Modeling of Two-phase Flow and Boiling with FLUENT

by

Nilanjana Basu, Andrey Troshko, and Greg Nurnberg

Fluent Inc.
Lebanon, New Hampshire
www.fluent.com

Presented at RELAP5 UGM, West Yellowstone, Montana
July 27, 2003

FLUENT
The Right Answer in CFD

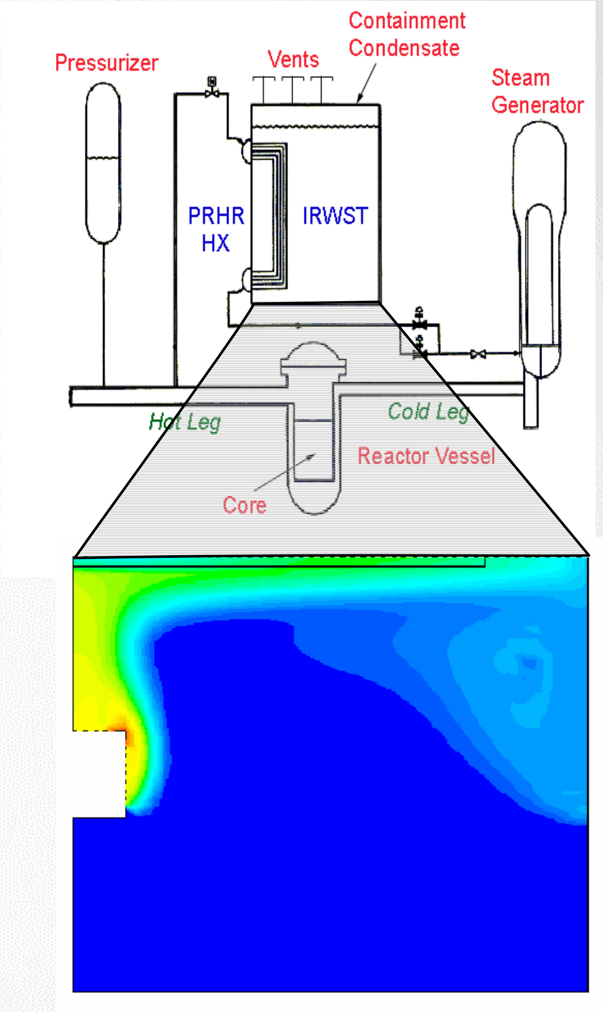
Outline

- FLUENT & RELAP5-3D© Coupling
- Multiphase models in FLUENT
- Boiling and two-phase flow Case studies with FLUENT
- Summary

FLUENT & RELAP5-3D© Coupling

Advantages

- Model entire system using 1 dimensional features of RELAP5-3D©
- Model some components of the system in detail using the 3 dimensional features of FLUENT
- Both the system and component behavior is more accurately predicted
- Boundary condition information is transferred back and forth between the two codes



FLUENT
The Right Answer in CFD

FLUENT & RELAP5-3D© Coupling (contd.)

Some key modeling capabilities in FLUENT to be utilized:

- **Turbulence**
- **Two-phase flow**
- **Flow through packed bed**
- **Neutronics-fluid interaction in the core region**

Focus of this presentation: Two-phase flow

Multiphase models in FLUENT

- Discrete Phase Model (DPM)
- Mixture Model
- Volume of Fluid Model (VOF)
- Eulerian Multiphase Flow Model

Multiphase models in FLUENT (contd.)

Discrete Phase Model (DPM)

- **Trajectories of particles/droplets/bubbles are computed in a Lagrangian frame.**
  - **–Particles can exchange heat, mass, and momentum with the continuous gas phase.**
  - **–Particle-Particle interaction is neglected.**
  - **–Turbulent dispersion can be modeled with stochastic tracking or a "particle cloud" model.**

- **Volume loading: volume fraction < 12%)**
- **Particulate Loading: Low to moderate.**

Application examples: Cyclones, spray dryers, particle separation and classification, aerosol dispersion, liquid fuel and coal combustion. etc.

FLUENT
The Right Answer in CFD

Multiphase models in FLUENT (contd.)

The Mixture Model

- Modeling N-phase flows.
- Solves the mixture momentum equation (for mass-averaged mixture velocity)
  - Inter-phase exchange terms depend on relative (slip) velocities
  - Turbulence and Energy equations are solved for the mixture
  - Only one of the phases may be defined as compressible.
- Solves the transport equation of volume fraction for each secondary phase.

FLUENT
The Right Answer in CFD

Multiphase models in FLUENT (contd.)

Applicability of Mixture Model

- **Flow regime:** **Bubbly flow, droplet flow, slurry flow.**
- **Volume loading:** **Dilute to moderately dense.**
- **Particulate Loading:** **Low to moderate.**
- **Turbulence modeling:** **Weak coupling between phases.**
- **Stokes Number:** **Stokes Number < < 1.**

Application examples: Hydrocyclones, bubble column reactors, solid suspensions, gas sparging.

FLUENT
The Right Answer in CFD

Multiphase models in FLUENT (contd.)

The Volume of Fluid Model (VOF)

- Model to track the position of the interface between two or more immiscible fluids.
- A single momentum equation is solved and the resulting velocity field is shared by all phases.
  - Surface tension and wall adhesion effects can be taken into account.
- Solves transport equation for volume fraction of each secondary phase.
- Recommended that simulation be performed in unsteady mode.

