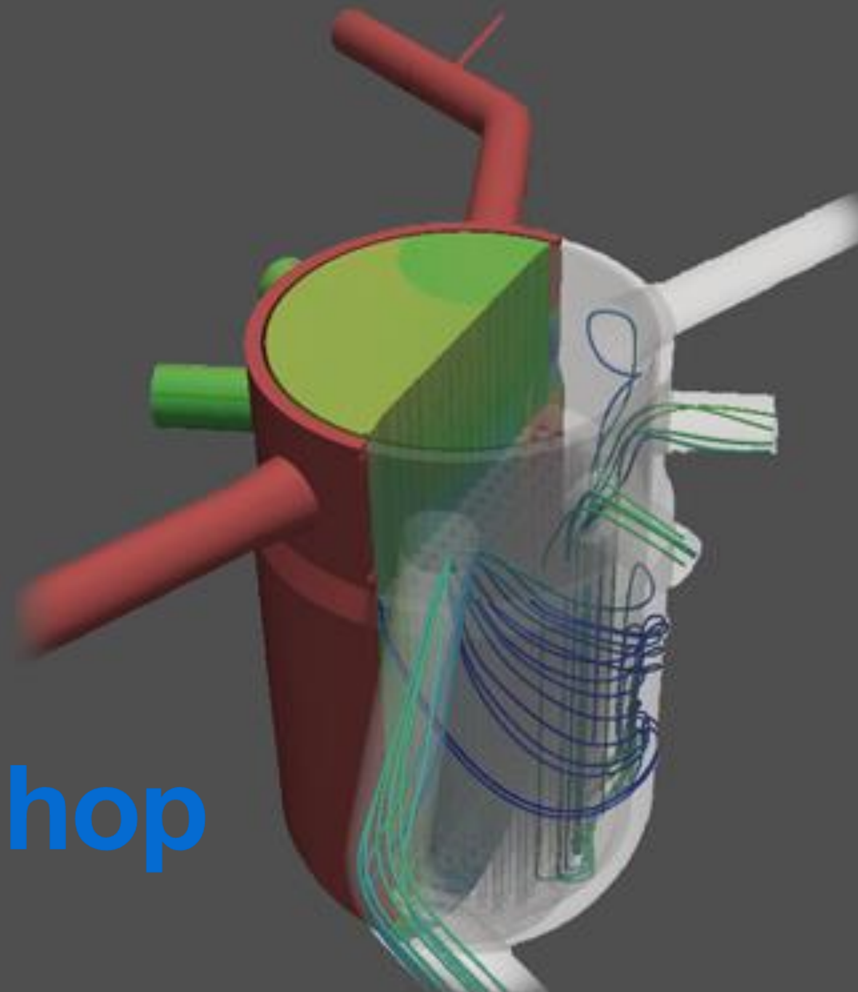
목　　차



10th CUPID Workshop

CUPID2.6 개요

윤 한 영

2022년 8월 23일

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

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**

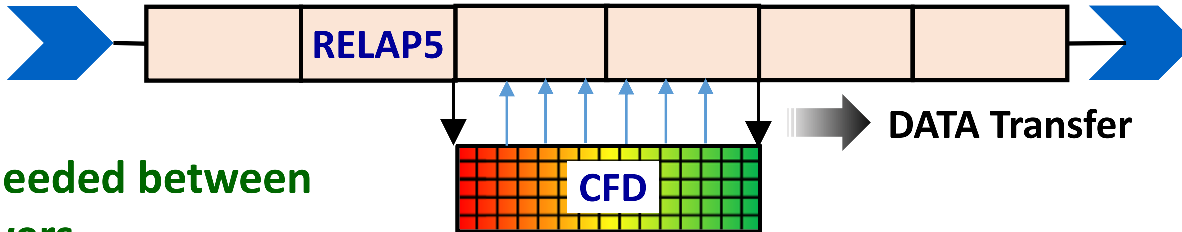
- **Development of a *Geometric Multi-Grid (GMG)* solver for *unstructured mesh***
  - CG solver: $\text{Time}_{CG} \propto N^{1.4}$
  - GMG solver: $\text{Time}_{GMG} \propto N^{1.0}$
  - The new GMG solver is *Easy to use* since the unstructured *coarse meshes are generated automatically*

Number of Cells	CPU Time (s)	
	CG	GMG
10^2	1	1
10^6	3.7 months	2.7 hours

Multi-Scale Simulation for a Fast Transient

Domain Overlapping

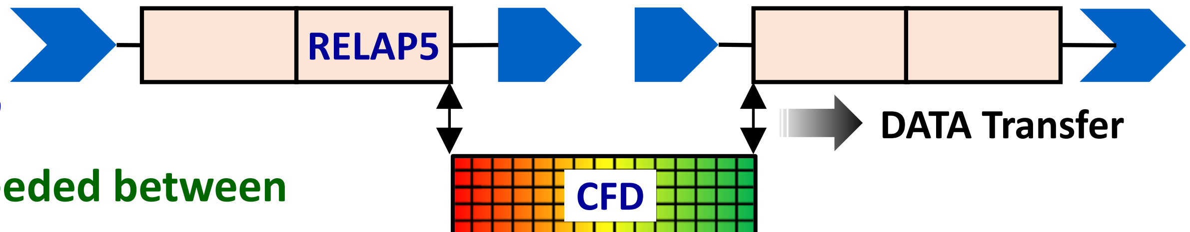
Data Transfer is needed between Two separate solvers



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

Domain Decomposition

Data Transfer is needed between Two separate solvers

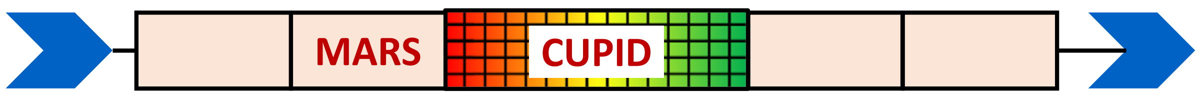


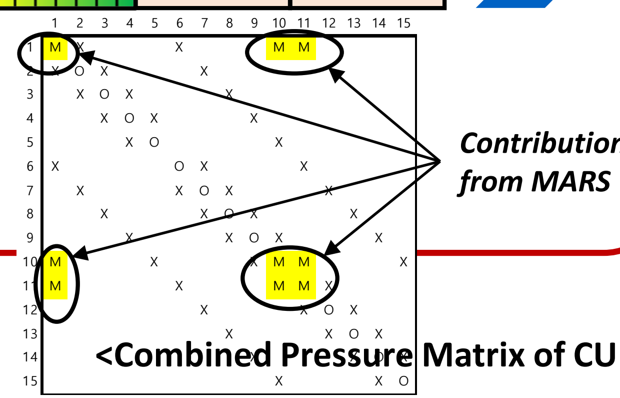
Single Domain

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.*

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***

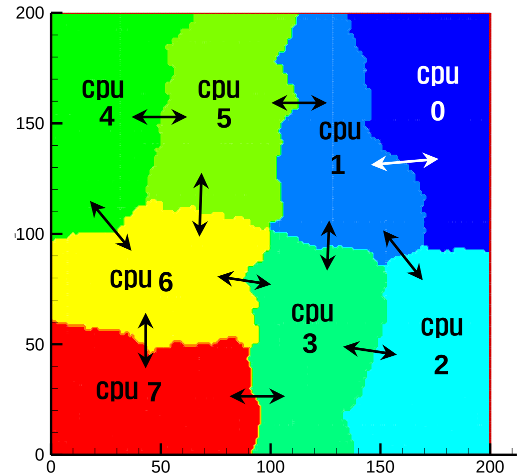
Numerical Models

2

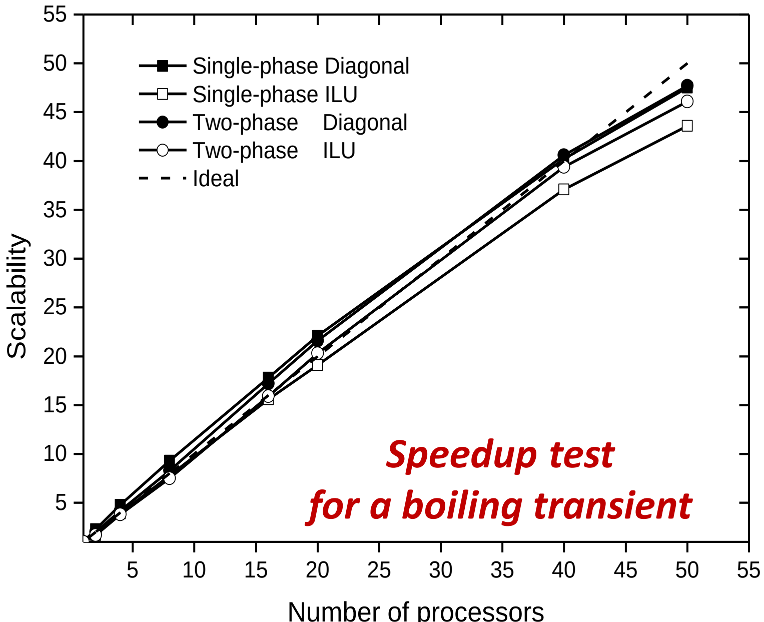
- Mathematical Models
- Parallel Computation
- Pre/Post Processors

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

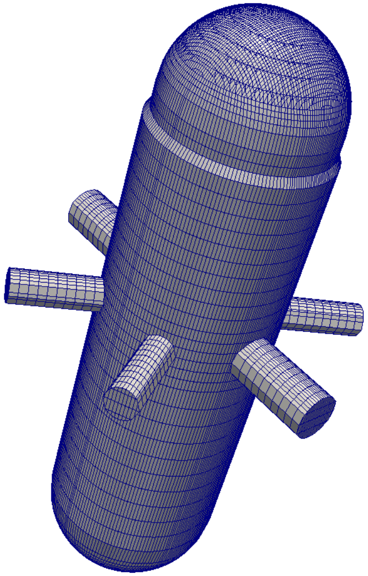
Mesh Generation

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

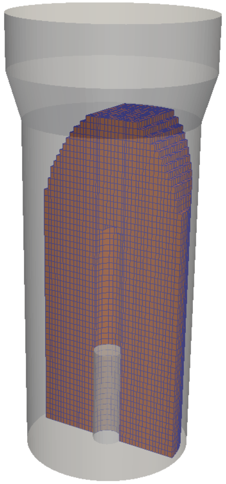
Post Processing

- Open Source Program: *Paraview*

RVMesh-3D



CUPID-SGP



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

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

CUPID/MASTER

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

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

CUPID-RV
CUPID-SG

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

CUPID-CT

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

CUPID/MARS/MASTER/FRAPTRAN

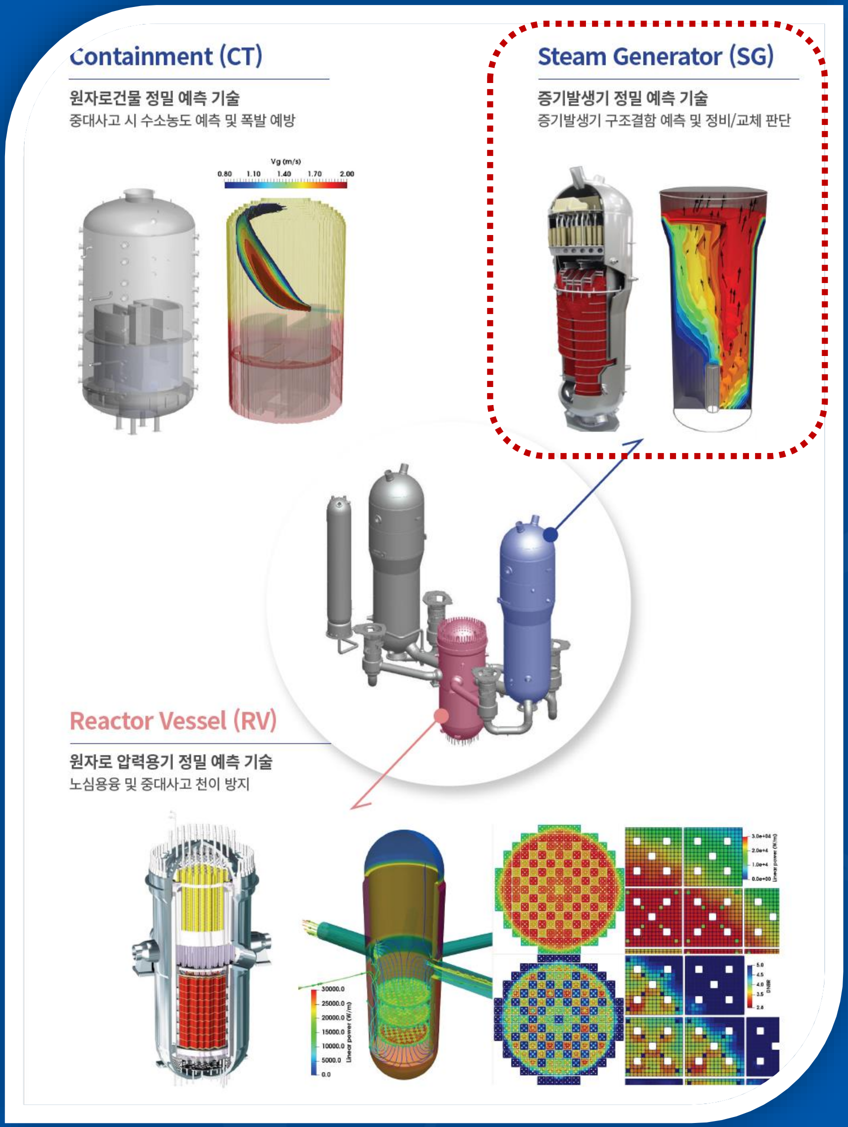
- Sub-channel models (13 cases)
- Multi-physics (CUPID/Fuel) coupling (2 cases)

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*)**

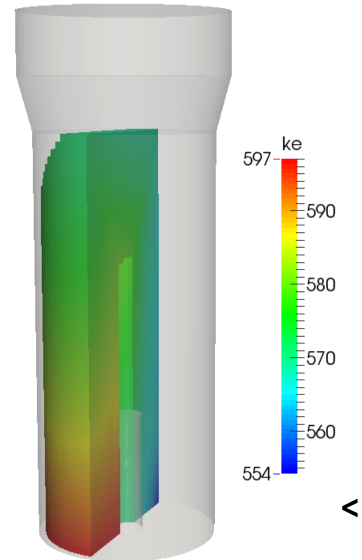
CUPID-SG

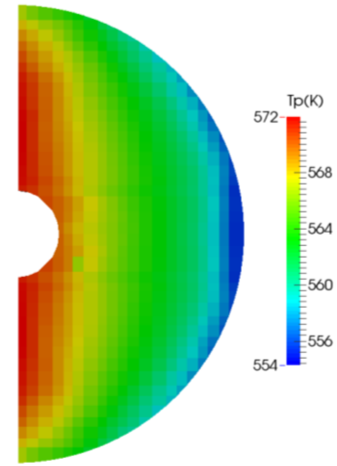

Containment (CT)
원자로건물 정밀 예측 기술
중대사고 시 수소농도 예측 및 폭발 예방
Steam Generator (SG)
증기발생기 정밀 예측 기술
증기발생기 구조결함 예측 및 정비/교체 판단
Reactor Vessel (RV)
원자로 압력용기 정밀 예측 기술
노심용융 및 중대사고 천이 방지

4

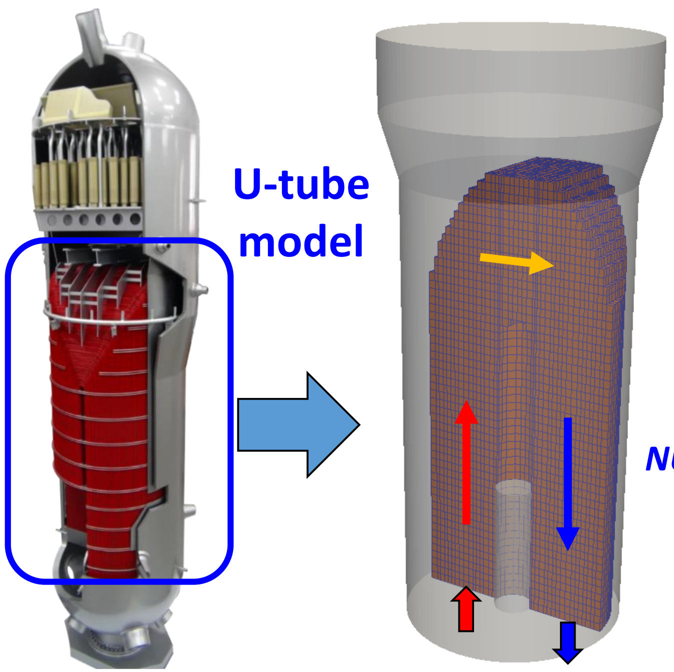
Steam Generator Analysis Code

- **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**

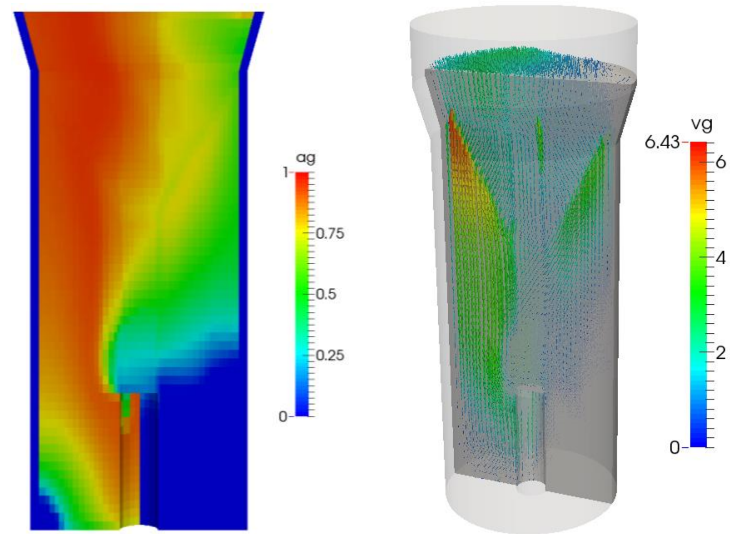




<Primary Coolant Temperature>



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



<Secondary Side Void Fraction and Velocity Vectors>

SG U-tube Model

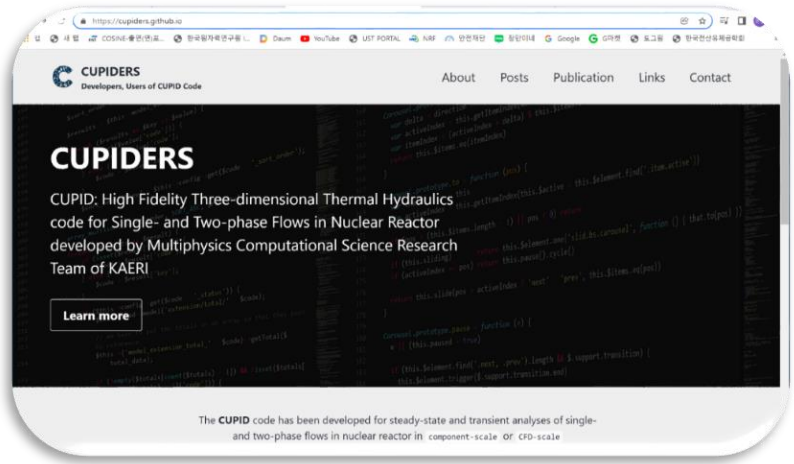
	Coarse mesh (29,780)	**Fine mesh (121,988)**	**Design value**
Number of U-tubes			13,102
Number of U-tube groups (ngroup)	**164**	**640**	
Total U-tube length	**256,996 m**	**256,100 m**	254060 m
Total U-tube heat transfer area	**15,380 m^2**	**15326 m^2**	15205 m^2
$error\left(1-\frac{Length\_tube_{mesh}}{Length\_tube_{design}}\right)$	**0.011**	**0.008**	
$error\left(1-\frac{HT\_Area_{mesh}}{HT\_Area_{design}}\right)$	**0.011**	**0.008**	

CUPID User Group

5

CUPID User Group

http://cupiders.github.io



Domestic Univ. & Research Inst.

Industries

2022년 기술실시
갱신 기관

Foreign Univ.

