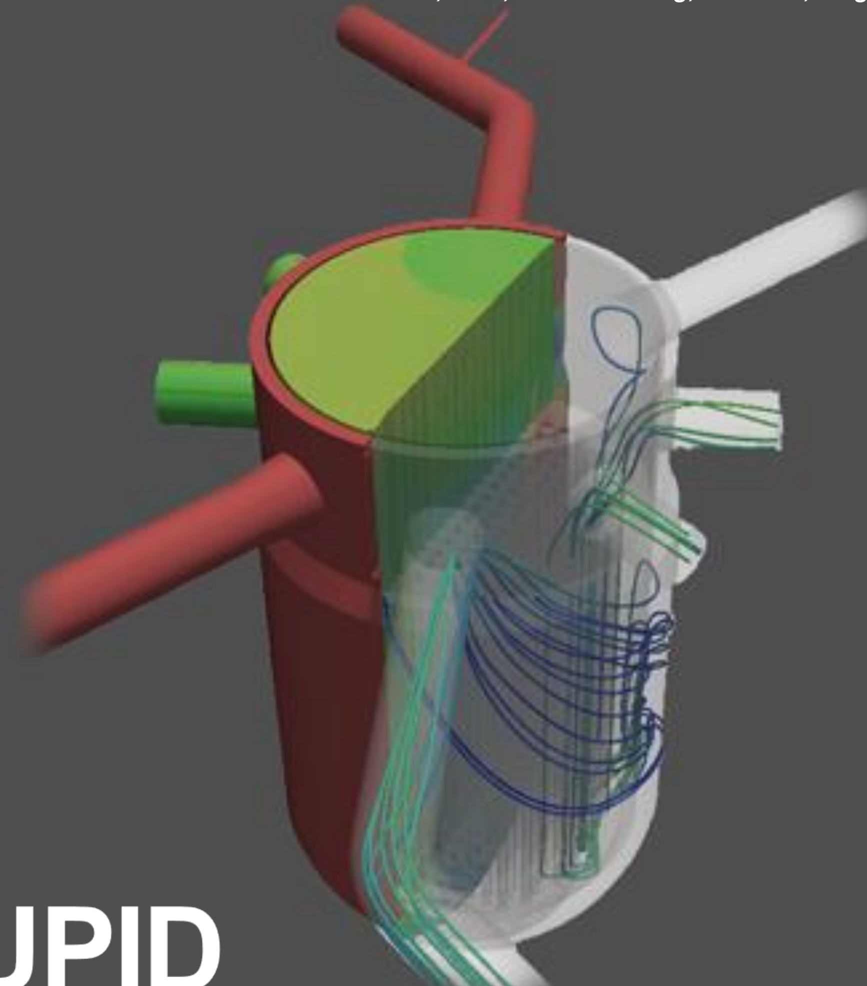
CONTENTS

19th International Topical Meeting on Nuclear Thermal Hydraulics
March 04, 2022, Virtual Meeting, Brussels, Belgium



CUPID Workshop

Introduction to CUPID

Han Young Yoon
March 04, 2022

CUPID Workshop

CONTENTS

Main Features of CUPID

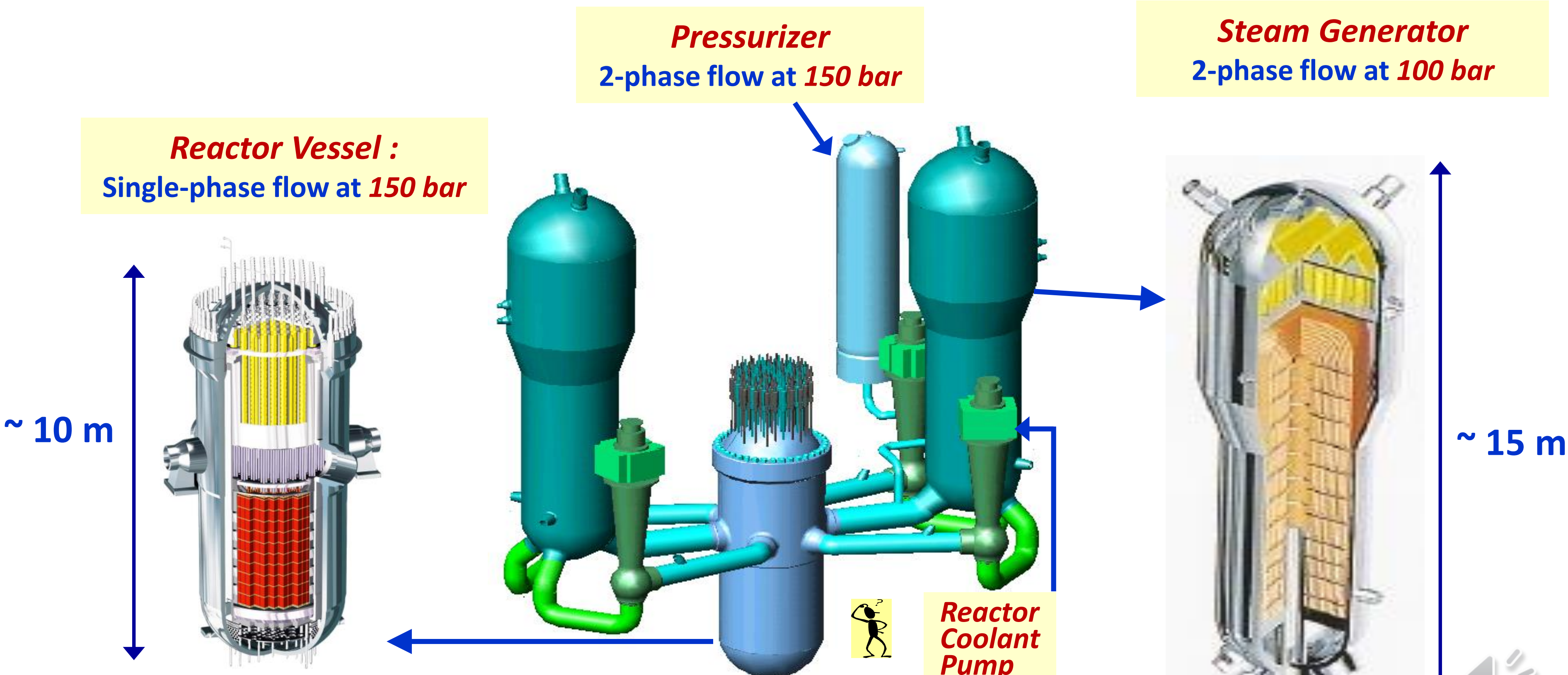
1

- Fast and Robust 3D 2-Phase Flow Solver
- Scalable Iterative Solver for a Large-scale Computing
- Multi-scale Simulation for a Fast Transient
- Multi-physics Simulation (TH/NK/FP)

TH: Thermal Hydraulics
NK: Neutron Kinetics
FP: Fuel Performance

Fast and Robust 3D 2-Phase Flow Solver (1/2)

- *Big and complicated* systems with transient *2-phase flows (1~150 bar)*
- The *CUPID* code has been developed for *steady-state and transient analyses* of *single- and two-phase flows* in nuclear reactor in *component-* or *CFD-scale*



OPR1000 MW (2775 MWt PWR)

http://www.doosanheavy.com/

Fast and Robust 3D 2-Phase Flow Solver (2/2)

- Commercial CFD mainly focuses on 3D fluid dynamics and ***has limited applications for 2-phase flows especially when a large phase change is involved***

Commercial CFD

- Numerical Models of ***3D Fluid Dynamics***
- High Performance Computing (***HPC***) Technology

CUPID

- Unique ***Numerical Model for a large phase change***
- CFD- or Component-scale ***Analysis of RV, SG, and CT***

Models and Correlations for Nuclear Reactors

- ***2-Phase Interface*** Momentum and Energy Transfer Models
- ***Multi-Scale/ Physics*** models

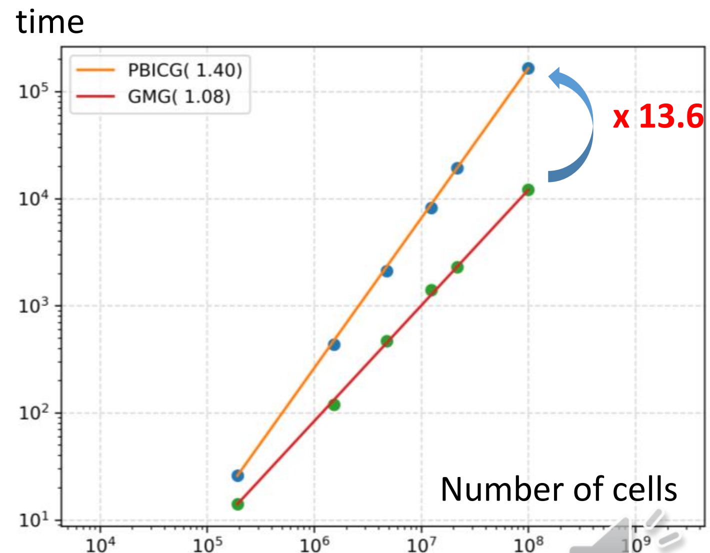
RV: *Reactor Vessel*, ***SG***: *Steam Generator*, ***CT***: *Containment*

Scalable Iterative Solver for a Large-scale Computing

- The most time-consuming part in CUPID is the ***"Pressure equation"*** solving module
  - The pressure equation takes more then ***90%*** of total computing time depending on the number of cells
  - The Conjugate Gradient ***(CG) solver is not scalable*** and we need to develop a ***new iterative solver*** which is ***scalable*** w.r.t the number of cells

Number of Cells	time_pressure / time_total (%)
191,800	**78.8**
1,533,600	**75.7**
4,773,600	**81.6**
12,357,600	**86.2**
21,683,700	**90.2**
107,968,000	**92.9**

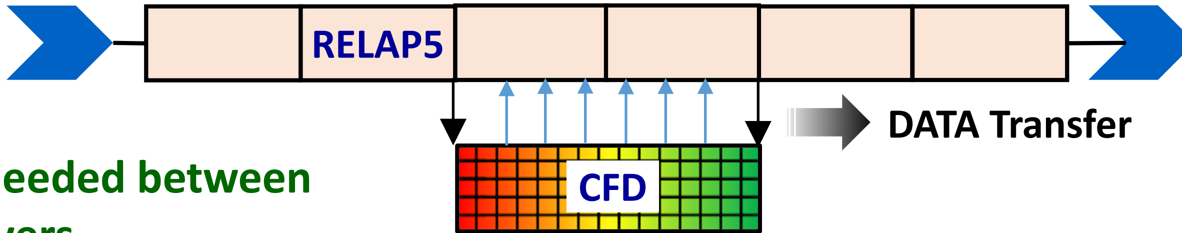
- Development of a ***Geometric Multi-Grid (GMG)*** solver for ***unstructured mesh***
  - CG solver: $\mathrm{Time}_{CG} \propto N^{1.4}$
  - GMG solver: $\mathrm{Time}_{GMG} \propto N^{1.0}$
  - The new GMG solver is ***Easy to use*** since the unstructured ***coarse meshes are generated automatically***



Multi-Scale Simulation for a Fast Transient

Domain Overlapping

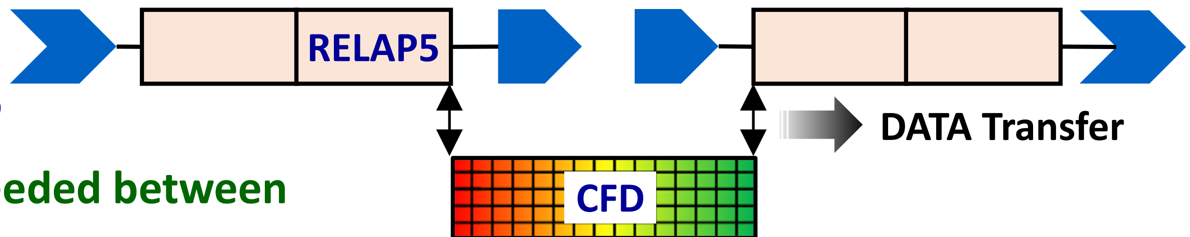
Data Transfer is needed between Two separate solvers



Ex) RELAP5/ CFX, FLEUNT, STAR-CCM+

Domain Decomposition

Data Transfer is needed between Two separate solvers

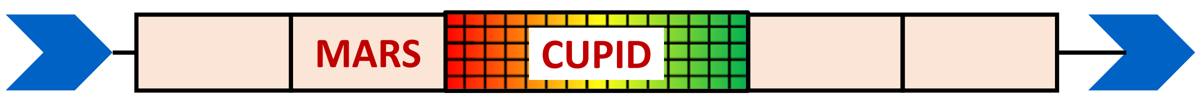


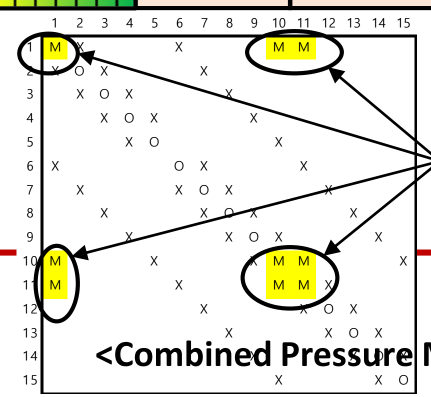
Single Domain

Single pressure solver matrix

: No need for the Data Transfer

⟶ *Versatile application to transient problems*





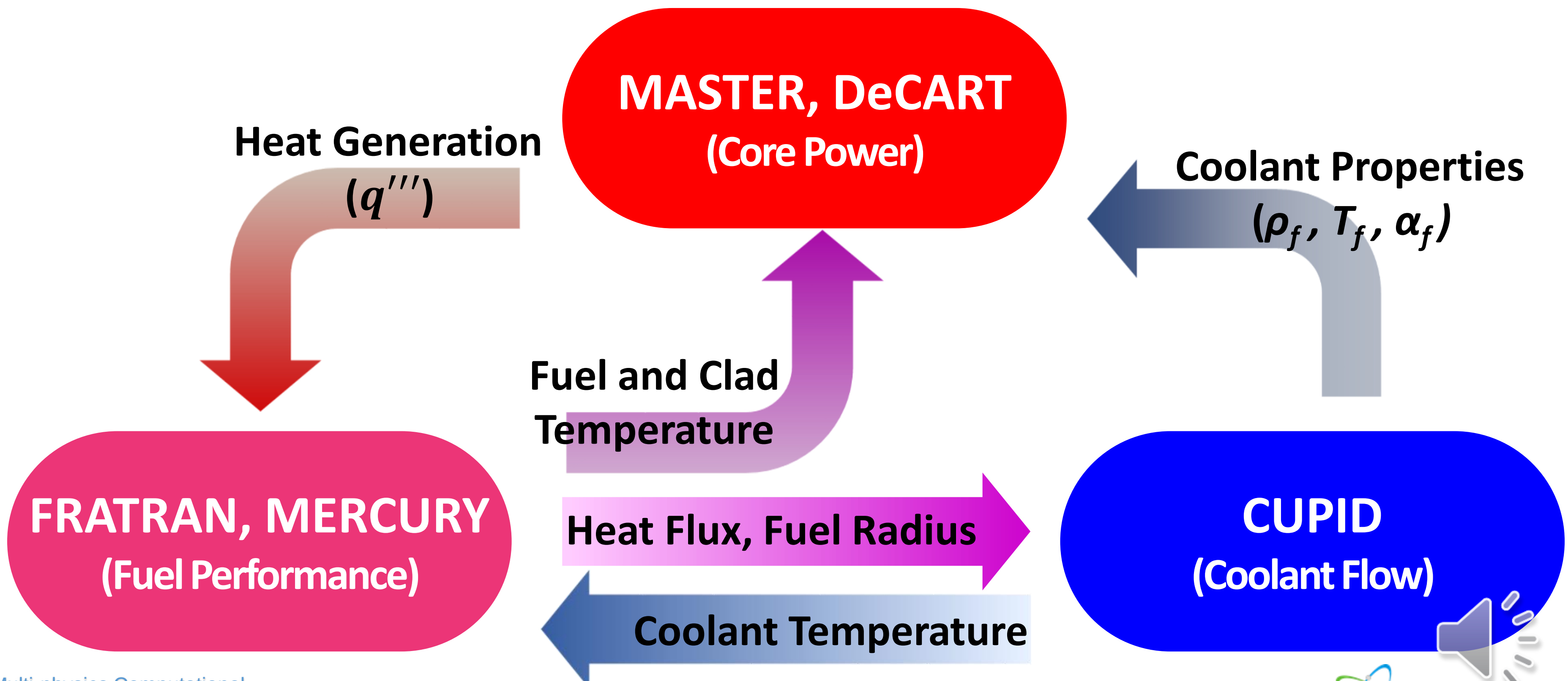
<Combined Pressure Matrix of CUPID>

** I.K.Park et al., Annals of Nuclear Energy, 2013.*

Multi-Physics Simulation (TH/NK/FP)

Coupling of Multi-Physics Codes

- **Neutron Kinetics Codes:** ***MASTER, DeCART***
- **Thermal Hydraulics Code:** ***CUPID***
- **Fuel Performance Codes:** ***FRAPTRAN, MERCURY***



Numerical Models

2

- Mathematical Models
- Parallel Computation
- Pre/Post Processors

Mathematical Models

Field Equations

- 3-Dimensional ***2-Fluid & 3-Field*** Mass, Mom., Eng. Equations
- Non-condensable Gas Equations (***He, H2, N2, Kr, Xe, Air, Ar, SF6***)
- 3-Dimensional ***Solid Conduction*** Equation
- ***Boron*** Transport Equation
- Interfacial Area Transport Equation

Numerical Methods

- Finite Volume Method (***FVM***)
- ***Unstructured Mesh***
- Semi-Implicit/***Fully-Implicit*** Scheme
- Pressure Solver: Bi-Conjugate Gradient (***BICG***), Multi-Grid (***MG***)
- ***Compressibility*** is considered

**** H.Y.Yoon et al., Numerical Heat Transfer, 2016.***

Physical Models

CFD-Scale

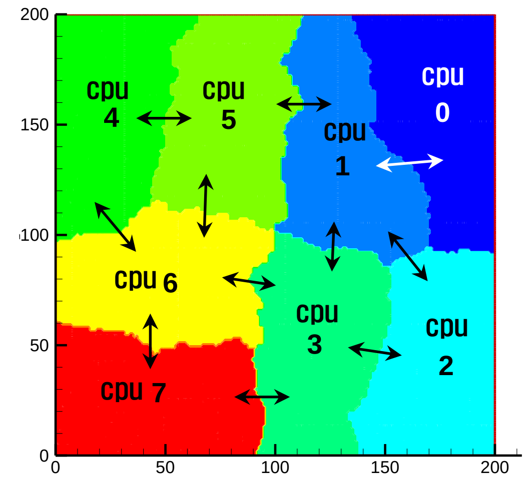
- Turbulence Model: ***k-e, SST, LES***
- ***2-Phase Topology*** Map,
- ***Drag, Lift Force, Interface Heat Transfer*** Models
- ***Radiation Heat Transfer*** Model

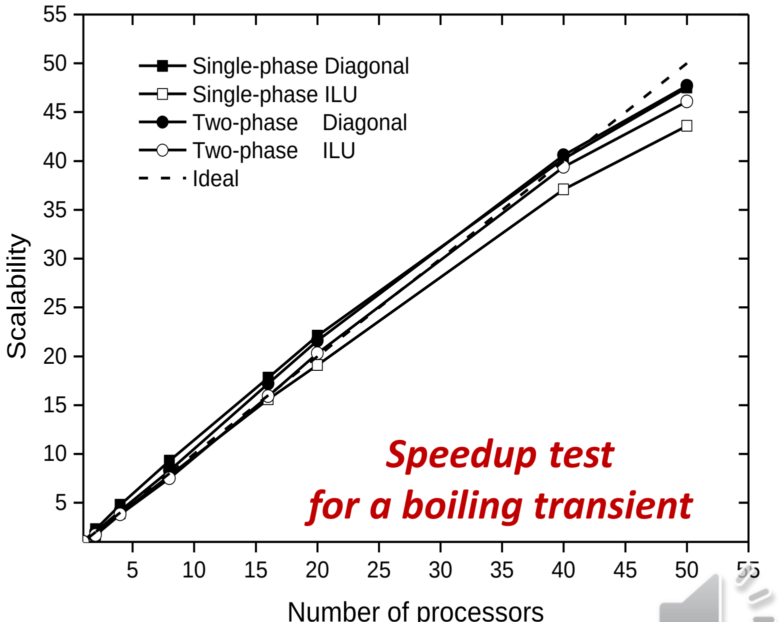
Component-Scale

- ***Porous Medium*** Model
- 2-Phase Flow Regime
- ***Models and Correlations*** Package for the safety Analysis
- ***Sub-channel Model***

Parallel Computation

- **Domain Decomposition**
  - Automatic domain decomposition using the ***METIS Library***
  - Manual decomposition
- ***MPI functions*** are used for the communication between different domains
- ***Highly Scalable*** parallel computing performance as the number of CPU increases





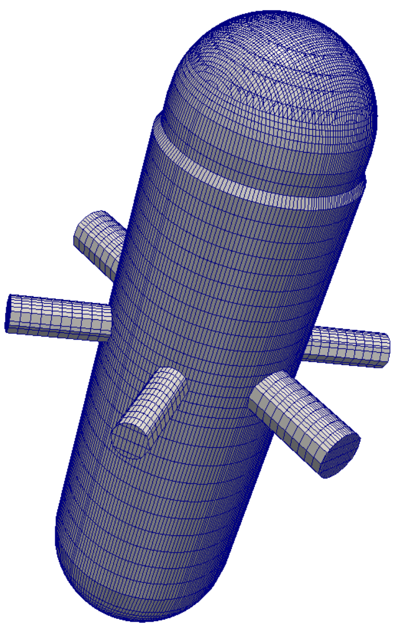
* *J.R.Lee et al., Journal of Mechanical Science and Technology, 2016.*

Pre/Post Processors

Mesh Generation

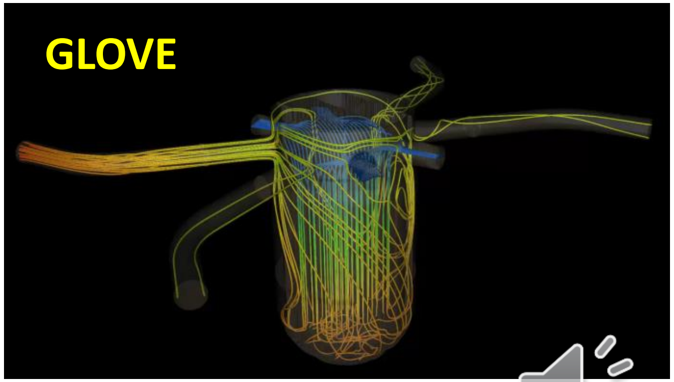
- **Mesh Input Data Structure: *OpenFOAM Format***
- **CFD-Scale Mesh Generation: *SALOME* (EDF)**
- ***Reactor Vessel Mesh* Generation at a Sub-channel Scale**
  - ***RVMesh-3D***

RVMesh-3D



Post Processing

- **Open Source Program: *Paraview***
- **Parallel Post Processing: *GLOVE* (KISTI)**



V&V and QA Programs

3

- V&V Program
- QA Program

Verifications & Validations

Extensive *Verification and Validation Consisting of 90 Test Problems*

CUPID Prototype (2010)

- Single-phase conceptual problems (8 Cases)

- Turbulence models (6 cases)
- Boron transport model (2 cases)

- 2-phase conceptual problems (14 Cases)

CUPID/MARS

CUPID/MASTER

- Multi-scale (CUPID/MARS) coupling (10 cases)

- Multi-physics (CUPID/MASTER) coupling (2 cases)

- Multi-species non-condensable gas models (4 cases)

CUPID-RV
CUPID-SG

CUPID-CT

CUPID/MARS/
MASTER/
FRAPTRAN

- SG models (2 cases)
- SET for LOCA analysis (13 cases)

- SET for containment analysis (7 cases)
- Wall boiling model (7 cases)

- Sub-channel models (13 cases)
- Multi-physics (CUPID/Fuel) coupling (2 cases)

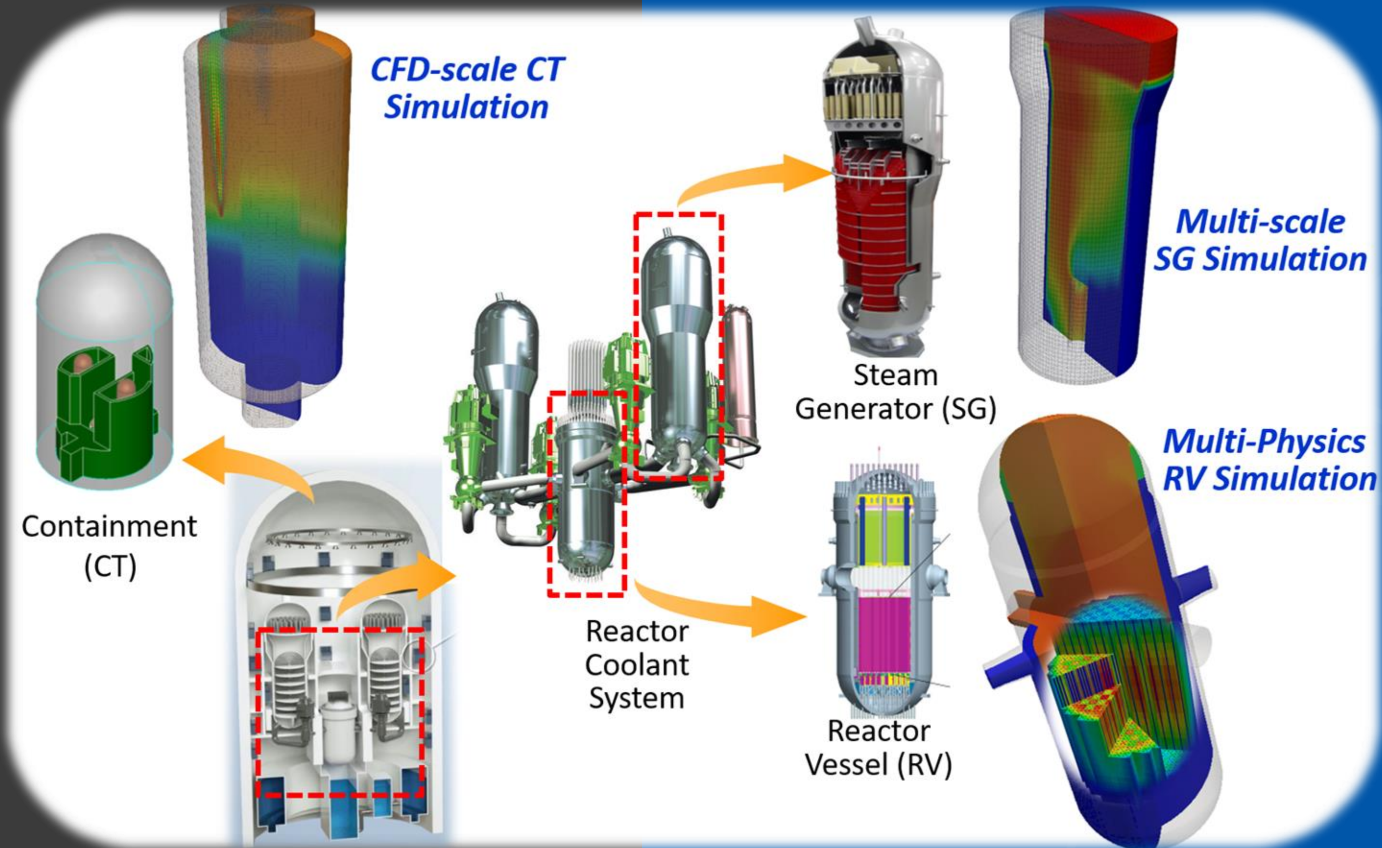
CUPID 2.5 (2021)

Quality Assurance

- **ASME NQA-1 (KEPIC-QAP)**
- **Documentation for QA**
- **CUPID Code Version Control System**
  - ***SVN (Subversion)*** **server/client type system**
    - **Centralized Version Control System (CVCS)**
  - **Store CUPID code and related documents to SVN server (Repository)**
  - ***Download*** **CUPID from SVN server (Checkout, Updated)**
  - ***Upload*** **newly developed coding to SVN server (Commit)**
    - ***V&V Calculation*** **→ V&V brief per** ***3 month*****, SVVR** ***per 1 year***
  - **All records are stored, traced back freely (*Traceback*)**

Major Applications

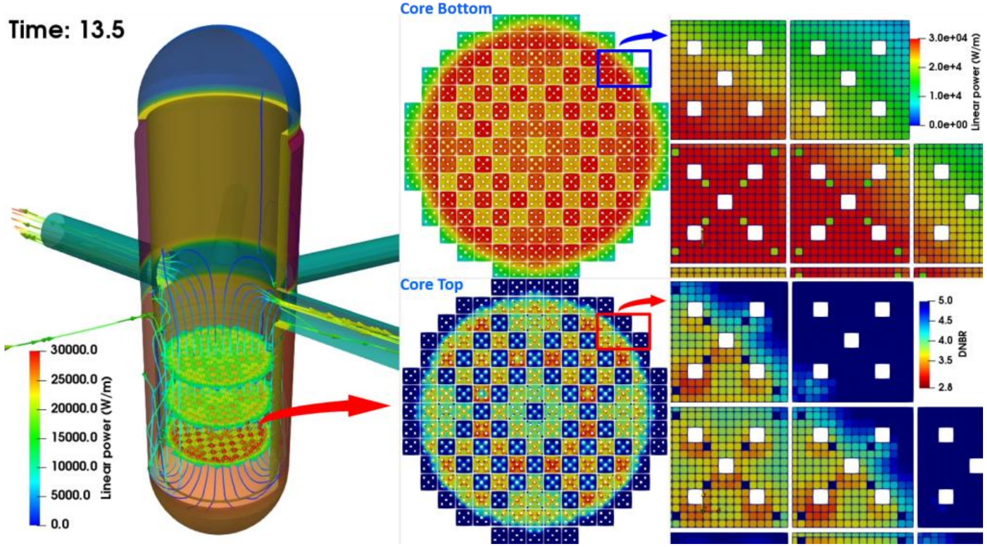
- Reactor Vessel
- Steam Generator
- Containment
- Special Components (SIT, PAFS)



4

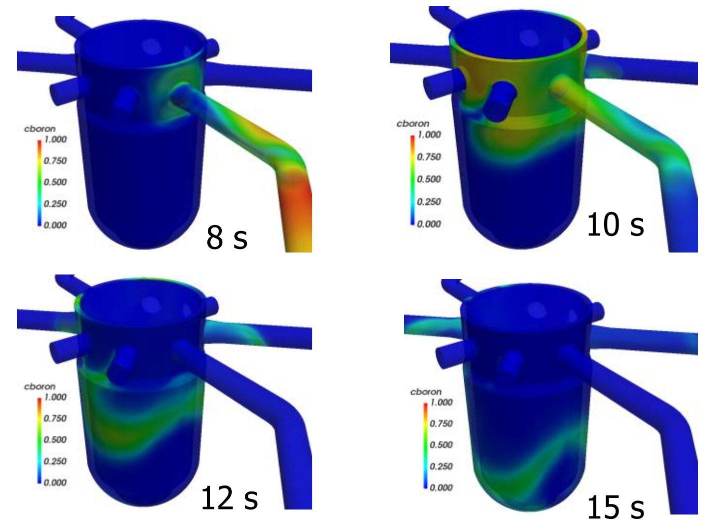
Reactor Vessel

» *Full Core Pin-wise Simulation* of the Steam Line Break Accident of OPR1000



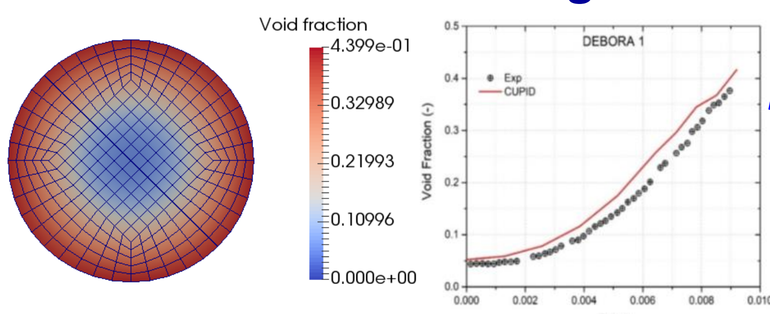
** H.Y.Yoon et al., Nuclear Science and Engineering, 2020.*

» Simulation of the *ROCOM (HZDR) flow mixing experiment* (IAEA/CRP)



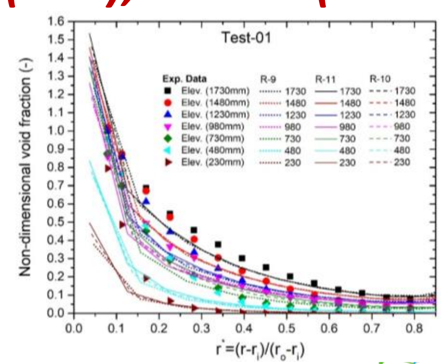
** Y.J.Cho et al., Nuclear Engineering and Design, 2019.*

» Validation of wall boiling models against *DeBORA (CEA), F-SUBO(KAERI)*



<DeBORA Simulation>

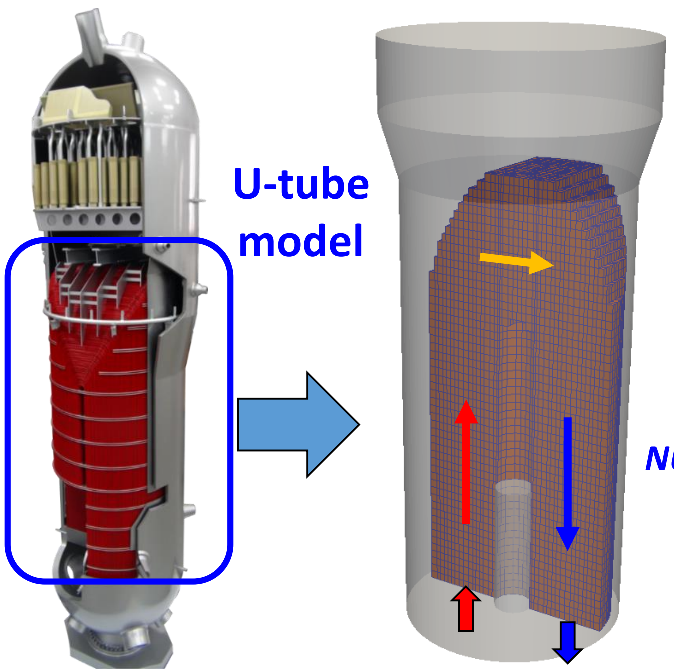
** Y. Alatrash et al., Nuclear Engineering and Technology, 2021.*



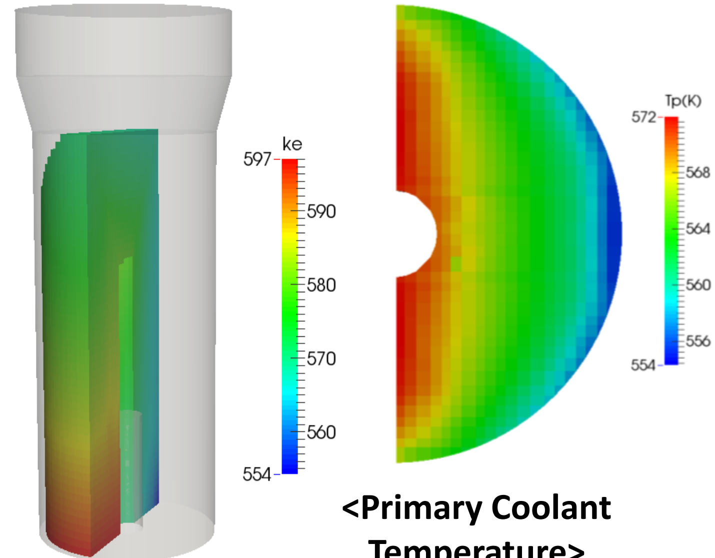
<F-SUBO Simulation>

Steam Generator

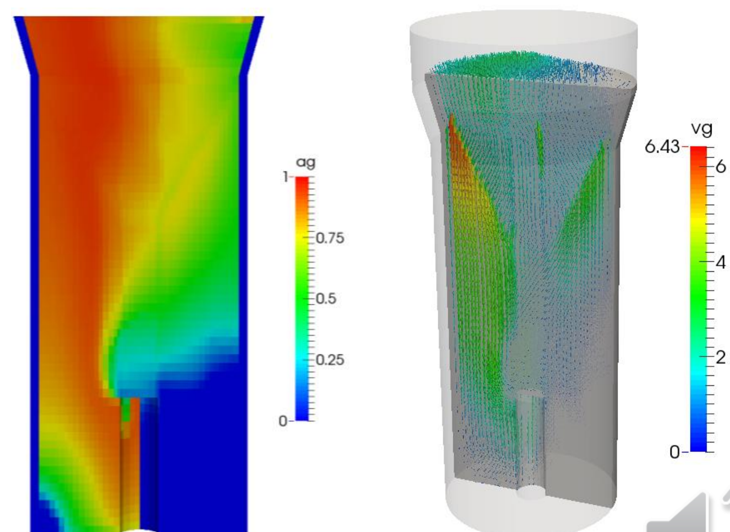
- PWR SG analysis code (*CUPID-SG*) has been developed based on the CUPID code
- All regions for *riser, downcomer, separator, and steam dome* are modeled
- A *U-tube model has been developed* where all U-tubes are grouped and connected with the secondary fluid cells



** H.Y.Yoon et al., NURETH-17, 2017.*



<Primary Coolant Temperature>



<Secondary Side Void Fraction and Velocity Vectors>

Containment

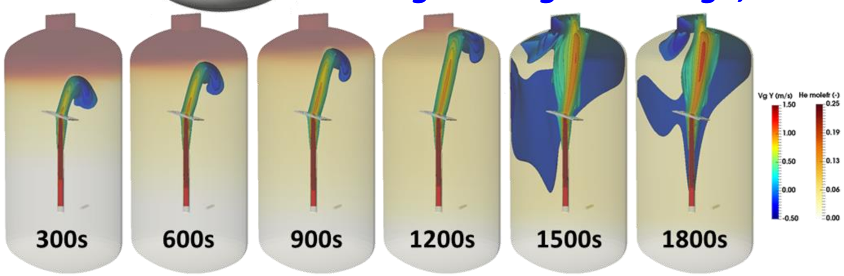
Containment Analysis Models

- ***Non-condensable gas***, ***Condensation*** models
- ***Radiation*** model

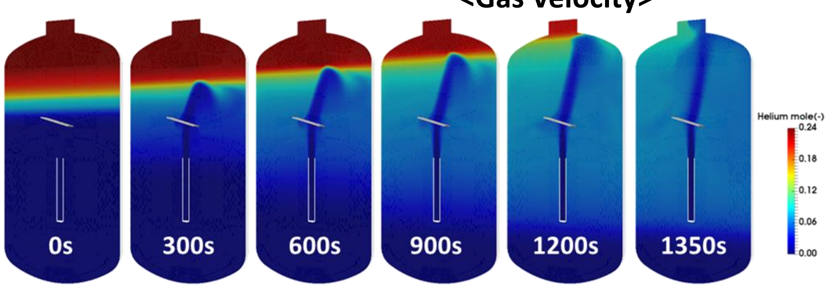
OECD/NEA *HYMERES-2*

- Helium layer erosion test using *PANDA* facility of PSI
- Vertical steam jet with obstructions (single/double plate, grid)

** J.H.Sohn et al., Nuclear Engineering and Design, 2021.*



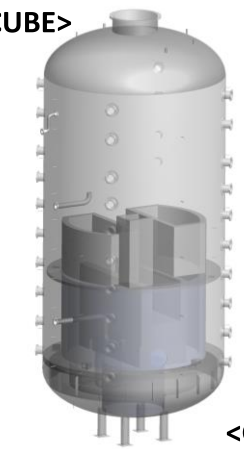
<Gas Velocity>

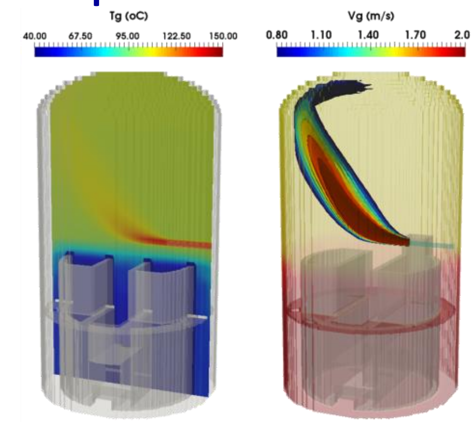


<Helium Moral Fraction>

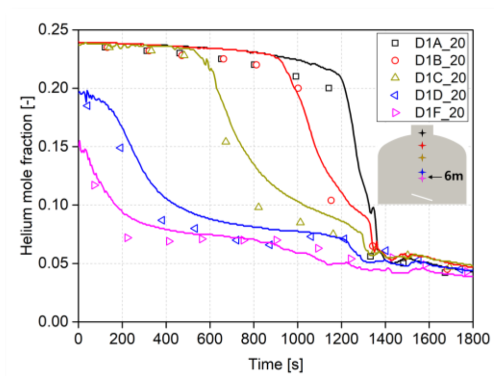
ATLAS-CUBE (KAERI)

- Steam injection test using CUBE facility connected with ALTAS
- Thermal stratification / Effect of structures as a passive heat sink

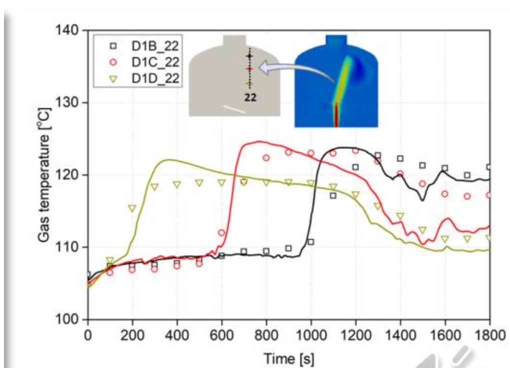




<Gas Temperature> <Gas Velocity>

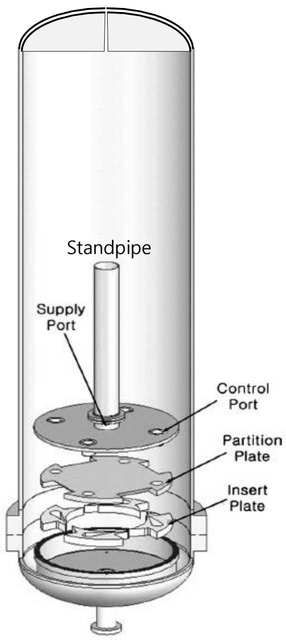


<Helium Moral Fraction Transient>

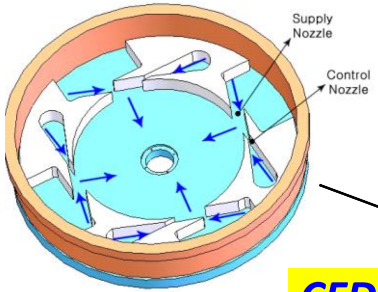


<Gas Temperature Transient>

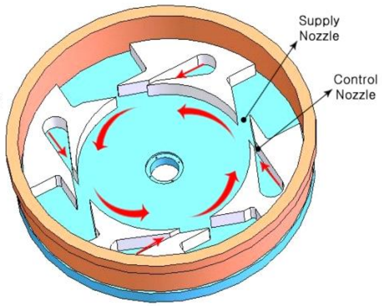
Special Component – Advanced SIT (APR1400)



(a) The internal structure

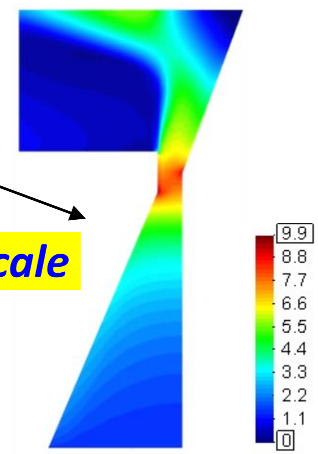


(b) High flow mode

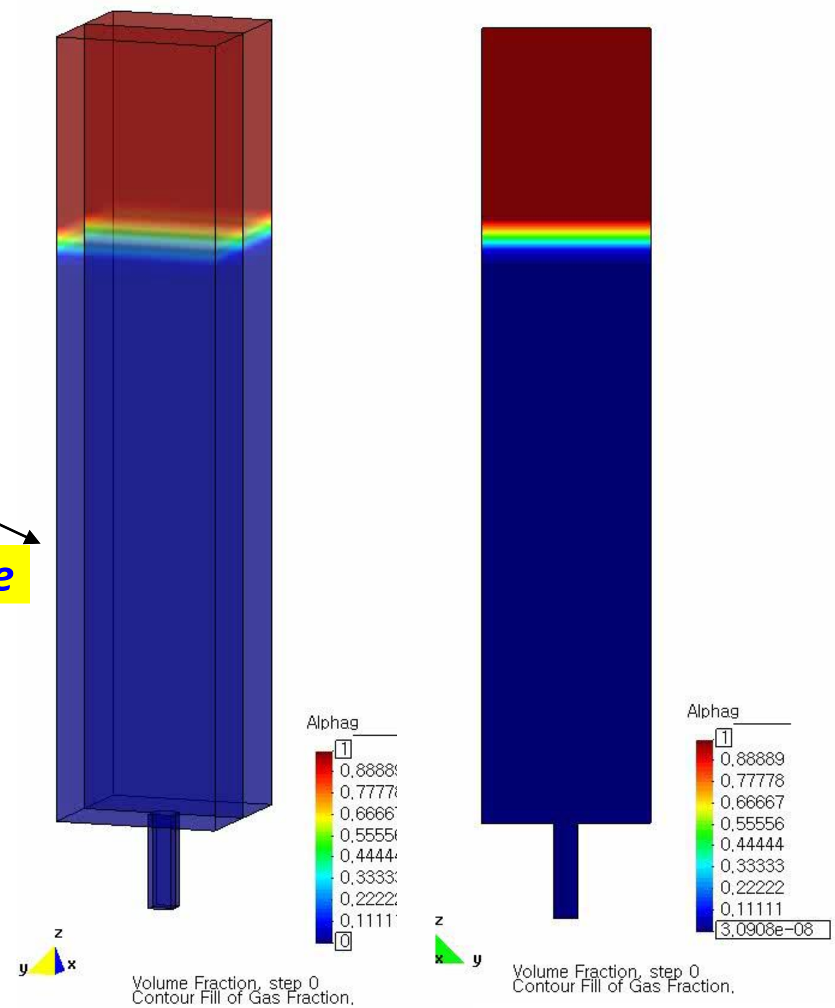


(c) Low flow mode

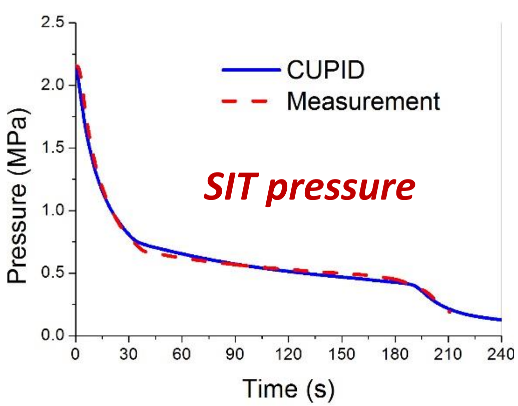
CFD-scale

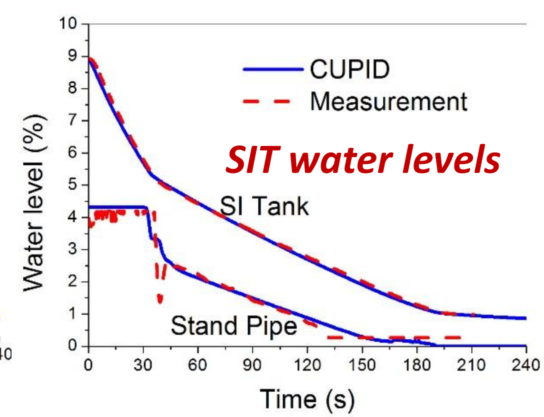


Component-scale



Comparison to the VAPER experiment (KAERI)





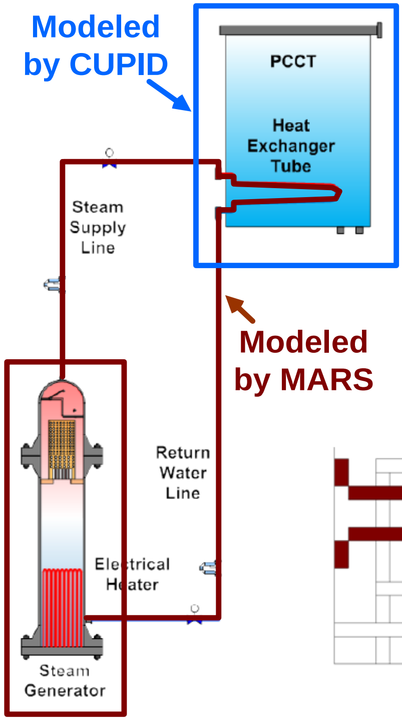
* H.Y.Yoon et al., Annals of Nuclear Energy, 2013.