FLUENT
The Right Answer in CFD

Multiphase models in FLUENT (contd.)

Applicability of VOF Model

- **Flow regime:** **Slug flow, stratified/free-surface flow.**
- **Volume loading:** **Dilute to dense.**
- **Particulate Loading:** **Low to high.**
- **Turbulence modeling:** **Weak to moderate coupling between phases.**
- **Stokes Number:** **All ranges of Stokes number.**

Application examples:Large slug flows, filling, off-shore oil tank sloshing, boiling, coating.

FLUENT
The Right Answer in CFD

Multiphase models in FLUENT (contd.)

The Eulerian Multiphase Model

- Solves continuity, momentum and energy equations for each phase.
  - Volume fractions characterize equation set for each phase.
    - Several models available to define inter-phase exchange coefficients.
    - Strong coupling makes this model more difficult to use than Mixture Model.

  - **Euler Granular option**: each granular phase is treated as a distinct interpenetrating granular 'fluid'.

  - **Heat and mass transfer between n-phases**: Ranz-Marshall (Euler/Euler), Gunn (Euler/granular) and user-defined models.

FLUENT
The Right Answer in CFD

Multiphase models in FLUENT (contd.)

Applicability of Eulerian model

- **Flow regime:** **Bubbly flow, droplet flow, slurry flow, fluidized beds, particle-laden flow.**
- **Volume loading:** **Dilute to dense.**
- **Particulate Loading:** **Low to high.**
- **Turbulence modeling:** **Weak to strong coupling between phases.**
- **Stokes Number:** **All ranges of Stokes number.**

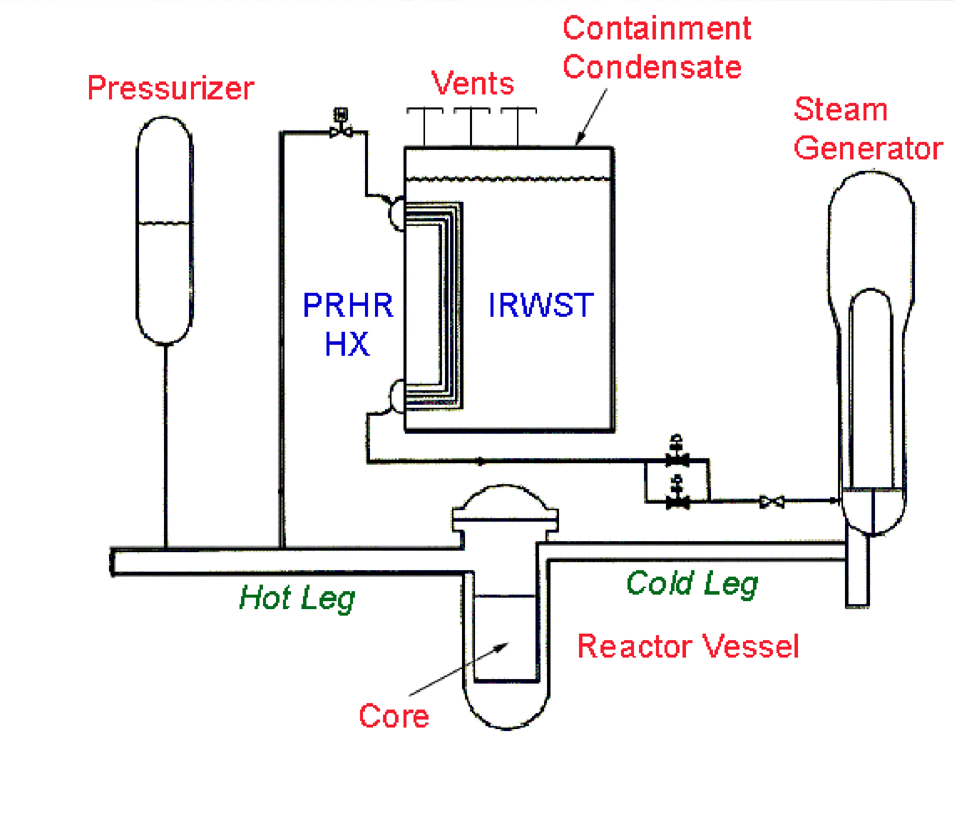
Application examples: High particle loading flows, slurry flows, sedimentation, hydro-transport, fluidized beds, risers, packed bed reactors.