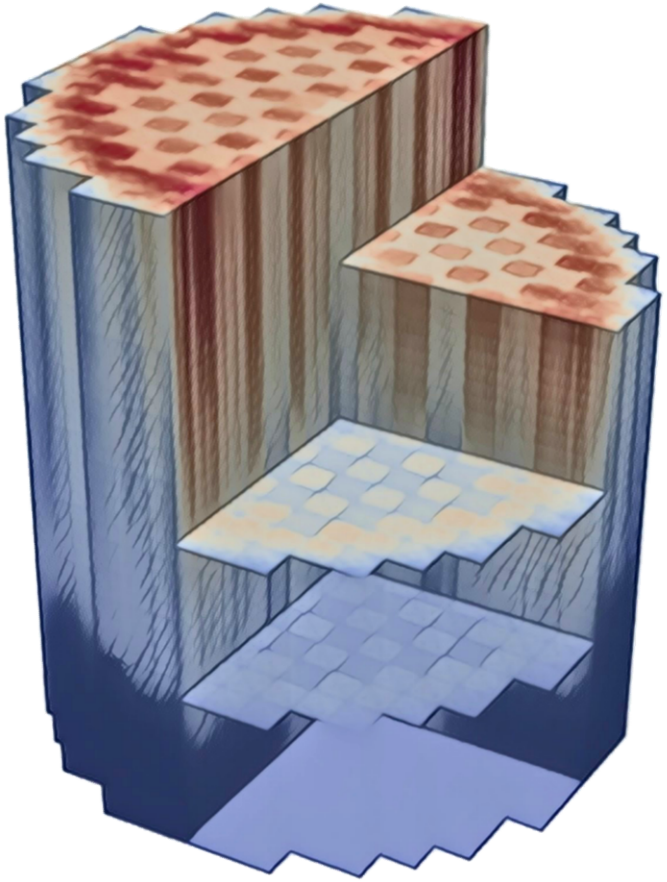
User Agreement

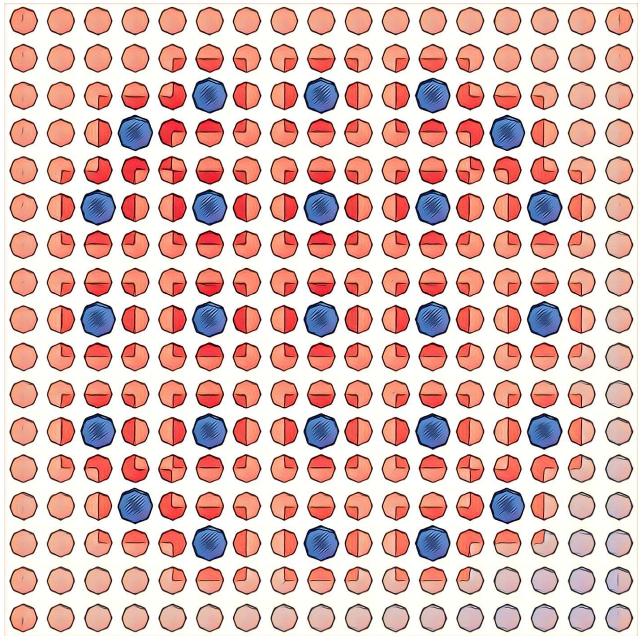
User Contract

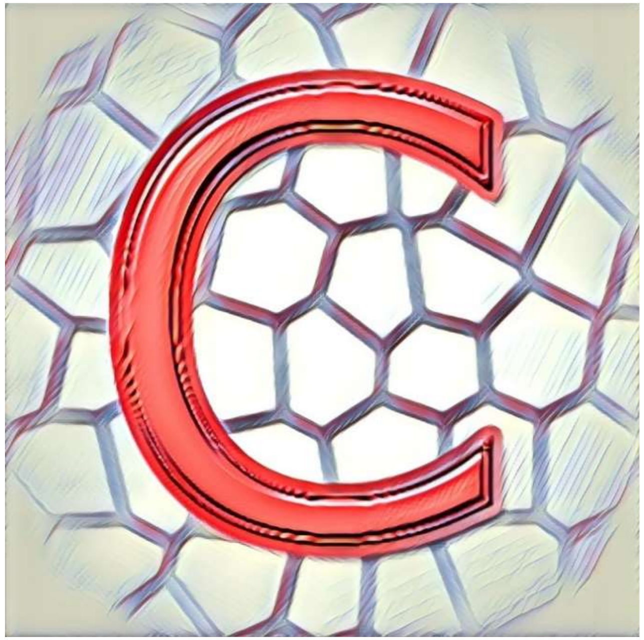
User Agreement

THANK YOU

hyyoon@kaeri.re.kr







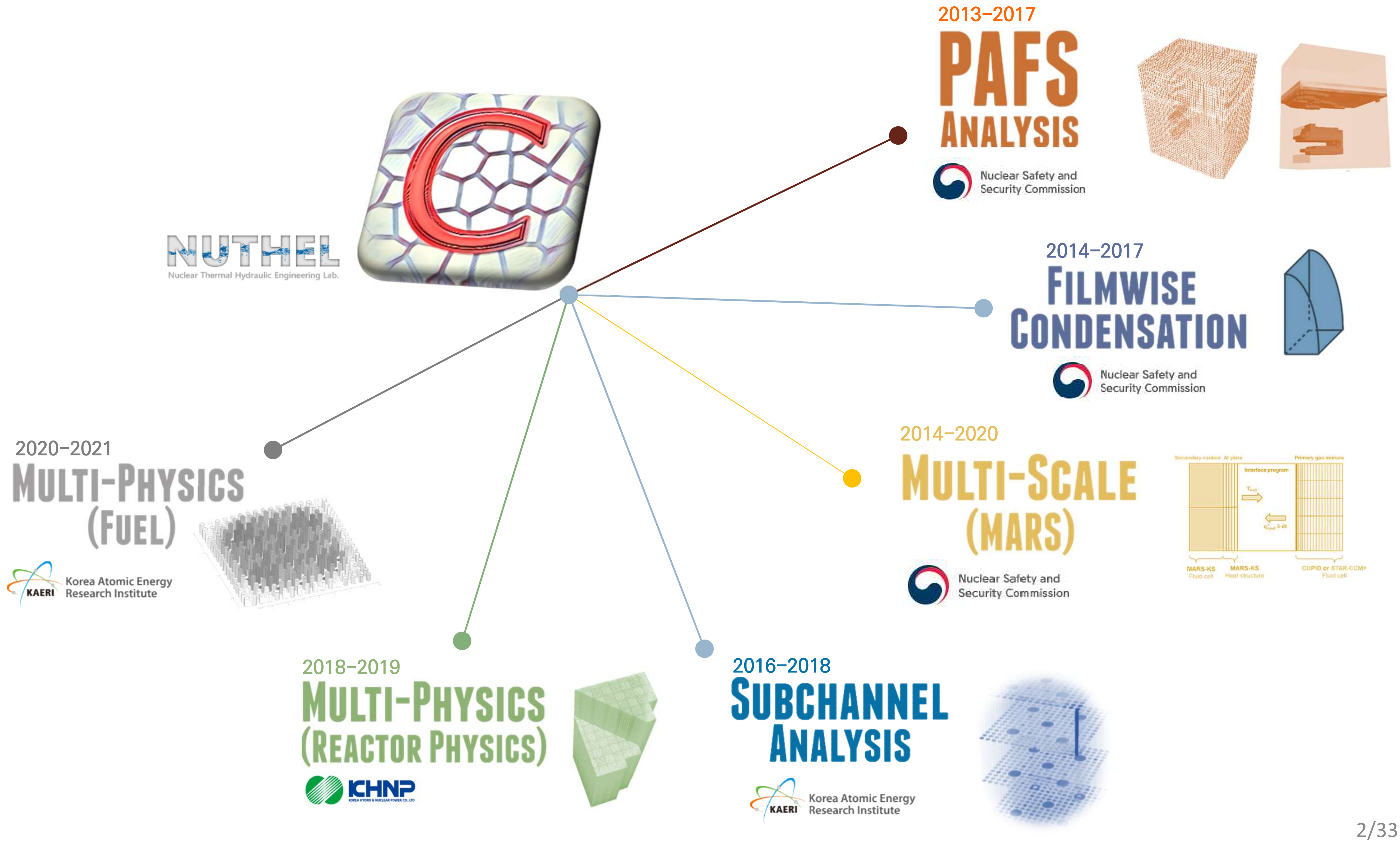
10th CUPIDERS Workshop

SNU-DNE-NUTHEL CUPID 활용 연구 현황 및 계획

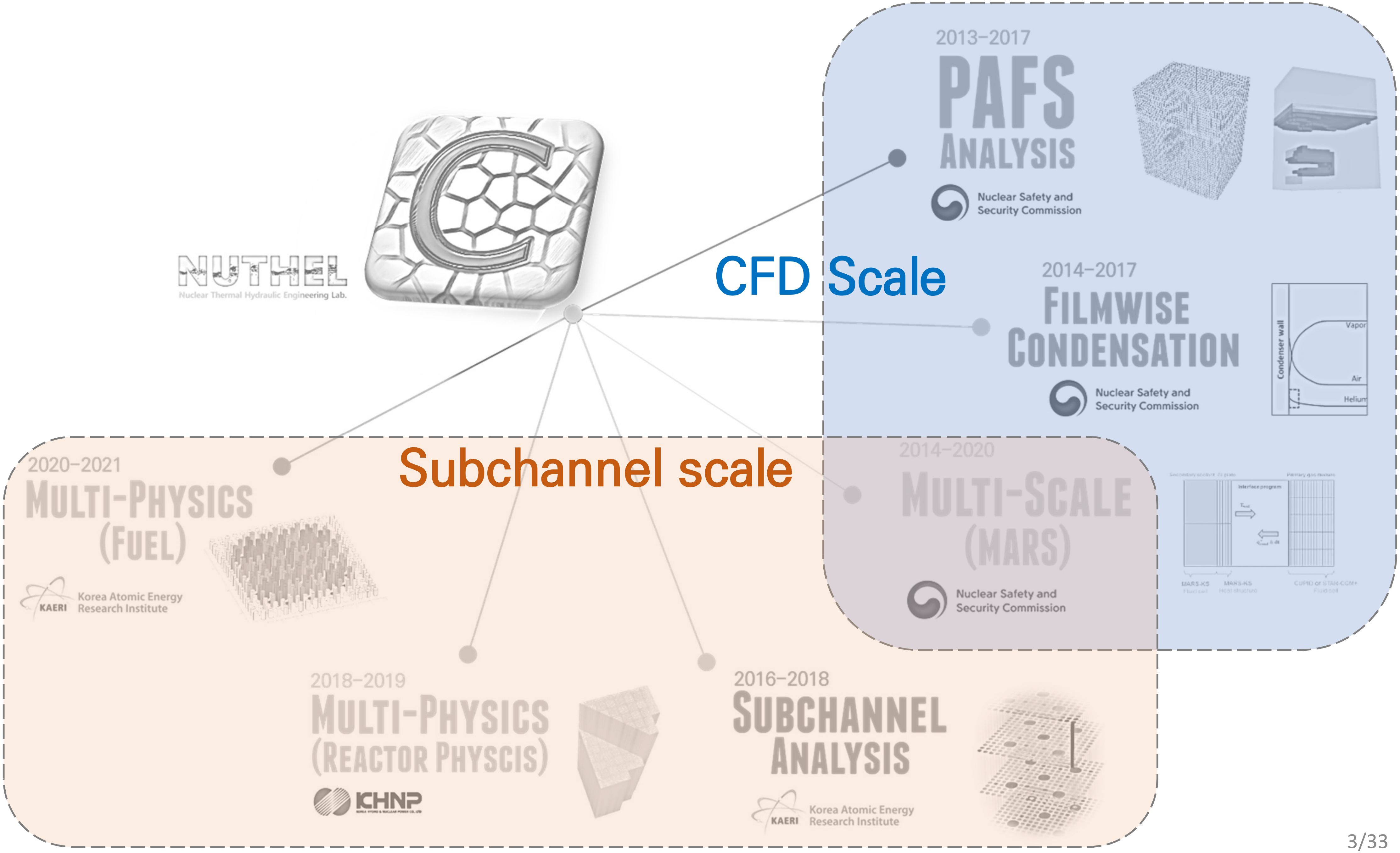
CUPIDERS Workshop, INTEC, KAERI, | 2022. 08. 23 | 서울대학교 원자핵공학과 | 조형규

NUTHEL
Nuclear Thermal Hydraulic Engineering Lab.

CUPID 활용 연구 이력 (2013-)



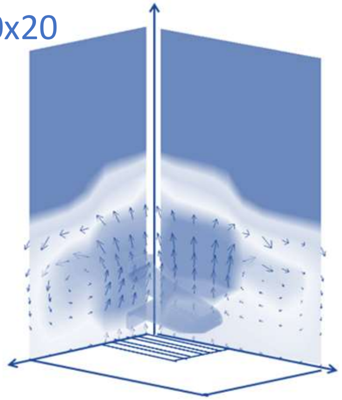
CFD & Subchannel analysis



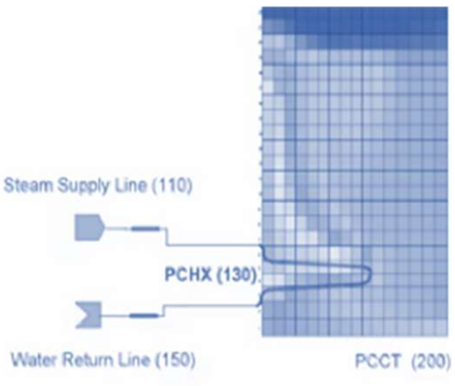
PAFS analysis using STH codes

MARS-MULTID (FNC)

APR+, 8x10x20



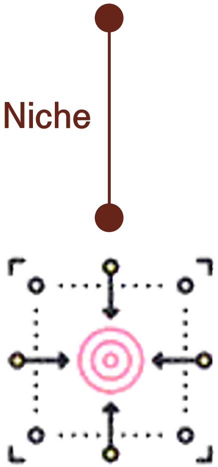
TRACE, PASCAL, 1x16x22 (KINS)



Turbulence

Coarse mesh

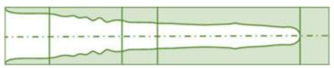
Niche



PAFS analysis using CFD codes

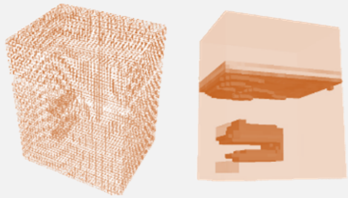
Two-phase models

In-tube condensation



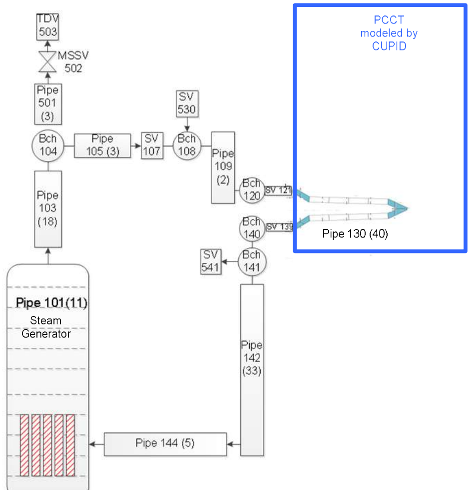
Computational cost

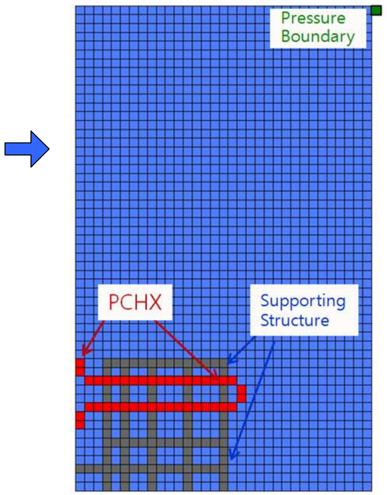
(Choi et al., KNS Autumn meeting, 2011)

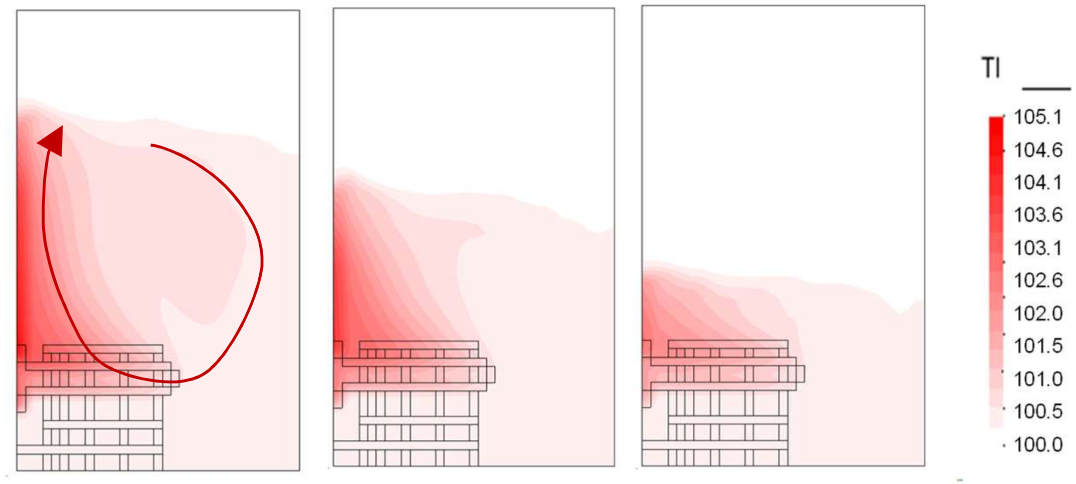


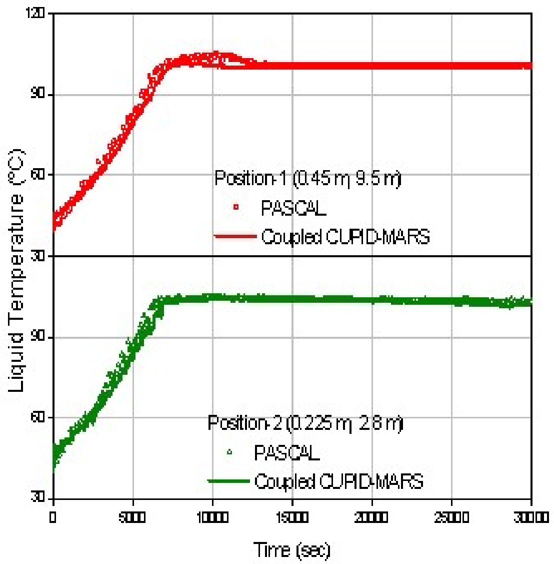
Cho, H.K. et al., Heat structure coupling of CUPID and MARS for the multi-scale simulation of the passive auxiliary feedwater system, NED2014.

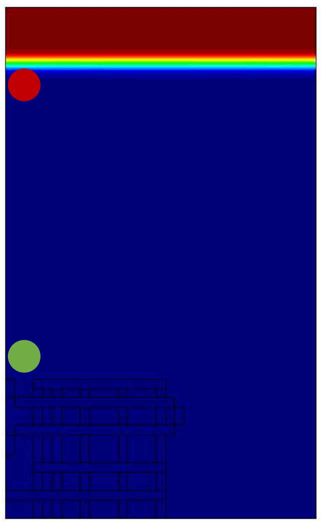
PASCAL analysis using CUPID−MARS (transient time: 28800 s)

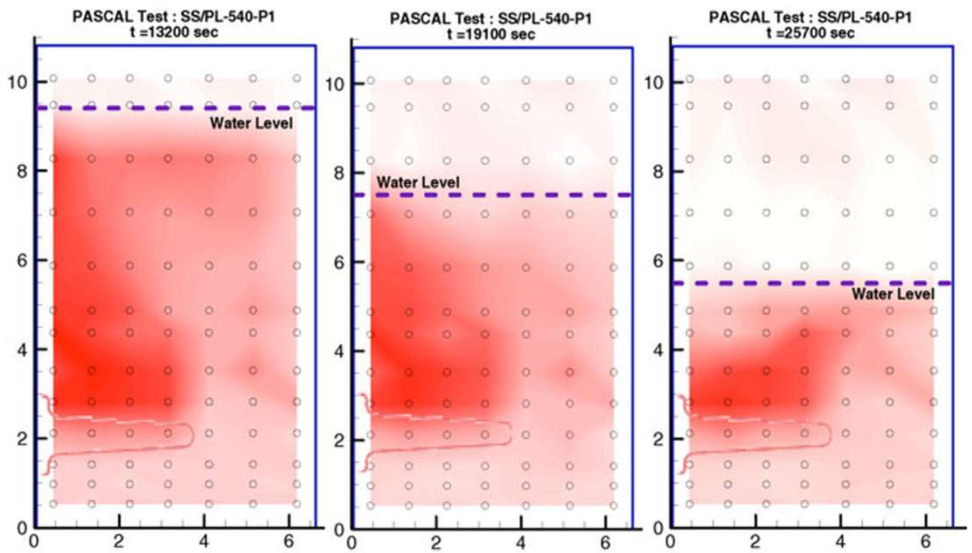






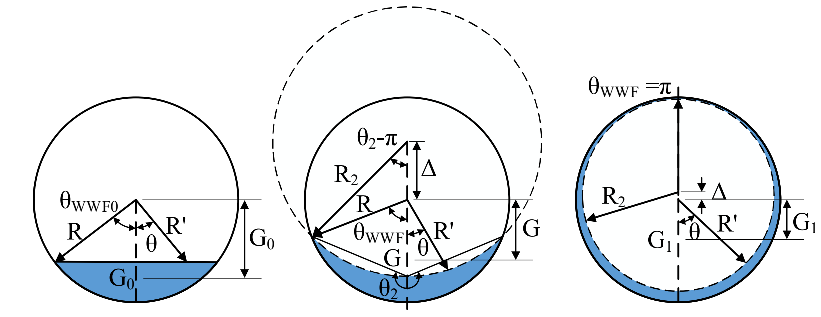






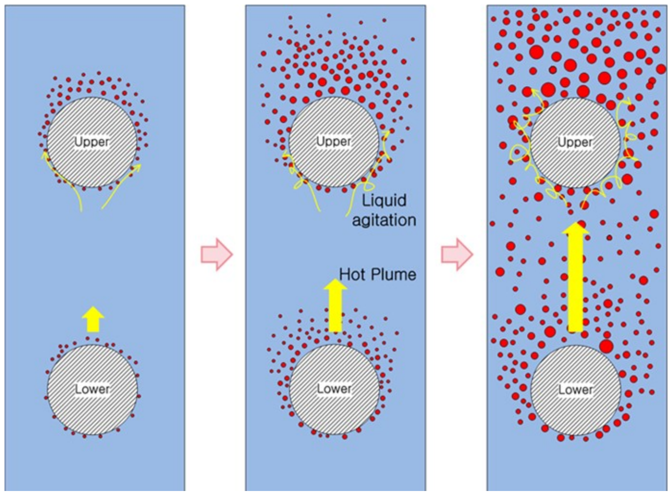
PASCAL analysis using CUPID-MARS

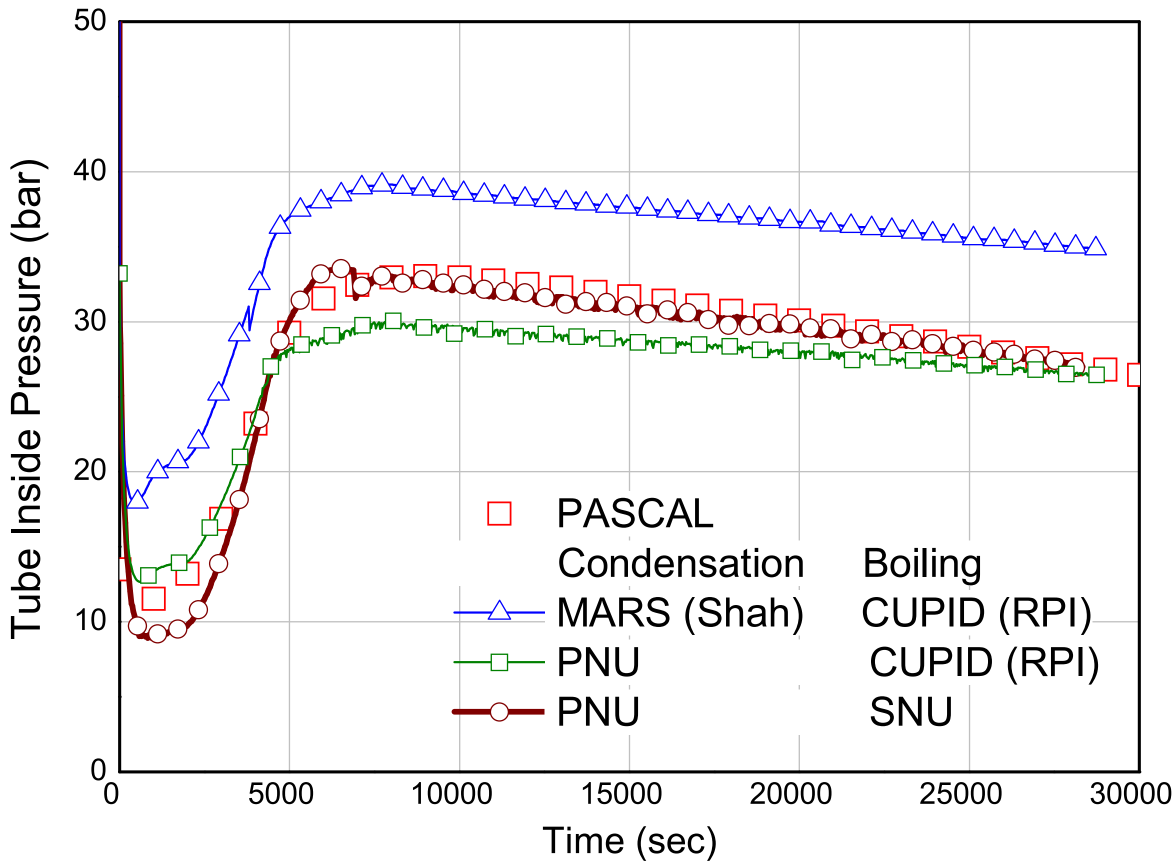
Condensation model: PNU model (Ahn, PhD thesis, 2018)

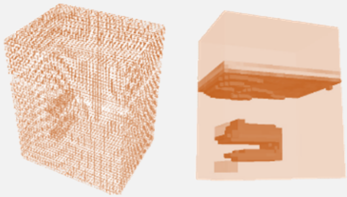


(a) Stratified-smooth flow (b) Stratified-wavy flow (c) Annular flow

Boiling model: SNU model (Jeon, PhD thesis, 2015)

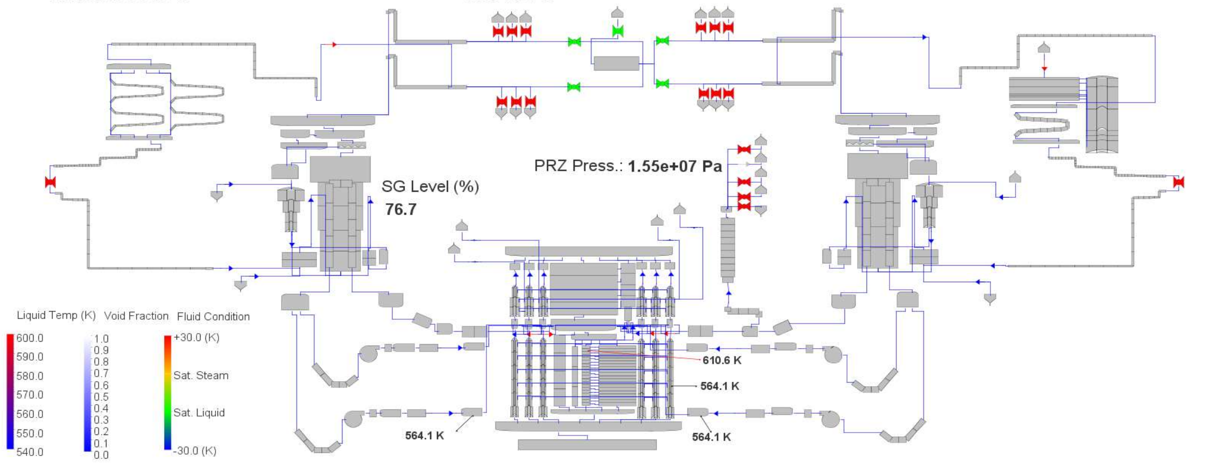


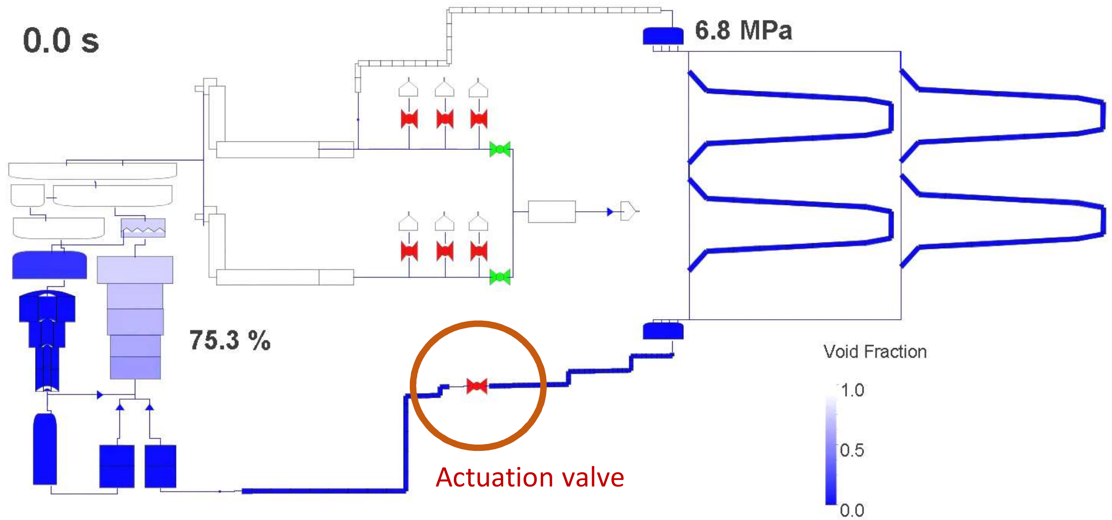




APR+ FLB analysis using CUPID-MARS

MARS nodalization for APR+ FLB

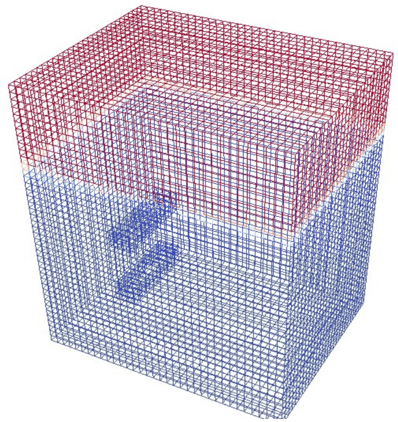




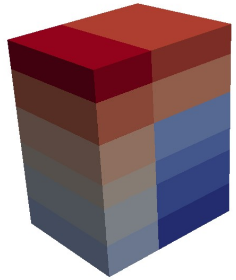
CUPID mesh for APR+ PCCT

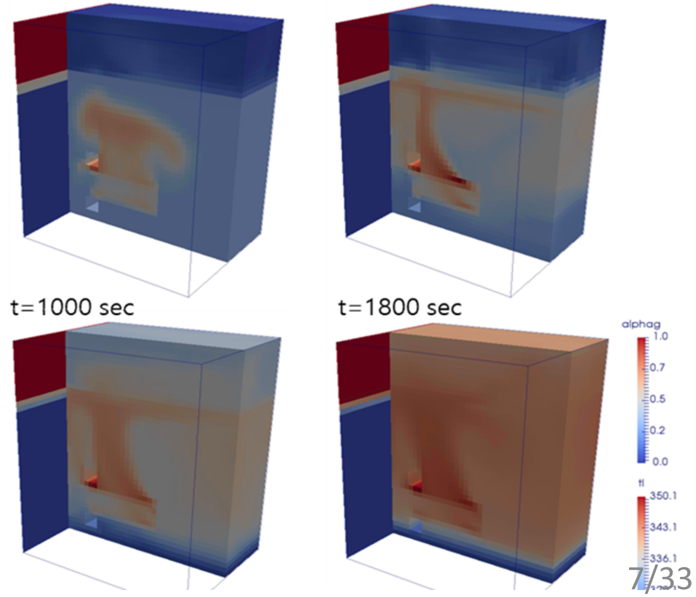
Nmesh: 38,340

Provided by S.J. Lee (KAERI)

