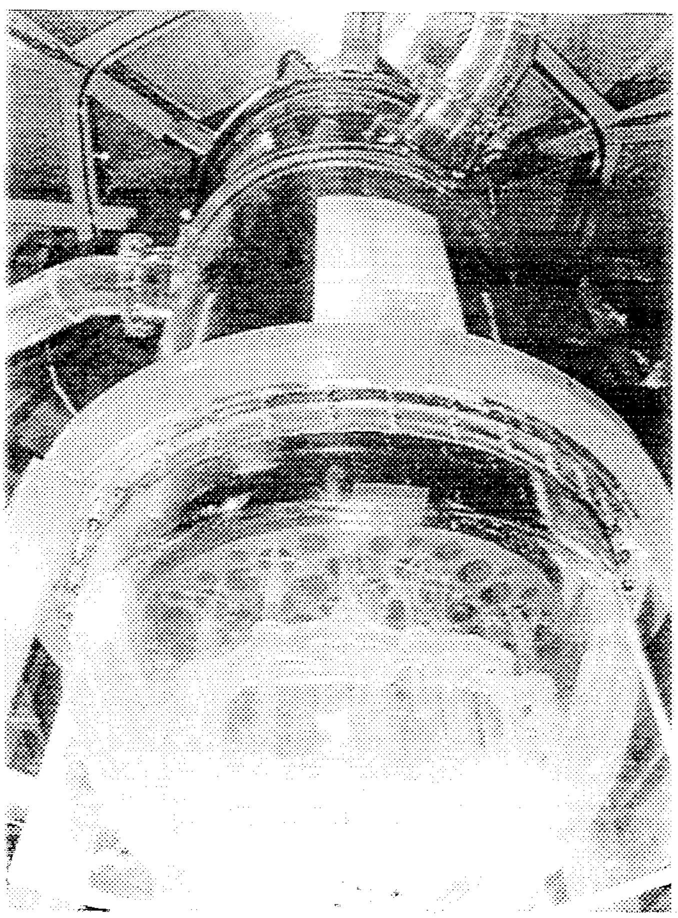

Figure 2 - BORA-BORA mock-up seen from below

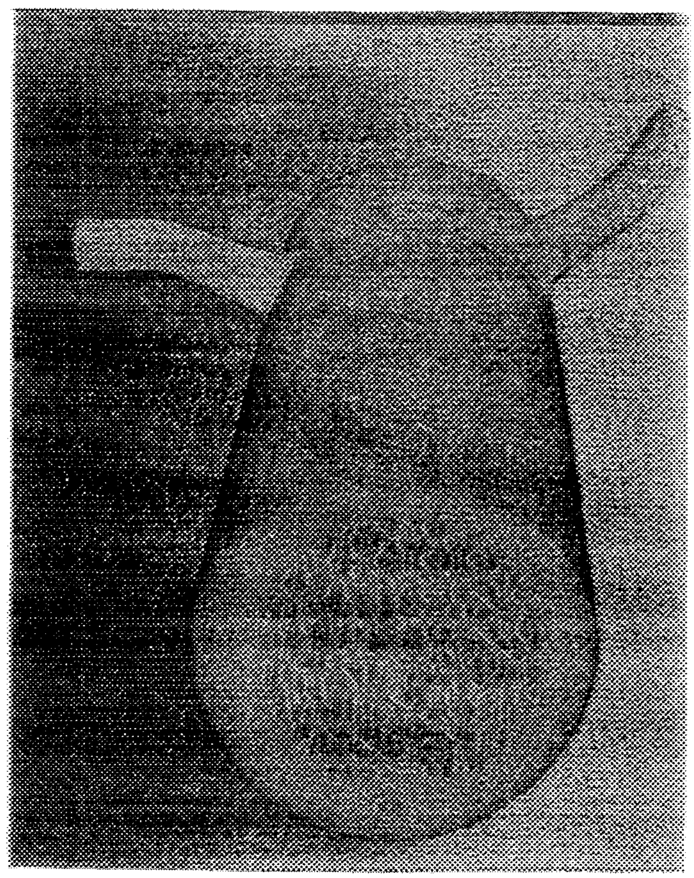

Figure 3 - Finite Element mesh seen from below

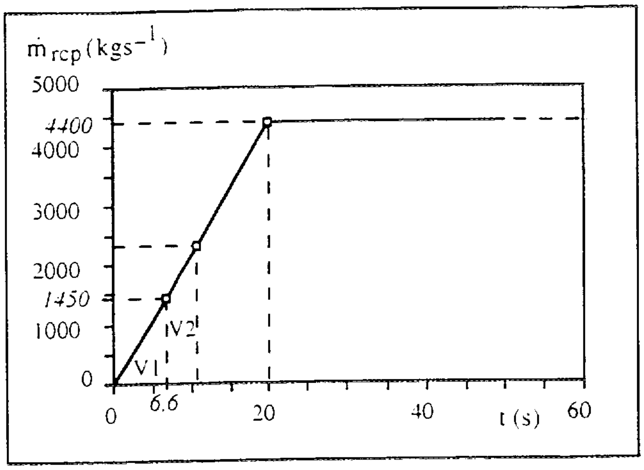

Figure 4 : Accurate Reactor Coolant Pump start-up fluid flow time history.
V1 : volume between the pump succion and the vessel,
V2 : 8 m^3 clear plug volume