FLUENT
The Right Answer in CFD

Boiling and two-phase flow Case studies with FLUENT

Advanced Pressurized Reactor

- The Advanced Pressurized Reactor is light water reactor being designed
- The In-containment Refueling Water Storage Tank (IRWST) is passive safety system for heat removal
- During a small break loss of coolant accident (SBLOCA) it allows steam to cool in a pool of water and escape through vents at the top

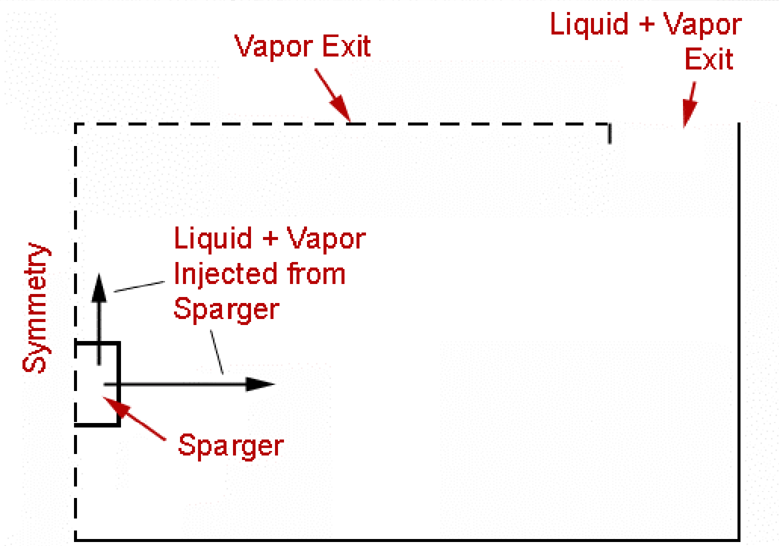


FLUENT
The Right Answer in CFD

Boiling and two-phase flow Case studies with FLUENT (contd.)

Advanced Pressurized Reactor

- **FLUENT is used to simulate the 2-phase flow in the IRWST**
- **The mixture is injected through a sparger**
- **The Eulerian multiphase model allows for separate transport equations for**
  - **liquid (water)**
  - **vapor (steam)**
- **The 2D model makes use of a porous region to allow only vapor to exit through most of the top boundary**

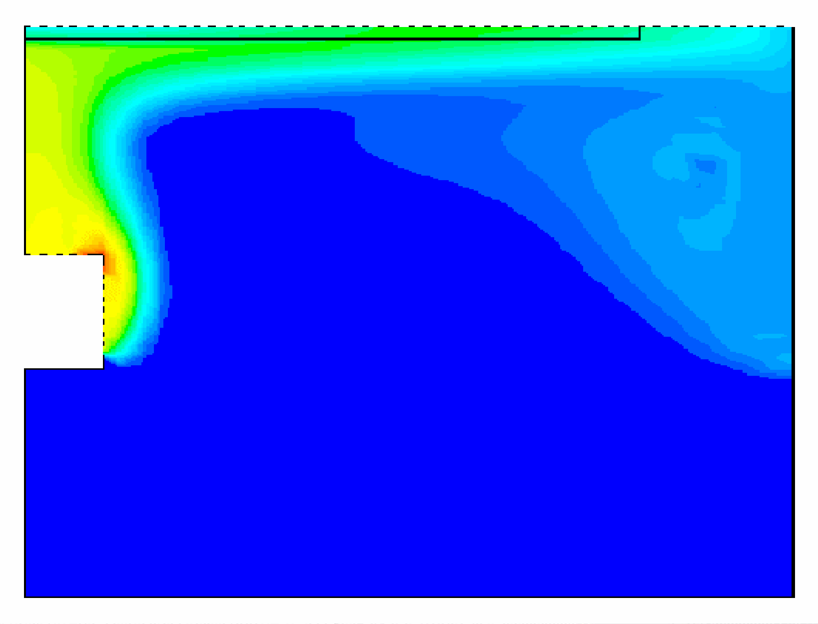


Boiling and two-phase flow Case studies with FLUENT (contd.)

Advanced Pressurized Reactor

- **Steady-state simulations are performed for different bubble sizes and vapor volume fraction**

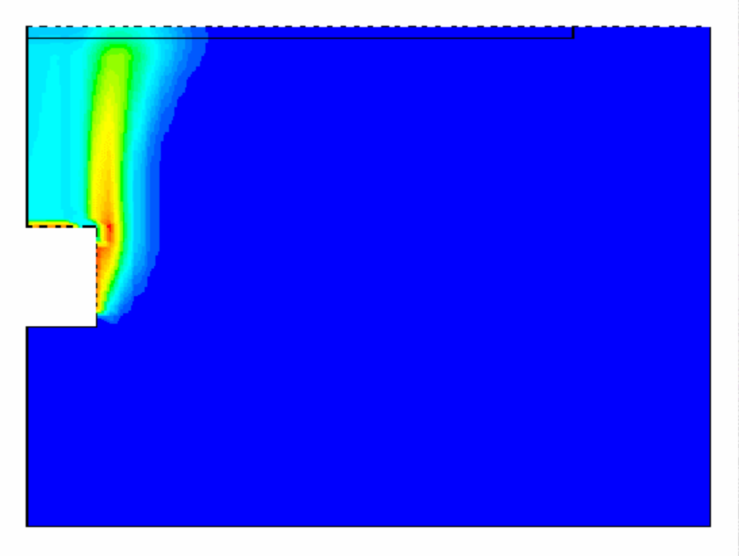
- **For 1mm bubbles and 40% vapor at the inlet, most vapor escapes but some is entrained in recirculation in the water near the side of the vessel**



Boiling and two-phase flow Case studies with FLUENT (contd.)