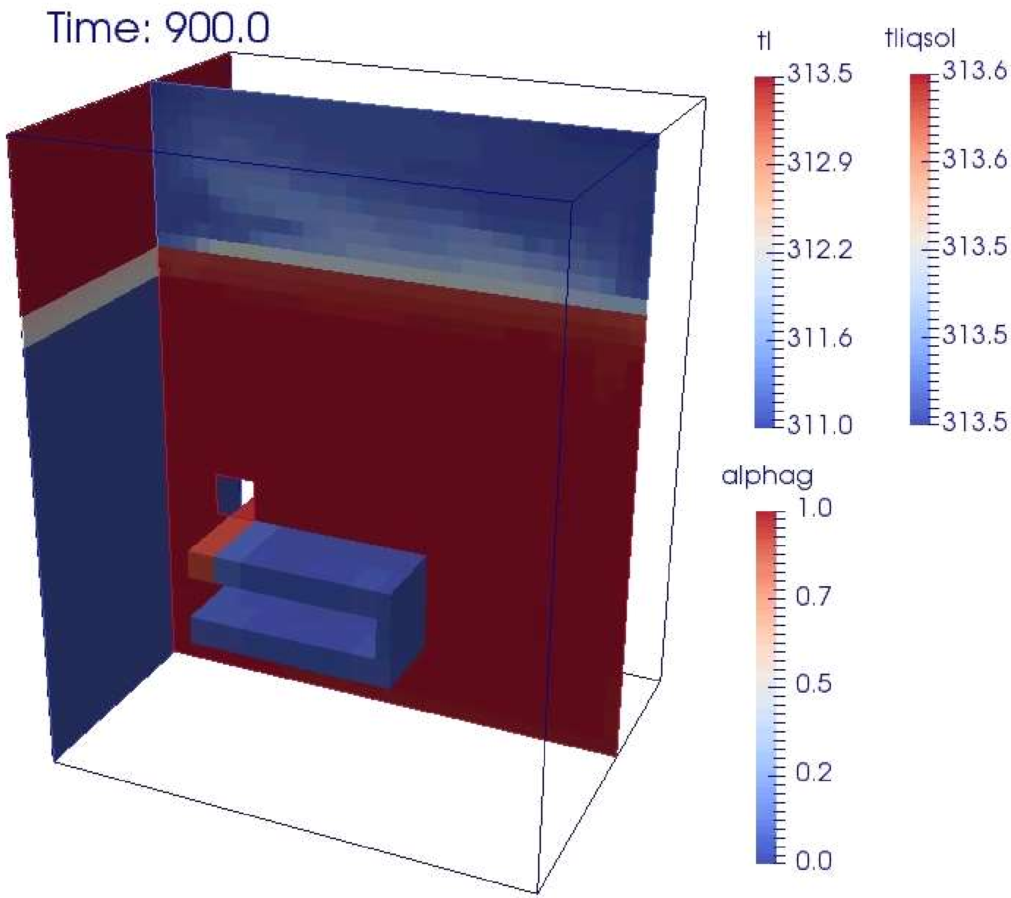


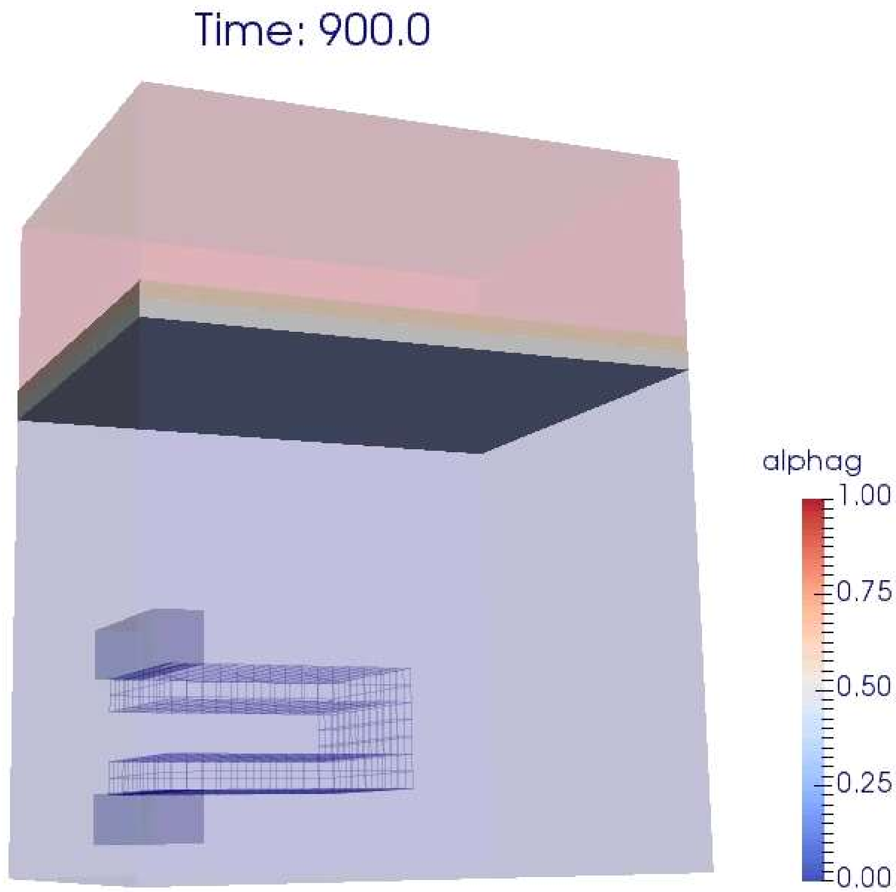
Domain decomposition





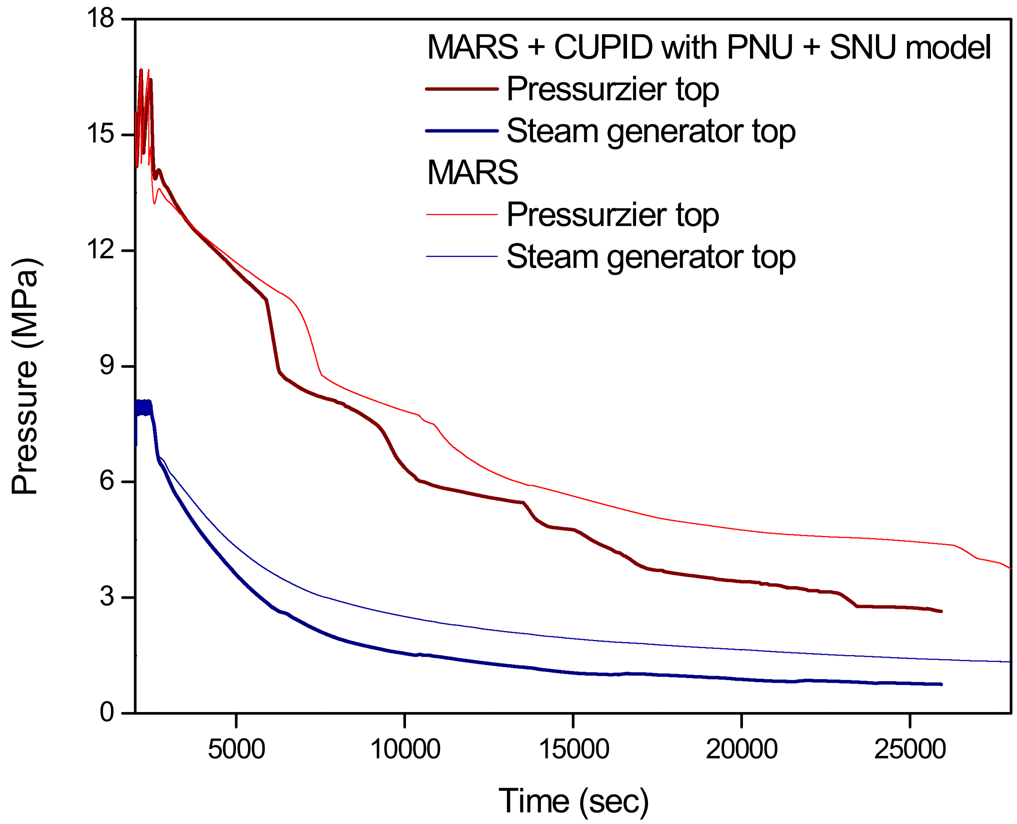
APR+ FLB analysis using CUPID-MARS

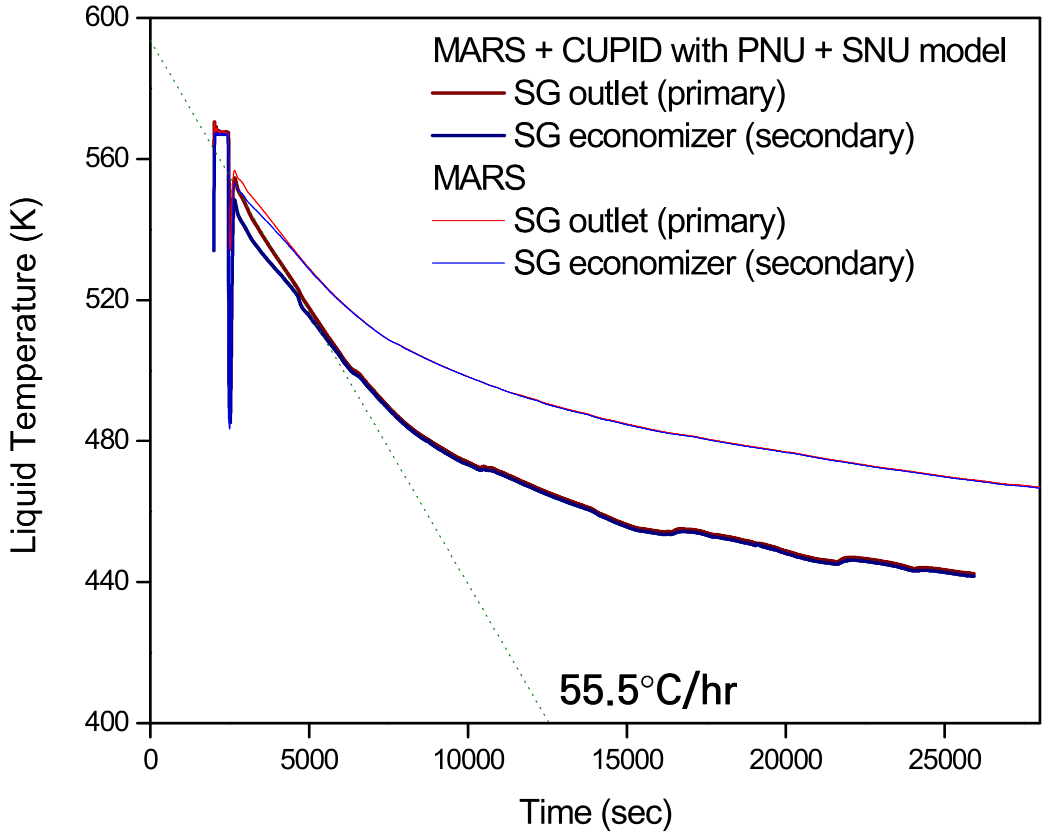




PAFS ANALYSIS

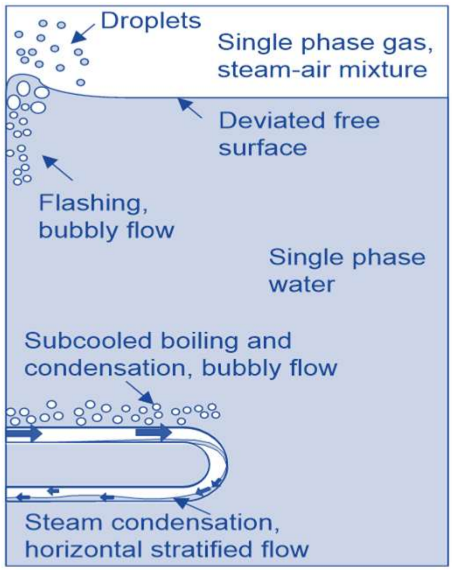
APR+ FLB analysis using CUPID-MARS





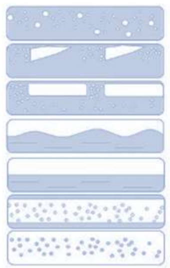
APR+ FLB analysis using CUPID-MARS

Challenges



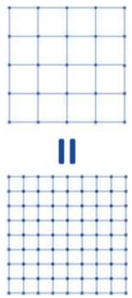
Turbulence model for separated two-phase flow

- Zero equation model (O)
- Two-equation model (X)



Flow regime map for large water pool

- Interface topology (O)
- IATE (X)



Mesh convergence

- Difficult to get converged solutions

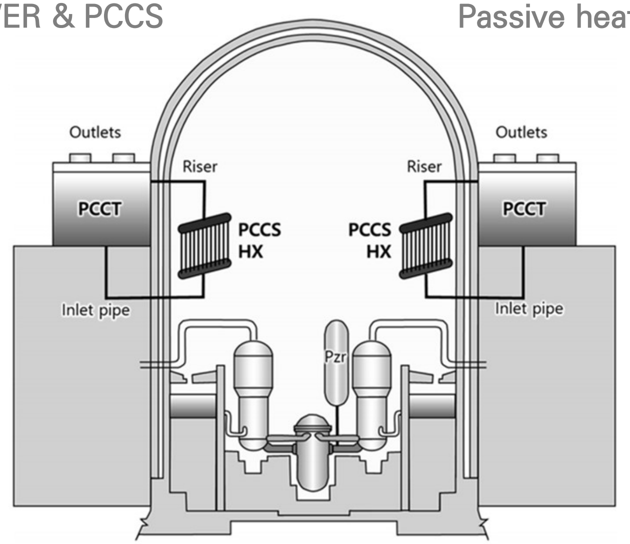
Constitutive relations

- No available experimental data for validation

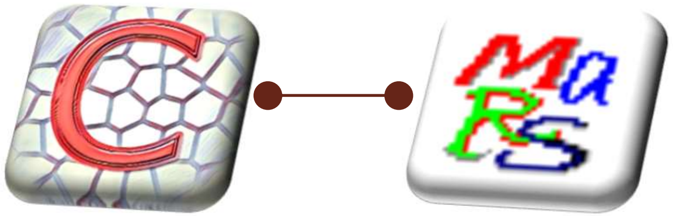
FILMWISE CONDENSATION

iPOWER & PCCS

Passive heat sinks



Multi-scale approach



Filmwise condensation model in the presence of NC gases

Single-phase approach

- Simple and efficient
- Not applicable to steam rich condition
- No slip condition for gas-liquid interface

Two-phase approach

- Wide applicability
  - ✓ Steam rich condition
  - ✓ Tall condenser wall
- Unstable
- Hard to handle condensate

Single-phase + film

- Less close to the real situation
- Steam rich condition
- Tall condenser wall
- Stable
- Easy to handle condensate
- Cannot handle thick film

Lee, J.H. et al., Improvement of cupid code for simulating filmwise steam condensation in the presence of noncondensable gases, NET, 2015.
Lee, J.H. et al., Simulation of wall film condensation with non-condensable gases using wall function approach in component thermal hydraulic analysis code CUPID, JMST, 2018

Filmwise condensation model for CUPID (modified Colburn–Haugen model)

- Species diffusion terms

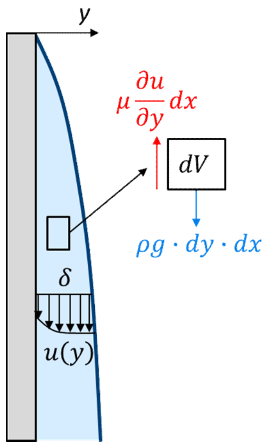
$$\underline{\nabla\cdot\left(\alpha_g\rho_g D\nabla X_n\right)} \qquad \underline{\nabla\cdot\left[\alpha_g\rho_g D(h_{nc}-h_v)\nabla X_n\right]}$$

- Heat balance on interface

$$H_{GI}(T_G-T_I)+m''h_{fg}-\frac{k_F}{\delta}(T_I-T_W)=0$$

- Liquid film equation

$$\frac{d}{dy}\left[\nu_l+E\right]\frac{dU_l}{dy}-\frac{1}{\rho_l}\frac{dP}{dz}+g\sin\theta=0$$



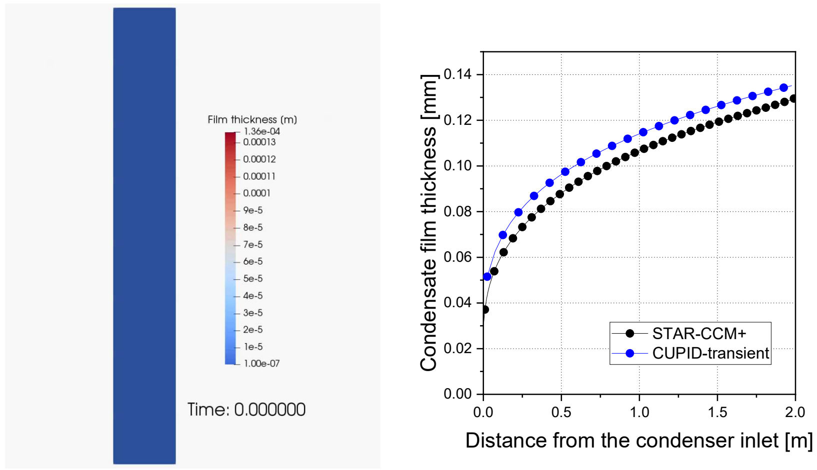
Sub-grid(wall film flow)

- Condensate film velocity profile
- Interface velocity

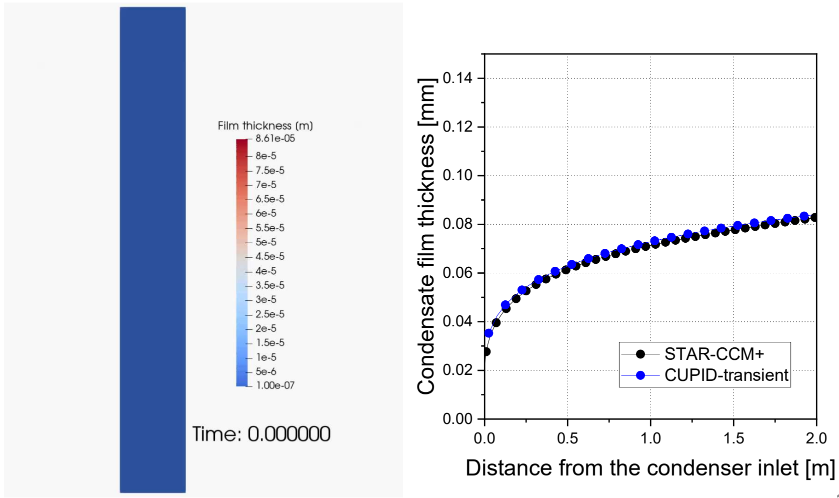
Fluid cell (gas mixture flow)

- Condensation rate
- Gas velocity & interfacial shear

P20-T50-V30-H65 (Forced convective condensation)



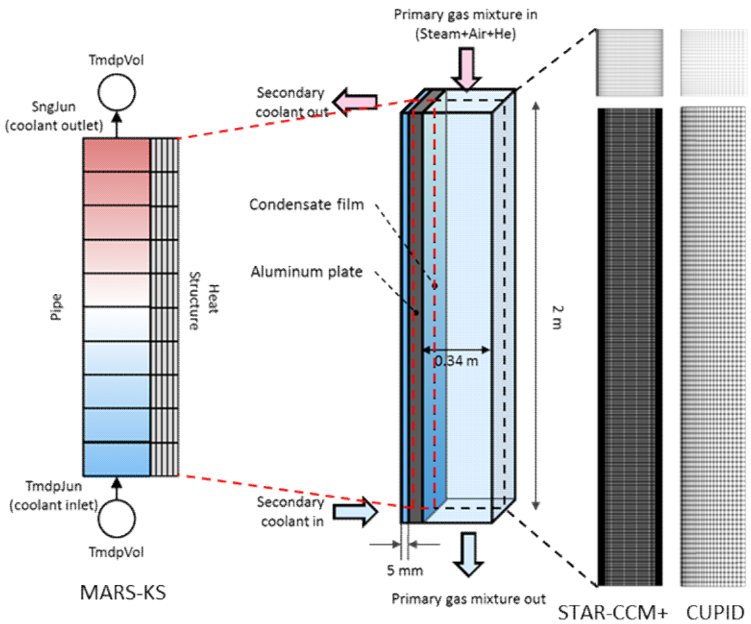
P05-T40-V06-H90 (Natural convective condensation)

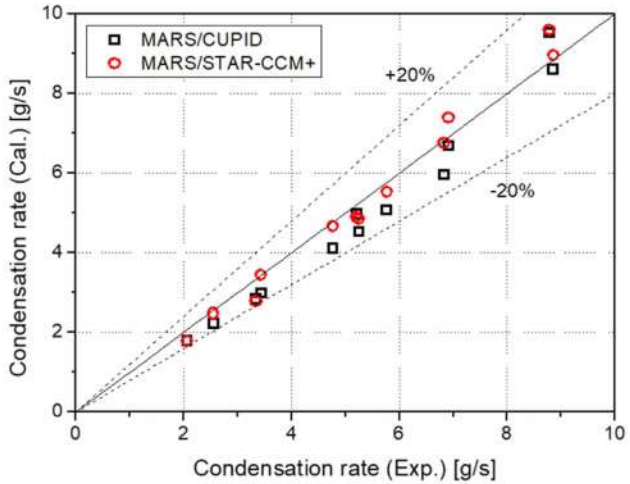


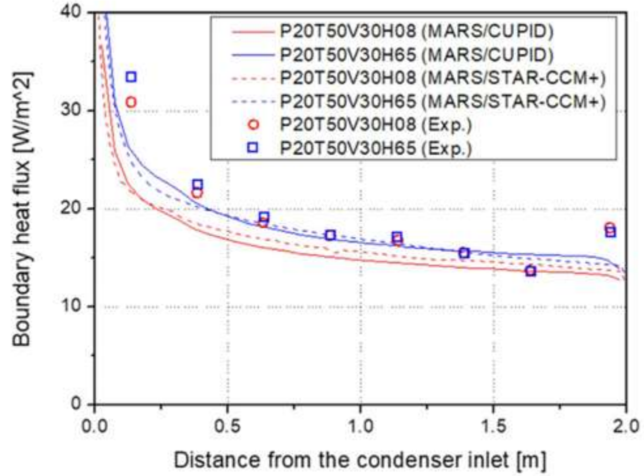
FILMWISE CONDENSATION

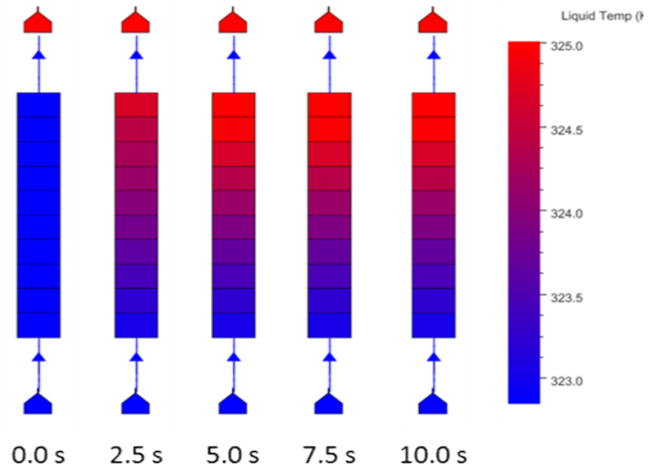
Lee, C.W. et al., Multi-scale simulation of wall film condensation in the presence of non-condensable gases using heat structure-coupled CFD and system analysis codes, NET, 2021

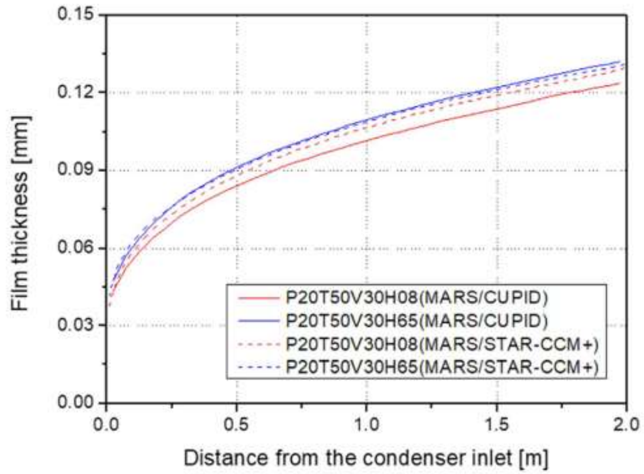
Multi-scale approach











Minor contributions

CUPID NCG diffusion model

1. $\nabla \cdot \left(\alpha_g \rho_g D \nabla X_n\right)$
2. $\nabla \cdot \left(\alpha_g \rho_g D \nabla (X_n q_{air})\right)$

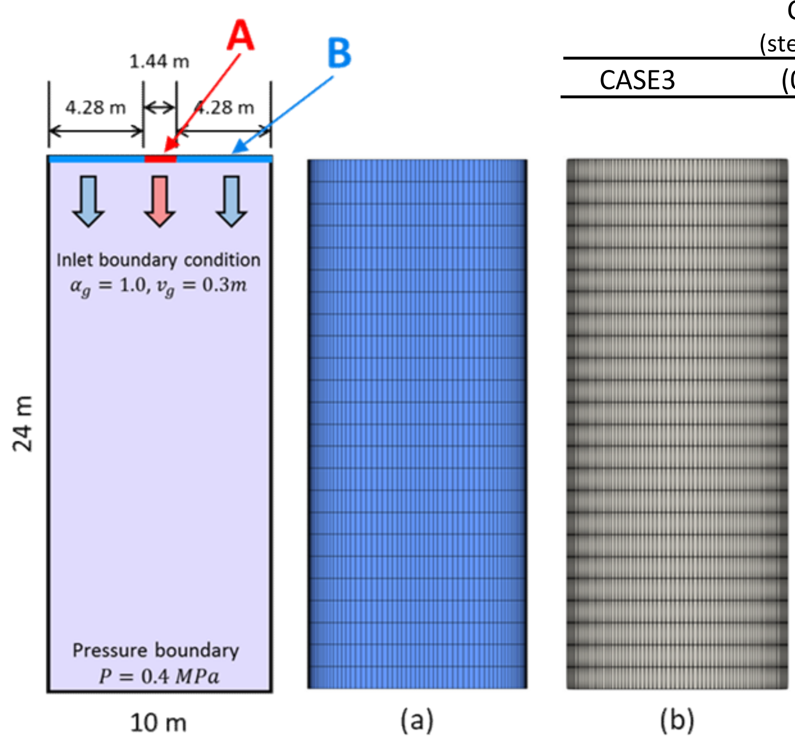
   $\nabla \cdot \left(\alpha_g \rho_g D \nabla (X_n q_{He})\right)$
3. $\nabla \cdot \left(\Sigma_f (h_s - h_{NCG}) \left(\alpha_g \rho_g D \nabla (X_n)\right)\right)$

