Business from technology

Adaptation and validation of OpenFOAM® CFD-solvers for nuclear safety related flow simulations

SAFIR2010 Seminar, 10.-11.3.2011, Espoo

Juho Peltola, Timo Pättikangas (VTT)

Tomas Brockmann, Timo Siikonen (Aalto)

Timo Toppila, Tellervo Brandt (Fortum)

Presentation

1. Motivation
2. What is OpenFOAM®?
3. Validation plan
4. Single-phase near wall treatment validation
5. Conjugated heat transfer validation
6. Two-phase solver development and validation
7. OECD/NEA T-junction benchmark
8. Summary

1. Motivation

Computational Fluid Dynamics (**CFD**) has become an increasingly popular tool for nuclear safety related thermo-hydraulic investigations.

There are several suitable CFD-codes available, both closed and open source.

The benefits of an open source CFD-code are:

- Transparency
- Infinite customizability
- Lack of licensing fees
  - → Feasible cost structure for massively parallel computations

2. What is OpenFOAM®?

"The OpenFOAM® (Open Field Operation and Manipulation) CFD Toolbox is a free, open source CFD software package produced by a commercial company, **OpenCFD Ltd**."

Originated in Imperial College London in early 90's, released as open source in 2004.

Unstructured 3D Finite Volume Method (FVM) for partial differential equation field problems.

A library of C++ modules that can be used create solvers, utilities and models. Comes with number of pre-built applications.

2. What is OpenFOAM®?

Compared to most other open source CFD codes, the **benefits** of OpenFOAM® are:

- A large, active and growing **user base**.
- Modern approach to mesh handling with **unstructured** and **polyhedral meshes**.
- Low level **parallelization** and an object oriented code structure that makes it fast and easy to implement new models and solvers in the top level code.

Distinct **drawbacks** of OpenFOAM® are:

- Lack of comprehensive, public, formal documentation
- A very steep user learning curve.
- Many of the features represent the state-of-the-art, but often lack the polish to directly apply them to practical engineering problems.

3. Validation plan

The work is focused on a few nuclear safety related applications at a time.

- The first application: single and two-phase flow inside a PWR fuel assembly.

The validation is based comparison of simulations to experimental data.

Systematic code verification is out of the scope of this project

- The OpenFOAM community with thousands of users is relied on to find the relevant code errors.

The validation plan is a living document that evolves during the project and code development.

4. Single-phase near-wall-treatment validation

1. test case: **Turbulent pipe flow**
   - **Re_D = 45 000, 80 000 or 169 000**

Solver: **buoyantPimpleFoam** (OpenFOAM® 1.7.x)

- Single-phase, heat transfer, compressible and incompressible
- NuFoam enhancements: Jayatilleke thermal wall function, external temperature boundary condition, post processing utilities.

Tested models:

- Two RANS* turbulence models: **standard *k-ε*** and **SST *k-ω***
- Different momentum and thermal wall functions
- Different mesh wall resolutions: **y^+ = 1...250**

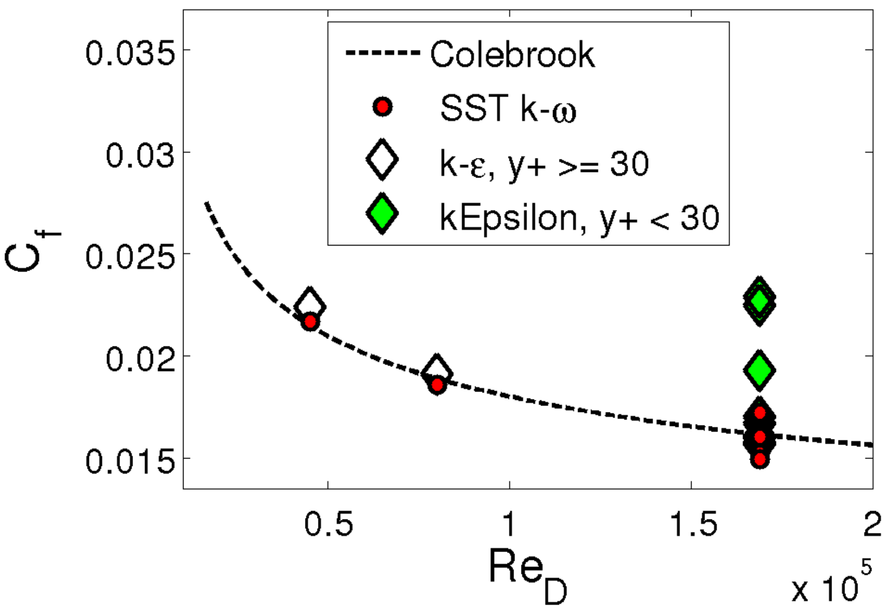
All-in-all: Results were reported for **48 RANS* simulations**