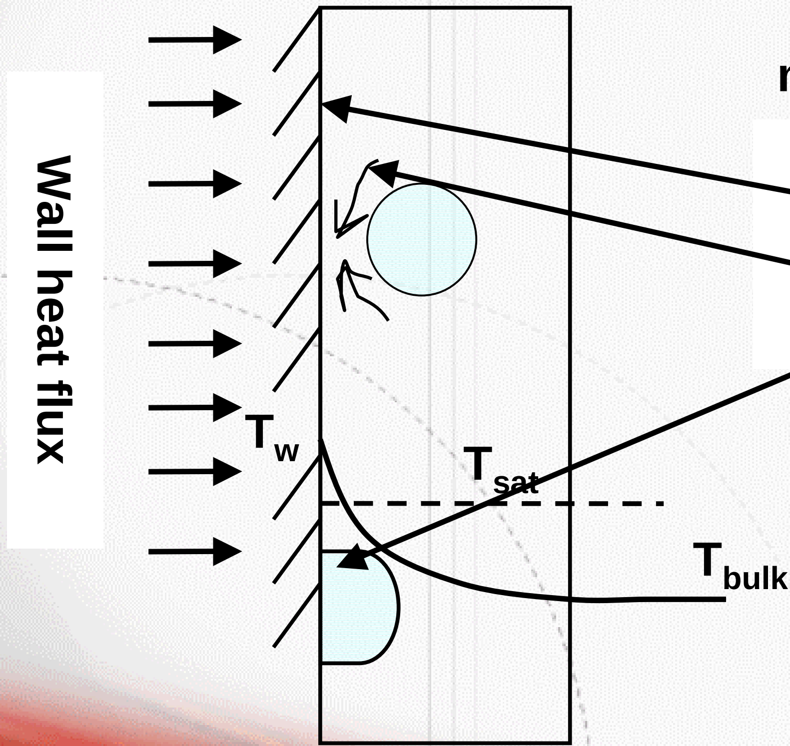
Advanced Pressurized Reactor

- For 100mm bubbles and 10% vapor at the inlet the flow is very different
- Larger buoyant forces cause steam to rise and escape quickly
- Results suggest that FLUENT is well suited to assist in the design of these systems



Boiling and two-phase flow Case studies with FLUENT (contd.)

Subcooled Nucleate Boiling



RPI (Rensselaer Polytechnic Institute) model of wall heat flux partitioning[1]

Wall heat flux =
Single phase heat flux
+ Quenching heat flux
+ Evaporation heat flux

Implemented as a source term in energy equation

A user-defined function (UDF) in FLUENT includes temperature-driven heat and mass transfer between phases[1]

[1]*Kurul, N., and Podowski, M. Z., 9th Int. Heat Trans. Conf. Jerusalem, p. 21-16, 1990.*

FLUENT
The Right Answer in CFD

Boiling and two-phase flow Case studies with FLUENT (contd.)

Subcooled Nucleate Boiling

An annular domain, with heated inner wall is simulated

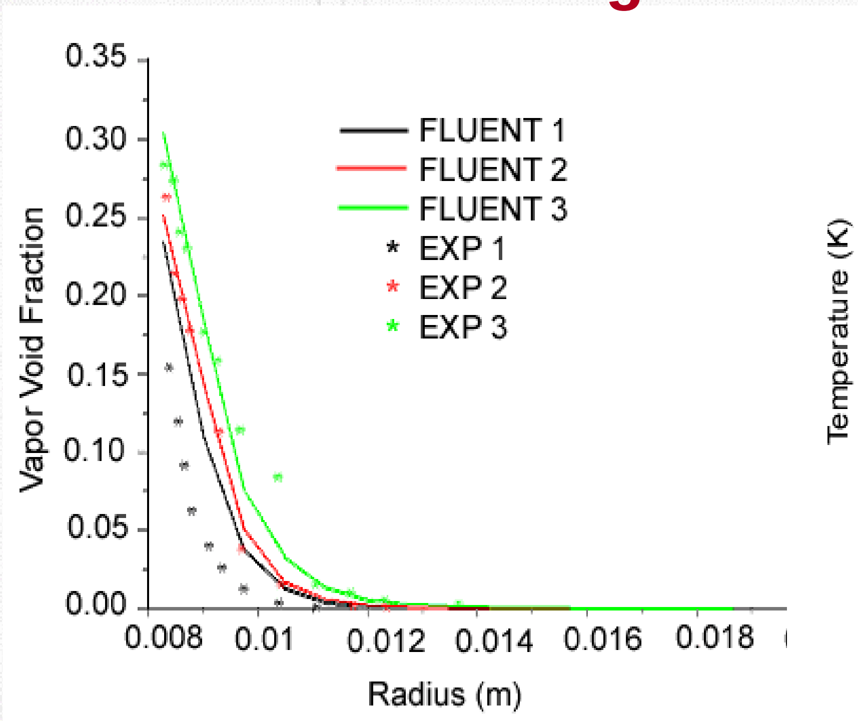
- **FLUENT 6.1 is used to simulate this process for three sets of experimental conditions[2] (below)**
- **User-defined functions are used with the Eulerian multiphase model to implement the RPI model[1] for**

Parameter	EXP 1	EXP 2	EXP 3
Inner wall heat flux, W/m^2	80,000	95,000	116,000
Fluid mass velocity, kg/m^2/sec	565	785	785
Mean liquid subcooling at test section inlet, ^{0}C	37.8	30.3	30.3