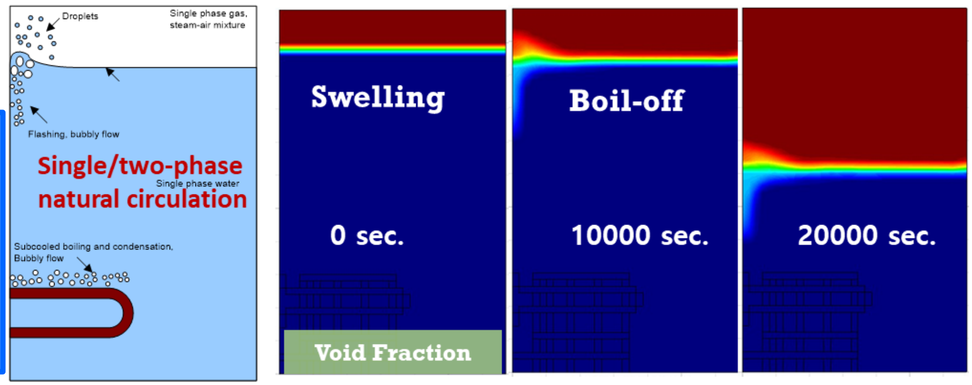
KAERI 한국원자력연구원 Korea Atomic Energy Research Institute

Special Component – PAFS (APR+)

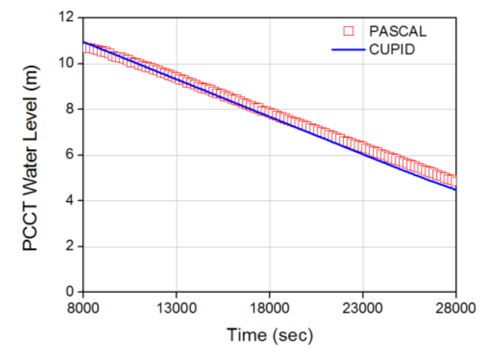
- ***PAFS* (Passive Auxiliary Feedwater System)**
  - **Removes residual heat by a *natural circulation loop***
  - **Validation Experiment: *PASCAL (KAERI)***
- **Simultaneous application of component and system scales (*CUPID/MARS*)**

** H.K.Cho et al., Nuclear Engineering and Design, 2014.*

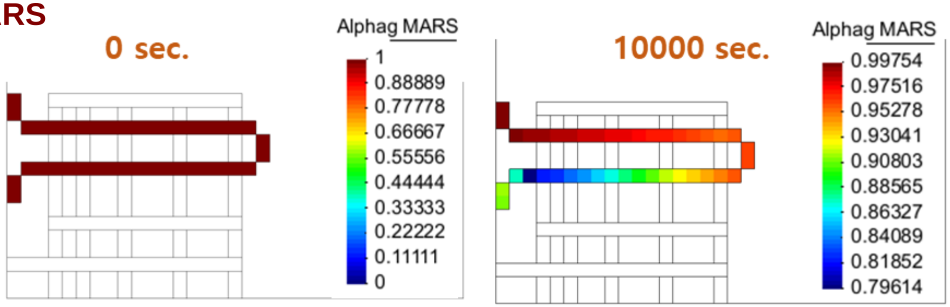




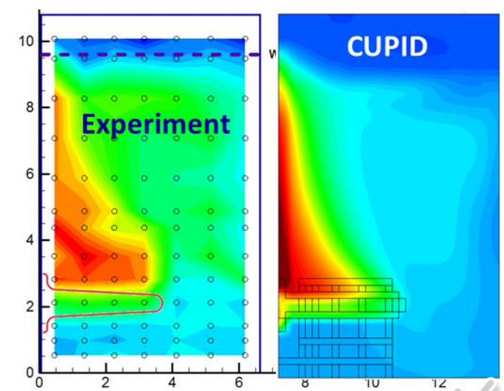
<PCCT Boil-off Transient by CUPID>



<Prediction of Water Level>



<HTX Tube Inside Condensation by MARS>



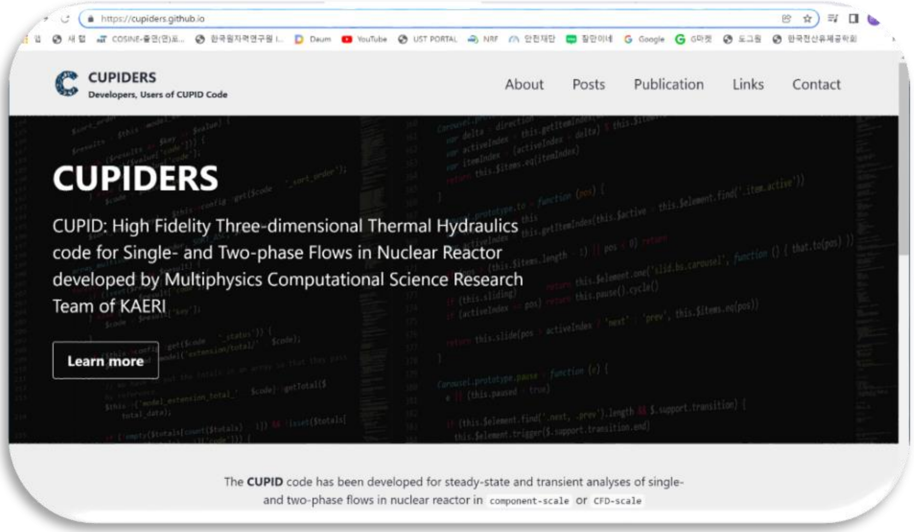
<Prediction of Water Temperature>

CUPID User Group

5

CUPID User Group

http://cupiders.github.io



Domestic Univ. & Research Inst.

Industries

Foreign Univ.

User Agreement

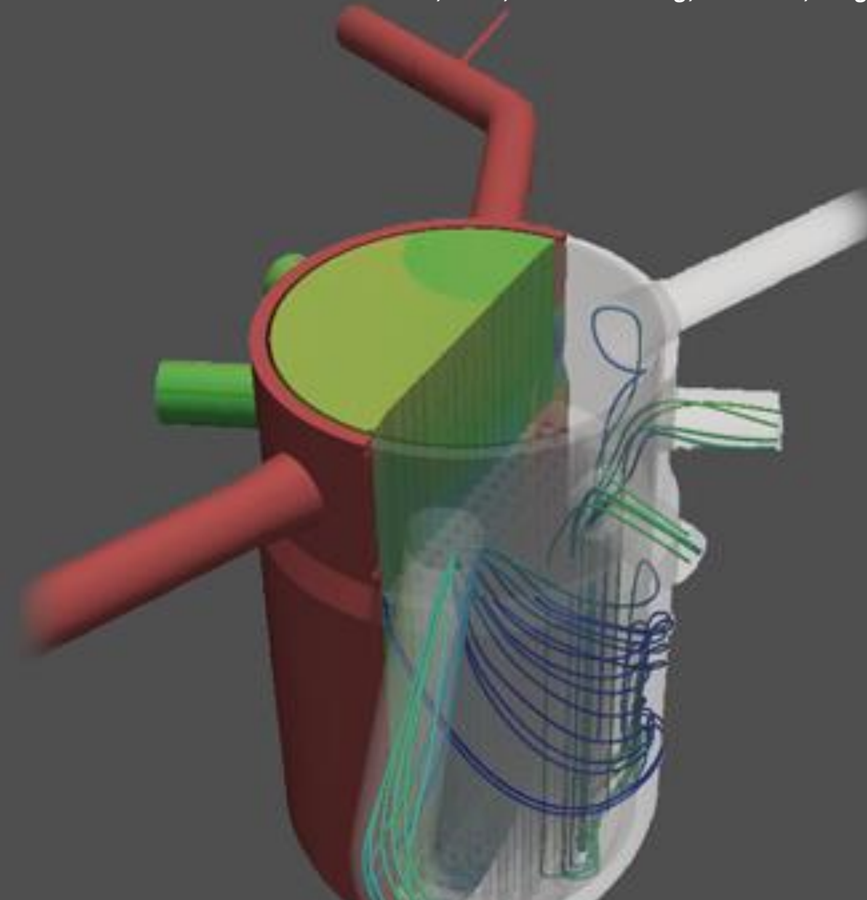
User Contract

User Agreement

THANK YOU

hyyoon@kaeri.re.kr

19th International Topical Meeting on Nuclear Thermal Hydraulics
March 04, 2022, Virtual Meeting, Brussels, Belgium



CUPID Workshop

Fast and Accurate 2-phase Flow Solution Scheme

Han Young Yoon

March 04, 2022

CUPID Workshop

CONTENTS

Development Strategy

1

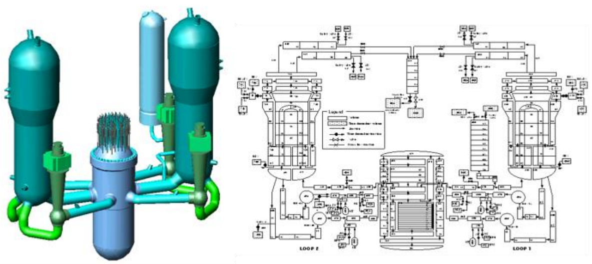
- Requirement for the Development
- Review of the Previous Methods

Requirement for the Development

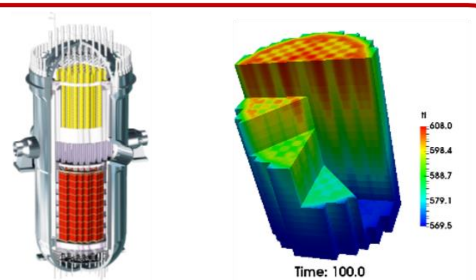
Analysis Scale

- **CFD-Porous (Sub-channel), CFD-RANS, CFD-LES**

System-scale



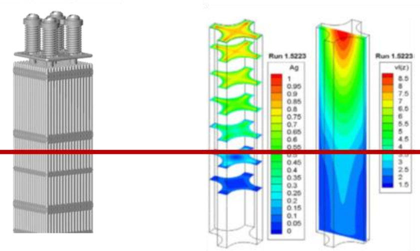
CFD-Porous (subchannel-scale)



CFD-RANS

CFD-LES

CFD-DNS



Major Applications

- **Nuclear Reactor Safety and Performance Analysis**
  - ***2-phase 3D TH model*** to deal with large phase change (150bar ~ 1bar)
  - ***High-resolution 1-phase 3D flow model*** with relevant turbulence models
- **3D TH Models for**
  - ***Reactor Vessel***
  - ***Steam Generator***
  - ***Containment***

Review of the Previous Methods

Analysis Code	Solution Methods	Limitations for Nuclear Reactor 3D Analysis	Adopted Features for CUPID
System Analysis Code: **RELAP5(NRC), MARS(KAERI), SPACE(KHNP)**	• FDM for staggered mesh • Semi-implicit Scheme • ***Pressure equation with phase change terms*** • ***2-phase models and correlations***	• Staggered mesh is ***hard to apply for a 3D geometry*** • Semi-implicit scheme is ***not efficient for a large calculation***	• Pressure equation with phase change terms • 2-phase models and correlations
CFD Code: **FLUENT, STAR-CCM+, OpenFOAM**	• ***FVM for unstructured mesh*** • ***Implicit Scheme*** • Incompressible pressure equation • ***Turbulence Models***	• ***Limited application to 2-phase flows*** with large phase change	• FVM for unstructured mesh • Implicit Scheme • Turbulence Models

Discretization Scheme

2

– Finite Volume Method with Unstructured Mesh

Finite Volume Method with Unstructured Mesh

- Useful for ***Complicated Geometries***
- Used in most of ***current CFD codes***
- Limitations
  - ***Good mesh quality*** is important for an accurate calculation
  - Difficult to apply a ***higher-order scheme***

Gradient

$$\int_V \nabla \phi dV = \int_S \phi d\vec{S} \approx \sum_f \phi_f \vec{S}_f$$

Diffusion

$$\int_V \nabla^2 \phi dV = \int_S \nabla \phi \cdot d\vec{S} \approx \sum_f \nabla \phi_f \cdot \vec{S}_f$$

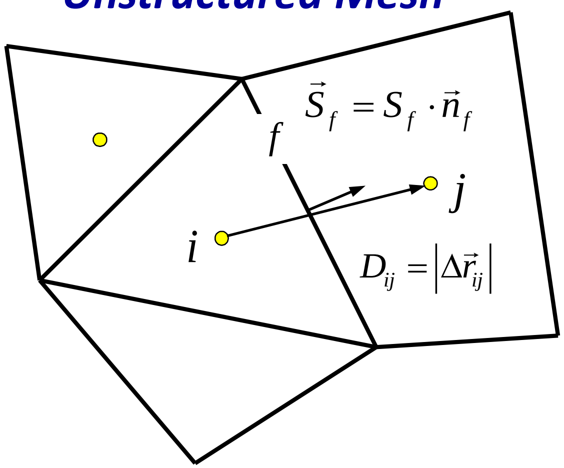
Convection

$$\int_V \nabla \cdot (\phi \vec{u})\, dV = \int_S \phi \vec{u} \cdot d\vec{S} \approx \sum_f \phi_{up} \Psi_f$$

where $\Psi_f = \{\xi \cdot \vec{u}_i + (1-\xi) \cdot \vec{u}_j\} \cdot \vec{S}_f$ *(Volume flux)*

$$\phi_{up} = \begin{cases} \phi_i, & \Psi_f \geq 0 \\ \phi_j, & \Psi_f < 0 \end{cases}$$

Unstructured Mesh



Solution Schemes

2

CUPID provides 3 different solution schemes

- Energy Coupled Method (Semi-implicit)
- Energy Decoupled Method (Semi-implicit)
- Energy Decoupled Method (Fully-implicit)

Energy Coupled Method (Semi-implicit)

- ***Mass and Energy Equations*** **are used for establishing the *pressure equation***
- **Used in the *system analysis code* such as RELAP5**
- ***Solution Procedure***

1. Momentum Eq.

$$\vec{u}^*_{k,i} = \vec{\gamma}_{k,i} - \beta_{k,i}\nabla P^n_i \qquad \vec{u}^{n+1}_{k,i} = \vec{u}^*_{k,i} - \beta_{k,i}\nabla \delta P_i$$

$$\sum_f \Psi^{n+1}_{k,f} = \sum_f \left[\Psi^*_{k,f} - \beta_{k,f} \frac{S_f}{\left|d\vec{r}_f\right|} \left(\delta P_j - \delta P_i \right) \right]$$

2. Linearization of Scalar

$$\delta \mathbf{x}_i = \mathbf{A}^{-1}_i \mathbf{s}_i + \sum_f \mathbf{A}^{-1}_i \sum_k \mathbf{c}_{k,f} \Psi^{n+1}_{k,f}$$

$$\delta \mathbf{x}_i = \left(\delta e_g, \delta e_l, \delta\alpha_g, \delta\alpha_l, \delta\alpha_d, \delta P \right)_i$$

3. Pressure Correction

$$\left(1 + \sum_f C_f \right) \delta P_i + \sum_f C_f \delta P_j = B_i$$

$$P^{n+1}_i = P^n_i + \delta P_i$$

$$\vec{u}^{n+1}_{k,i} = \vec{u}^*_{k,i} + \beta_{f,i}\nabla \delta P_i$$

4. Scalar Update

$$\mathbf{x}^{n+1}_i = \mathbf{x}^n_i + \delta \mathbf{x}_i$$

Energy Decoupled Method (Semi-implicit)

- ***Mass eq.*** is used for establishing the ***pressure eq.***
- Similar to that used in ***the CFD code*** but different in the ***implicit calculation of the phase change term***
- ***Solution Procedure***

1. Momentum Eq.

$$\sum_f \Psi_{k,f}^{n+1} = \sum_f \left[\Psi_{k,f}^{*} - \beta_{k,f} \frac{S_f}{\left| d\vec{r}_f \right|} \left(\delta P_j - \delta P_i \right) \right]$$

2. Combined Mass Eq.

$$\sum_k \left[\frac{\nabla \cdot \left(\alpha_k \rho_k \vec{u}_k \right)}{\rho_k} \right] = \Gamma_v \left(\frac{1}{\rho_g} - \frac{1}{\rho_l} \right) - \sum_k \left(\frac{\alpha_k}{\rho_k} \frac{\partial \rho_k}{\partial t} \right)$$

$$\frac{1}{V_i} \sum_k \frac{\left(\alpha_k \rho_k \right)_f}{\rho_{k,i}} \Psi_{k,f}^{n+1} = \Gamma_{v,i}^{n+1} \left(\frac{1}{\rho_{g,i}} - \frac{1}{\rho_{l,i}} \right)$$

$$- \sum_k \left[\frac{\alpha_k}{\rho_k \delta t} \left(\frac{\partial \rho_k}{\partial P} \delta P + \frac{\partial \rho_k}{\partial e_k} \delta e_k + \frac{\partial \rho_k}{\partial X_n} \delta X_n \right) \right]$$

3. Pressure Correction

4. Linearization & Update Scalar

Phase change term should be treated implicitly

Values at previous the time step

Energy Decoupled Method (Fully-implicit) (1/2)

- ***Convection and Diffusion*** **terms are calculated** ***implicitly*** **in the** ***"Fully-implicit Scheme"***

	Semi-implicit	Fully-implicit
Energy Coupled	O	X
Energy Decoupled	O	O

Momentum Equation

Implicit terms

Semi-implicit	Fully-implicit
$\alpha_g\rho_g \frac{\vec{u}_g^* - \vec{u}_g^n}{\delta t} + \nabla\cdot\left(\alpha_g\rho_g\vec{u}_g\vec{u}_g\right)^n - \vec{u}_g\nabla\cdot\left(\alpha_g\rho_g\vec{u}_g\right)^n = -\alpha_g\nabla P_i^n + \nabla\cdot\left(\alpha_g\mu_g\nabla\vec{u}_g\right)^n + C_d\left(\vec{u}_l^* - \vec{u}_g^*\right) + SRC_g^n$ Interfacial friction: $C_d\left(\vec{u}_l^* - \vec{u}_g^*\right)$	① Phase link step $\alpha_g\rho_g \frac{\vec{u}_g^* - \vec{u}_g^n}{\delta t} = -\alpha_g\nabla P_i^n + C_d\left(\vec{u}_l^* - \vec{u}_g^*\right) + SRC_g^n$ Interfacial friction: $C_d\left(\vec{u}_l^* - \vec{u}_g^*\right)$ ② Space link step $\alpha_g\rho_g \frac{\vec{u}_g^{**} - \vec{u}_g^*}{\delta t} + \nabla\cdot\left(\alpha_g\rho_g\vec{u}_g^{**}\vec{u}_g^n\right) - \vec{u}_g^{**}\nabla\cdot\left(\alpha_g\rho_g\vec{u}_g^n\right)^n = \nabla\cdot\left(\alpha_g\mu_g\nabla\vec{u}_g^{**}\right)$ Convection: $\vec{u}_g^{**}\vec{u}_g^n$; Diffusion: $\vec{u}_g^{**}$

Energy Decoupled Method (Fully-implicit) (2/2)

» Energy Equation (ex. Vapor Energy)

Implicit terms

Semi-implicit

$$V_i \delta t^{-1} \alpha_{g,i}^n \rho_{g,i}^n \left(e_{g,i}^{n+1} - e_{g,i}^n\right) +$$
$$\sum_f \left[\left(\alpha_g \rho_g e_g\right)_f^n - e_{g,i}^n \left(\alpha_g \rho_g\right)_f^n \right] \Psi_{g,f}^{n+1} =$$
$$+V_i \delta t^{-1} \frac{\alpha_{g,i}^n P_i^n}{\rho_{g,i}^n} \delta\rho_{g,i} +$$
$$\frac{P_i^n}{\rho_{g,i}^n} \sum_f \left[\left(\alpha_g \rho_g\right)_f^n - \rho_{g,i}^n \alpha_{g,f}^n \right] \Psi_{g,f}^{n+1}$$
$$+\sum_f \left(\alpha_g \vec{q}_g\right)_f S_f + V_i \frac{P_{s,i}^n}{P_i^n} H_{ig} (T_i^{s,n+1} - T_{g,i}^{n+1})$$
$$-V_i \left(\frac{P_i^n - P_{s,i}^n}{P_i^n} \right) H_{gf} (T_{g,i}^{n+1} - T_{l,i}^{n+1})$$

Interfacial heat transfer

Fully-implicit

① Phase link step

$$V_i \delta t^{-1} \alpha_{g,i}^n \rho_{g,i}^n \left(e_{g,i}^* - e_{g,i}^n\right) =$$
$$+V_i \delta t^{-1} \frac{\alpha_{g,i}^n P_i^n}{\rho_{g,i}^n} \delta\rho_{g,i} + \frac{P_i^n}{\rho_{g,i}^n} \sum_f \left[\left(\alpha_g \rho_g\right)_f^n - \rho_{g,i}^n \alpha_{g,f}^n \right] \Psi_{g,f}^{n+1}$$
$$+\sum_f \left(\alpha_g \vec{q}_g\right)_f S_f + V_i \frac{P_{s,i}^n}{P_i^n} H_{ig} (T_i^{s,*} - T_{g,i}^*)$$
$$-V_i \left(\frac{P_i^n - P_{s,i}^n}{P_i^n} \right) H_{gf} (T_{g,i}^* - T_{l,i}^*)$$

Interfacial heat transfer

② Space link step

$$V_i \delta t^{-1} \alpha_{g,i}^n \rho_{g,i}^n \left(e_{g,i}^{n+1} - e_{g,i}^*\right) =$$
$$-\sum_f \left[\left(\alpha_g \rho_g\right)_f^n \left(e_{g,f}^{n+1} - e_{g,i}^{n+1}\right) \left(\alpha_g \rho_g\right)_f^n \right] \Psi_{g,f}^{n+1} + \sum_f \kappa_f \left[\frac{T_{g,j}^{n+1} - T_{g,i}^{n+1}}{\Delta r_{ij}} \right] \vec{S}_f$$

Convection

Diffusion

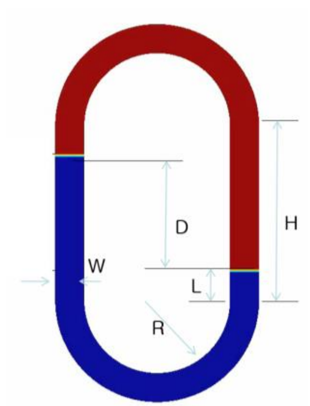
Verification & Validation

4

- Verification of Different Solution Scheme
- Validation for the Reflood Test Problems

Verification of Different Solution Schemes

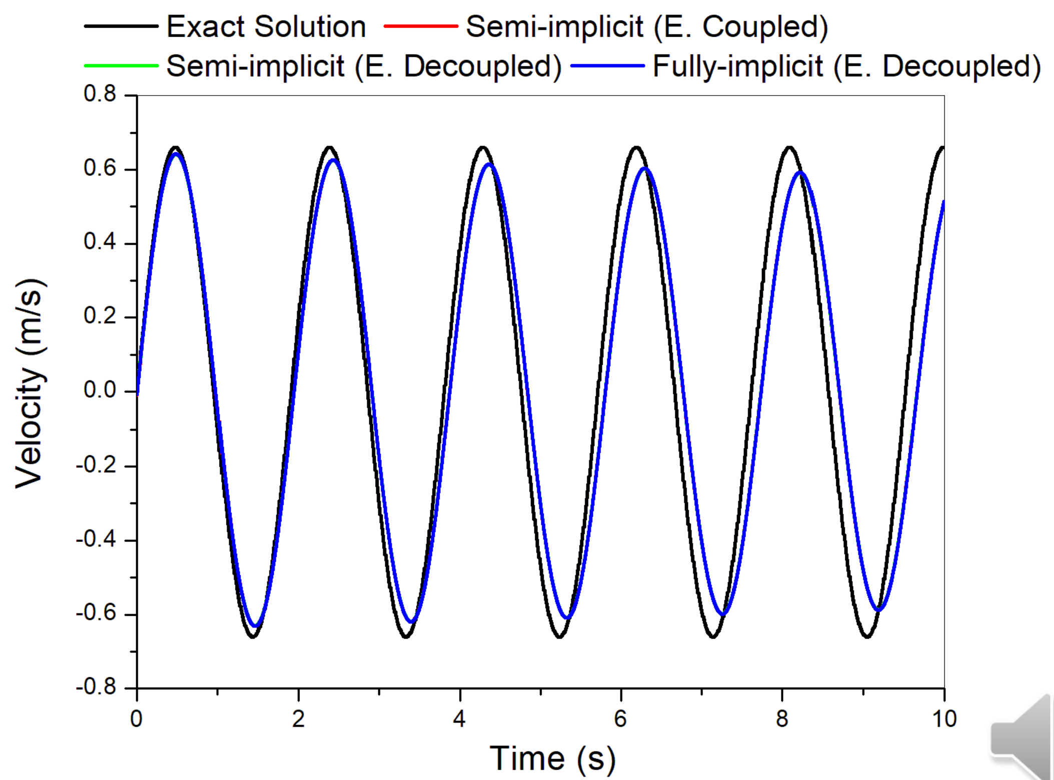
- ***3 different schemes are compared*** **using the Verification and Validation matrix consisting of** ***90 test cases*****.**
- ***Gravity-driven Flow Oscillations***
  - **3 solution schemes show the same result**



- D = 0.4 m
- R = 0.25 m
- H = 0.6 m
- L = 0.1 m
- W = 0.1 m

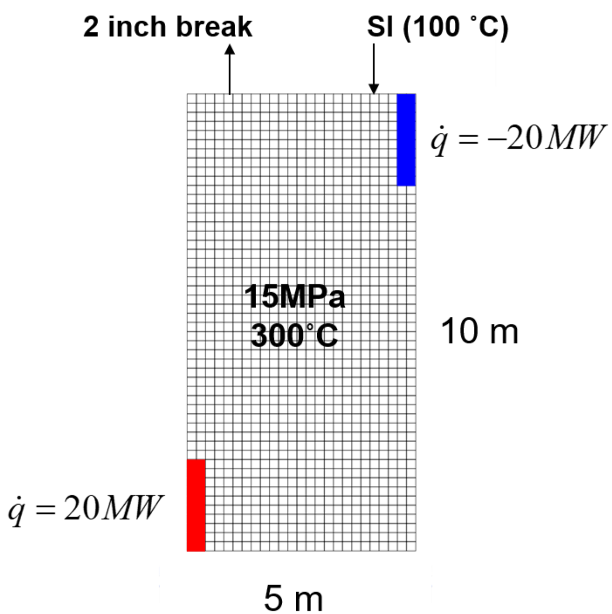
Liquid velocity at
the bottom of the channel

$$u(t) = H_0 \sqrt{\frac{2g}{L}} \sin\left(\sqrt{\frac{2g}{L}} t\right)$$

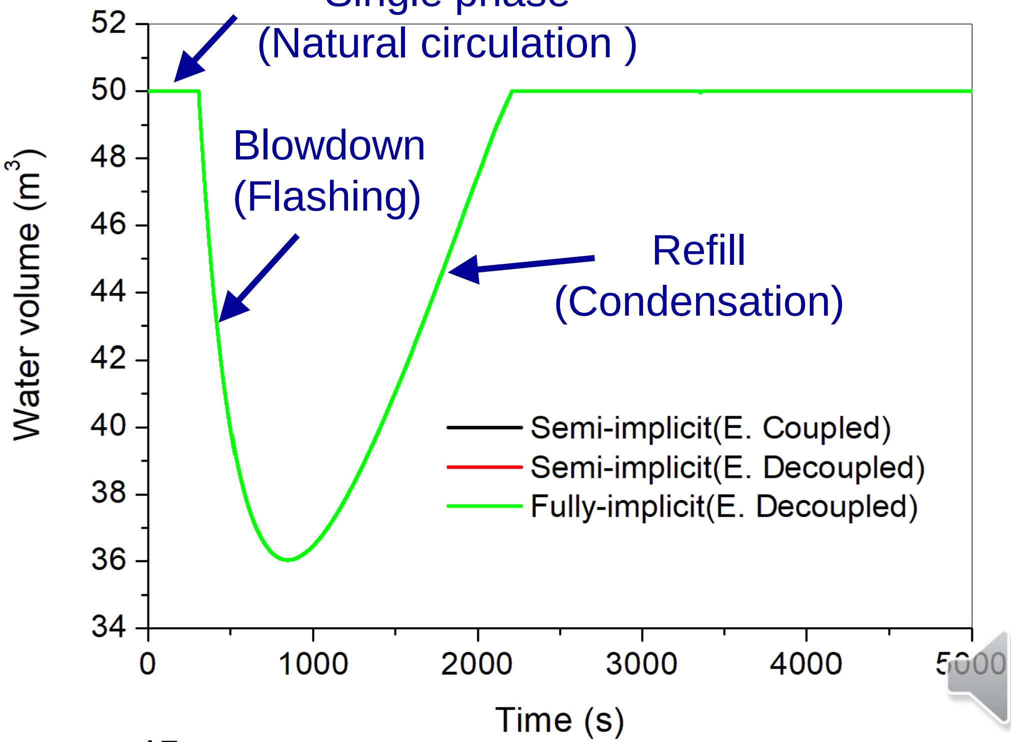


Verification of Different Solution Schemes

- ***2-phase flow numerical test*** **to simulate the *Blowdown and Refill* Phenomena**
  - **To test *Numerical stability* when *a large phase change is involved***
  - **Fast Transient: *Single phase liquid → Blowdown → Refill***
  - **3 solution schemes show the same result**



2500 2D cells (25x50)



Validation for the Reflood Test Problems (1/2)

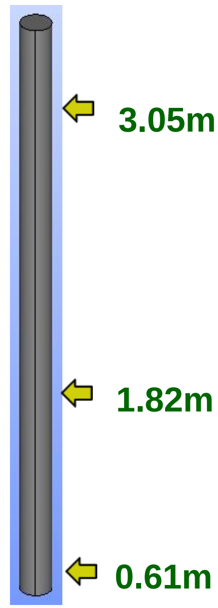
FLECHT-SEASET

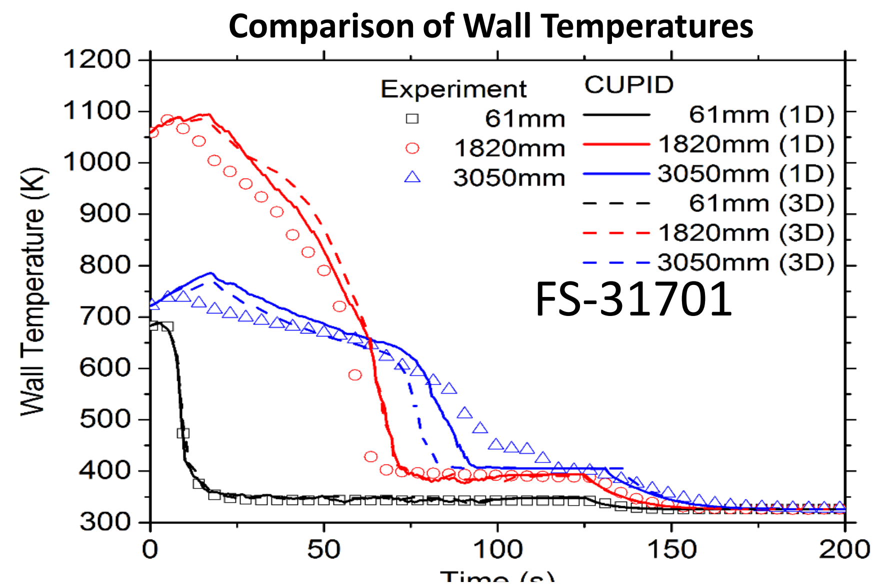
- Full-Length Emergency Core Heat Transfer – Separate Effects and System Effects Test (EPRI/NRC, 1986)
- 17x17 rod bundle test

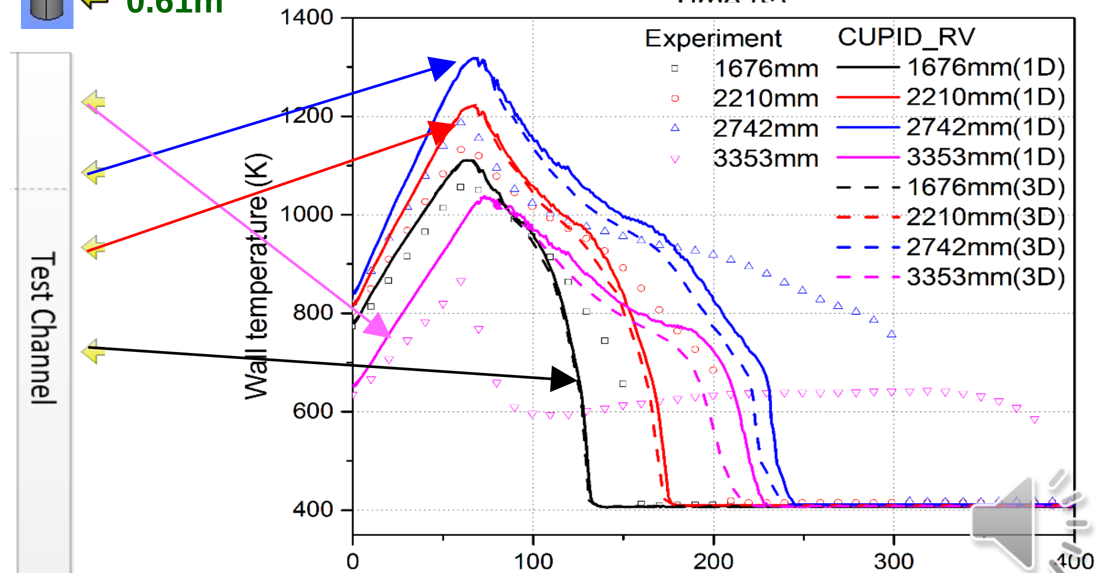
RBHT

- Rod Bundle Heat Transfer test (Pen State/NRC, 2012)
- 7x7 rod bundle in a square array

2-phase flow Models and correlations have been implemented in CUPID-RV for the analysis of *blowdown*, *refill*, and *reflood* phenomena following LBLOCA



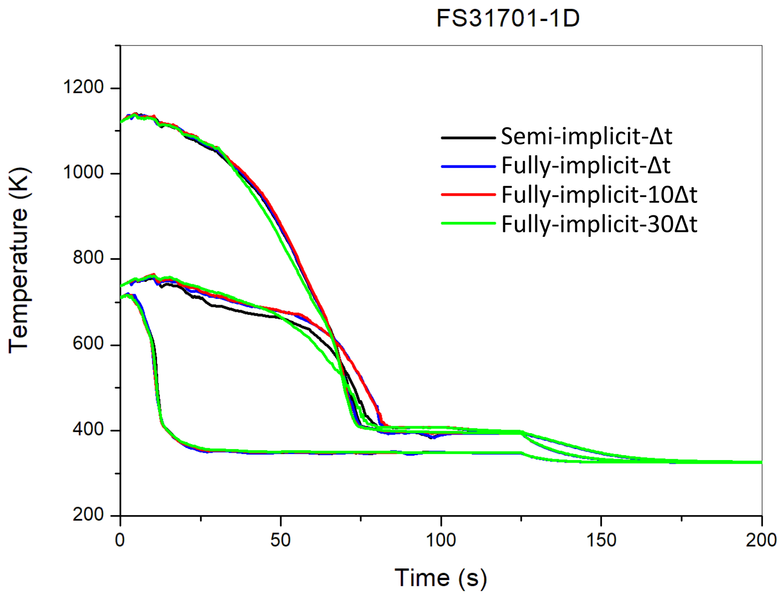


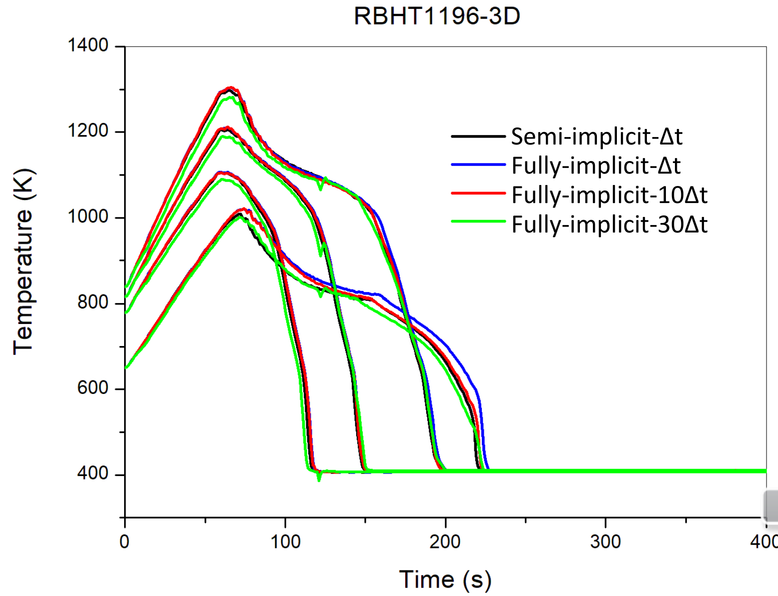


Validation for the Reflood Test Problems (2/2)

FS31701-1D	Computation time (s)
Semi-implicit (Δt)	69
Fully-implicit (Δt)	100
Fully-implicit (10Δt)	8
Fully-implicit (30Δt)	3

RBHT1196-3D	Computation time (s)
Semi-implicit (Δt)	1792
Fully-implicit (Δt)	2225
Fully-implicit (10Δt)	278
Fully-implicit (30Δt)	108





Summary

5

Summary

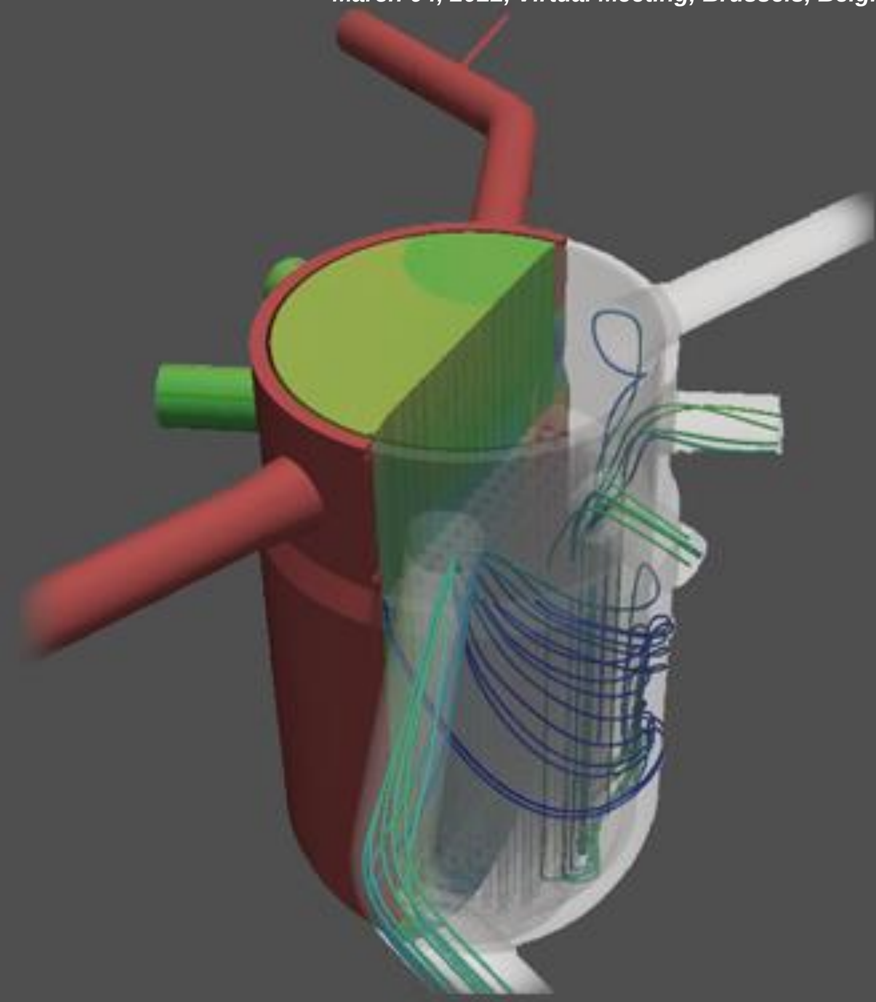
- **Discretization Method of CUPID**
  - **FVM for the modeling of complicated 3D geometries**
- **3 Solution Methods are available in CUPID**
  - **Energy Coupled Semi-implicit Scheme**
    - Used in the system analysis code and provides a reference solution
  - **Energy Decoupled Semi-implicit Scheme**
    - Improved numerical stability due to the nearly symmetric pressure matrix
  - **Energy Decoupled Fully-implicit Scheme**
    - Allows a large CFL number for a fast 2-phase flow calculation
- **The fully-implicit scheme provides a *fast and robust simulation* of the *transient 2-phase flow with a large phase change* that is important for the analysis of LWR thermal hydraulics**

THANK YOU

hyyoon@kaeri.re.kr

Korea Atomic Energy Research Institute · 한국원자력연구원 · KAERI

19th International Topical Meeting on Nuclear Thermal Hydraulics
March 04, 2022, Virtual Meeting, Brussels, Belgium



CUPID Workshop

Highly Scalable Iterative Solver (Geometric Multi-Grid Method for Unstructured Mesh)

Seongju Do
March 04, 2022

CUPID Workshop

CONTENTS

WHY ‘Multi-Grid’?

1

WHY 'Multi-Grid'?

- **One of the most time-consuming part in CUPID is the "Poisson equation" solving module.**
  - **Total cost of solving pressure matrix is 40-90%.**
  - **The Conjugate Gradient (CG) solver is**
    - Widely used in commercial CFD software
    - Not scalable w.r.t the number of cells *N*

$$CG : t_{CPU} \propto N^{1.5}$$

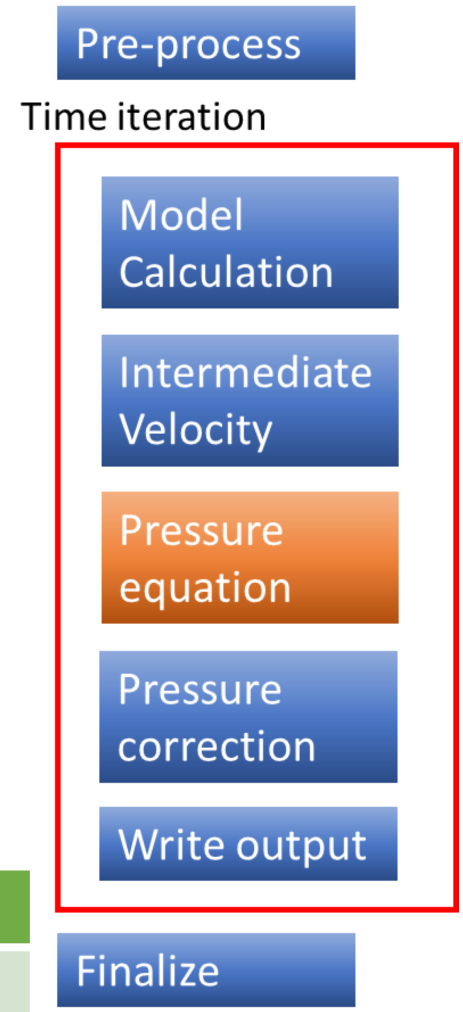
$$PBICG : t_{CPU} \propto N^{1.4}$$

  - **Development of Multi-Grid(MG) solver which is scalable w.r.t the number of cells**

$$MG : t_{CPU} \propto N^{1.0}$$

What does the exponent mean?

#Cells	CPU time of CG	CPU time of MG
10,000	1 min	1 min
1,000,000	$100^{1.5}$ min ≈ 17 hours	100^{1} min ≈ 1.7 hours
100,000,000	$10{,}000^{1.5}$ min ≈ 694 days	$10{,}000^{1}$ min ≈ 7 days



<CUPID work flow>

WHAT is ‘GMG’?

2

WHAT is GMG?

