Proceedings of PVP2002
2002 ASME Pressure Vessels and Piping Conference
August 5-9, 2002, Vancouver, BC, Canada

PVP2002-1461

CFD-Tool for Thermal-Hydraulics Pressurised Thermal Shock analysis. Qualification of the *Code_Saturne*.

A. MARTIN[1], S. BELLET[2]

1 EDF/Research and development Division/Fluid Mechanics and Heat Transfer.
6, Quai Watier - B.P. 49 - 78401 CHATOU CEDEX 01 - FRANCE
Tel : 33-1-30877622 Fax : 33-1-30878086 Email : Alain-cc.Martin@der.edf.fr

2 EDF/ Industry – Basic Design Department.
12-14, Avenue Dutrievoz - 69628 VILLEURBANNE CEDEX – France

Abstract

This paper explains the numerical program concerning the new thermalhydraulic *Code_Saturne* qualification for Safety Injection studies. Within the frame of the plant life time project, an analysis has shown that the most severe loading conditions are generated by a pressurised injection of cold water in the downcomer of a Reactor Pressure Vessel. For this kind of transients, a thermal hydraulics study has to be carried out in order to better adjust the accurate distribution of the fluid temperature in the downcomer. For that, the numerical tools have to be able to simulate the physical phenomena present during the Pressurised Thermal Shock. (PTS). For this qualification task, we have investigated one configuration related to an injection of cold water particularly in cold leg but also in a downcomer. One experiment test case has been studied and this paper gives a comparison between experiment and numerical results in terms of temperature field.

Introduction

Within the frame of the plant lifetime project, the assessment of the French Reactor Pressure Vessel (RPV) integrity has been performed according to a specific approach derived from the codified fast fracture analysis (RCCM code, ZG appendix), based on a selection of subclad defects and a set of loading transients. This analysis has shown that the most severe loading conditions are given by the small break loss of coolant accidents due to the pressurised injection of cold water into the downcomer of the RPV. For these PTS transients, a thermal hydraulics analysis has to be carried out in order to better adjust the distribution of fluid temperature in the downcomer and the heat transfer coefficients on the inner RPV surface. In this scenario, the thermal loading generated induces a strong loading on the vessel (cold shock). A thermalhydraulic analysis constitutes a new approach which can lead to a better knowledge of the thermalhydraulic conditions on the cooling of the vessel. Before beginning reactor computations, a qualification phasis is necessary to show the *Code_Saturne* capabilities to simulate the physical characteristics present in this kind of scenarii. This paper reports the thermalhydraulic studies performed with *Code_Saturne*. The geometrie used and described consists of a EPRI-Creare 1/5 scale mixing facility, which has a geometry type of PWR that includes one cold leg, a planar downcomer with or without an internal thermal shield and an High Pressure Injection.

This paper presents the CREARE facility, the *Code_Saturne* software developed under E.D.F Quality Assurance Policy and also, the Star CD code used by Framatome ANP and used here for the comparison between numerical tools in addition of the experiment results. After that, the numerical simulation concerns firstly, the transport and the mixing of cold water (Safety Injection) in a hot environment through the cold leg and the downcomer. Secondly, experiment results are available with different probes located mainly in the cold leg (stalks are used to analyse the thermal layers) but also just at the beginning of the downcomer, area directly concerned by the thermal loading interesting in the safety studies. In fact, some probes are located respectively on the external wall (representative of the vessel wall) and on the internal wall (representative of the core barrel). Globally, we analyse the thermal behaviour of a cold leg and a downcomer in its fluid part. The fluid behaviour can be assess and in a same time, the numerical tools can be compared with the experiments results. This task is used to qualify the numerical tools, used in the frame of the plant lifetime project.

The EPRI/CREARE geometry

Figure 1 shows the geometry of the EPRI/CREARE (NUREG/CR-3822, EPRI) facility at 1/5 scale.