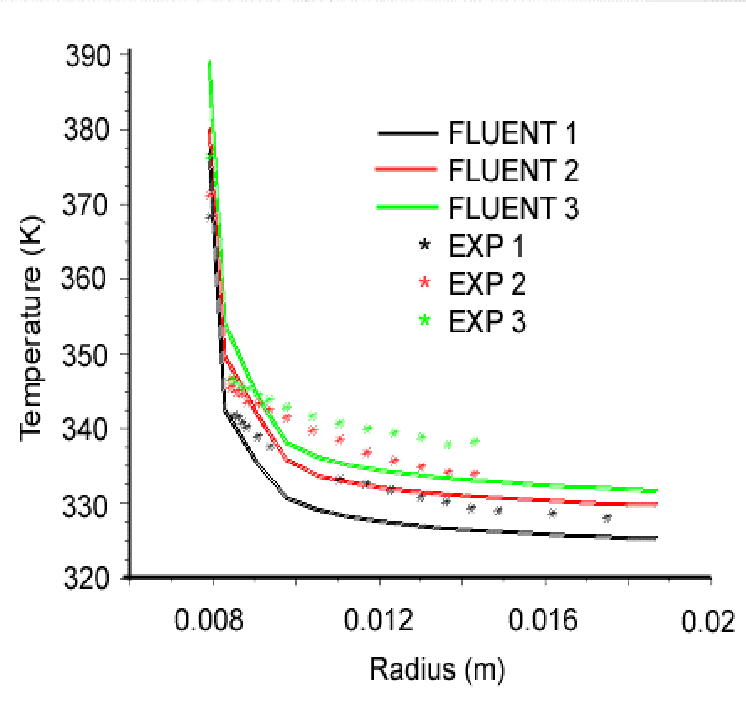
[2]*Roy, R. P., Velidandla, V., and Kalra, S. P., ASME J. Heat Trans. 119, 754-766 (1997).*

Boiling and two-phase flow Case studies with FLUENT (contd.)

Subcooled Nucleate Boiling



Radial profiles of vapor void fraction prediction



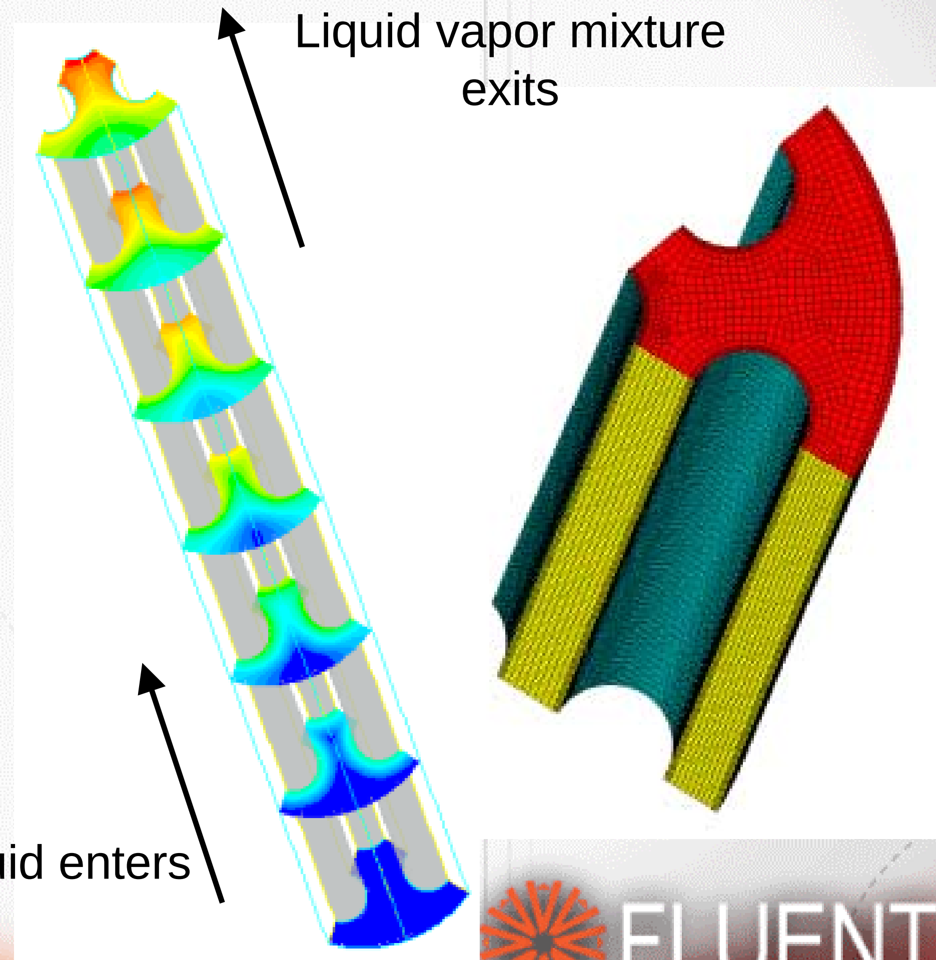
Temperature predictions are in acceptable

FLUENT
The Right Answer in CFD

Boiling and two-phase flow Case studies with FLUENT (contd.)

