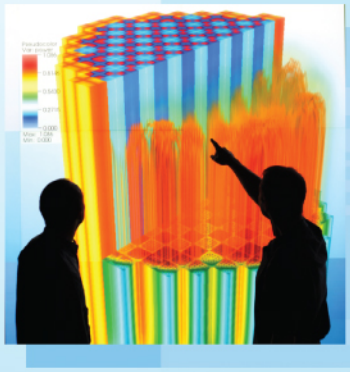
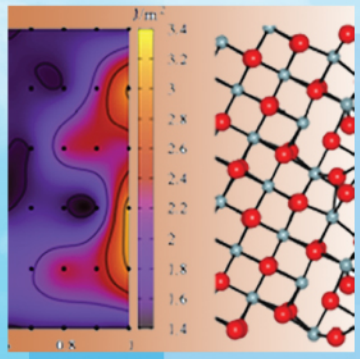
CASL-U-2011-0281-000-a

Power uprates
and plant life extension



Engineering design
and analysis

Science-enabling
high performance
computing



Fundamental science

Plant operational data

Consortium for Advanced Simulation of LWRs

L3:THM.CFD.P3.03

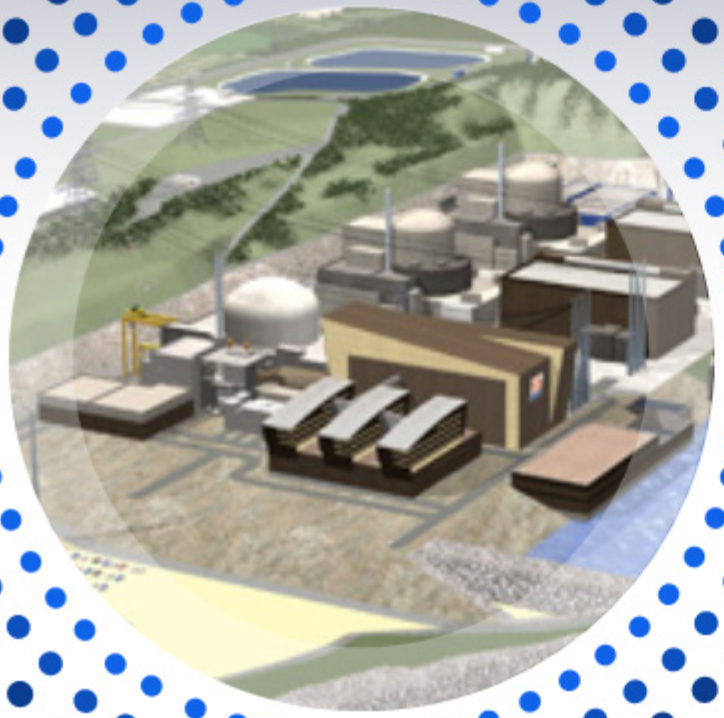
Presentation: NEPTUNE Multi-phase Local Computational Fluid Dynamics (CFD)

Robert Lowrie
Los Alamos National Laboratory

August 24, 2011

U.S. DEPARTMENT OF ENERGY | Nuclear Energy

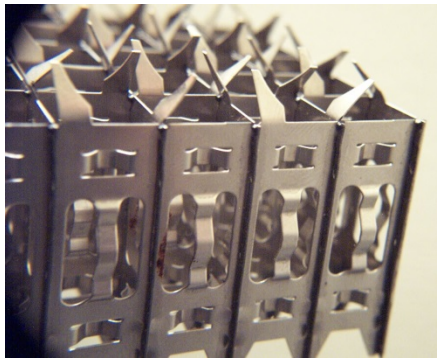
CASL-U-2011-0281-000-a

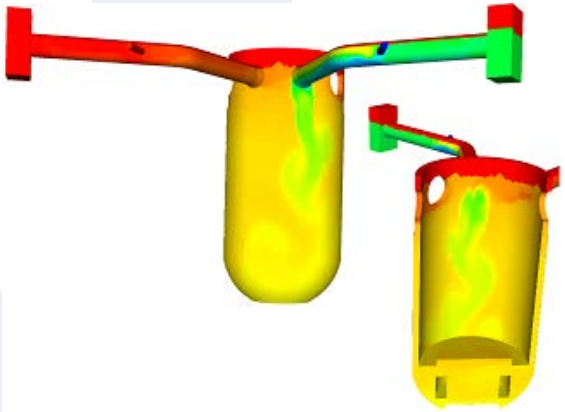


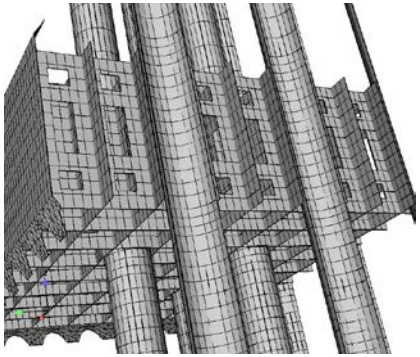
NEPTUNE_CFD Multi-phase local CFD

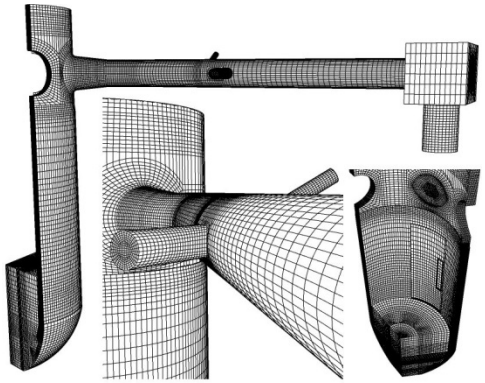
Outline

- Local two-phase thermal-hydraulics challenges
- Development framework: NEPTUNE project
- Overview of NEPTUNE_CFD
- Some applications
- Perspectives