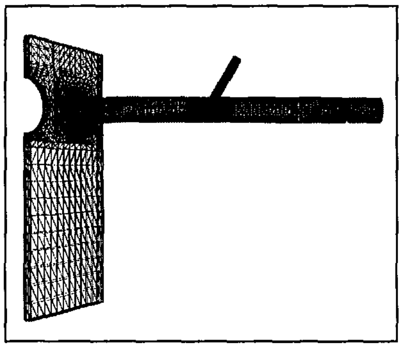


Figure 1 : Creare geometry

It's a transparent acrylic facility designed for operation near atmospheric pressure. The downcomer is represented by a planar section having width and height comparable to a 90° sector of a reactor downcomer. The downcomer includes certain internal geometries such as the hot leg only represented by the space required by it in the downcomer geometry. The vessel wall is flat and a thermal shield is implemented below the cold leg nozzle and spans the full width of the 90° downcomer section. In the experiment and computation comparison, we used the Creare geometry with and without thermal shield. The High Pressure Injection is modelled on the cold leg with an 60° angle. Experiment data are available end for our comparison, they are issued mainly from thermocouples located in the cold leg and in the downcomer close to the vessel wall and the core barrel.

The *Code_Saturne* and Star CD codes

The Code_Saturne code

In close relation with the developments carried out within the two in-house thermohydraulics codes N3S-EF (A. Martin and al, 2000 and 2001) and ESTET-ASTRID, EDF embarked in 1996 on a program aiming at unifying the potentialities of the two products within the same CFD package: Code_Saturne. Indeed, N3S-EF takes advantage from Finite Elements Techniques for complex geometries while ESTET-ASTRID provides refined physical modelling on a structured finite volumes mesh, particularly for two-phase flow.

Code_Saturne 1.0 is well suited for 2D, 2D axi-symmetric and 3D calculation of steady or unsteady single phase laminar and turbulent flows (with classical approaches such as k-epsilon and second order turbulence models). This CFD package is coupled to several modules such as Lagrangian approach, radiative transfer module (grey gas) and the SYRTHES code for conjugate heat transfer between fluid and solid. The numerical basis of Code_Saturne is a Finite Volume approach associated with reconstruction to evaluate on strongly non-orthogonal meshes. For incompressible flows, a centred scheme is used for diffusion terms, and convection is addressed by upwind, second-order upwind, centred or other classical second order techniques. For multi-phase flows, two major approaches are investigated : the first one based on the ASTRID algorithm, while the second one is inherited from hyperbolic methods coupling mass, momentum and turbulent kinetic energy. Code_Saturne supports any kind of elements (tetrahedra, hexahedra, prism, pyramids...) in any kind of numerical mesh (conform or not, hybrid,...).

The STAR CD code

The computation code named STAR CD (version 3.) has been applied by Framatome ANP to various thermal hydraulics studies based on a 3D finite volume type meshing (Cochet, 1999). This code is qualified for a wide range of incompressible laminar and turbulent flows with or without heat transfer. This code solves the Reynolds averaged Navier–Stokes equations for steady and unsteady incompressible flow with k-epsilon turbulence model. The time discretization is based on a PISO method.

In order to check the validity of the code for cold water injection into the downcomer, a specific qualification has been performed by Framatome ANP on experimental data from representative experiments of the physical phenomena involved during the SBLOCA (thermal layers, high-density injection, thermal transfer between fluid and structures).

Analysis of the main local physical phenomena

Knowledge of fluid mixing is interesting in order to analysis of plant overcooling transients where coolant water is injected at high pressure and hence pressurised thermal shock at the reactor vessel is theoretically possible. This paper presents the results leading us to better master the local physical phenomena at the key points. This approach enables us to take into account the fluid mixing during its transport along cold legs and downcomer and confirms the code capabilities to simulate correctly the thermal layers and the fluid separation in the downcomer.

As final result, these codes have to be assessed as being adapted to simulate realistically PTS transients. In the CREARE test case, we injected cold water into stagnant primary water with a zero mass flow rate through the loop. For a best examination of data, thermocouples or probes have been located in key locations and more particularly along and on all height of the cold leg and at the beginning of the downcomer, under the cold leg, close to the vessel wall and the core barrel.

In a first step, the fluid physical phenomena are presented in cold legs and into the downcomer for the *Code_Saturne* code. After that, the experiment and numerical results are presented in term of temperature values given by the different thermocouples (*Code_Saturne* and Star CD). These data are used for the experiment and numerical comparison.

Initial and Boundary Conditions for CREARE test case.

Creare Initial conditions

At the beginning of the computation, we consider the pressure at 1 bar. The table below sums up the initialisation used into the CREARE geometry in term of temperature. The considered value of the mass flow rate is zero.