Boiling flow in nuclear reactor

- **Flow in nuclear fuel assembly**
  - **Pressure 50 atm**
  - **Re_{liq}=300,000**
  - **Heat flux 0.522 MW/m^2**
  - **Inlet subcooling 4.5 K**
  - **y_+=100**



FLUENT
The Right Answer in CFD

Boiling and two-phase flow Case studies with FLUENT (contd.)

Boiling flow in nuclear reactor

- Condensation or evaporation at surface of bubbles in free stream
- Turbulent dispersion of bubbles if liquid flow is turbulent
- Additional turbulence created by bubbles
- Modified lift force to account for vortex shedding by bubbles

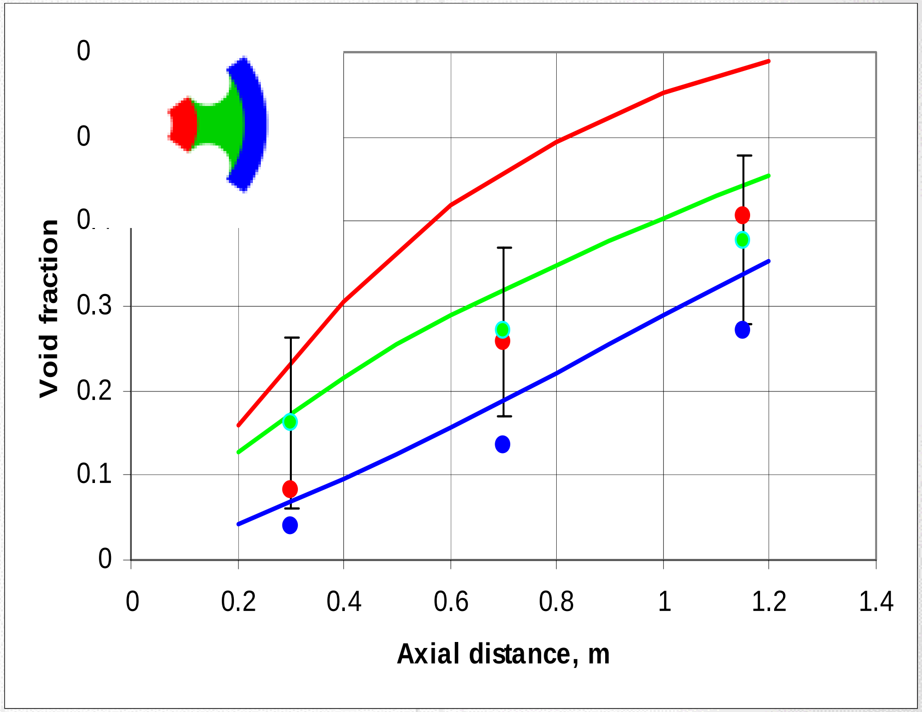
FLUENT
The Right Answer in CFD

Boiling and two-phase flow Case studies with FLUENT (contd.)

Boiling flow in nuclear reactor

- **Wall temperature is defined by bisection method from flux partitioning**

- **~3-4 hours to get converged solution on 2GHz CPU**
  80,000 cells

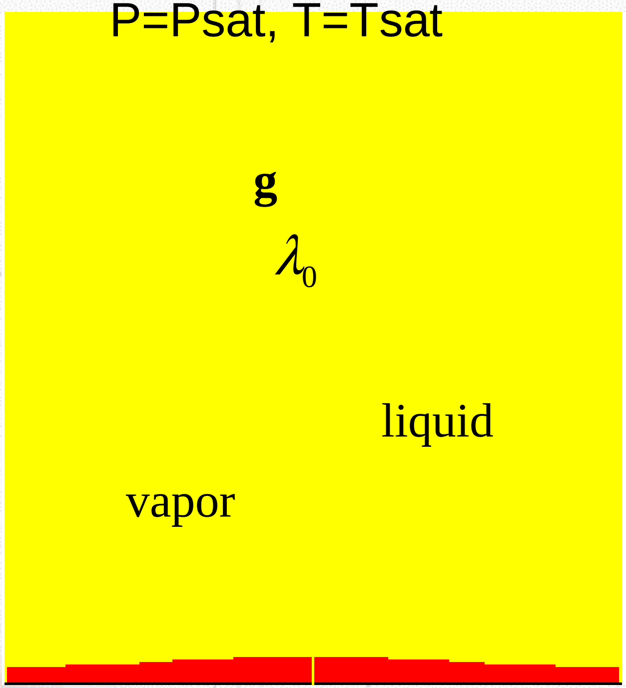


Comparison with experiment for vapor void fraction

Boiling and two-phase flow Case studies with FLUENT (contd.)