General species transport equation

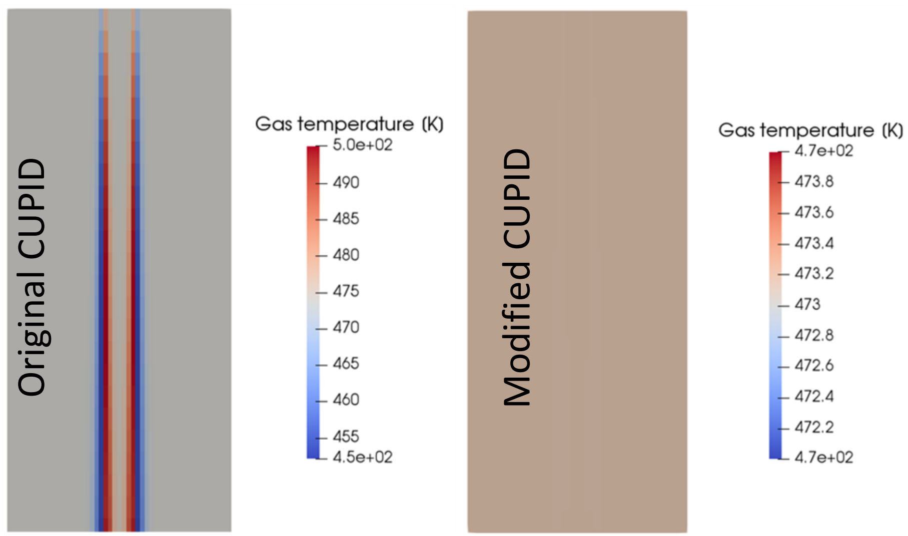
1. $\nabla \cdot \left(\alpha_g \rho_g D_{air} \nabla X_n q_{air} + \alpha_g \rho_g D_{He} \nabla X_n q_{He}\right)$
2. $\nabla \cdot \left(\alpha_g \rho_g D_{air} \nabla (X_n q_{air})\right)$

   $\nabla \cdot \left(\alpha_g \rho_g D_{He} \nabla (X_n q_{He})\right)$
3. $\nabla \cdot \left(\Sigma_f (h_s - h_{air}) \left(\alpha_g \rho_g D_{air} \nabla (X_n q_{air})\right)\right) + \nabla \cdot \left(\Sigma_f (h_s - h_{He}) \left(\alpha_g \rho_g D_{He} \nabla (X_n q_{He})\right)\right)$

	Central inlet (steam, helium, air)	Side inlet (steam, helium, air)
CASE3	(0.0, 0.6, 0.4)	(0.0, 0.3, 0.7)



(a) (b)



Minor contributions

CUPID RBLA condensation model

$$\dot{m}''_{v,i} = -\frac{1}{1-Y_{v,i}}\rho D_{va}\frac{\partial Y_{v,i}}{\partial n}$$

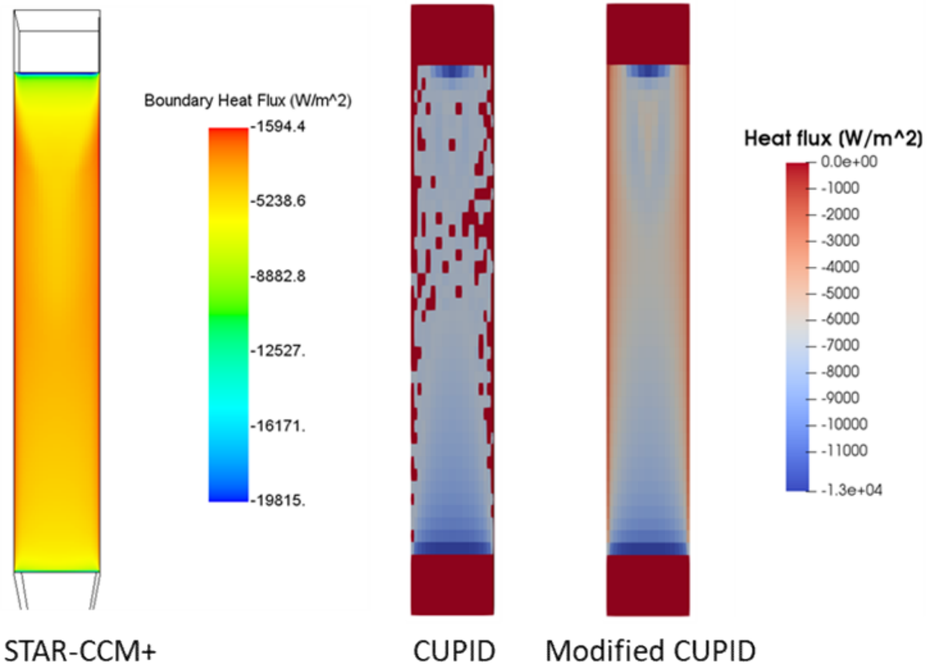
$$Y_{v,i} = \frac{M_v X_{v,i}}{M_v X_{v,i} + M_{air} X_{air,i}} = \frac{M_v X_{v,i}}{M_v X_{v,i} + M_{air}(1-X_{v,i})}$$

General RBLA condensation model

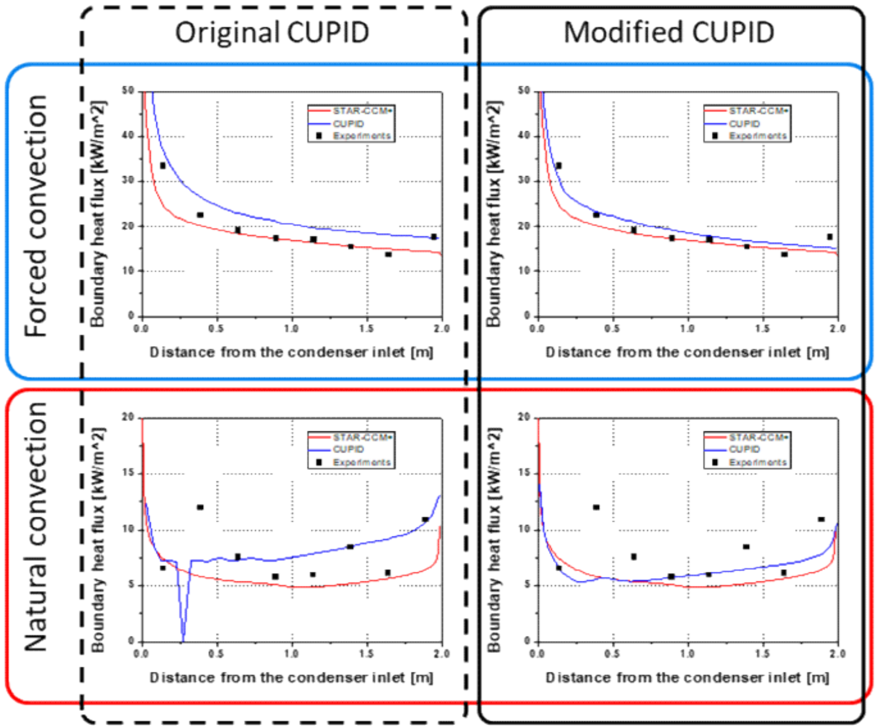
$$\dot{m}''_{v,i} = -\frac{1}{1-Y_{v,i}}\rho D_{v}\frac{\partial Y_{v,i}}{\partial n}$$

$$Y_{v,i} = \frac{M_v X_{v,i}}{M_v X_{v,i} + M_{ncg} X_{ncg,i}} = \frac{M_v X_{v,i}}{M_v X_{v,i} + M_{bg}(1-X_{v,i})}$$

$$\left(M_{bg} = \frac{M_{He}X_{He} + M_{Air}X_{Air}}{X_{He} + X_{Air}}\right)^*$$

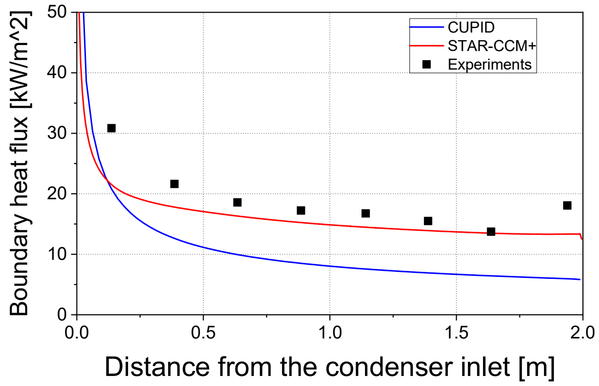


- Condensation heat flux distribution (P05-T40-V06-H90)



- Comparison of condensation heat flux between original and modified CUPID

Minor contributions



STAR-CCM+

- The **high-*y*+ wall treatment** implies the wall-function-type approach: this treatment **assumes that the near-wall cell lies within the logarithmic region of the boundary layer.**
- The **low-*y*+ wall treatment** is suitable only for low-Reynolds number turbulence models: this treatment **assumes that the viscous sublayer is properly resolved.**
- The all-*y*+ wall treatment is a hybrid treatment that attempts to emulate the high-*y*+ wall treatment for coarse meshes, and the low-*y*+ wall treatment for fine meshes. It is also formulated with the desirable characteristic of producing reasonable answers for meshes of intermediate resolution (that is, when the wall-cell centroid falls within the buffer region of the boundary layer).

CUPID

- There are default $\boldsymbol{k-\varepsilon}$ base model and Low-Re $\boldsymbol{k-\varepsilon}$ **model** in CUPID
- However, there is no low-y+ wall treatment which is needed to calculate with resolved boundary layer meshes

- $\tau_w = \rho C_\mu^{1/4}\sqrt{k}\frac{\kappa}{\ln(y^+E)}U_t$
- $u_\tau = \frac{\kappa}{\ln(\frac{\rho\Delta y u_\tau}{\mu}E)}U_t$
- $y^+ = \frac{\rho\Delta y u_\tau}{\mu}$

FILMWISE CONDENSATION

LESSONS LEARNED

Minor contributions =

Commercial codes are very useful for verification!

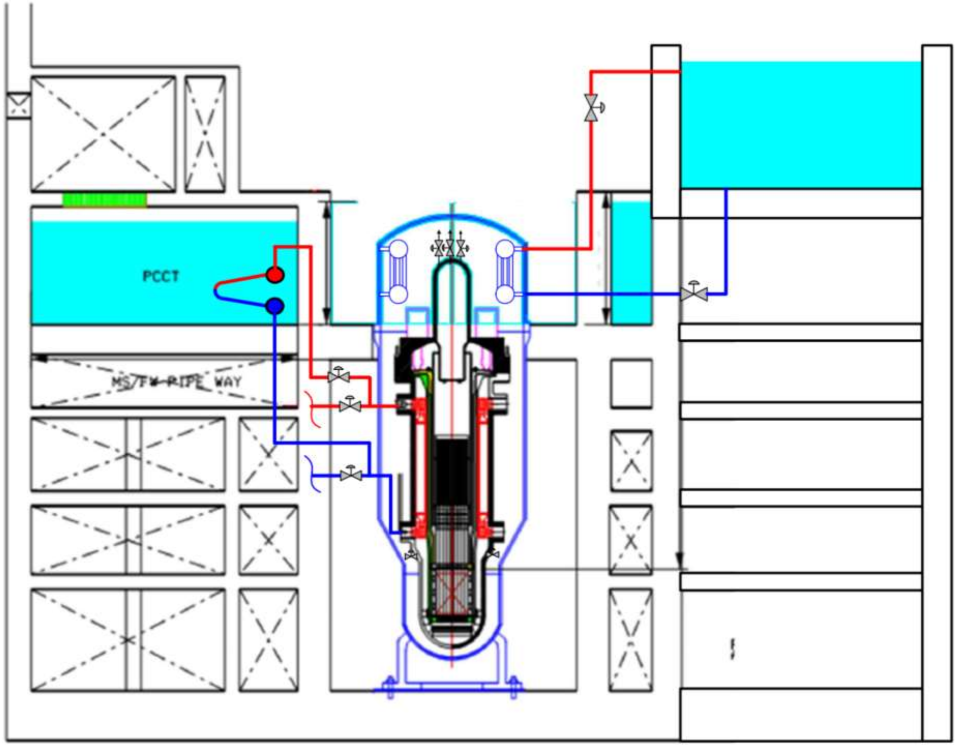
Especially, for turbulence model.

Check local values!

Gas properties selection
Interface or bulk?

Turbulence model
Wall function or RBLA?

Hope to be used for iSMR analysis

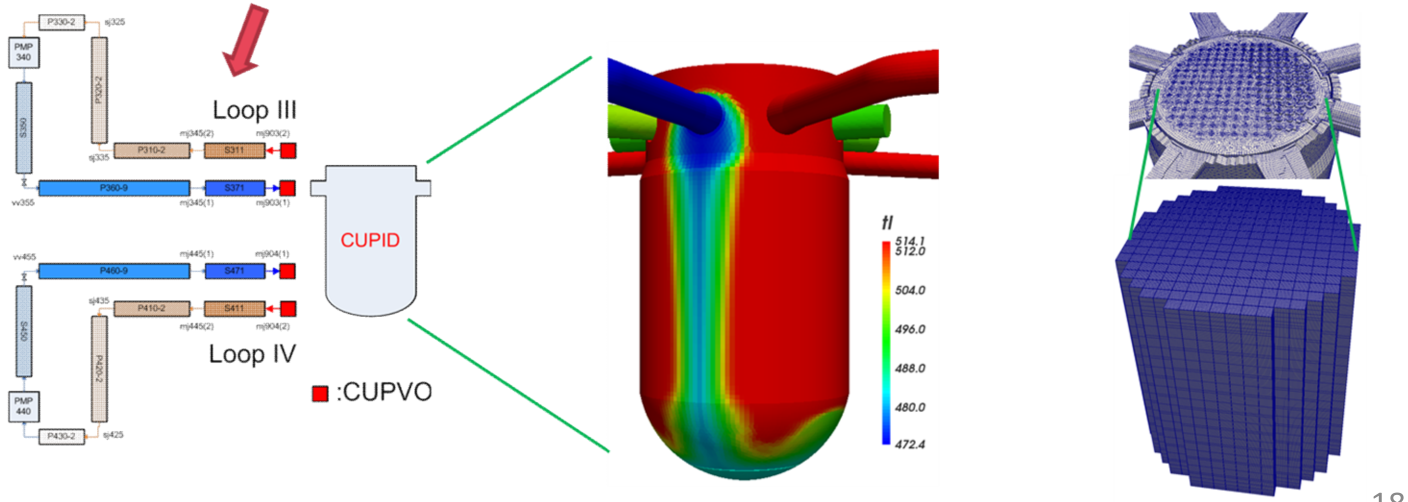


CASL RISE OF CTF

COMPETITIVE ADVANTAGE

Multi-scale approach

1D system scale (MARS) ⟷ 3D CFD scale (CUPID) ⟷ 3D Component scale (CUPID)



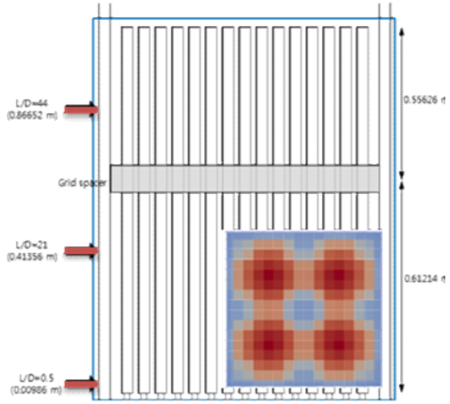
S.J. Yoon, Application of CUPID for subchannel-scale thermal–hydraulic analysis of pressurized water reactor core under single-phase conditions, NET, 2018

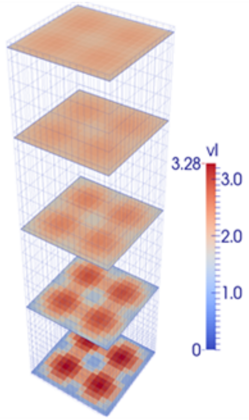
Implementation of subchannel TH models and validation

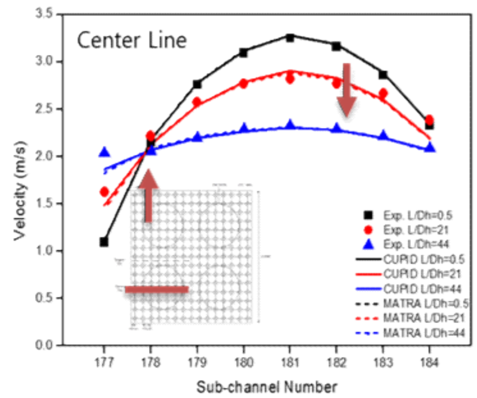
Validation of CUPID for unheated single-phase flow

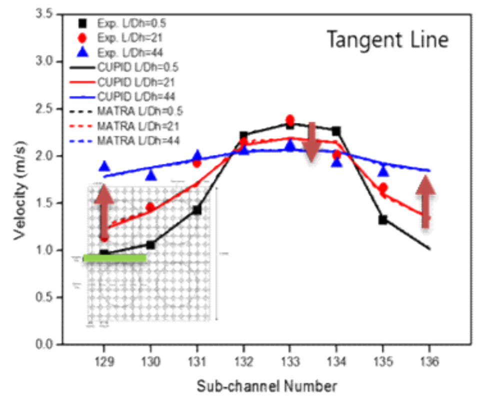
CE 15x15 inlet jetting test

Errors along the center line: 8.2 %
Errors along the tangent line: 9 %



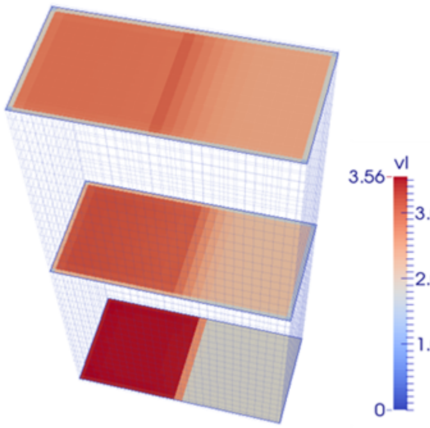


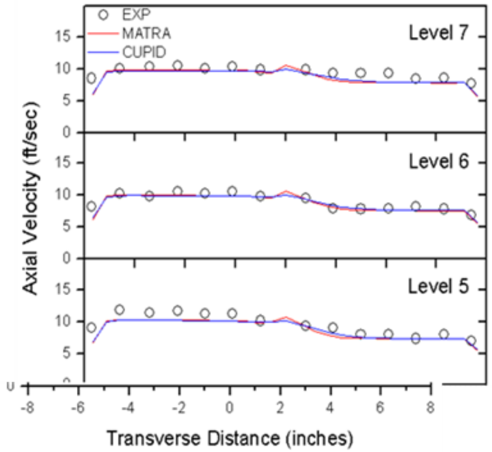


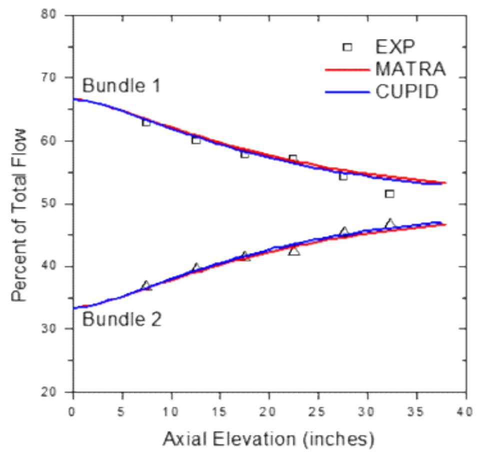


Inlet velocity distribution and measurement elevations

WH 14x14 two-assembly inlet blockage test



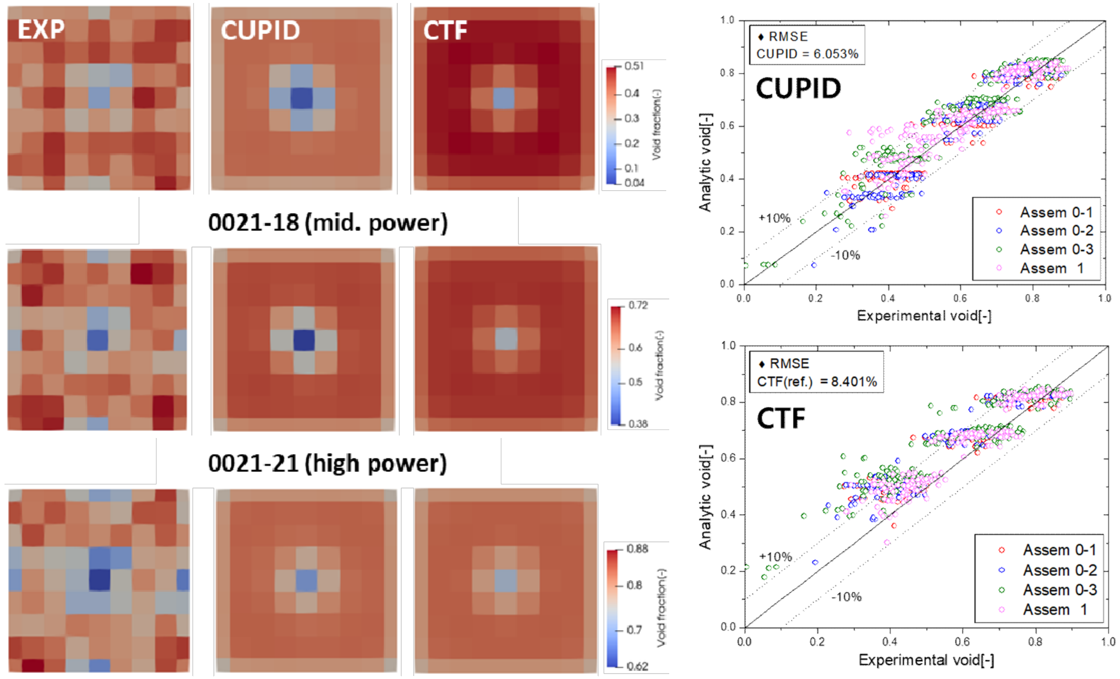




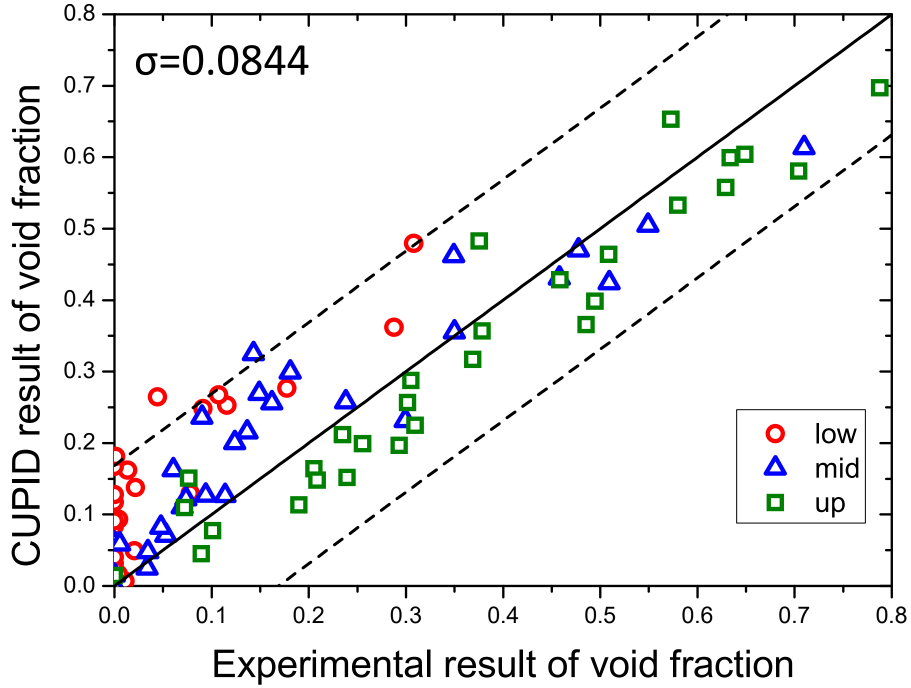
Implementation of subchannel TH models and validation

Validation for heated two-phase flows

BFBT

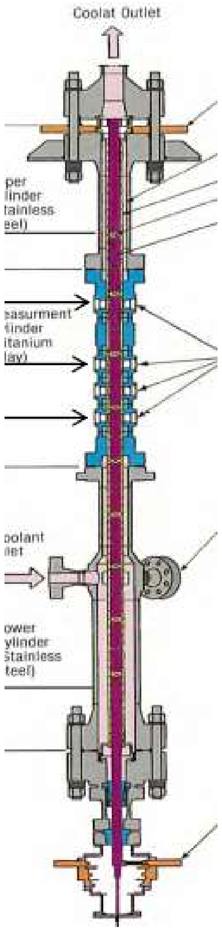


PSBT



Implementation of subchannel TH models and validation

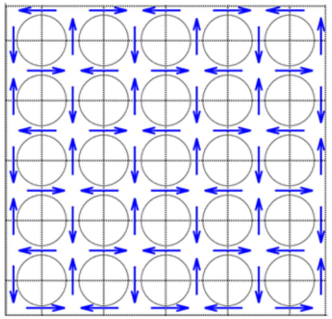
Mixing vane model

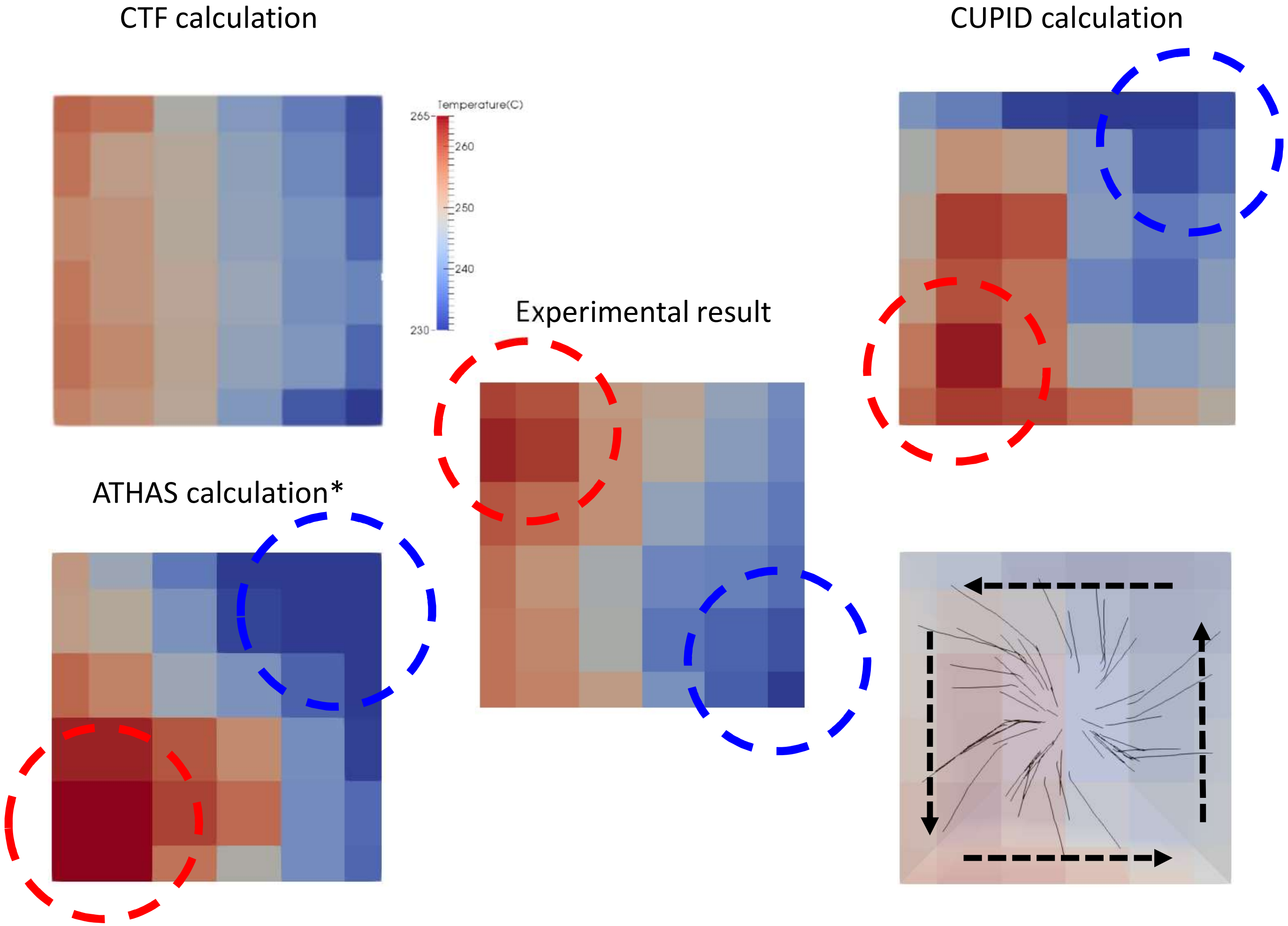


봉다발 출력 분포

1.00	1.00	0.25	0.25	0.25
1.00	1.00	1.00	0.25	0.25
1.00	1.00	0.25	0.25	0.25
1.00	1.00	1.00	0.25	0.25
1.00	1.00	0.25	0.25	0.25