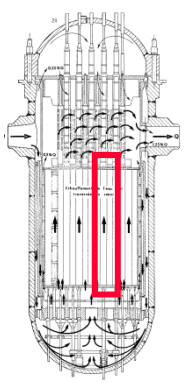


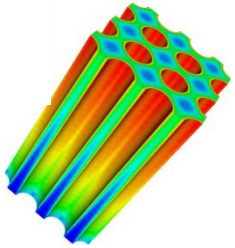
Local two-phase TH challenges

- Nuclear thermal-hydraulics
  - Involved in many design, competitiveness and safety issues of NPP
- Component and system scales
  - Improved models and closure terms, validation against new experiments
  - Extension of 3D and coupling capabilities (e.g. CATHARE 3)
- The advent of CMFD
  - Improved knowledge of local flow phenomena is a key issue for numerous design and safety issues
    - Fuel efficiency: with respect to Critical Heat Flux, design of spacer grids, CHF occurrence itself (thermal mixing, wall-to-fluid transfer, CHF prediction)
    - Pressurized Thermal Shock, involved in plant life-extension (mixing in stratified two-phase flow)
    - Many others: SG Tube vibrations, CRUD deposition, cavitation,…

Major challenges: CHF in PWR

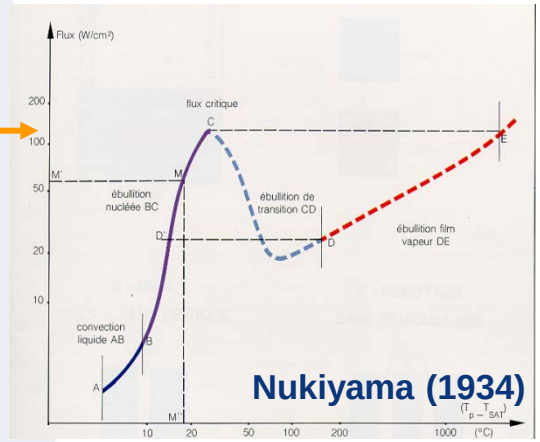
- Critical Heat Flux: value of heat flux leading to fuel rod damage
  - **DNB occurs: Departure from Nucleate Boiling**
  - **A vapour film isolates the fuel from the water: the fuel heats up sharply and suddenly**



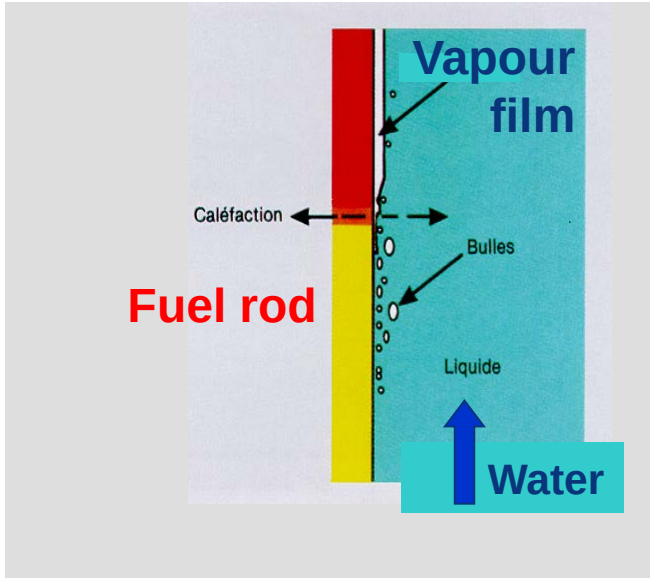


Heat Flux

CHF



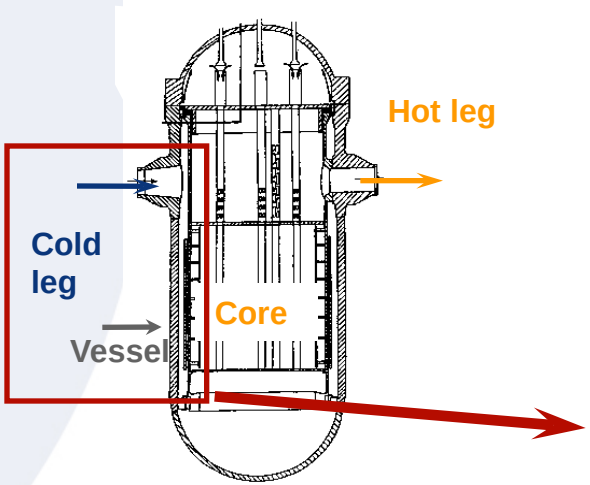
Twall - Tsat

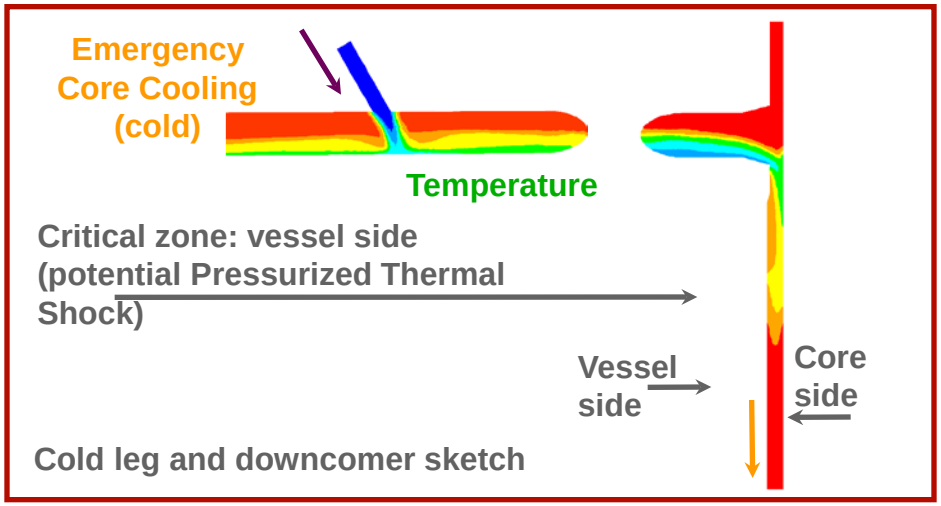


- Basic physical mechanisms not yet really understood
- **→ Get reliable CFD simulations for boiling bubbly flows in fuel assemblies**

