Department for Business, Energy and Industry Strategies (BEIS)

Smart Models for Reactor Components

Project FORTE - Nuclear Thermal Hydraulics Research & Development

Personnel involved:

Dr Bo Liu *
Prof Shuisheng He *
Dr Charles Moulinec †
Dr Juan Uribe ‡

* Department of Mechanical Engineering, The University of Sheffield, Sheffield, S1 3JD, UK
† Science and Technology Facilities Council, Daresbury Laboratory, Warrington, WA4 4AD, UK
‡ EDF Energy R&D UK Centre, Manchester, M13 9PL, UK

Outline of this presentation

1. Development of a novel Coarse-Grid Sub-Channel CFD

1.1 Background

1.2 Methodology development

1.3 Validation & Application

1.3.1 Validation through the baseline model

1.3.2 Mesh matter

1.3.3 Application for 3-D complex flows

1.4 Conclusions

2. A benchmarking study of a 2x2 SCWR bundle

2.1 Overview

2.2 Experimental rig

2.3 Model descriptions

2.4 Results and discussion

2.5 Conclusions

1. Development of Sub-Channel CFD (SubChCFD)

1.1 Background

Motivation

- Reactor design/safety cases rely on the system/sub-channel codes (0/1-D).
- CFD becomes more mature and starts playing a role in assisting design/safety cases.
- Can CFD be used as a **mainstream** tool in reactor design?

0/1-D approach		Conventional CFD
System/sub-channel code	**Coarse-Grid CFD**	**RANS** **Hybrid RANS/LES** **LES** **DNS**

Advantages

0/1-D approach

- Mature in technology
- Abundant validated empirical correlations
- Low computing cost
- Routinely use in engineering

Conventional CFD

- High resolution results
- Detailed information of the local flow
- Low dependence on experience
- High flexibility for non-standard operation scenarios
- Potential to account for complex conditions

Disadvantages

0/1-D approach

- Only works for the validated cases it covers
- Cannot account for ‘complex’ conditions (e.g. fuel distortion, buoyancy, cross flow)
- Limited information in local flow details
- High dependence on experience

Conventional CFD

- High computing cost, especially the case of high fidelity methods
- Long simulations turnover time
- Application restrictions in Re (e.g. DNS) and/or domain size
- Difficulties to quantify uncertainties

1.1 Background

Vision

To develop a modern **CFD-based** **'sub-channel'** framework for nuclear power plant (SubChCFD)

- Combining advantages of CFD & traditional engineering tools.
- Based on a standard CFD solver whilst embracing the sub-channel correlations for model closure.
- using a two-level mesh system,

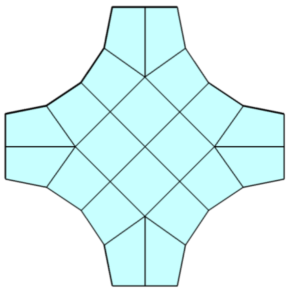
  (i) A **Coarse-grid computing mesh**, on which a typical CFD solver is used to resolve the inviscid flow with corrections for diffusion and turbulent mixing.

  (ii) A **filtering mesh**, which aligns with the mesh used in the traditional methods (sub-channel codes), enabling the use of existing engineering correlations to account for the integral effects of wall shear and heat transfer.

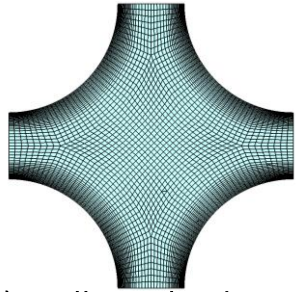
1.1 Background

Benefits & advantages

- Significant reduction in computing cost compared to conventional CFD

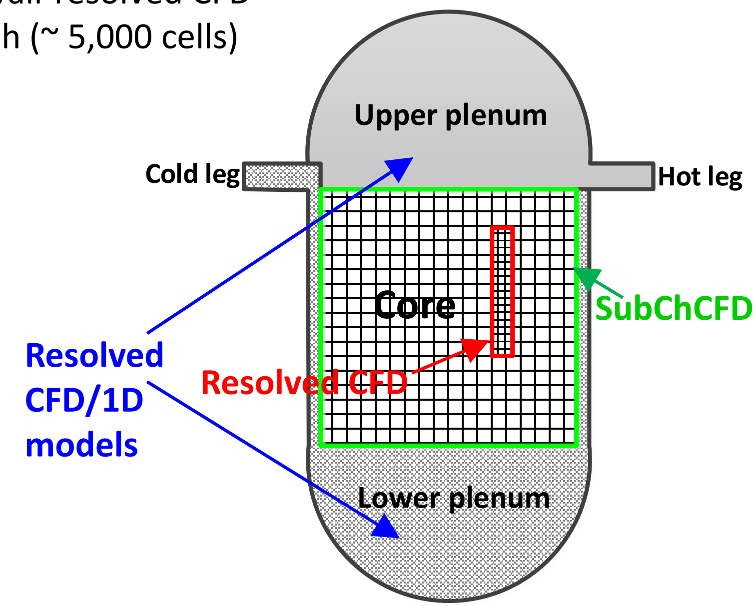


(a) SubChCFD mesh (20 cells)

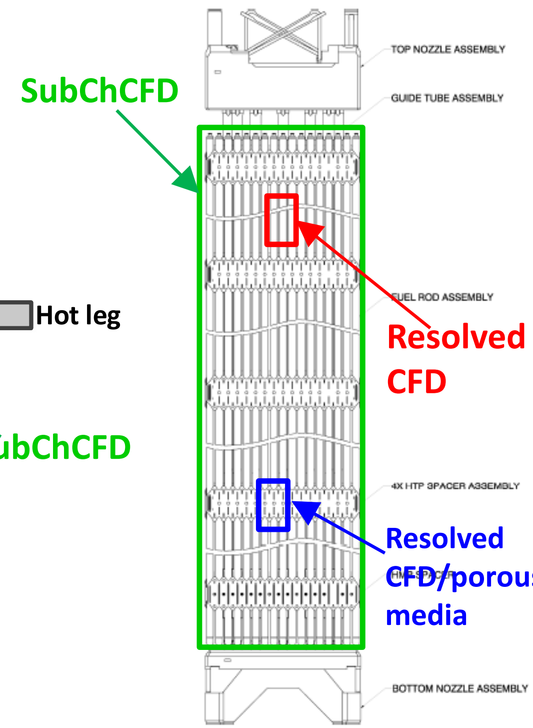


(b) Wall-resolved CFD mesh (~ 5,000 cells)

- Potential to be applied for very large systems
- Overall Full 3-D result
- Ease of coupling with resolved CFD and/or porous media approach
- Providing locally desirable resolution of the results for regions of interest



A whole reactor core



A full fuel assembly

1.2 Methodology

Details of concepts

Computing mesh

- Using very coarse mesh to reduce computing cost (mesh resolution between CFD and the sub-channel codes)
- Large structures (e.g. fuel rods) are meshed explicitly
- Complex structures (e.g. spacers) are accounted for using various methods, including,
  - Porous medium
  - Momentum source term
  - Coupling with resolved CFD

Core flow region

- Inviscid flow is captured using the coarse mesh
- Turbulent mixing is accounted for using simple turbulence model/correlations
- Model parameters are calibratable according to specific circumstances

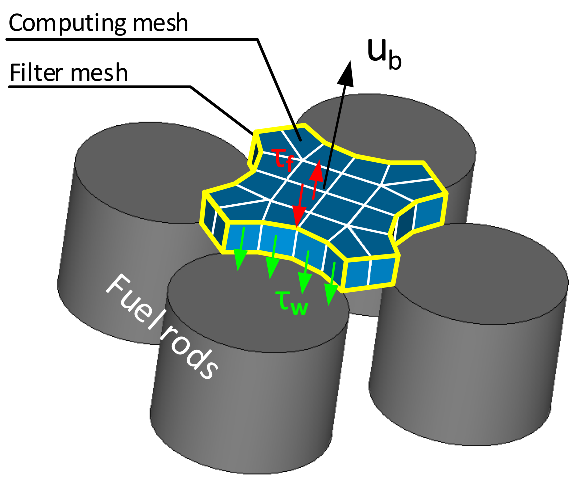
Near-wall region

- Using engineering standard correlations to ensure correct integral effects of friction and heat transfer
  - Sub-channel-based frictional factor correlation
  - Sub-channel-based Nusselt number correlation

1.2 Methodology

Implementation

- **Currently implemented in Code_Saturne**



- **Two-level mesh system**
  - Filtering mesh

    Obtaining sub-channel-level quantities by spatial averaging

$$\varphi_{sub,j} = \sum_{i\in j}\left(\frac{V_i}{V_{sub,j}}\right)\varphi_i$$

  - Computing mesh

    Solving governing equations

$$\frac{\partial \rho\vec{u}}{\partial t} + \nabla\cdot\rho\vec{u}\otimes\vec{u} = -\nabla p + \nabla\cdot\bar{\sigma}$$

$$\iiint_{\Omega}\frac{\partial \rho\vec{u}}{\partial t}dV + \oiint_{S}\vec{u}\left(\rho\vec{u}\cdot\vec{n}\right)dA = -\oiint_{S}\left(\bar{I}p\cdot\vec{n}\right)dA + \oiint_{S}\bar{\sigma}\cdot\vec{n}dA$$

$$= -\oiint_{S}\left(\bar{I}p\cdot\vec{n}\right)dA + \iint_{S_w}\bar{\sigma}\cdot\vec{n}dA + \iint_{S_f}\bar{\sigma}\cdot\vec{n}dA$$

where $S = S_w \cup S_f$

$$\iint_{S_f}\bar{\sigma}\cdot\vec{n}dA = \iint_{S_f}\left(\mu+\mu_t\right)\left(\nabla\vec{u}+\nabla\vec{u}^T - \frac{2}{3}\delta\nabla\cdot\vec{u}\right)\cdot\vec{n}dA$$

$$\mu_t = \rho l_m^{\ 2}\sqrt{2S_{ij}S_{ij}}$$

$$l_m = 0.09\delta$$

$$\iint_{S_w}\bar{\sigma}\cdot\vec{n}dA = -\frac{1}{4}f\,\frac{1}{2}\rho_{sub}\vec{u}_{sub}\left|\vec{u}_{sub}\right|\iint_{S_w}dA$$

$$f = \left[a + b_1(P/d-1) + b_2(P/d-1)^2\right]/\mathrm{Re}^n$$

Implementation

- **Engineering standard correlations for Square-lattice rod bundles**

  - Frictional factor correlation

$$f = \left[a + b_1(P/D-1) + b_2(P/D-1)^2\right]/\mathrm{Re}^n$$

Sub-channel type	1.0 ≤ P/D ≤ 1.1			1.1 ≤ P/D ≤ 1.5		
	a	b_1	b_2	a	b_1	b_2
Laminar						
Interior	26.37	374.2	-493.9	35.55	263.7	-190.2
Edge	26.18	554.5	-1480	44.40	256.7	-267.6
Corner	28.62	715.9	-2807	58.83	160.7	-203.5
Turbulent						
Interior	0.09423	0.5806	-1.239	0.1339	0.09059	-0.09926
Edge	0.09377	0.8732	-3.341	0.143	0.04199	-0.04428
Corner	0.09755	1.127	-6.304	0.1452	0.02681	-0.03411