	Creare Mix 2
Temperature	65,5°C

The fluid physical properties are taken at the pressure 1 bar.

Creare Boundary conditions

The studies take into account the simulation of the Safety Injection by HPI (High Pressure Injection). The cold temperature is taken into account by these injections. The table below sums up the boundary conditions used in the three simulations in term of temperature and velocity injection.

	Creare Mix 2
Temperature	17.8°C
velocity	0.125m/s

Fluid results in the cold leg and into the downcomer

The main physical phenomena are located in the cold leg and in the area of the downcomer the most interesting thermal loading point of view. Close to the cold water injection, this cold water causes the appearance of thermal layers induced by density effects. These effects are generated by the difference between the water injection in the nozzle (about 17°C) and the initial temperature (66°C). During the thermal transient, the physical phenomena present in the cold legs have a real influence on the local fluid behaviour in the downcomer. Figure 3 shows the development of the thermal layers in the cold legs .

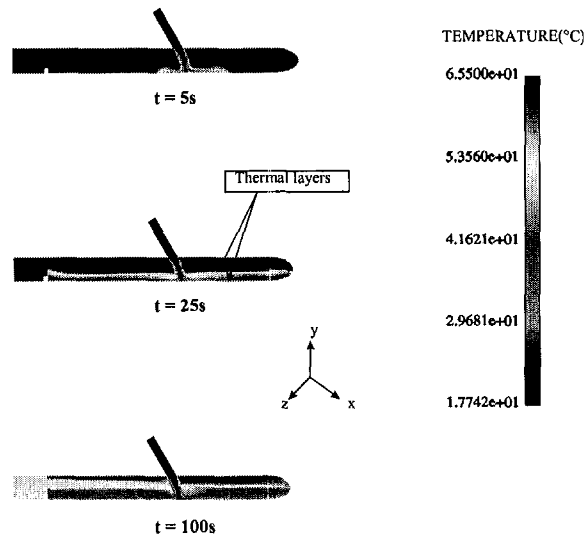
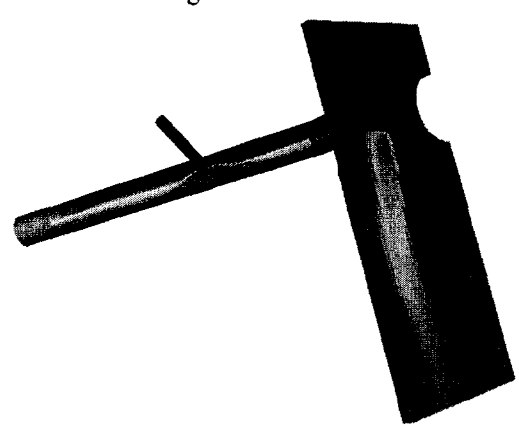
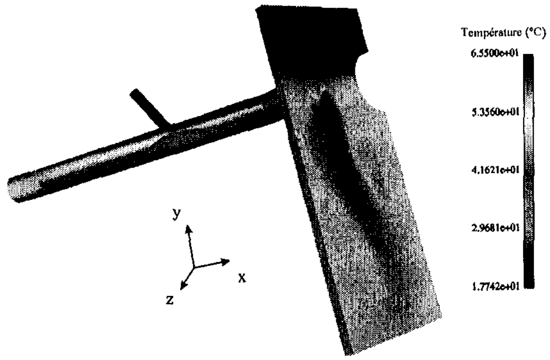


Figure 3 : Thermal layers for CREARE test case

In fact, a cold stream flows along the bottom of the cold leg and a hot stream flows along the top of the cold leg countercurrent to the cold stream. Figure 4 shows the cold stream from the cold leg penetrated into the downcomer as a plume which fluctuates during the transient.



t=45s



t=100s

Figure 4 : General view of flow behaviour

Temperature measurements in the downcomer indicate that, due to the mixing in the cold leg and at the cold leg/downcomer interface, the temperature of the fluid is of course significantly higher than the temperature injection. Figure 5 illustrates the profile and the path followed by the flow when it arrives to the connection zone (RPV/cold leg fillet) and in the downcomer.