Major challenges: PTS in PWR

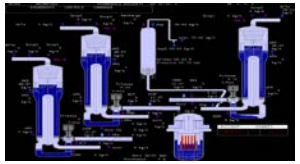
- Overview of PTS (Pressurized Thermal Shock) issue
  - In case of a LOCA, safety injections feeds cold legs and vessel downcomer with (really) cold water
  - The induced thermal shock is a risk for vessel mechanical resistance, increased by material ageing (fluence)
  - Local temperature evolution in the pressure vessel wall is a key parameter for the structural analysis





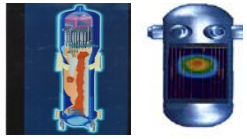
The NEPTUNE project

- The NEPTUNE project: EDF/CEA/IRSN/AREVA-NP joint development programme since 2001
  - **To meet the industrial needs in the field of nuclear thermal-hydraulics**
  - **By preparing the new generation of industrial two-phase flow codes**
- The objective of the software platform is to address:
  - **The whole range of modelling scales**
  - **Multi-scale and multi-disciplinary calculations**
  - **Industrial studies as well as R&D work**
- WP structure
  - **Software development**
  - **R&D: models and numerical methods**
  - **Experimental programmes: validation**

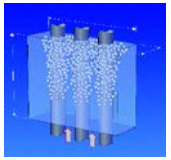


SYSTEM SCALE

CATHARE 3

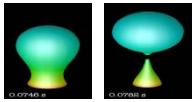


COMPONENT SCALE



C(M)FD SCALE

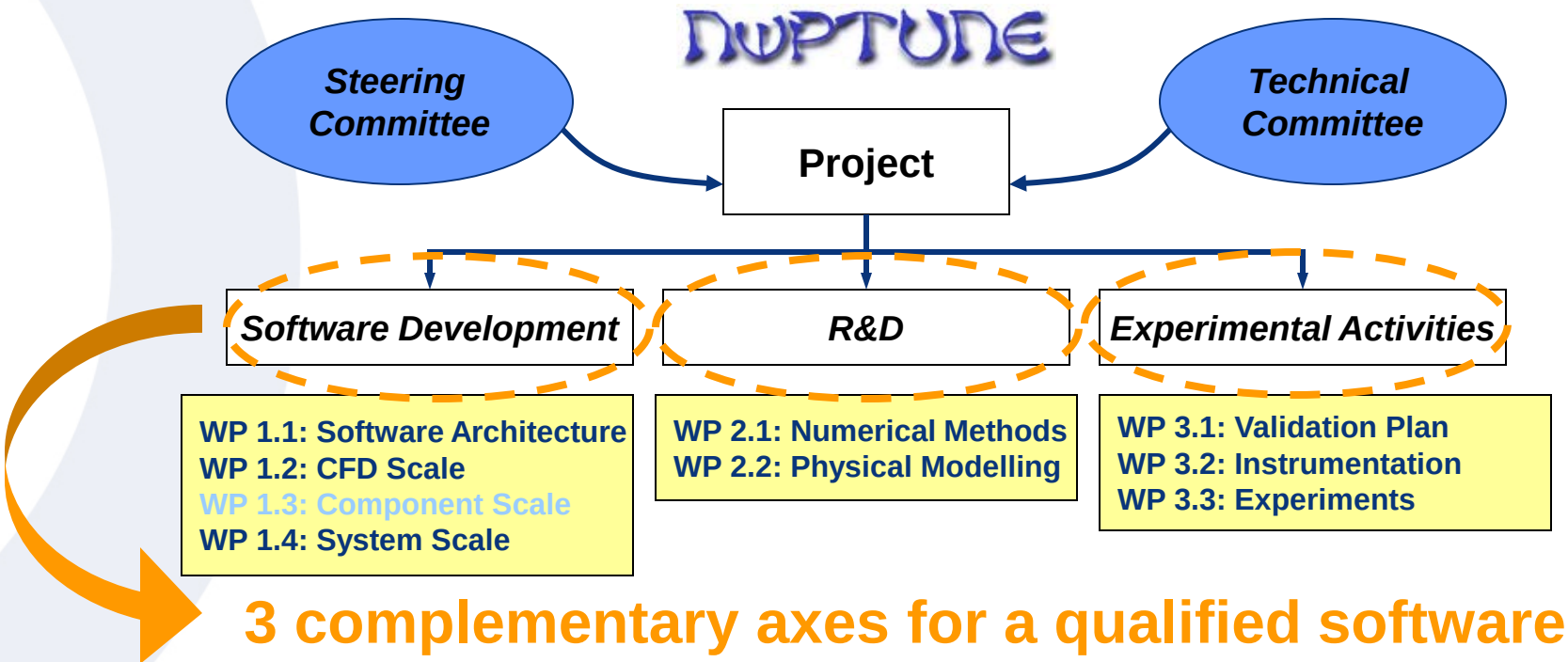
NEPTUNE_CFD



DNS SCALE

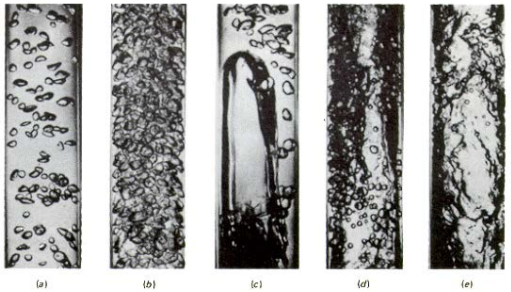
The NEPTUNE project

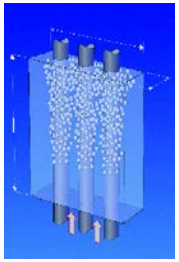
- Resources
  - ~25 men.year per year (2/3 CEA, 1/3 EDF), 5 PhD, 5 sites
  - 2011 Funding shares: 50 % EDF, 30 % CEA, 10 % IRSN, 10 % AREVA NP