  - Nusselt number correlation

$$Nu_\infty = \psi (Nu_\infty)_{c.t.}$$

$$Nu_{\infty,c.t.} = 0.023\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.4}$$ when the fluid is heated

$$Nu_{\infty,c.t.} = 0.023\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.3}$$ when the fluid is cooled

$$\psi = 1 + 0.9120\,\mathrm{Re}^{-0.1}\,\mathrm{Pr}^{0.4}(1 - 2.0043e^{-B})$$

$$B = \frac{D_h}{D}$$

Conditions of availability,

$$3\times10^3 \le \mathrm{Re} \le 10^6$$

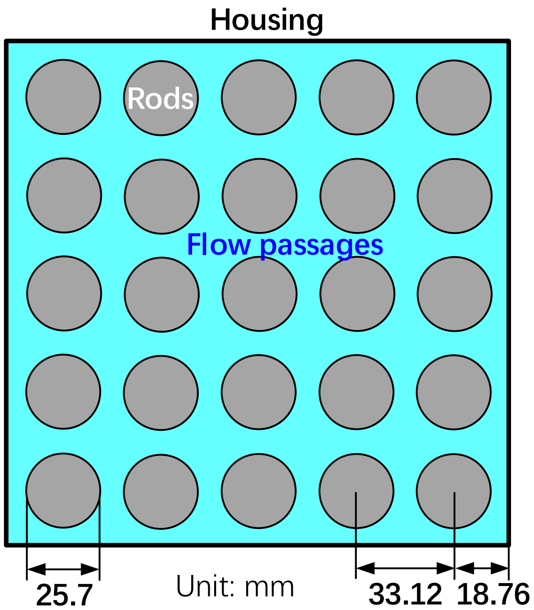
$$0.66 \le \mathrm{Pr} \le 5.0$$

$$1.0 \le \mathrm{P}/D \le 1.8$$

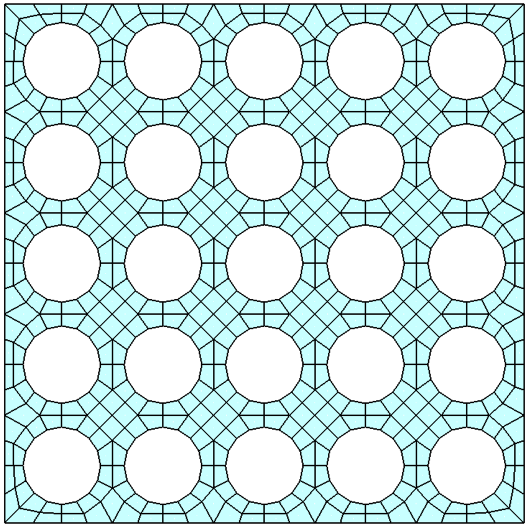
1.3 Validation & Application

2-D 5x5 bundle

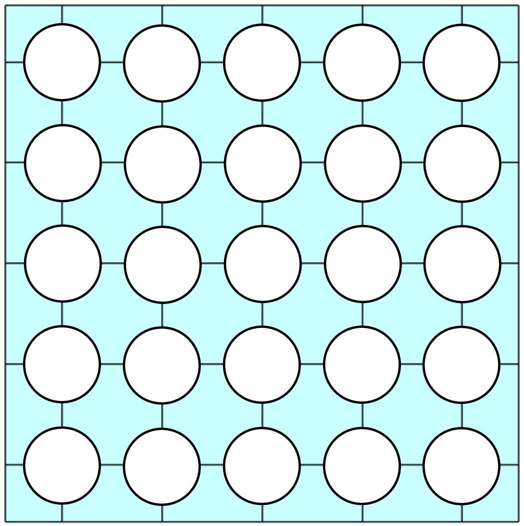
- **OECD/NEA MATiS-H benchmark experiment**
  - Bare bundle case
  - About 3 time bigger than a real PWR rod bundle
  - Axial periodic boundaries (pseudo 2-D)
  - Water at T=35 °C, p=1.5 bar, u_m=1.5 m/s, Re=50250, Q = 200 kW/m^2
  - $\bar{D}_h$ = 0.0243 m, P/D = 1.3



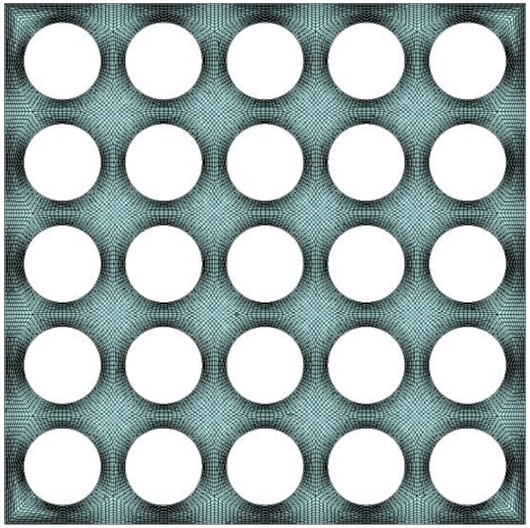
- **Mesh**



Computing mesh



Filtering mesh

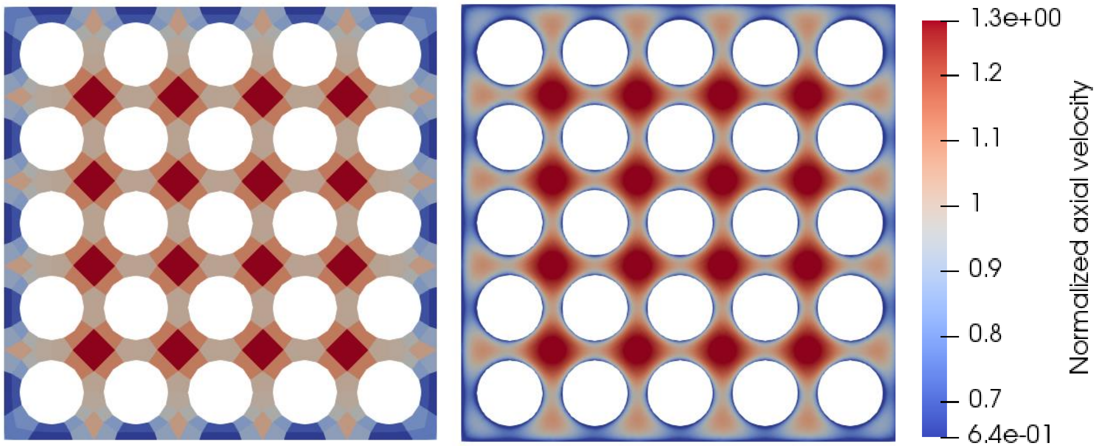


CFD mesh (reference model)

1.3.1 Validation through the baseline model

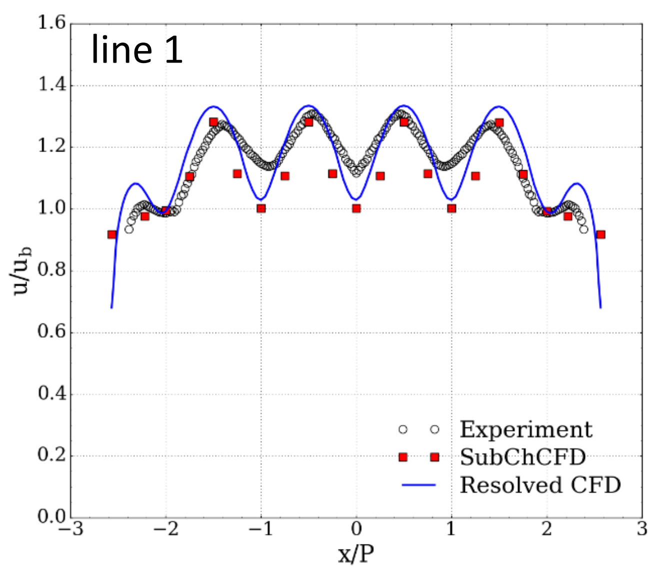
Case <u>2D 5^2A</u> (5x5 bundle with uniform heating)

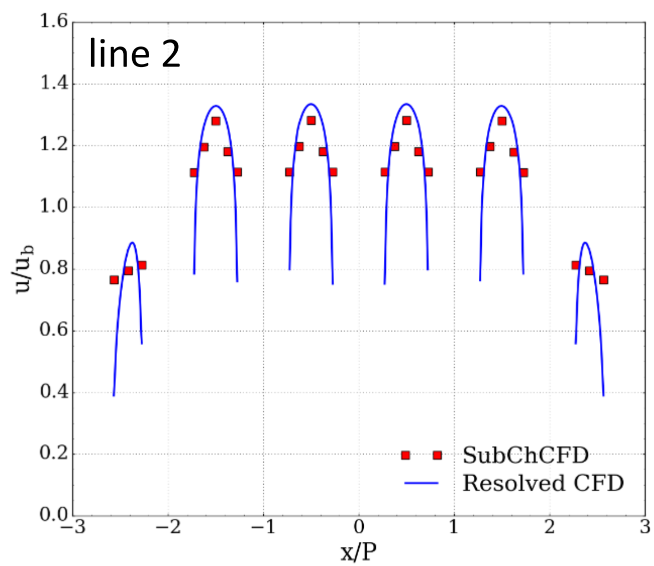
- **Velocity field**

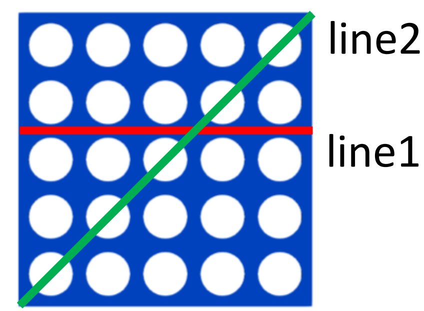


(a) SubChCFD (b) Resolved CFD

- Experimental data is available for Line-1
- Performing as expected
- Some details of the flow pattern can be captured



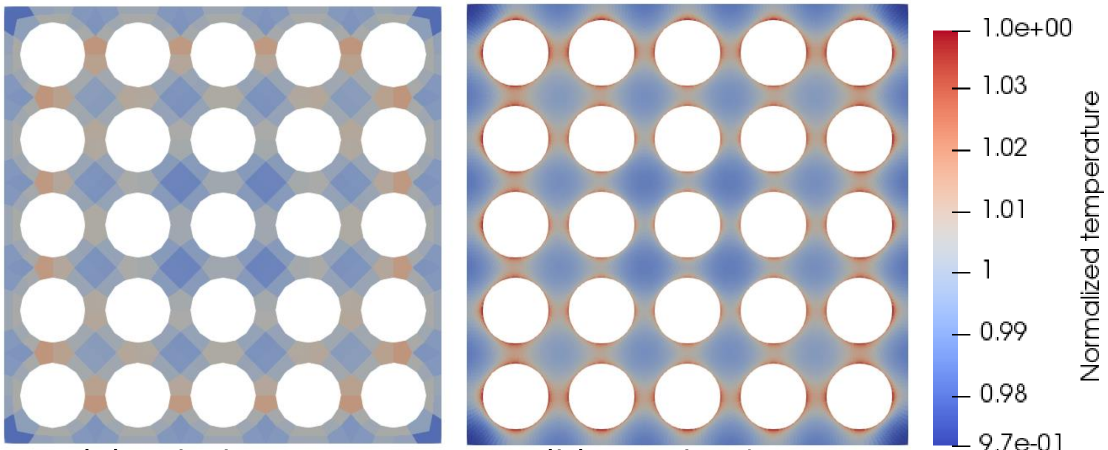




1.3.1 Validation through the baseline model

Case 2D 5^2A (5x5 bundle with uniform heating)

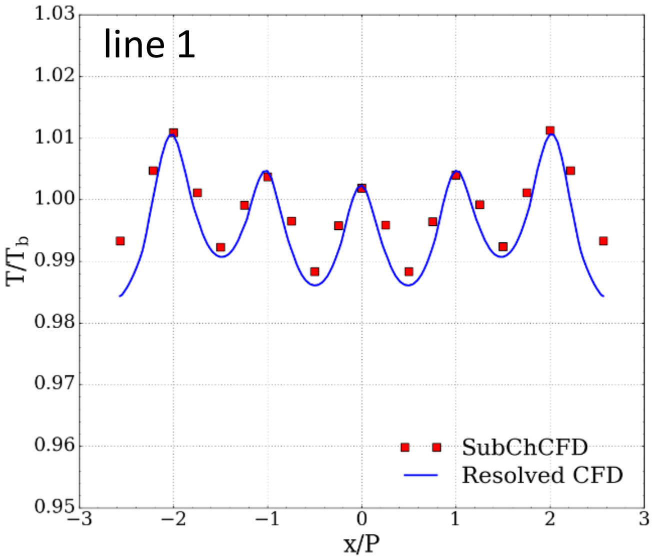
- **Temperature field**

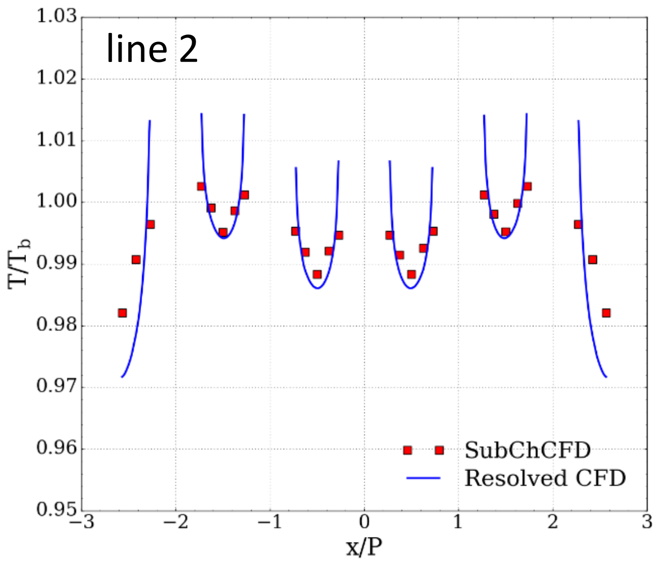


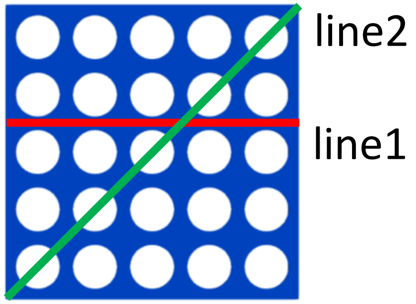
(a) SubChCFD (b) Resolved CFD

- Heat sink is added to the energy equation for thermal periodicity

$$S_{ad} = \frac{\Phi_{net}\vec{u}\cdot\vec{e}_z}{\int_\Omega \vec{u}\cdot\vec{e}_z d\Omega} = \frac{\Phi_{net}u_z}{\int_\Omega \vec{u}_z d\Omega}$$