혼합날개 방향





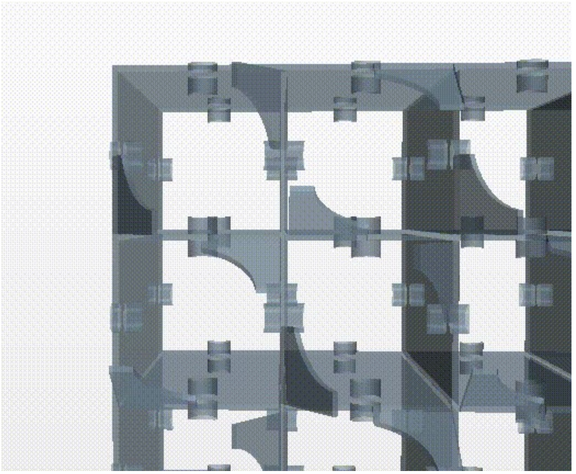
Implementation of subchannel TH models and validation

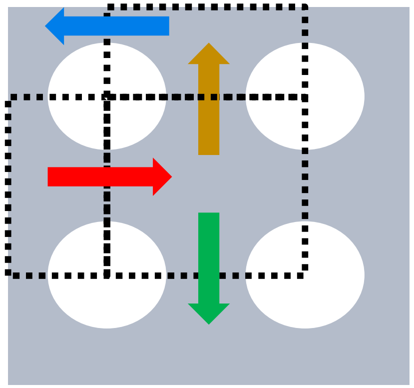
Mixing vane model (grid directed cross flow model)

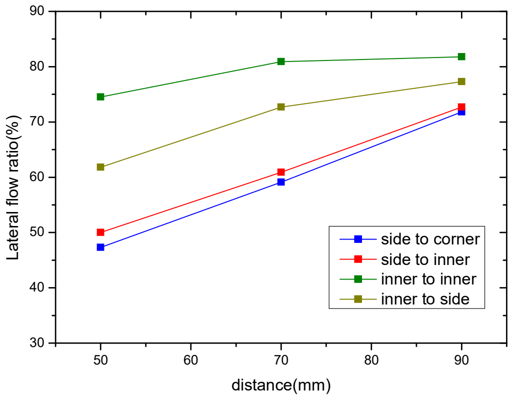
Lateral convection factor from STAR-CCM+

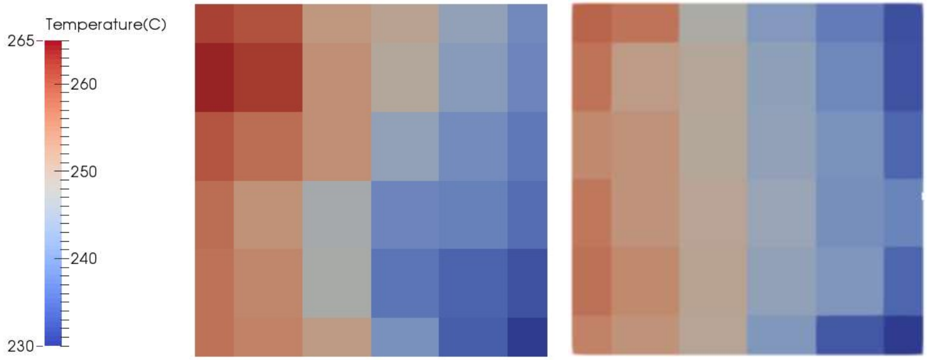
$$M_k = f^2 u_l \rho_l A \times u_l$$

f : Lateral convection factor



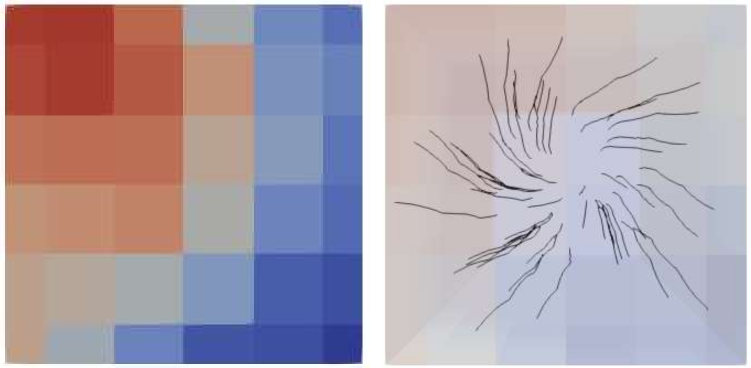






Experimental result

CTF



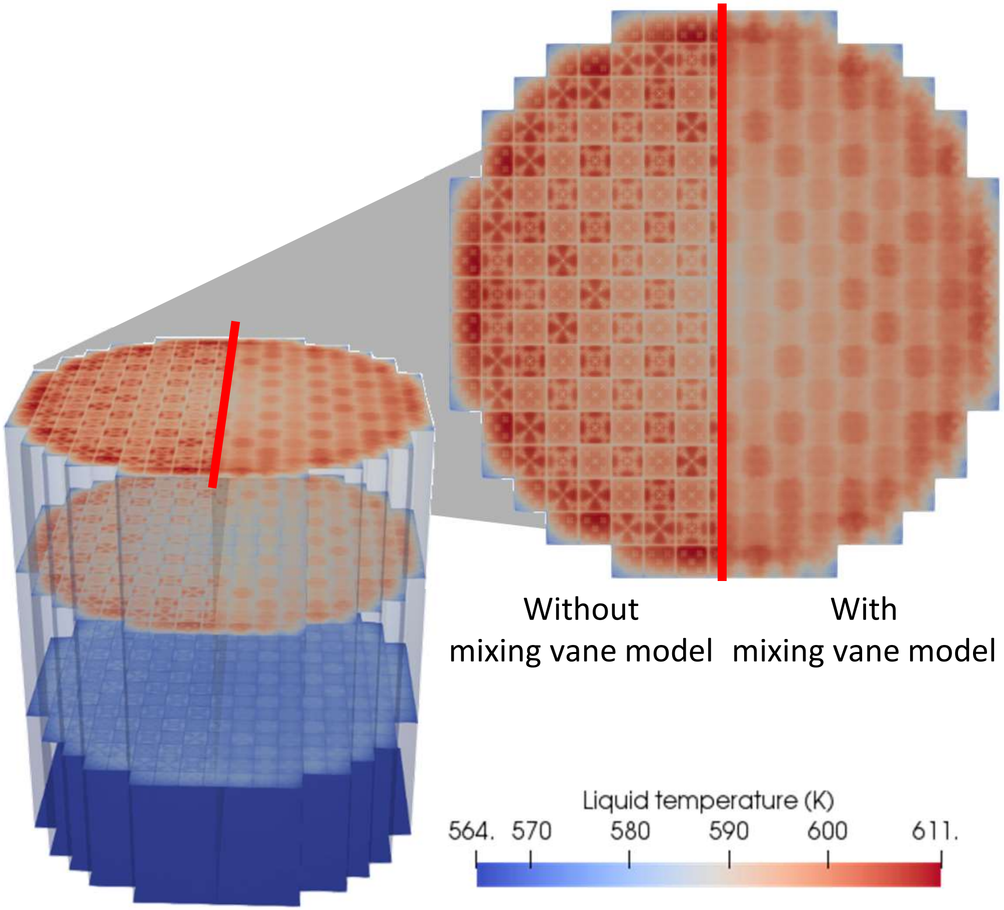
CUPID

CUPID, stream line

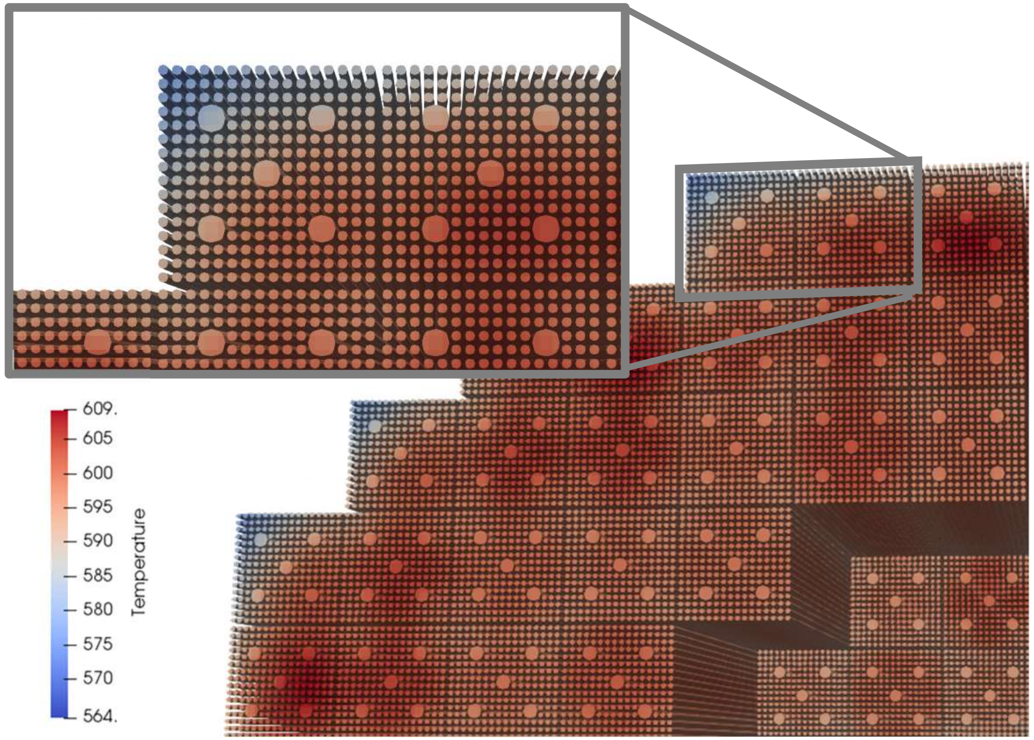
Whole core simulation using CUPID subchannel module

APR1400, $\beta = 0.005$

Liquid temperature

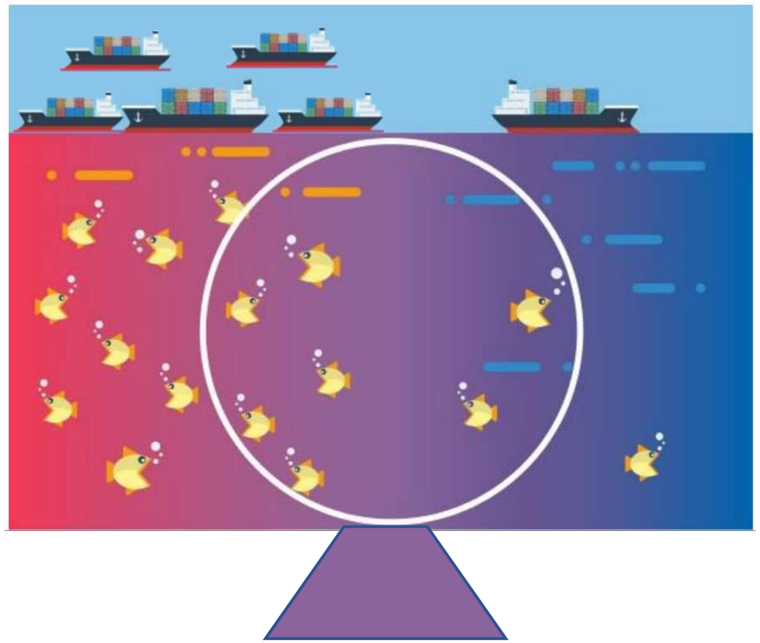


Cladding outer surface temperature



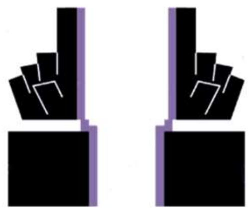
Subchannel Analysis

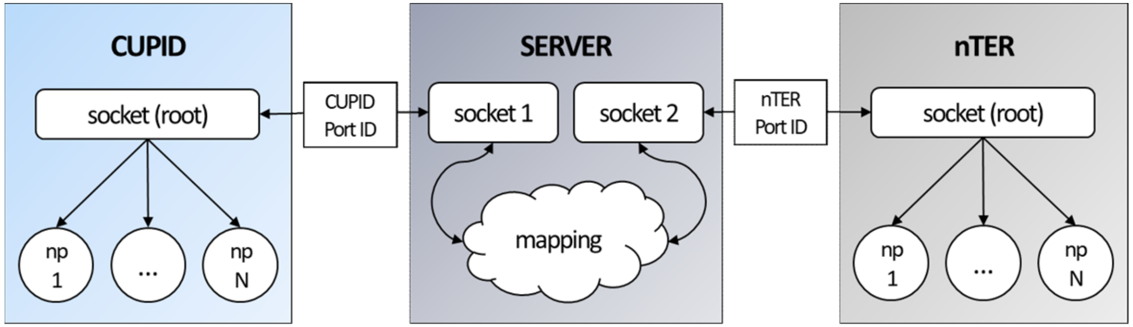
Too Many Competitors



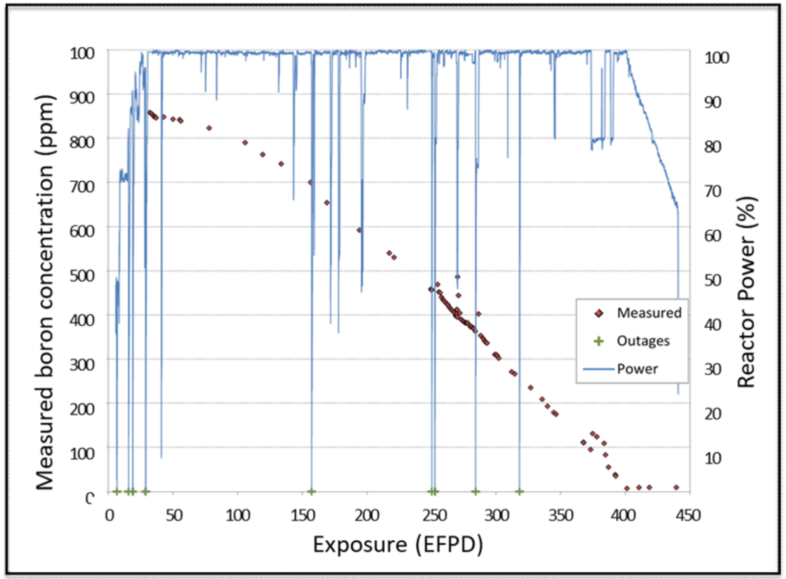
Multi-Scale

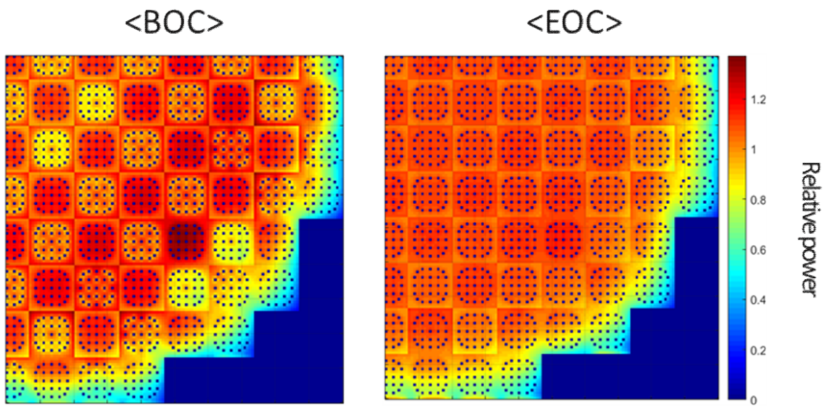
Anything else ?



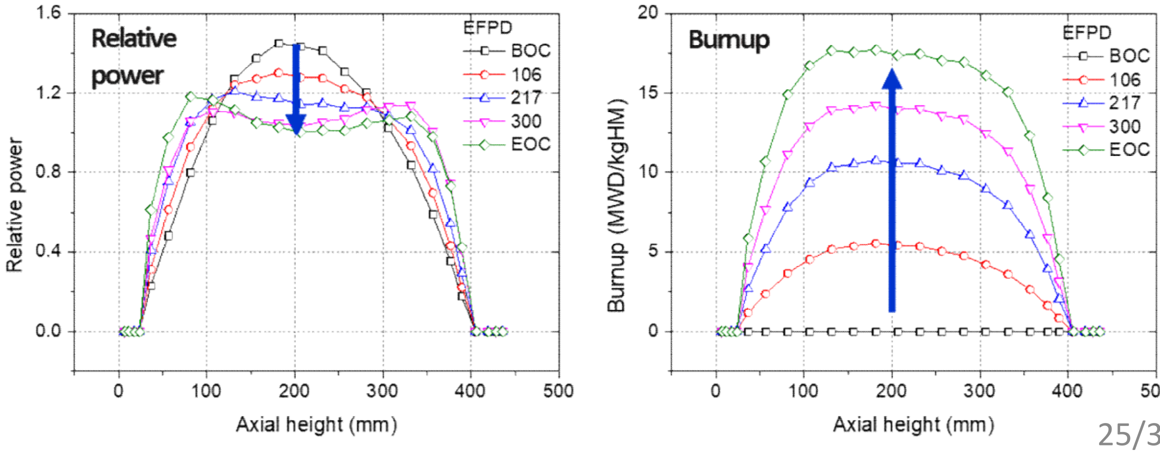


VERA Benchmark



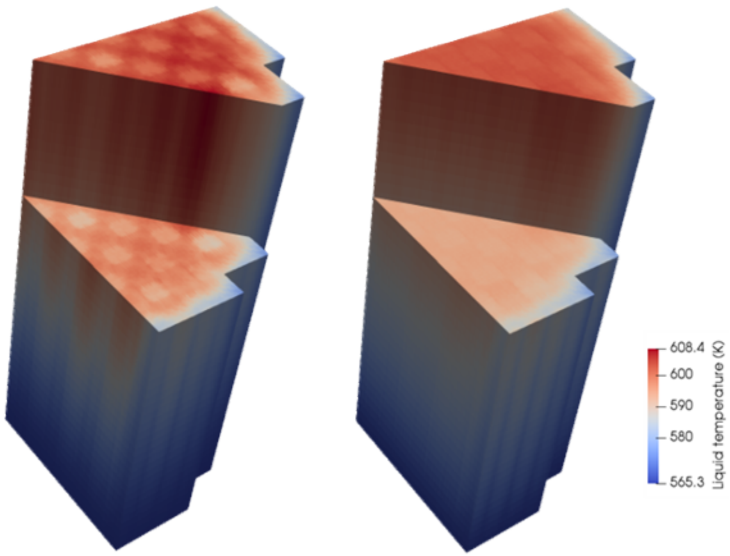


2-D relative pin power distribution

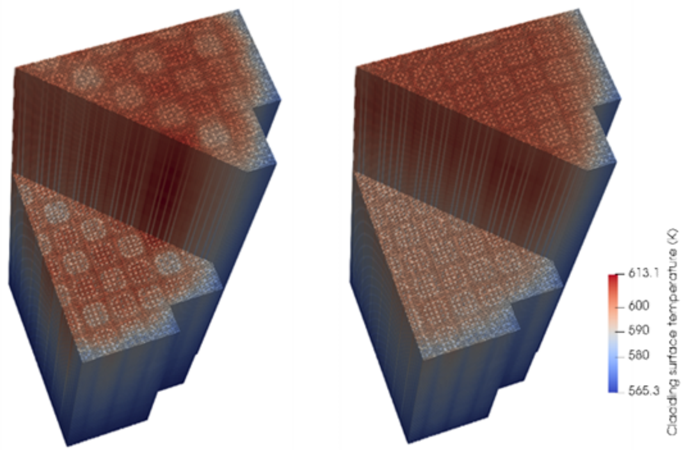


MULTI-PHYSICS (REACTOR PHYSICS)

<BOC> <EOC>

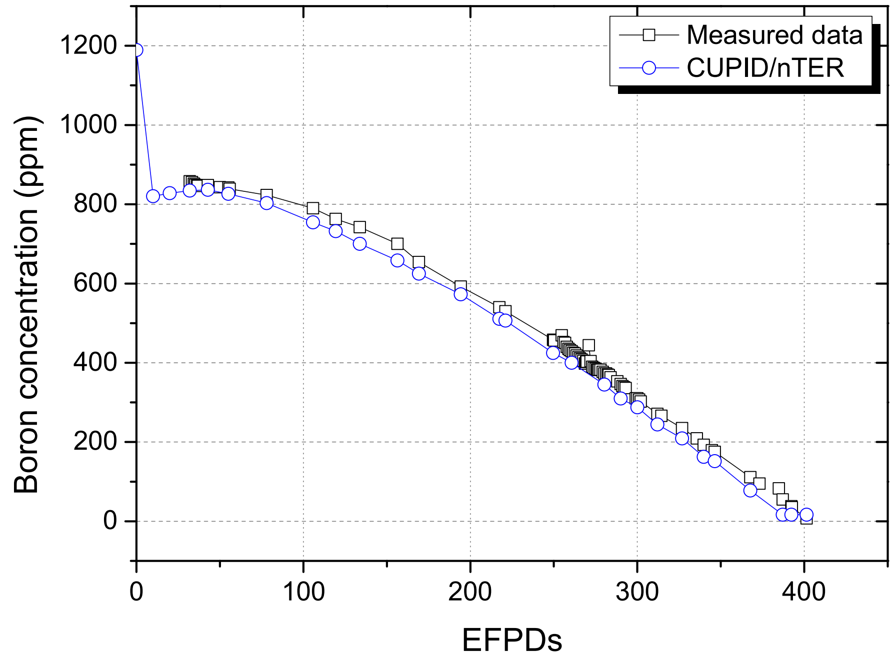


Liquid temperature distribution



Cladding surface temp.