Comparison MG / AMG / GMG

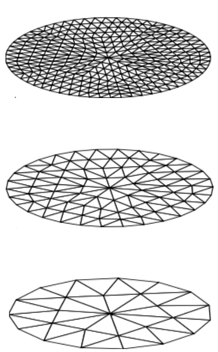
[1] Cao Lu et al. "A Hybrid Geometric + Algebraic Multigrid Method with Semi-Iterative smoother" NUMERICAL LINEAR ALGEBRA WITH APPLICATIONS, 2013

	Classic MG	AMG	GMG
Memory requirement	low	high	low
Operator complexity	less costly	more costly	less costly
Irregular domains	not robust	robust	Moderate
user friendliness	less friendly	more friendly	Moderate

Mesh

Mesh 2

Mesh 3



Coarsening
- Manually

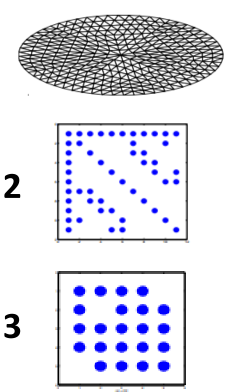
Interpolation
(Geometric)

<MG>

Mesh

Matrix 2

Matrix 3



Coarsening
- Automatically

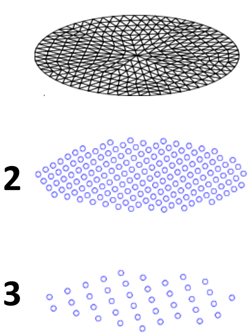
Interpolation
(algebraic)

<AMG>

Mesh

Graph 2

Graph 3



Coarsening
- Automatically

Interpolation
(Geometric)

<GMG>

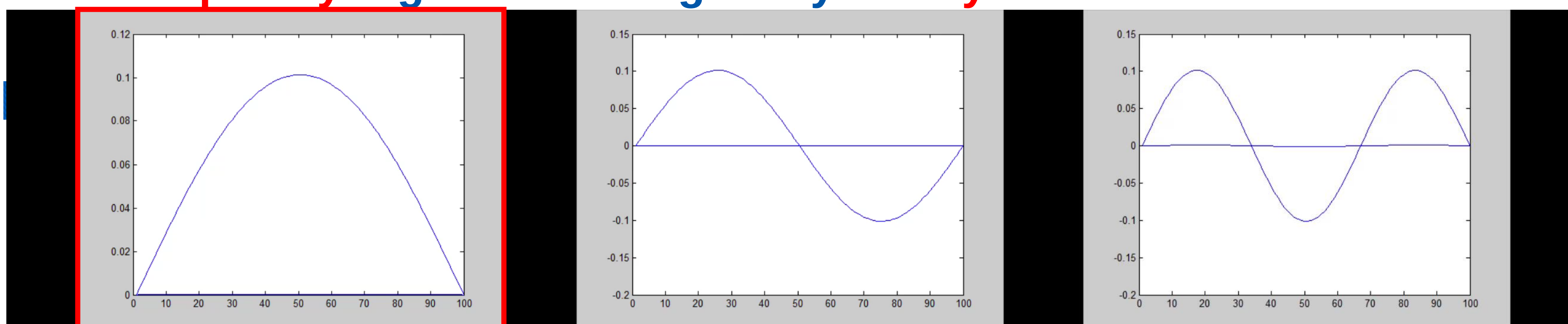
GMG Algorithm

3

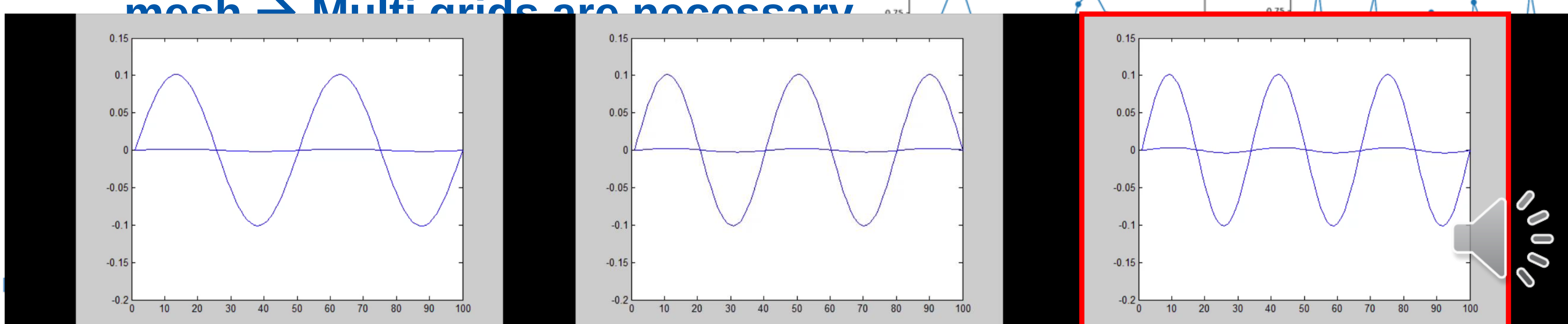
GMG algorithm

Jacobi/Gauss-Seidel solver

- One of the simplest iterative linear solver
- Computational complexity for convergence is $O(N^2)$
  - Impossible to use in practical areas
- **High-frequency** parts of the error converge **quickly**, while the **low-frequency** regions converge very **slowly**.



- Resolve small error on coarse mesh → Multi grids are necessary



GMG algorithm

Elements of MG solver

➢ Coarse mesh generator

- Semi-coarsening method

➢ Data transfer between coarse and fine meshes.

- Interpolator (coarse → fine)
- Restrictor (fine → coarse)

➢ Smoother

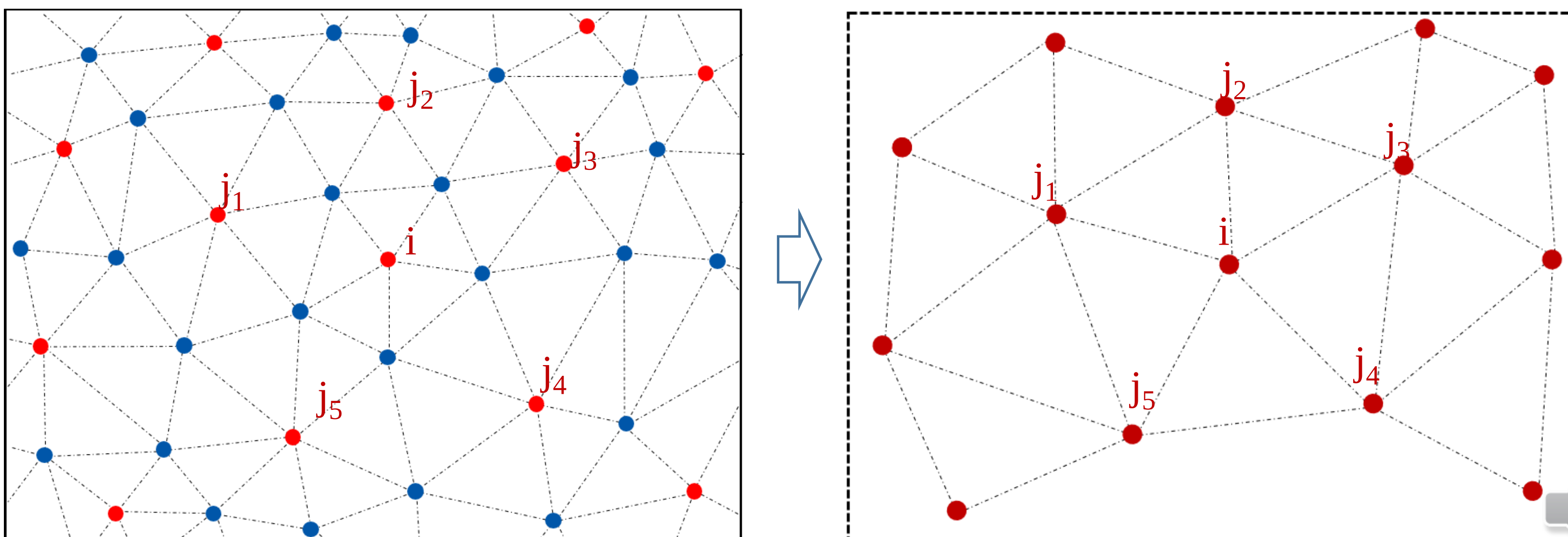
➢ V-cycle iteration

GMG algorithm

[2] Herve Guillard, Node-nested multi-grid with Delaunay coarsening, (1993)

Coarse mesh generator

- **Only the finest mesh is required as an input data**
  - Automatic mesh generation on coarse levels
- **Node-coarsening by MIS(Maximum Independent Set [2])**
  - Initially, mark all the nodes in finer mesh '**green**'
  - And then, for each node in a finer mesh:
    - ✓ If the node is **green** add this node to the list of **red nodes** and mark its neighbor as the **blue nodes**.
    - ✓ Otherwise, go to the next node.

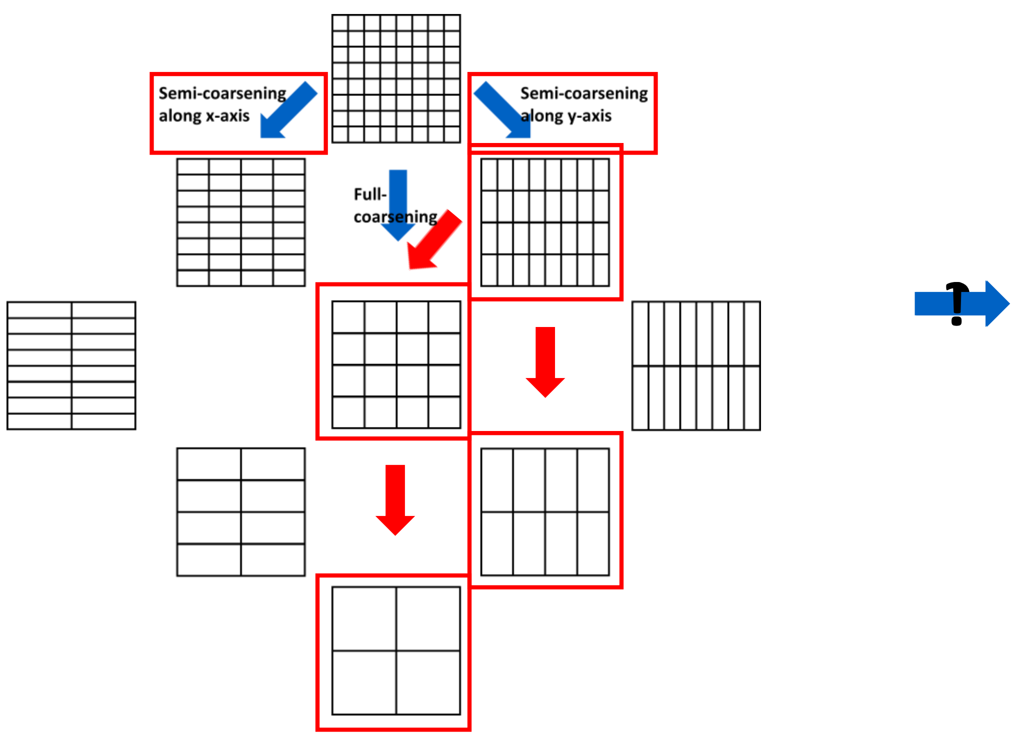


GMG algorithm

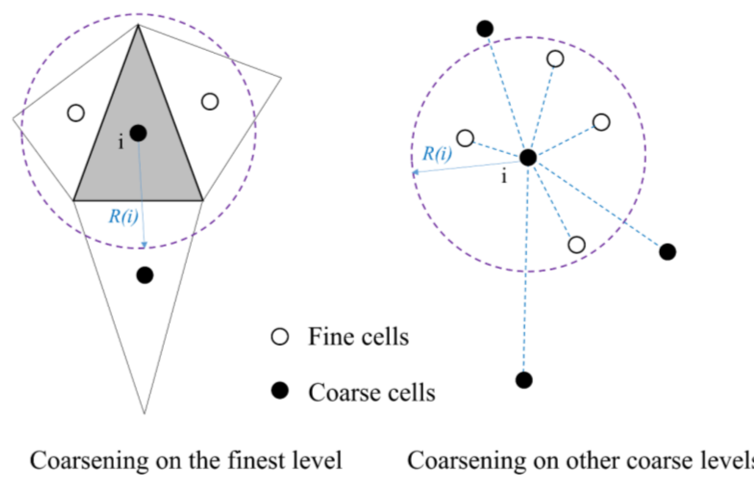
Semi-Coarsening technique [3]

[3]E.MORANO, Coarsening Strategies For Unstructured Multigrid Technique with Application to Anisotropic Problem, *SIAM J.SCI.COMPUT.Vol.20, No2.pp.393-415* (1998)

- In case of high aspect ratio mesh, the convergence speed of MG solver could be slower.
- Semi-coarsening algorithm in structured mesh is extended to unstructured mesh.



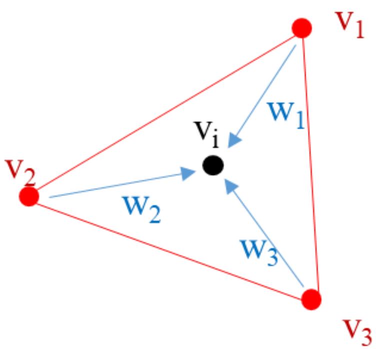
<Unstructured mesh>



GMG algorithm

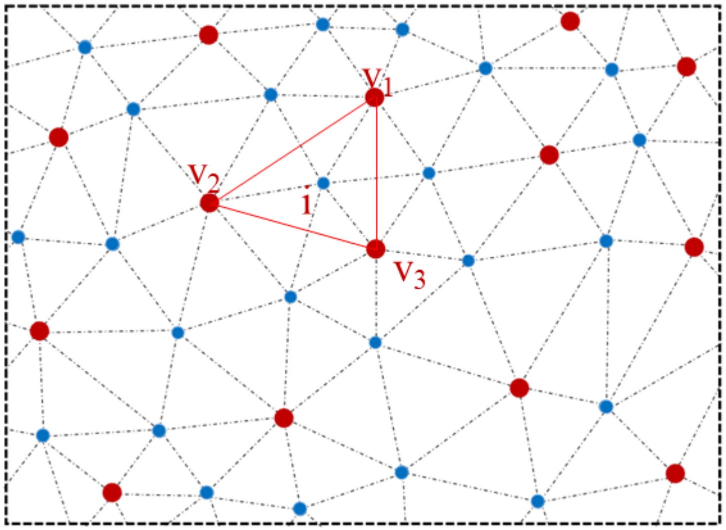
Data transfer between coarse and fine meshes

- Interpolator (coarse → fine)
  - ✓ Inverse distance

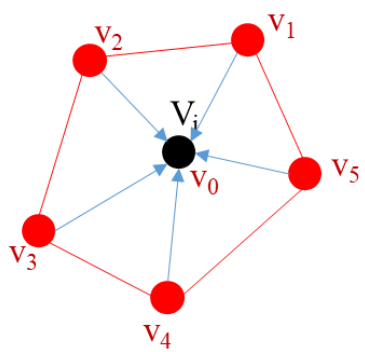


$$v_i = \frac{w_1 v_1 + w_2 v_2 + w_3 v_3}{w_1 + w_2 + w_3}$$

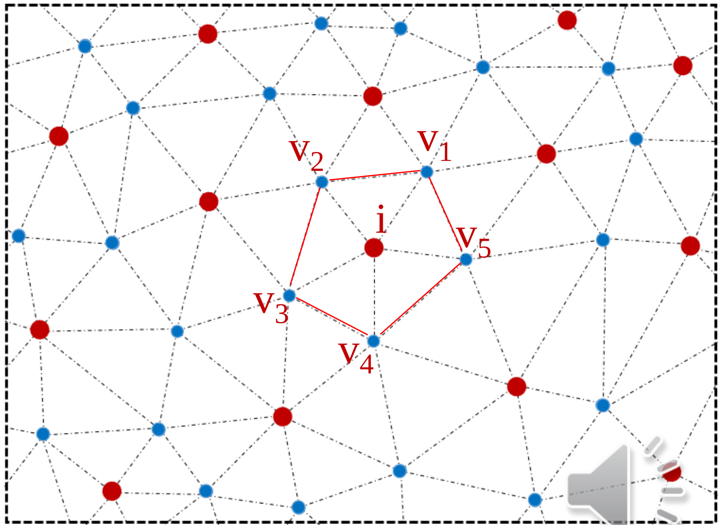
$$w_k = \frac{1}{d(v_i, v_k)}$$



- Restrictor (fine→ coarse)
  - ✓ Injection



$$V_i = v_0$$



GMG algorithm

- **Smoother**
  - **Jacobi**
  - **Gauss-Seidel**
  - **Weighted Jacobi**
  - **Weighted Gauss-Seidel (SOR)**

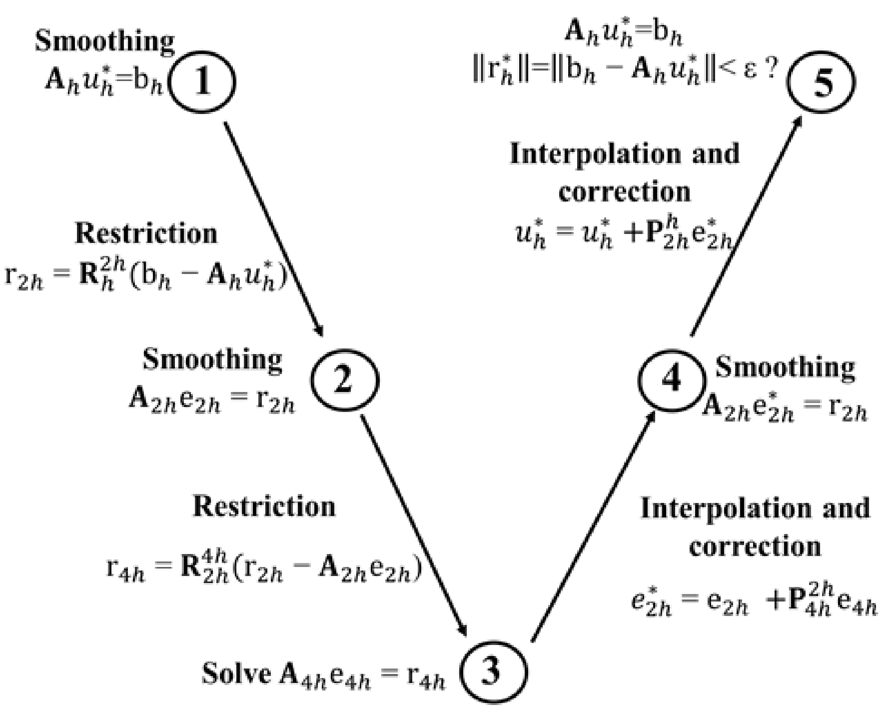
- **In CUPID, 2-3 times SOR sweeps are performed on each meshes.**

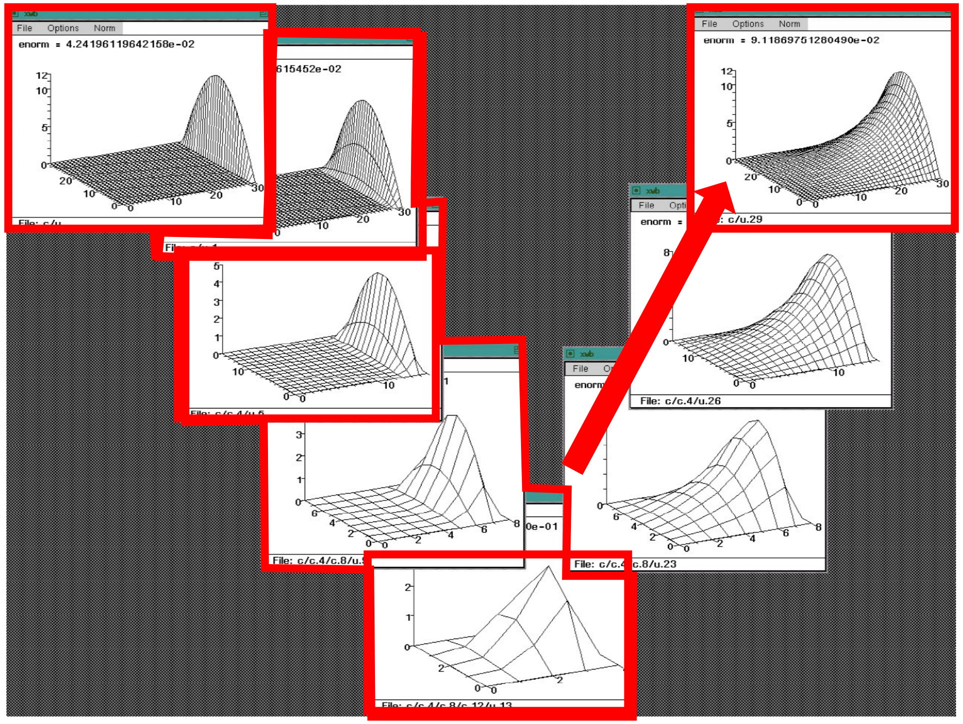
GMG algorithm

[3] William L. Briggs et al. “A multigrid tutorial” SIAM, 2000
[4] http://www.mgnet.org/mgnet/tutorials/xwb/smoother.html

V-cycle iteration [3]

- Smoothing residual vectors for each level
- Correct solution by adding smoothed residual vector





V-cycle workbench [4]

Speedup Test

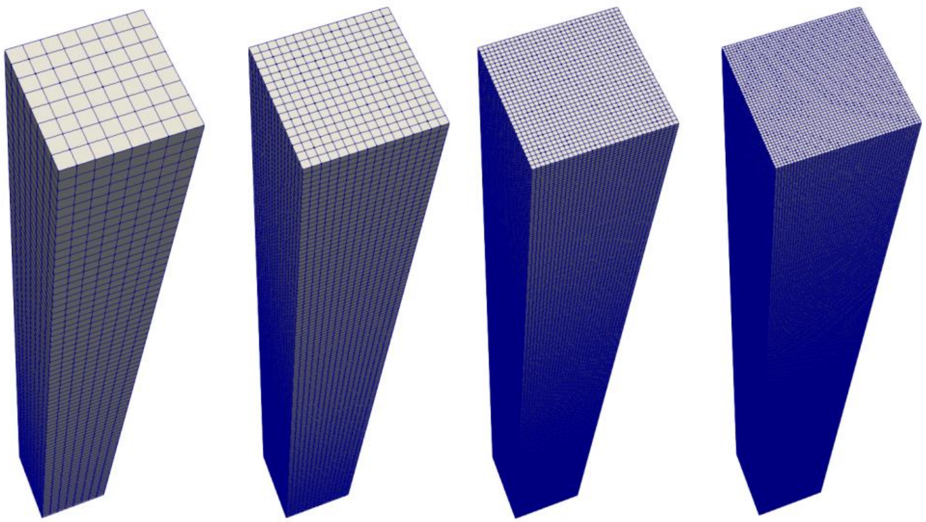
4

- 3D boiling flow (two-phase/structured)
- 3D Reactor Vessel problem (single-phase/unstructured)
- 3D Channel flow (single-phase/unstructured/ ~100 million cells)

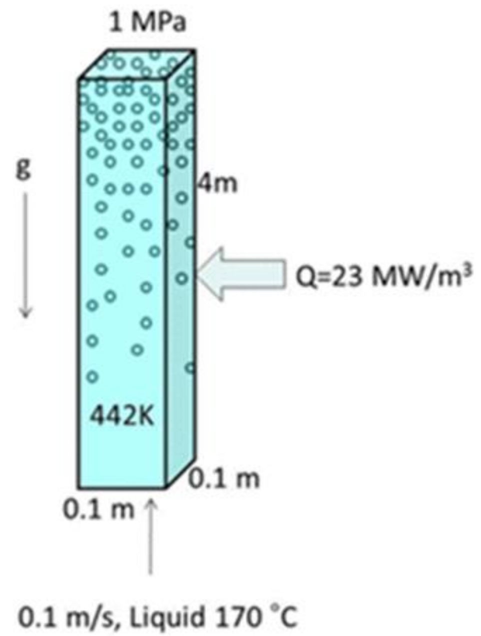
Speedup Test – 3D boiling (1/2)

3D boiling test

- Two-phase simulation
- 4 kinds of structured meshes are used to evaluate the performance of GMG solver



5,120 | 40,960 | 327,680 | 1,105,920

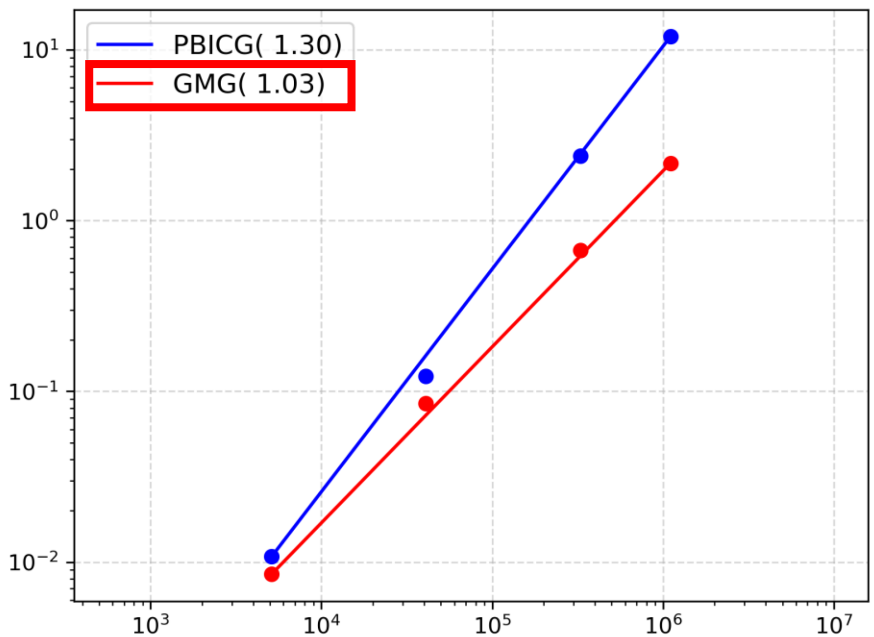


<Test setup>

Speedup Test – 3D boiling (2/2)

3D boiling test

Result of automatic mesh coarsening



<Computational complexity>

Comparison of PBICG / GMG

- The number of iteration in GMG is *constant* regardless of the number of cells.
- The larger the problem size, the greater the benefit of GMG.

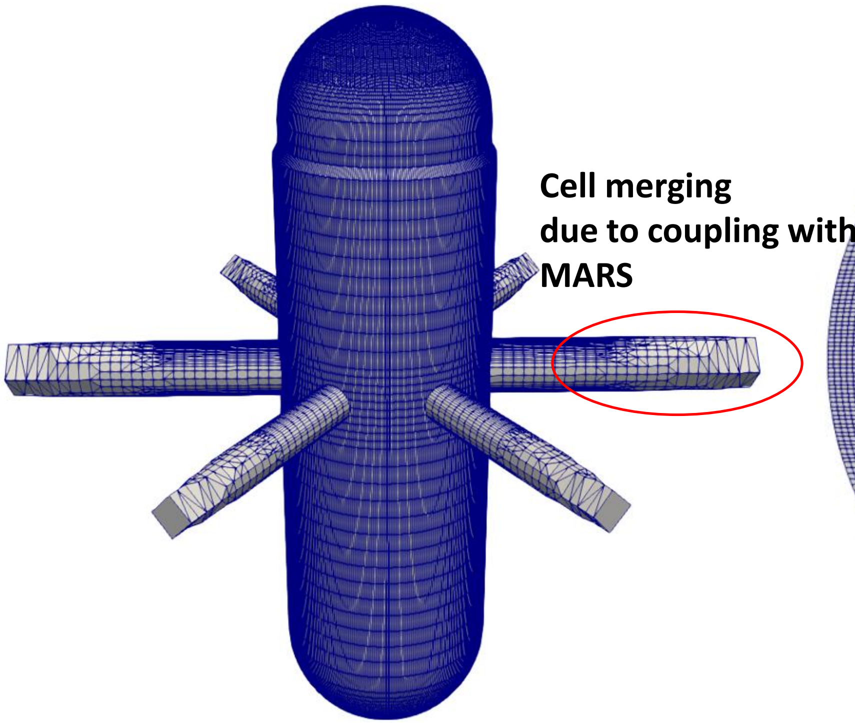
Case	Num. cells	PBICG iteration	GMG iteration	Speed up [times]
Mesh 1	5,120	37	10	0.88
Mesh 2	40,960	63	10	1.44
Mesh 3	327,680	124	10	3.91
Mesh 4	1,105,920	174	10	5.40

Increasing! **Constant !**

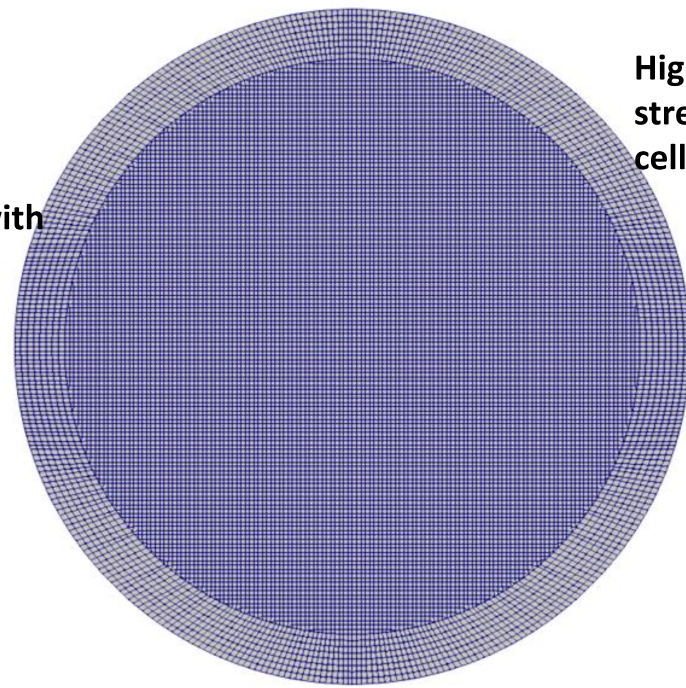
Speedup Test – RV problem (1/2)

RV test

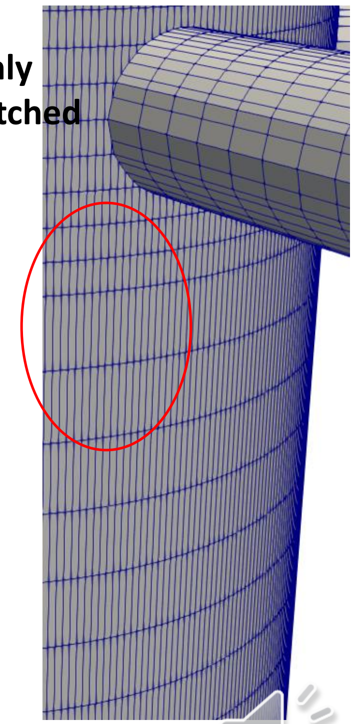
- Polygonal cells / High aspect ratio mesh
- 4 kinds of unstructured meshes are used to evaluate the performance of GMG solver



<RV mesh>



<Cross section>

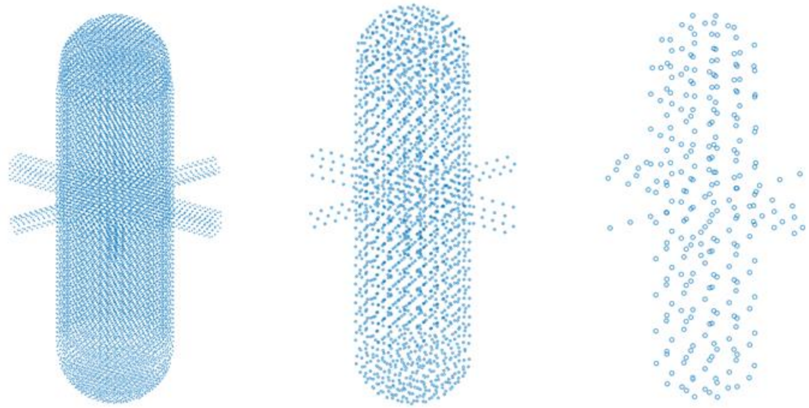


<Junction>

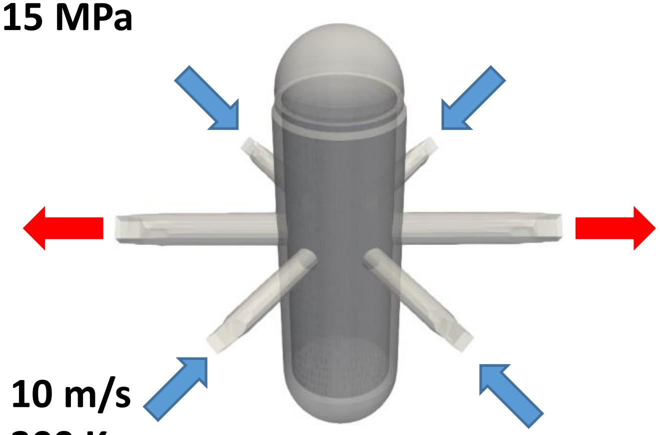
Speedup Test – RV problem (2/2)

RV test

Result of automatic mesh coarsening



<Nodes after coarsening>

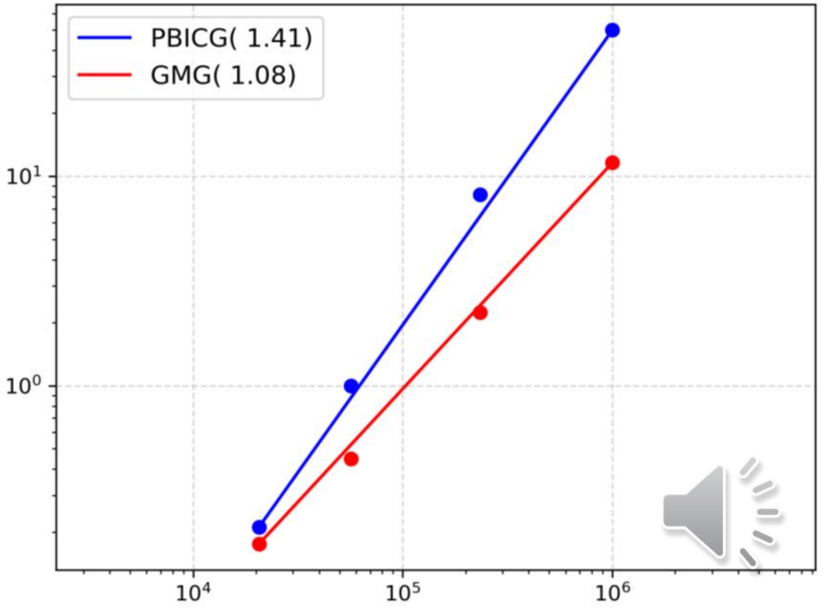


<Test setup>

Comparison of PBICG / GMG

Case	Num. cells	PBICG iteration	GMG iteration	Speed up [times]
Mesh 1	20,619	121	28	1.273
Mesh 2	56,654	171	22	2.246
Mesh 3	234,122	263	26	3.617
Mesh 4	1,003,086	317	28	4.245

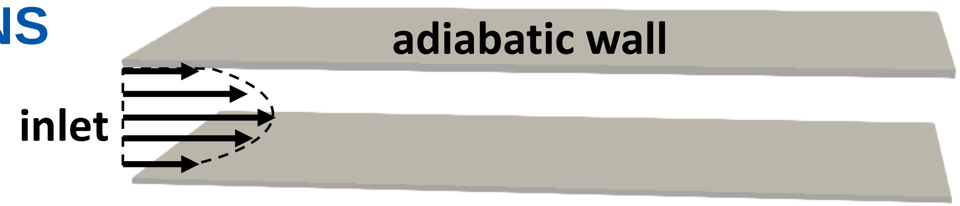
Almost constant



Speedup Test – Channel flow (1/2)

Single phase channel Flow

- **Well-known problem in the DNS community**



<Channel configuration>

- **Computational setup**
  - Domain

$$4\pi\delta \times 2\delta \times \frac{4\pi\delta}{3} \quad (\delta = 0.01m)$$

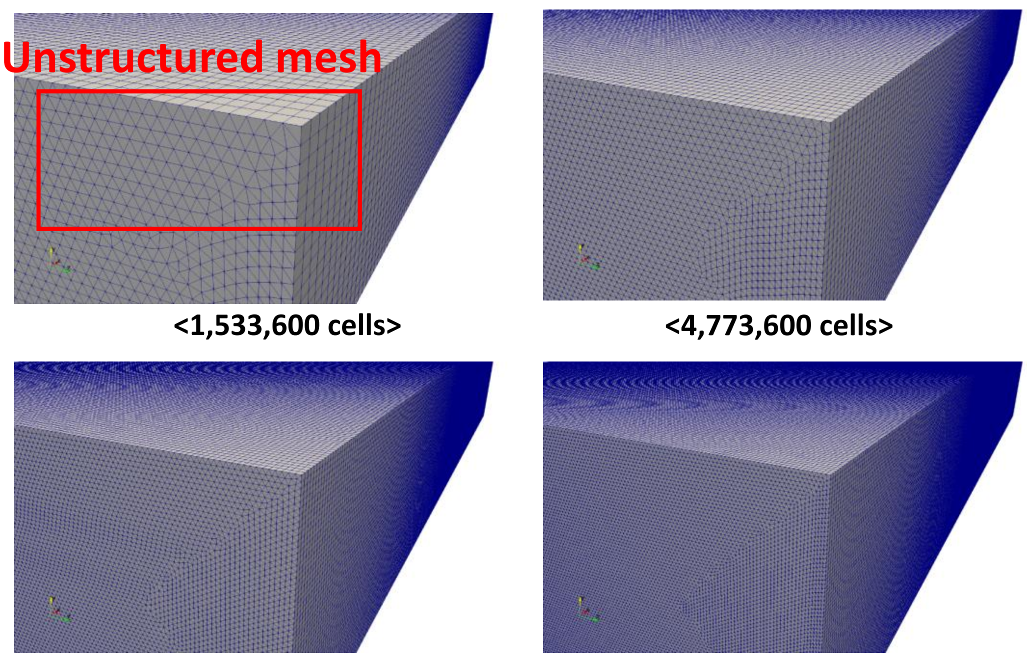
  - 6 kinds of unstructured meshes

191,800(mesh1)	1,533,600(mesh2)
4,773,600(mesh3)	12,357,600 (mesh4)
21,683,700(mesh5)	107,968,000(mesh6)

  - Low Reynolds number

$$\mathrm{Re} = \frac{U_h \delta}{\nu} = 283.4$$

  - Inlet condition
    - ✓ parabolic velocity profile from DNS data



<1,533,600 cells> <4,773,600 cells>

<12,357,600 cells> <21,683,700 cells>

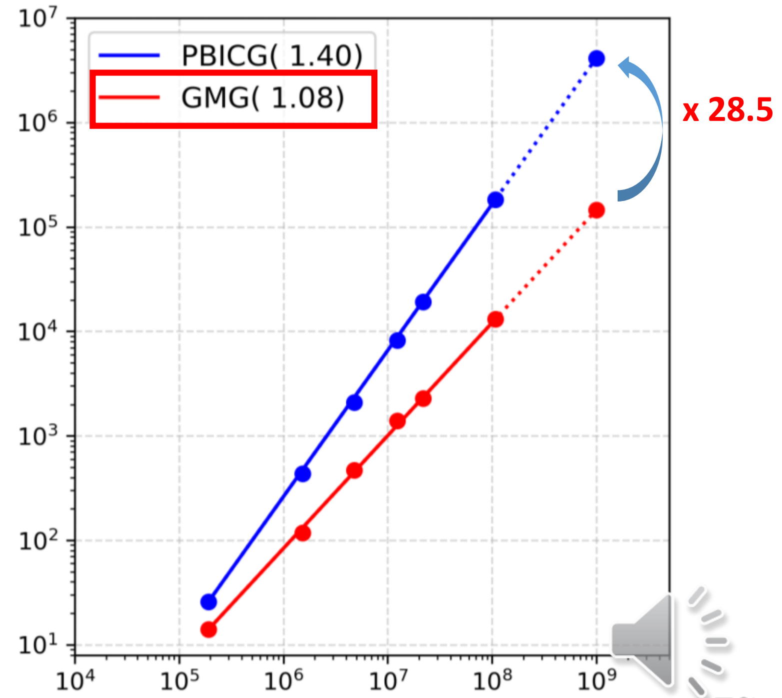
<Computational meshes>

Speedup Test – Channel flow (2/2)

➢ Comparison of PBICG / GMG

- The overall trend is similar to the previous problems.
- The complexity exponent of GMG is **1.08**, which is slightly larger than the theoretical value 1.
- In the case of **1 billion** cells, GMG is predicted to improve matrix solving performance by about **28.5** times.

Case	Num. cells	PBICG iteration	GMG iteration	Speed up [times]
Mesh 1	191,800	152	21	1.84
Mesh 2	1,533,600	216	27	3.66
Mesh 3	4,773,600	463	27	4.57
Mesh 4	12,357,600	640	27	5.85
Mesh 5	21,683,700	892	28	8.44
Mesh 6	107,968,000	1472	29	13.63



Speedup Test – Total computational time

Calculation load

1st part

Pressure Solve

Computational cost: $O(n^{1.4})$ for PBICG

Do Until converge ← depends on n
  Do cell loop
    (PBICG algorithm)
  End Do
End Do

2nd part

Modeling

Summation

UDFN

...

Computational cost: $O(n)$

Do cell loop
  (Do something.)
End Do

Do face loop
  (Do something.)
End Do

- **How much can GMG solver contribute to reduction of whole CPU time?**
- **Computational time of 'Pressure solving' is dominant as 'n' increases.**

Total calculation

Pressure solve

Num. cells	$\frac{t_{PBICG}}{t_{total}}$	Speed up (pressure)	Speed up (total)
191,800	0.566	∞	2.30
1,533,600	0.739	3.66	2.160
4,773,600	0.816	4.57	2.758
12,357,600	0.840	5.85	3.293
21,683,700	0.902	8.44	4.881
107,968,000	0.946	13.63	8.069
1,000,000,000	0.987	28.52	21.005

1 day → 1 hour

Summary

- GMG solver can **stably** and **efficiently** solve the **single/multi-phase** problem on **unstructured** meshes.

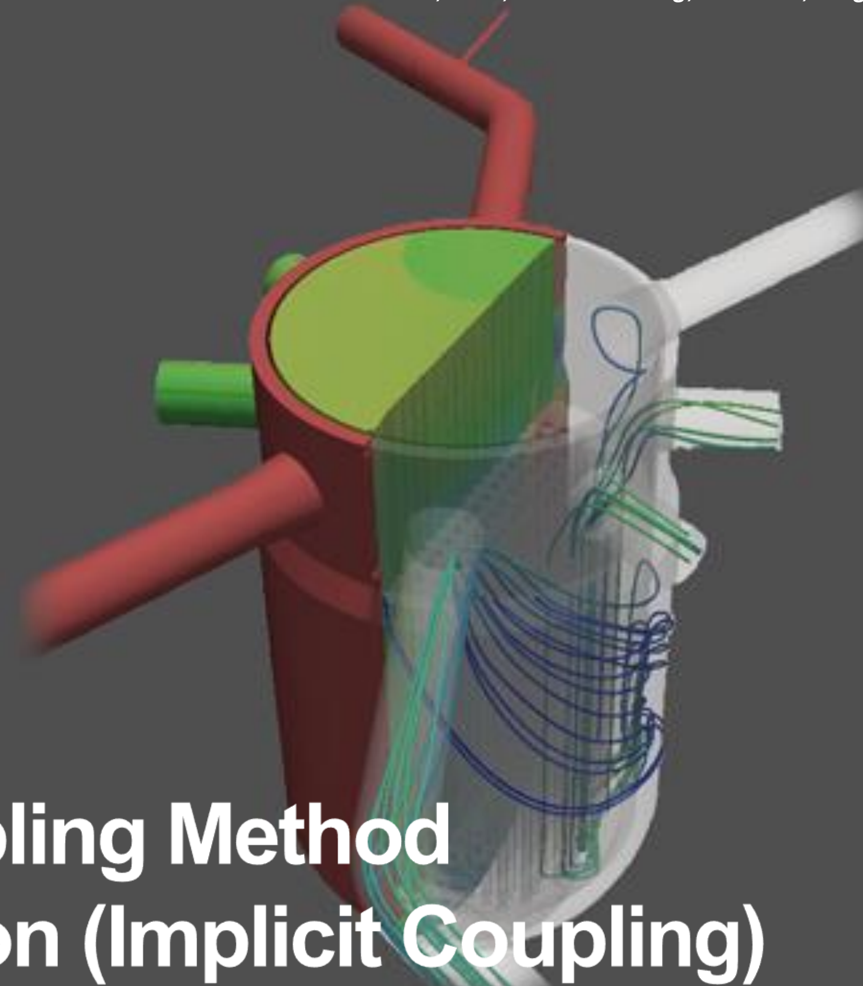
- The performance gain of GMG solver becomes **more larger** as the number of mesh increases.