Film boiling

- **Using VOF modeling in Fluent**



$\frac{3\lambda_0}{2}$

symmetry

symmetry

Wall
T=Tsat+10K

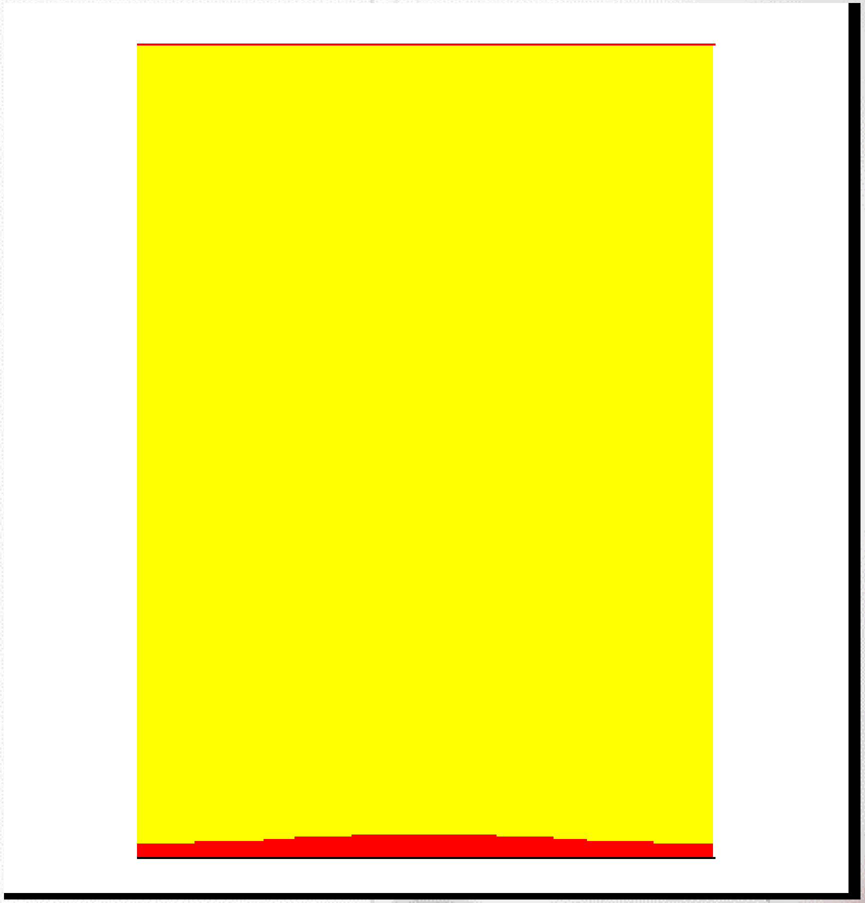
$$\lambda_0 = 2\pi\left(\frac{3\sigma}{(\rho_l - \rho_g)g_y}\right)^{1/2} = 0.0778m$$

Boiling and two-phase flow Case studies with FLUENT (contd.)