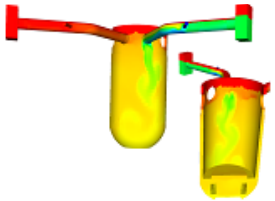


Overview of NEPTUNE_CFD

- Development team: EDF and CEA
  - **Development, validation, maintenance, installation, training, hot-line**
- Users (~50)
  - **EDF, CEA, IRSN, AREVA-NP**
  - **Academic collaboration (IMFT)**
  - **European projects partners (NURISP)**
- Main features
  - **3D and local two-phase flow analysis**
  - **Generalized multi-field model**
  - **Physical models**
    - **Turbulence (k-ε and RSM)**
    - **Interfacial area and polydispersion models**
    - **Set of models for boiling bubbly flows**
    - **Set of models for stratified steam-water flows**
    - **Conjugate heat transfer**
  - **Numerics**
    - **3D fully unstructured cell center Finite Volume**
    - **Iterative coupling of equations**
    - **Parallel by distributed memory (domain splitting)**

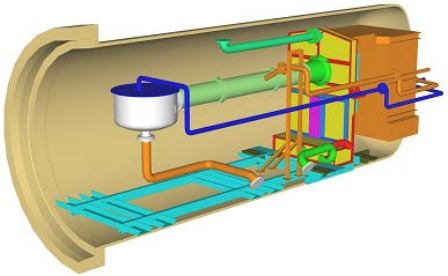






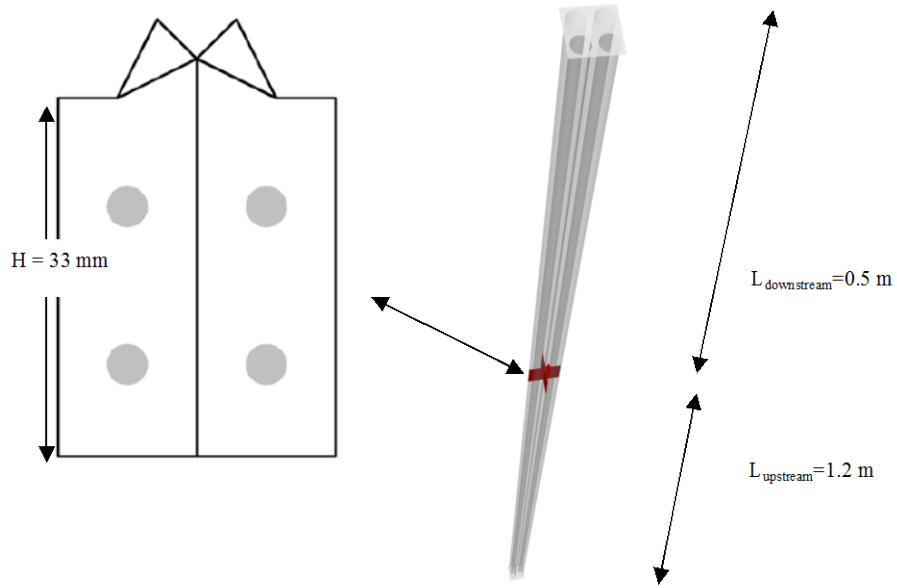
Verification & Validation strategy

- Physical and numerical benchmarks
  - **Definition of tests cases and acceptability criteria**
  - **Classical benchmark and more complex cases**
- Physical validation for boiling bubbly flows (for DNB)
  - **Existing data for adiabatic and boiling bubbly flows**
  - **New comprehensive experimental programs**
    - **Adiabatic bubbly flow tube**
    - **Adiabatic bubbly flow bundle+grid**
    - **Condensation tube**
    - **Convective boiling flow tube**
    - **Global tube**
    - **Global bundle+grid**
- Physical validation for PTS-related models
  - **Validation on existing experiments**
  - **New dedicated international program (TOPFLOW-PTS)**



Application to mixing grid analysis

- Objectives
  - Gain knowledge about the two-phase boiling flow through a spacer grid with mixing vanes
  - Assess the impact of turbulence models on the target variables supposed to be related to DNB (max void fraction at the wall, max wall temperature, mixing efficiency,…)
  - Impact of vane orientation on the target variables
- Configuration
  - 2x2 bundle, simple grid
  - PWR core conditions penalised to cause boiling
    - 155 bar; 330 °C; 3000 kg/m^2/s;1.6 MW/m^2
    - Vapor fraction up to 70%
  - Meshes: 1.5 to 7.6 Mcells

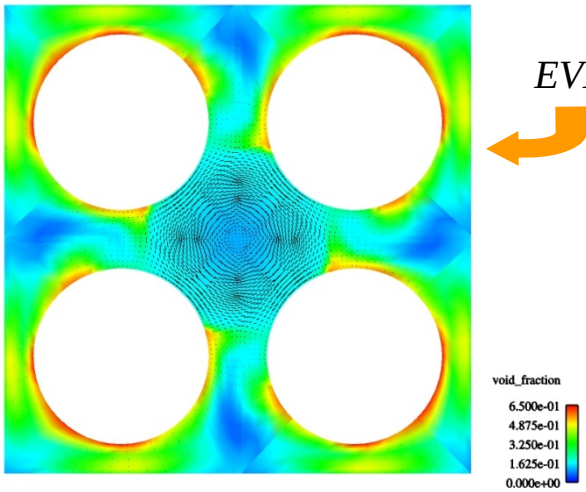


spacer grid with mixing vanes

2x2 rod bundles.