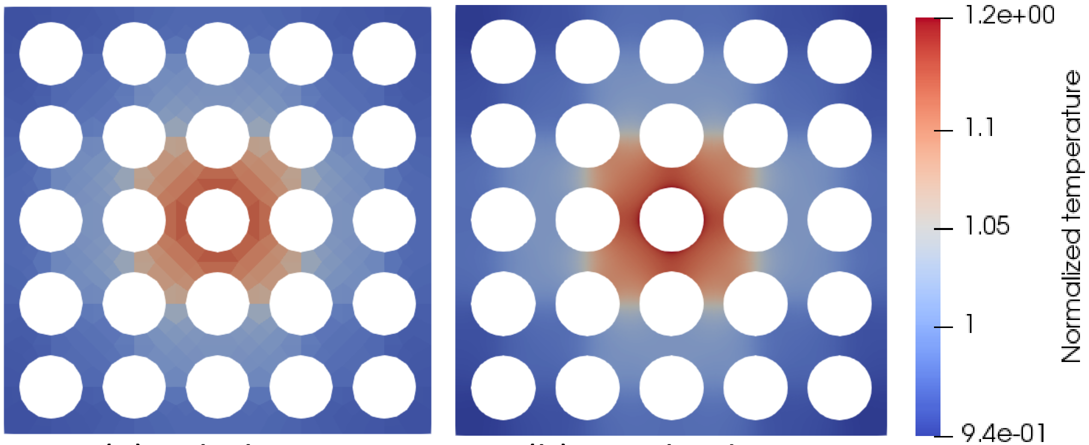




1.3.1 Validation through the baseline model

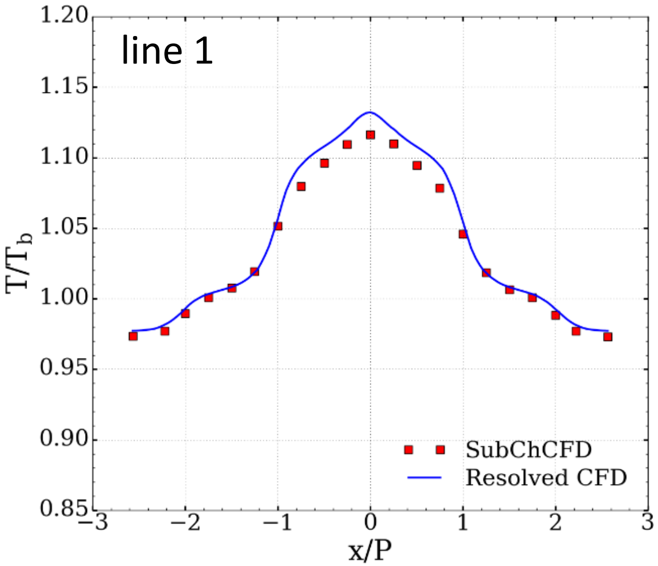
Case 2D 5^2B (5x5 bundle with non-uniform heating)

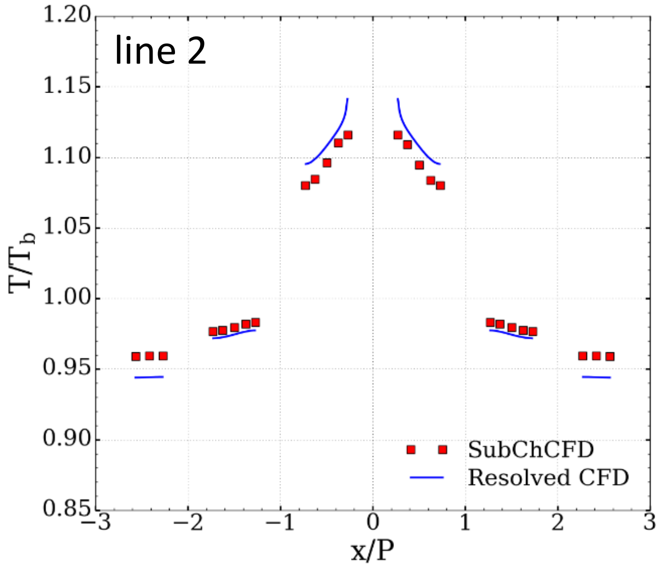
- **Temperature field**

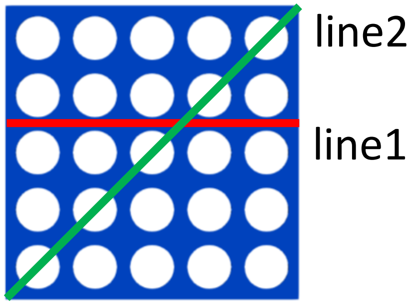


(a) SubChCFD (b) Resolved CFD

- Only the centre rod with heating (200 kW/m^2)
- Adiabatic walls for other rods and the housing
- Trend of temperature distribution is well captured



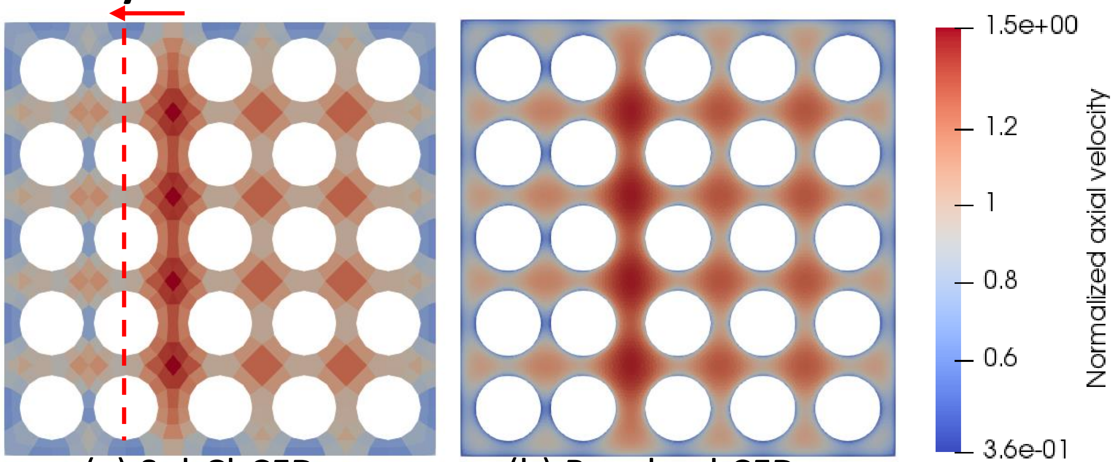




1.3.1 Validation through the baseline model

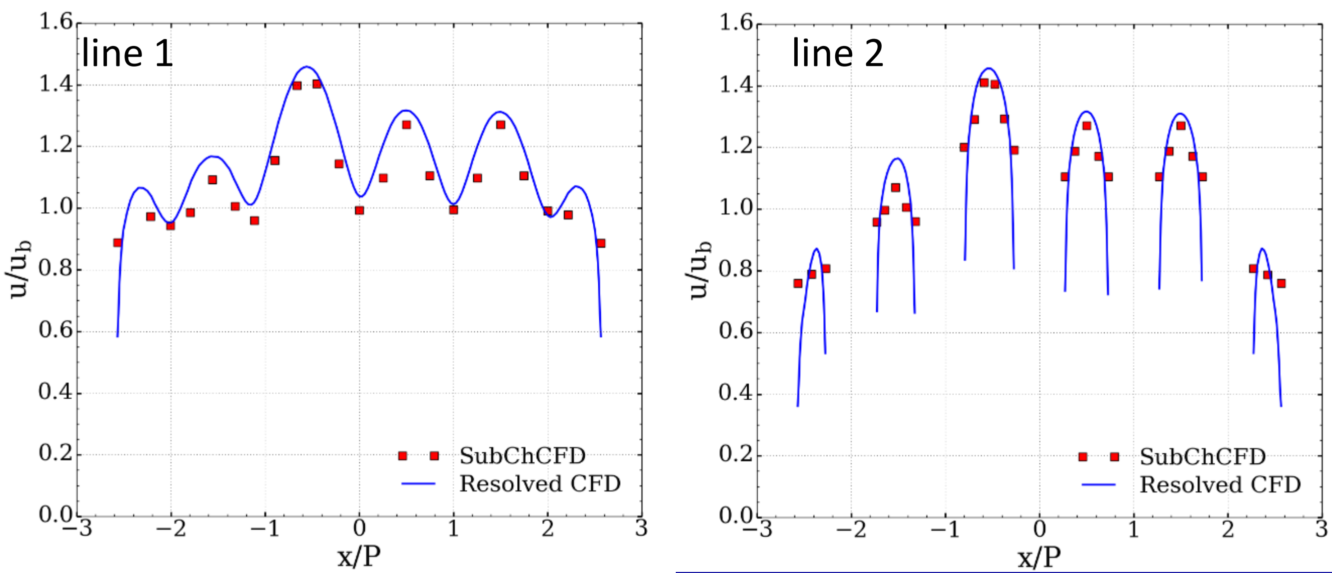
Case 2D 5^2C (Distorted 5x5 bundle)

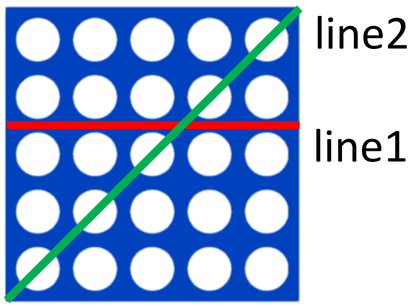
- **Velocity field**



(a) SubChCFD (b) Resolved CFD

- One column of the rods shifted to one side
- Mesh is slightly modified to fit the new geometry
- Velocity re-distribution is well captured

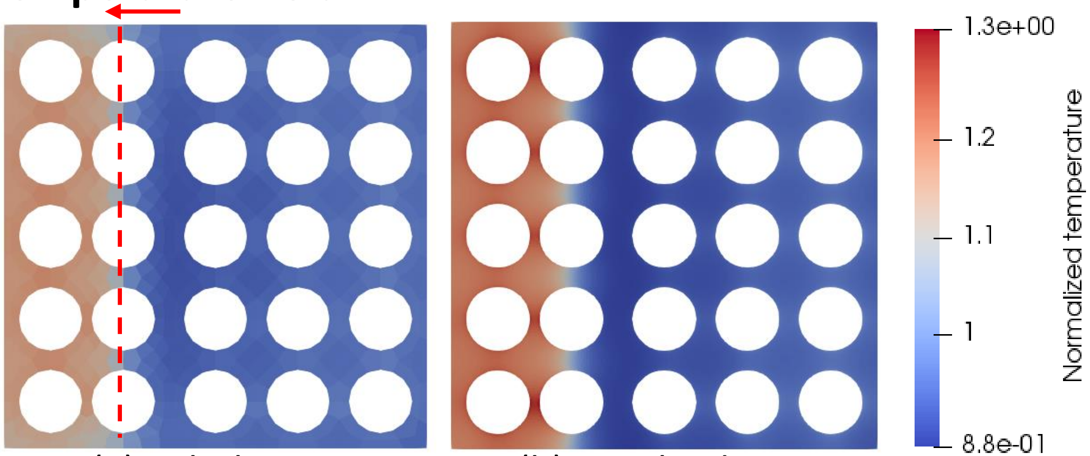




1.3.1 Validation through the baseline model

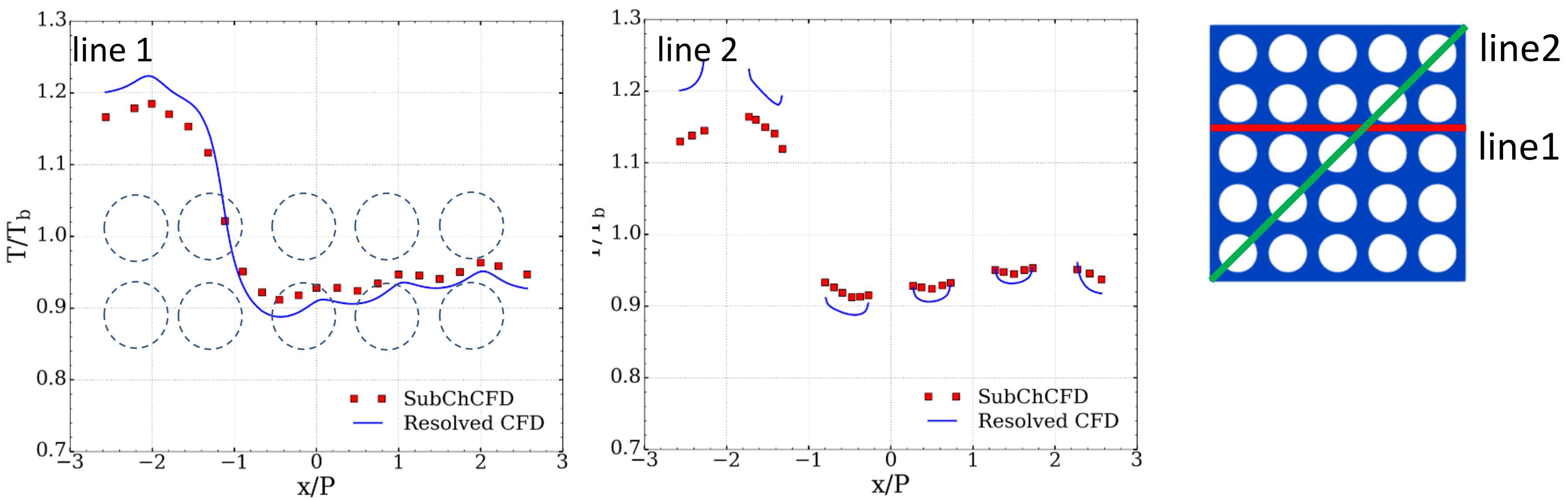
Case 2D 5^2C (Distorted 5x5 bundle)