- GMG solver can be used in calculations on **unstructured** meshes **without additional user's effort**

THANK YOU

sjdo@kaeri.re.kr

19th International Topical Meeting on Nuclear Thermal Hydraulics
March 04, 2022, Virtual Meeting, Brussels, Belgium



CUPID Workshop

Unique Multi-Scale Coupling Method for a Transient Calculation (Implicit Coupling)

Ik Kyu Park
March 04, 2022

CUPID Workshop

CONTENTS

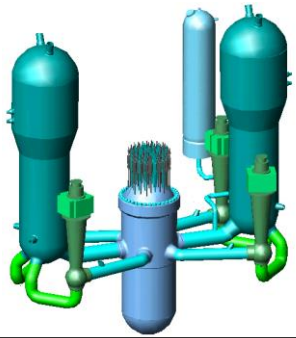
Multi-Scale Method for PWRs

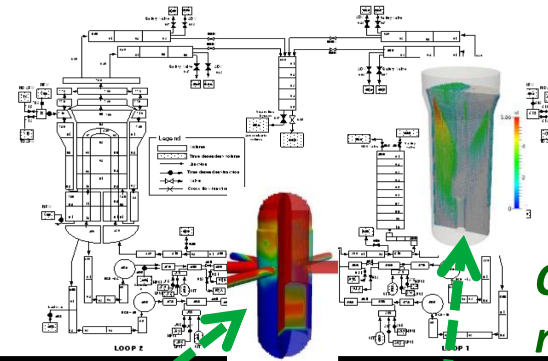
1

- Overview of Multi-Scale Approach
- Multi-Scale Coupling Strategy of CUPID

Multi-Scale Approach in Nuclear T/H

System-scale





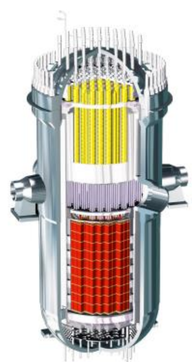
~10^{0} m

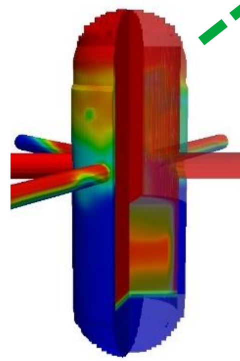
Combined multi-scale calculation

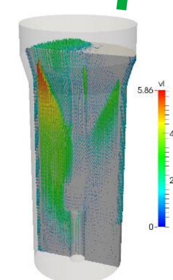
CUPID-RV

CUPID-SG

Component-scale
(CFD-Porous)
(subchannel-scale)



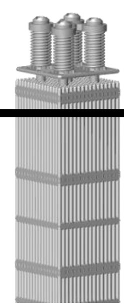


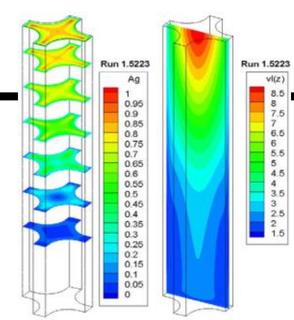


~10^{-3} m

CFD-RANS

CUPID-CFD





~ 10^{-4} m

CFD-LES

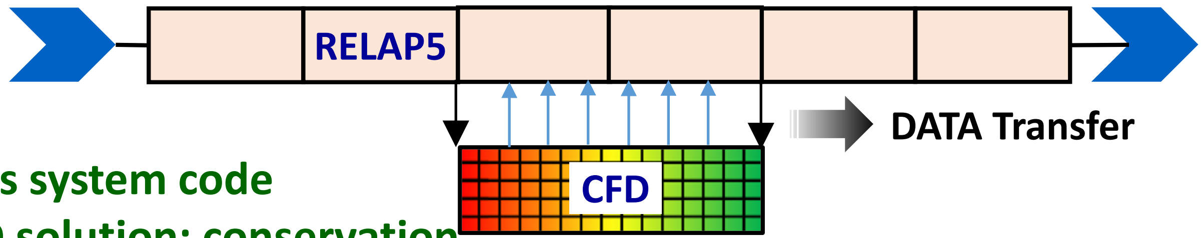
~ 10^{-5} m

CFD-DNS

~ 10^{-6} m

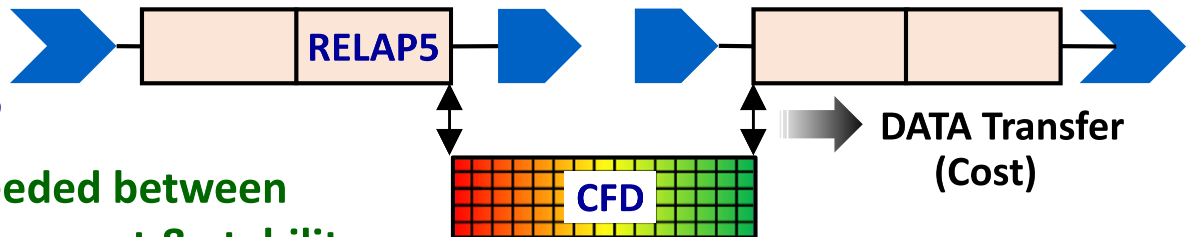
Overview of Multi-Scale Methods

Domain Overlapping



CFD code overlaps system code solution with CFD solution: conservation

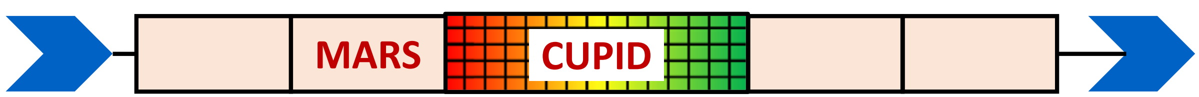
Domain Decomposition



Data Transfer is needed between two separate solvers: cost & stability

Ex) RELAP5/ CFX, FLEUNT, STAR-CCM+

Single Domain

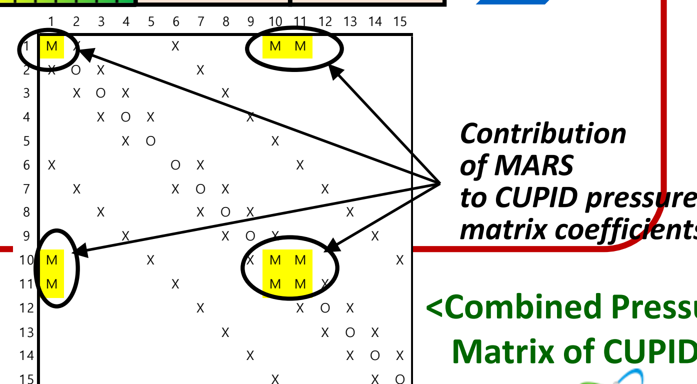


System analysis code developed by KAERI

Single pressure solver matrix

: No need for the Data Transfer

→ *Versatile application to transient problems*



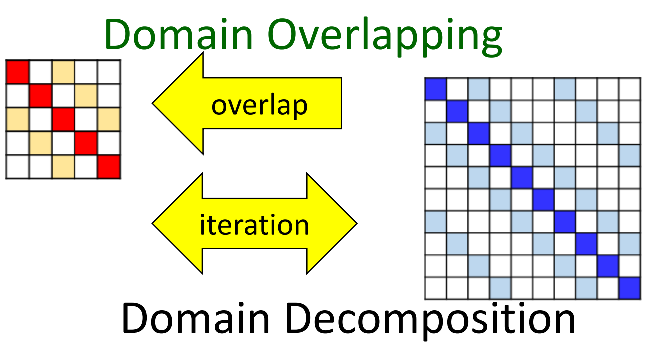
<Combined Pressure Matrix of CUPID>

** I.K.Park et al., Annals of Nuclear Energy, 2013.*

Multi-Scale Coupling Strategy of CUPID

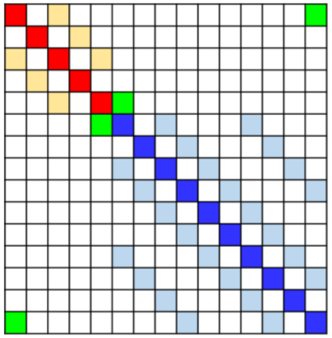
Comparison Multi-Scale Coupling Methods

Method	Characteristics	Limitation	Applications
Domain Overlapping	Transfer B.C. Overlapping solutions	Mapping Conservation Fast transient	CATHARE2/TrioCFD SFR Natural Convection
Domain Decomposition	Iterate two solvers transferring B.C.	Fast transient	ATHLET/OpenFOAM ROCOM PKL3 Test 1.1 Flow Mixing
Single Domain (Implicit coupling)	Build a single solver matrix combining two domains	Two source codes should be accessible.	MARS/CUPID APR1400 MSLB Accidents



Combined Single Domain with a Single solver

-manipulate pressure matrix coefficients



Multi-Scale Coupling Method

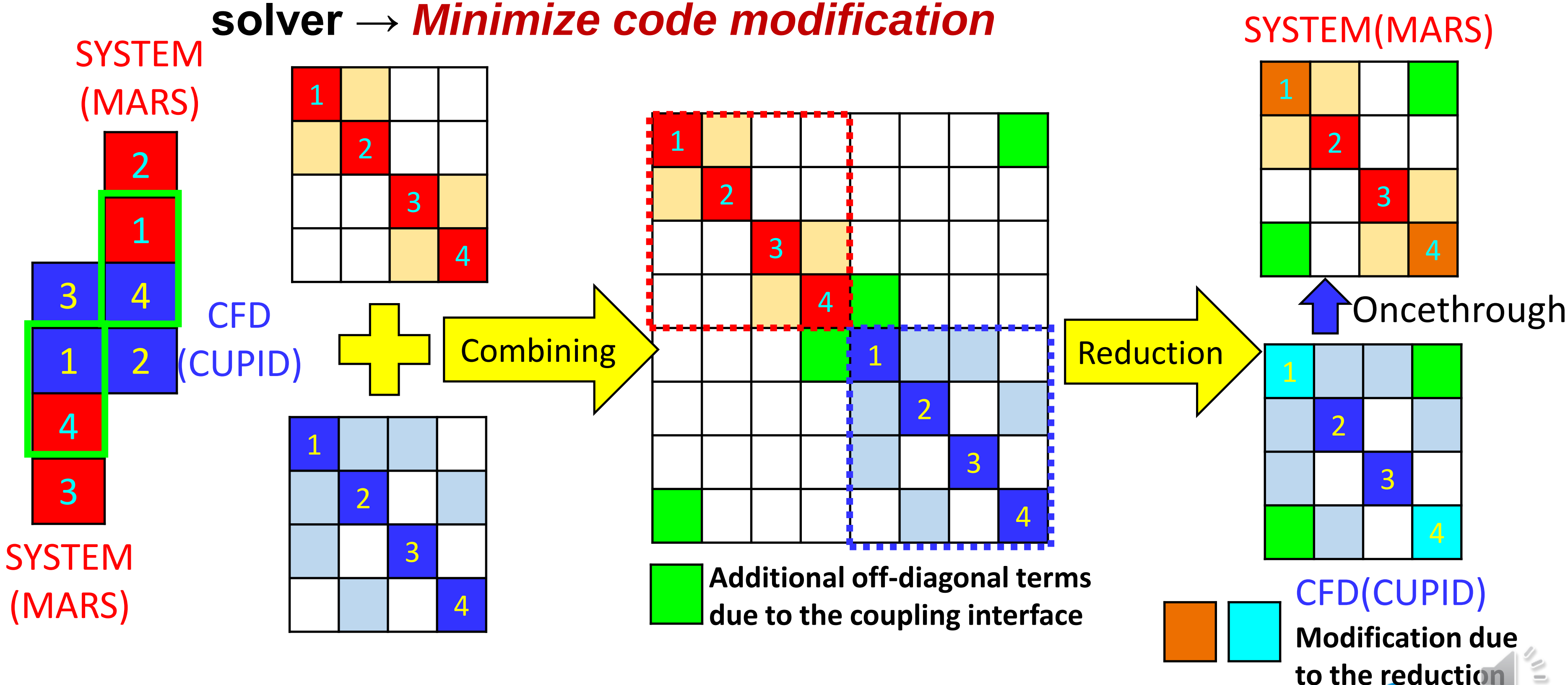
2

- Implicit Multi-Scale Coupling Method
- Verification of the Implicit Coupling Method

Implicit Multi-Scale Coupling Method (1/3)

Strategy of the Coupling Method

- **Solve a *single matrix* for the combined single domain**
  - **The pressure matrices of two codes are *combined***
  - ***Matrix reduction* is needed to fit the matrix into each code solver → *Minimize code modification***



Implicit Multi-Scale Coupling Method (2/3)

Contribution of Coupling Face to Pressure Solver

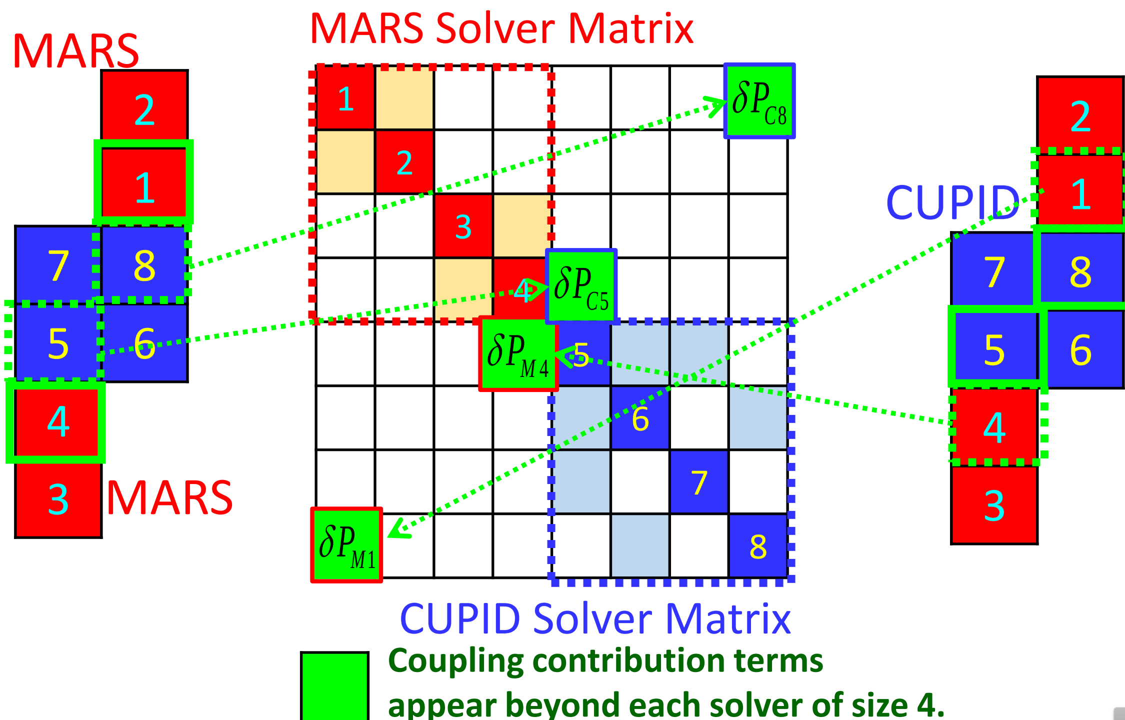
- **Pressure correction eq. from mass & energy eqs.**
  - Relation btw volume flow & pressure correction from momentum eq.
- **Neighbor cell effects appears on off-diagonal terms**
  - Need to eliminate off-diagonal terms contributed by the partner code

$$\delta P_i = b_i + \sum_{f=1,NB} S_f V_f^{n+1}$$

$$S_f V_f^{n+1} = \alpha_f + \beta_f \left(\delta P_j - \delta P_i\right)$$

$$\boxed{\delta P_i} = b'_i + \sum_{f=1,NB} \left(\beta_f \left(\delta P_j - \delta P_i\right)\right)$$

j=Neighbor cell

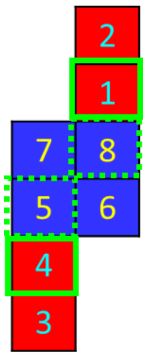


Implicit Multi-Scale Coupling Method (3/3)

Reduction to fit the matrix into each code solver

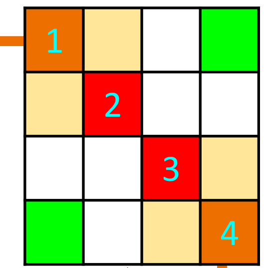
- Relation of pressure corrections btw coupled cells in MARS
- Eliminate MARS contribution in CUPID pressure correction eqs.
- Solve pressure matrix from CUPID to MARS

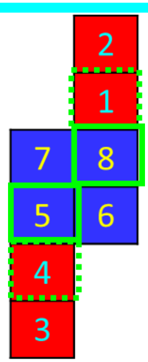
MARS



$$\beta'_1 \delta P_1 - \beta_2 \delta P_2 = b_1 + \beta_{C8} \delta P_{C8}$$
$$\beta'_2 \delta P_2 - \beta_1 \delta P_1 = b_2$$
$$\beta'_3 \delta P_3 - \beta_4 \delta P_4 = b_4$$
$$\beta'_4 \delta P_4 - \beta_3 \delta P_3 = b_4 + \beta_{C5} \delta P_{C5}$$

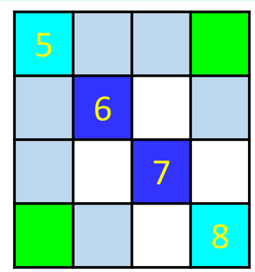
$$\delta P_{M4} = \varepsilon_2 + \eta_{2,1} \delta P_{C5} + \eta_{2,2} \delta P_{C8}$$
$$\delta P_{M1} = \varepsilon_1 + \eta_{1,1} \delta P_{C5} + \eta_{1,2} \delta P_{C8}$$





$$\beta'_5 \delta P_5 - \beta_6 \delta P_6 - \beta_7 \delta P_7 - \beta_{M4} \delta P_{M4} = b_5$$
$$\beta'_8 \delta P_8 - \beta_6 \delta P_6 - \beta_7 \delta P_7 - \beta_{M1} \delta P_{M1} = b_8$$

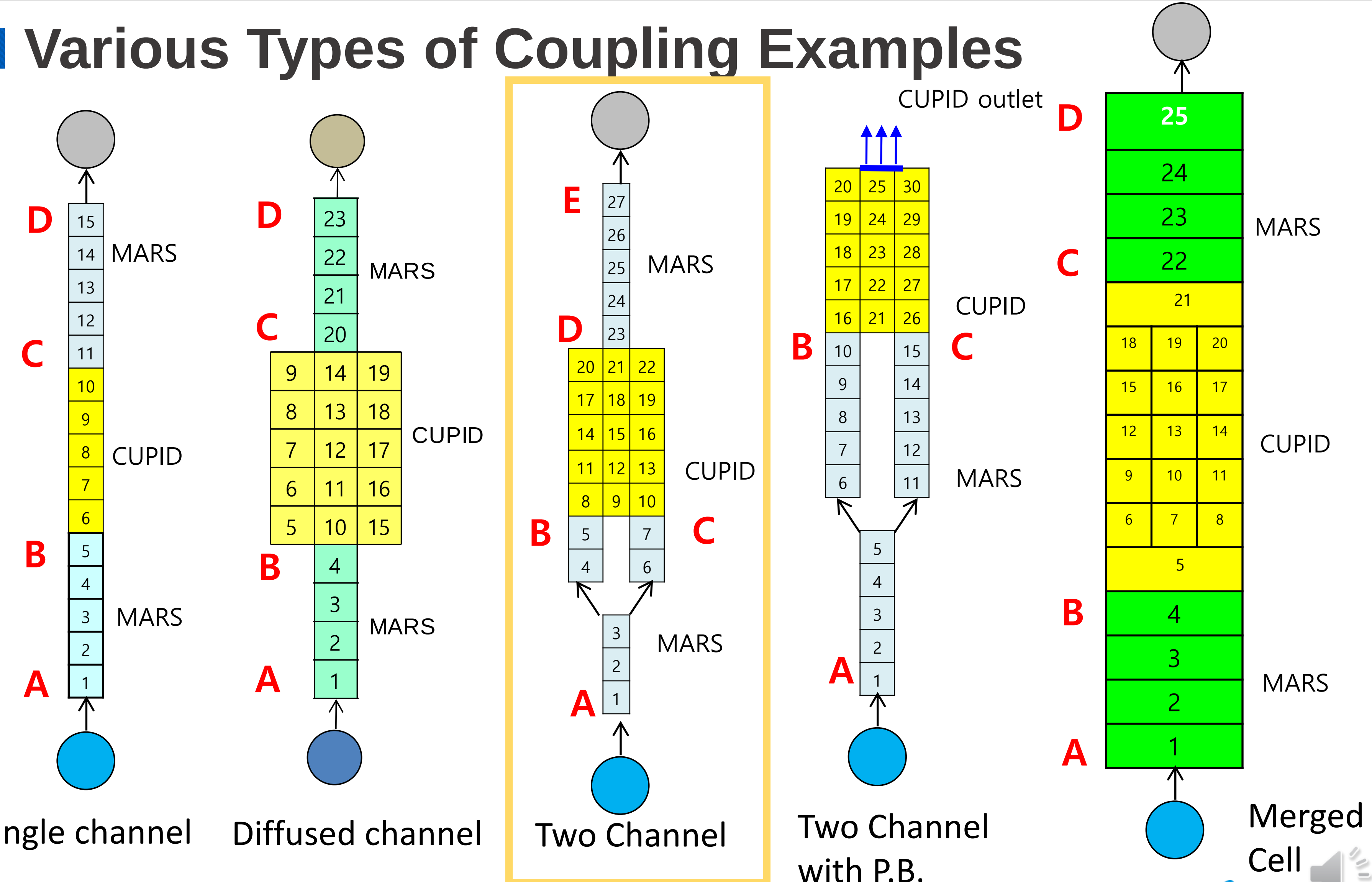
$$\beta'_5 \delta P_5 - \beta_6 \delta P_6 - \beta_7 \delta P_7 + \beta_{M5} \left(\eta_{1,1} \delta P_{C5} + \eta_{1,2} \delta P_{C8} \right) = b'_1$$
$$\beta'_8 \delta P_8 - \beta_6 \delta P_6 - \beta_7 \delta P_7 + \beta_{M1} \left(\eta_{2,2} \delta P_{C8} + \eta_{2,1} \delta P_{C5} \right) = b'_{10}$$



CUPID

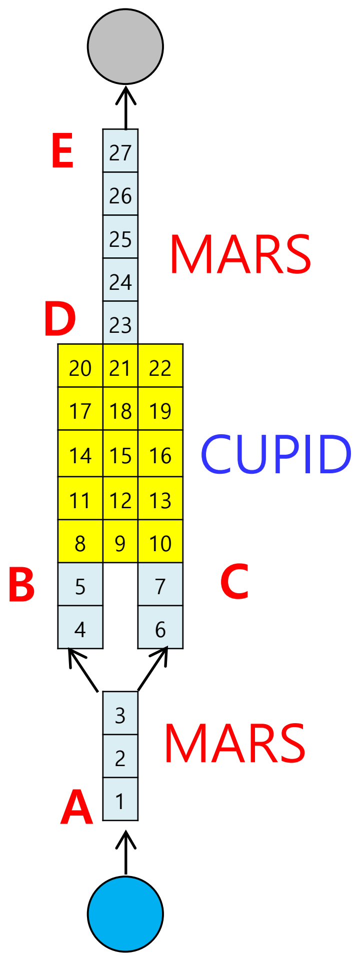
Verification of Implicit Flow Coupling (1/3)

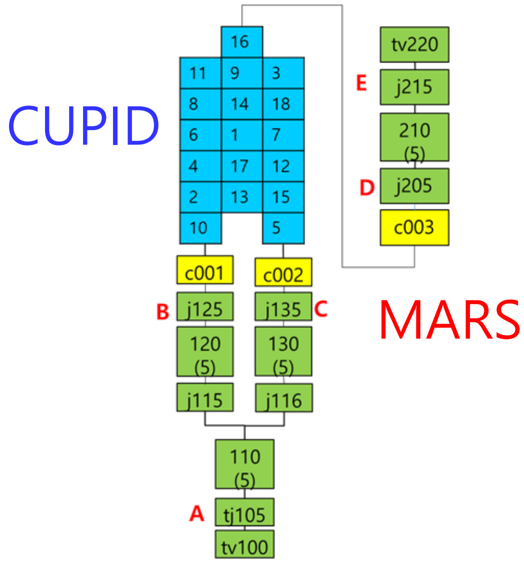
Various Types of Coupling Examples



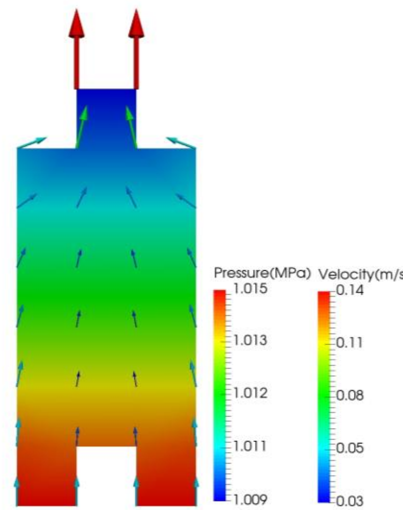
Verification of Implicit Flow Coupling (2/3)

Two-Channel Coupling Example

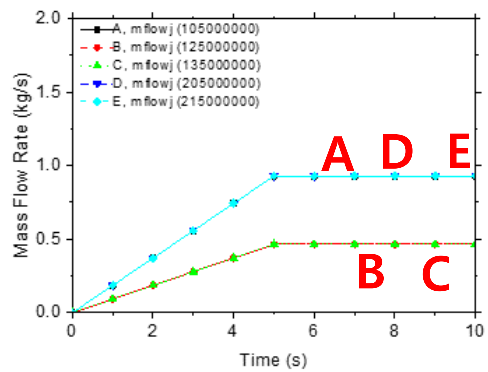




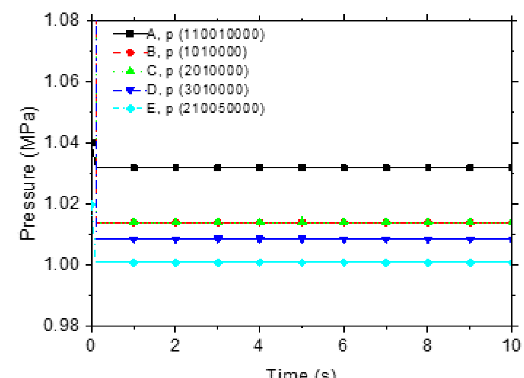
Calculation Mesh



Pressure contour & Velocity vector



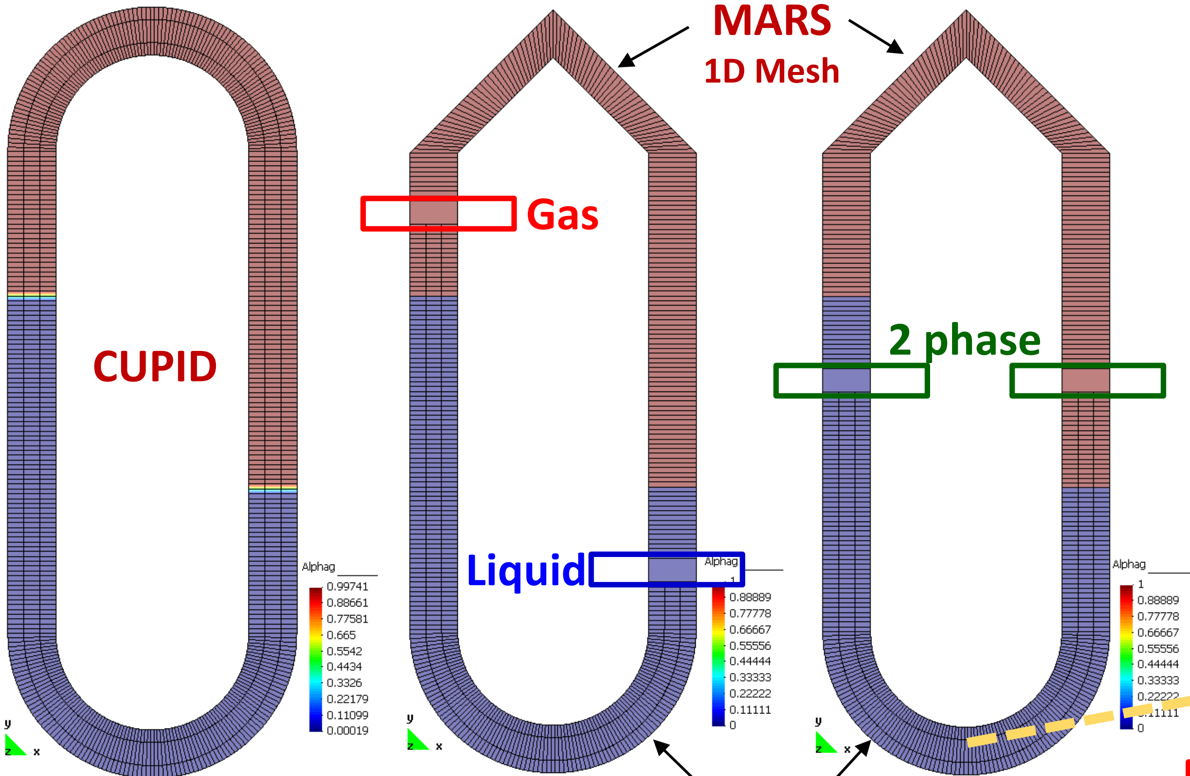
Mass flow rate at A,B,C,D,E



Pressure at A,B,C,D,E

Verification of Implicit Flow Coupling (3/3)

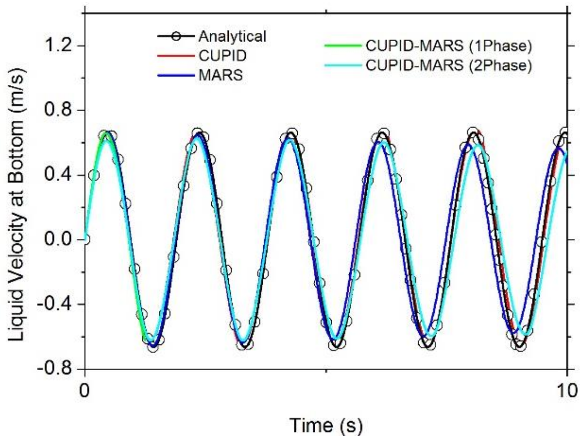
» Liquid Column Oscillations in O-tube



CUPID Standalone with 2D Mesh

CUPID-MARS (1phase coupling)

CUPID-MARS (2phase coupling)



<Comparison of Liquid Velocities at the Tube Bottom>

Analytical Solution of Liquid Velocity

Gas phase coupling mesh

Liquid phase coupling mesh

2 phase coupling mesh

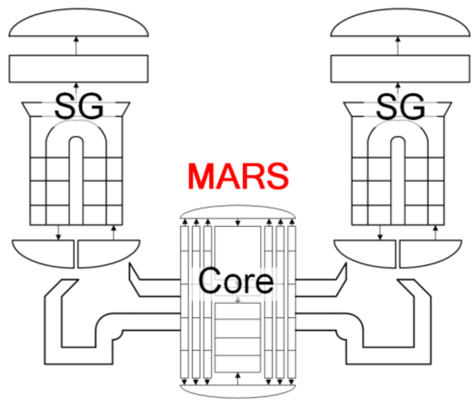
Nuclear Reactor Application

3

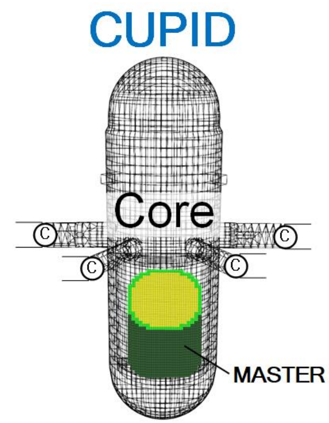
– Calculation Procedure for the Coupled Code

– Coupled Analysis of PWR MSLB

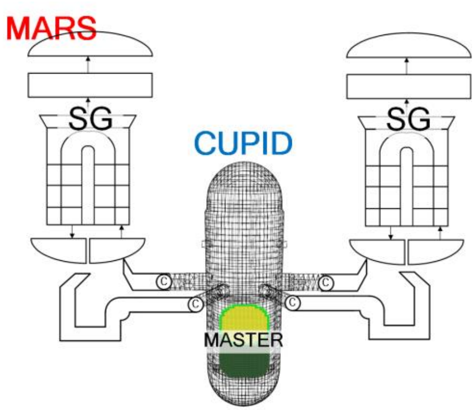
Calculation Procedure for the Coupled Code



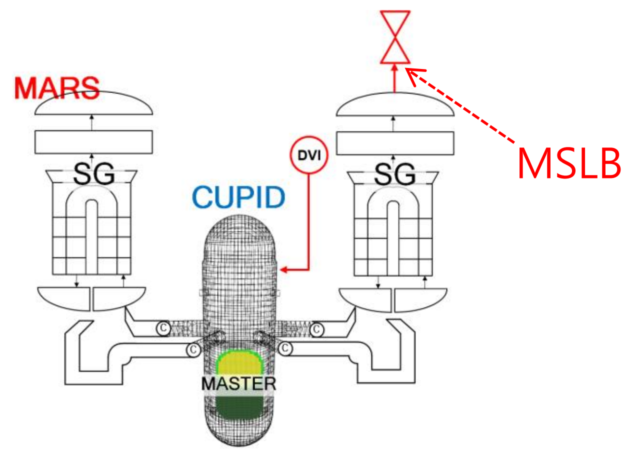
① 1D Reactor System *Steady*



② 3D RPV *Steady*

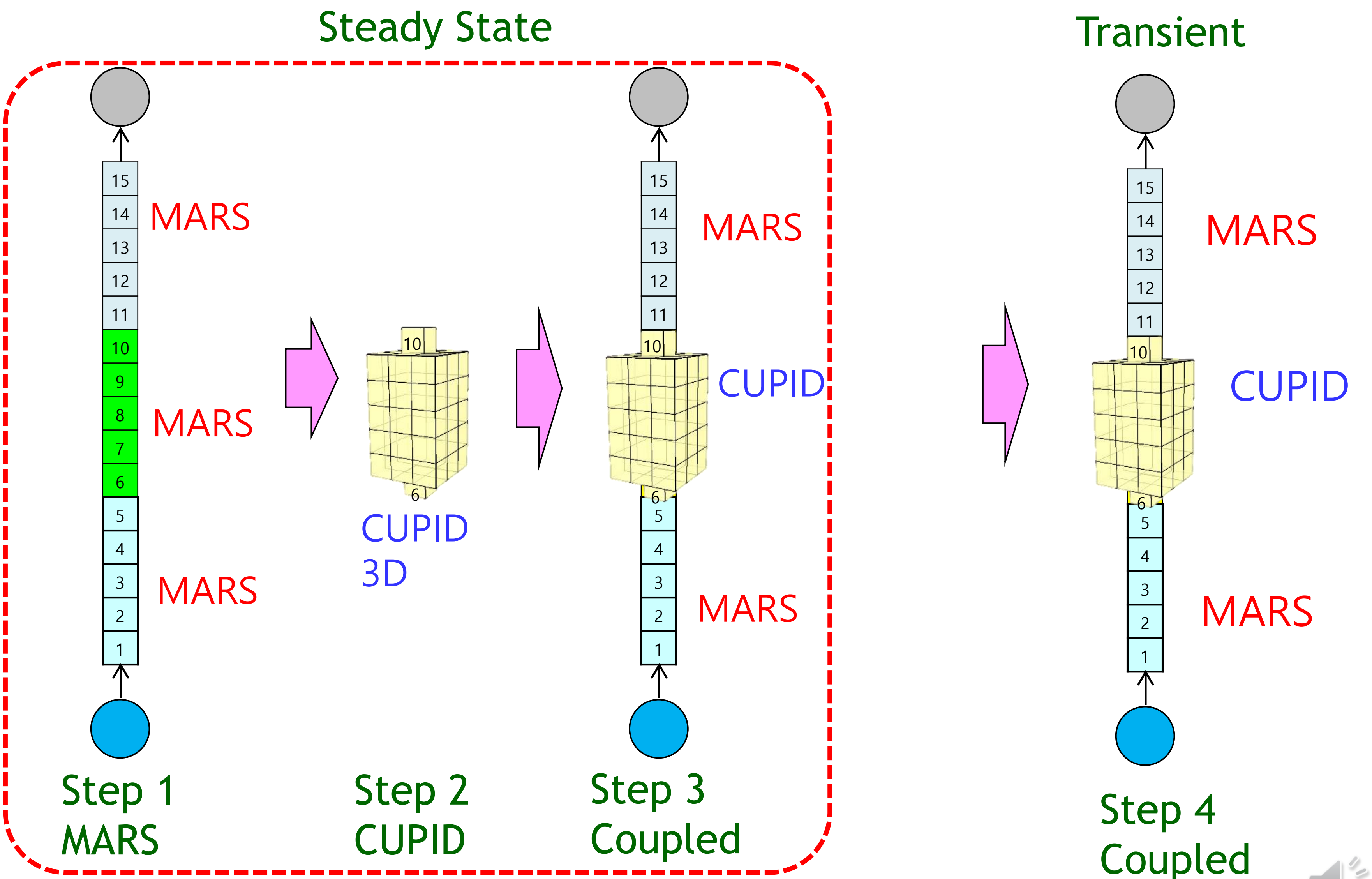


③ 1D/3D Coupled *Steady*



④ 1D/3D Coupled *Transient*

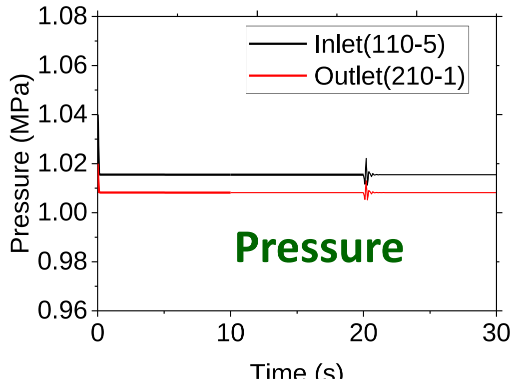
Verification of the Calculation Procedure (1/2)

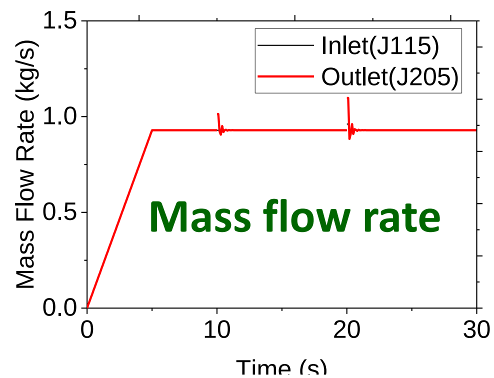


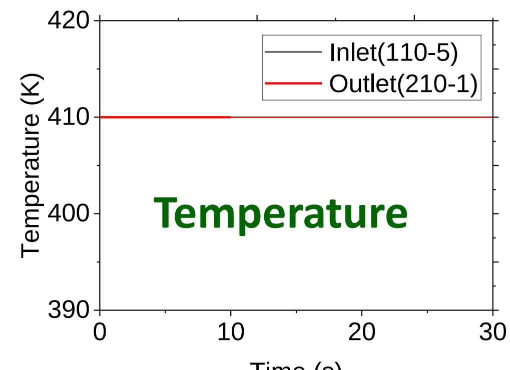
Verification of the Calculation Procedure (2/2)

Calculation Results

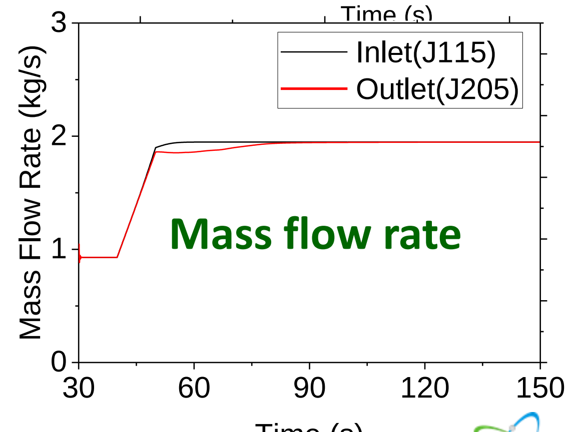
- **Steady state calculations (Step 1,2,3: 10s,20s,30s) provide consistent pressure, mass flow rates, and temperatures.**
  - Coupled steady state can be achieved quickly by combining 1D and 3D standalone steady state .



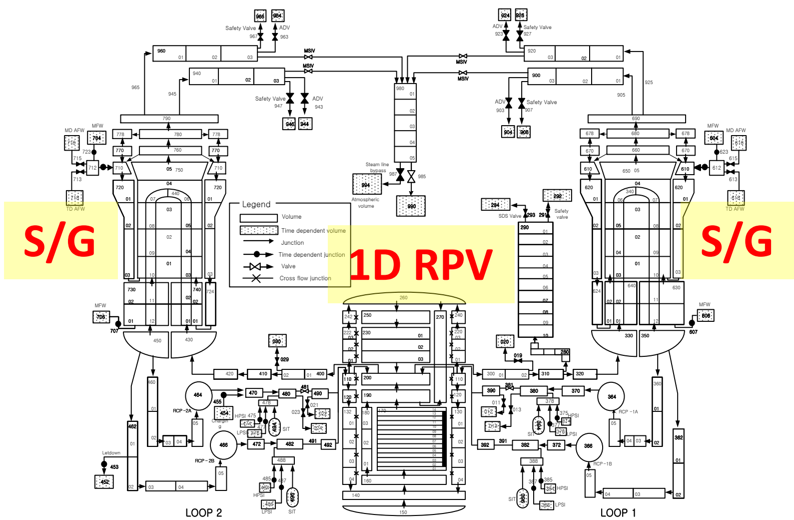




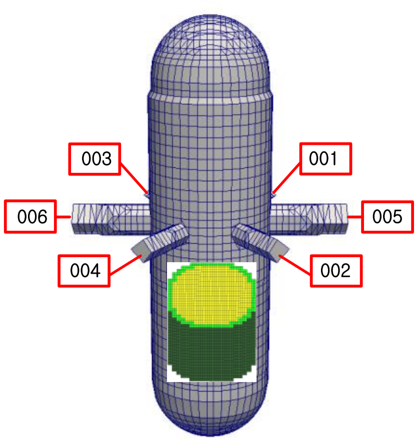
- **Transient calculation(Step 4) can be done using the coupled steady state.**
  - Nuclear safety analysis can be conducted with this suggested procedure.



Coupled Analysis of PWR MSLB (1/2)

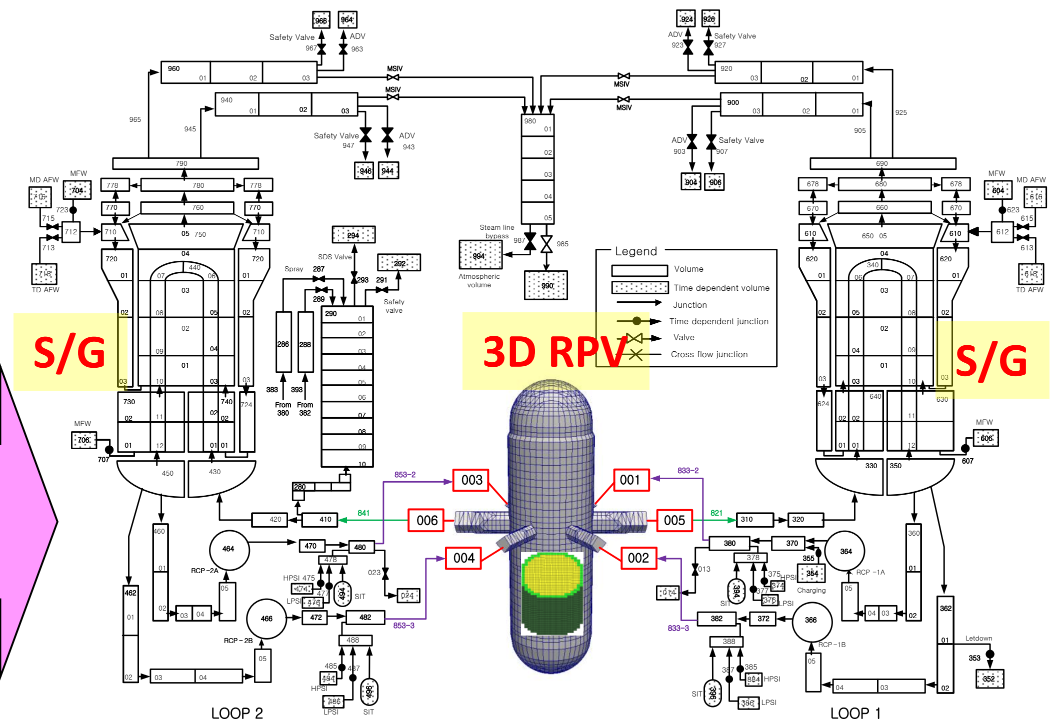


Step 1: 1D Mesh for Reactor System



Step 2: 3D Mesh for 3D RPV

» **Coupling of 1D and 3D meshes for the OPR1000 MSLB Accident Analysis**



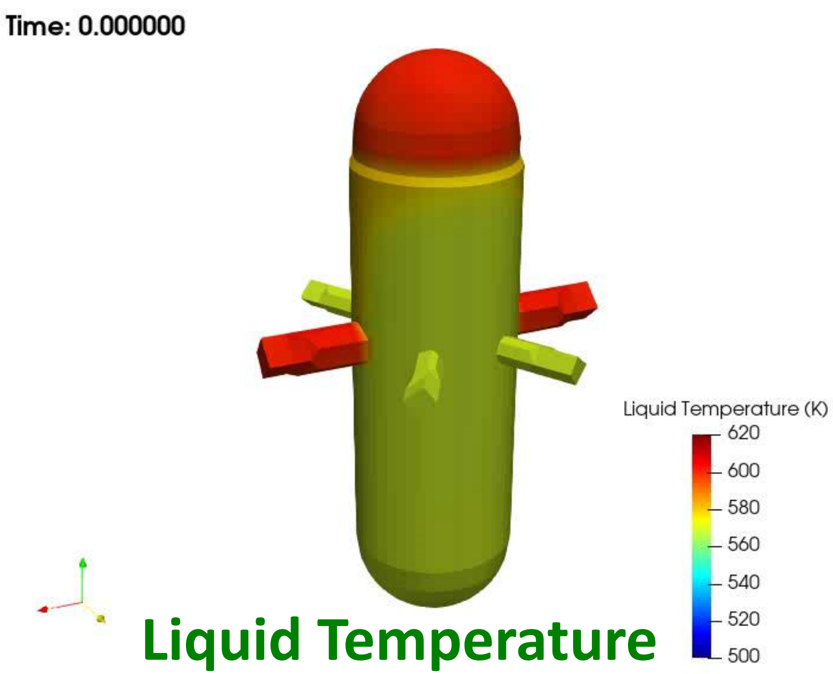
Step 3,4: Multi-scale Mesh for a 1D/3D Coupled Reactor System

Coupled Analysis of PWR MSLB (2/2)

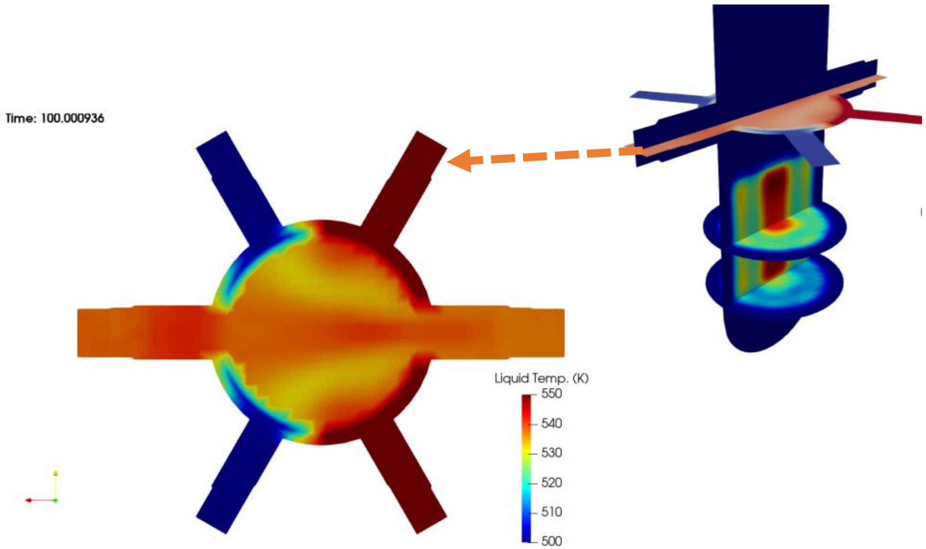
Verification of Coupled Safety Analysis Method

➢ Set up PWR MSLB calculation set in assembly-scale

- Confirm the proper simulation of MSLB including SLB major feature like asymmetric coolant temperature.
- Computation times is 3600s for 21055 cells, 8 cores, 100s transient.
- CUPID/MARS is efficient and practical enough for safety analysis.