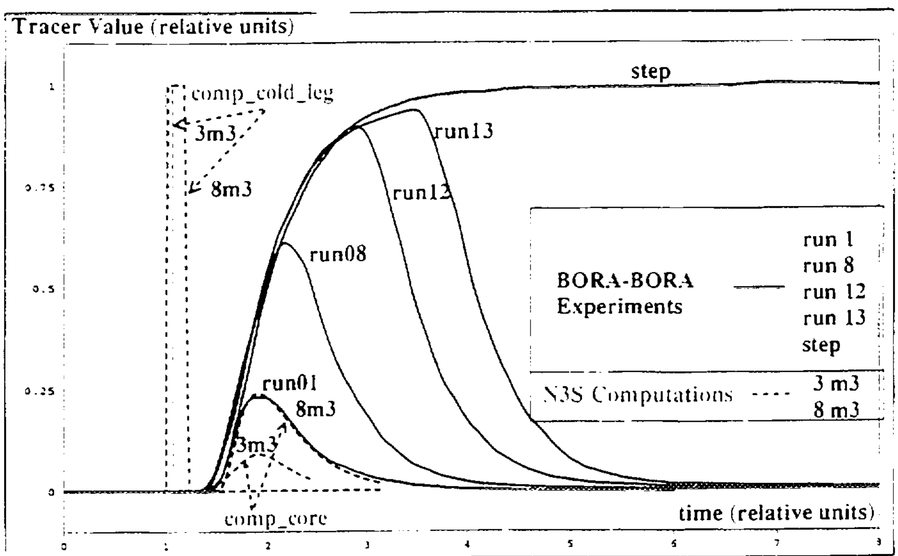

Figure 5 : Clear water plug transient mixing with steady state fluid flow conditions.
Mean concentrations measured and computed at the core inlet.

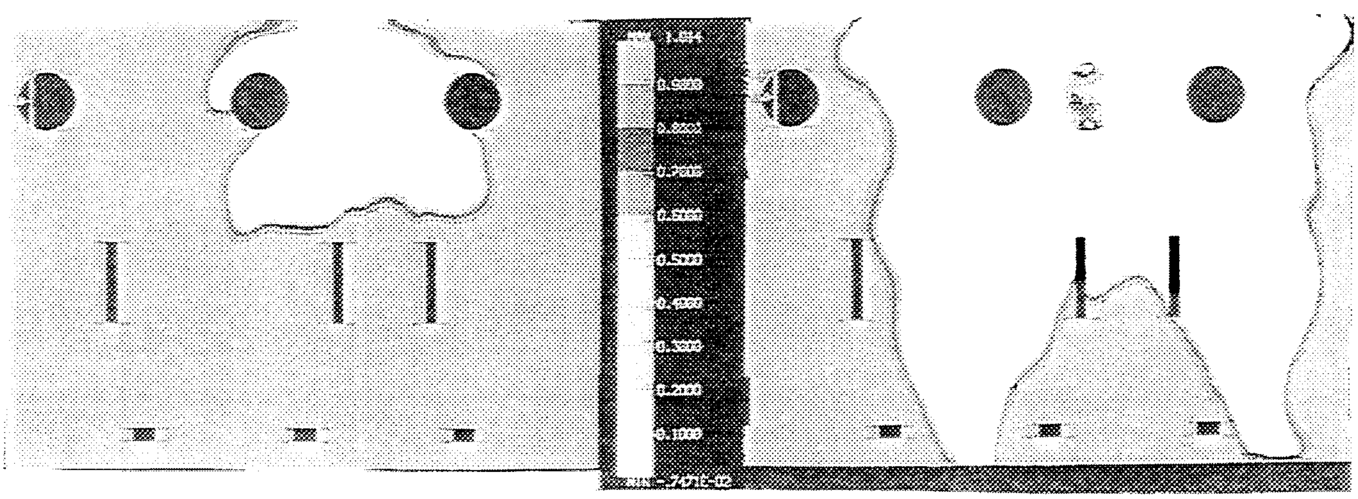

Figure 6 : 8 m^3 clear water plug transient mixing with the RCP transient fluid flow conditions.
Concentration maps of the tracer in the downcomer computed by N3S for two different time steps.