- **Temperature field**



(a) SubChCFD (b) Resolved CFD

- Temperature re-distribution is also well captured
- Magnitude may be improved by adjusting model parameters, like turbulent Prandtl number



1.3.2 Mesh matter

Mesh dependency test (2D 5x5 bundle)

Bare bundle case based on the OECD/NEA MATiS-H benchmark experiment (P/D=1.3, Re=50,250)

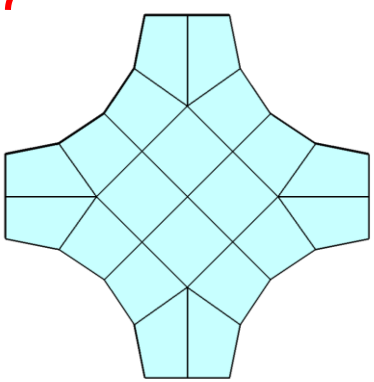
Resolved reference model

- ✓ Axial periodic
- ✓ K-epsilon model (non-wall cells)
- ✓ Scalable wall function (wall cells)
- ✓ No. of mesh cells: 54,528

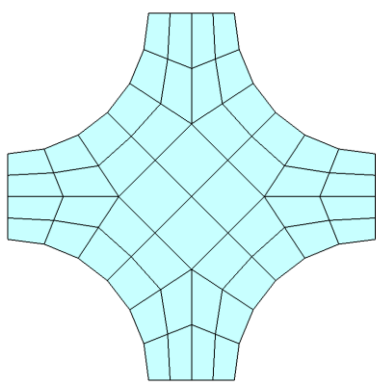
Coarse-grid model

- ✓ Axial periodic
- ✓ Mixing length model (non-wall cells)
- ✓ Sub-channel correlations (wall cells)
- ✓ No. of mesh cells: 1,120 (Mesh-1)
  1,568 (Mesh-2)
  2,176 (Mesh-3)

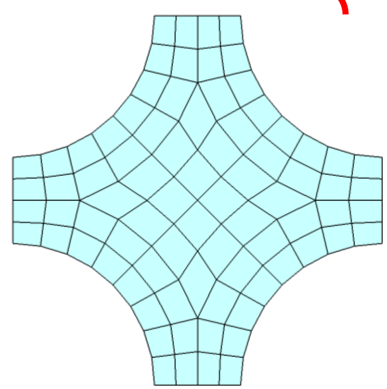
Three tested meshes for SubChCFD



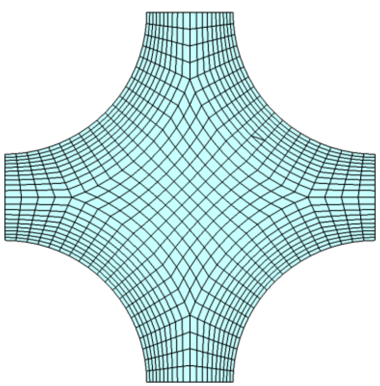
(a) Mesh-1



(b) Mesh-2



(c) Mesh-3

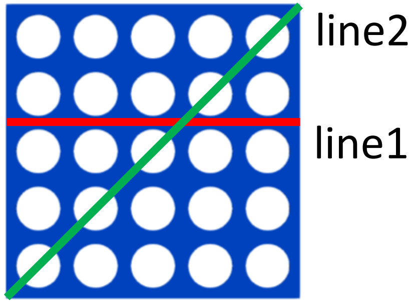


(c) CFD mesh (reference model)

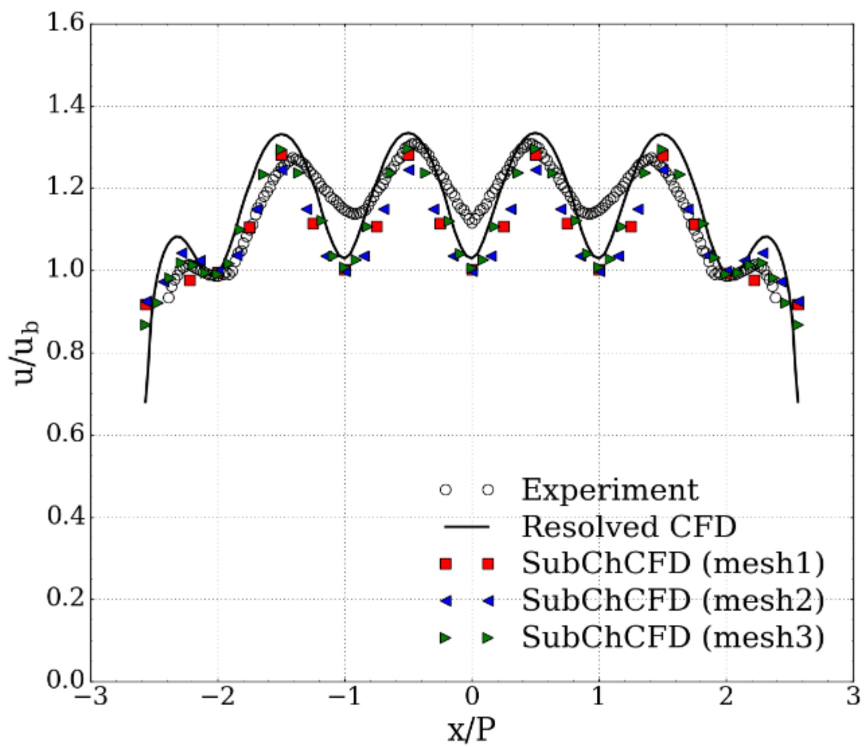
1.3.2 Mesh matter

Mesh dependency test (2D 5x5 bundle)

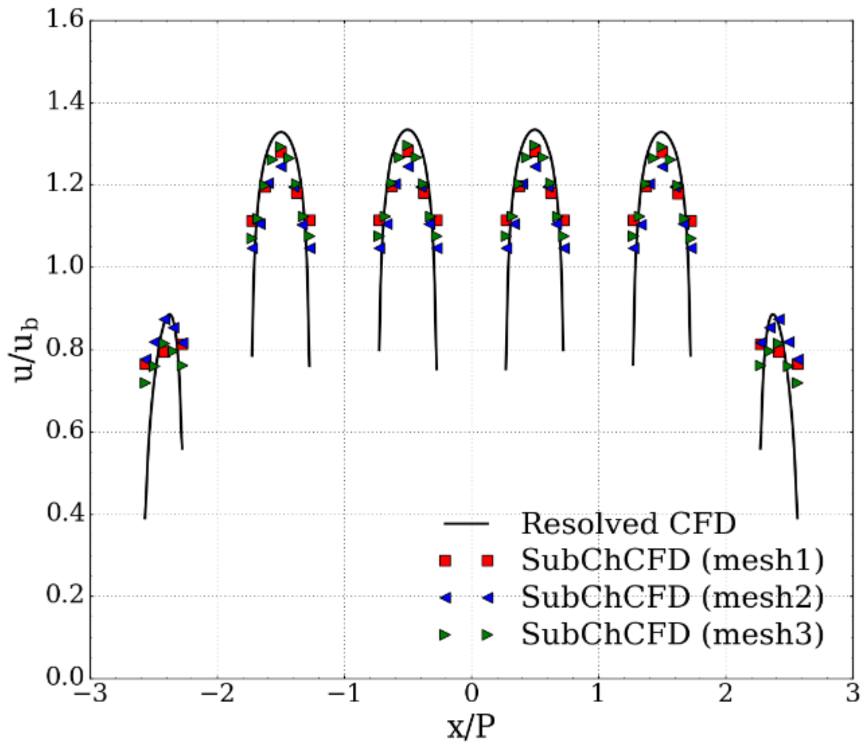
- Overall, results are **insensitive** to the three tested meshes
- Mesh-1 is slightly inadequate for the edge/corner sub-channels
- Results using Mesh-1 deviates slightly from the other two meshes



Axial velocity profiles



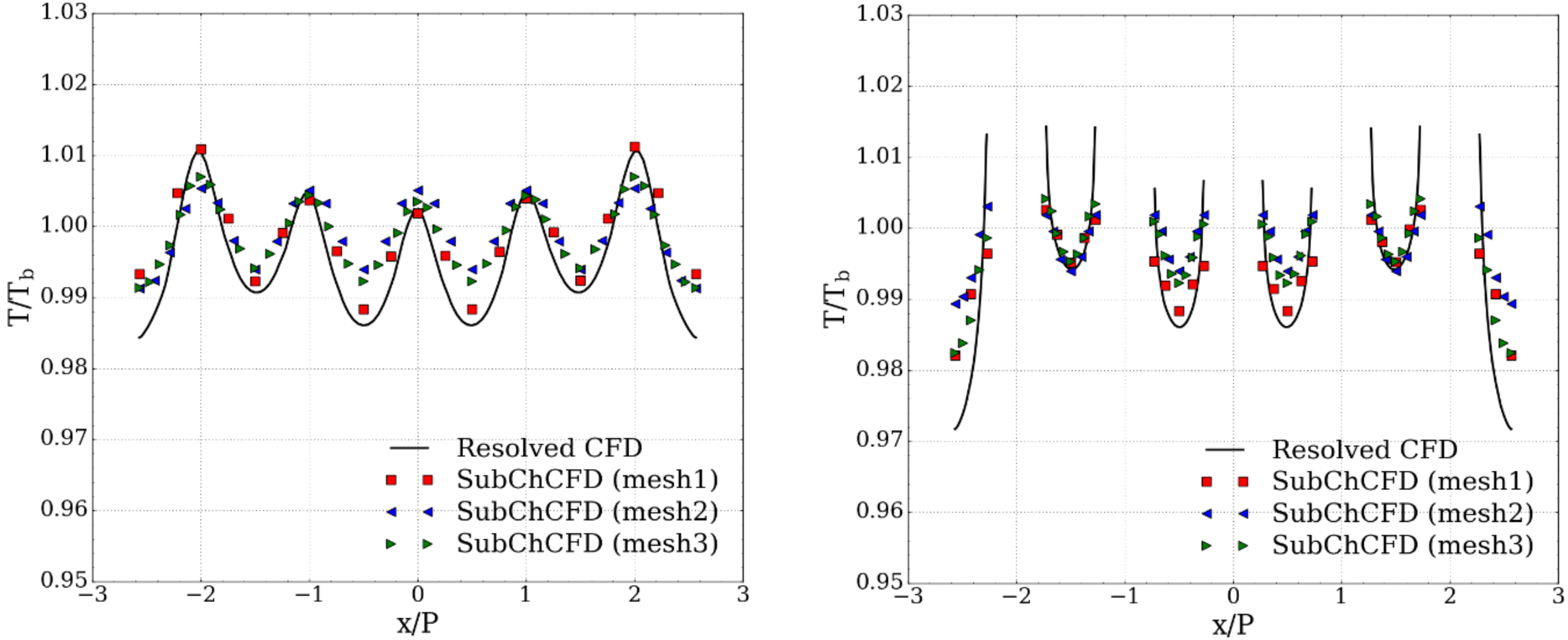
(a) Plot over line 1



(b) Plot over line 2

Mesh dependency test (2D 5x5 bundle)

Temperature profiles



(a) Plot over line 1

(b) Plot over line 2

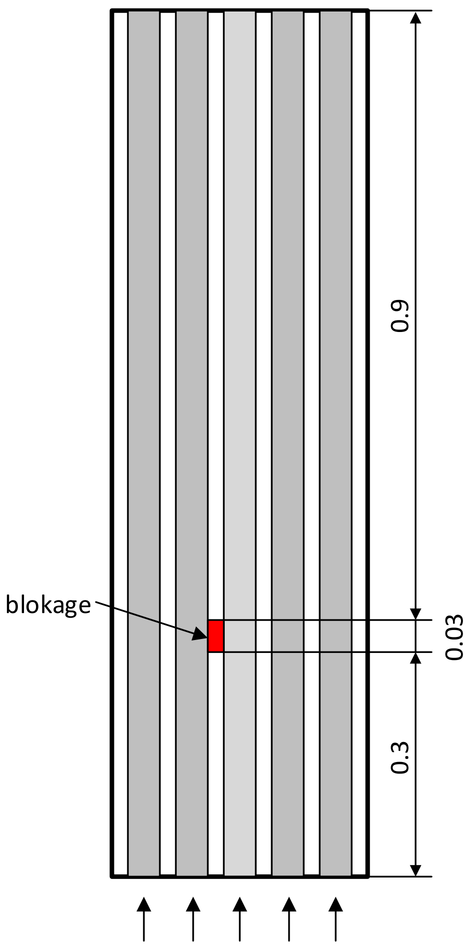
Mesh strategy for SubChCFD

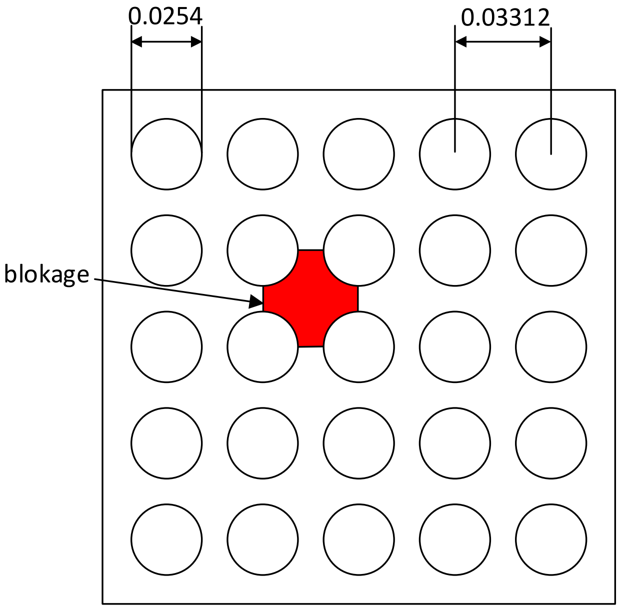
- Numerical diffusion may be significant for other cases, e.g. strong lateral flow
- The mesh generation strategy of SubChCFD needs to be **standardised**