Liquid Temperature



Liquid Temperature at Hot Leg Level

Summary

4

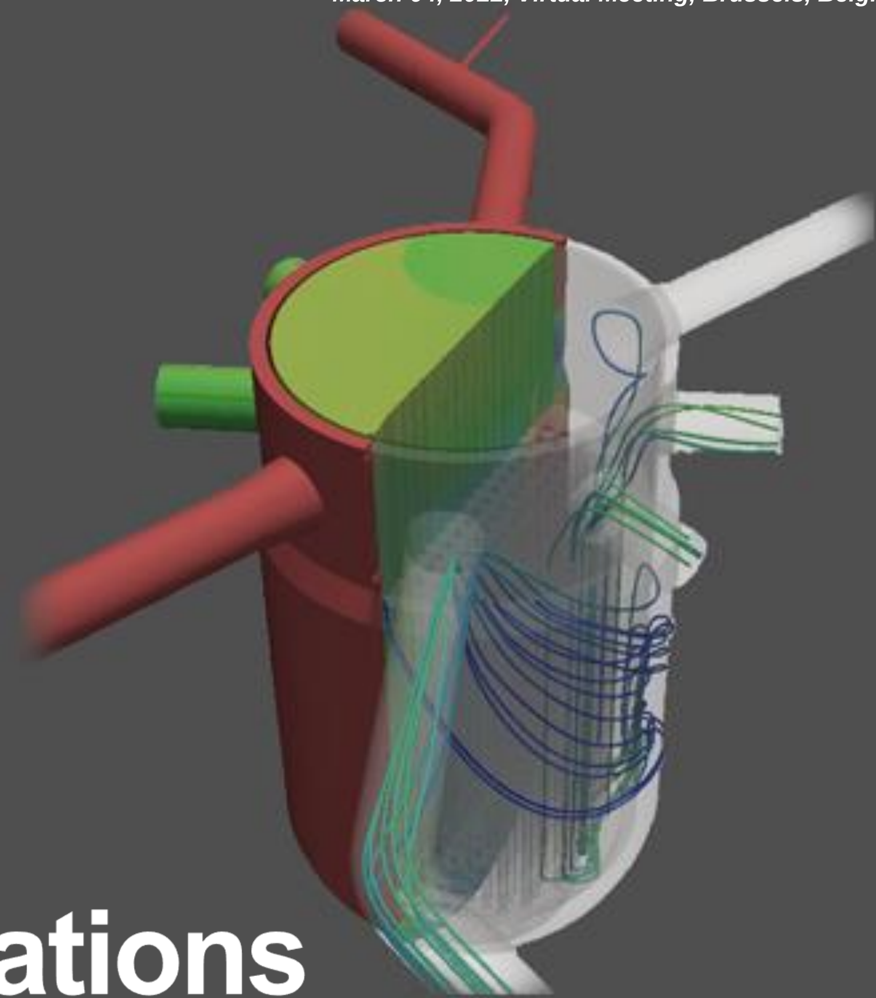
Summary

- Reactor system analysis code, MARS, is coupled with, 3D reactor vessel T/H code, CUPID-RV, implicitly in a single domain
  - No needs to transfer data and to iterate each solver
  - Fast and robust calculation of a transient
- The coupled CUPID/MARS code was verified using the various types of mesh coupling
  - Arbitrary types of coupling interfaces are allowed
- CUPID/MARS was successfully applied to the PWR MSLB accident analysis with a practical computation time
  - Calculation time of CUPID/MARS was 3600 s at the assembly-scale (21055 cells, 8 cores, 100s transient)

THANK YOU

gosu@kaeri.re.kr

Korea Atomic Energy Research Institute
KAERI
한국원자력연구원



CUPID Workshop
CFD Scale Applications

Yun-Je Cho
March 04, 2022

Korea Atomic Energy Research Institute · 한국원자력연구원 · KAERI

CUPID Workshop

CONTENTS

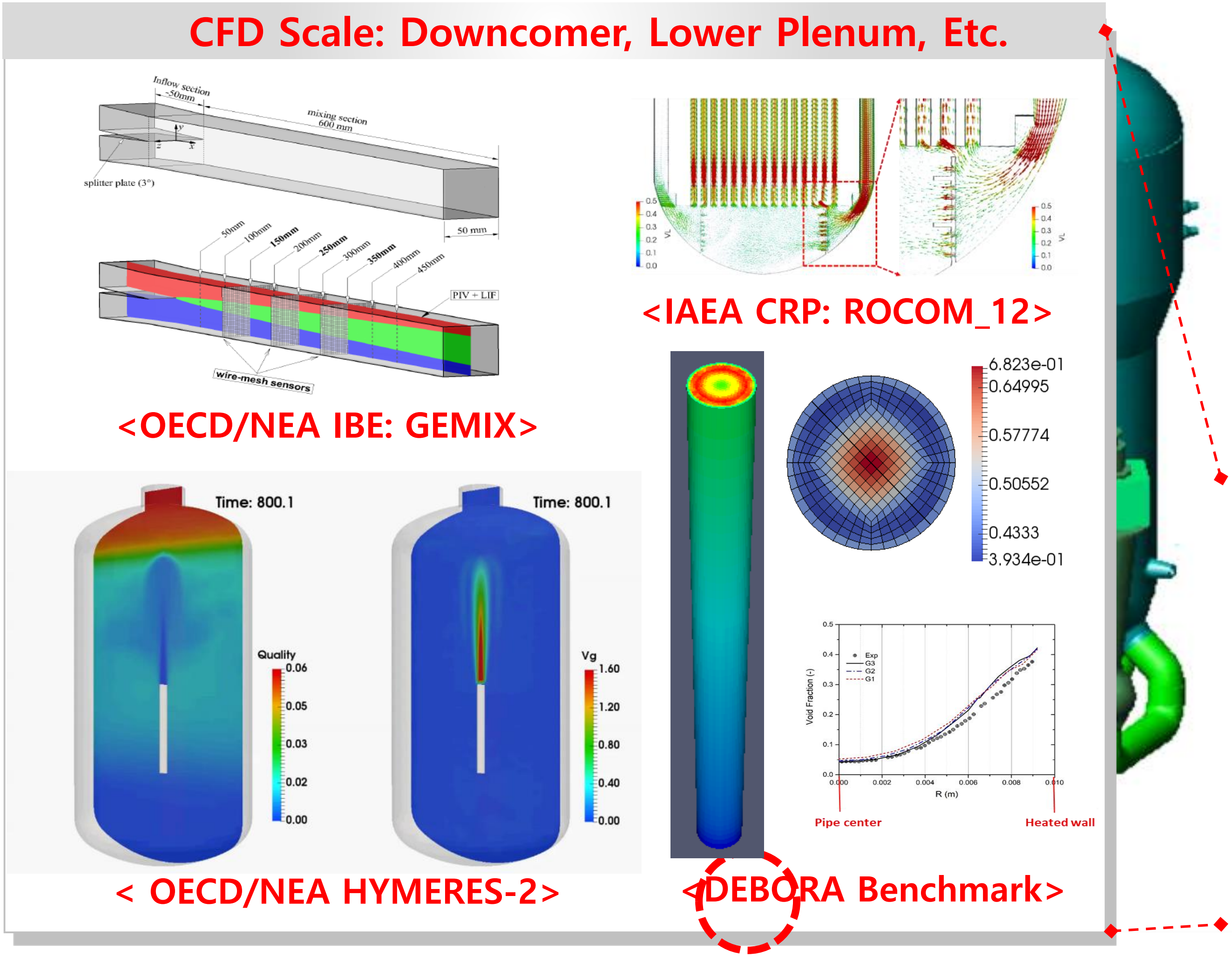
International CFD Benchmark

1

- CFD Applications using CUPID
- International Benchmark

Introduction

CFD-Scale Applications using CUPID



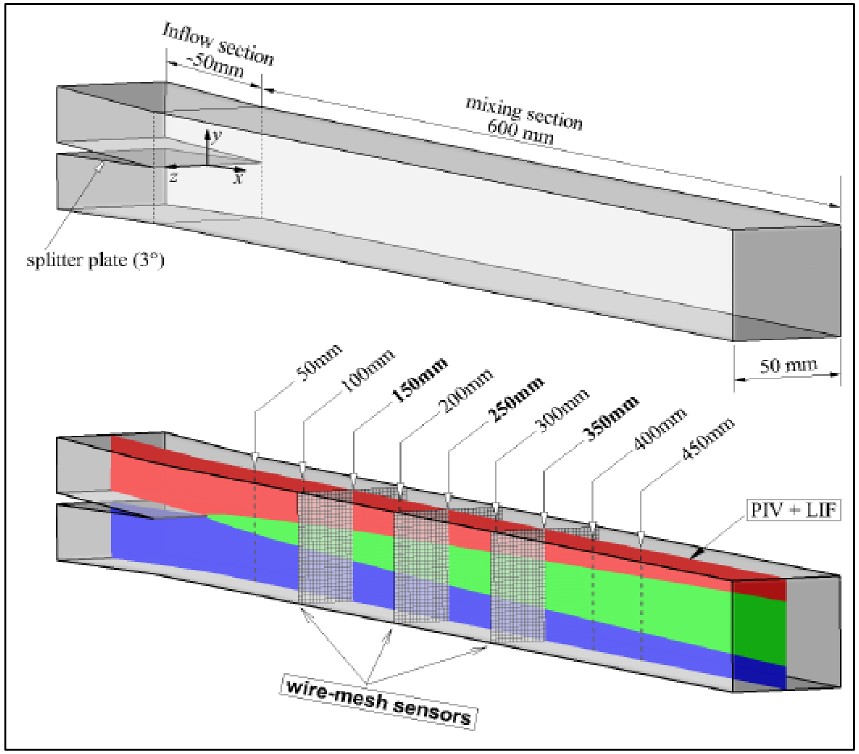
OECD/NEA IBE-4 (GEMIX)

2

- IBE-4 GEMIX
- Calculation Results
- Synthesis Report

OECD/NEA Benchmark (1/3)

IBE-4: GEMIX (2016)



Inlet velocity	0.6 m/s	1.0 m/s
Global Re	30000	50000
Δρ=0%, ΔT=0K	N339	N337
Δρ=1%, ΔT=5K	N320	**N318**
	Open	**Blind**

13 submissions

CFD-CODE

ANSYS (CFX)	3
ANSYS (FLUENT)	2
STAR-CCM	2
Code_Saturne	2
CUPID	1
TrioCFD	1
P2REMICS	1
OpenFOAM	1

TURBULENCE MODEL

k-eps	7	**CUPID**
k-omega	4	
LES	2	
RSM	1	

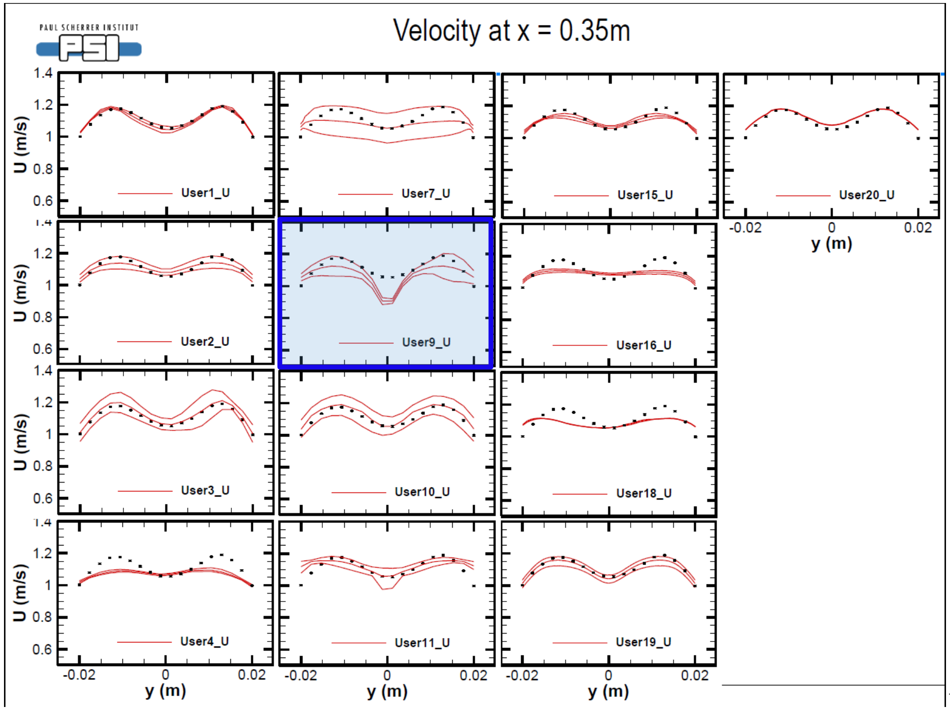
Number of grid

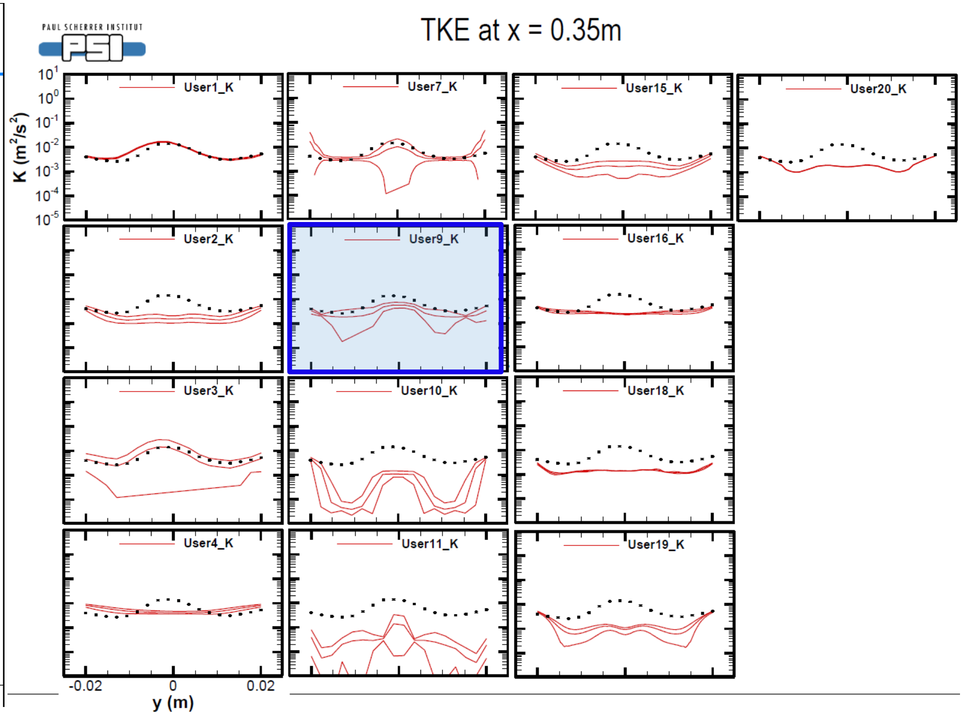
	59850 **(2D)**	← *minimum*
CUPID	156260	
	300000	
	20644596	← *maximum*
	3696000	
	5753458	
	3596992	
	675168	
	712704	
	62418 **(2D)**	
	1637784	
	652320	
	813276	

OECD/NEA Benchmark (2/3)

Calculation Results

- Velocity
- Turbulent kinetic energy





OECD/NEA Benchmark (3/3)

Final Result

➢ Thickness of mixing layer

	user	FoM	Ranking
	1	0.031418	2
	2	0.186935	7
	3	0.02999	1
	4	0.476981	13
CUPID	7	0.156025	6
	9	0.09458	5
	10	0.069638	4
	11	0.274574	11
	15	0.203641	8
	16	0.215331	10
	18	0.294725	12
	19	0.033593	3
	20	0.213123	9

➢ Turbulent kinetic energy

	user	FoM	Ranking
	1	3.867411	2
	2	6.002381	6
	3	3.463244	1
	4	6.352827	7
CUPID	7	4.662649	4
	9	4.86622	5
	10	11.21324	12
	11	12.31548	13
	15	6.523958	8
	16	4.401786	3
	18	10.81458	11
	19	7.109226	9
	20	9.406994	10

IAEA CRP (ROCOM)

3

- IAEA CRP
- ROCOM Test
- Computational Setup
- Computational Mesh
- Calculation Results

IAEA CRP

Coordinate Research Project (CRP)

- **Title: Application of Computational Fluid Dynamics Codes for Nuclear Power Plant Design**
  - Purpose: to address the application of CFD computer codes to optimize the design of water cooled nuclear power plants
  - Period: February 2013 ~ October 2019
  - 16 participants : Canada/CNL, China/Jiao Tong University, France/CEA Grenoble, France/AREVA, France/EDF, Germany/HZDR, India/BARC, Italy/University of Pisa, Republic of Korea/KAERI, Russian Federation/GIDROPRESS, Russian Federation/VNIIAES, Switzerland/Goldsmith Transactions, USA/MIT, USA/Texas A&M University, Algeria/CNRB, and USA/Westinghouse
  - Four Benchmark problems: Boron Dilution, PTS, two rod bundle tests

Description of ROCOM (1/2)

ROCOM_12 Test (HZDR)

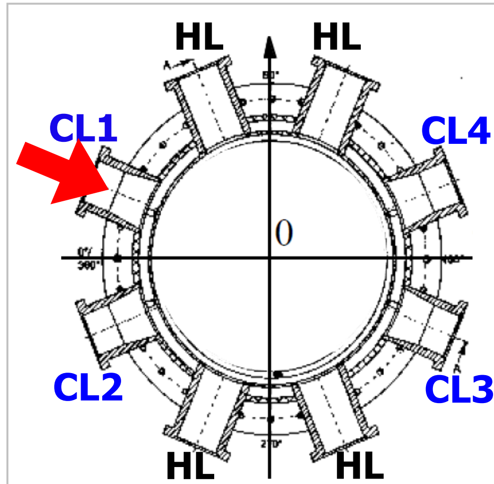
➢ Slug Mixing Experiments

- Prototype: German KONVOI reactor
- To simulate 'Boron dilution transients'
- Injection of water from one cold leg

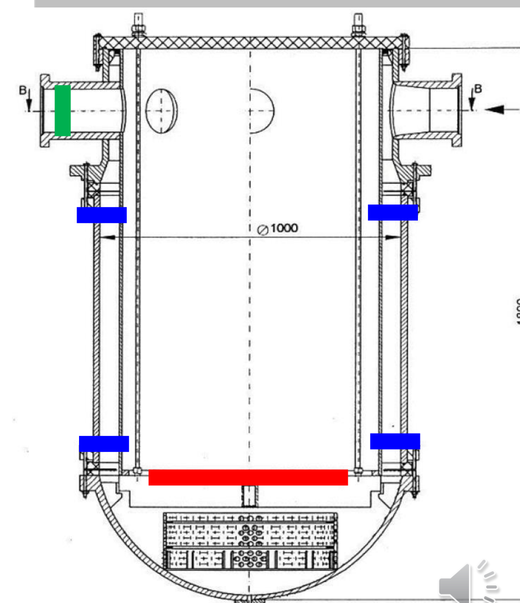
Ramp length	Volumetric flow rate	Slug volume (salted water)
14 s	185.0 m^3/h	8.0 m^3

- Wire mesh sensor: conductivity change

Core inlet plane (15x15)



<Schematic of ROCOM>



Model and Numerical Setup

Turbulence Models

- Standard k- ε model & Low Reynolds number model
- RNG k- ε model & Realizable k- ε model
- SST k- ω model

Boron Transport Equation

$$\frac{\partial}{\partial t}\left[(1-\alpha_g)\rho_l C_B\right]+\nabla\cdot(\alpha_l\rho_l C_B\vec{u}_l)+\nabla\cdot(\alpha_d\rho_l C_B\vec{u}_d)=0$$

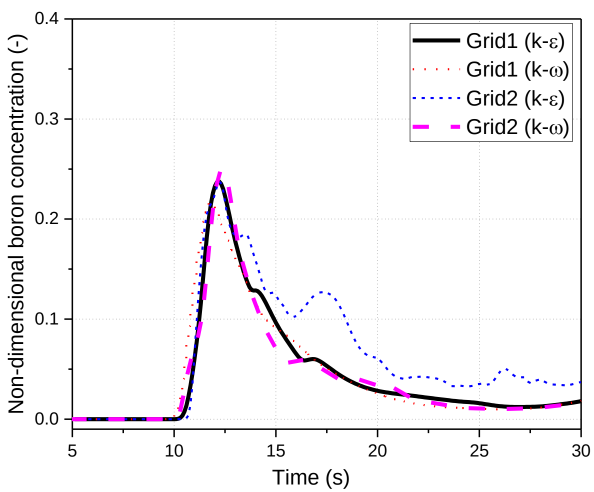
Baseline Calculation Case

Baseline case	Setup
Mesh	Reference grid
Turbulent model	Standard k-ε model
Convection scheme	2^{nd} order upwind
Solution Scheme	Implicit SMAC scheme

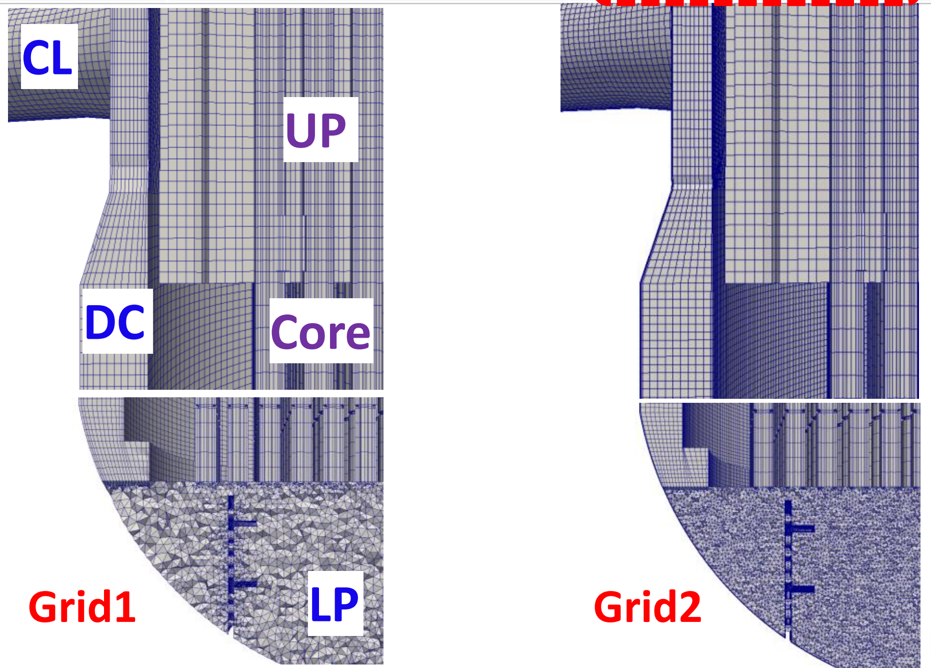
Mesh Sensitivity Test

Additional Grids for Sensitivity Test

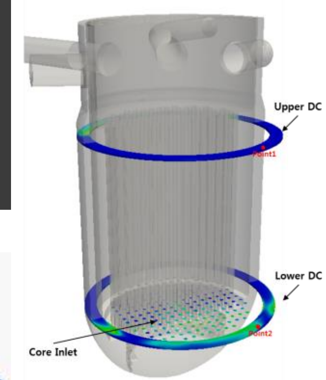
Grid	Core+UP	CL+DC+LP	Y+
Coarse	2.2M	1.15M	Y+>300
Grid1		2.45M	30<Y+<300
Grid2		10.21M	Y+<5

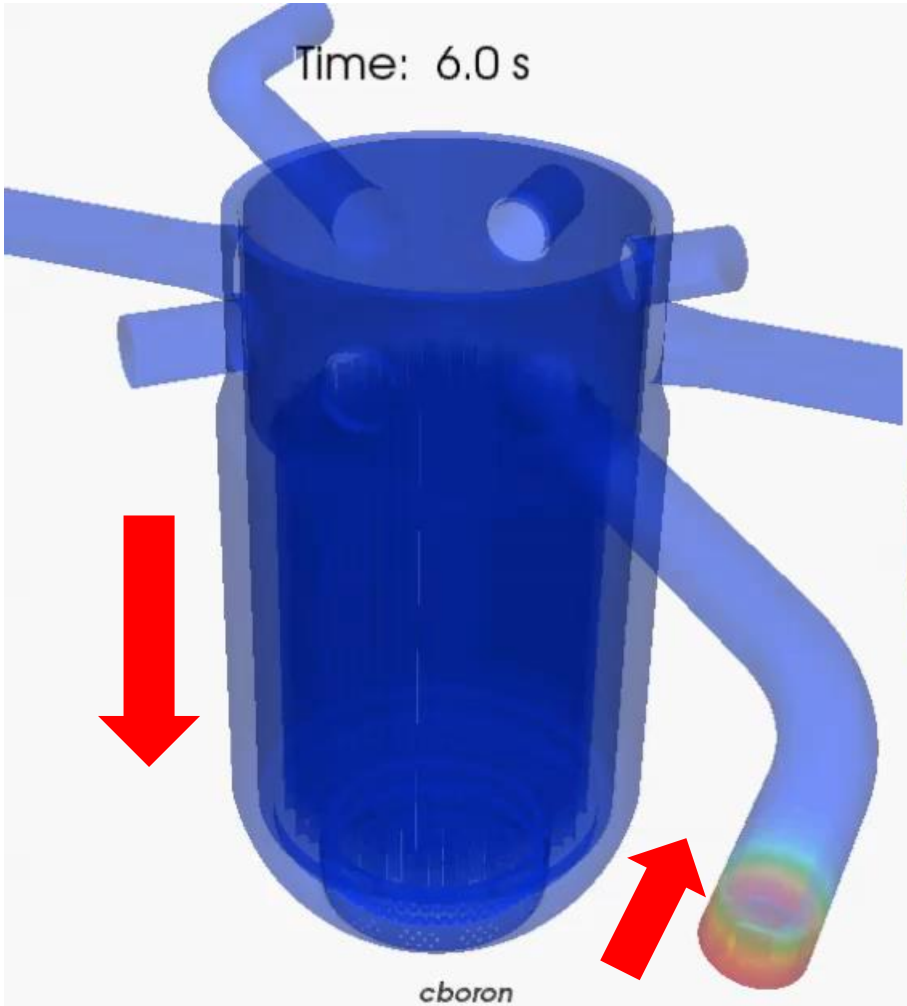


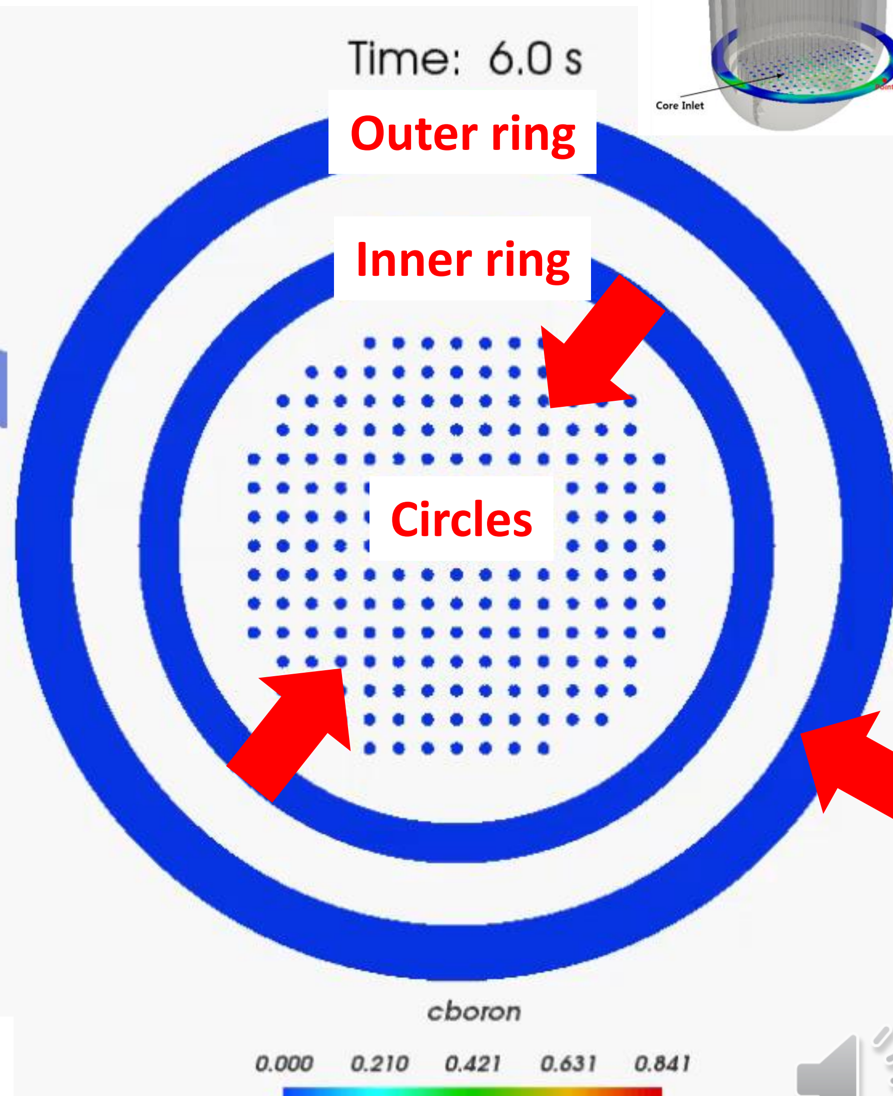
Convergence Test



Overall Mixing Behavior

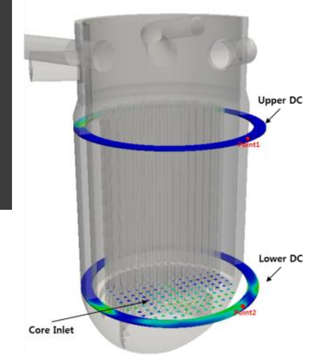




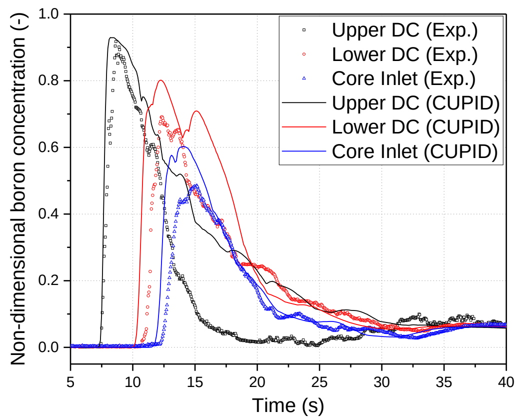


Non-dimensional boron concentration

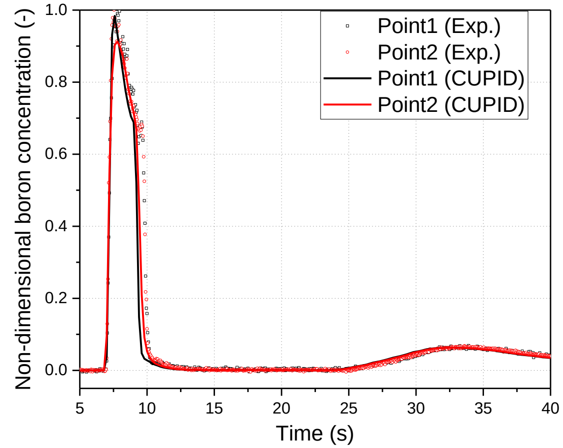
Quantitative Comparisons



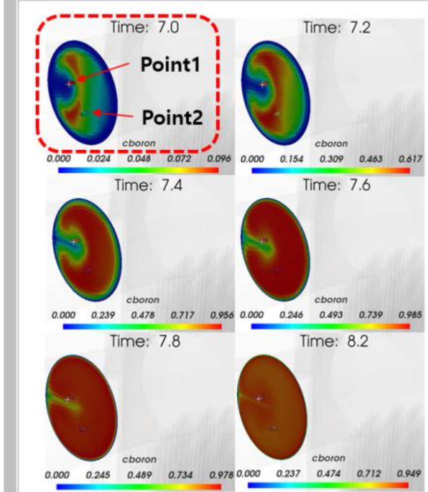
Averaged & Local concentration

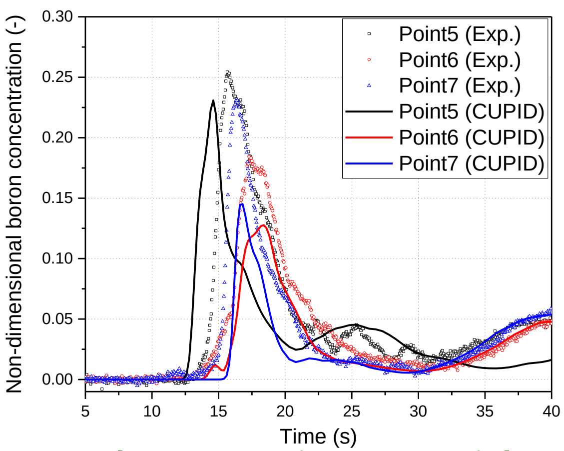


Maximum concentration at upper DC, Lower DC, Core inlet

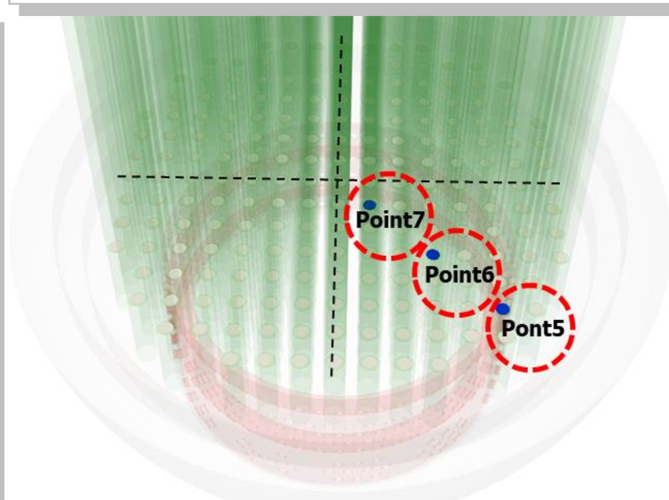


Local concentration at cold leg





Local concentration at core inlet



Synthesis Report (1/2)

IAEA CRP: Boron dilution benchmark

- Competition with CFX, Star-CCM$^+$, and OpenFOAM
- Use of different models and meshes
- Prediction of hydraulic resistance of perforated drum and turbulence mixing in downcomer

Participant	Code	Turbulence model	Mesh
HZDR	CFX 18	SST	6.5 M mixed cells
VNIIAES	Star-CCM+	Realizable k-ε with two layer wall model	Unknown number of mixed cells
BARC	OpenFOAM	One equation LES with delta cube root	8 M or 19 M of mixed cells
KAERI	CUPID 2.0	k-ε with Chen's low-Re number model	4.6 M mixed cells

Synthesis Report (N. Boyan, PSI)

Synthesis Report (2/2)

IAEA CRP: Boron dilution benchmark

- The first place in three of ranking system

PAUL SCHERRER INSTITUT PSI

Boron Dilution Benchmark

Overall ranking of contributions

Number of best predictions

1.	KAERI	5
2.	VNIIAES	3
=2.	BARC	3
4.	HZDR	2

Using a scoring system (3, 2, 1)

1.	KAERI	23
2.	VNIIAES	20
3.	BARC	18
4.	HZDR	17

Accumulated RMS of errors

1.	KAERI	1.12
2.	HZDR	1.13
3.	BARC	1.14
4.	VNIIAES	1.53

Synthesis Report (N. Boyan, PSI)

OECD/NEA HYMERES-2

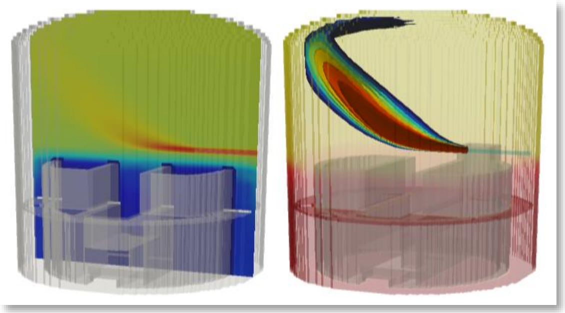
4

- HYMERES-2 Project
- PANDA Test
- Computational Setup
- Calculation Results

Overview of HYMERES-2

HYMERES-2

- **HYMERES-2 : 2017.01 – 2021.06 (HYMERES : 2013 - 2016)**
- **Main objective**
  - To improve the understanding of the containment phenomenology during postulated severe accident with release and distribution of hydrogen
- **Main topics of HYMERES-2**
  - Erosion of helium layer by steam jet/plume interacting with various obstruction geometries
  - Thermal radiation effects
  - Suppression pressure pool and BWR systems
  - Performance of safety components

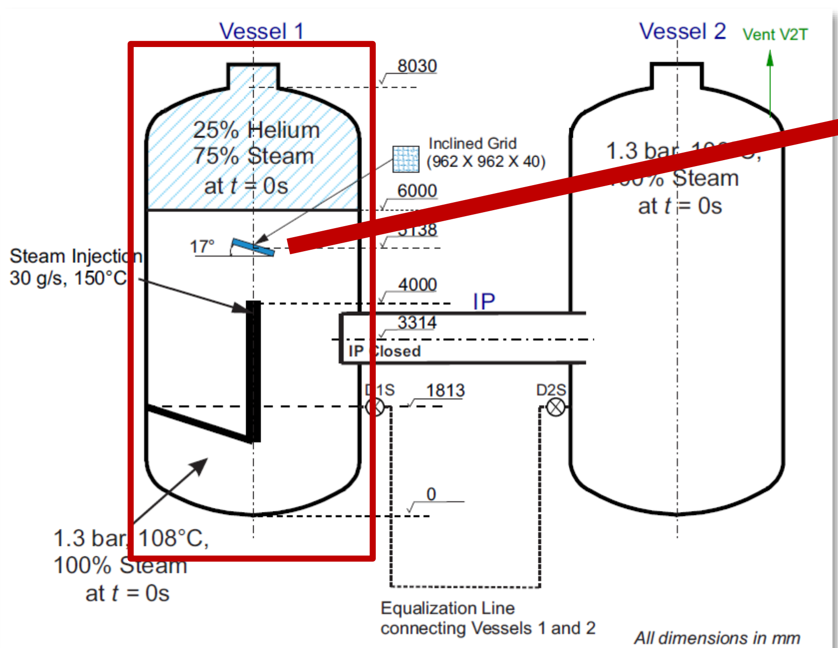


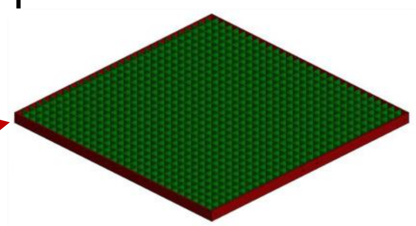
- **Experimental Data**
  - PANDA tests (PSI, Switzerland)
  - Open cases, 1 blind benchmark case

Description of PANDA Test

Blind Benchmark (H2P1_10)

- Given initial and boundary conditions **without test results**
- Erosion of **helium stratification** by vertically injected **steam jet**
- **Flow obstruction** (grid-shape) blocked the steam jet.
  - Inclined grid : 0.962m x 0.962m x 0.04m,
  - Installed at 5.138m, inclined 17° to horizontal plane





<Test configuration and nominal conditions>

Initial conditions (Nominal value)	
Steam mole fraction (< 6m)	1.0
Helium mole fraction (> 6m)	0.25
Wall temperature [℃]	108
Gas temperature [℃]	108
Boundary conditions (Nominal value)	
Pressure [bar]	1.3
Steam mass flow rate [g/s]	30
Steam temperature [℃]	150

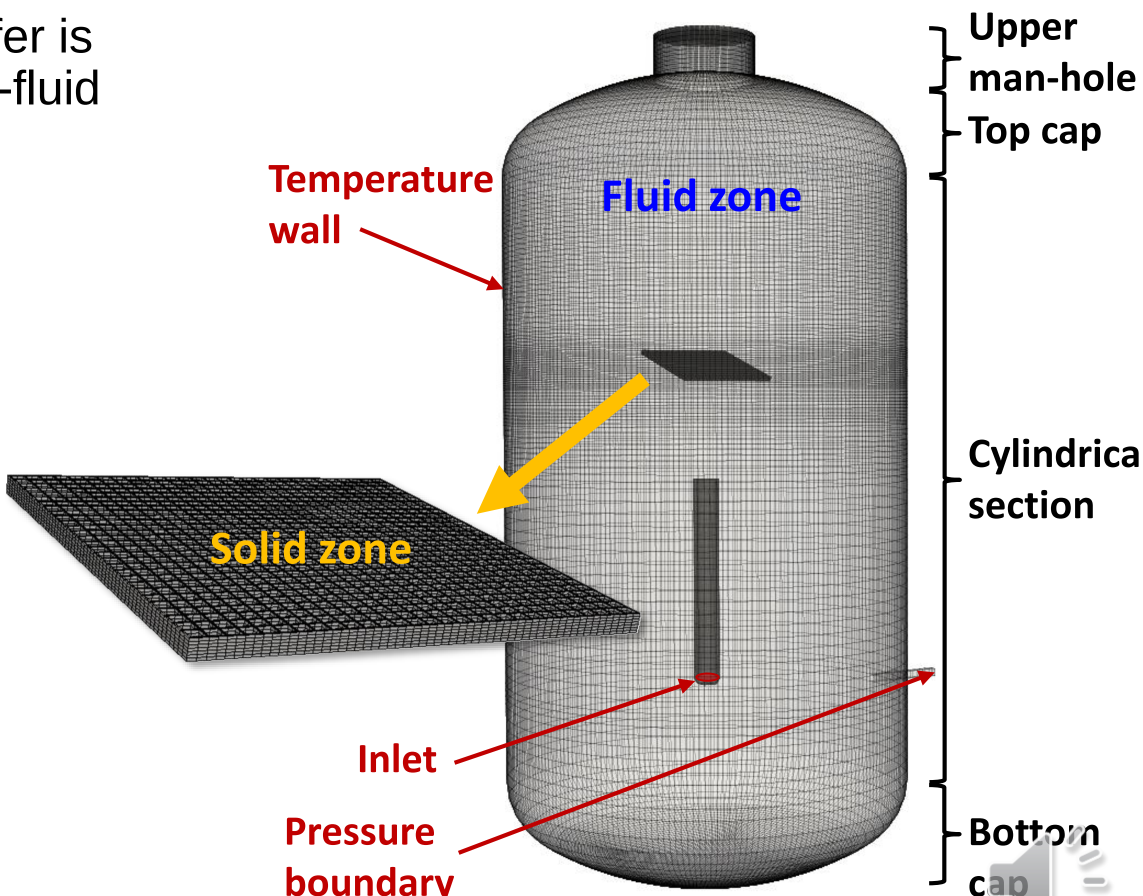
Computational Mesh

Configuration

- 3-D geometry of the vessel
- Flow obstruction : solid zone
  - Conjugated heat transfer is considered at the solid-fluid interface.

Computational mesh

- Hexahedron mesh
- 2.4M cells
  - Fluid zone : 2,290,376
  - Solid zone : 94,192

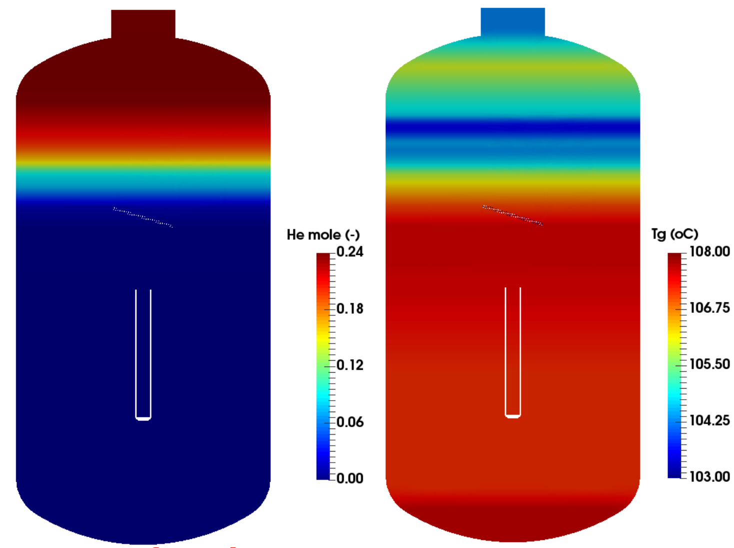


<Computational mesh>

Initial & Boundary Conditions

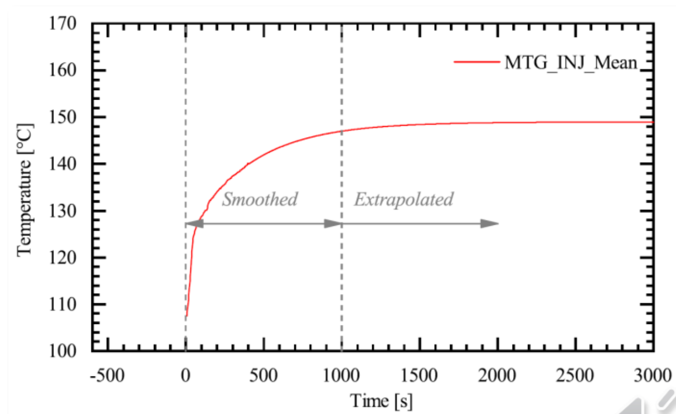
Initial Condition

- **Helium concentration : Experimental data along the height at central axis**
- **Gas temperature: Experimental data along the height at central axis**



Boundary Conditions

- **Inlet boundary**
  - Experimental data suggested as calculation requirements
  - Temperature of injected steam is increased 108℃ to 150℃ during 1000s
- **Wall boundary**
  - Followed calculation requirements
  - Lid : constant at 101℃
  - Cylindrical wall : decreasing as ~0.12K/100s



<Steam injection temperature>

Physical Models

Turbulence Model

- **Standard *k-ε* model with standard wall function**
- **Turbulence buoyancy effect is considered by adding buoyancy production term to source term of *k* and *ε* equation.**

$$G_k = -\vec{g}\frac{\mu_t}{\rho Pr_t}\nabla\rho$$: buoyancy term for *k*

$$G_\varepsilon = \frac{\varepsilon}{k}C_{\varepsilon 1}C_{\varepsilon 3}G_k$$: buoyancy term for *ε*

Radiative Heat Transfer

- **P-1 model is applied.**

- Transport equation of incident radiation (*G*)

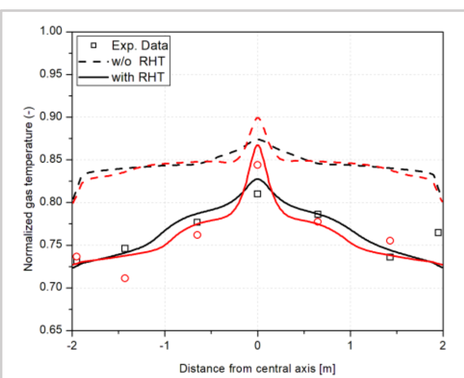
$$\nabla\cdot\left(\frac{1}{3(\kappa+\sigma_s)-A_1\sigma_s}\nabla G\right)-\kappa G+4\kappa\sigma T^4=0$$

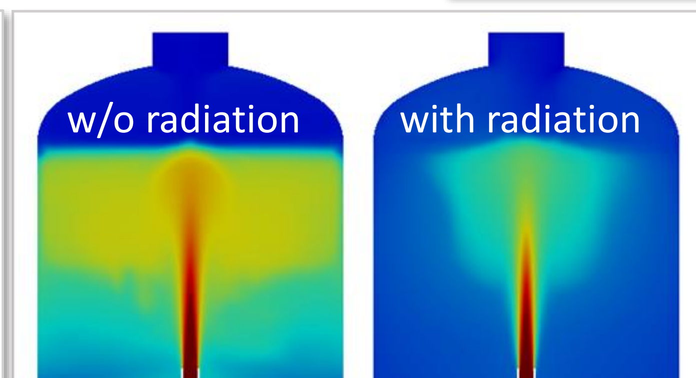
- Radiative heat flux

$$\vec{q}_{rad} = -\frac{1}{3(\kappa+\sigma_s)-A_1\sigma_s}\nabla G$$

$$\vec{q}_{rad,wall} = \frac{\varepsilon_w}{2(2-\varepsilon_w)}(4\sigma T_w^4 - G_w)$$

$$S_{rad} = -\nabla\cdot\vec{q}_{rad}$$ ← added to source term of gas-phase energy conservation equation





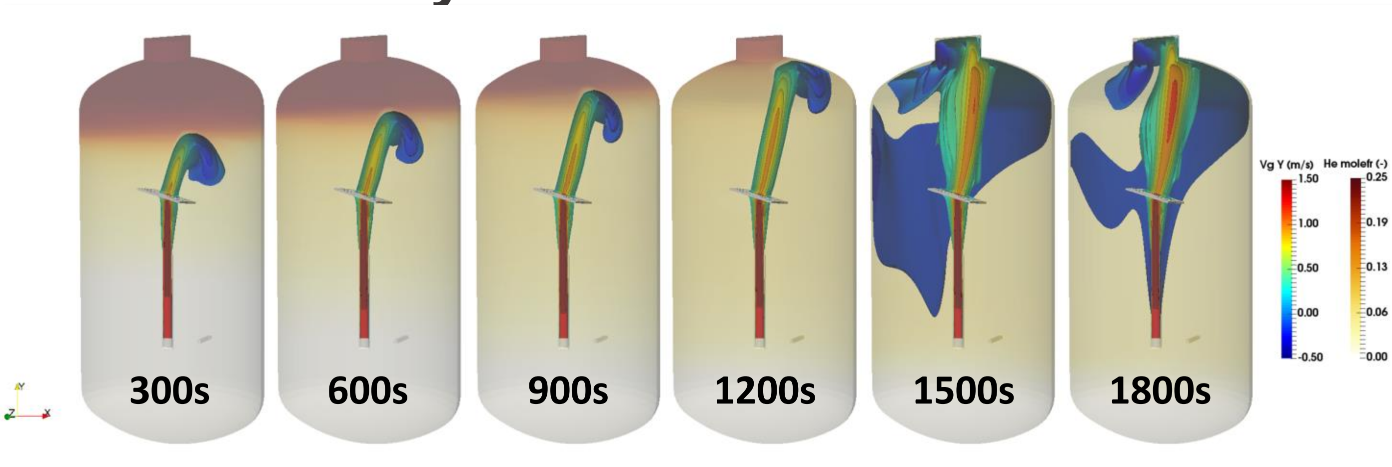
<Comparison of results with and without RHT model>

* Left : Gas temperature profiles

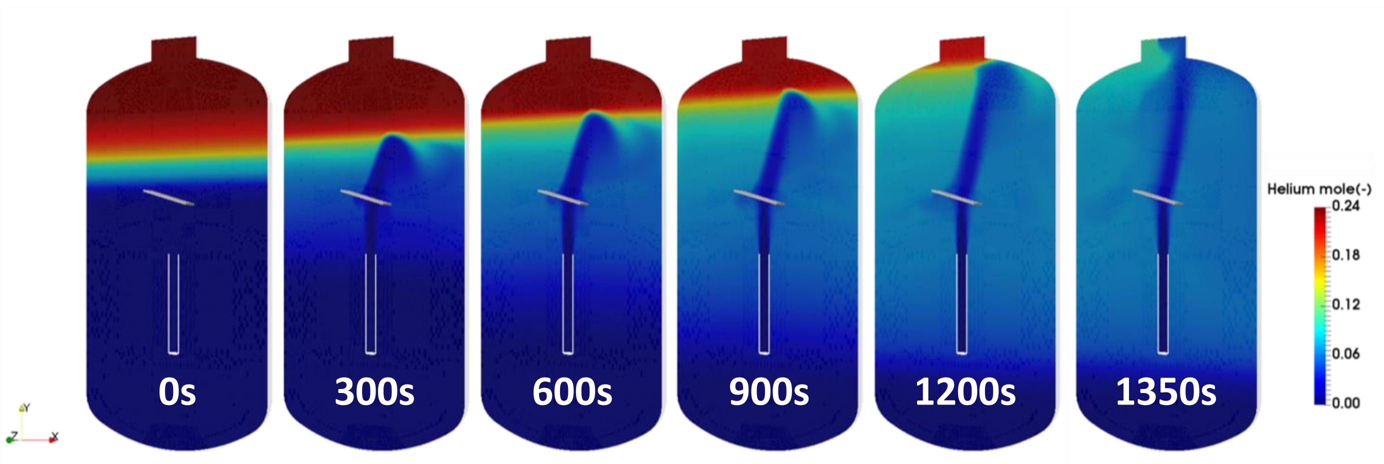
* Right : Distribution of gas temperature

Overall Behavior

Gas velocity



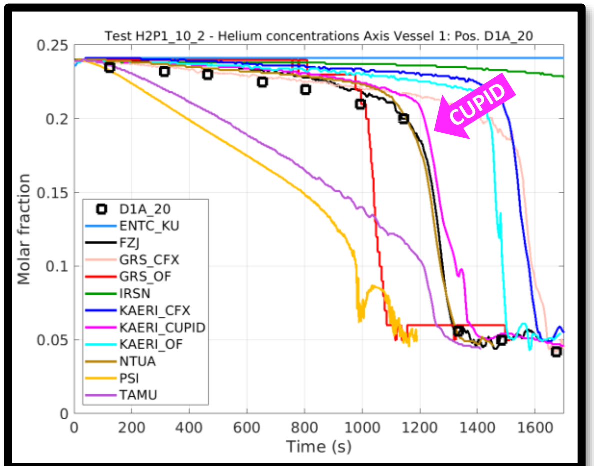
Helium concentration



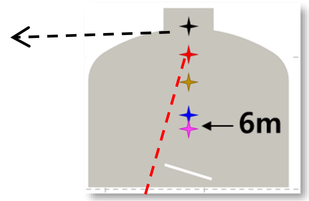
- The jet is inclined to the right in the figure due to the flow obstruction.
- As the jet rises, the stratified helium is gradually eroded.
- After the jet reached the top wall of the vessel, helium was distributed almost uniformly inside the vessel.