* Reynolds Averaged Navier-Stokes (RANS): Turbulent fluctuations are modelled and local mean properties are solved.

4. Single-phase near-wall-treatment validation

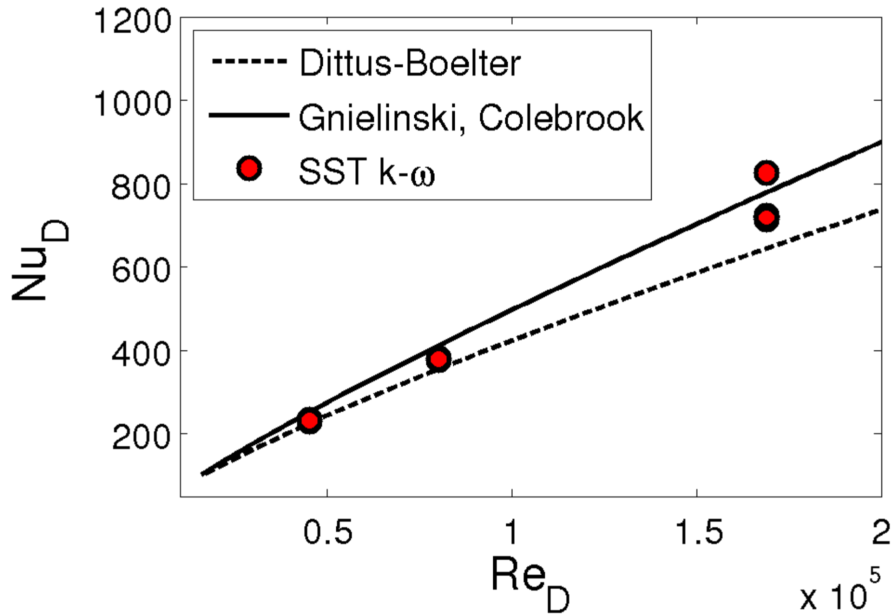
Pressure loss compared to Colebrook correlation:

Std. k-ε:	$y^+ >= 30$:	errors < 4%
	$y^+ < 30$:	inaccurate
SST *k-ω*:	$y^+ = 1..250$:	errors < 9 %



Heat transfer compared to the Gnielinski correlation (simulated friction):

$y^+ > y^+_{thermal}$:	error < 8 %
$y^+ < y^+_{thermal}$:	error < 1 %



y^+ is dimensionless distance from wall to first cell centre of the computational mesh

5. Conjugated heat transfer validation

Solver: **chtMultiRegionFoam** (OpenFOAM® 1.7.x)

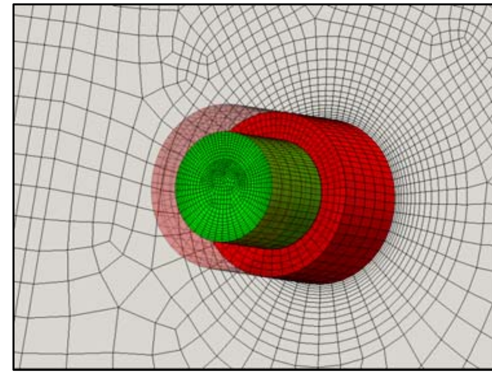
- Allows multiple temperature coupled fluid and solid regions
- Coupling by special boundary conditions