Boron concentration results and measured data



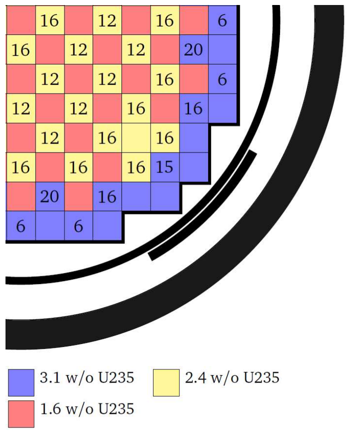
Comparison of VERA benchmark results

	CUPID/nTER	VERA-CS**
CPU requirements	6 nodes 148 cores	180 nodes 4307 cores
# axial planes (TH, neutronics)	(40, 24)	(53, 59)
# state-points	28	37
Average number of iterations between neutronics & TH	9.5	11.1
Mean boron difference (ppm)	-26±13	-24±19

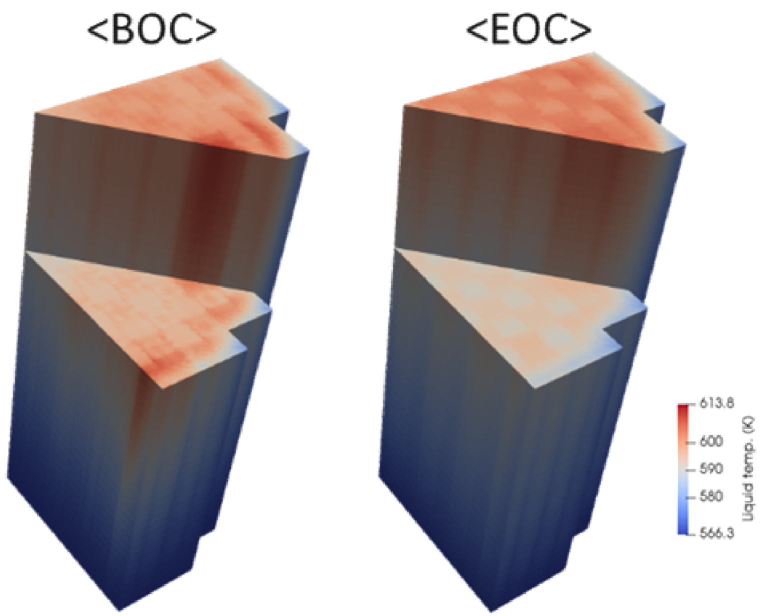
Average runtime/state-point >> 76.5 min / 35.9 min

BEAVRS Benchmark

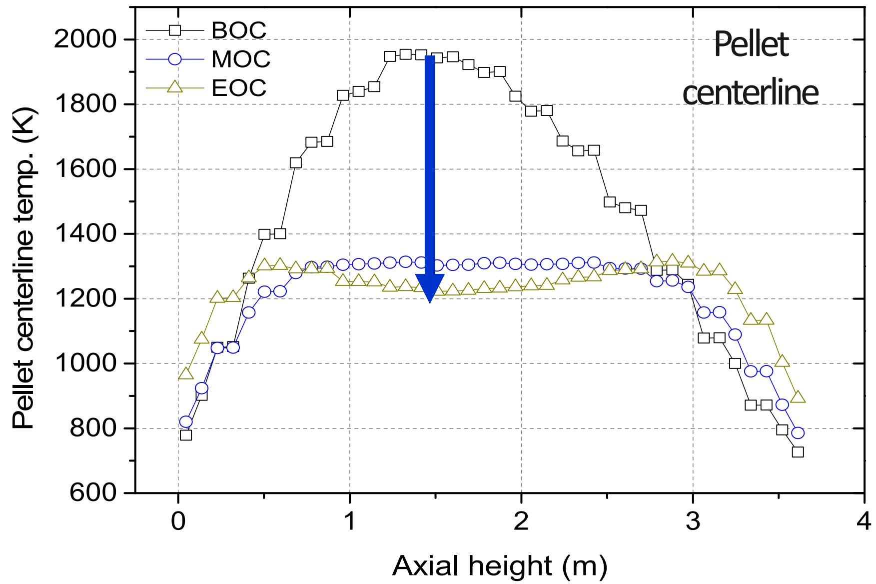
Core fuel and poison loading pattern
(BEAVRS (left) and VERA (right))

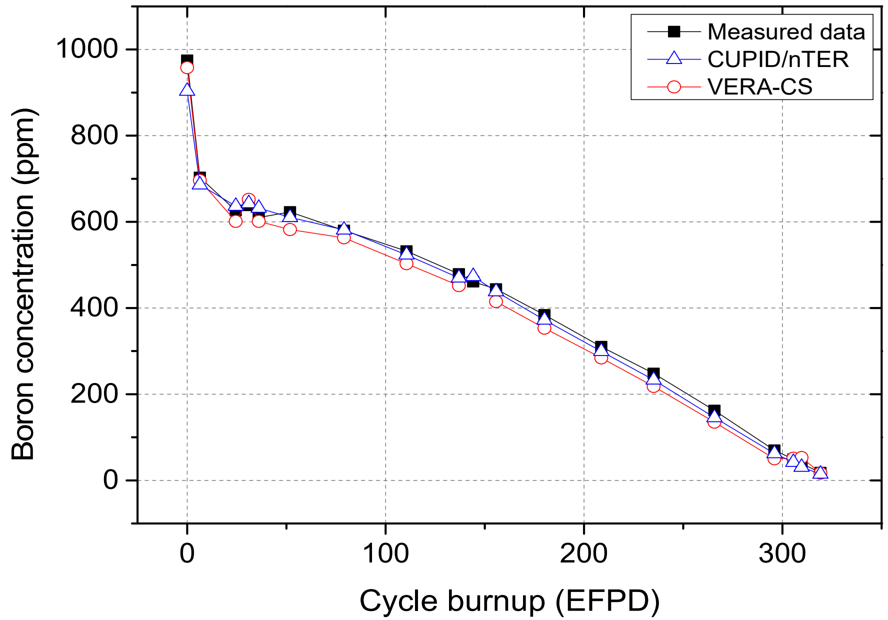


	H	G	F	E	D	C	B	A
8	2.1	2.6 20	2.1	2.6 20	2.1	2.6 20	2.1	3.1 12
9	2.6 20	2.1	2.6 24	2.1	2.6 20	2.1	3.1 24	3.1
10	2.1	2.6 24	2.1	2.6 20	2.1	2.6 16	2.1	3.1 8
11	2.6 20	2.1	2.6 20	2.1	2.6 20	2.1	3.1 16	3.1
12	2.1	2.6 20	2.1	2.6 20	2.6	2.6 24	3.1	
13	2.6 20	2.1	2.6 16	2.1	2.6 24	3.1 12	3.1	
14	2.1	3.1 24	2.1	3.1 16	3.1	3.1		
15	3.1 12	3.1	3.1 8	3.1	Enrichment Number of Pyrex Rods			



Liquid temperature

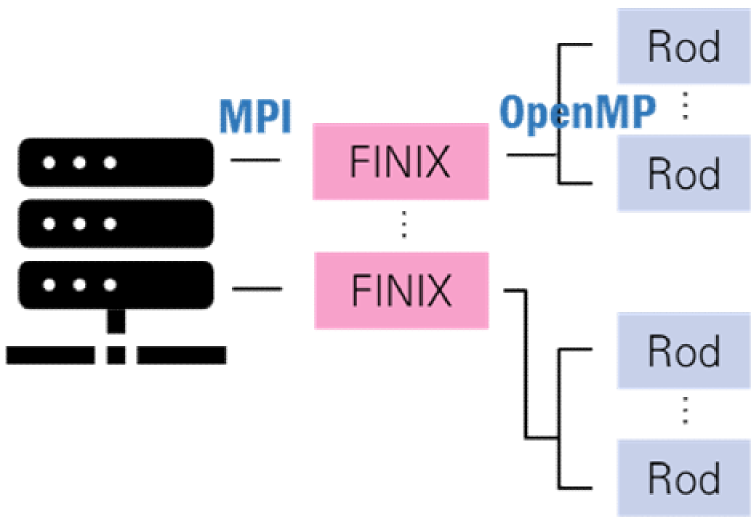


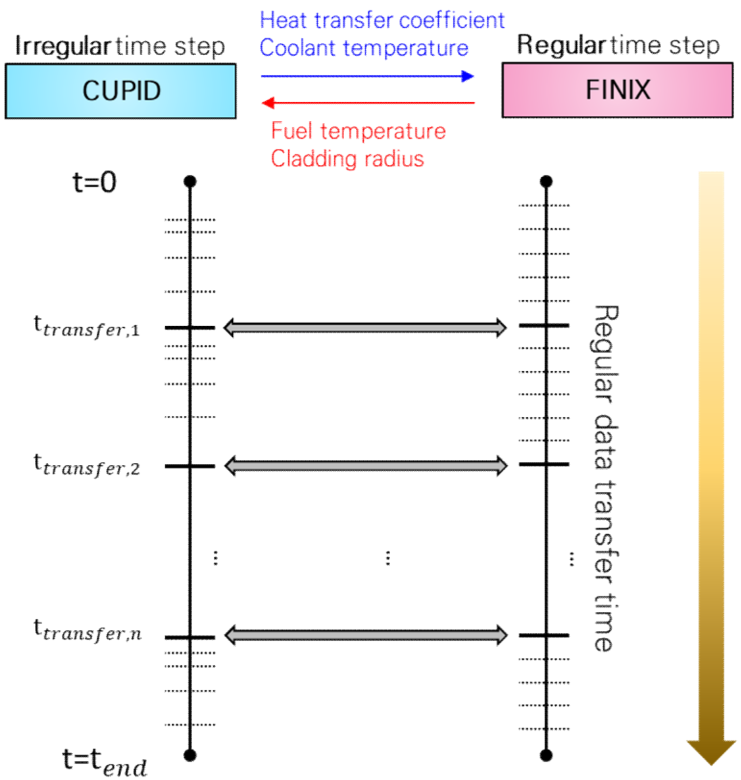


FINIX VTT

Fuel performance code
Both steady-state and transient analyses

Parallelization of FINIX



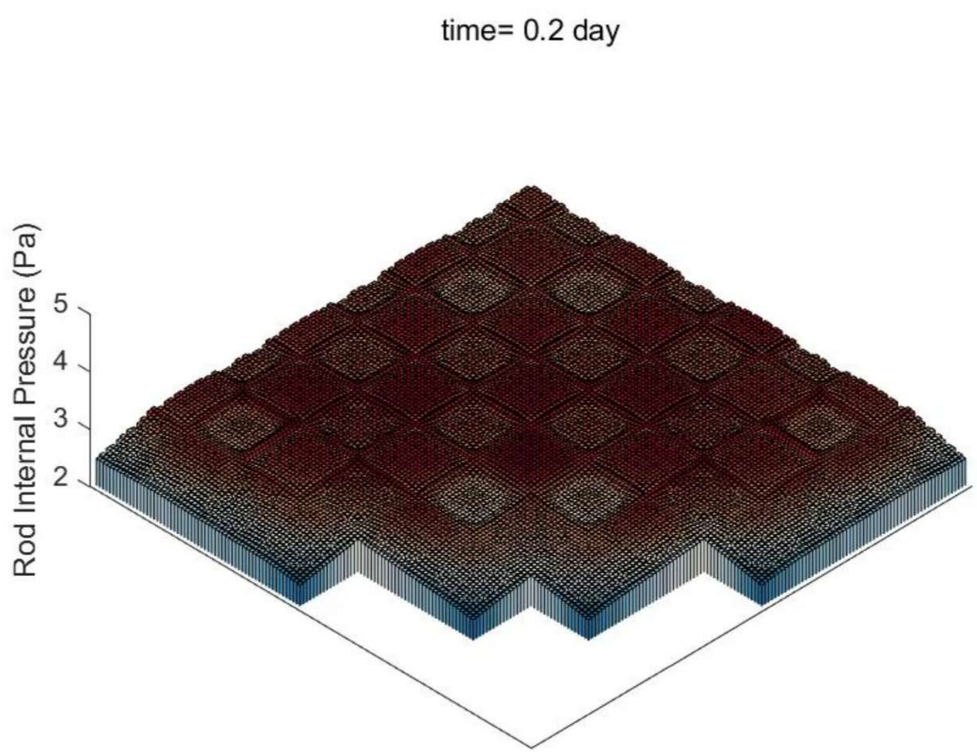


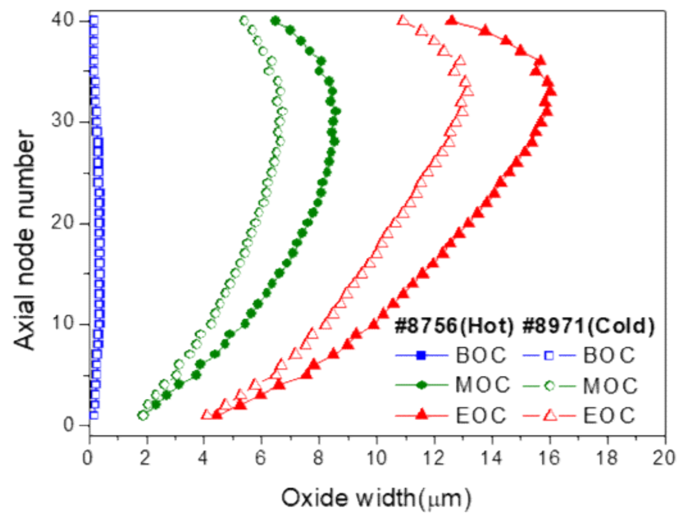
Data exchange of CUPID/FINIX

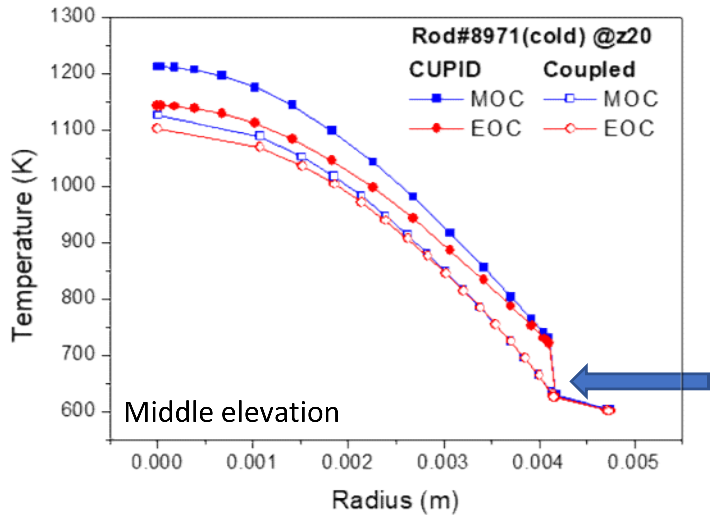
FINIX VTT

Fuel performance code
Both steady-state and transient analysis

Whole core pin-by-pin analysis (VERA benchmark)







Research Plan Using CUPID

핵연료코드 통합 CUPID 부수로 모듈 검증 및 개선 (2021–2026)

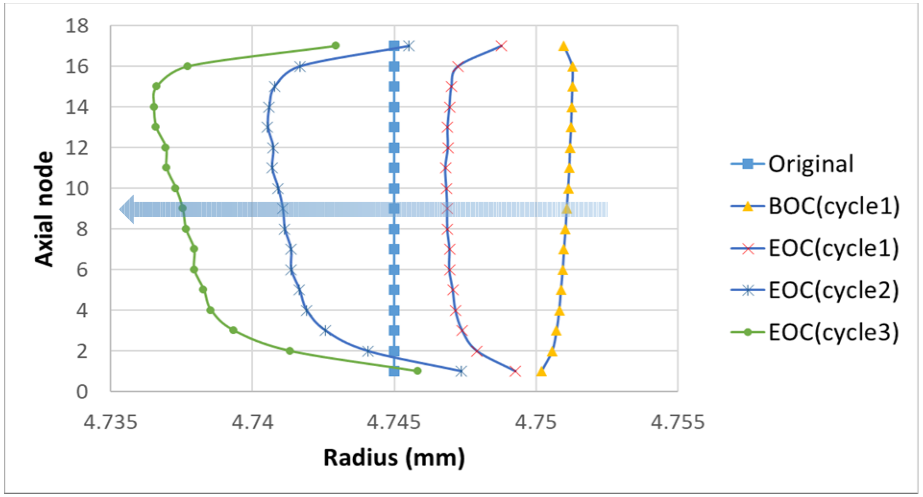
GIFT

Fuel performance code for steady-state

Pin-wise initial conditions

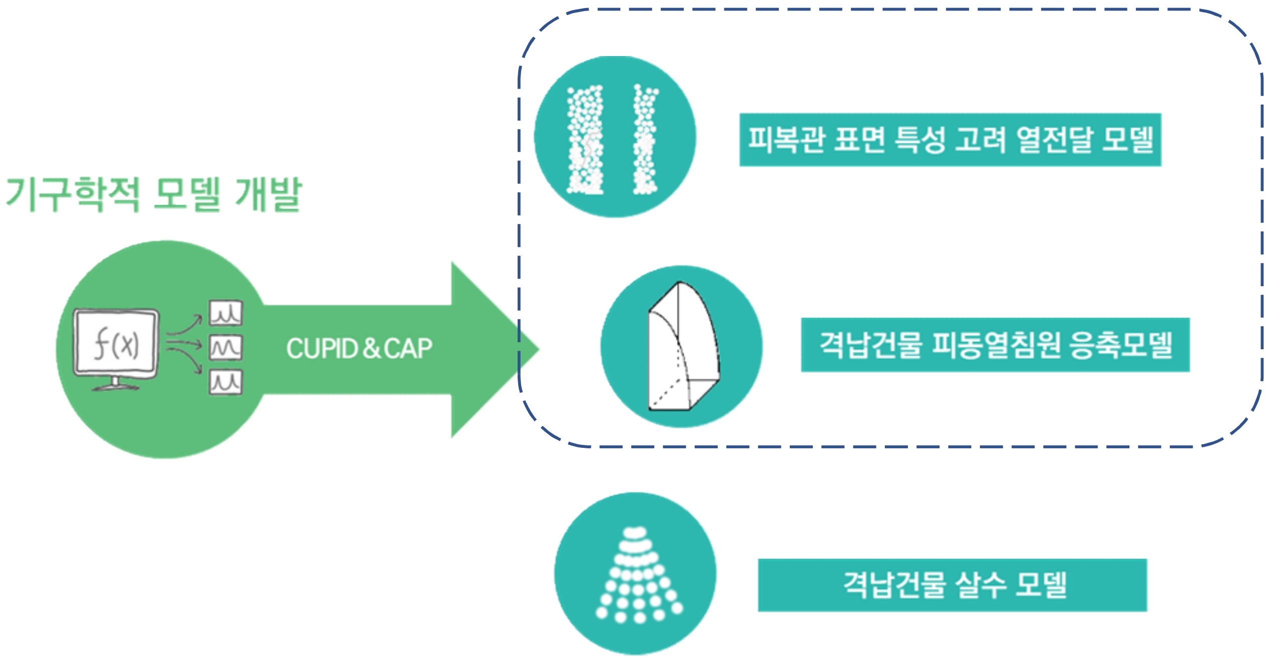
FRAPTRAN

Fuel performance code for transient



Research Plan Using CUPID

최적 통합해석체계 기구학적 모델 및 검증기술 개발 (2022~2029)



Closing remarks

Software Quality

Feasibility Prototype

- Validate individual features
- Check feasibility
- Extremely important

Preliminary Product Prototype

- Interplay of components
- Identify limitations & shortcomings
- Selected features

Demonstration Prototype

- Most of the features & functions
- Start user testing

Production Software

- All features
- Fully optimized
- Production ready

Acknowledgment

Technical support of developers

NUTHEL
Nuclear Thermal Hydraulic Engineering Lab.

Filmwise Condensation	J.H. Lee (FNC) C.W. Lee (SNU, PhD student)
Subchannel Analysis	S.J. Yoon (CRI) S.B. Kim (KINS)
Multi-Physics (Reactor Physics)	Y.Y. Choi (MIT, PhD student)
Multi-Physics (Fuel)	J.K. Park (KINS)

10TH ANNIVERSARY

서울과학기술대학교 SEOULTECH

The 10th CUPIDERS Workshop
August 23th, 2022; KAERI, Daejeon

OECD/NEA IBE에 대한 개요, 종합 그리고 개인적 추가 연구 (IBE-2)

김 정 우

기계시스템디자인공학과

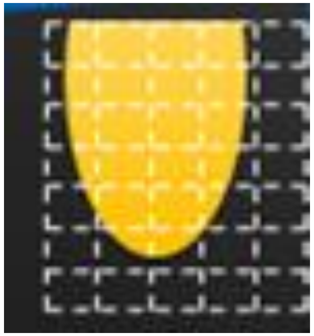
서울과학기술대학교

Introduction

• For safety improvement of nuclear power plants, it is necessary to understand and estimate thermal-hydraulics phenomena of various flow types existing there.

• Many traditional reactor system codes are modelled as networks of 1-D or 0-D volumes. However, there are important 3-D aspects of the system's thermal hydraulics. Typical instances in NRS problems include: mixing and stratifications and many other situations.

• Natural circulation, mixing and stratification is essentially 3-D nature, and representing such complex flows by pseudo 1-D approximations may not just be oversimplified, but misleading, producing erroneous conclusions.



Introduction

• Application of CFD codes in such a field requires validated models, especially models of turbulence and good capacity to treat complex geometries of very different sized scales.

•For single-phase CFD applications, these devolve around the traditional limitations of computing power, controlling numerical diffusion, the appropriateness of the established turbulence models, and so on.

• Complex configurations arising in industrial situations have led people to consider "canonical" situations that may be identified in industrial flows: wall shear flows, free shear flows, and impinging flows.

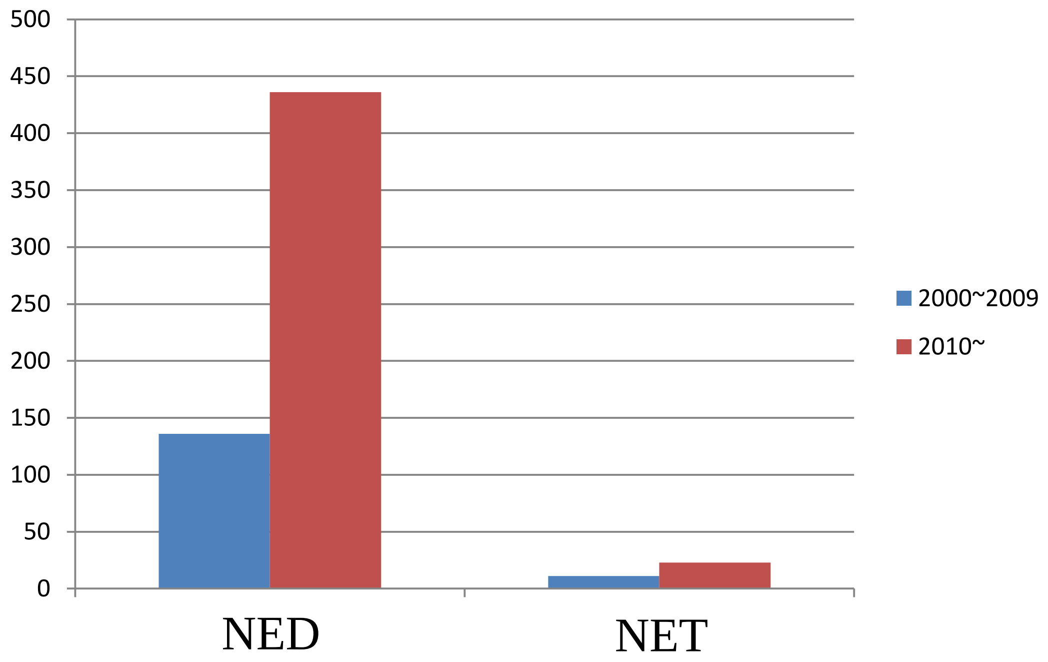
NRS problems where CFD analysis brings real benefits (NEA/CSNI/R(2014)11)

NRS problem	System classification	Incident classification
Mixing: stratification	Primary circuit	Operational
Heterogeneous flow distribution	Primary circuit	Operational

Introduction

원자력 발전소 안전성 향상을 위하여, 원자력 발전소 내 존재하는 다양한 열수력 현상을 정확히 해석하고 이해할 필요가 있음.

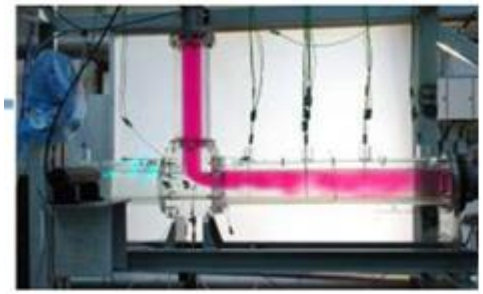
기존의 1차원 시스템 해석 코드는 3차원적인 현상을 예측하는 데 분명히 한계가 있기 때문에, 3차원 CFD 스케일 해석 방법에 대한 관심이 최근 증대되고 있음.



History of OECD/NEA International Benchmark Exercise

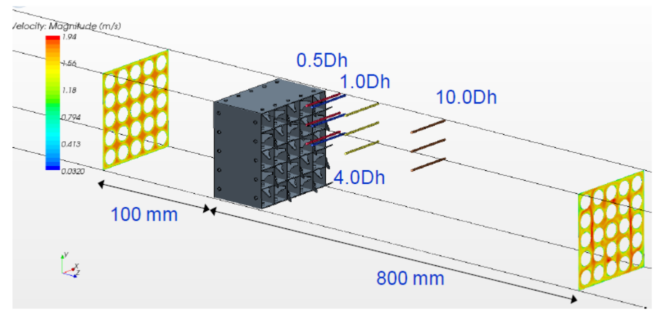
IBE-1: T-junction (2010)

열손상
비정상



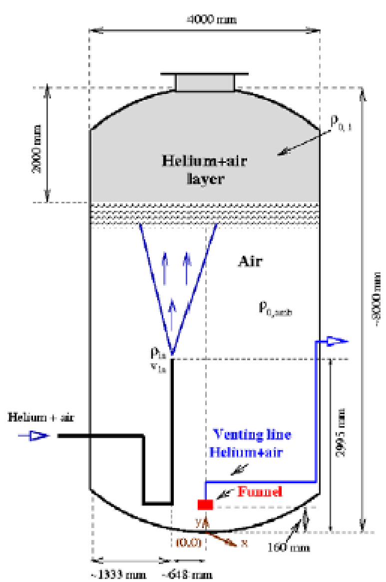
IBE-2: Grid spacer (2012)

난류혼합
발달영역



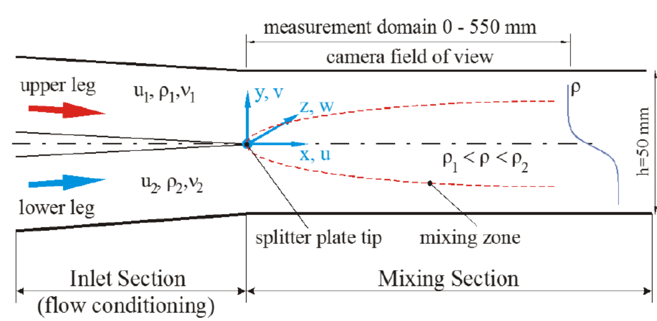
IBE-3: PANDA (2014)

성층화
과도상태



IBE-4: GEMIX (2016)

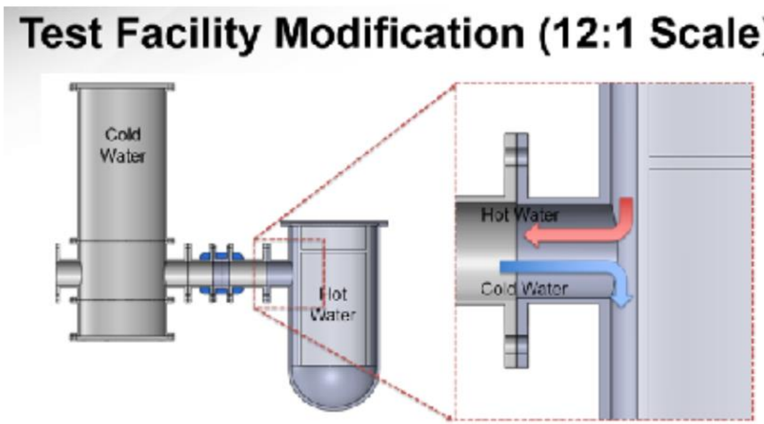
난류혼합
발달영역



History of OECD/NEA International Benchmark Exercise

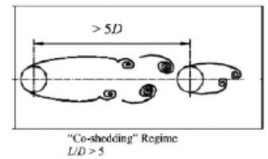
- History of OECD/NEA International Benchmark Exercise

IBE-5: Cold leg (2018)

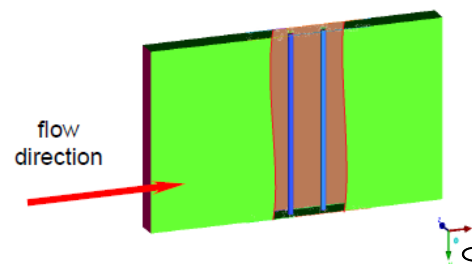


과도혼합문제

IBE-6: FSI (2020)