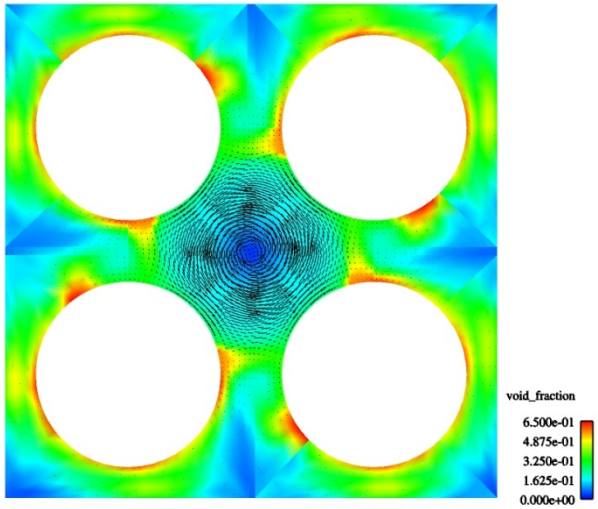
Application to mixing grid analysis

- Void fraction and liquid velocity
  - **Downstream mixing vanes**



$EVM \quad (K-\varepsilon)$

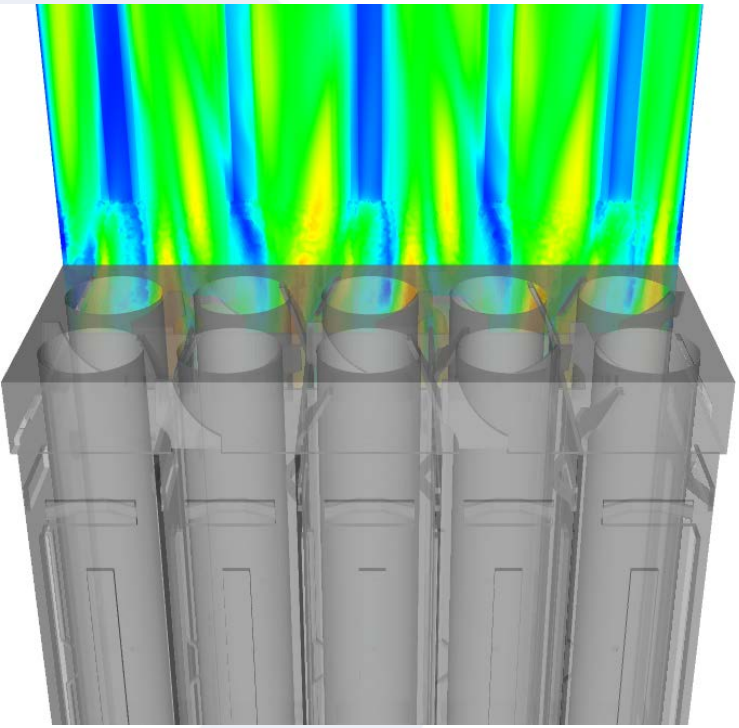
$RSM \quad (R_{ij}-\varepsilon)$

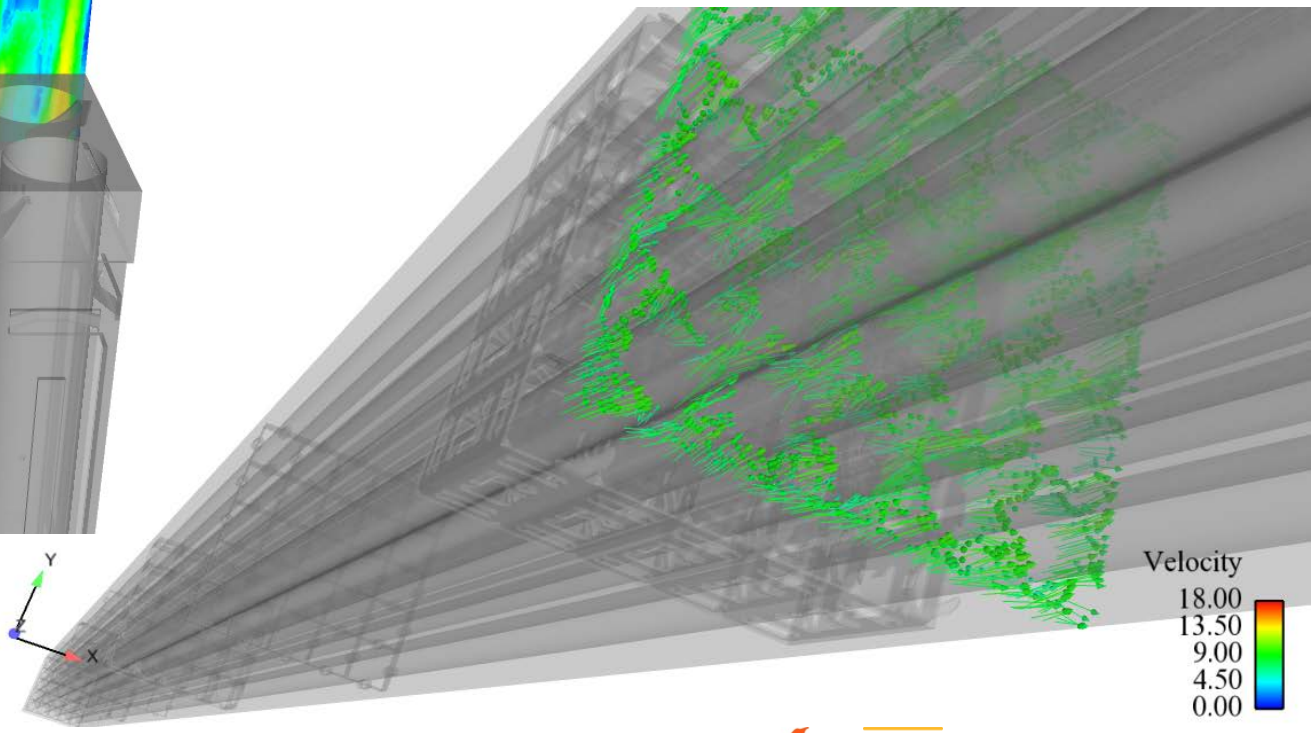


- Visualization of the “anti-void” effect of the vanes
- Difference between the 2 turbulence models results
  - Strong underestimation of rotation by k-ε
  - Void fraction slightly higher with RSM
  - Temperature gradient higher with RSM