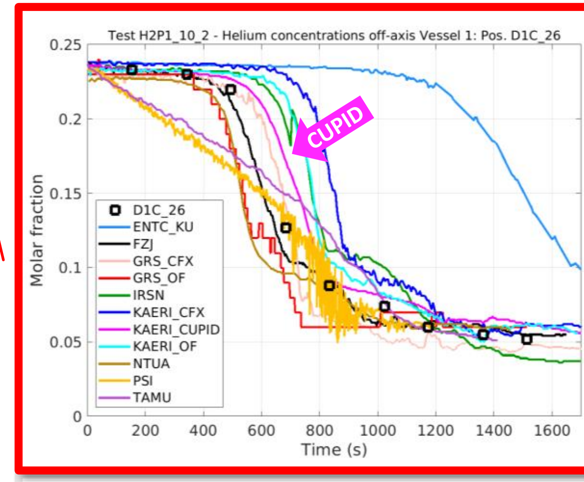
Helium Concentration

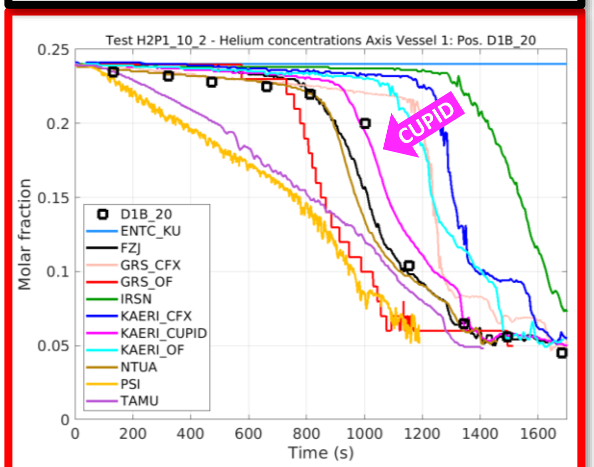
Evolution of helium concentration over time



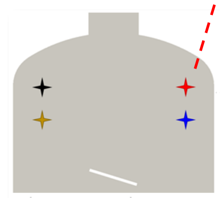
Central axis

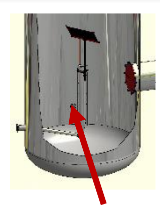




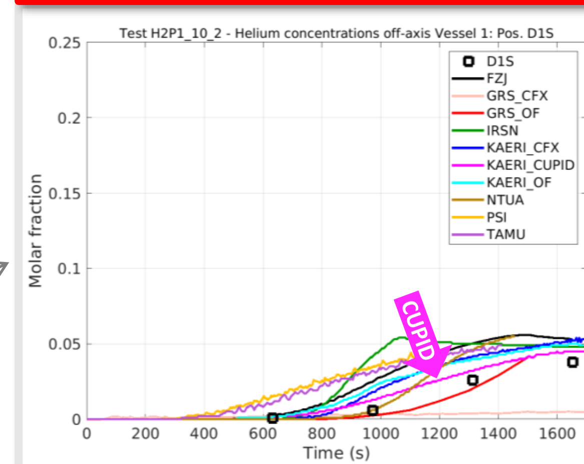


Off axis





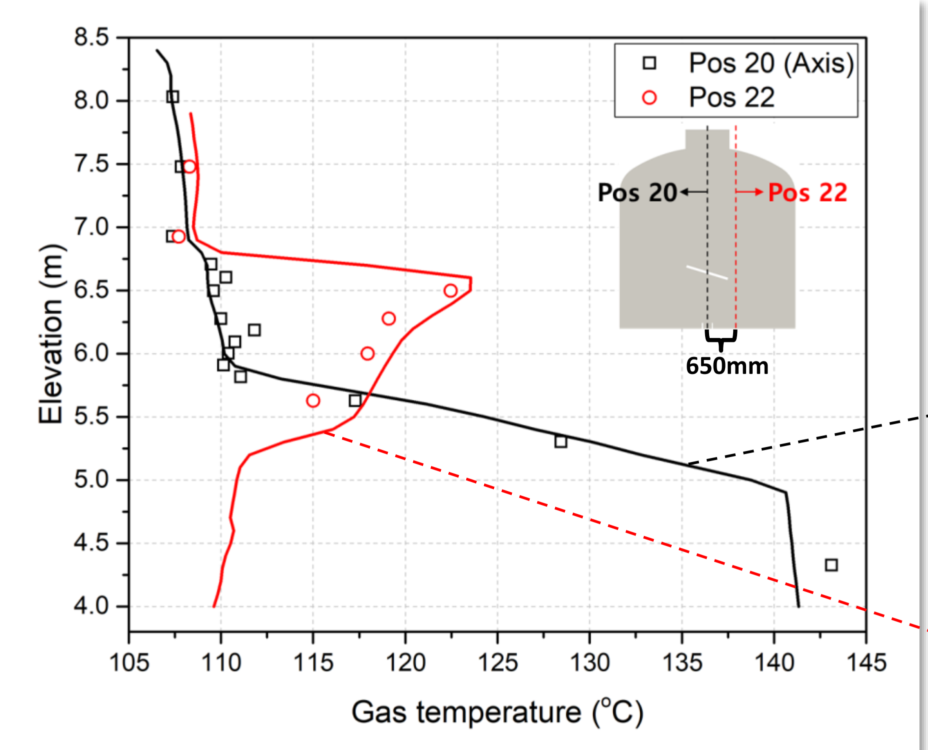
Equalisation line



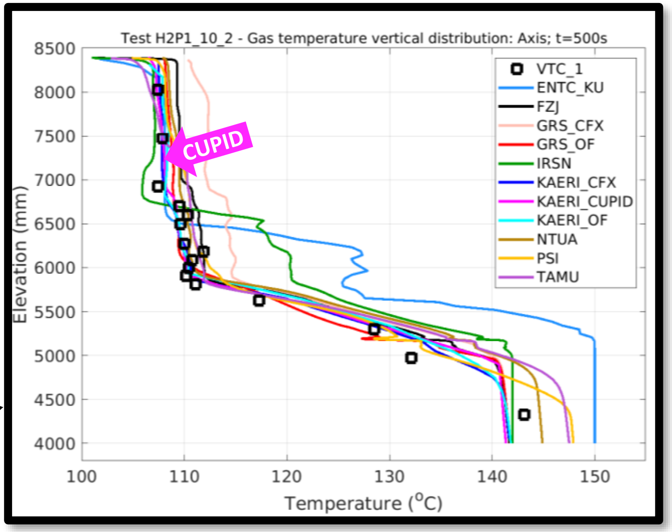
- CUPID predicted fairly well the experimental data in which the helium stratification was completely eroded at 1300 s.
- Overall calculation results of CUPID were excellent among the other results.

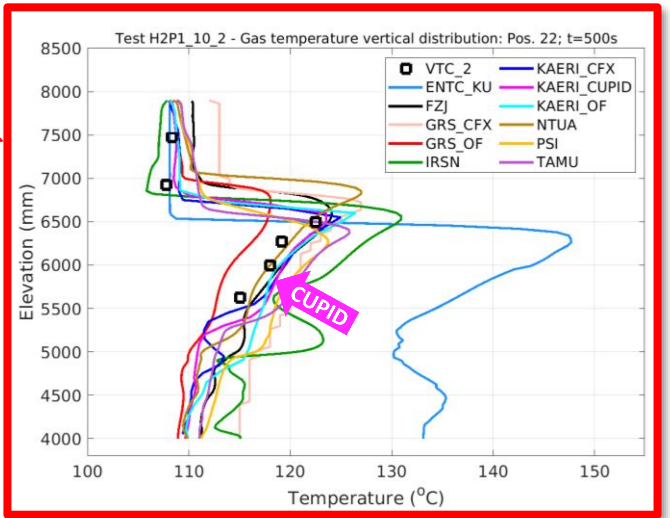
Gas Temperature

Distribution of gas temperature at specific time



Vertical temperature profiles (500s)





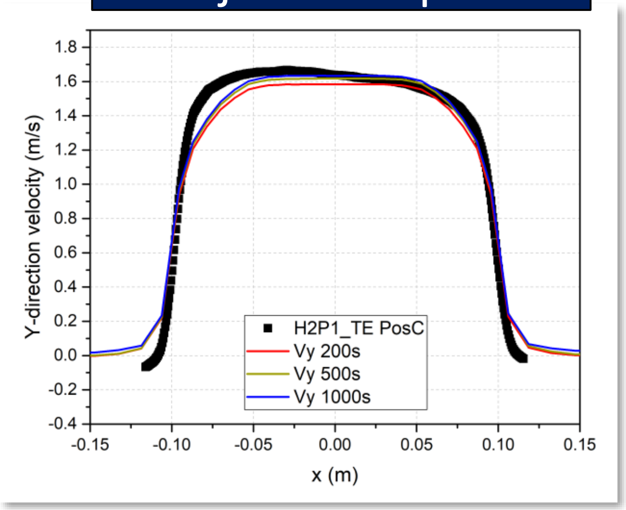
- The distribution of gas temperature in vertical direction agreed well with the experimental data.

Velocity Profile

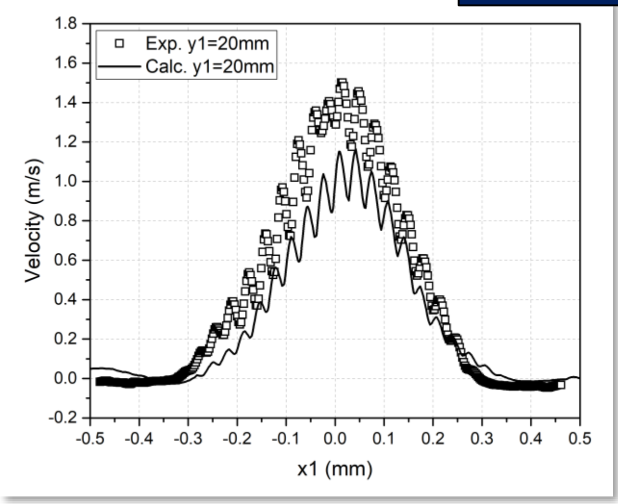
Distribution of Y-direction velocity at specific time

At Injection Pipe Exit

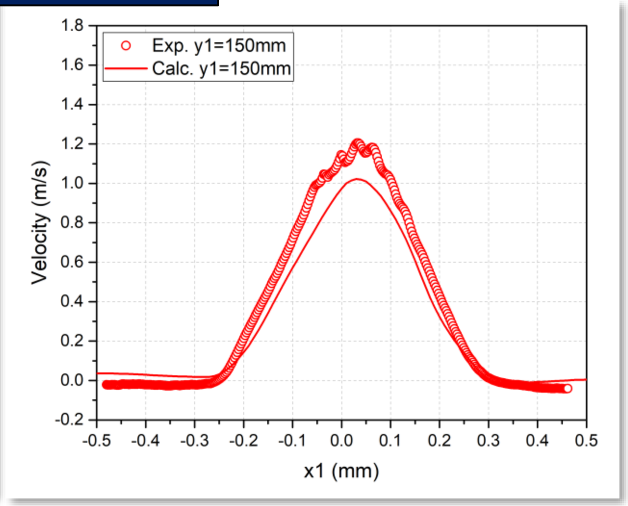
At PIV Area



<Time at 200s, 500s, 1000s>



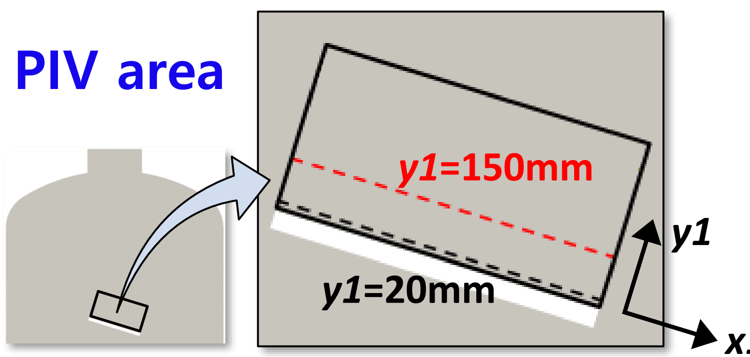
<Time at 846s, *y1*=20mm>



<Time at 846s, *y1*=150mm>

- Y-direction velocity at exit of injection pipe was calculated symmetrically and good agreement with experimental data.
- The velocity profile at *y1*=20mm, well reproduced the jagged shape.

PIV area



DEBORA BENCHMARK (CEA)

5

- Organization and Objectives
- Description of DEBORA Experiment
- Physical Models
- Calculation Results

Organization and Objectives

DEBORA Benchmark

- **Organized by CEA (France) and hosted by the Neptune project**
- **24 institutes from 15 countries confirmand their participating**
- **Main goals**
  - Lead the way towards a **unified method** for testing and validating CMFD closures under high pressure conditions in simple geometry
  - Addressing some aspects of challenges in boiling flows CMFD modeling

- **Two phases**
  - **Phase 1: open tests (October 2021- March 2022)**

    14 selected cases with already published data, and opening of some supplementary data
  - **Phase 2: blind tests (June 2022- November 2022)**

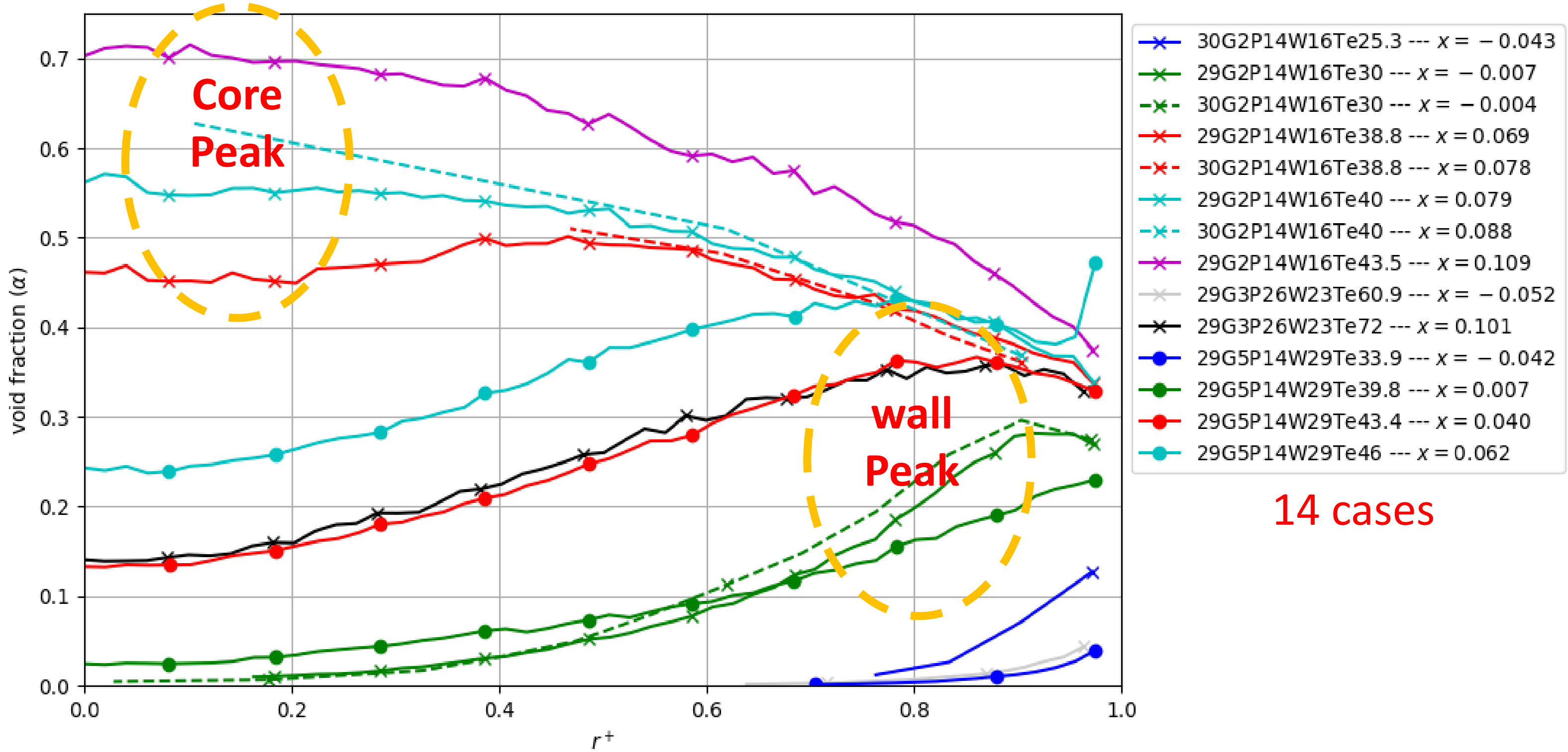
    4 additional cases in blind conditions

Organization and Objectives

Phase 1

Challenges

- A wide range of void fraction (up to 70%)
- Different positions of the peak values
- Single set of closure model for the whole database



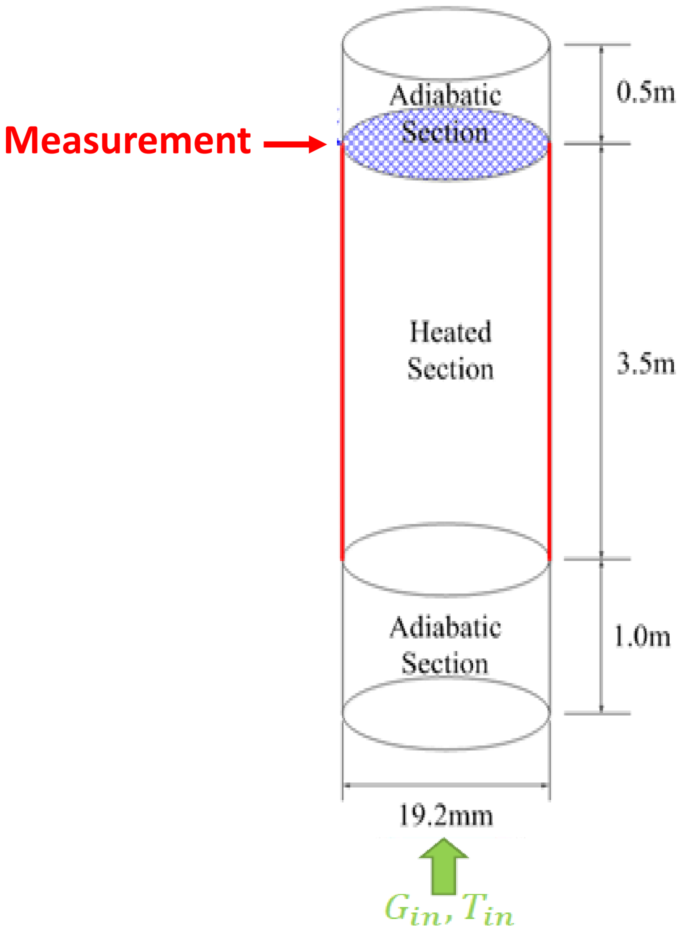
14 cases

Description of DEBORA Experiment

» Subcooled flow boiling under high-pressure conditions

Main Characteristics

Test Section	pressurized **pipe**
Working fluid	**R-12**
Pressure range	1,46 to 3.0 MPa (R-12) 9.0 to 17.0 MPa (water/steam)
Measurement	Radial profiles of boiling parameters at **one elevation**
Country	France **(CEA)**
Year	**2001**



Numerical and Physical Models (1)

Wall Heat Flux Partitioning

- The rate of **vapor generation** at the wall is computed by the Wall Heat Flux Partitioning model (WHFP)

WHFP "RPI boiling model"

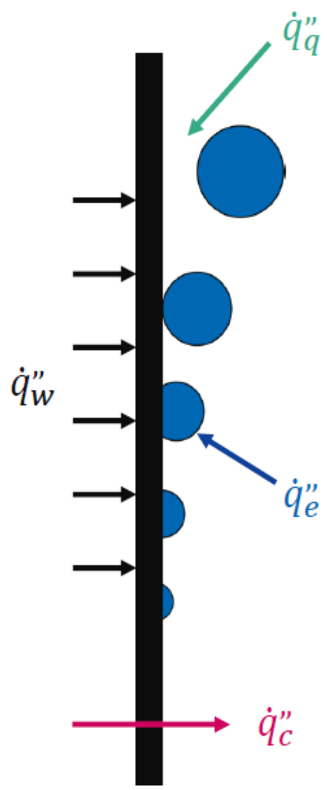
$$\dot{q}''_w = \dot{q}''_c + \dot{q}''_q + \dot{q}''_e$$

Single-phase Convection | Quenching | Evaporation

$$\dot{q}''_c = \left(1 - A_{2f}\right) h_c \left(T_w - T_l\right)$$

$$\dot{q}''_q = A_{2f} h_q \left(T_w - T_l\right)$$

$$\dot{q}''_e = N'' f \left(\frac{\pi}{6} D^3_{dep}\right) \rho_g h_{fg}$$



Sub models

- Bubble departure diameter (D_{dep})
- Bubble departure frequency (f)
- Nucleation site density (N)

Numerical and Physical Models (2)

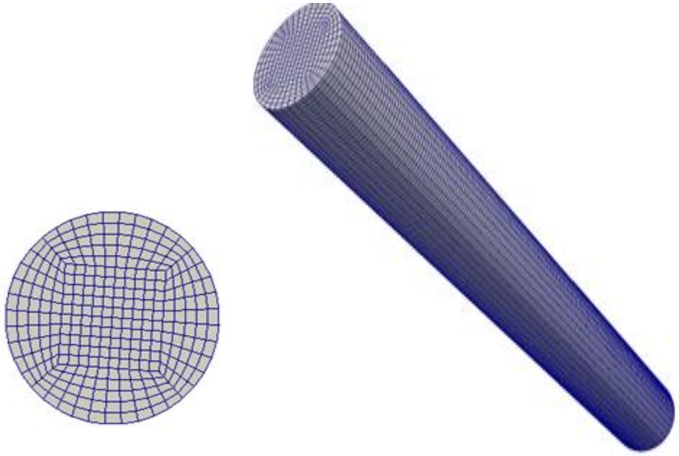
Interfacial Non-drag Force Model

- **Generated bubbles movement in radial direction**
  1. Bubble Lift force: push the bubble in a direction orthogonal to the main flow
  2. Turbulent dispersion force: spread particles and smear gradients
  3. Wall lubrication force: pushes the bubbles away from the wall

Parameters	Model
Wall Boiling	
Active nucleation site density	Hibiki-Ishii
Bubble departure diameter	Unal
Bubble departure frequency	Cole
Non-Drag forces	
Wall lubrication force	Antal
Bubble lift force	Tomiyama
Turbulence dispersion force	Gosman
Others	
Turbulence	Standard k-ε
Bubble induced turbulence	Kataoka
Bubble diameter (SMD)	Alatrash (KAERI)
Interfacial heat transfer	Ranz and Marshall

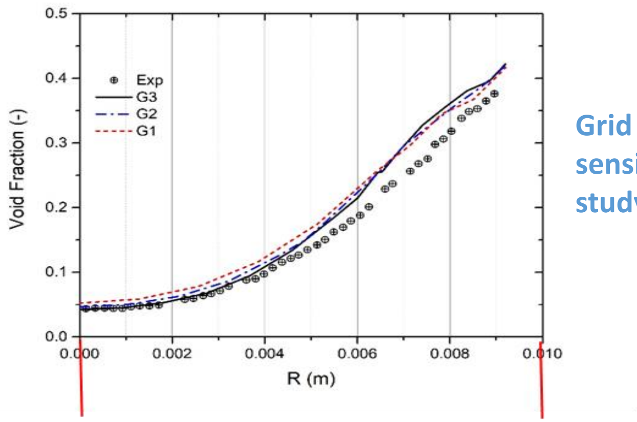
Mesh Generation

Computational Mesh and Test Matrix



<Full representation of the CFD domain>

G1=60000, G2=86400 and G3=117600)



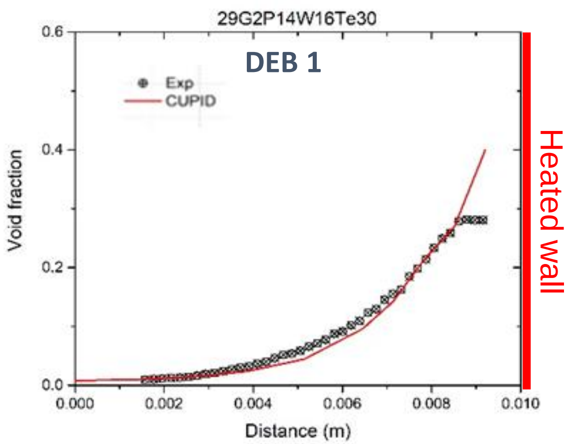
Grid sensitivity study

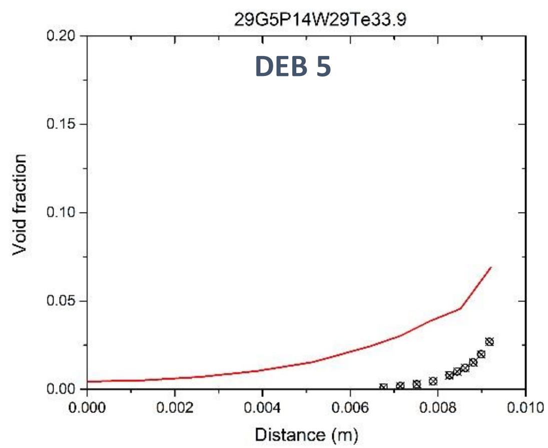
	Pressure (MPa)	Inlet subcooling (K)	Heat flux (W/m^2)	Mass flowrate (Kg/m^2s)	Test number
DEB 1	1.46	26.2	76240.0	2030	29G2P14W16Te30
DEB 2	1.46	14.4	76260.0	2022	29G2P14W16Te38.8
DEB 3	1.46	16.2	76260.0	2022	29G2P14W16Te40
DEB 4	1.46	12.5	76260.0	2024	29G2P14W16Te43.5
DEB 5	1.46	21.2	135000	5063	29G5P14W29Te33.9
DEB 6	1.46	15.7	135000	5085	29G5P14W29Te39.8
DEB 7	1.46	11.53	135000	5063	29G5P14W29Te43.4
DEB 8	1.46	9.53	135000	5070	29G5P14W29Te46

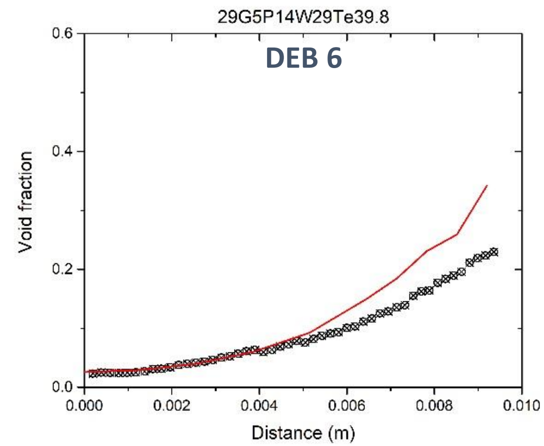
Position of the void fraction peak Shifted from wall to the bulk

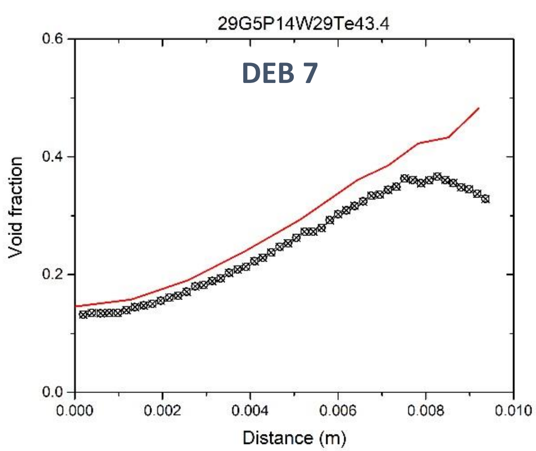
Calculation Results (1)

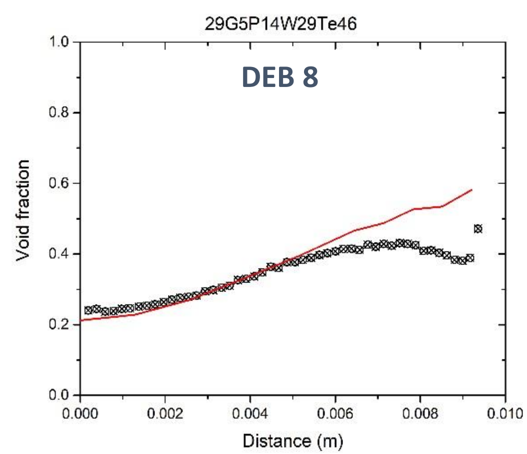
Wall Peaking Cases







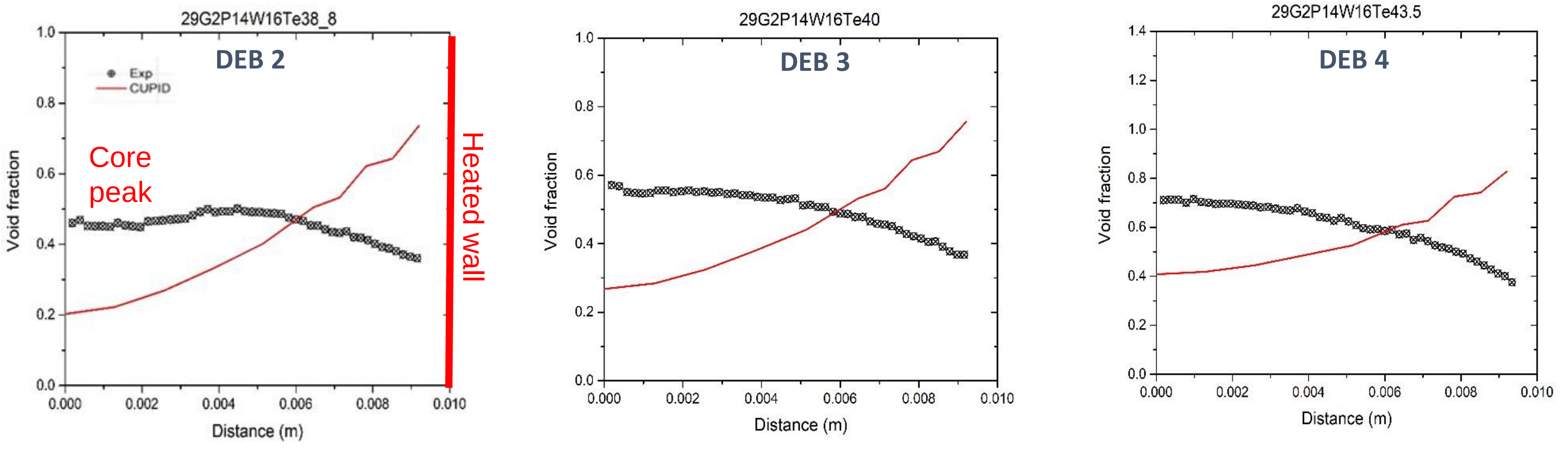




CUPID predicted the void fraction radial distribution well in the cases when the peak position is near the wall

Calculation Results (2)

Core Peaking Cases



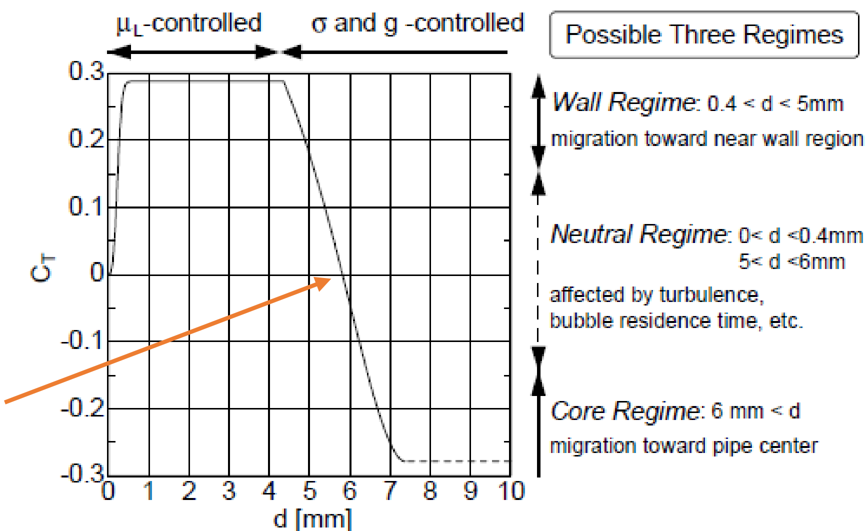
- **Calculations still predict the wall peaking trend.**
- **Change of the void fraction maximum position is caused by the lift force.**
- **Void fraction shifting are not captured correctly using the default setting (Original Tomiyama model)**

Modification of Lift Force Model

Modifying Tomiyama model

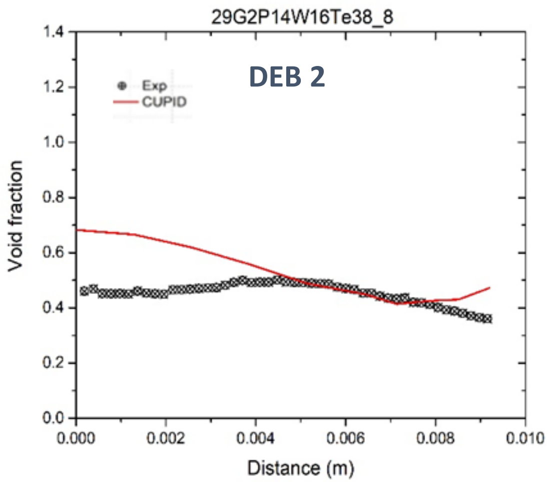
- **Change of the lift force coefficient sign: bubble sizes larger than 5.8 mm**
- **Tomiyama model: developed under atmospheric pressure conditions**
- **At high pressure bubble sizes are smaller.**
- **Modification of the Tomiyama model to change the sign of the lift force coefficient at bubble sizes larger than 0.7 mm**

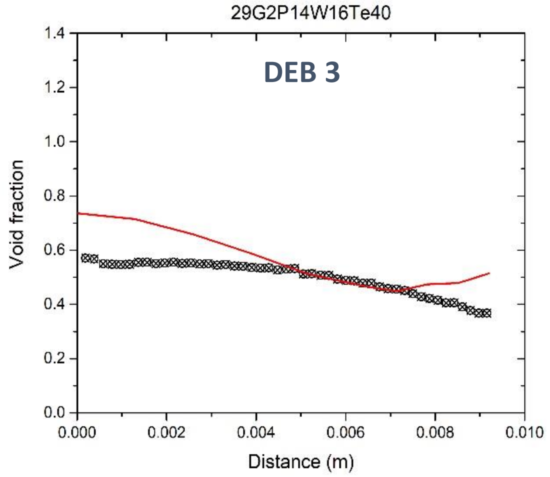
Change of lift force coefficient sign

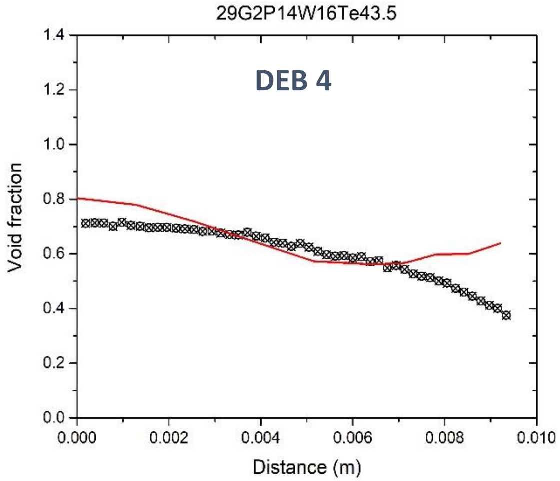


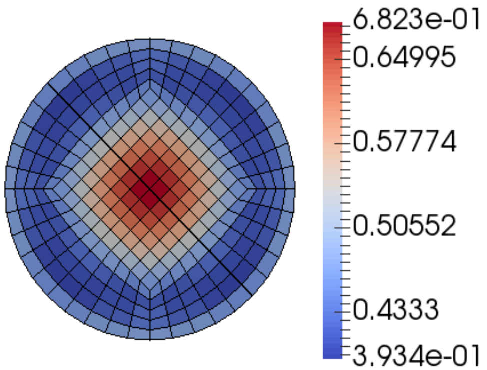
Improvement of Core Peak Prediction

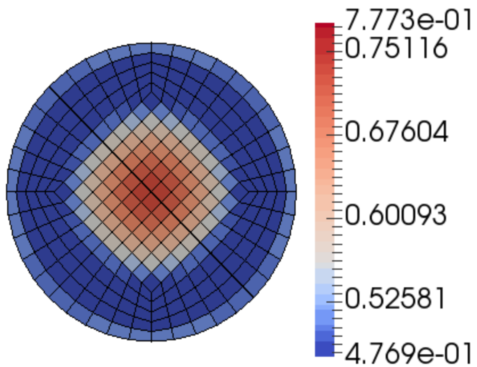
Effect of Modification

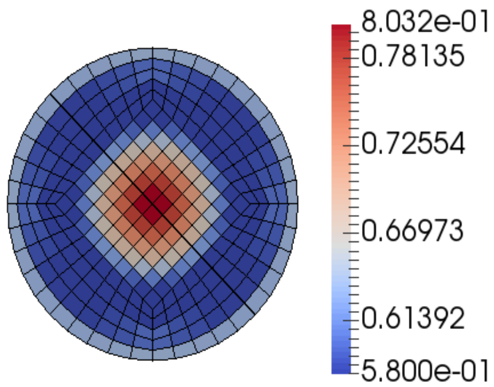












Conclusions

6

Conclusions

Validation of CUPID via international benchmarks

- Radiation model, turbulence model, WFHP model, non-drag force models

Summary

OECD/NEA IBE-4

- Turbulence mixing due to the density difference
- Modified k-e model

IAEA CRP

- Diffusion due to the concentration difference
- Turbulence mixing in complex geometries
- Low Reynolds number k-e model

OECD/NEA HYMERES-2

- Thermal stratification with radiation model
- Turbulence mixing in complex geometries
- Standard k-e model with modified buoyancy term

DEBORA Benchmark

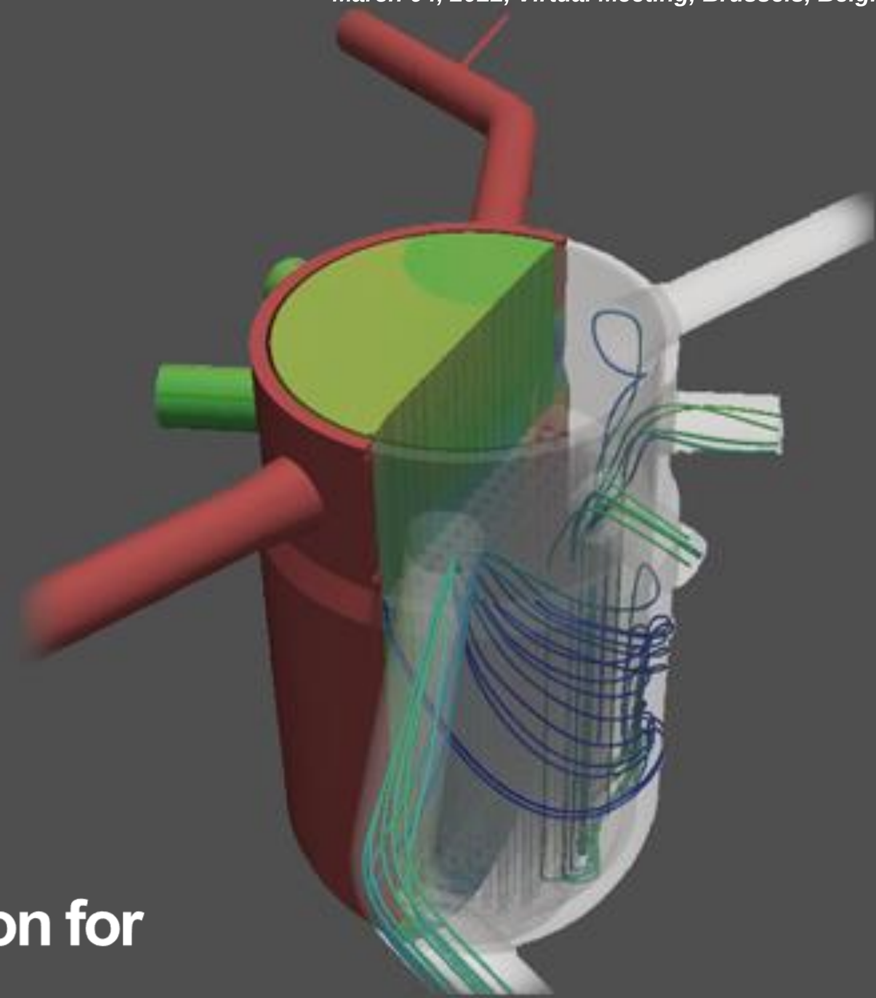
- Wall heat flux partitioning model
- Modification of bubble size criteria for bubble lift force

THANK YOU

yjcho@kaeri.re.kr

Korea Atomic Energy Research Institute
KAERI
한국원자력연구원

19th International Topical Meeting on Nuclear Thermal Hydraulics
March 04, 2022, Virtual Meeting, Brussels, Belgium



CUPID Workshop

Reactor Vessel 3D Mesh Generation for Safety Analysis

Seongju Do
March 04, 2022

CUPID Workshop

CONTENTS

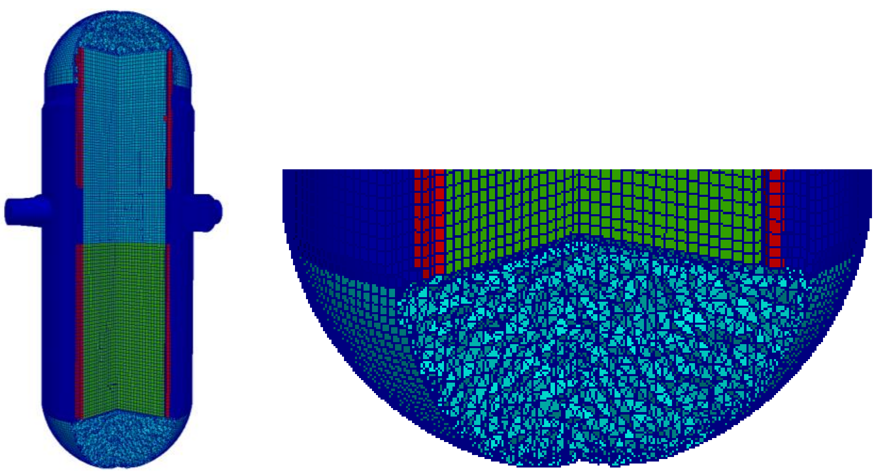
WHY RV Mesh 3D?

1

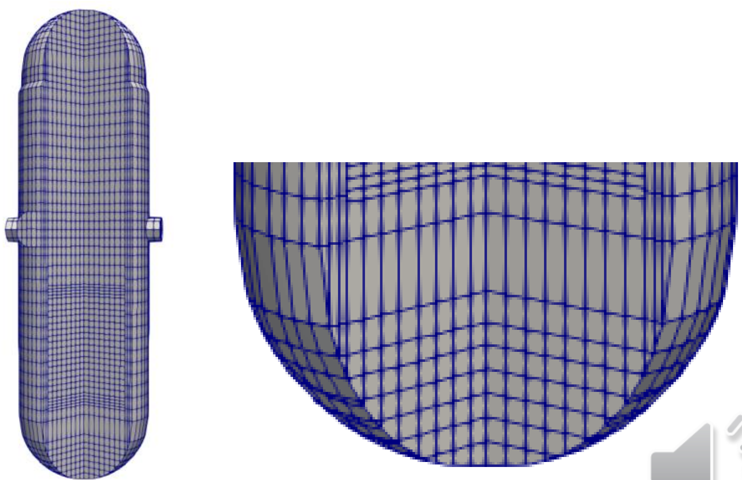
WHY 'RV Mesh 3D'?