1.3.3 Applications for 3-D complex flows

Case 3D 5^2D (5x5 bundle with local blockage)

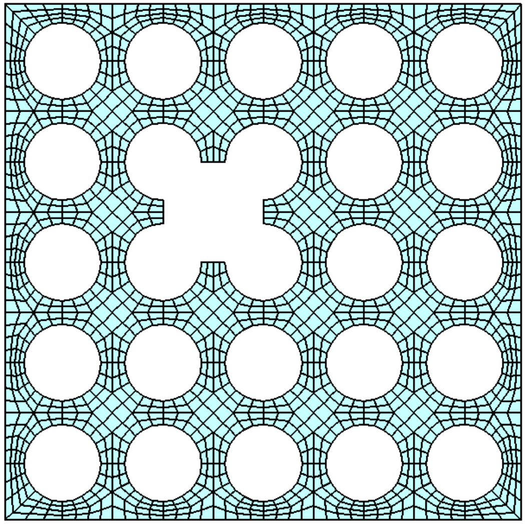
- **Geometry & Mesh**





Unit: m

- Fully developed inlet velocity (u_{mean}=1.5m/s)

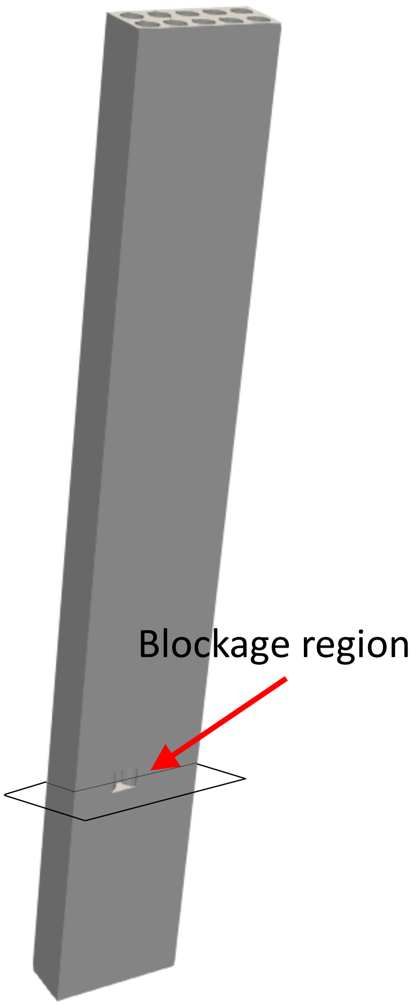


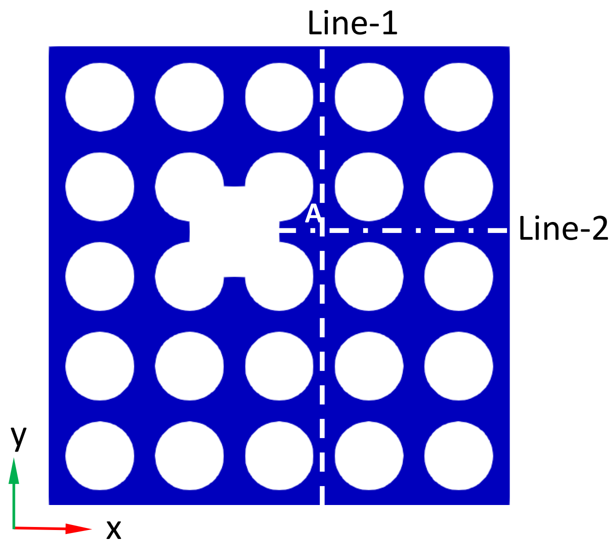
- Mesh resolution: based on Mesh-2
- Total number of cells: **0.645M** (**21M** for the reference model)

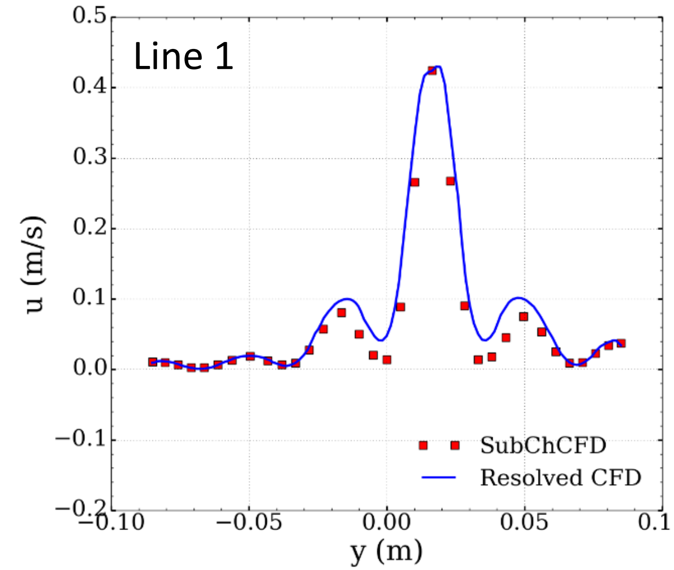
1.3.3 Applications for 3-D complex flows

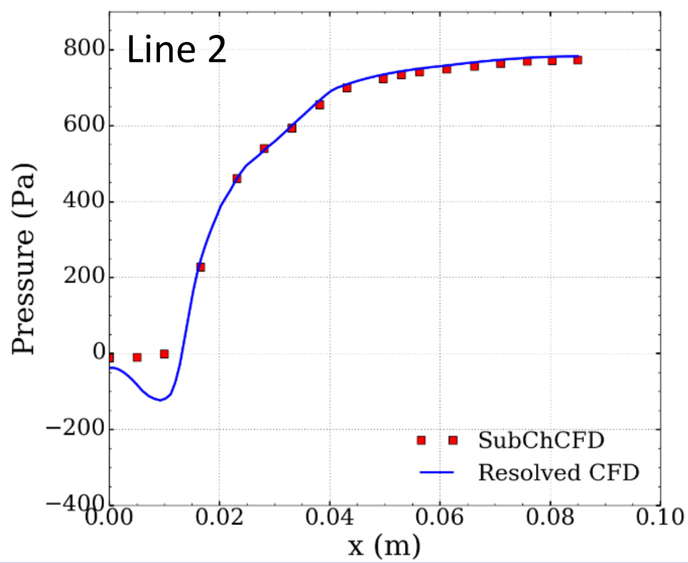
Case 3D 5²D (5x5 bundle with local blockage)

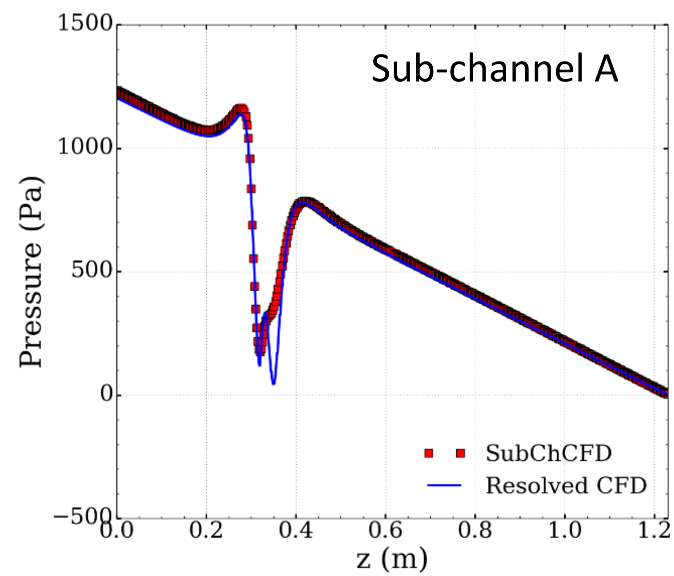
- **Results**







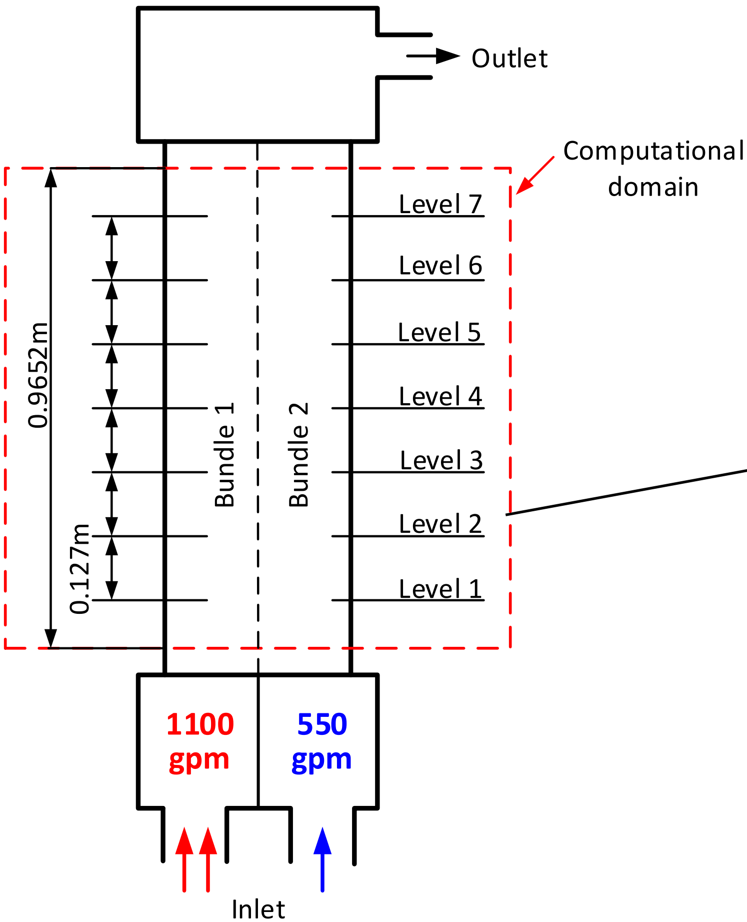




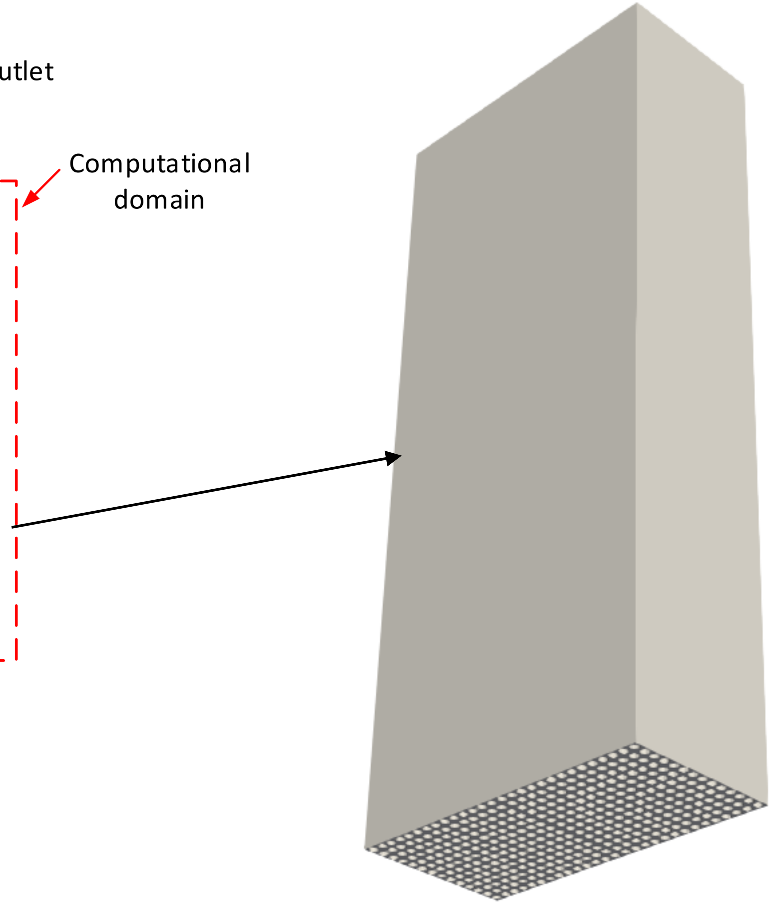
1.3.3 Applications for 3-D complex flows

Case 3D 14²A (Mixing in parallel assembly)

- **Computational domain**



Schematic of the facility[7]