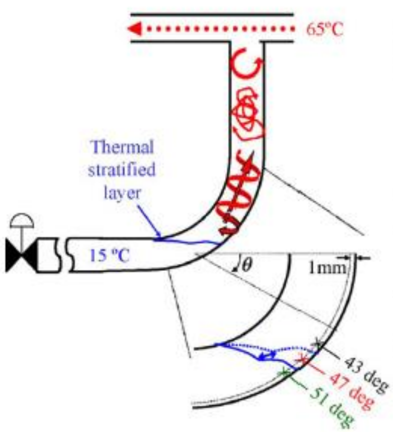


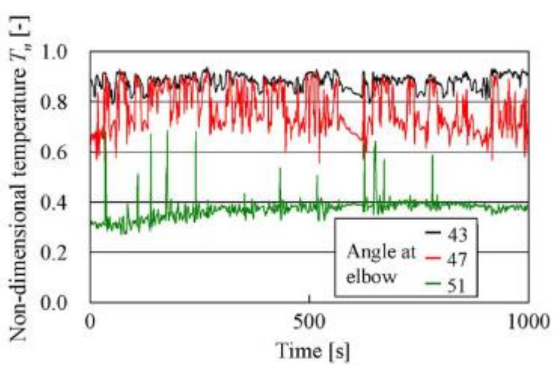
Sumner, "Two circular cylinders in cross-flow…", 2010

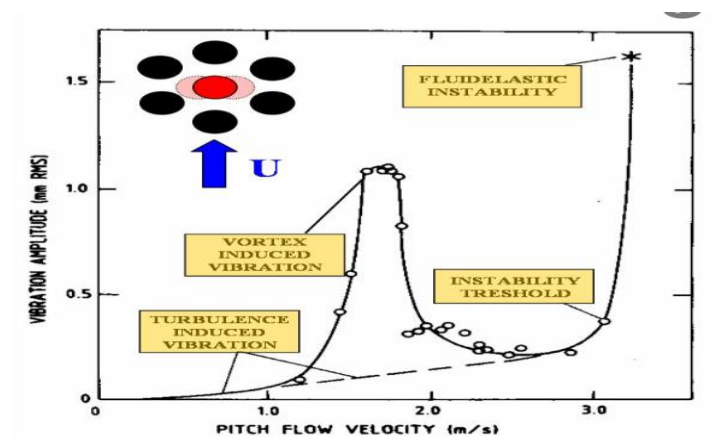


유체-구조물 상호작용

IBE-7: Thermal mixing (2022)







History of OECD/NEA International Benchmark Exercise

Table 1: NRS problems requiring CFD with/without coupling to system codes

	NRS problem	System classification	Incident classification	Single- or multi-phase
1	Erosion, corrosion and deposition	Core, primary and secondary circuits	Operational	Single/Multi
2	Core instability in BWRs	Core	Operational	Multi
3	Transition boiling in BWR/determination of MCPR	Core	Operational	Multi
4	Recriticality in BWRs	Core	BDBA	Multi
5	Reflooding	Core	DBA	Multi
6	Lower plenum debris coolability/melt distribution	Core	BDBA	Multi
7	Boron dilution	Primary circuit	DBA	Single
8	Mixing: stratification/hot-leg heterogeneities	Primary circuit	Operational	Single/Multi
9	Heterogeneous flow distribution (e.g. in SG inlet plenum causing vibrations, HDR experiments, etc.)	Primary circuit	Operational	Single

	NRS problem	System classification	Incident classification	Single- or multi-phase
10	BWR/ABWR lower plenum flow	Primary circuit	Operational	Single/Multi
11	Water hammer condensation	Primary circuit	Operational	Multi
12	PTS (pressurised thermal shock)	Primary circuit	DBA	Single/Multi
13	Pipe break – in-vessel mechanical load	Primary circuit	DBA	Multi
14	Induced break	Primary circuit	DBA	Single
15	Thermal fatigue (e.g. T-junction)	Primary circuit	Operational	Single
16	Hydrogen distribution	Containment	BDBA	Single/Multi
17	Chemical reactions/combustion/detonation	Containment	BDBA	Single/Multi
18	Aerosol deposition/atmospheric transport (source term)	Containment	BDBA	Multi
19	Direct-contact condensation	Containment/ Primary circuit	DBA	Multi
20	Bubble dynamics in suppression pools	Containment	DBA	Multi
21	Behaviour of gas/liquid surfaces	Containment/ Primary circuit	Operational	Multi
22	Special considerations for advanced (including Gas-Cooled) reactors	Containment/ Primary circuit	DBA/BDBA	Single/Multi
23	Sump strainer clogging	Containment	DBA	Single/Multi

DBA – Design Basis Accident; BDBA – Beyond Design Basis (or Severe) Accident; MCPR – Minimum Critical Power Ratio

History of OECD/NEA International Benchmark Exercise

IBE-1: T-junction (2010)

- 29 submissions
- LES calculations: 19
- Numbers of grid points: 300,000 to 70 million

IBE-2: Grid spacer (2012)

- 25 submissions
- LES calculations: 6, (U)RANS calculations: 15
- Numbers of grid points: 3.3~110 million

IBE-3: PANDA (2014)

- 19 submissions
- LES calculations: 3, (U)RANS calculations: 13
- Numbers of grid points: 4,000 to 4.3 million

IBE-4: GEMIX (2016)

- 13 submissions
- LES calculations: 2, (U)RANS calculations: 11
- Numbers of grid points: 59,850 to 20 million

History of OECD/NEA International Benchmark Exercise

IBE-5: Cold leg (2018)

- 10 submissions
- LES calculations: 8
- Numbers of grid points: 0.2M to 500M
- Uncertainty calculations: 5

IBE-6: FSI (2020)

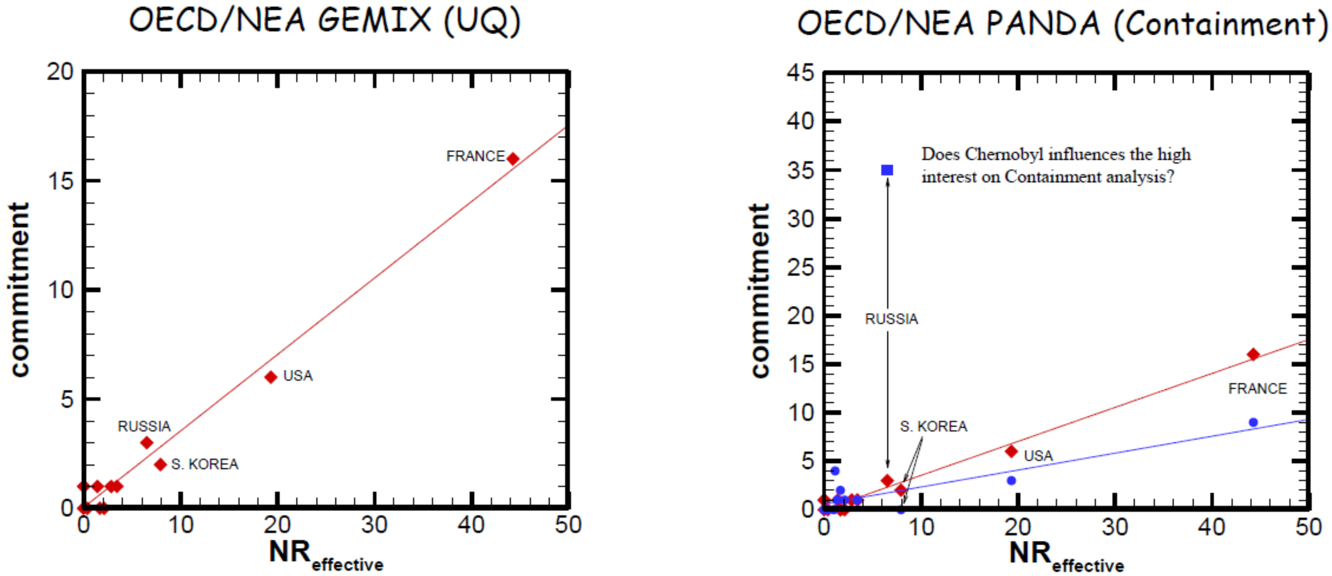
- 10 submissions
- LES calculations: 4, (U)RANS calculations: 4

History of OECD/NEA International Benchmark Exercise

WGAMMA 미팅 (2016.6.13) 자료에서 발췌

$$NR_{effective} = \text{Number of reactors} \times \text{nuclear share in the country}$$

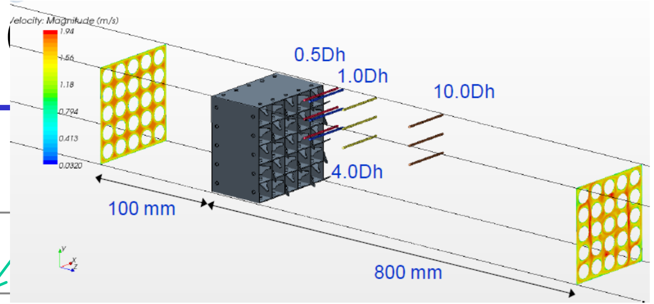
$$\text{Commitment} = \text{Number of registered participants} \times \text{successful submissions}$$

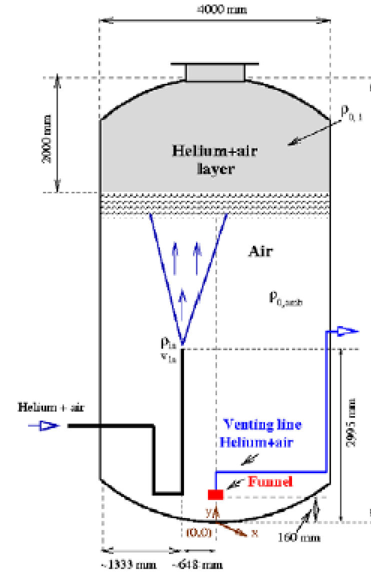


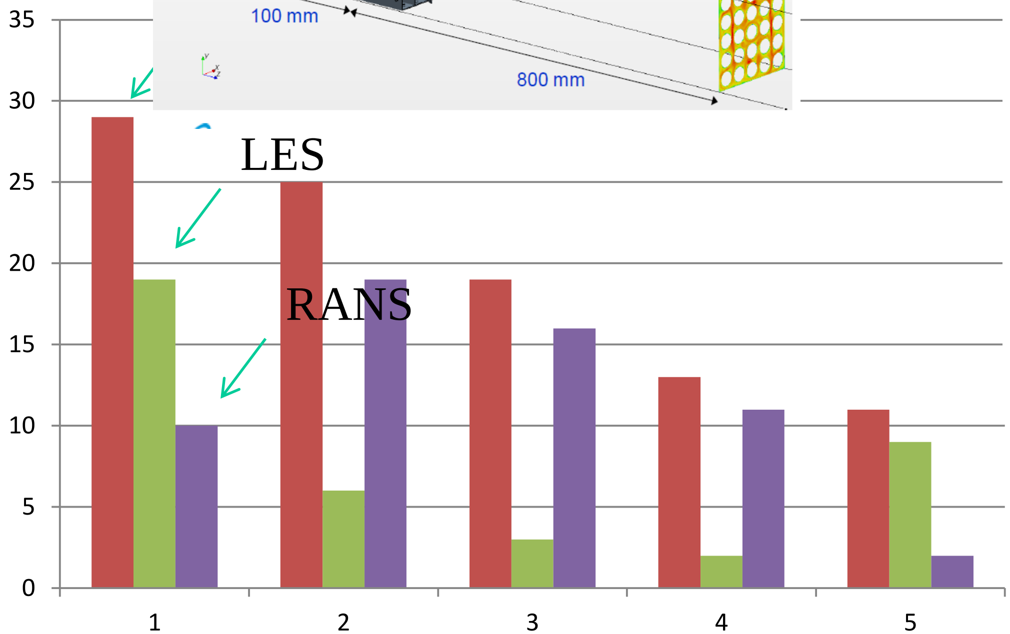
From the slopes we can say that, in general, countries show more interest on UQ than containment!

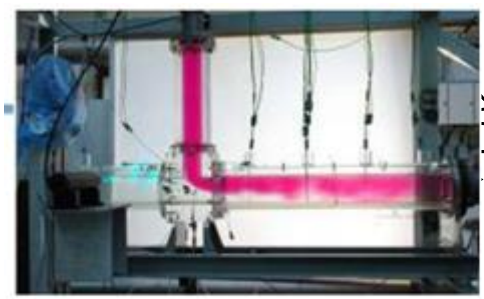
Is this true or is just a problem of dissemination?

Summary of ... l Benchı

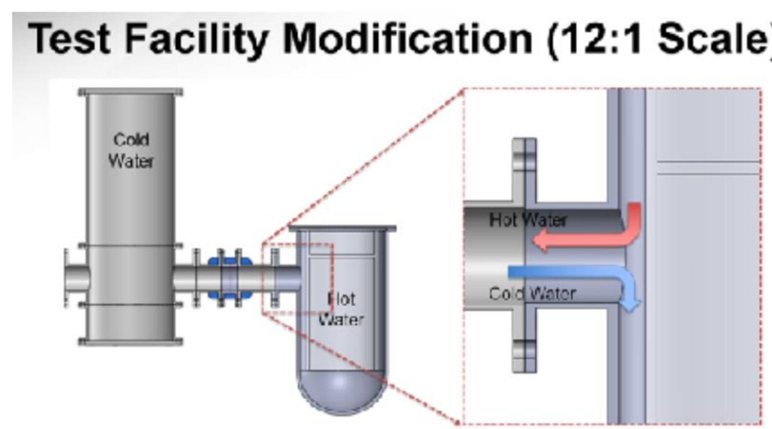








ission 중에서 LES가 차지하는 비중이 높아지고 있음
상 LES가 적합한 문제와
음.

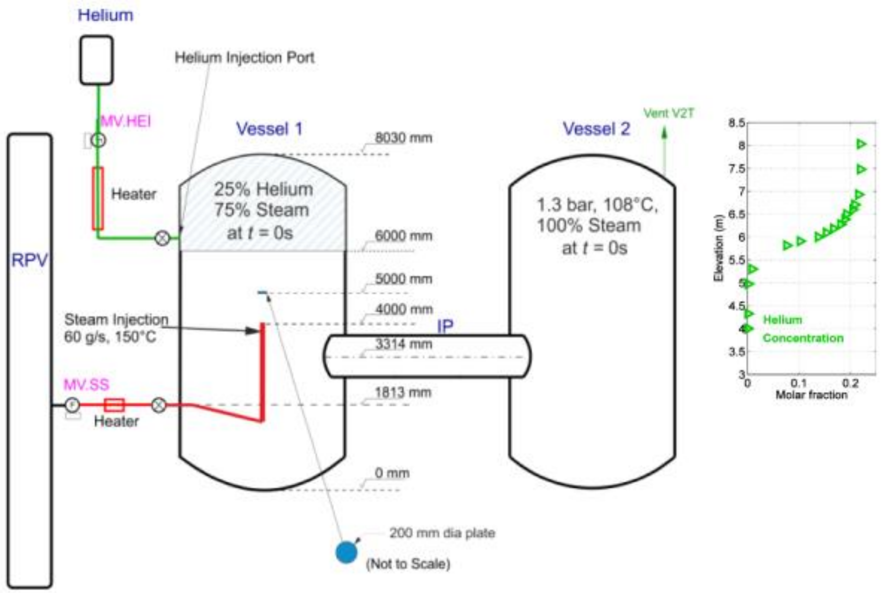


Summary of OECD/NEA International Benchmark Exercise

격자수 (단위: M)

600

Max. gri



n. g

Table 1
Summary of submissions for the blind benchmark.

Organisation *Contribution*	Code	Number of cells $\times 10^3$
AERB	FLUENT 16	163
BARC	CFD-ACE+	164
CIEMAT	FLUENT 15	135
FZJ	CFX-15	565
GRS	CFX-15	1412
IBRAE *IBRAE*	CFX-12	110
IBRAE *IBRAE2*	FLUENT 14.5.7	355
KAERI	OpenFOAM 2.3.1	158
S/NRA/R	FLUENT 15	655
PSI *PSIF*	FLUENT 15	560
PSI *PSIG*	GOTHIC 8.1	20
SPICRI	CFX-13	1284
VTT	FLUENT 16	2872

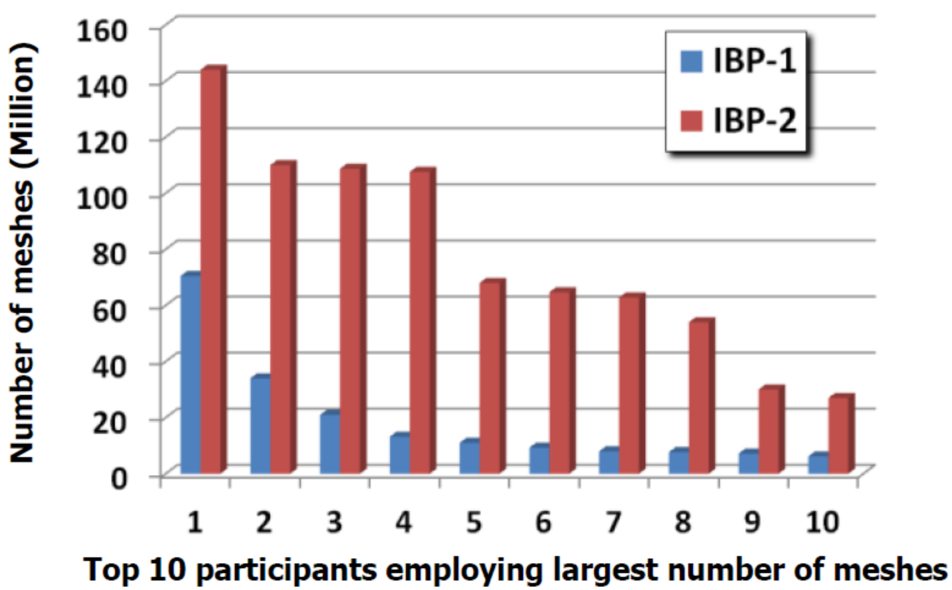
IBE-1에서는 격자조밀도와 정확도가 어느 정도 ㅂ
조밀도와 정확도는 큰 관련이 없었음.

CFD4NRS-4 Synthesis 발표자료에서 발췌

Grid Dependency of Errors

❑ Comparison between IBE-1 and IBE-2 In Terms of Meshes

- Increase in the Number of Mesh Used in IBE-2 mainly Due to:
  - Complexity of the Geometry and in Resultant Flow Behavior,
  - Improvement in Computing Power in Nuclear Community Recently.



Summary of OECD/NEA International Benchmark Exercise

Boyd (2016, NED)

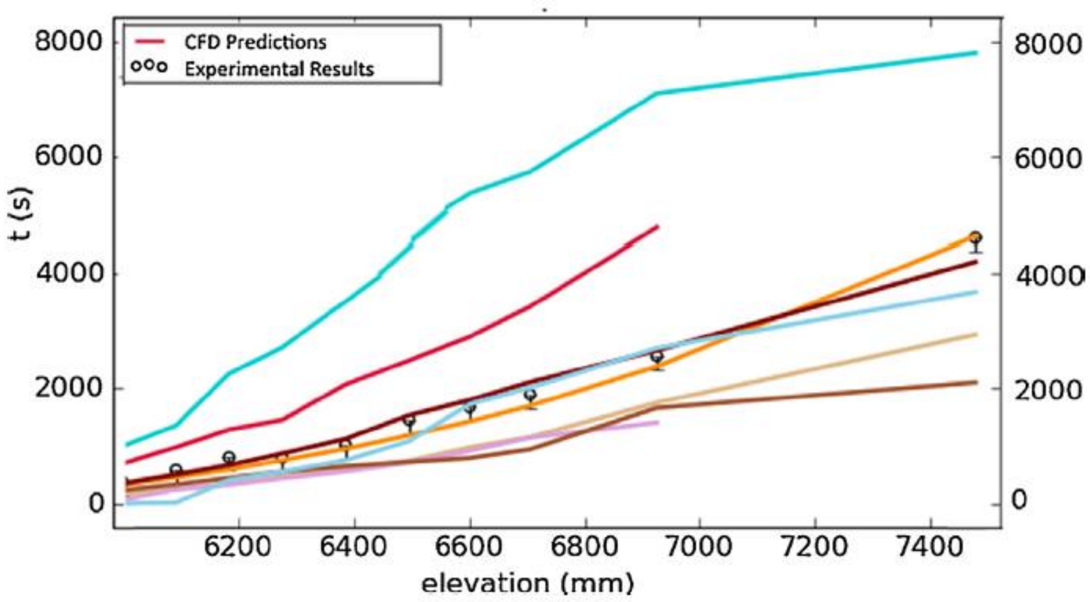


Fig. 1. Predicted helium layer erosion times compared to test data.

난류모델 등 물리모델이 불확실도에 많은 역할을 함.

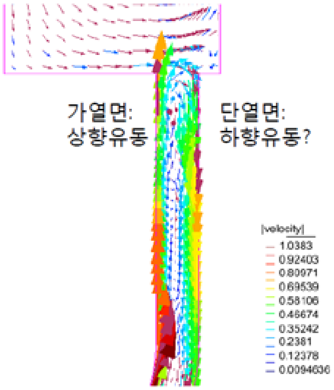
Summary of OECD/NEA International Benchmark Exercise

불확실도에 대한 고려

정재준 (2015, 안전해석 심포지엄)

안전해석 인프라 강화

- **Verification and Validation**
  - 계통분석코드 >> 2-Phase CFD 코드
  - 계통분석코드 >> 중대사고코드
- **Uncertainty Quantification**
  - 계통분석코드: 성숙 단계
  - 2-Phase CFD 코드, 중대사고코드: ?

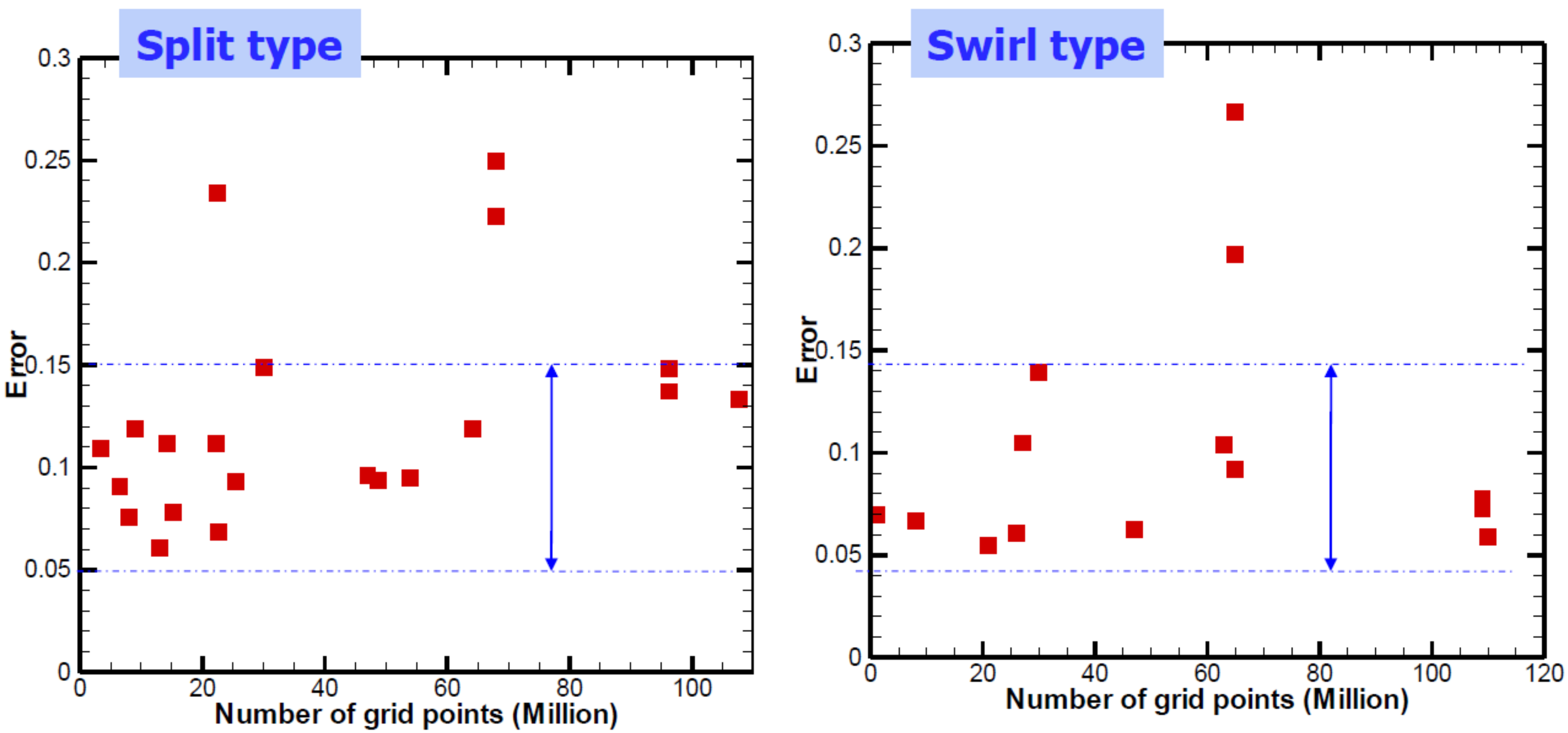


OECD/NEA IBE-2에 대한 개인적 추가 연구

CFD4NRS-4 Synthesis 발표자료에서 발췌

Grid Dependency of Errors

❑ **Ranking by the Number of Mesh: IBE-2 (at y=0.5P, z=1.0D$_H$)**



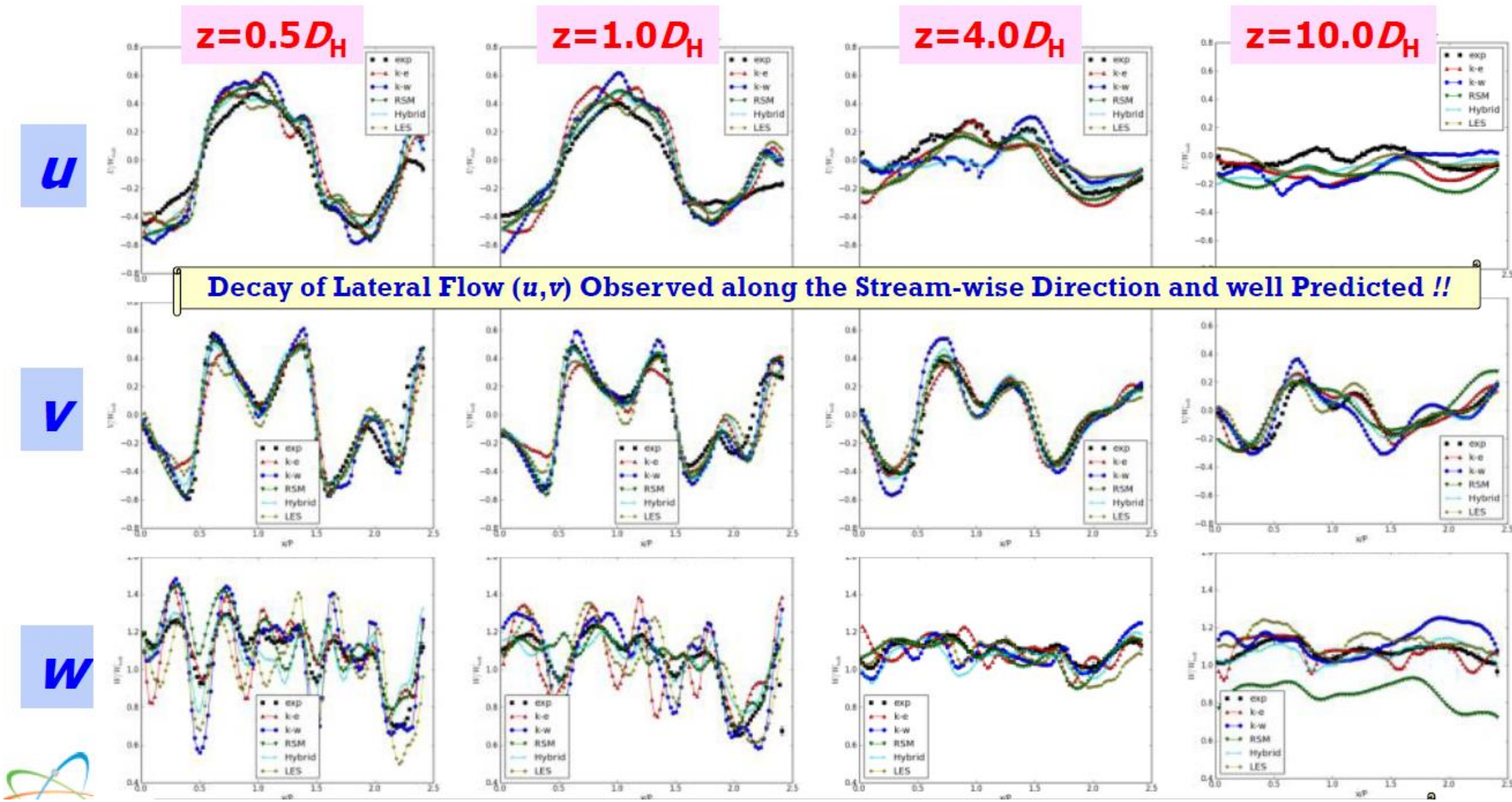
No Large Error Found even with Smaller Number of Mesh !!