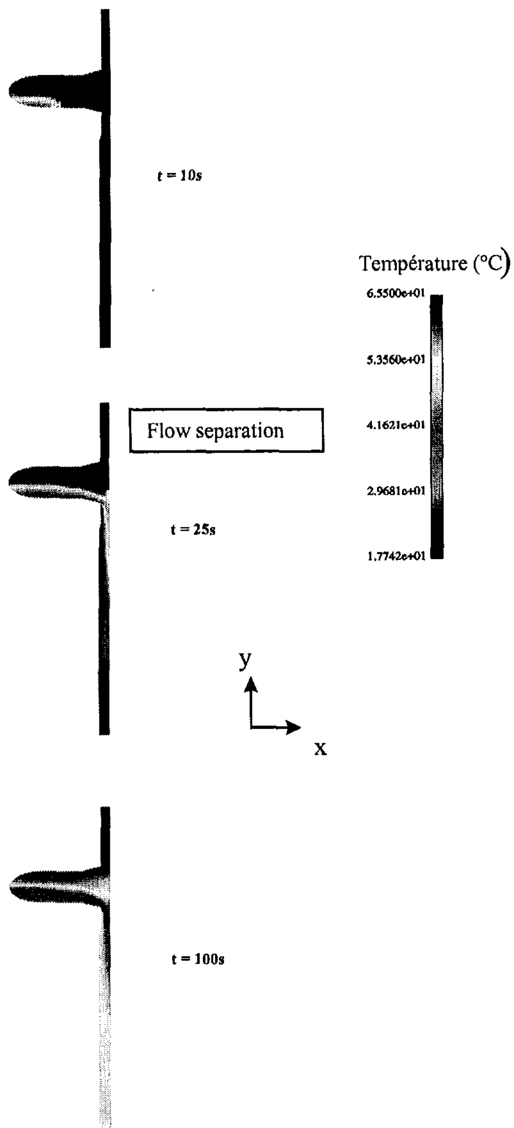


Figure 5 : Realistic local behaviour of fluid flow in the connection zone(inlet vessel nozzle)

The main result in this area concerns the fluid flow separation. This behaviour strongly depends on the fluid characteristics (velocity, thermal layers...), the recirculation phenomena in the downcomer and the fillet radius.

During the transient, the fluid fluctuates on both side of the downcomer walls. The cold fluid flows alternatively along the RPV and along the core barrel. So, about thermal loading of the RPV point of view the thermal shock for the vessel will be less hard. For the qualification task and quantification point of view, different measurements have been realised in the cold leg and into the downcomer. Figure 6 illustrates the location of the probes and stalks on the CREARE facility.

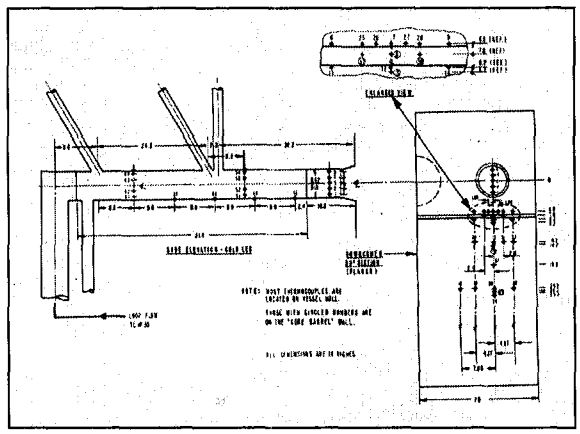


Figure 6 : CREARE Instrumentation

Figure 7 shows the good agreement between experiment and numerical results in term of temperature measurements. The thermal layers are correctly predicted with a good transition between the hot and the cold stream (probe 33). The probe 34 is located just in front of the nozzle injection. We can check the good synchronisation between experiment and numerical fluid flow.

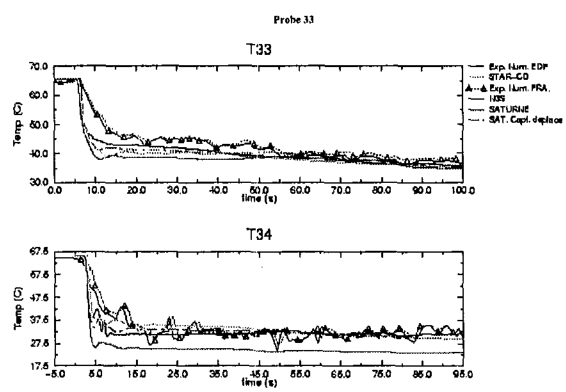


Figure 7 : Temperature probes in cold leg

Figure 8 illustrates a stalk located just before the connection with the downcomer. The results show a good agreement between numerical results and experiment values. The thermal layers are well predicted as well as in cold part than the hot part of the leg.