Film boiling

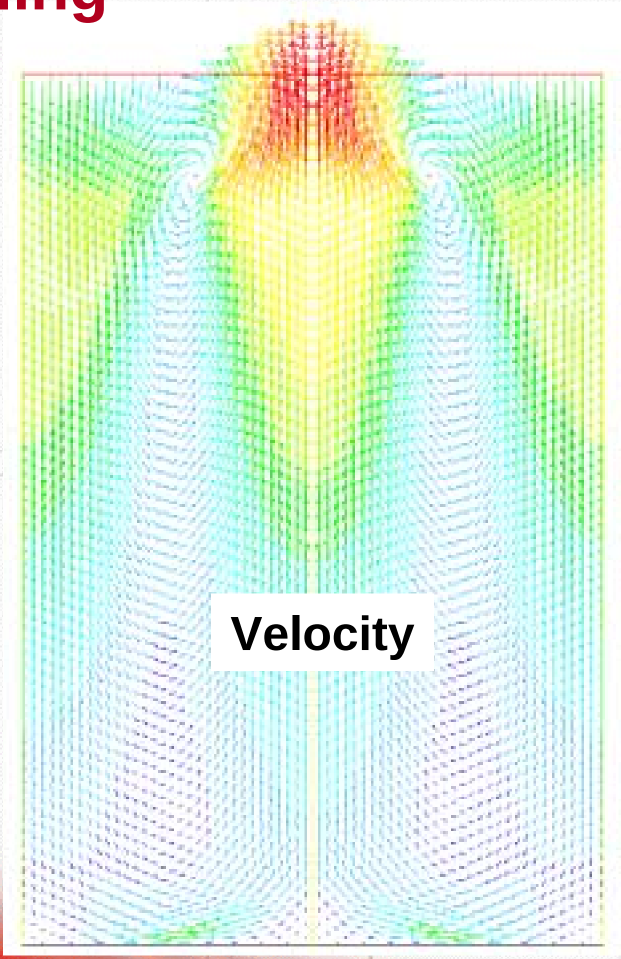
Animation

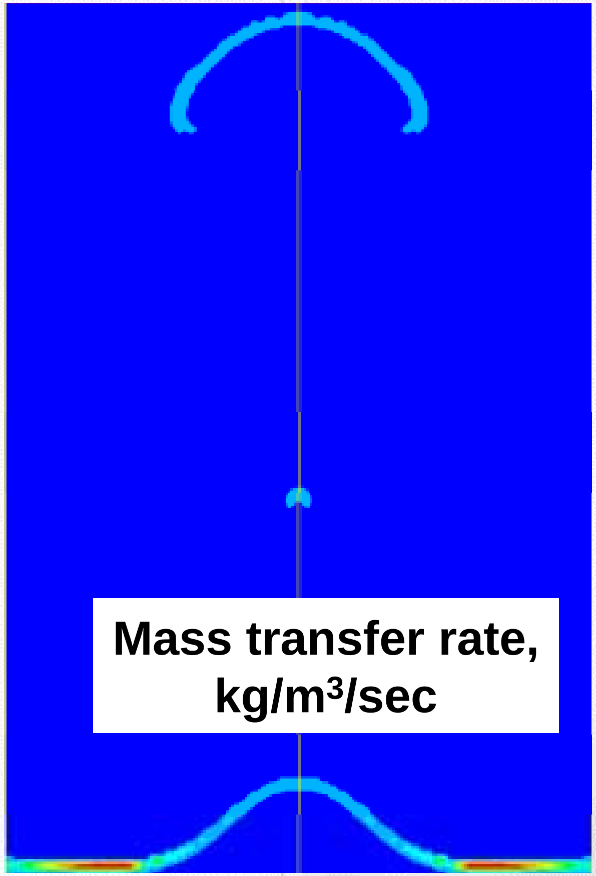
Contours of volume fraction of the vapor



Boiling and two-phase flow Case studies with FLUENT (contd.)

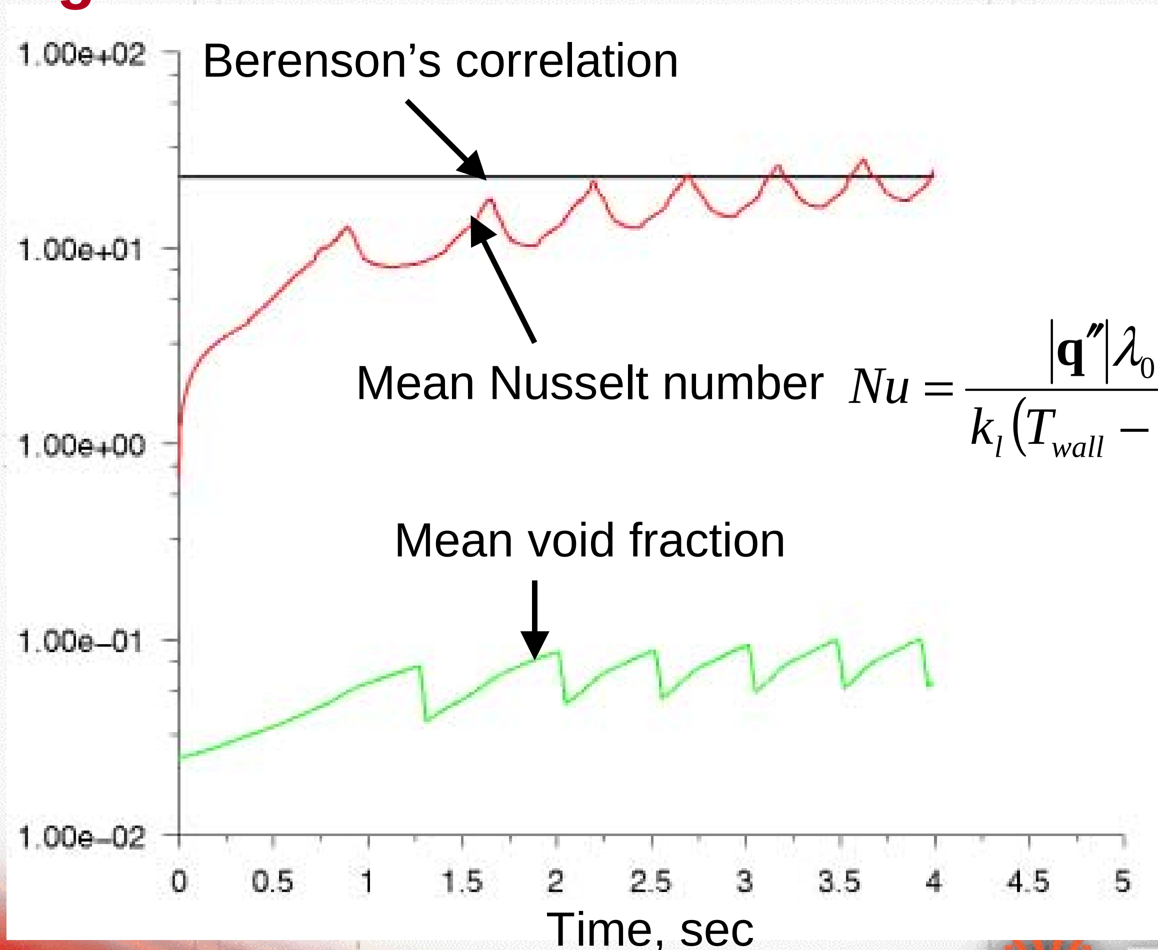
Film boiling





Boiling and two-phase flow Case studies with FLUENT (contd.)

Film boiling



$$Nu = \frac{|\mathbf{q}''|\lambda_0}{k_l\left(T_{wall} - T_{sat}\right)}$$

Summary

- Case studies of nucleate boiling and film boiling with FLUENT have been presented.
- These case studies demonstrate that FLUENT can successfully model two-phase flow and boiling.
- Two-phase modeling capabilities will enhance Reactor thermal hydraulic study using FLUENT-RELAP5 coupling

FLUENT
The Right Answer in CFD