Application to mixing grid analysis

- Future work
  - Full length 5x5 bundle with detailed grid

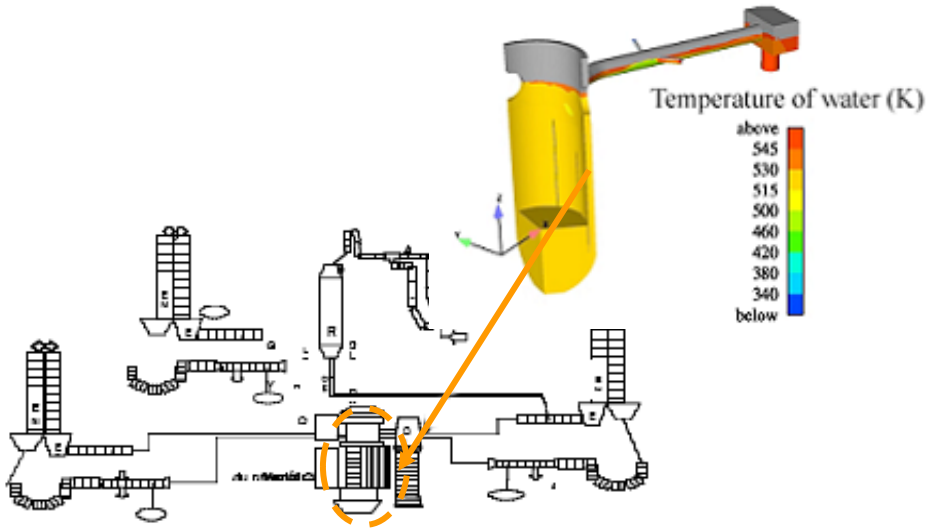




Application to PTS analysis

- In incidental transients such as LOCA, cold water is injected into the cold leg ➔ **potential Pressurized Thermal Shock**

- Compared to (1D) correlations approaches, validated (3D) CFD codes + conjugate heat transfer shall provide
  - **Improved precision**
  - **Reduced conservatisms**

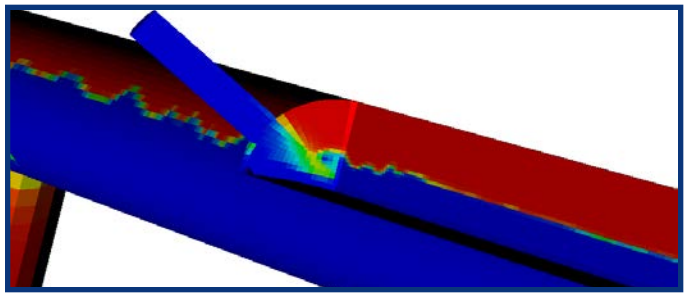


- Methodology
  - **System scale (CATHARE) one way coupling**
  - **3D CFD + conjugate heat transfer**
  - **Structural analysis**

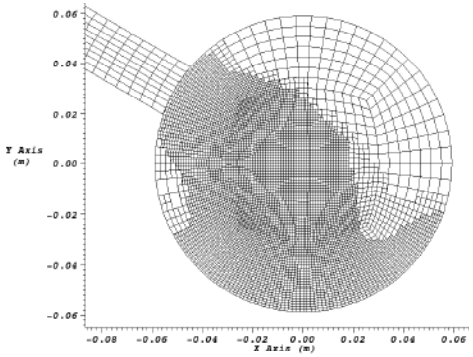
- Objective: extend the methodology to two-phase configurations

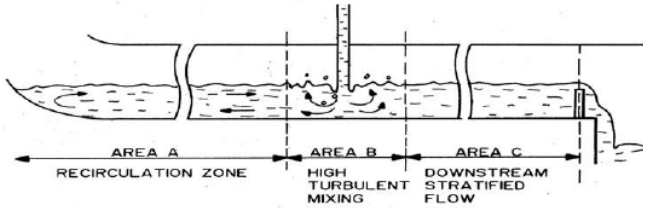
Application to PTS analysis

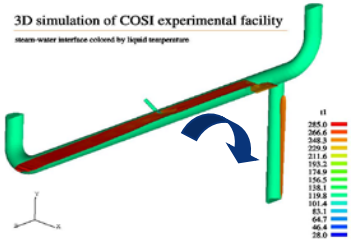
- Complex CFD modelling with steam & water in the hot leg
  - Free surface + Friction + Turbulence
  - Jet + Bubble entrainment + Turbulence
  - Condensation + Turbulence

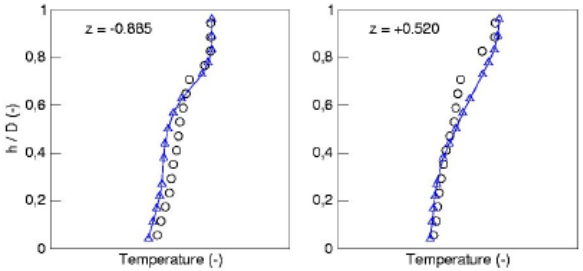


- Need for validation
  - Start with existing COSI experiment (cold leg 1:5, downcomer 1:100 in volume)
    - **Two-phase flow with cold jet and direct condensation**
    - **Liquid temperature and global condensation**