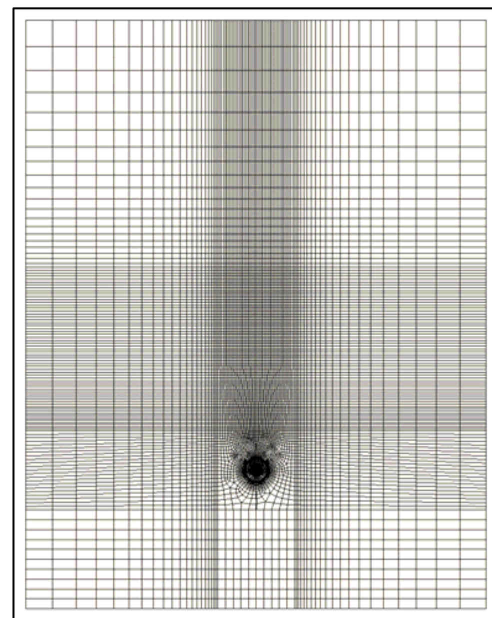
1. test case: **Turbulent pipe flow in a copper pipe**

- Tested against: buoyantPimpleFoam and an analytical heat transfer coefficient.
- Results match: Relative pressure loss difference 10^{-5} Wall temperature difference 0.005 K

2. test case: **Water pipe suspended in air**

- Heat transfer: Water→Copper → Air
- Three simulations: Free convection at $Ra_D = 10^5$ Forced convection at $Re_D = 120$ using two different channel widths.