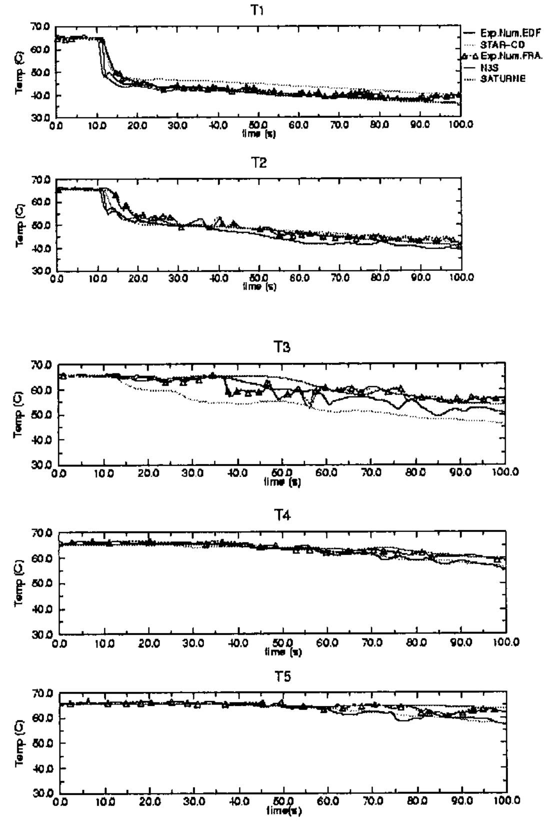


Figure 8 : Stalk with probes 1-2-3-4-5

A first result into the downcomer leads to point more particularly to the area just at the vertical below the connection zone.

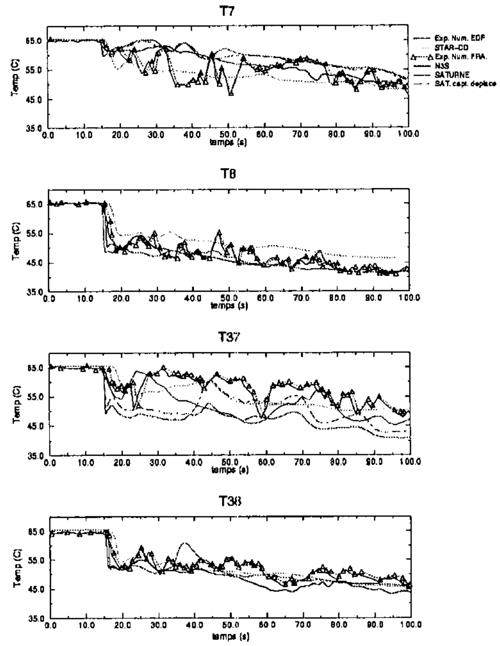


Figure 9 : Probes(Creare) 7-8-37-38

In fact, whatever the test case taken into account (Creare with or without thermal shield), two probes are systematically located here and there of the downcomer (on the internal and external wall). The probe 8 (core barrel) value is colder than the probe 7 and so, this result indicates the good prediction of the flow separation at this location. Figure 10-12 illustrates more particularly the code capabilities to simulate the local flow separation and the mixing flow which fluctuates in the downcomer.

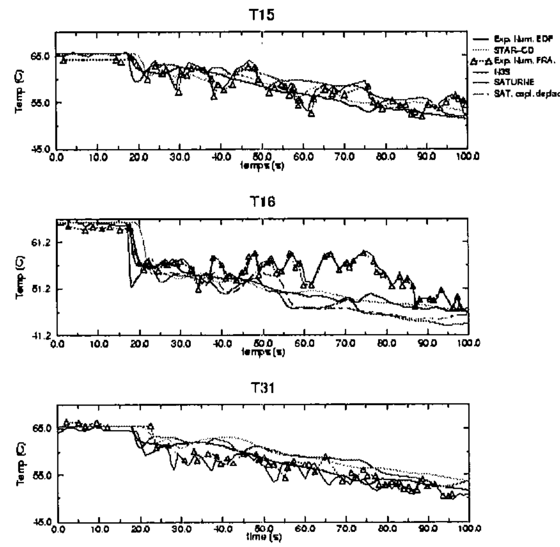


Figure 10 : Probes 15, 16, 31

On these figures, we can see the unstable characteristics of the flow which depends notably of its evolution in the cold leg. Some parameters such as the thickness or the height of the thermal layer can lead to this behaviour.