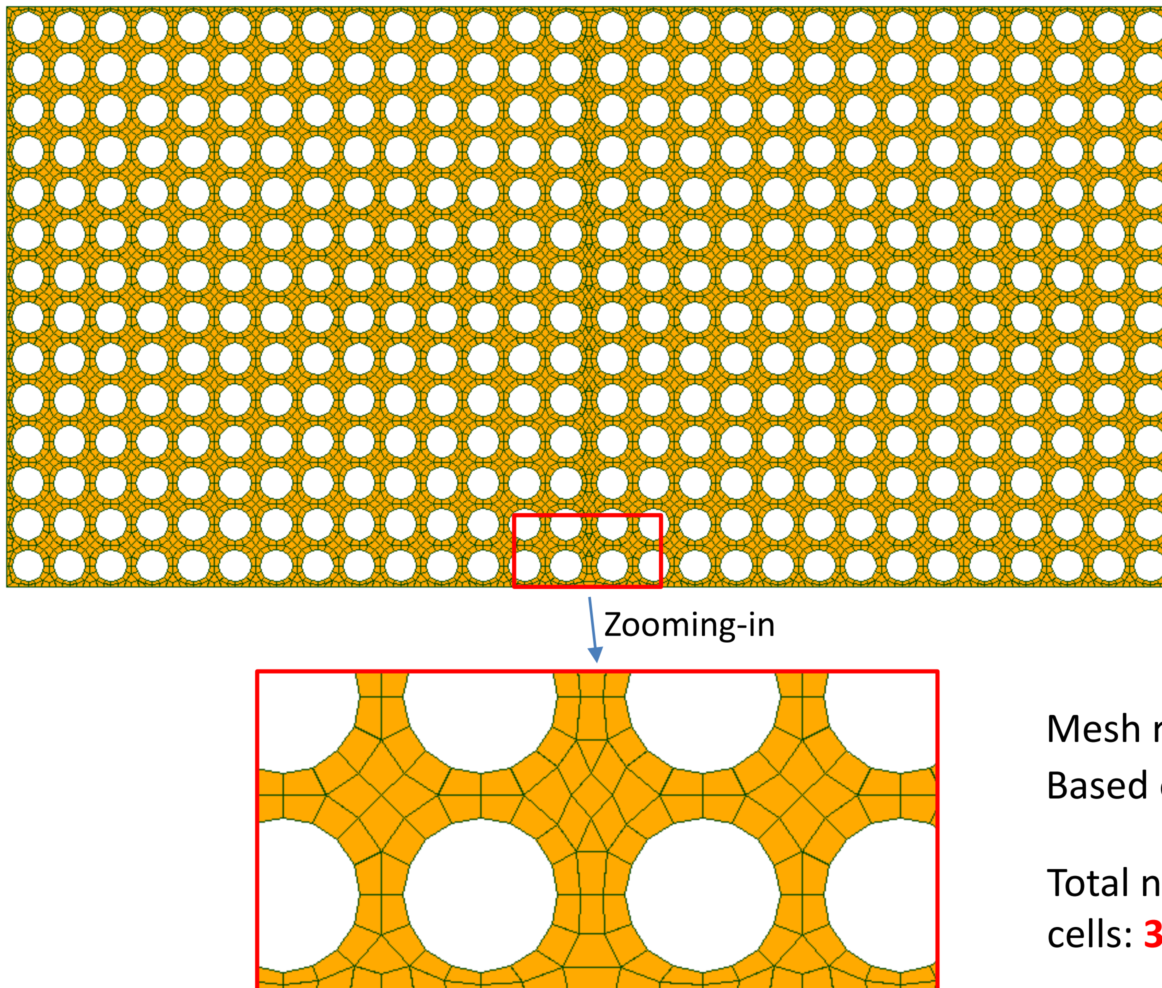
Parallel 14x14 bundle

- Different input flow rate
- D = 0.0108m
- P/D = 1.28

[7] S.J. Yoon, S.B. Kim, G.C. Park, H.Y. Yoon, H.K. Cho. Application of CUPID for Subchannel-scale Thermalehydraulic Analysis of Pressurized Water Reactor Core Under Single-phase Conditions. Nuclear Engineering and Technology, 50, 54-67 (2018)

Case 3D 14^2A (Mixing in parallel assembly)

- **Mesh**



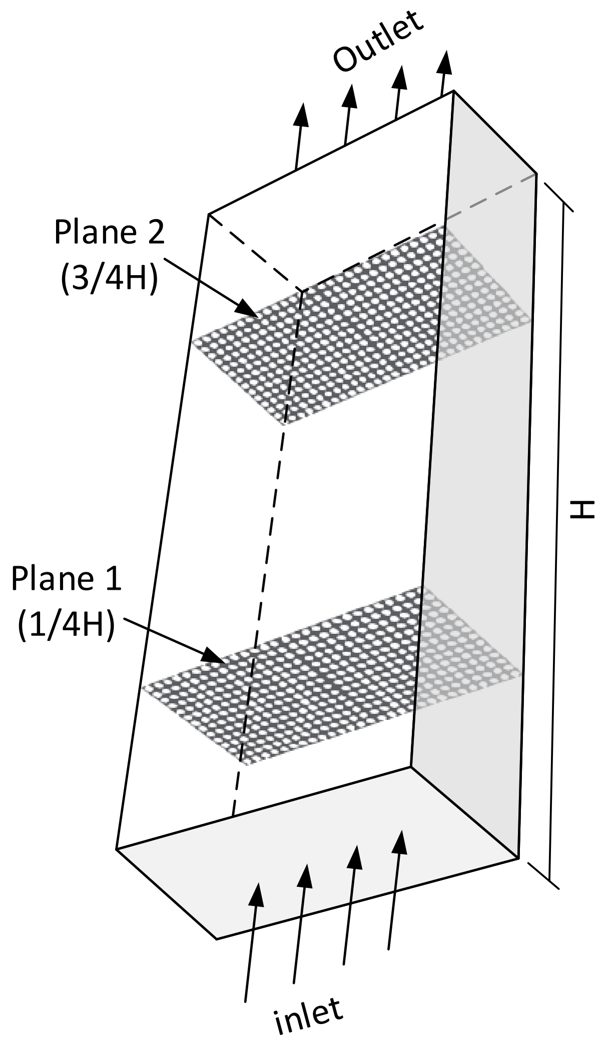
Mesh resolution:
Based on Mesh-1

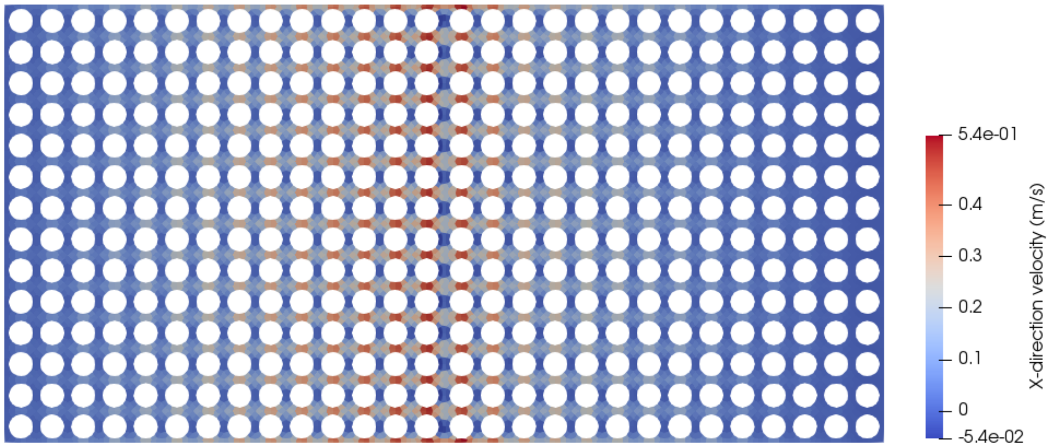
Total number of cells: **3.3M**

1.3.3 Applications for 3-D complex flows

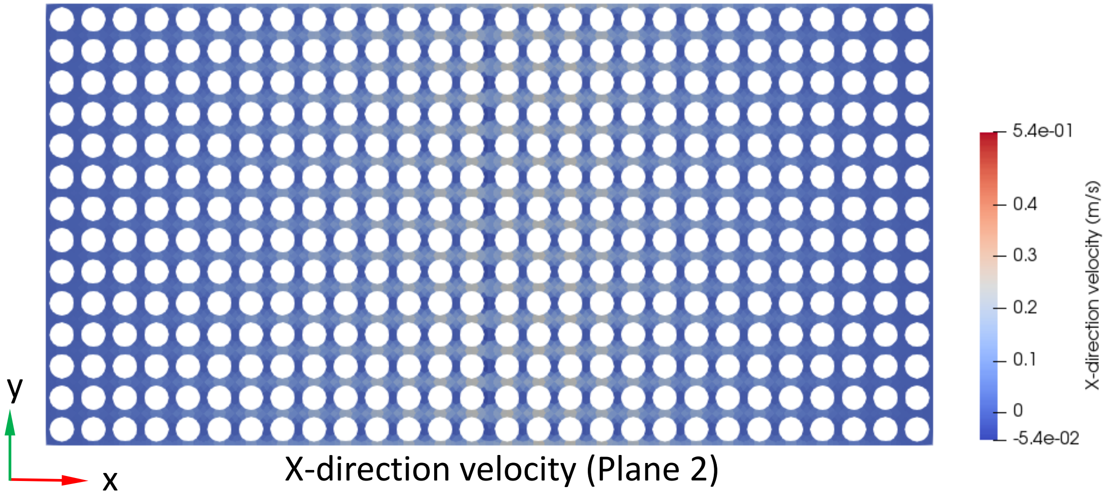
Case 3D 14²A (Mixing in parallel assembly)

- **Cross-flow velocity distribution**





X-direction velocity (Plane 1)

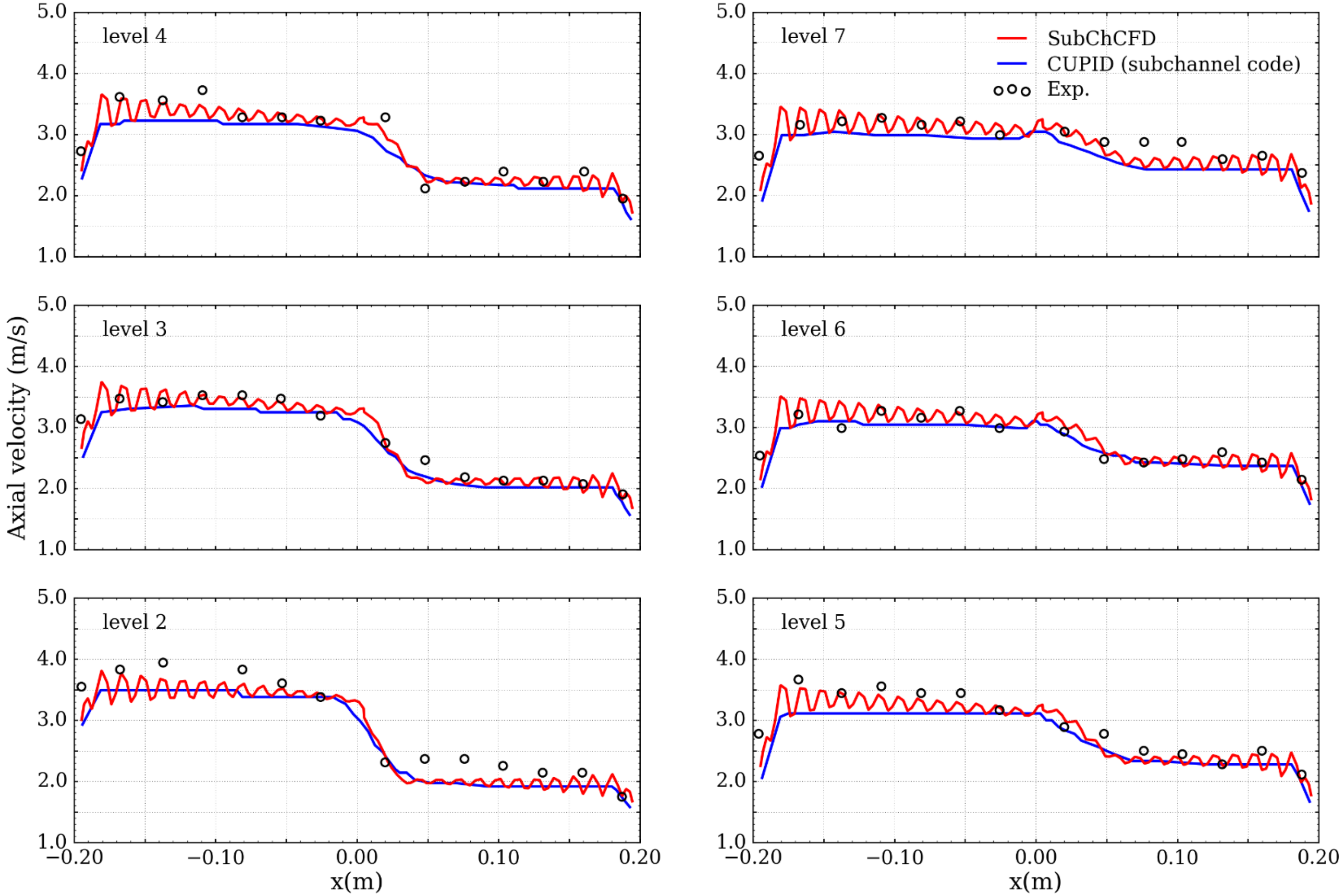


X-direction velocity (Plane 2)

1.3.3 Applications for 3-D complex flows

Case 3D 14^{2}A (Mixing in parallel assembly)

- **Comparison against experiment & sub-channel code CUPID**



1.4 Conclusions

- A CFD based Sub-channel framework (SubChCFD) has been developed to reduce the computational cost in modelling of large components/core of nuclear reactors;
- The model has to date been validated using the following test cases;

Case ID	Case description	momentum	Heat transfer	Validation method
2D1	Single sub-channel	✓	✓	Resolved CFD
2D5^{2}A	5x5 PWR bundle (regular)	✓	✓	Resolved CFD & Exp.
2D5^{2}B	5x5 PWR bundle (non-uniform heated)	✓	✓	Resolved CFD
2D5^{2}C	5x5 PWR bundle (with rods shifted)	✓	✓	Resolved CFD
3D5^{2}D	5x5 bundle with local blockage	✓		Resolved CFD
3D14^2	Parallel 14x14 bundle	✓		Sub-channel code & Exp.