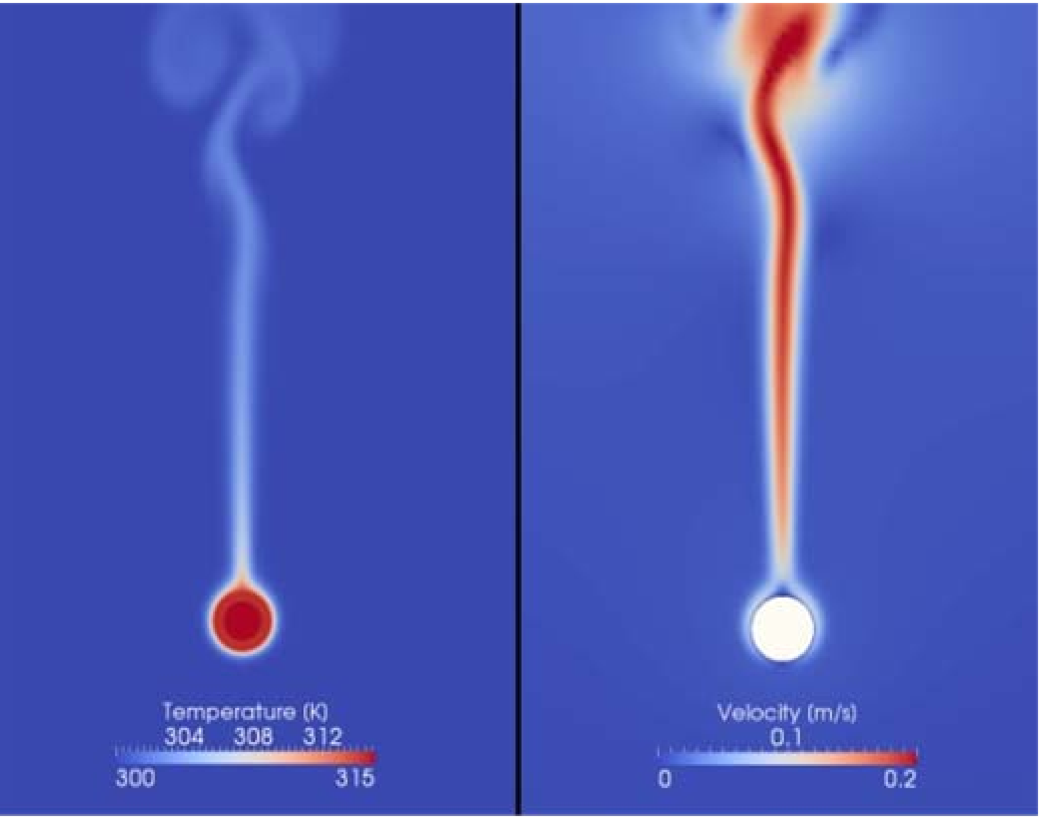




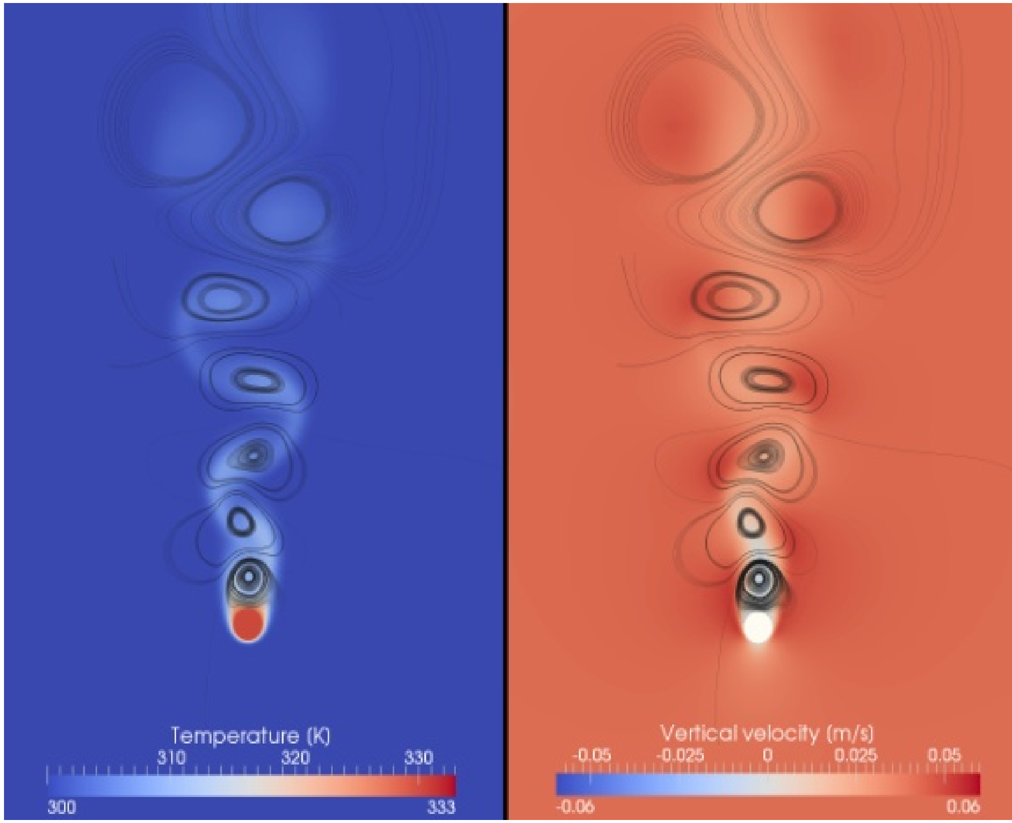
5. Conjugated heat transfer validation

Copper water pipe suspended in air

Free convection



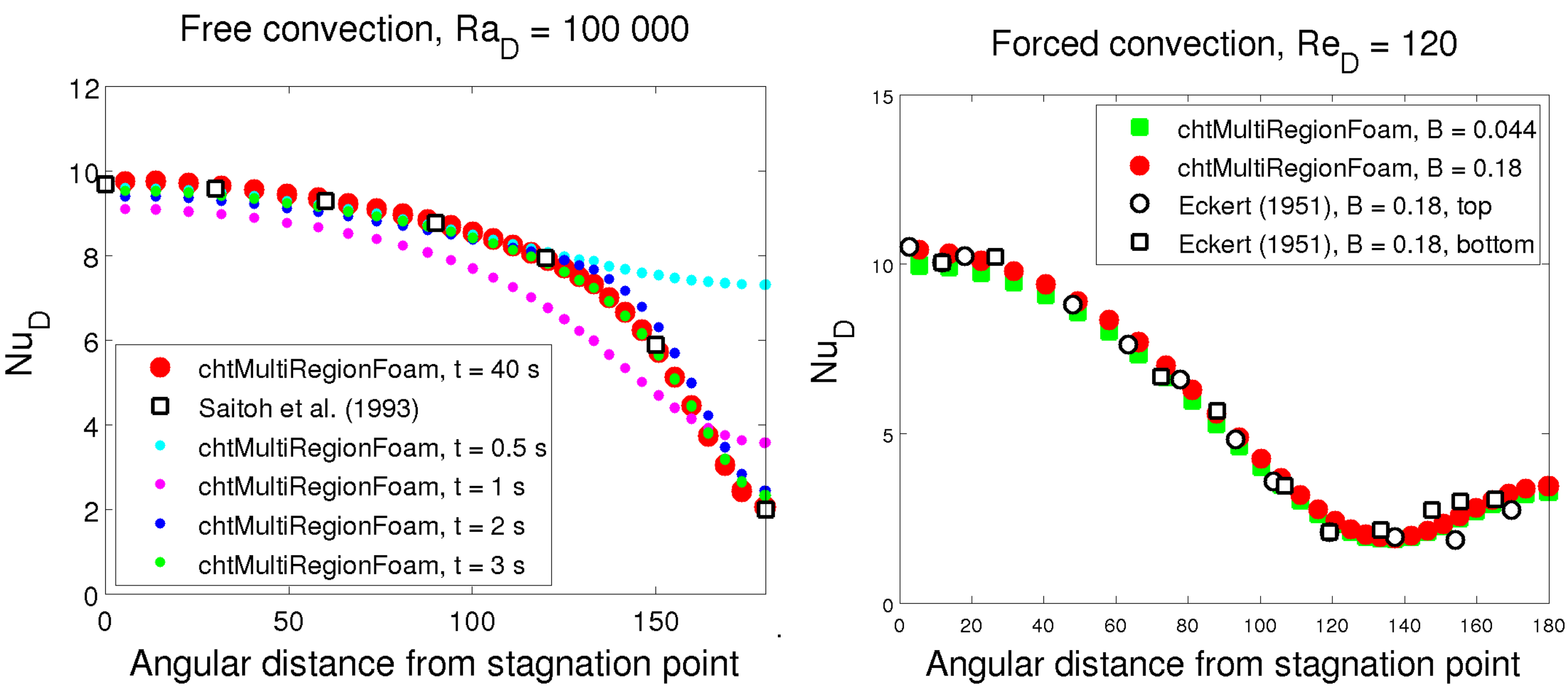
Forced convection



Temperature Velocity Temperature Vertical velocity

5. Conjugated heat transfer validation

Example of results: **External convection Nusselt number**



6. Two-phase solver development and validation

OpenFOAM 1.7.x: Existing two-fluid solvers: bubbleFoam and twoPhaseEulerFoam

twoPhaseEulerFoam was selected as starting point:

- More general and provides all the functionality the bubbleFoam does.
- Two **incompressible** phases with **constant material properties**
- *k*-*ε* turbulence model for the continuous phase
- **No interfacial models for bubbles**
- **No heat transfer**

To implement enhancements, a new solver **twoPhaseNuFoam v0.1** was created:

- Based on the twoPhaseEulerFoam
- Coupling for bubble interfacial forces
- Bubble induced turbulence models