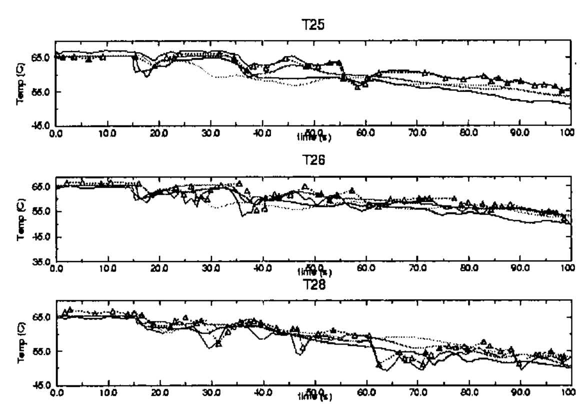


Figure 11 : Probes 26,28

This figure also shows (probes 26-28), the good prediction of the two numerical tools about experiment values located here and there of the vertical plume issued from the cold leg. This result is important because it shows the fluctuations of the plume into the downcomer and shows the code capabilities to reproduce this local physical phenomenon.

In a first time, the plume flows vertically because, there are much buoyancy effects, but progressively the plume will fluctuate with the decreasing of these effects. The figure 12 illustrates some thermocouples located in the lower part of the downcomer. In this area, the cooling is more uniform in the thickness of the downcomer.

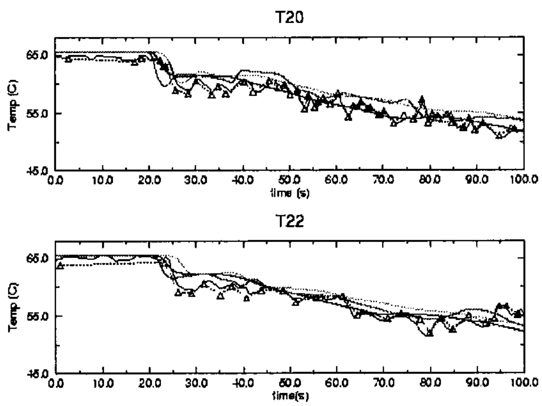


Figure 12 : Probes 20, 22

Conclusion

The 3D local thermalhydraulic computations have lead us to assess the *Code_Saturne* capabilities to represent the physical phenomena linked to the cold water injection in a hot environment. This task enters within the frame of the plant life time project. The results of the CREARE test case was available with the numerical results associated and issued from the task qualification of the Star CD code realised by Framatome ANP in this context. The different results show a good agreement. The main results are located in the connection zone into the cold leg and in the downcomer just below the cold leg. The thermal layers and the temperature levels are correctly predicted and the flow separation of the fluid along the RPV wall which is a important physical phenomenon has been in a prominent position. This qualification shows the capacity of *Code_Saturne* to simulate the local phenomena during the Safety Injection.

Acknowledgements :

The authors are grateful to Mrs S. Laurent and Mr Ph. Dolleans from Framatome ANP company for the Star CD results providing and for their precious analysis.

REFERENCES

Martin A. and al, “ Assessment of the pressure vessel integrity in PTS conditions. Thermal Hydraulic study of a safety injection in a three loop PWR plant with the Finite Element code N3S “. 2000 ASME PVP Seattle, Washington

Martin and al, “ CFD-Tool for Thermal-Hydraulics Pressurised Thermal Shock analysis.Qualification of the Finite Element Code N3S”. 20010ASME PVP Atlanta, Georgia

NUREG/NP-3822 LA-10132-MS. SOLA-PTS : A Transient, Three-Dimensional Algorithm for Fluid-Thermal Mixing and Wall Heat Transfer in Complex Geometries.

EPRI/NP-2312 Project 347-1 Creare Incorporated. Fluid and thermal Mixing in a Model of cold leg and downcomer with loop flow.

EPRI - Transient Cooldown in a Model of Cold leg and downcomer. NP 3118 - Research Project 2122-3