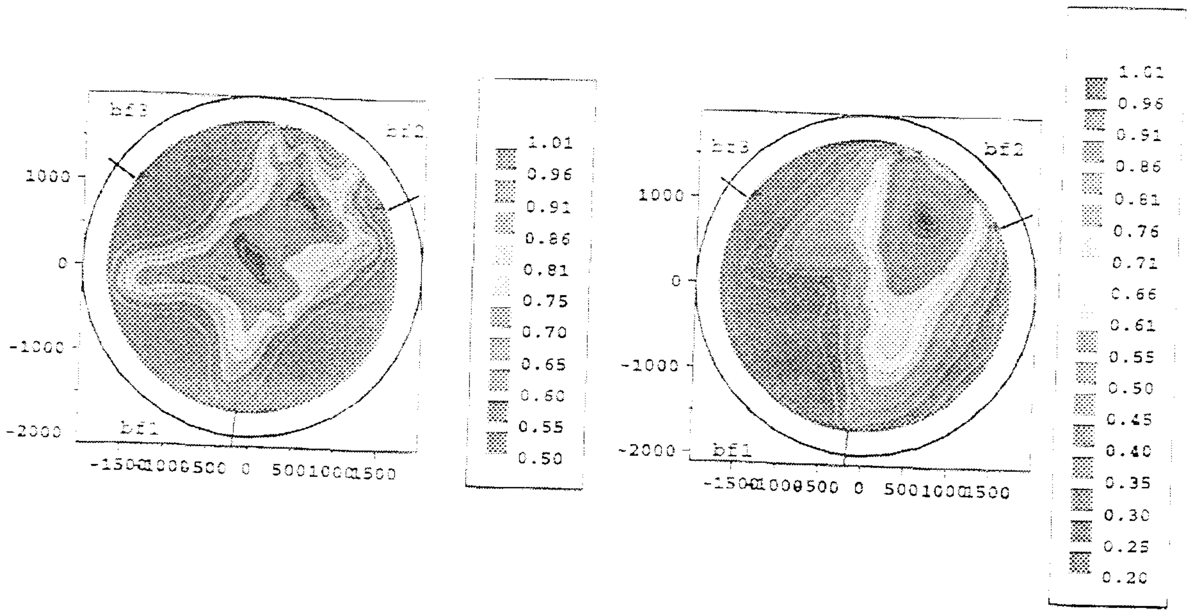

Figure 7 : 8 m^3 clear water plug transient mixing.
Concentration maps of the tracer at the core inlet computed by N3S.
Left : with steady state fluid flow conditions.
Right : with the RCP transient fluid flow conditions.

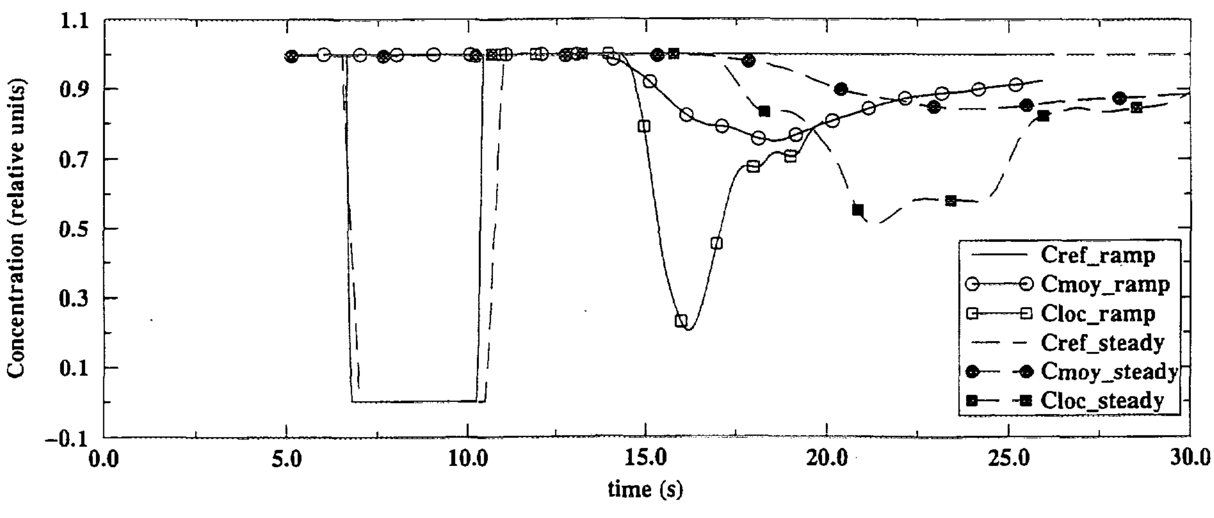

Figure 8 : 8 m^3 clear water plug transient mixing.
Concentration time histories of the tracer computed by N3S.
Subscript ref : concentration in the cold leg.
Subscript moy : spatially averaged concentration at the core inlet.
Subscript loc : minimum concentration at the core inlet.
steady : with steady state fluid flow conditions.
ramp: with the RCP transient fluid flow conditions.