6. Two-phase solver development and validation

New **bubbleModel** library for bubble interfacial force models was created. It provides:

- **Local bubble properties**: Reynolds number, aspect ratioetc.
- A selection of **user selectable sub models**:

Drag, C_d

- Tomiyama (2002)
- Tomiyama (1995)
- Schiller-Nauman (1935)
- Constant C_d

Drag Swarm correction

- Tomiyama (1995)
- none

Lift, C_l

- Tomiyama (2002)
- Constant C_l

Virtual Mass, C_{vm}

- Lamb (1879)
- Constant C_{vm}

Wall lubrication force

- Frank (2004)
- Generalized Tomiyama (2003)
- Antal (1991)
- None

Bubble aspect ratio

- Vakrushev & Effremov (1970)
- Constant

Turbulent dispersion

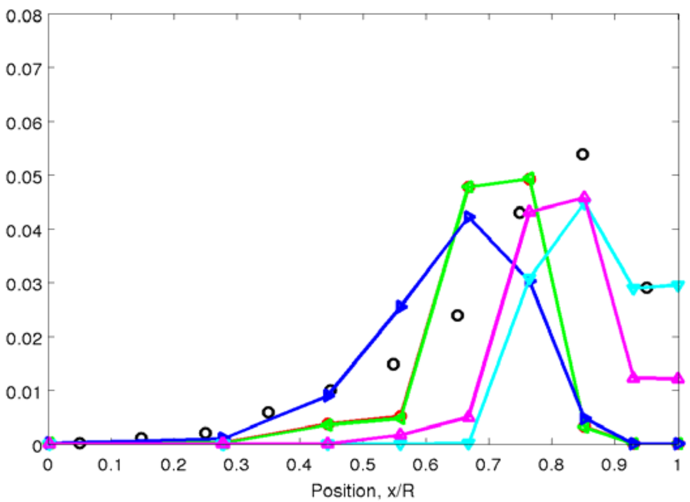
- Burns (2004)
- Bertodano (RPI) (1992)
- Gosman (1992)
- None

6. Two-phase solver development and validation

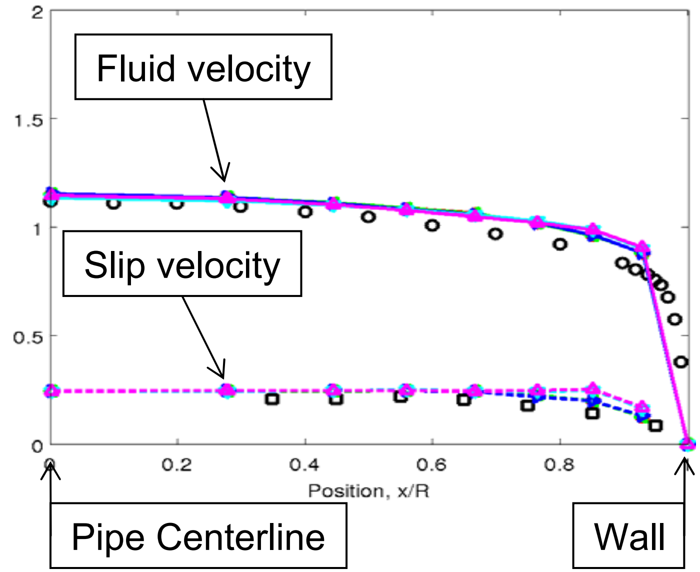
The first test cases are **vertical bubbly flows**: Hosokawa & Tomiyama (2009)
FZD Rossendorf MT-Loop 074

Example of results: **Hosokawa & Tomiyama (2009) Case 3**:
Different bubble induced turbulence and wall lubrication force models:

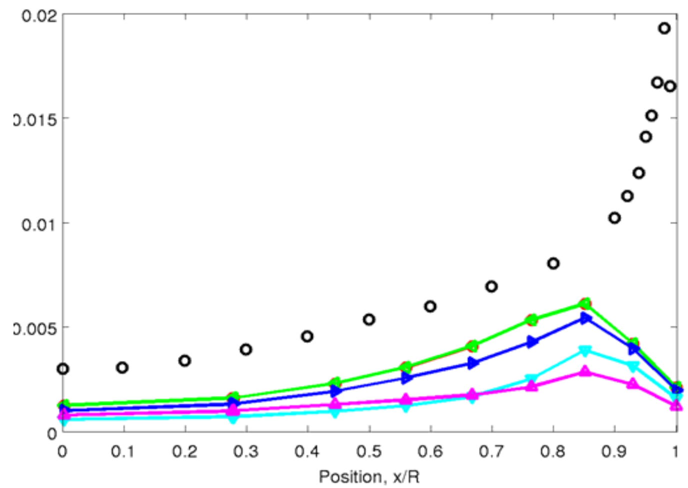
Gas volume fraction



Velocities



Turbulent kinetic energy



6. Two-phase solver development and validation

Velocities and volume fraction behave qualitatively correctly.

Different interfacial models affect the flow as expected.

Problematic turbulence modelling:

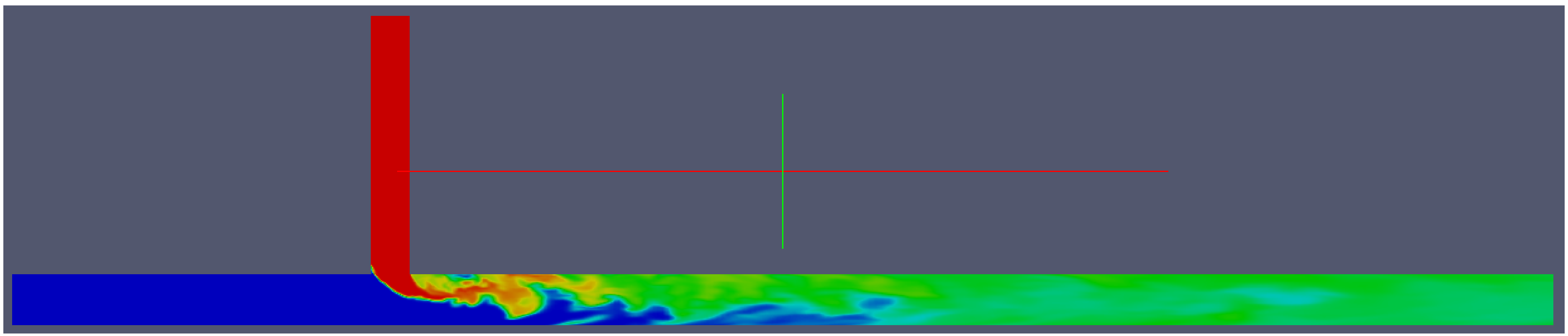
- Doesn’t match single-phase results at the dilute limit!
- Predicted turbulence kinetic energy is significantly lower than experimental results in Hosokawa & Tomiyama (2009) test case.

Planned and on-going development:

- Turbulence model overhaul
- Heat transfer
- Variable material properties, compressibility
- Boiling and condensation

7. Simulation of the T-junction Flow

- A blind benchmark exercise was arranged by OECD/NEA in 2010 to simulate turbulent mixing of warm and cold flows in a T-junction
- Transient information of the mixing is important, because heat fluctuations create stresses on the ducts, which may eventually lead to cracks and leaks.