- Overall, the model works very well and will be further developed for more complex scenarios,
  - Mixed/free convection
  - High non-equilibrium turbulence
  - Coupling feature

2. Benchmarking study of a 2x2 SCWR bundle

2.1 Overview

- **Why Super Critical Water-cooled Reactor (SCWR)?**
  - Recognised as one of the six proposed designs of the Gen IV advanced reactors
  - High thermal efficiency, compact system structure, and low capital cost

- **IAEA Coordinated Research Project (CRP)** on Understanding and Prediction of Thermal-Hydraulics Phenomena Relevant to SCWRs (**2015-2019**)
- The CRP organizes a number of **benchmark exercises** among other activities to establish an understanding of the capability of CFD for SCWR.

Third Research Coordination Meeting (RCM) in Madison, Wisconsin, USA, 26-29 June 2017

- **Members/participants**

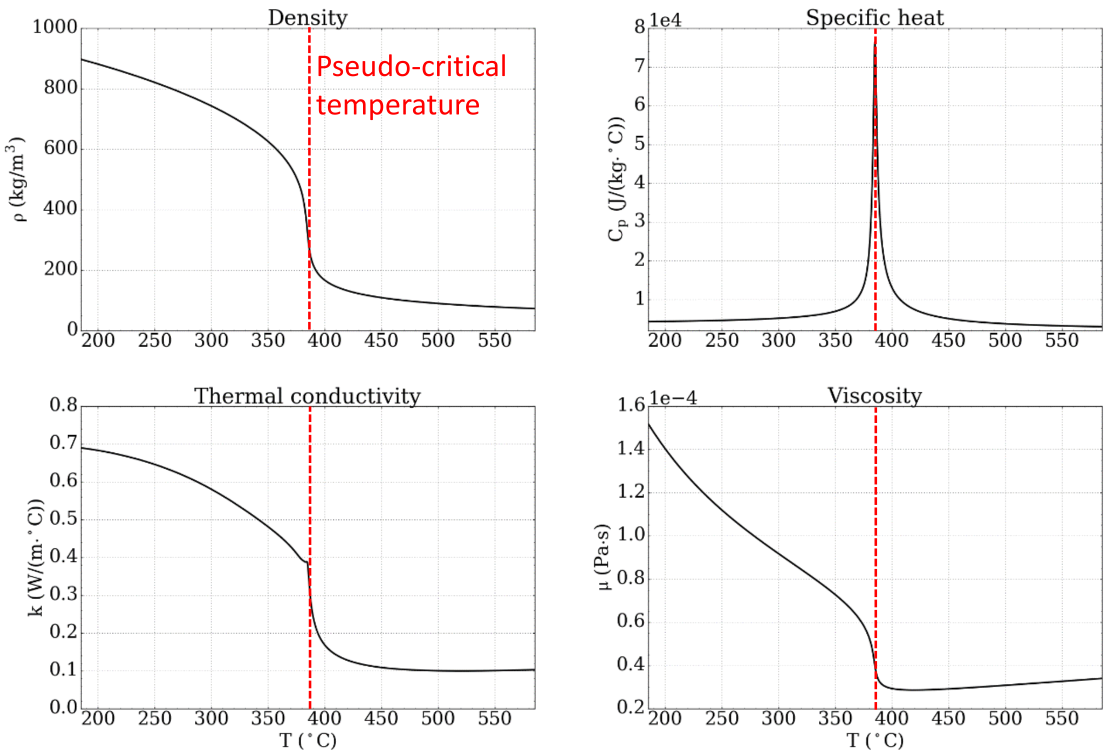
  Canadian Nuclear Laboratories (Canada), University of Ontario Institute of Technology (Canada), China Institute of Atomic Energy (China), Shanghai Jiao Tong University (China), Karlsruhe Institute of Technology (Germany), Budapest University of Technology & Economics (Hungary), Bhabha Atomic Research Centre (India), University of Pisa (Italy), JSC OKB Gidropress (Russia), National Technical University of Ukraine (Ukraine), **University of Sheffield (UK)**, University of Wisconsin - Madison (USA)

2.1 Overview

- **Goals of the benchmarking exercise**
  - Compare tools, models from different research groups/institutions
  - Establish the capability of CFD

- **Challenges in CFD modelling for SCWR**
  - High sensitivity to mesh, numerical scheme, turbulence model, near-wall treatment, model implementation, due to
    - Drastic properties change
    - Complex physics, including,
      - buoyancy
      - thermal expansion
      - flow acceleration
      - flow laminarisation
      - heat transfer deterioration

Physical properties of water at 25 MPa



2.2 Experimental rig

Flow loops

UW high-pressure heat transfer primary and secondary flow loops

3D model of the primary loop

1) High Pressure Pump
2) Orifice Flow Meter
3) Bypass Orifice
4) **Heated Test Section**
5) Heat Exchanger
6) Bypass Valve

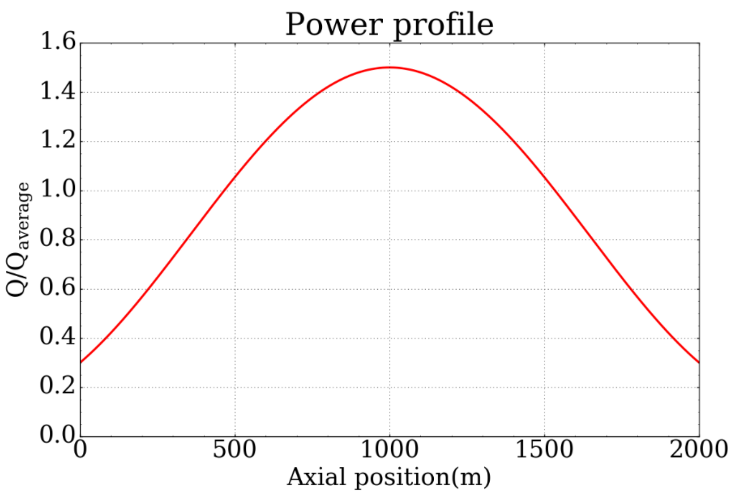
2.2 Experimental rig

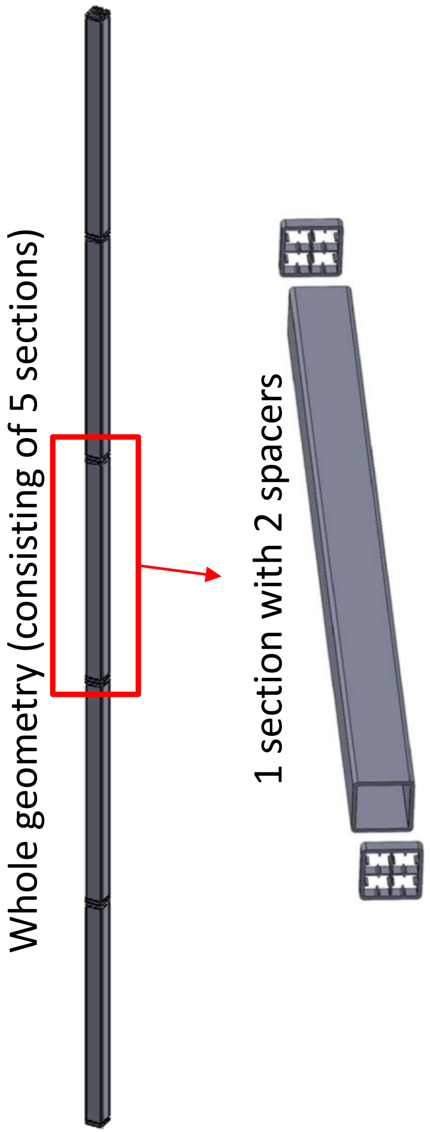
Test section

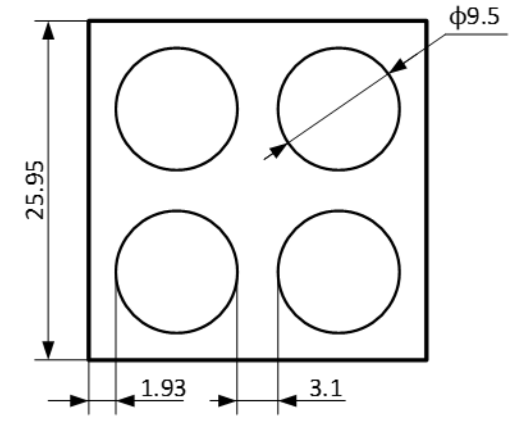
- 2x2 bundle with spacers
- Enclosed in a square housing
- Rod diameter: 9.5mm
- Pitch/Diameter: 1.326
- Non-uniform power profile

$$\dot{q}(z) = q_{av}\left\{\theta_0 + \theta_1 \cos\left[2\theta_2\left(\frac{z}{L} - 0.5\right)\right]\right\}$$

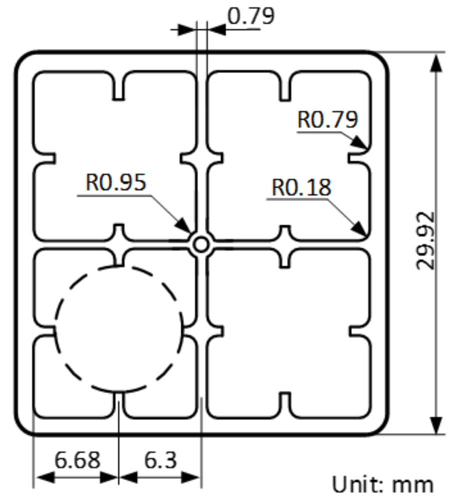
$\theta_0 = 0.8187458177$
$\theta_1 = 0.6812541823$
$\theta_2 = 2.436354311$







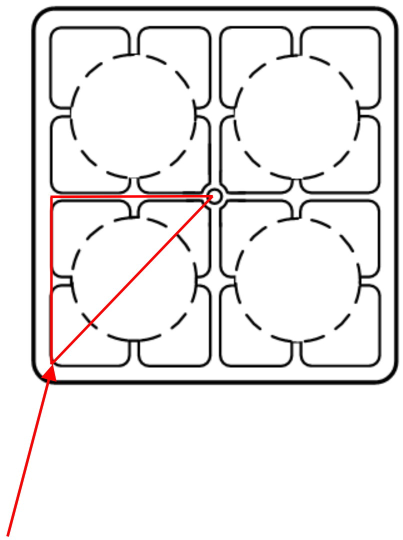
(a)



(b)

2.3 Model descriptions

Geometry



Studied domain: 1/8 representative section

Mesh

(a) Cross section view (without spacers)   (b) Cross section view (with spacers)

Total number of mesh cells: 10.3M

Model setups

- Transient
- NIST property database
- Gravity
- K-omega-SST
- 2-scale wall function
- Buoyancy production
- Solving enthalpy
- SIMPLEC
- Spatial 2nd order
- Temporal 1st order

Studied cases

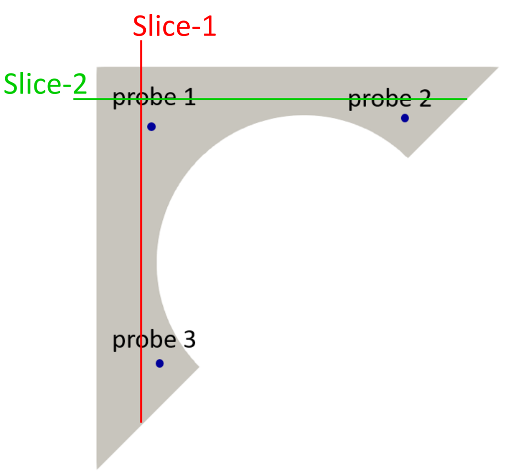
Group	Case	P_{in} (MPa)	T_{in} (°C)	G (kg/m^2·s)	U_0 (m/s)	Re_0	q_{av} (kW/rod)
I	A	8.26	121.8	2201	2.329	66925.4	10.07
	B	8.28	149.6	1447	1.571	54841.6	24.96
II	C	25.0	346.0	844	1.323	79329.1	47.8
	D	25.0	340.0	450	0.6869	40959.7	32.9