- **3D mesh generator dedicated to the PWR vessel geometry is vital.**
  - **Include reactor core, downcomer(DC), upper/lower plenum(UP/LP) and hot/cold leg**
  - **Practical number of meshes (less than 10 million)**
  - **Most importantly, maintain structured mesh in the core region for the application of subchannel model**
  - **Applicable for different PWR geometries**

It's hard to apply commercial pre-processors



<Generated by commercial pre-processor>



<Generated by RV Mesh 3D>

Algorithms

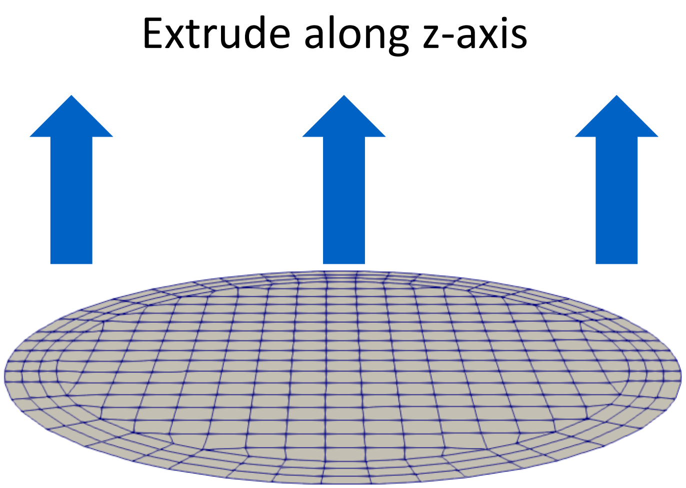
2

- Plane extrusion
- Cut-cell method
- Enhancement of mesh quality

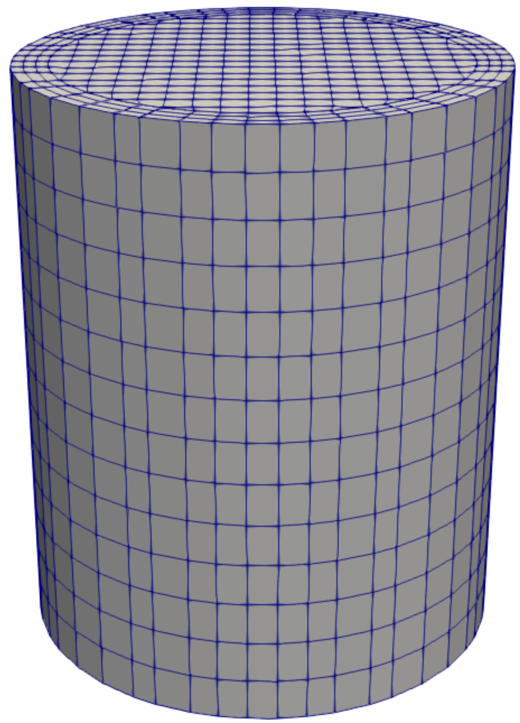
Plane extrusion

Core / Downcomer(DC) region

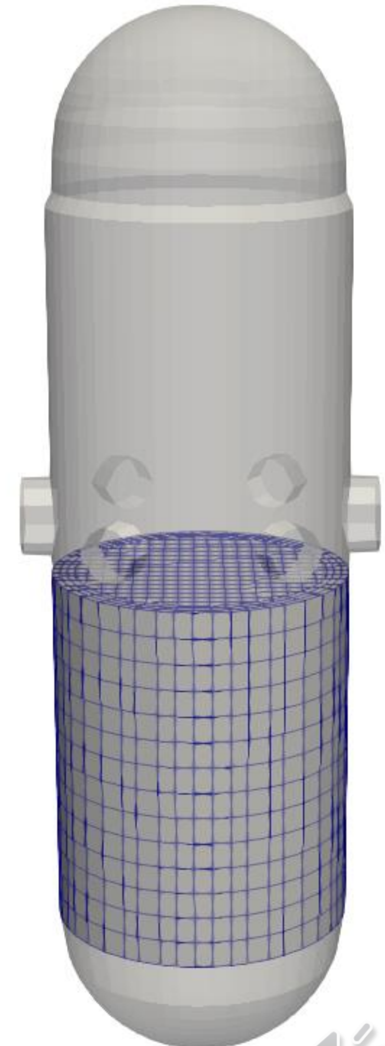
- **2D Mesh generation**
- **Plane extrusion along z-direction**



<2D plane extrusion>



<3D volume mesh>

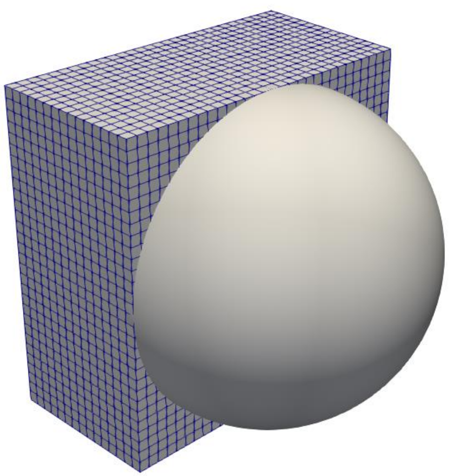


<Core / DC part>

Cut-cell method

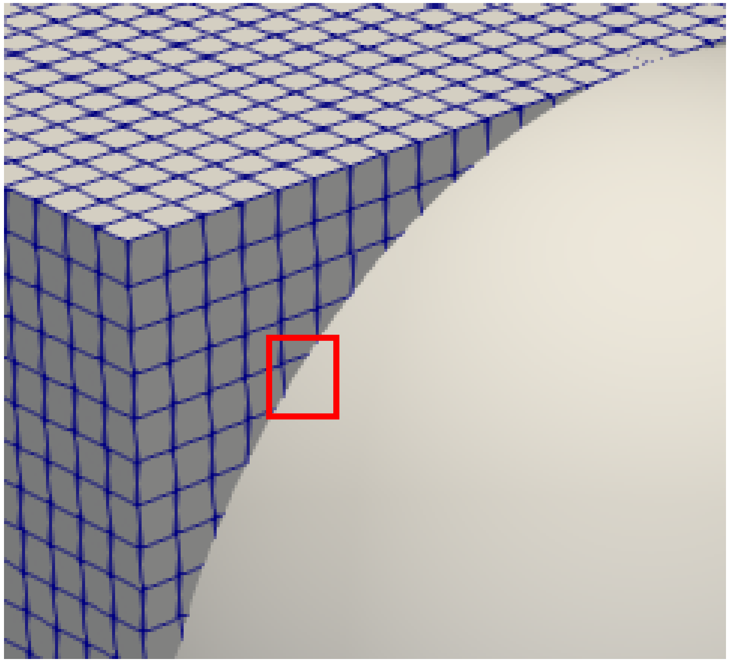
Representation of the Curved Surfaces

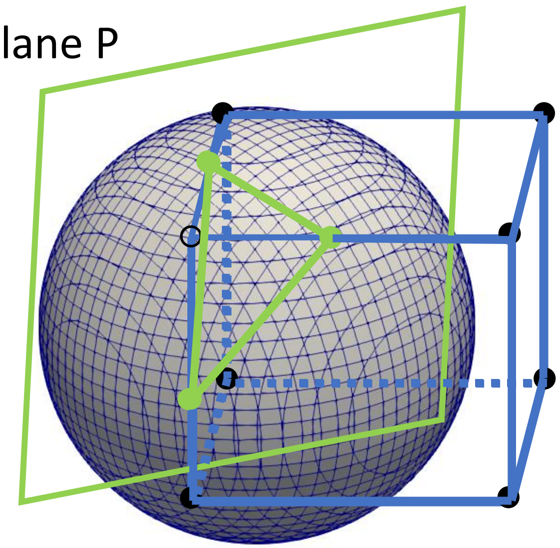
- **Upper / lower plenum**
- ***Cut-cell method*** **is applied for the curved faces**
  - Cut-cell approach uses background Cartesian grid with special treatments being applied to cells which are cut by solid bodies.



<Sphere is immersed in base mesh>

<Base mesh>

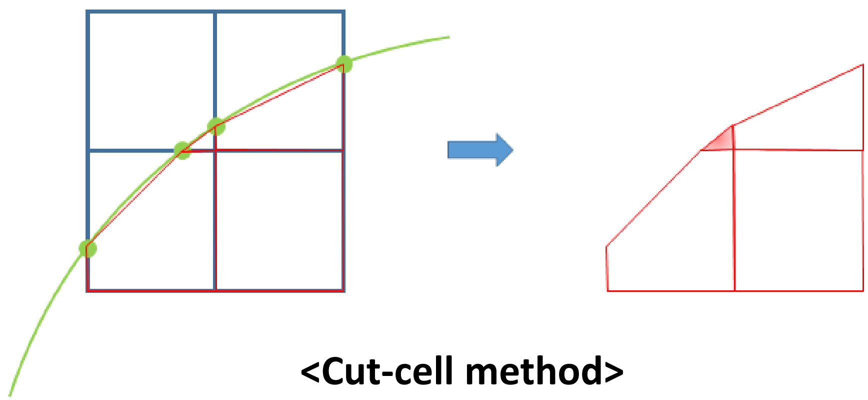




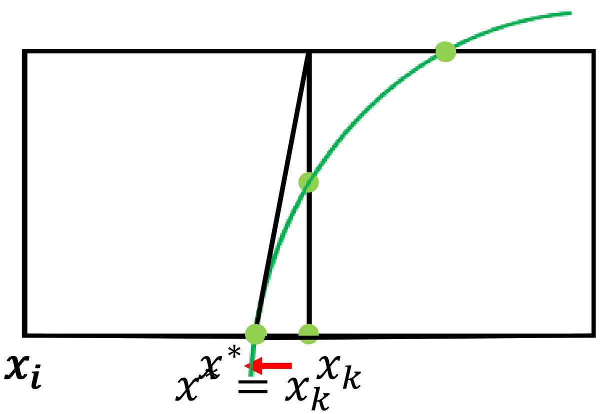
<After cut-cell algorithm>

Enhancement of mesh quality

- Cut-cell method may generate **small cells**
  - Small cells cause numerical instability and small time step size.
- The generation of small cells can be suppressed by **transforming the base grid.**

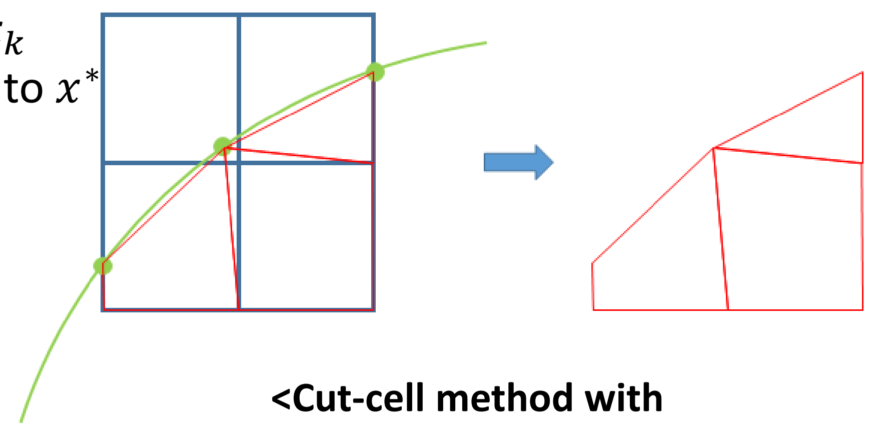


<Cut-cell method>



<Quality enhancement approach>

x^* is close to x_k
➔ move x_k to x^*

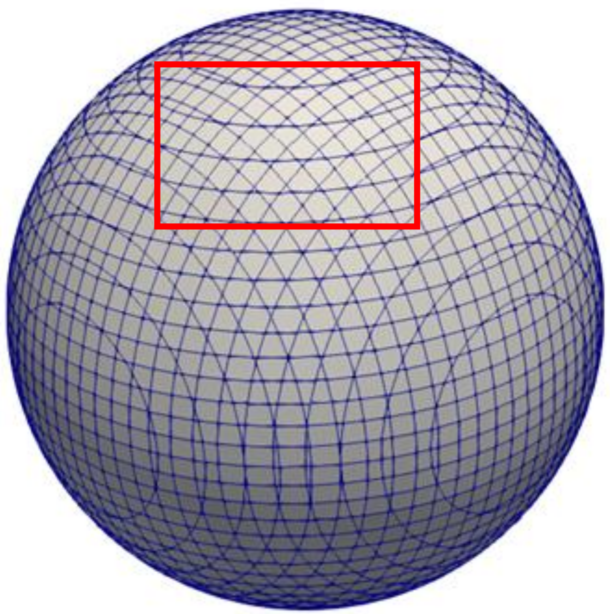


<Cut-cell method with quality enhancement>

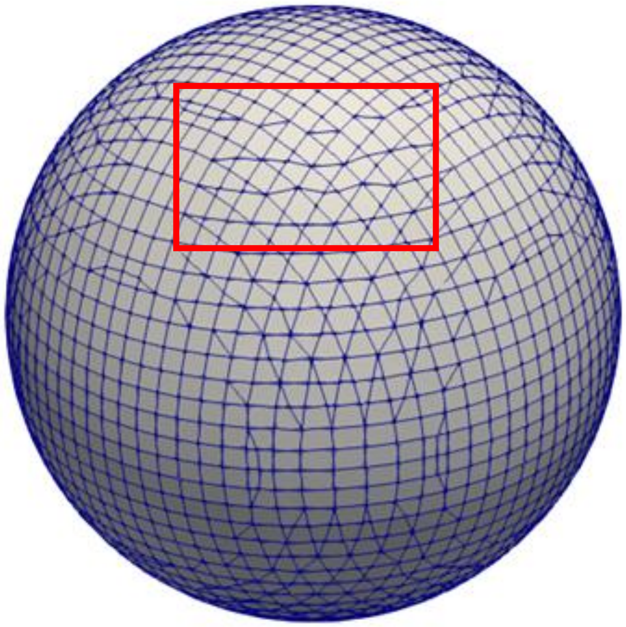
Enhancement of mesh quality

Results after the enhancement

- **Small cells are removed with the algorithm.**
- **There is no geometric distortion**



<Before enhancement>



<After enhancement with ε = 0.1>

Applications

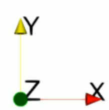
3

- OPR1000/APR1400
- NuScale (TerraPower, US)
- iSMR (KHNP, Korea)

OPR1000/APR1400

Mesh Generation Procedure

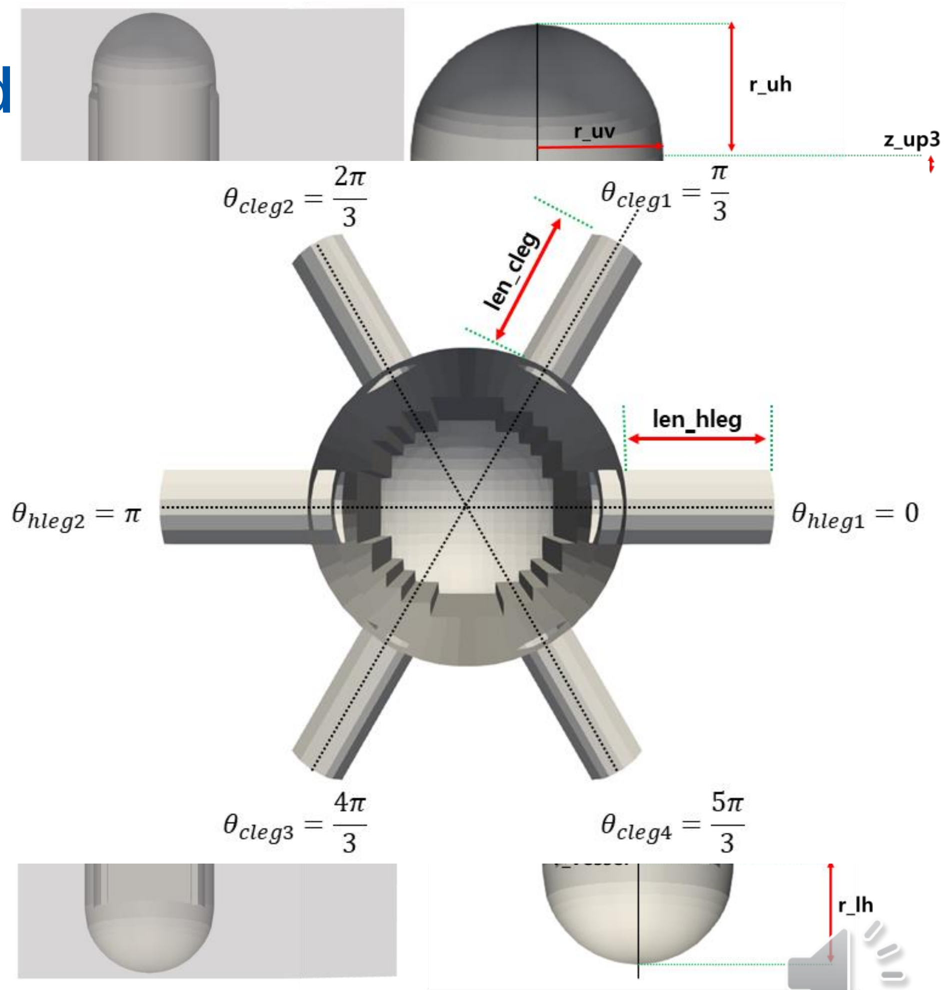
- **Generate 2D plane**
- **Core / DC region**
  - Extrude the 2D plane
- **LP/UP region**
  - Cut-cell method
- **Hot/cold legs**
  - Cell splitting



OPR1000/APR1400

User-friendliness of RVMesh3D

- **Text-based input**
- **User inputs are minimized**
  - Geometrical information
    - ✓ Heights
    - ✓ Radius
    - ✓ Angle/length of legs
    - ✓ FA configuration
  - Mesh information
    - ✓ Mesh resolution (assembly/subchannel)



OPR1000/APR1400

Mesh generation of OPR1000/APR1400

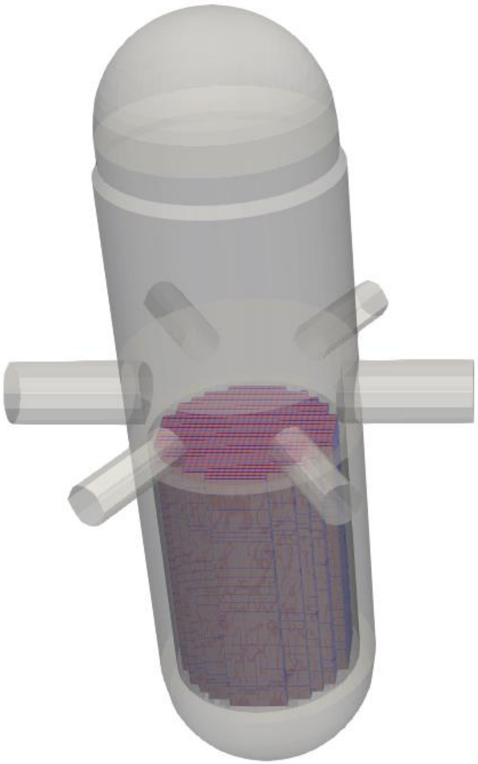
Main geometrical differences

- Radius of vessel
- Assembly configuration
  - ✓ 15x15 Grid / 17x17 Gı

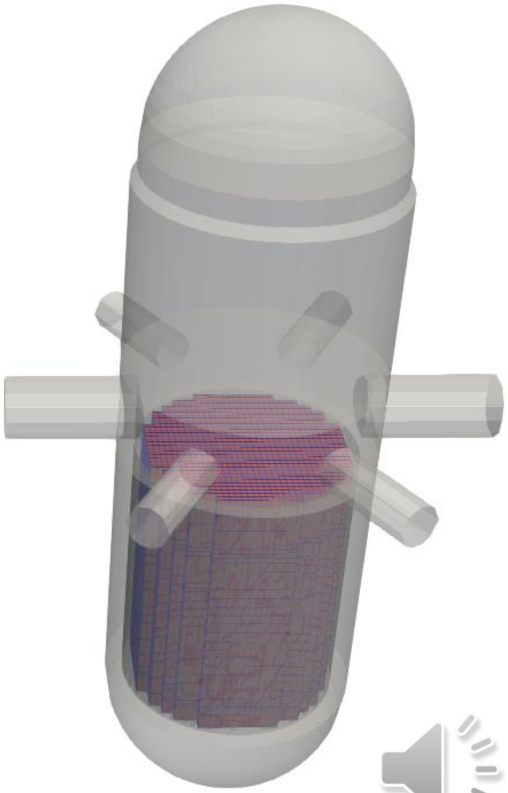
```
!------------------------MASTER nz = 28----------------------------!
&PLANE_2D
  r_core = 1.7526d0 !3.505200d0*0.5d0
  df0 = 0.208740d0
  witdh_dc = 0.3048d0      !...r_vessel=r_core+witdh_dc
  rhlg = 0.6d0  !...'rhlg' is a just some sufficiently large value rather than 'real
  n_layer_dc = 3
  nx_assem = 15
  ny_assem = 15
  nx_pin = 16
  ny_pin = 16
  pitch_rod_rod  = 0.012852d0     ! pitch1, rod-to-rod pitch
  pitch_rod_wall = 0.007980d0      ! pitch2, rod-to-wall distance

  resolution_base = 2  ! 1: 1x1, 2:2x2, 3:4x4, 4:8x8, 5:rod-rod.
  resolution_core = 2  ! 1: 1x1, 2:2x2, 3:4x4, 4:8x8, 5:rod-rod.
  MARS_coupling = .false.
/

&ASSEMBLY_MAP_2D
  mask_assem(1:15, 1) = 0 , 0 , 0 , 0 , 0 , 1 , 1 , 1 , 1 , 1 , 0 , 0 , 0 , 0 , 0
  mask_assem(1:15, 2) = 0 , 0 , 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0 , 0 , 0
  mask_assem(1:15, 3) = 0 , 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0 , 0
  mask_assem(1:15, 4) = 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0
  mask_assem(1:15, 5) = 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0
  mask_assem(1:15, 6) = 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1
  mask_assem(1:15, 7) = 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1
  mask_assem(1:15, 8) = 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1
  mask_assem(1:15, 9) = 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1
  mask_assem(1:15,10) = 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1
  mask_assem(1:15,11) = 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0
  mask_assem(1:15,12) = 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0
  mask_assem(1:15,13) = 0 , 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0 , 0
  mask_assem(1:15,14) = 0 , 0 , 0 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 1 , 0 , 0 , 0
  mask_assem(1:15,15) = 0 , 0 , 0 , 0 , 0 , 1 , 1 , 1 , 1 , 1 , 0 , 0 , 0 , 0 , 0
/
```

r_
nx
ny
ma

<OPR1000>

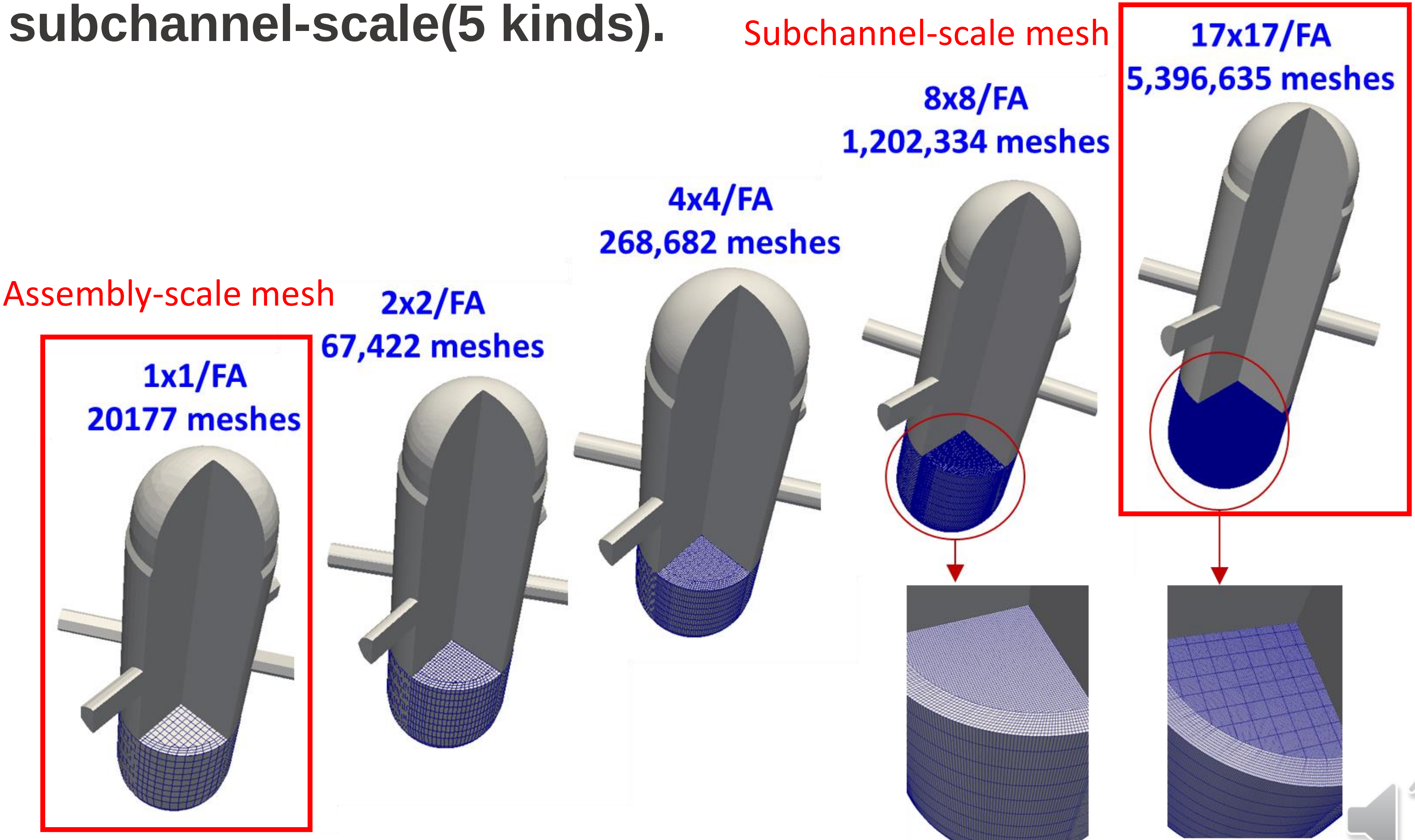


<APR1400>



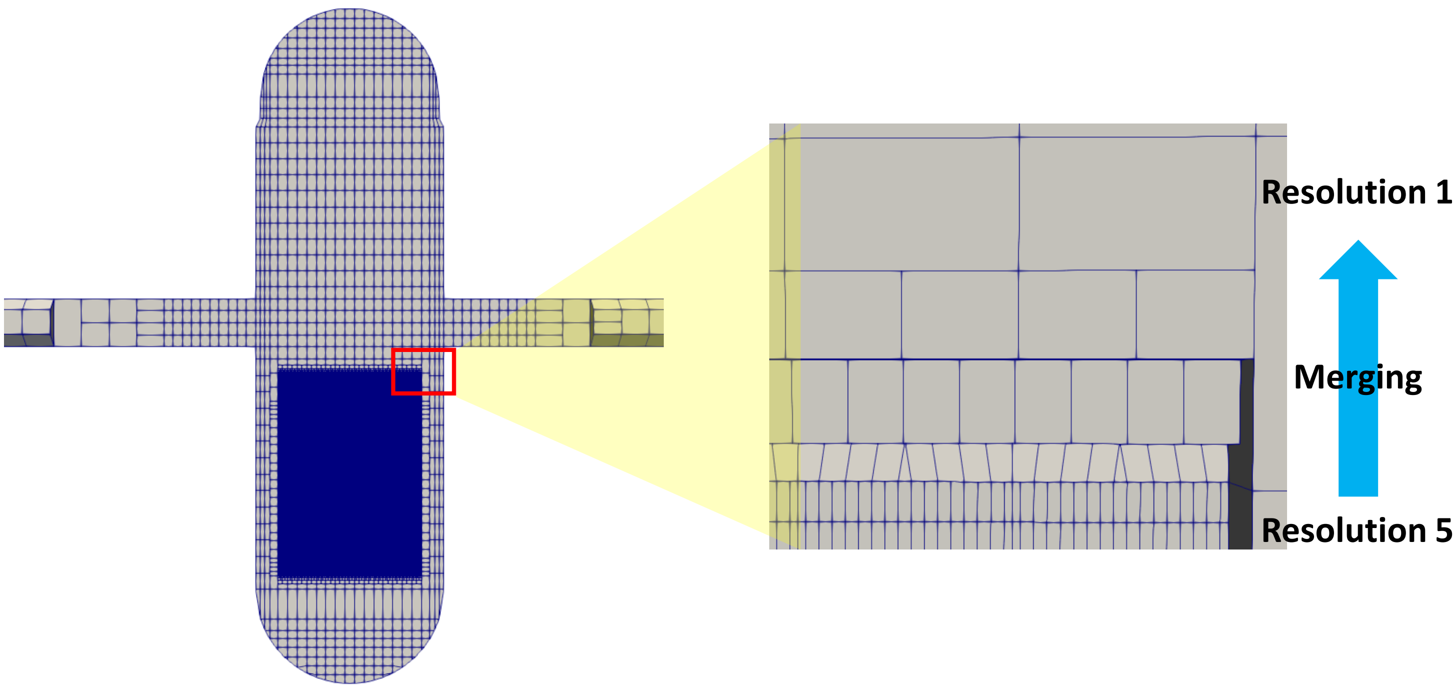
OPR1000/APR1400

- Mesh resolution can be controlled from assembly-scale to subchannel-scale(5 kinds).



Mesh Optimization

- To reduce simulation time, ***mesh adaptation*** technique is applied.
- High resolution mesh is utilized in **core region** only.
- The number of mesh is reduced by about **75%**.



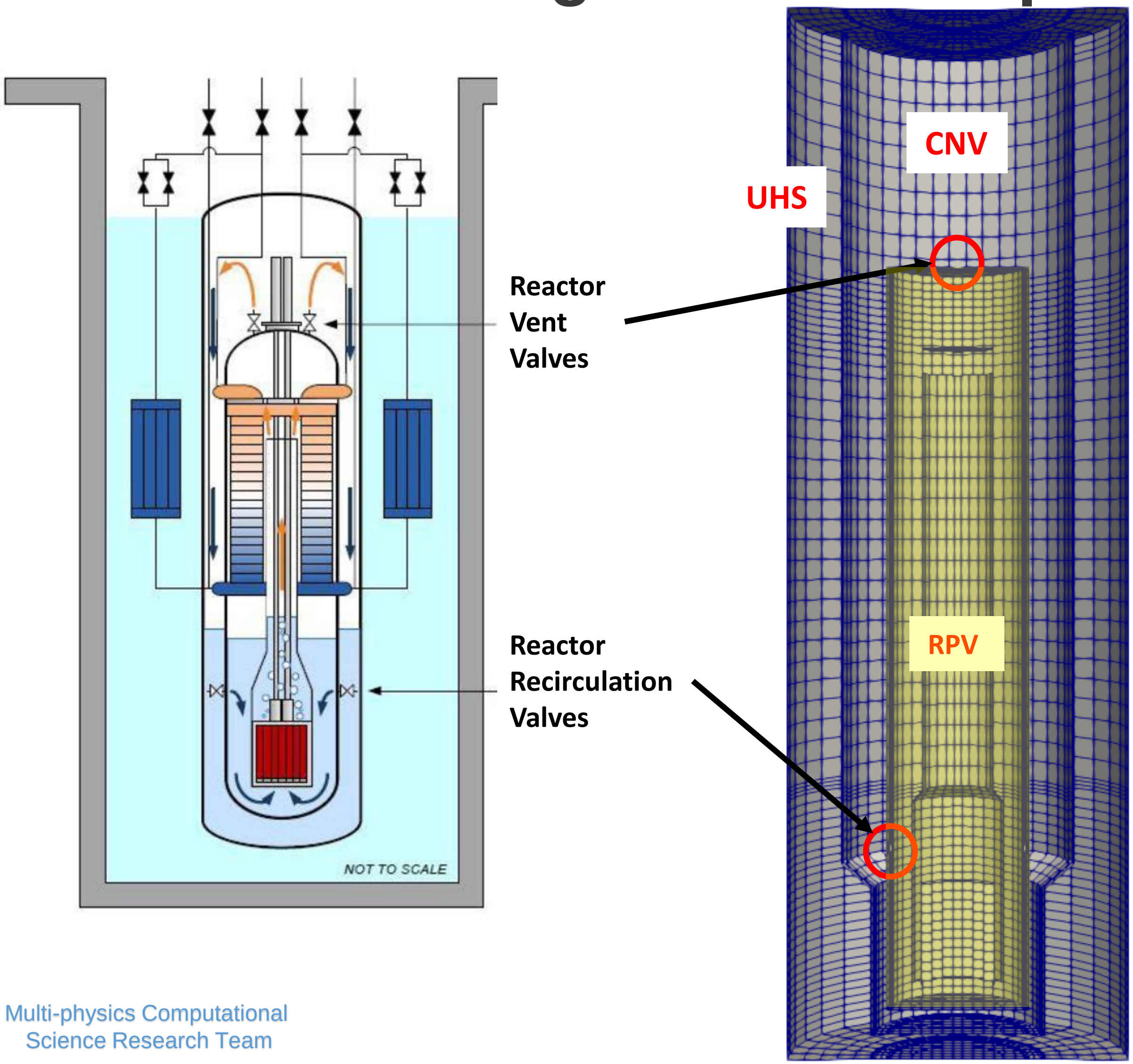
Case	Num. cells
1x1/FA	20,177
2x2/FA	67,422
4x4/FA	268,682
8x8/FA	1,202,334
17x17/FA	5,396,635
Adaptive	1,390,578

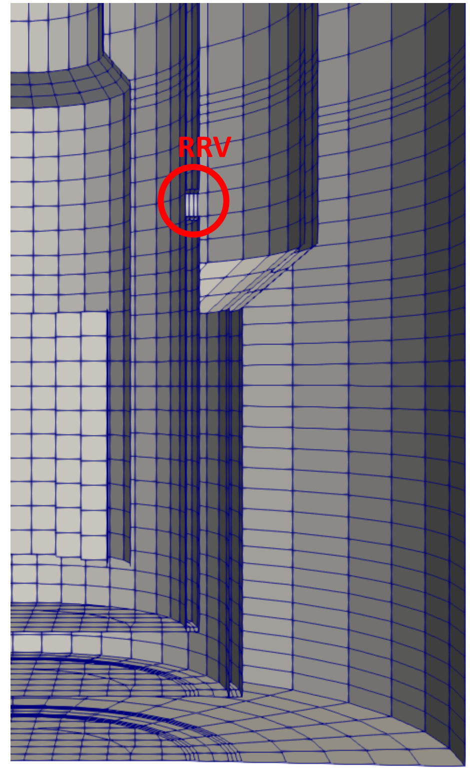
75% reduction

NuScale (US SMR)

Schematic diagram

Computational mesh

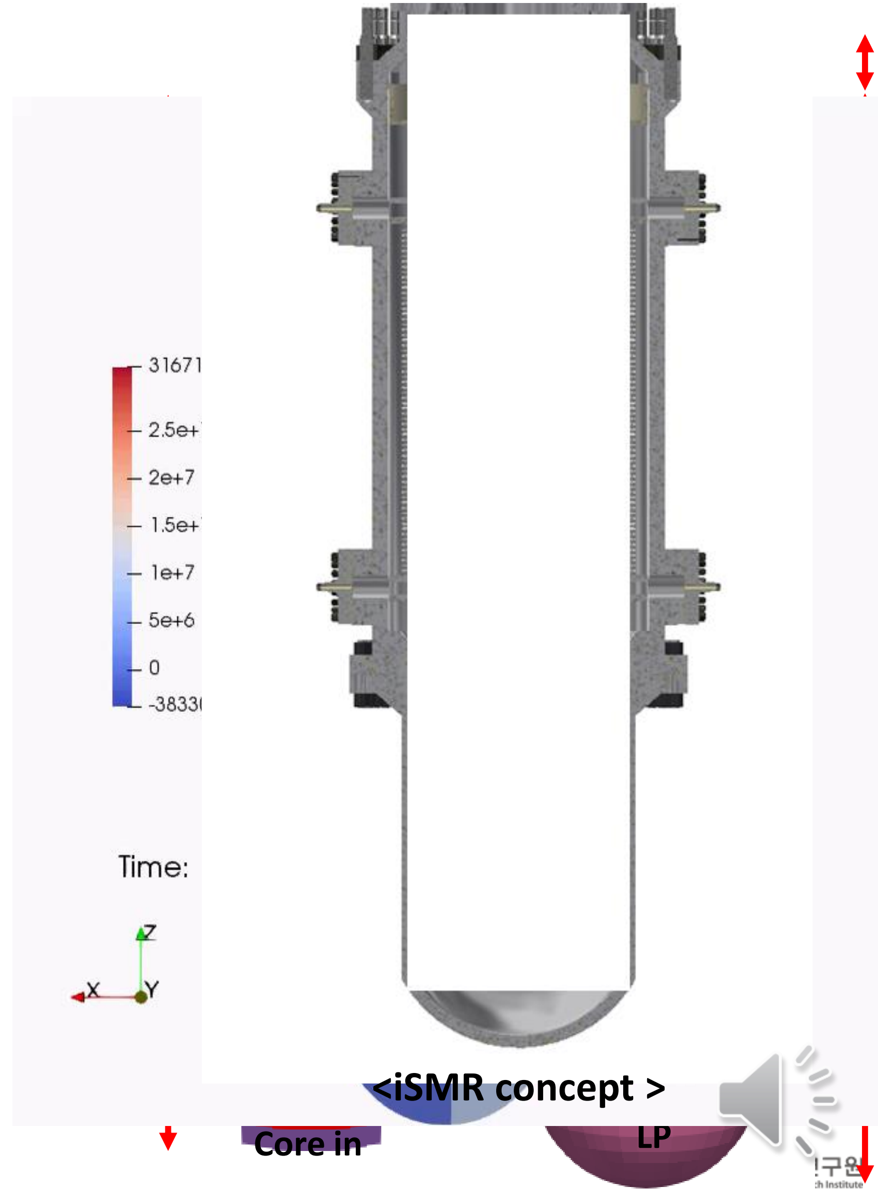




- RVV/RRV flow path is considered to simulate LOCA.
- 3D LOCA simulation is currently in progress.

iSMR(Korean SMR)

- iSMR design is in progress in Korea.
- CUPID and RVMesh3D are used to simulate **natural circulation** phenomenon.



Summary

- Maintain **structured mesh in the core region** for the application of **subchannel model**

- Practical number of meshes (**1.5 million** for **subchannel scale**)

- Applicable for **different PWR** and **SMR** geometries with **simple user input**

THANK YOU

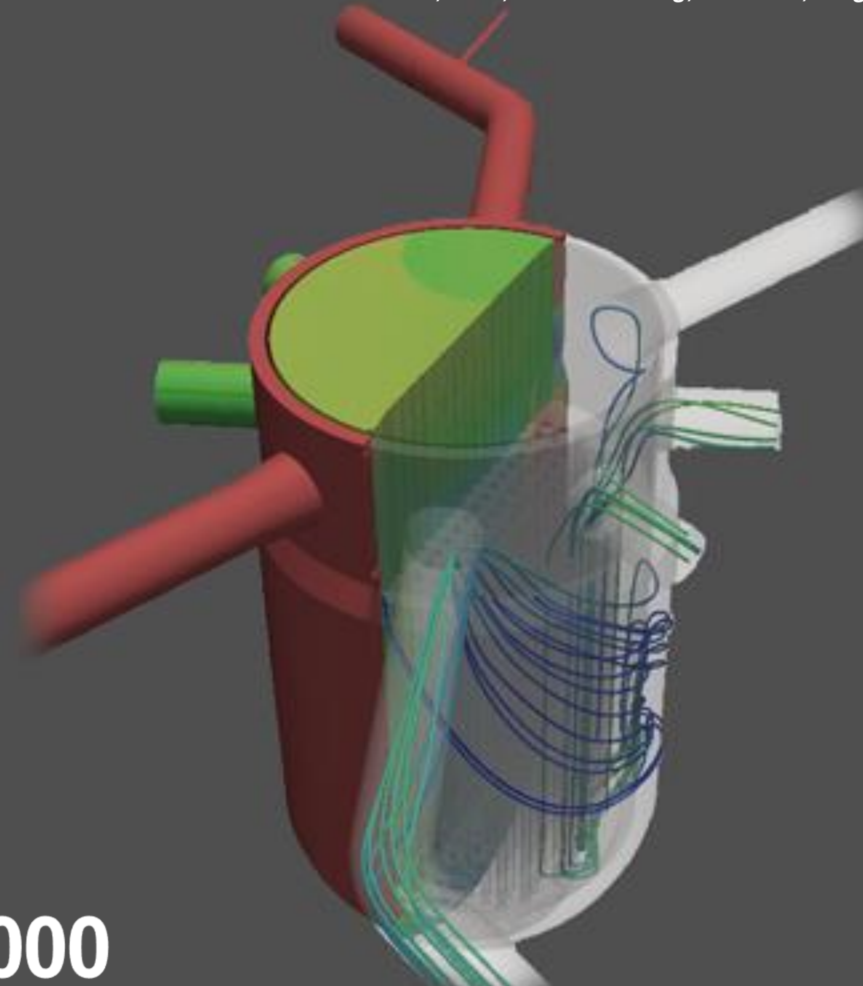
sjdo@kaeri.re.kr

Korea Atomic Energy Research Institute

KAERI

한국원자력연구원

19th International Topical Meeting on Nuclear Thermal Hydraulics
March 04, 2022, Virtual Meeting, Brussels, Belgium



CUPID Workshop

Pin-wise Full Core Safety Analysis of OPR1000

Jae Ryong Lee
March 04, 2022

CONTENTS

WHY 3D Safety Analysis ?