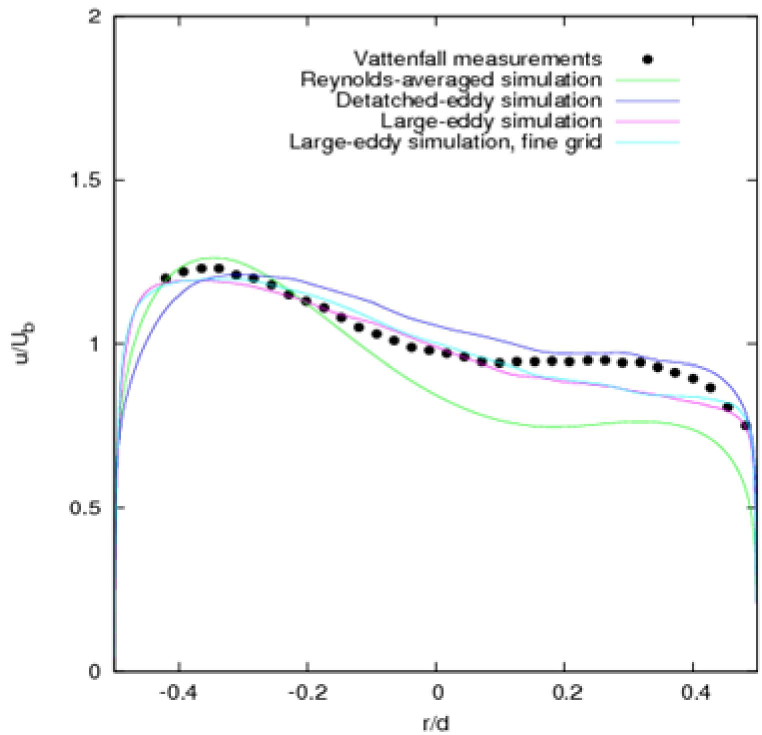
Instantaneous temperature distributions from detached-eddy simulation.

7. Simulation of the T-junction Flow

- Three common turbulence modelling approaches were applied in the work
  - Large-eddy simulation (LES)
    - Large-scale turbulence is simulated time-accurately and subgrid-scale effects are modelled
    - Dynamic Smagorinsky turbulence model applied (Piomelli and Liu)
  - Reynolds-Averaged Navier-Stokes simulation (RANS)
    - Turbulence effects are completely modelled, and only time-averaged solution is solved in the simulation
    - SST k-omega turbulence model
  - Detached-eddy simulation (DES)
    - Affordable version of LES. The flow is solved like RANS near surfaces, but like LES elsewhere.
    - Spalart-Allmaras turbulence model (delayed version)

7. Simulation of the T-junction Flow

- Verdict of the results
  - LES gave satisfactory results in the mixing flow, but finer grid would be necessary to simulate turbulent boundaries accurately.
  - LES results ranked **5/29 in velocity** and **21/29 in surface temperature** comparisons.
  - DES gave improved flow distributions transient heat fluctuations near surfaces. The temperature error is still over 5% of total range in places.
  - RANS solution was stable and transient fluctuations were not apparent. The flow distributions developed into incorrect state.



Time-averaged velocity field at a distance of 3.6 D from the junction on a vertical line

Summary: single-phase

Excellent results in the laminar horizontal cylinder test case.

- Validation of the basic flow and heat transfer solutions.

Validation of the RANS turbulence model near wall treatment:

- Satisfactory, but not perfect.
- The *k-ε* model gave more accurate results on coarse meshes.
- SST *k-ω* model gave reasonably good results on all the meshes.

Conjugated heat transfer:

- Flow solution matched the validated single-region results
- Heat conduction in the pipe wall matched analytical results.
- Confirmed to work with simultaneous compressible an incompressible regions.

OECD/NEA T-junction:

- In the blind exercise our LES results ranked 5/29 in velocity and 21/29 in surface temperature comparisons.

Summary: two-phase

The existing OpenFOAM® two-fluid solvers lack important sub models for bubbly flows.

A new solver was created to enhance two-phase capabilities

- Qualitatively the results behave as expected
- Turbulence model produces incorrect results
- The solver still requires significant development for nuclear safety applications.

Since the end of the two-phase portion of this project, weaknesses in the two-phase turbulence modelling that affect the results have been identified and corrected in a VTT internal development version of the solver, and partially in the official 1.7.x release.