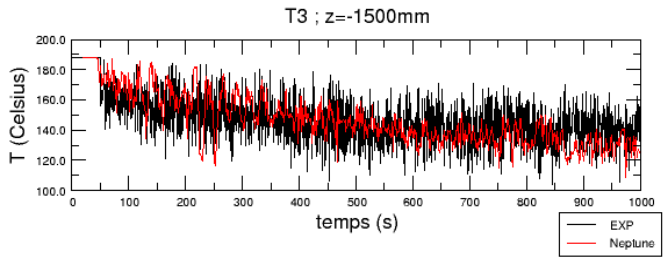


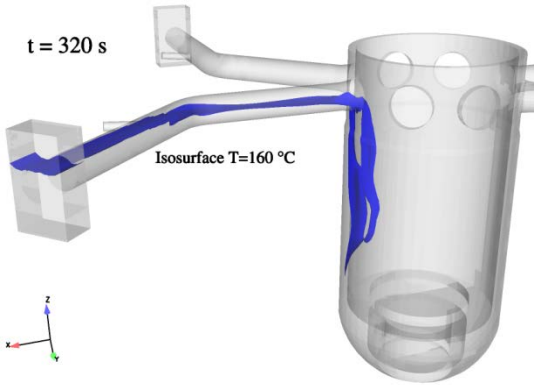




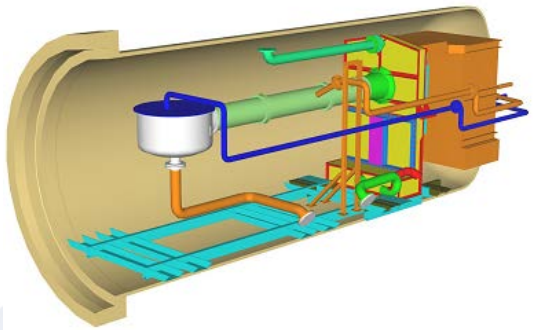
Application to PTS analysis

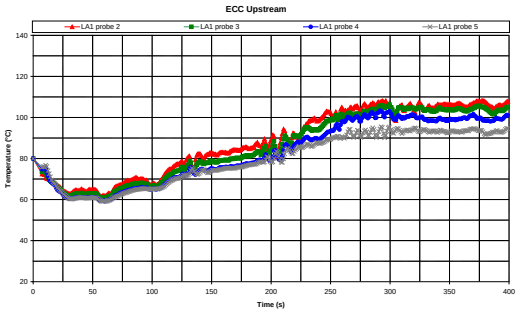
- **UPTF experiment (full scale 1300 Konvoi; air / water: no condensation)**
  - **Thermal mixing with free surface**
  - **Conjugate heat transfer (SYRTHES)**

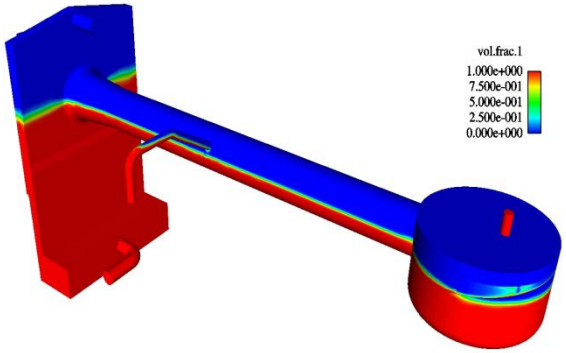




- **New integral experiment with all the physics: TOPFLOW-PTS** (HZDR, EDF, CEA, IRSN, AREVA, PSI, ETHZ)
  - **Pre-test NEPTUNE_CFD computations used to adjust the test matrix**