1

— 3D Safety Analysis Issues

— Multi-Scale & Multi-Physics (MSMP) Approach to Safety Analysis

3D Safety Analysis Issues

Steam Line Break (SLB)

Safety issue of SLB accident

- Increase of heat removal due to steam line break
- Local power increase and radially asymmetric distribution
- **DNBR** Margin ※ DNBR: Departure from Nucleate Boiling Ratio

System T/H Analysis for Non-LOCA

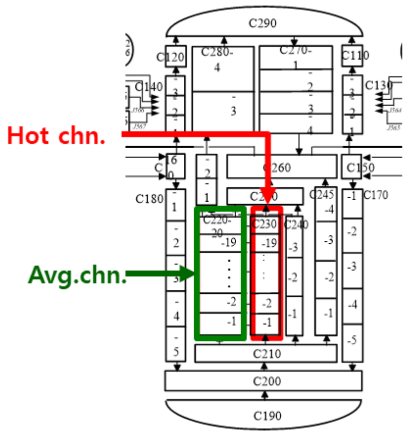
1D nodalization

- Hot channel modeling for DNBR evaluation

Limitation of 1D approach

- **Axial flow** ONLY
- Neutron power using **point-kinetics**
- **Simplified geometric parameter**
  - ✓ Hydraulic diameter, heated diameter

➔ **Conservative safety analysis results**



System TH nodal for PWR

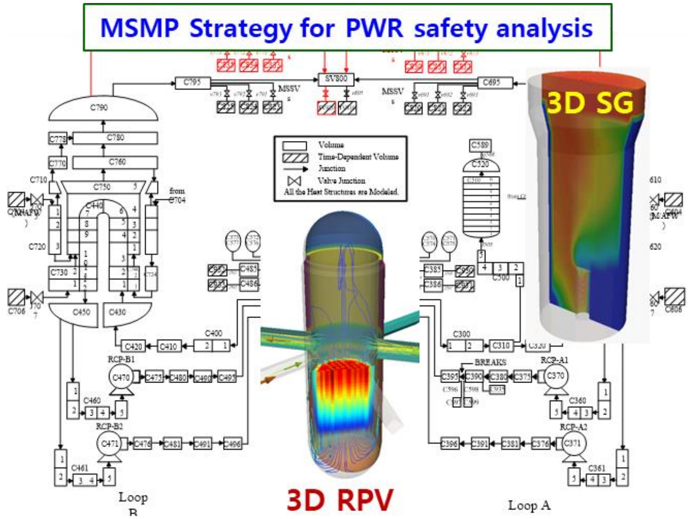
Multi-Scale & Multi-Physics (MSMP) approach

Multi-Scale T/H

- ***3D (subchannel T/H)*** **resolution for region of interest**
  - **Reactor pressure vessel**, steam generator
  - Desirable spatial resolution for 3D resolution
    - ✓ Ex. **Subchannel scale** for core
  - Realistic multi-dimensional flow behavior
    - ✓ Radial flow behavior in core, two-phase flow in secondary side of SG
- ***1D (Sys. T/H)*** **resolution for the rest of RCS**

Multi-Physics (N/K, F/P)

- ***Pin-wise fuel behavior***
  - 3D power distribution
    - ✓ Neutron kinetics (N/K) code
  - Realistic fuel rod status
    - ✓ Fuel performance (F/P) code



Multi-Scale and Multi-Physics (MSMP) Configuration

2

– Multi-Scale & Multi-Physics (MSMP) Strategy

– MARU* Platform

※ MARU
(Multi-physics Analysis Platform for Nuclear Reactor SimUlation)

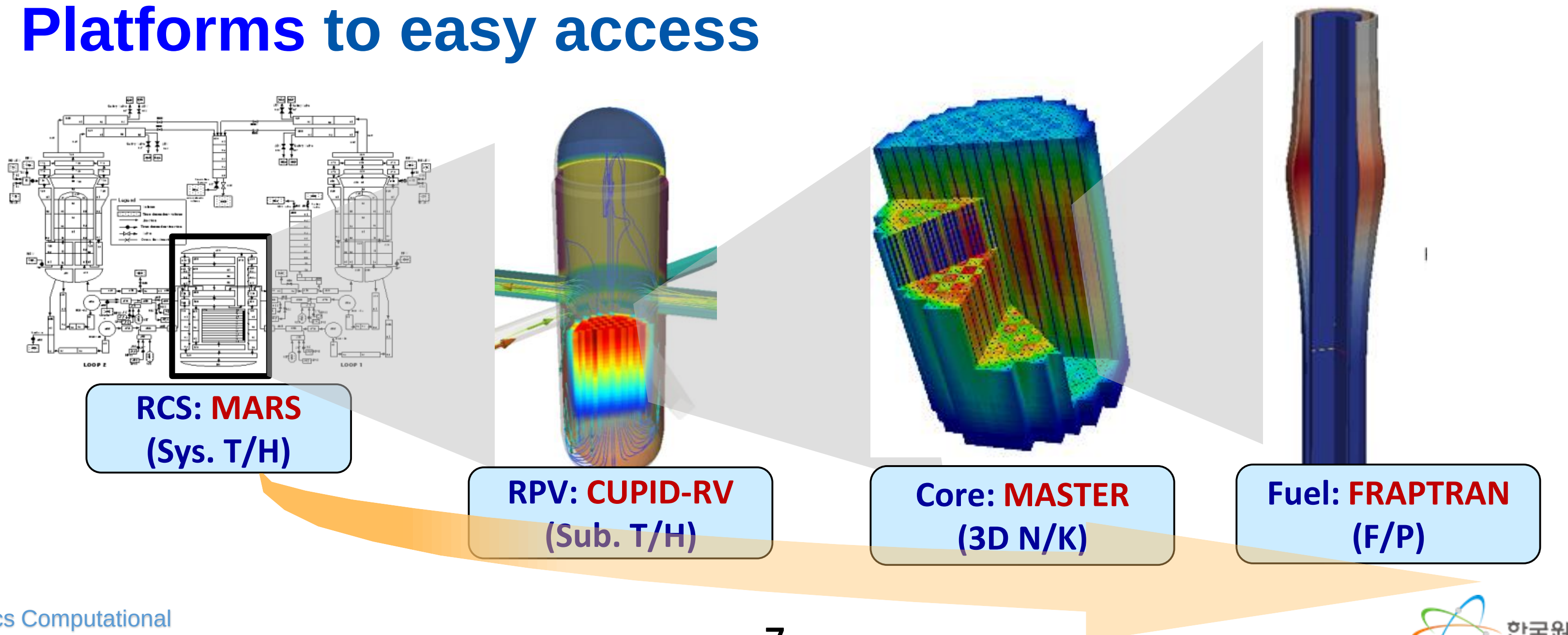
Multi-Scale & Multi-Physics (MSMP) Strategy

MSMP Simulation Scope

➢ Entire RCS is considered

Region	features	Code	Coupling
RCS	System-scale T/H	MARS	Source-to-source
RPV	Subchannel-scale T/H	CUPID-RV	
Reactor core	Fuel performance	FRAPTRAN	
	3D neutron diffusion	MASTER	Dynamic Link Library (DLL)

➢ Platforms to easy access



MARU Platform

Background

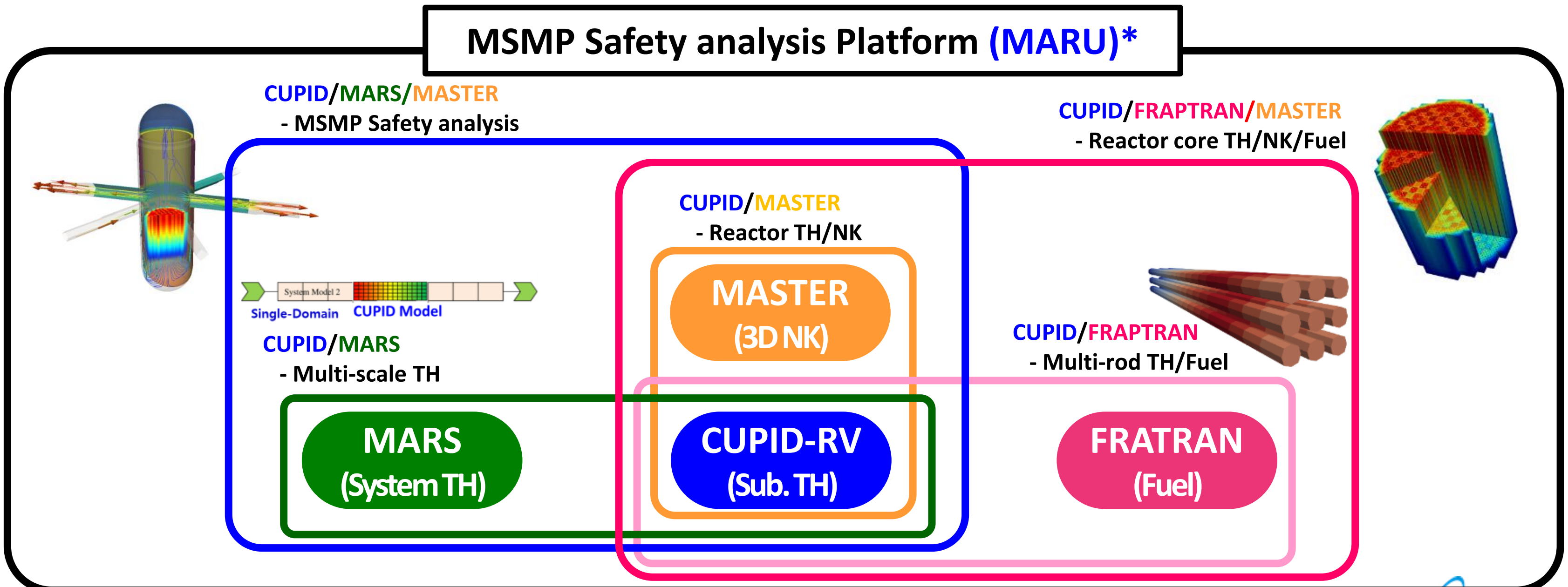
Limited coupled simulation

- Multi-scale T/H , Multi-physics (T/H &
- Necessity for **integrated tools** for M

Code	Physics	Ownership	Year
MARS	System T/H	KAERI	2006
CUPID-RV	Subchn. T/H	KAERI	2017 (Ver.2.5)
MASTER	Nodal N/K	KAERI	2013 (Ver.4)
FRAPTRAN	Fuel Per.	US NRC	2014 (Ver.2)

MARU (Multi-physics Analysis Platform for Nuclear Reactor SimUlation)

Platform for 3D MSMP Safety analysis



Platforms Structure

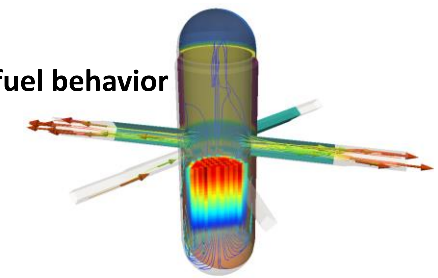
TCP/IP socket communication

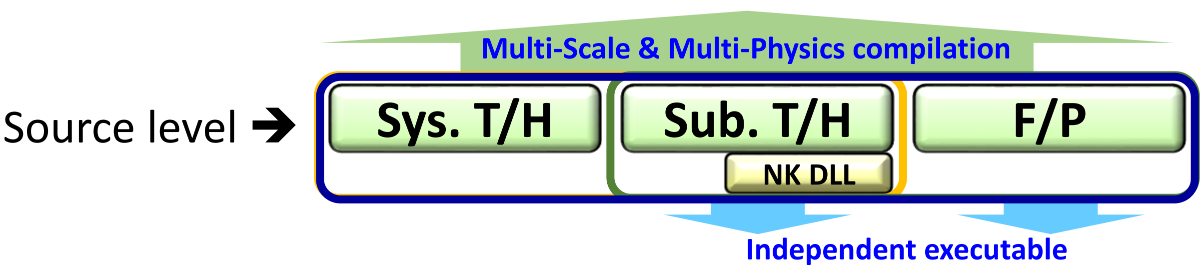
- Server (Linux) ←→ Client (Windows)

Multiple source-to-source compilation among codes as user needs

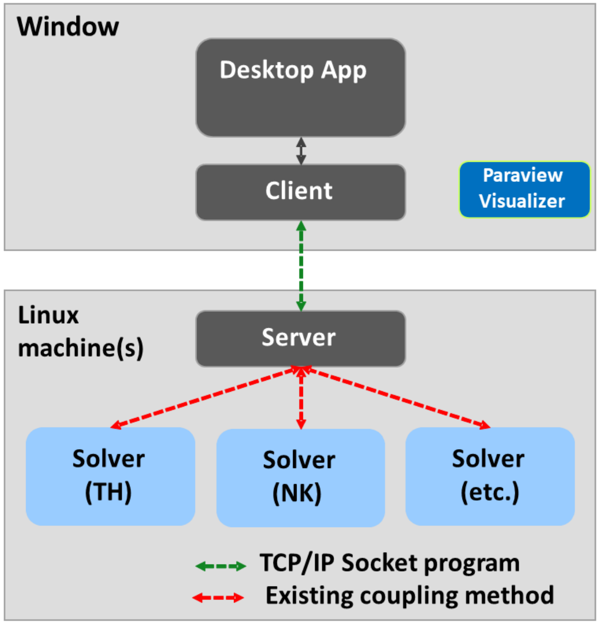
- Equivalent source level between T/H and F/P
  - ✓ N/K is weak coupled by DLL

Full core pin-wise fuel behavior
With RCS status





MARU configuration



MSMP Safety Analysis

3

— MSMP Simulation of OPR1000 SLB Accident

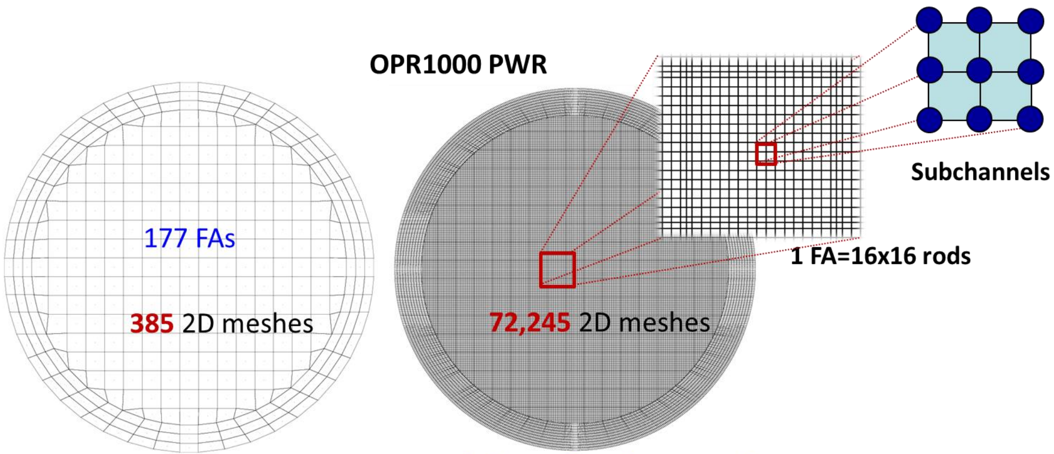
— Improvement of Safety Margin (DNBR)

3D Reactor Pressure Vessel Modeling

Subchannel-scale RPV Computational Geometry

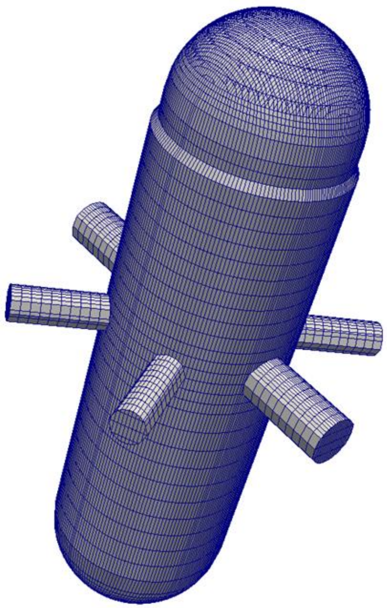
Body-fitted RPV mesh

- **In-house** RPV mesh generator (***RVMesh3D***)
- Reactor core, downcomer, upper/lower plenum, and hot/cold leg
- Practical number of meshes (Currently **1.3M**)
- **Subchannel T/H resolution** for core region



Assembly-scale 2D mesh

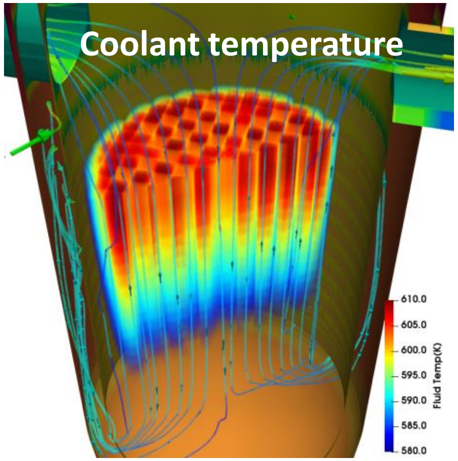
Subchannel-scale 2D mesh

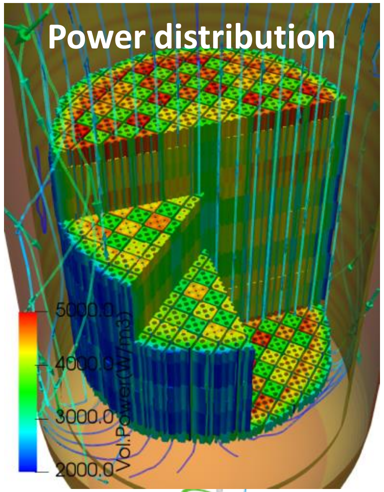


Body-fitted 3D Mesh

MSMP Simulation of OPR1000 SLB Accident

Steady State (End of Cycle Full Power)





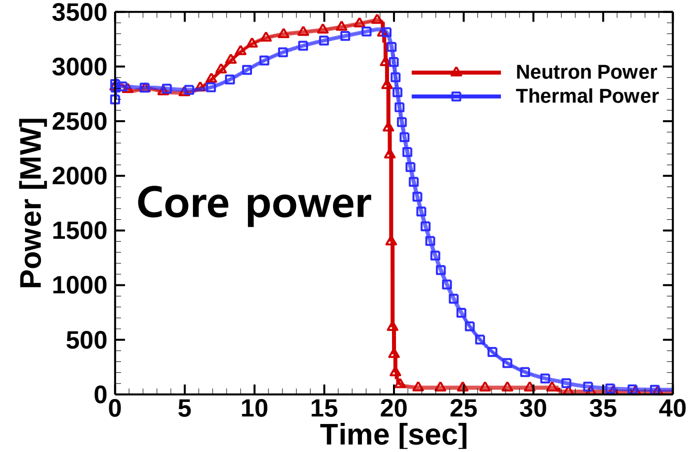
MSMP Simulation of OPR1000 SLB Accident

Sequence of Events and Major Parameters

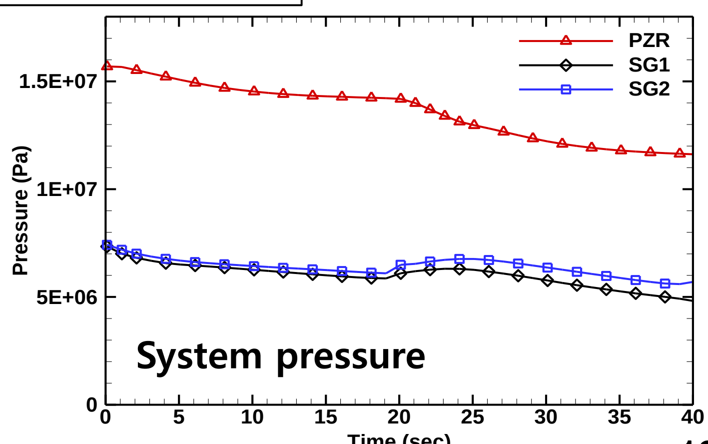
Sequence of events

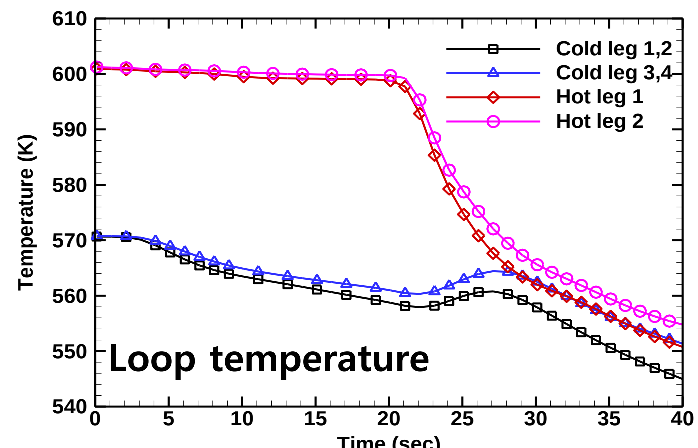
Time(sec)	Event	Setpoint
0.0	Steam line break occurs	
18.4	**Overpower trip setpoint reached**	**121 %**
18.6	Turbine Trip	
19.1	Rod begins to drop	
34.0	Low SG1 setpoint reached	5.44 MPa
35.1	MSIV1 closed	

MSMP result



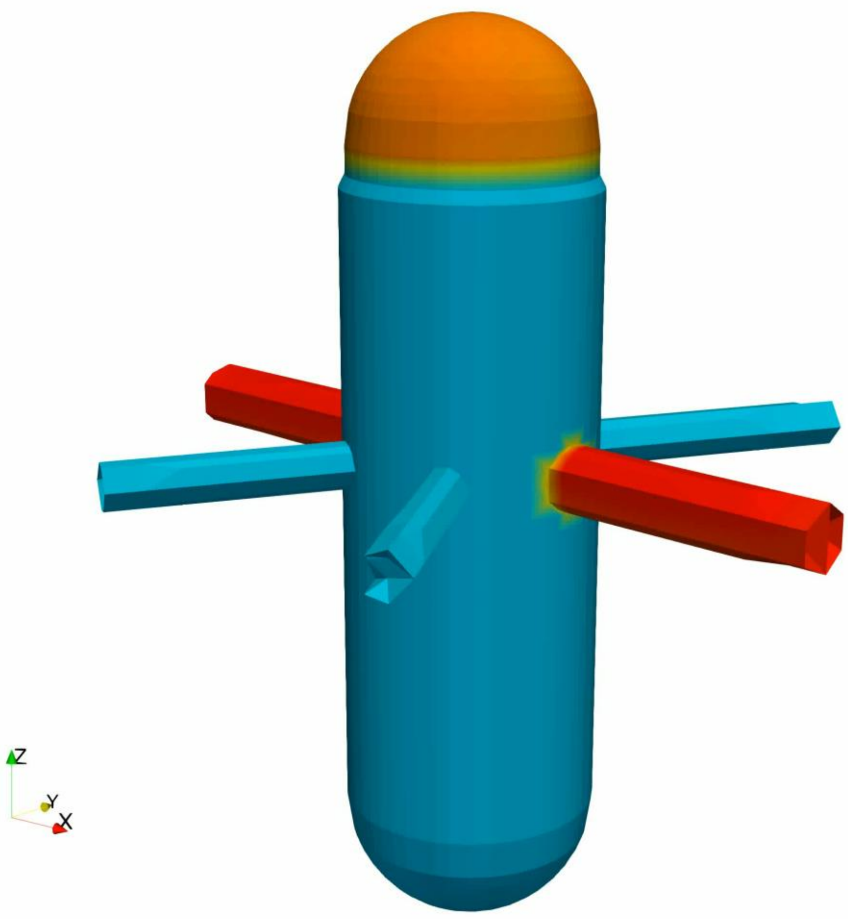
System TH result





MSMP Simulation of OPR1000 SLB Accident

Power & DNBR Distribution



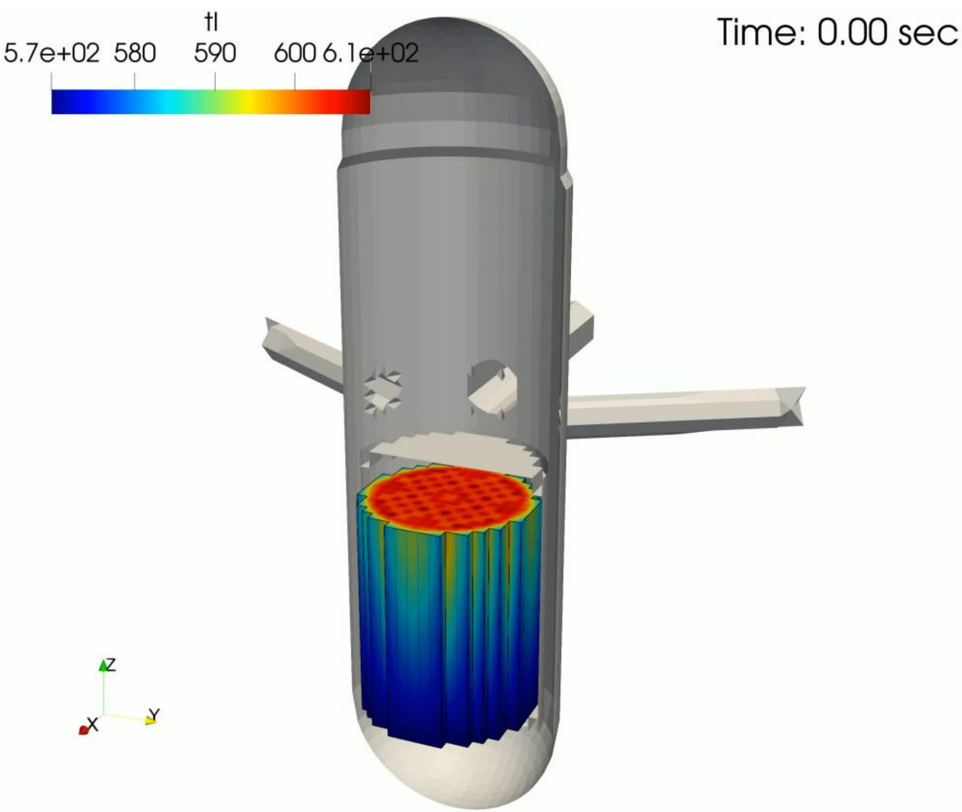
Sequence of events

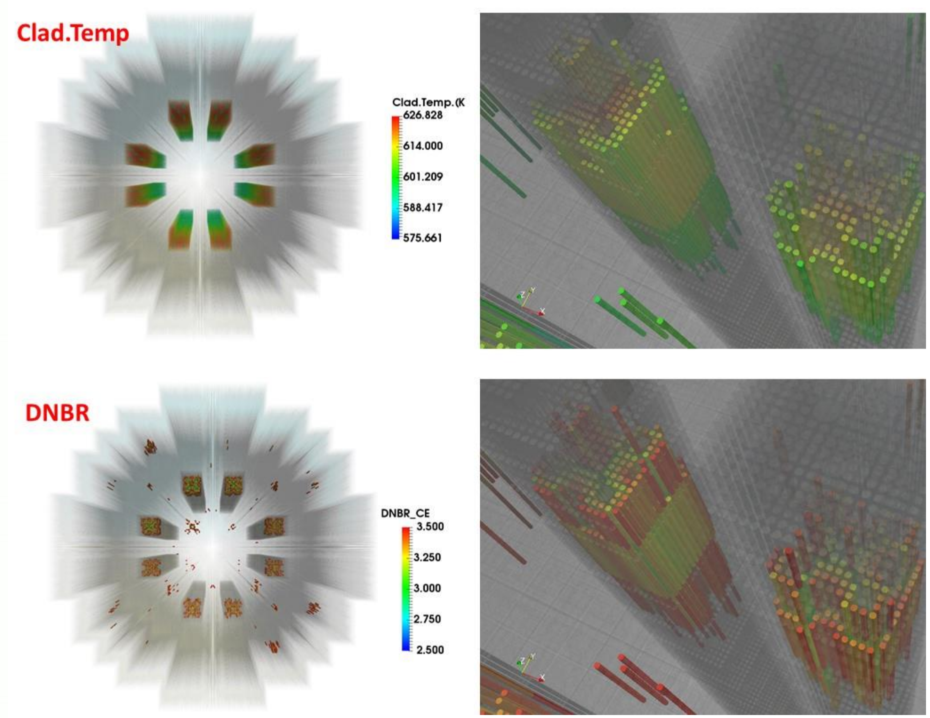
Time(sec)	Event
0.0	Steam line break occurs
18.4	**Overpower trip setpoint reached**
18.6	Turbine Trip
19.1	Rod begins to drop
31.0	Void begins to form in RV Upper P.
34.0	Low SG1 setpoint reached
35.1	MSIV1 closed

Performance	
Problem time	**100 sec**
Resources	Intel® Xeon® Gold 6230R CPU @ 2.10GHz
Number of Procs	300
Computing time	**120 min**

MSMP Simulation of OPR1000 SLB Accident

Fuel Rod Visualization (Clad temperature & DNBR)





Improvement of Safety Margin (DNBR)

Safety Margin in SLB Accident

- **Minimum DNBR in fuel assembly**

※ DNBR: Departure from Nucleate Boiling Ratio

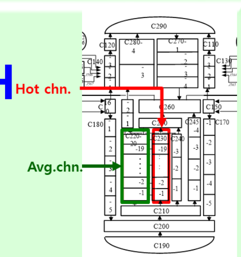
  - Key parameter to ensure safety margin for SLB accident

- **Enhancement of safety margin for MSMP approach**
  - 30% larger than 1D result

Methodology	MDNBR
1D System-scale TH	2.020
MSMP (without Turbulent mixing)	2.331
MSMP (with Turbulent mixing)	2.615

Enhancement of Safety Margin

1D System-scale TH
- Axial flow ONLY
- Hot pin assumption
- Point kinetics



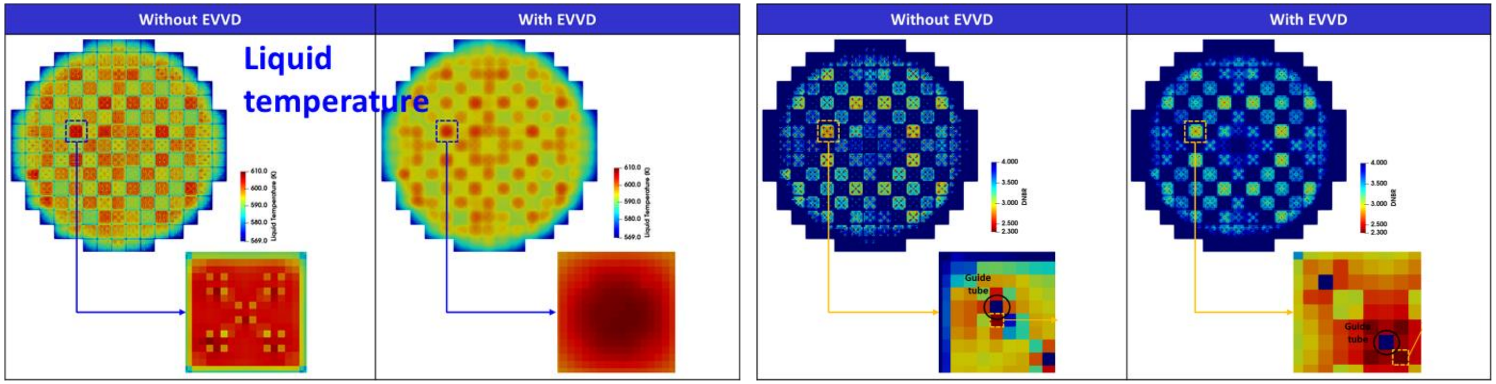
System TH nodal for PWR

3D Full core rod-wise MSMP
- Radial flow dispersion
- Pin-by-pin power distribution
- Channel-by-channel geometric parameters

Key parameters to enhance MDNBR in MSMP

Key parameter 1: 3D Coolant Flow

- **3D radial flow dispersion with turbulent mixing**
  - Impossible to consider radial flow mixing in 1D safety analysis
  - **3D Radial flow including turbulent mixing enhances coolability**
  - **Ensure additional safety margin**

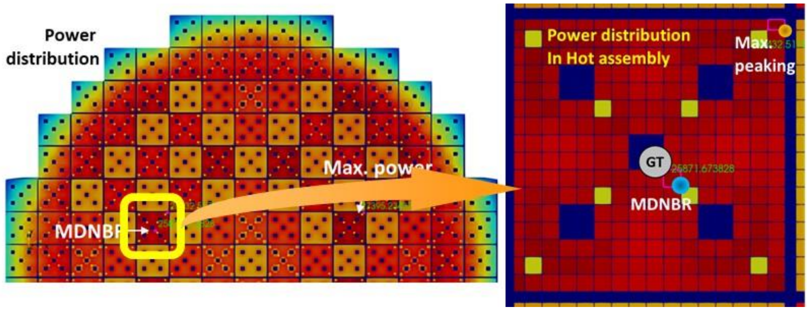


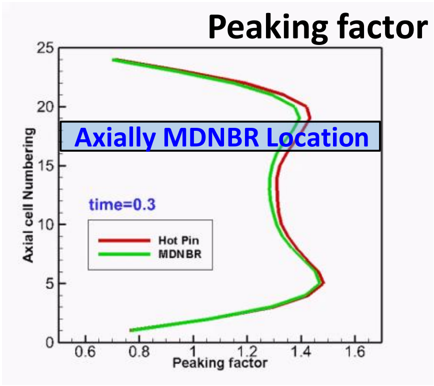
Key parameters to enhance MDNBR in MSMP

Key parameter 2: Realistic fuel power (with N/K code)

Power output of fuel assembly (pin-wise power)

- Co-simulation with N/K produces detailed rod-scale power
- MDNBR does not meet the Hot Pin assumption
  - ✓ 1D Safety analysis: Hot pin assumption to occur MDNBR
- Mitigate 1D conservative assumption
- **Ensure additional safety margin**





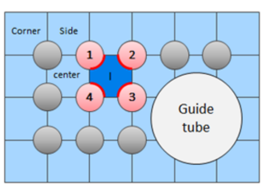
Key parameters to enhance MDNBR in MSMP

Key parameter 3: Non-identical geometric parameters

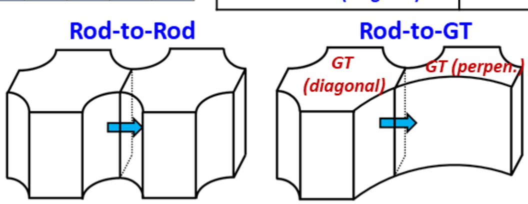
➢ Various subchannel information

- Subchannel-scale resolution yields various geometric parameters
- CHF can be evaluated according to subchannel type
- **Ensure additional safety margin**

Various heated diameters



Subchannel type	Heated dia. [m]
Rod-to-Rod	0.012637
1D System TH	0.013012
Rod-to-GT (perpendicular)	0.013855
Rod-to-GT (diagonal)	0.016922



MDNBR distribution for subchannel type

Subchannel type	Heated dia. [m]	K1	q''_{CHF}	MDNBR
Rod-to-Rod	0.012637			
1D System TH	0.013012			
Rod-to-GT (perpendicular)	0.013855			
Rod-to-GT (diagonal)	0.016922			

Full core Pin-wise Fuel Performance

4

- Pin-wise F/P code Coupling
- Evaluation of Pin-wise Fuel Performance in SLB Accident

Pin-wise Fuel Performance code Coupling (1/4)

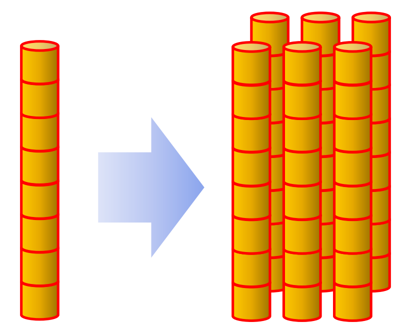
Fuel Performance Code

- **US NRC FRAPTRAN code**
  - Single fuel rod behavior
  - Coupled with system T/H code

How to Couple for Source Level Multiple Fuels

- **Mapping between fuel's cell and fluid cell**
  - **CUPID** (sub. T/H): Fuel nodes of Internal heat structure model
  - **FRAPTRAN** (F/P): Single-rod Fuel nodes
- **Extend fuel code for multiple fuels**
  - Extend coupling variables for multi-rods
    - ✓ Including modification of F/P code (variables' array)
  - Call fuel code as many as the number of fuels

```
DO i=1,Nrods
  Call FRAPTRAN(i)
ENDDO
```



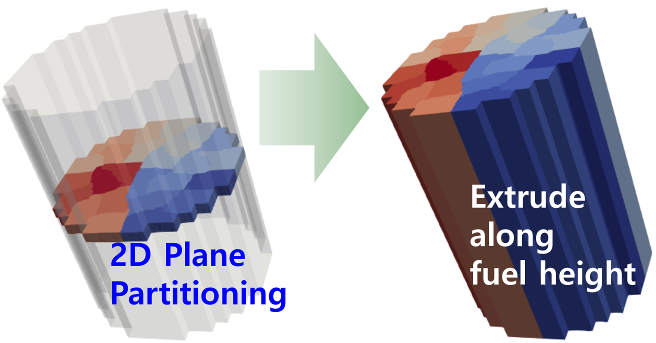
Single fuel
a(1:nz)

Multi fuels
a(1:nz,1:Nrod)

Pin-wise Fuel Performance code Coupling (2/4)

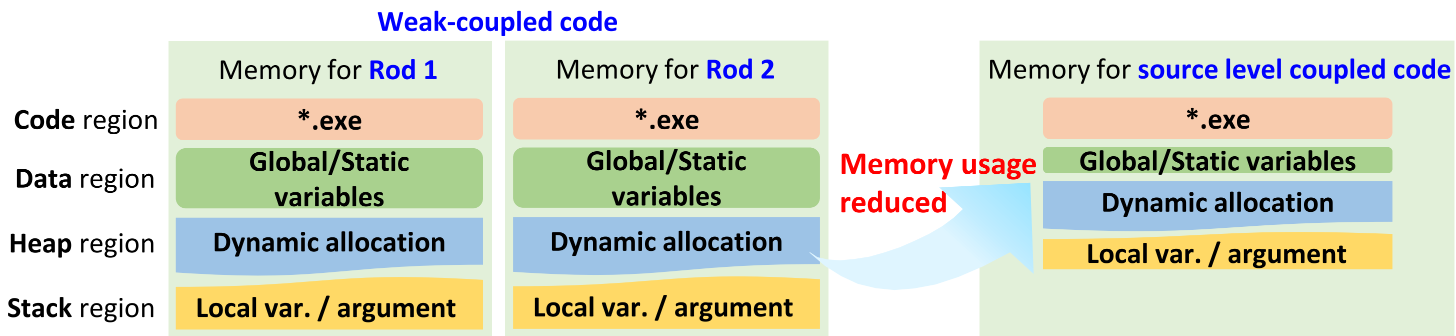
Parallel for Pin-wise Full Core Simulation

- **Prerequisite for simple mapping**
  - Single fuel is not partitioned
- **Domain partitioning by METIS**
  - Partitioned 2D plane
  - Extrude along fuel height



Parallel Computing in HPC Environment

- **SPMD (Single Program, Multiple Data)**
  - Source-to-source compilation
  - Efficient memory usage for massive multiple fuels calculation.

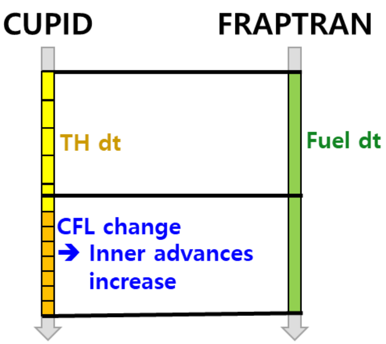


Pin-wise Fuel Performance code Coupling (3/4)

Time Advance

Time-step control between TH and Fuel code

- Time-marching scheme is implemented
  - ✓ Time marching between CUPID and FRAPTRAN
  - ✓ The number of inner advances determined
- Final time step determined by system T/H (MARS) and sub. T/H (CUPID)

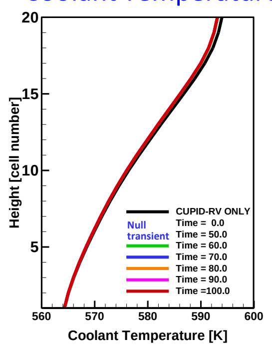


Verification(1) – 3x3 Fuel Rods

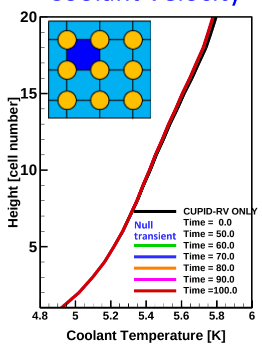
CUPID-RV/FRAPTRAN

- Fuel behavior from FRAPTRAN

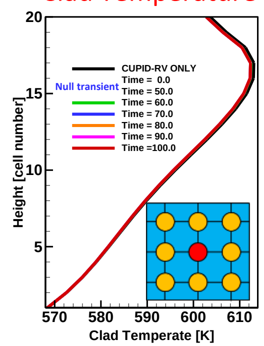
Coolant Temperature



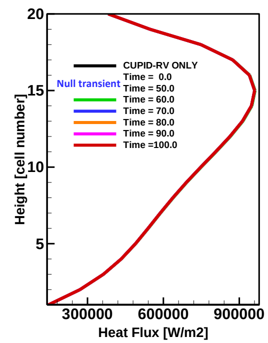
Coolant Velocity

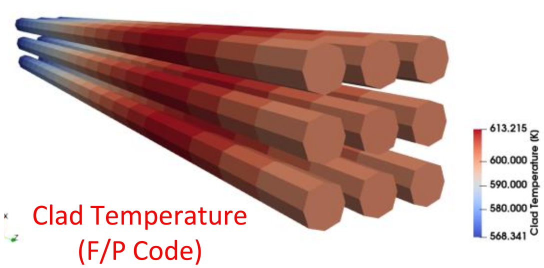


Clad Temperature



Heat Flux



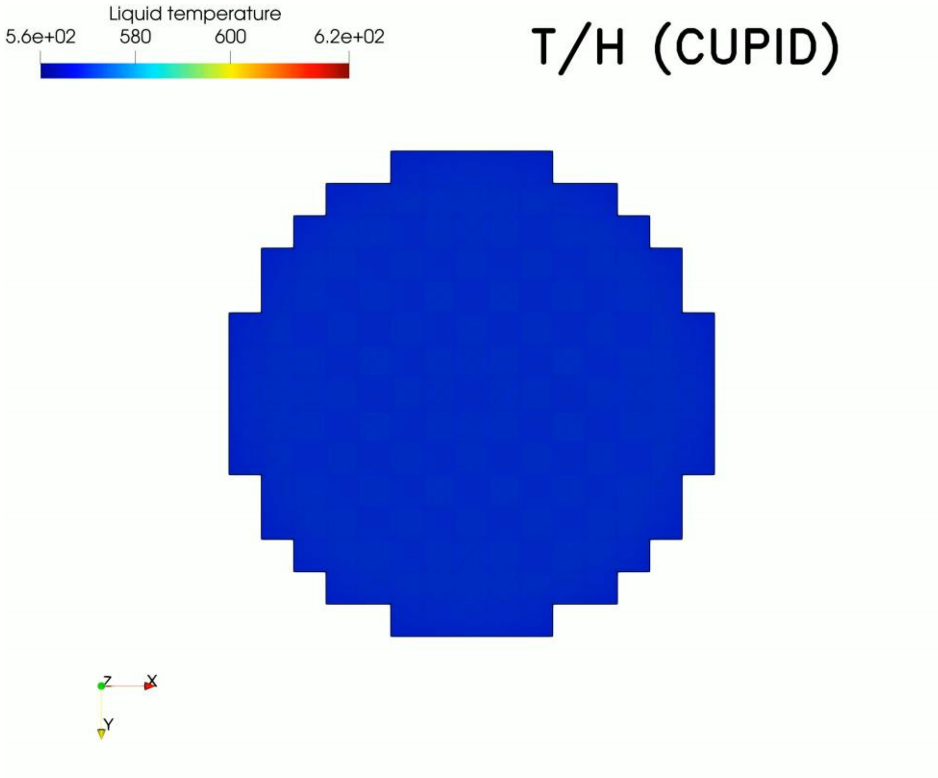


Clad Temperature (F/P Code)

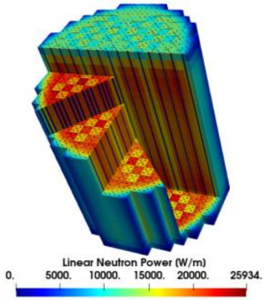
Pin-wise Fuel Performance code Coupling (4/4)

Verification(2) – LWR Full Core

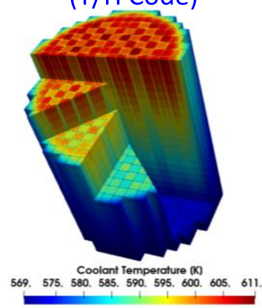
- Steady state of OPR1000 core region
- CUPID-RV/MASTER/FRAPTRAN
  - Sub. T/H & N/K & F/P coupled simulation



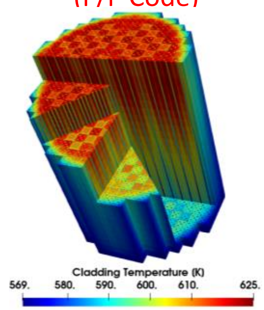
Neutron Power (N/K Code)



Coolant Temperature (T/H Code)



Clad Temperature (F/P Code)



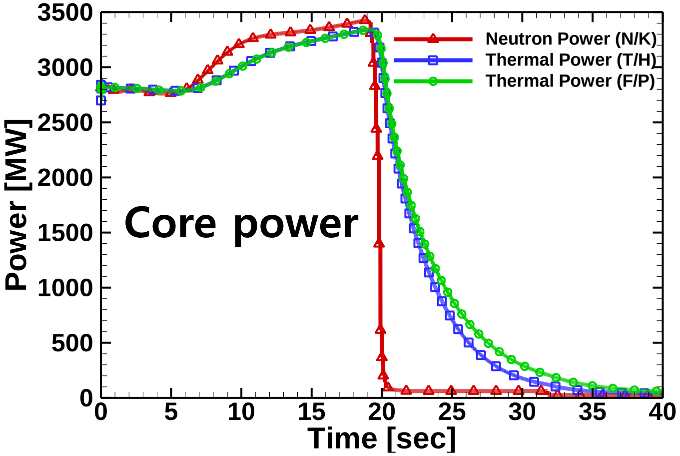
Evaluation of Pin-wise F/P in SLB Accident

Evaluation by Fuel code

Steam line break accident (SLB)

- Fuel behavior calculated by FRAPTRAN
- Radial conduction is dominant
- Similar pattern with simple heat structure model of T/H code
- MDNBR expected lower slightly

Methodology	MDNBR
1D System-scale TH	2.020
MSMP (w/o F/P)	2.615
MSMP (w/i F/P)	2.563



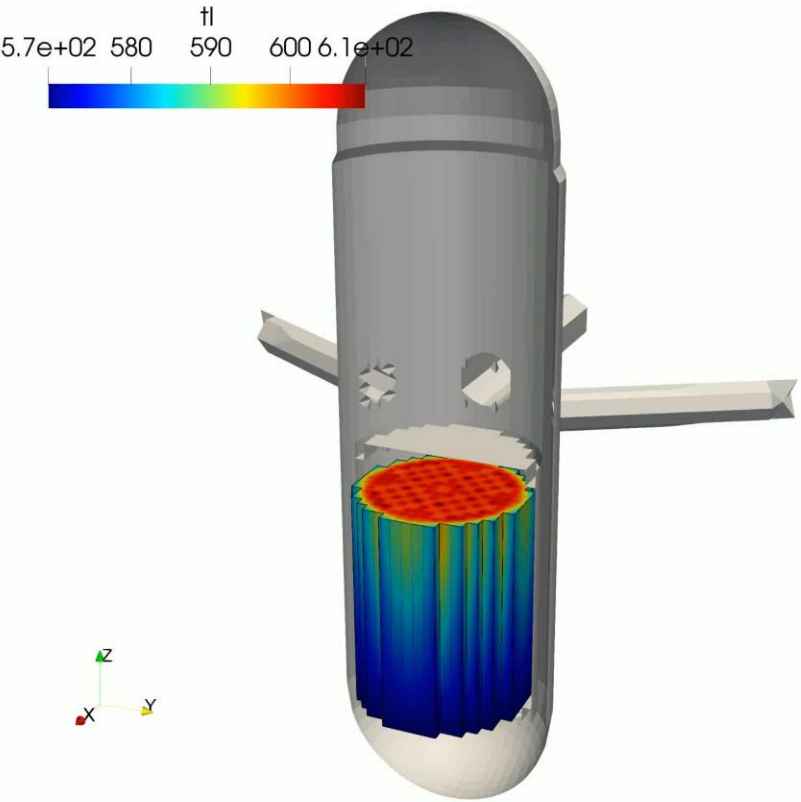
Capability for pin-wise fuel behavior

- Other accidents such as RIA can be accessed (On-going)
  - ✓ F/P code becomes considerably important in safety analysis for RIA accident

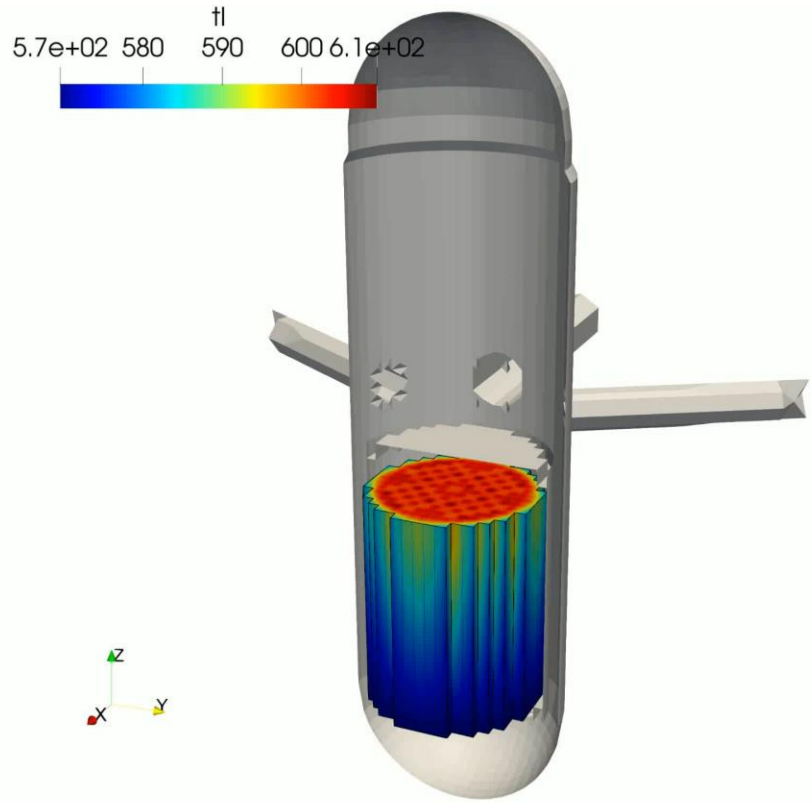
Evaluation of Pin-wise F/P in SLB Accident

Fuel Rod Visualization (Clad temperature & DNBR)

Without F/P code



With F/P code

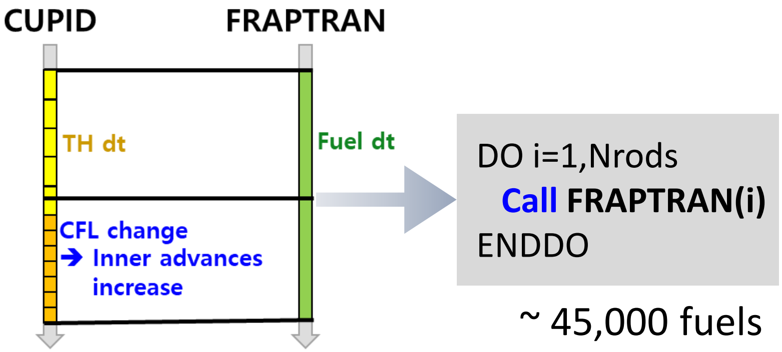


Evaluation of Pin-wise F/P in SLB Accident

Performance of F/P-Coupled Code

Pin-wise F/P calculation

- OPR1000 → about 45,000 fuels
- 45,000 F/P calls

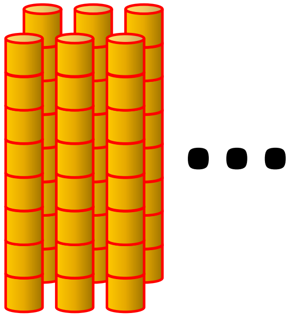


SLB accident safety analysis

- 33% increase of computing time
- Needs to optimize (On-going)

Performance	
Problem time	**100 sec**
Resources	Intel® Xeon® Gold 6230R CPU @ 2.10GHz
Number of Procs	300
Computing time	**160 min (w/I FRAPTRAN)**
	120 min (w/o FRAPTRAN)

~ 45,000 fuels



Multi fuels
a(1:nz,1:Nrod)

Summary

4

Summary

Summary

3D Safety Analysis for SLB

- MSMP (Multi-scale & Multi-Physics) approach
- Realistic RPV modeling with in-house mesh generator
- Platform for T/H & N/K & F/P code

Necessity for 3D MSMP Simulation

- Detailed Visualization inside of RPV
  - 3D Visualization of full core fuel power distribution
- Enhancement of safety margin
  - Realistic MDNBR evaluation
  - Improvement of MDNBR designed by 1D safety analysis
- Pin-wise fuel evaluation by F/P code coupling
  - Extend F/P code to pin-wise full core fuel analysis of LWR
  - Qualitatively reasonable fuel behavior
  - Should be applied to RIA

THANK YOU

jrlee@kaeri.re.kr