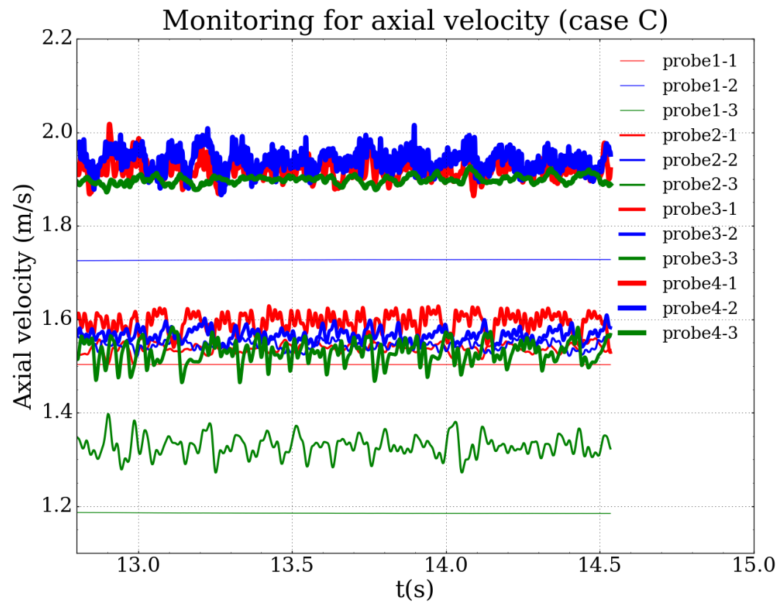
2.4 Results & discussion

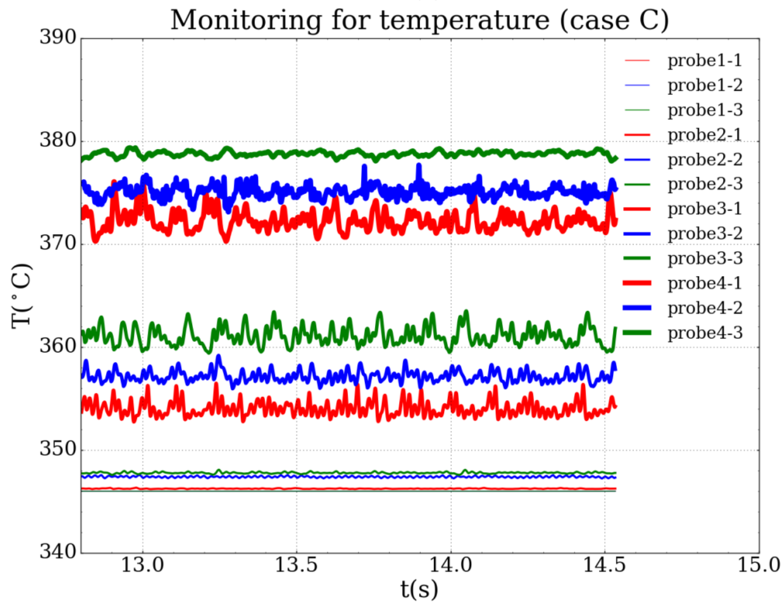
Transient flow

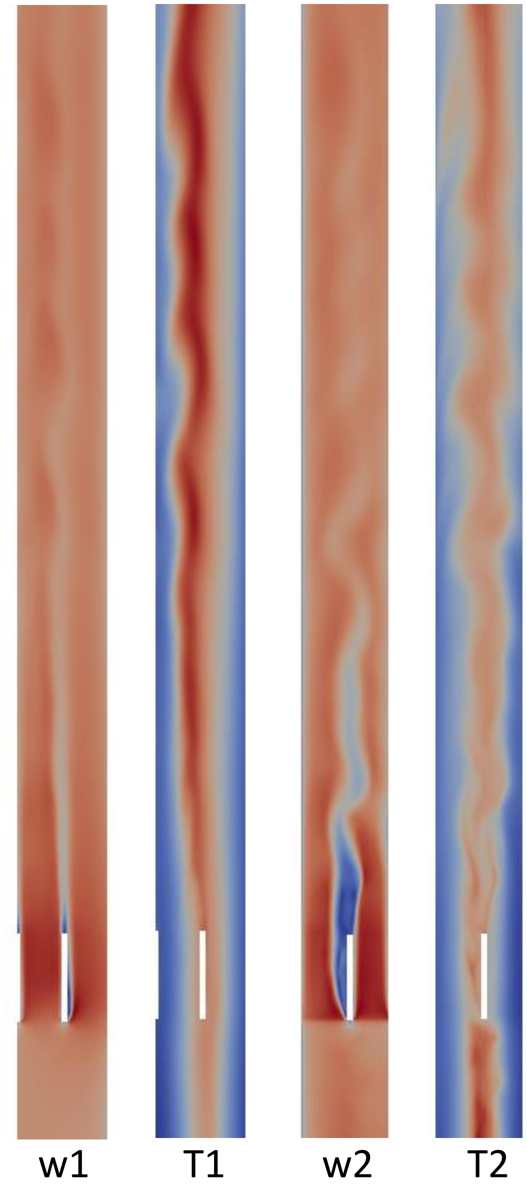
Probes locations

Probe set	Probe No.	Axial location (m)
1	1	-0.14
	2	
	3	
2	1	0.1
	2	
	3	
3	1	0.5
	2	
	3	
4	1	1.0
	2	
	3	



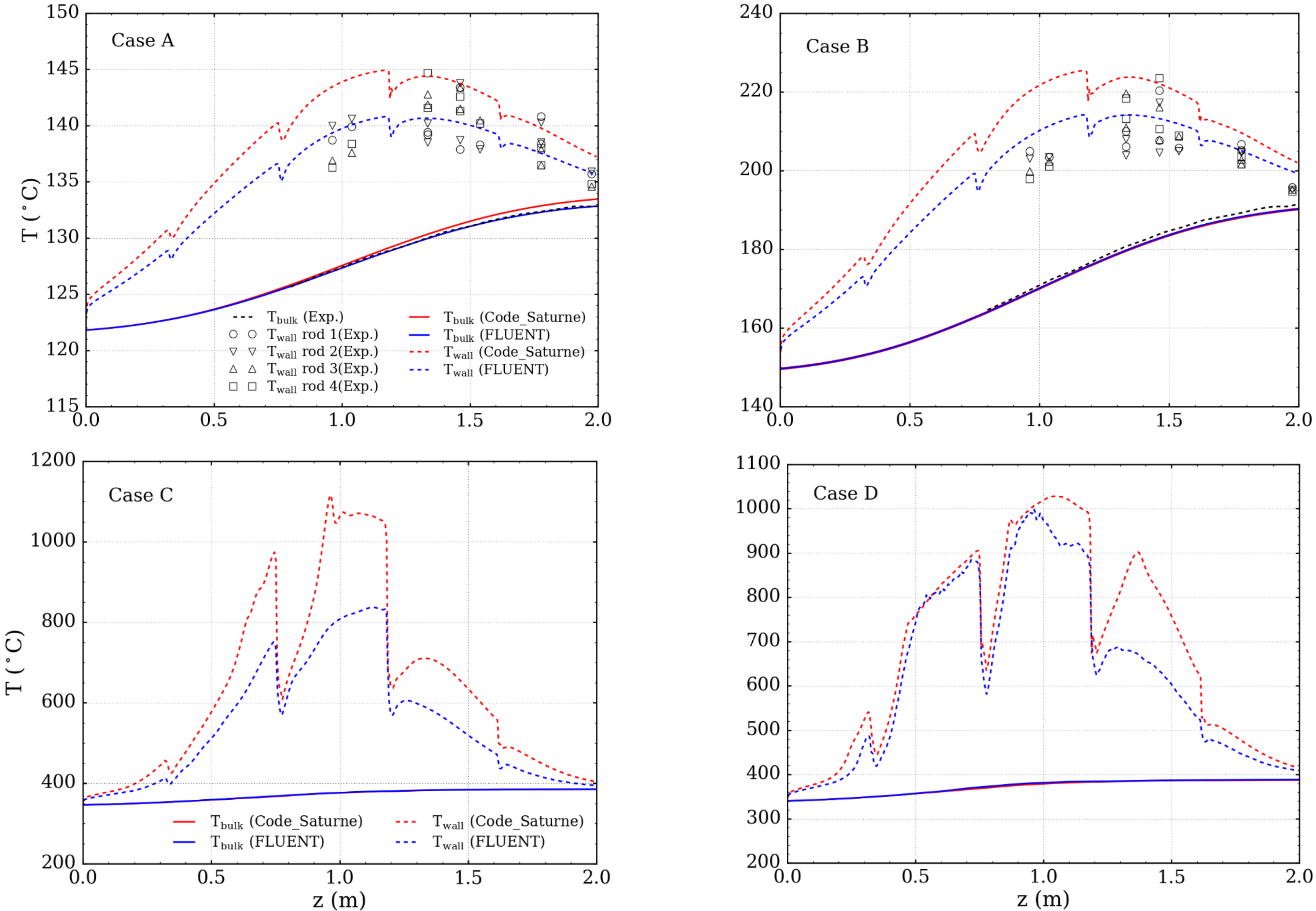






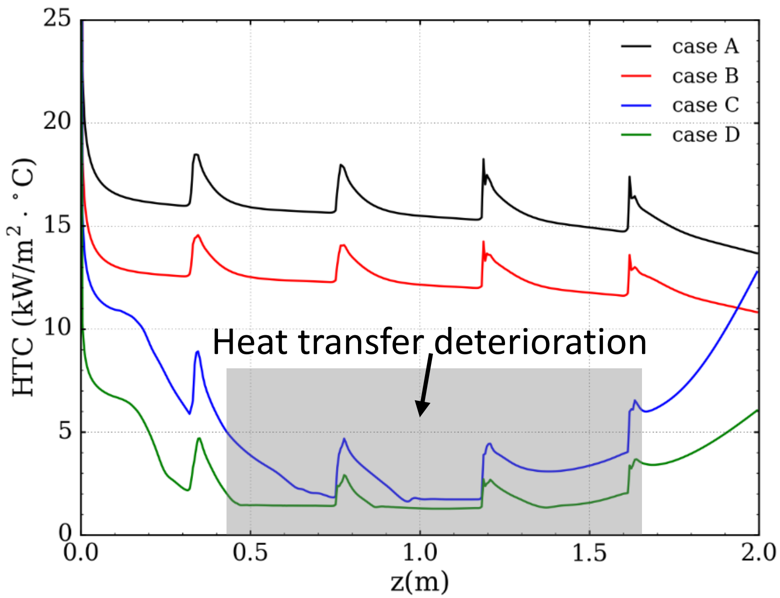
2.4 Results & discussion

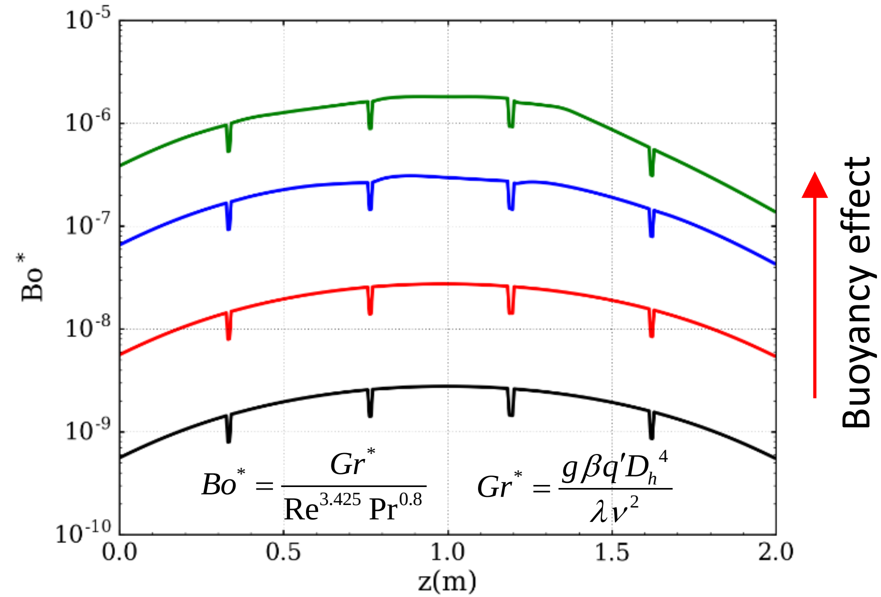
Wall temperature (Including comparisons against Fluent)



Other information

Code_Saturne results





- Heat transfer deterioration happens in both Case-C and Case-D
- The strongest buoyancy effect appears in Case-D, then Case C

2.5 Conclusions

- When using k-ω-SST model, Code_Saturne predicts higher wall temperatures than Fluent, which is more significant for the mixed convection Case-C;
- Both codes predicted the flow laminarisation and heat transfer deterioration in the mixed convection Case-C and Case-D, and Code_Saturne tends to be more responsive to the buoyancy effects;
- The non-uniform heating of the bundle is the reason to account for the recovery of the heat transfer coefficient in the mixed convection cases;
- Results **have been submitted** to the benchmarking organiser. A comprehensive assessment of different methods, models, and tools will be available.


QUESTIONS