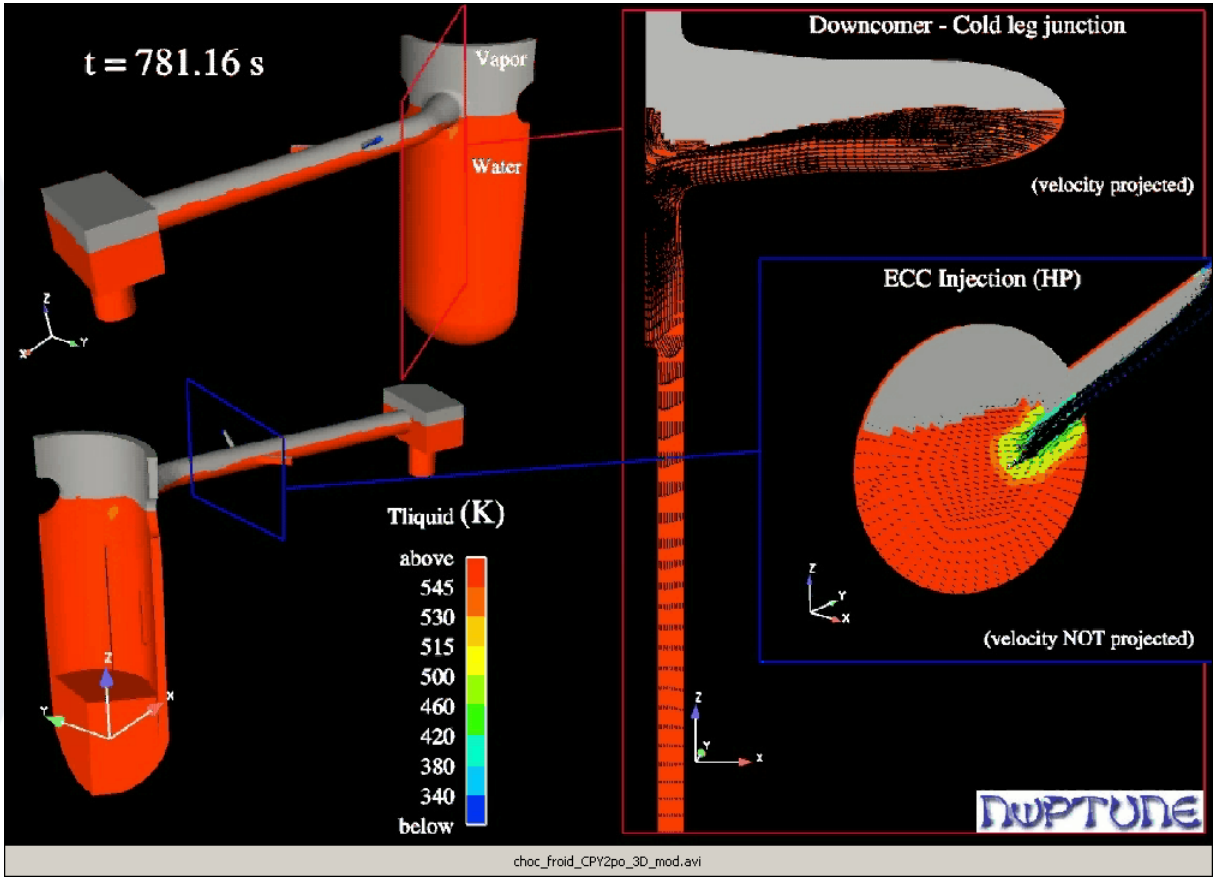






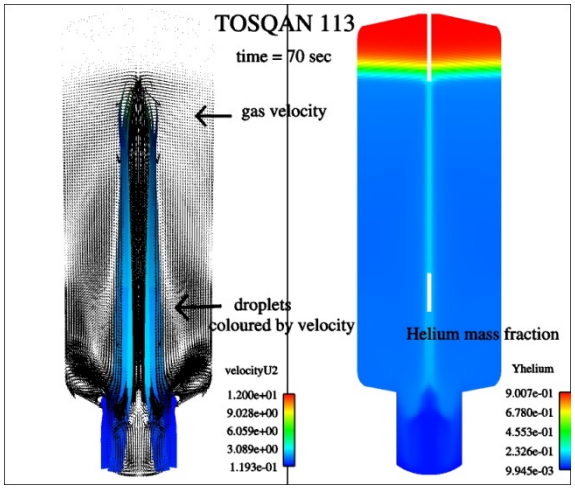
Application to PTS analysis

- First *test* of full scale two-phase scenario (2005): 900 MWe CPY 2'' LOCA) with
  - System scale (CATHARE) one way coupling; 3D CFD + conjugate heat transfer; Condensation

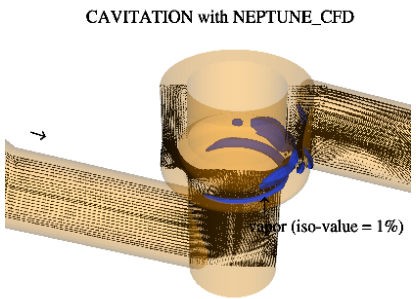


Some other applications in progress

- **Containement**
  - **Spray modeling**
  - **Wall condensation**
  - **Gas mixing**



- **Cavitation and valve qualifacation**
  - **Pressure constraints**
  - **Thermal shocks**
  - **Cavitation**



- **High fluxes device (fusion)**
  - **Very high-heat fluxes**
  - **Special CHF-increasing designs**

NURESIM (2005-2008) – NURISP (2009-2011)

Some profitable contributions to NEPTUNE_CFD :

- The European Project NURESIM : 01/02/2005 – 31/12/2008
  - 18 partners: EDF, CEA, FZD, GRS, UCL, JSI, KFKI-AEKI, NRI, U-Pisa, VTT, LUT, PSI, ASCOMP, KTH, UNI-KA, TU-Delft, UP-Madrid, INRNE
- 13 partners using NEPTUNE_CFD (Thermal-Hydraulics WP)
  - Some contributions to NEPTUNE_CFD
  - Verification: NURESIM enables to increase the quality of NEPTUNE_CFD
    - Eg.: corrections thanks to comparisons by KFKI (VVER)
  - Benchmark and validation
    - Numerous comparisons on various test-cases
  - First developments in NEPTUNE_CFD, very early in NURESIM
    - Eg.: wall functions, surface tension, …
  - Framework to better know the partners and exchange technically
    - Eg.: technical exchanges with JSI and UCL
  - NEPTUNE_CFD supports the comparison to FLUENT/CFX
    - Eg.: 3 users (UCL, PSI, JSI) mention the good numerical performance of NEPTUNE_CFD compared to FLUENT and CFX

…Thank you for your attention…

- Reference (general)

  - **A. Guelfi, D. Bestion, M. Boucker, P. Boudier, P. Fillion, M. Grandotto, J.-M. Hérard, E. Hervieu, P. Péturaud, NEPTUNE - A New Software Platform for Advanced Nuclear Thermal-Hydraulics, Nuclear Science and Engineering, Volume 156, Number 3 , Pages 281-324, July 2007**
  - **N. Méchitoua, M. Boucker, J. Laviéville, J.-M. Hérard, S. Pigny, and G. Serre, An unstructured finite volume solver for two-phase water-vapour flows based on an elliptic oriented fractional step method, in 10th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH 10), October 2003**

NEPTUNE_CFD balance equations

- Basis: classical two-fluid one pressure approach, including mass, momentum and energy balances for each phase, extended to *m* phases:

Two mass balance equations:

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla.\left(\alpha_k \rho_k \underline{V}_k\right) = \Gamma_k$$

The interfacial transfer terms of mass, momentum and heat.

Two momentum balance equations:

Reynolds stress tensor

$$\frac{\partial \alpha_k \rho_k \underline{V}_k}{\partial t} + \nabla.\left(\alpha_k \rho_k \underline{V}_k \underline{V}_k\right) = -\alpha_k \underline{\nabla} p + \underline{M}_k + \alpha_k \rho_k \underline{g} + \nabla.\left[\alpha_k \left(\underline{\underline{\Sigma}}_k + \underline{\underline{R}}_k\right)\right],$$

Two total enthalpy balance equations:

$$\frac{\partial}{\partial t}\left[\alpha_k \rho_k \left(h_k + \frac{V_k^2}{2}\right)\right] + \nabla.\left(\alpha_k \rho_k \left(h_k + \frac{V_k^2}{2}\right)\underline{V}_k\right) = \alpha_k \frac{\partial p}{\partial t} + \alpha_k \rho_k \underline{g}.\underline{V}_k$$

Wall transfer model for nucleate boiling

$$+\Gamma_k \left(h_{ki} + \frac{V_k^2}{2}\right) + \Pi'_k A_i + q'''_{wk} - \nabla.\left[\alpha_k \left(\underline{q}_k + \underline{q}_k^T\right)\right]$$

turbulent heat flux