Flo

&

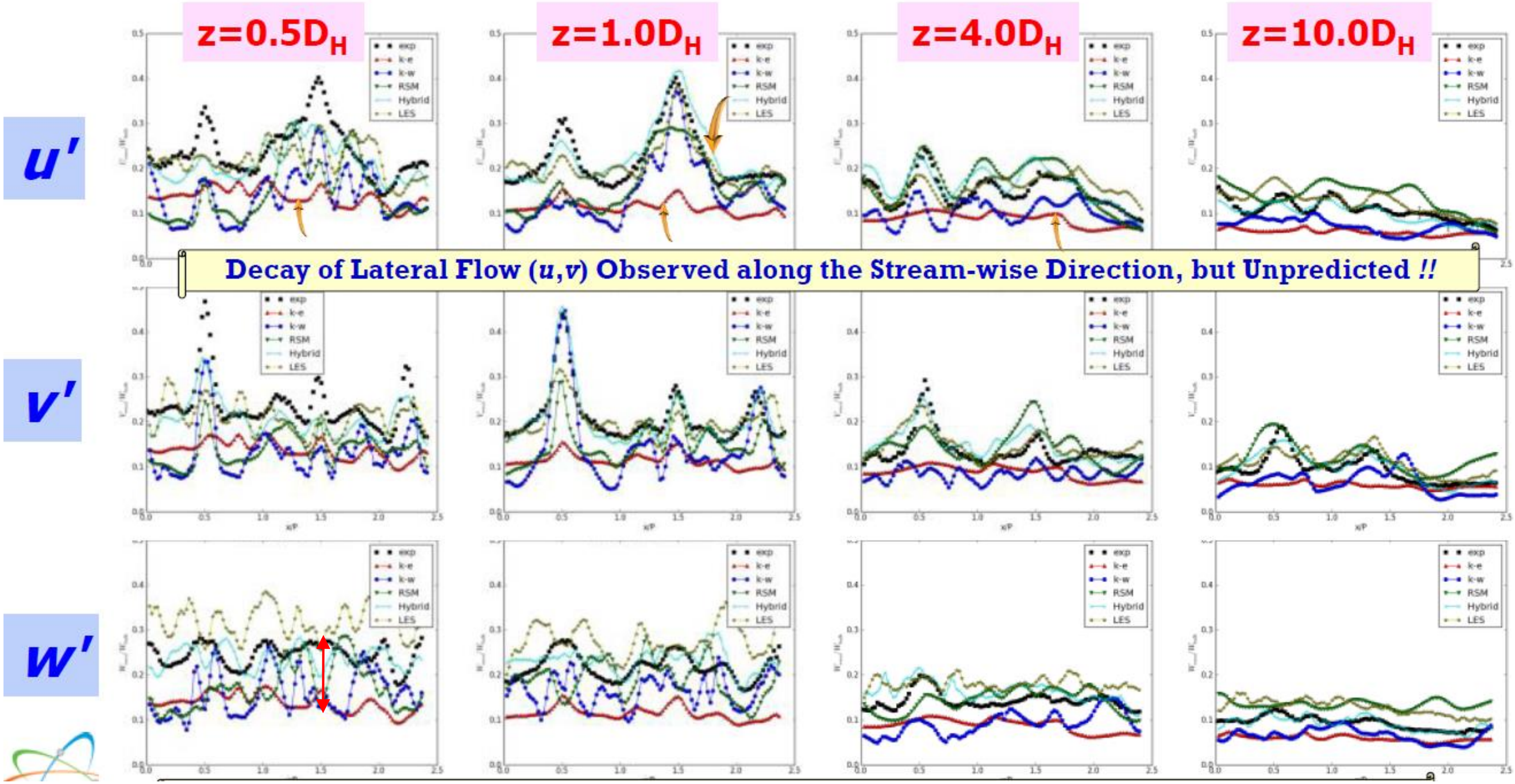
KAERI

❑ Evolution of Mean Velocity along the Flow (along y=0.5P)



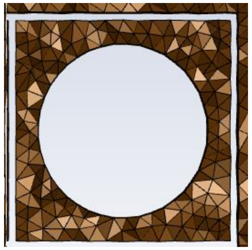
Decay of Lateral Flow (u,v) Observed along the Stream-wise Direction and well Predicted !!

OECD/NEA IBE-2에 대한 개인적 추가 연구

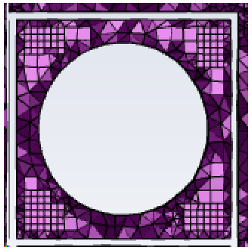


OECD/NEA IBE-2에 대한 개인적 추가 연구

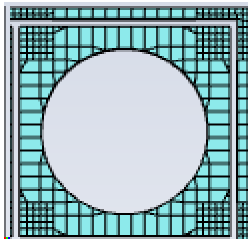
Tet



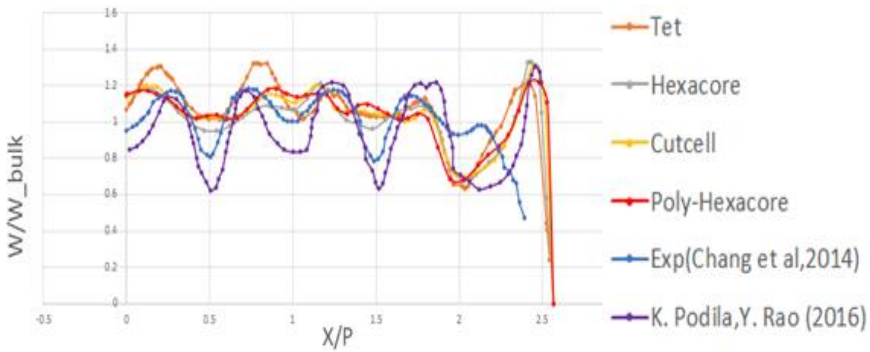
Hexacore

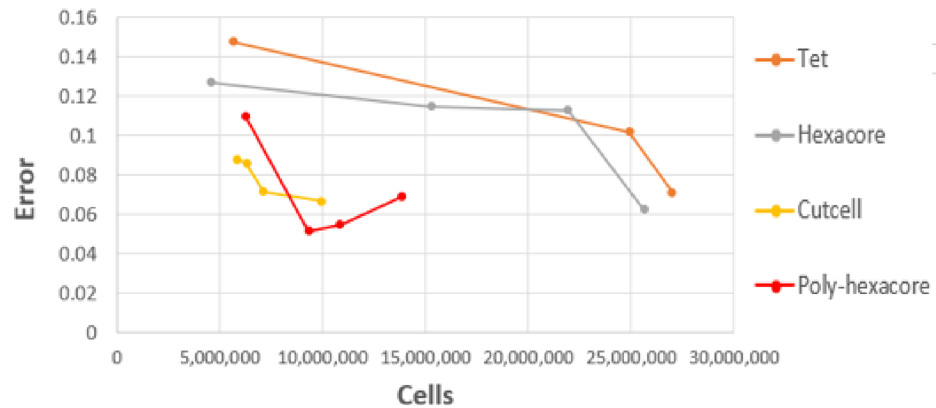


Cutcell



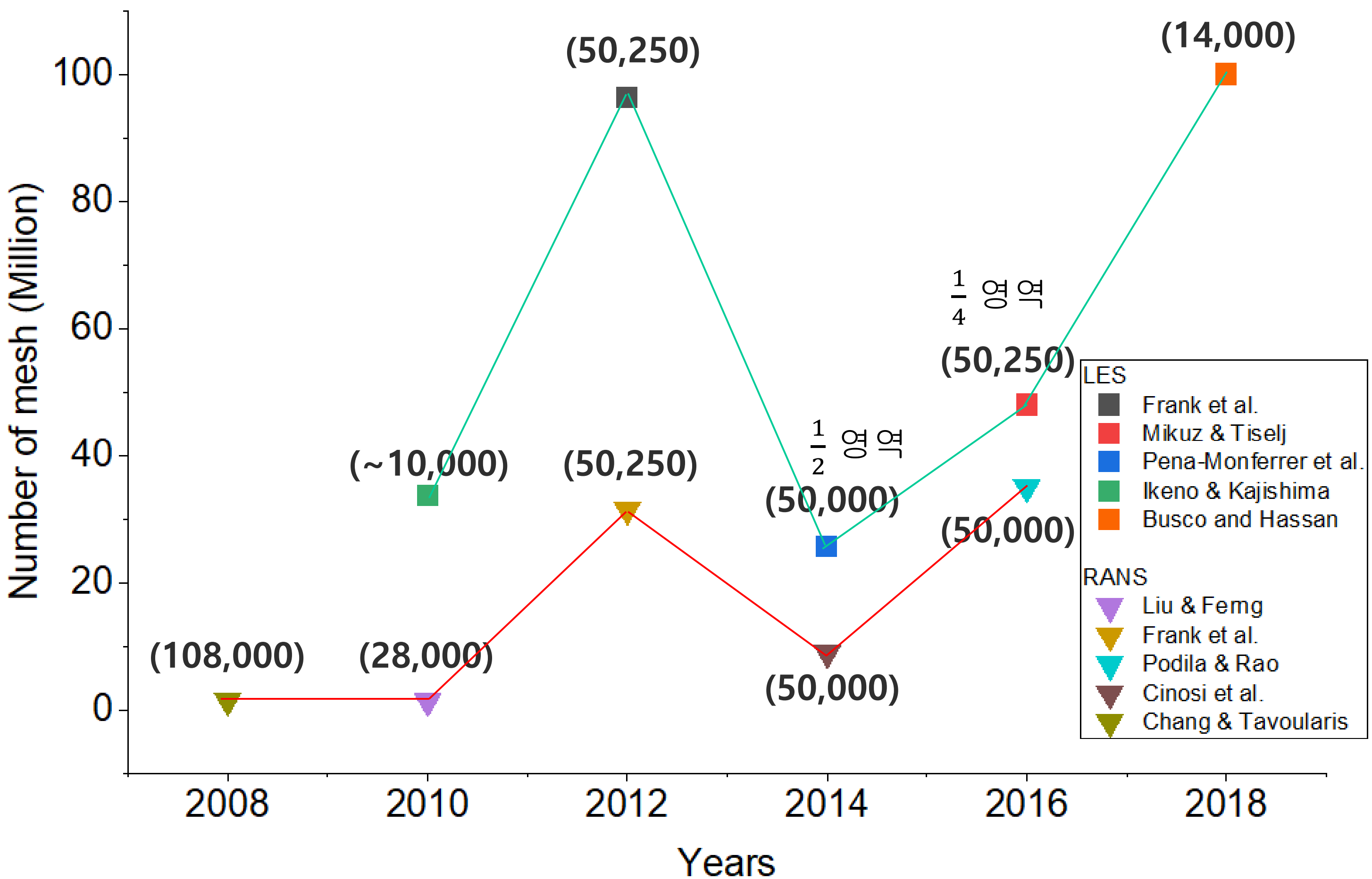
Poly-hexacore





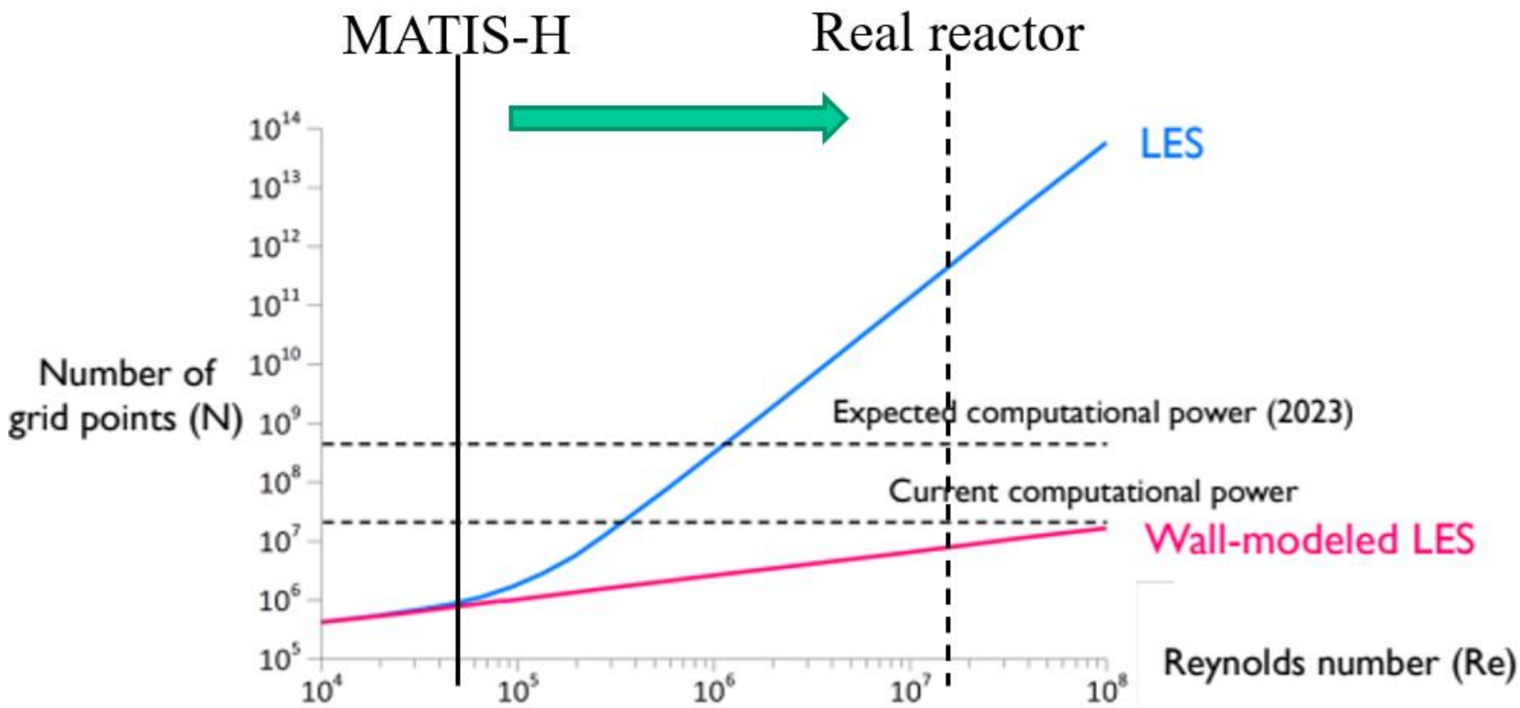
- 격자구성 방식에 따라 결과 차이가 있으나, 격자수렴된 결과는 유사함.

OECD/NEA IBE-2에 대한 개인적 추가 연구



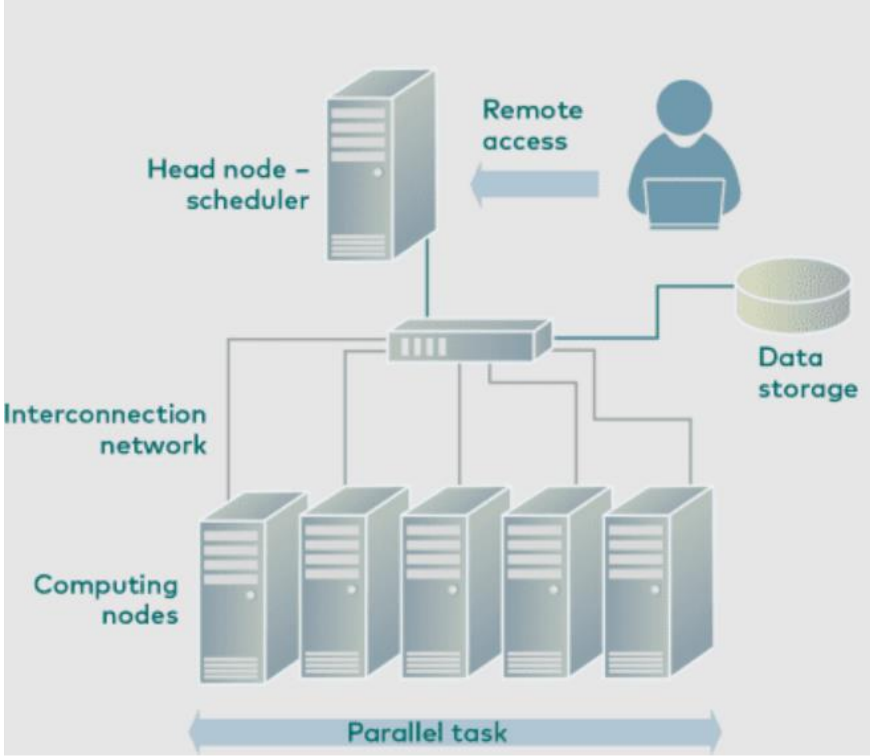
* (Reynolds Number)

OECD/NEA IBE-2에 대한 개인적 추가 연구



Piomelli and Balaras, Annu. Rev. Fluid Mech. 2002

OECD/NEA IBE-2에 대한 개인적 추가 연구

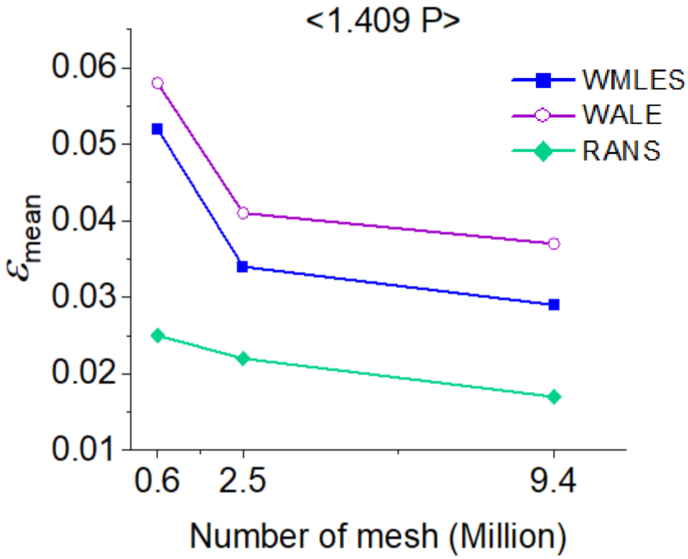


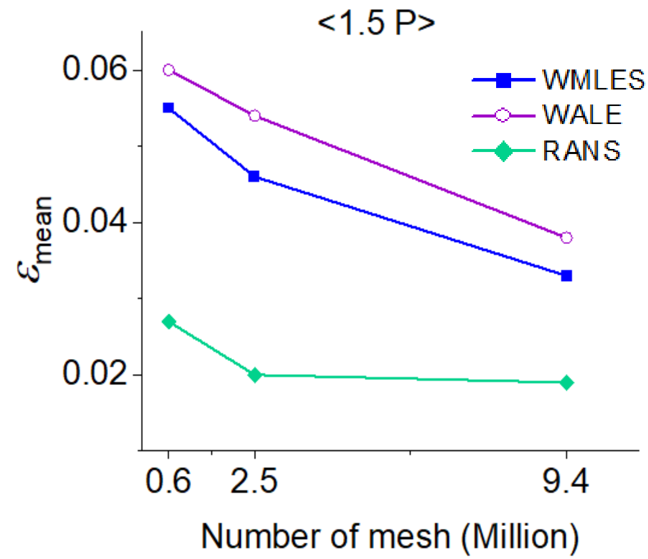
(코어 120개 기준)

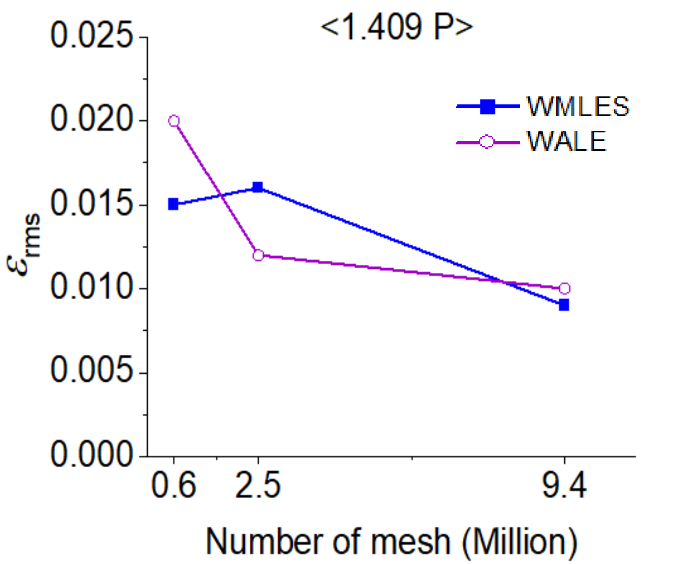
하드웨어유형			
H0 Type, Batch / Intel Xeon Platinum 8151 3.4GHz			
코어/서버	24	총 코어	120
메모리/서버	384.0 GB	총 메모리	1.9 TB
스토리지/서버	192.0 GB	기본제공 스토리지	960.0 GB
추가 스토리지	0 Bytes	총 스토리지	960.0 GB

- 고성능 컴퓨팅은 데이터를 신속히 처리하고 복잡한 계산을 수행하는 것을 의미함.
- 가장 잘 알려진 유형의 HPC 솔루션은 슈퍼 컴퓨터이며, 수천 개의 컴퓨팅 노드가 포함되어 병렬 처리 계산이 가능하며 서로 네트워크로 연결되어 있음.

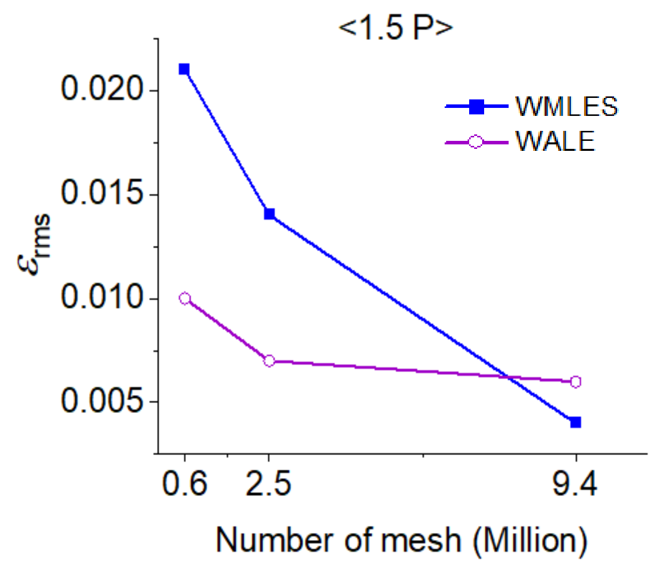
OECD/NEA IBE-2에 대한 개인적 추가 연구





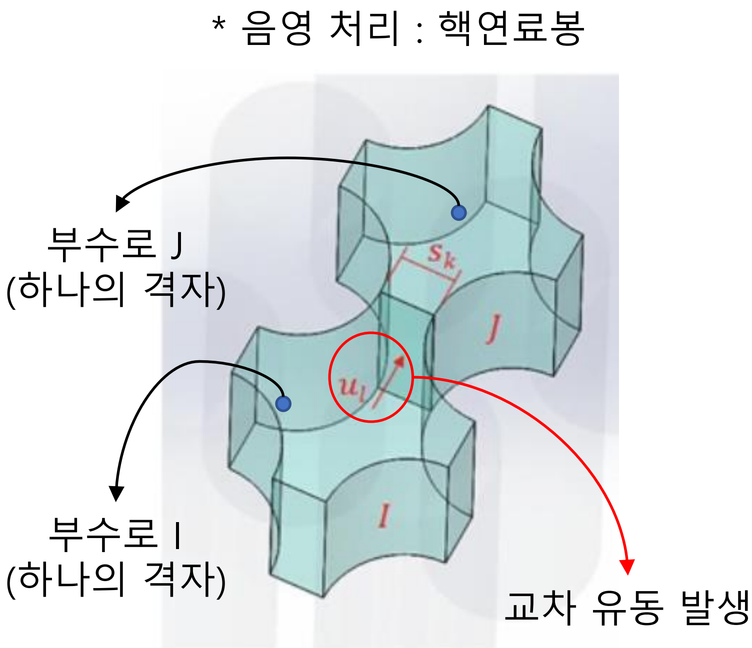


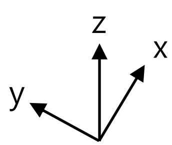
[핵연료봉 근처 영역]



[핵연료봉 간극의 중심 영역]

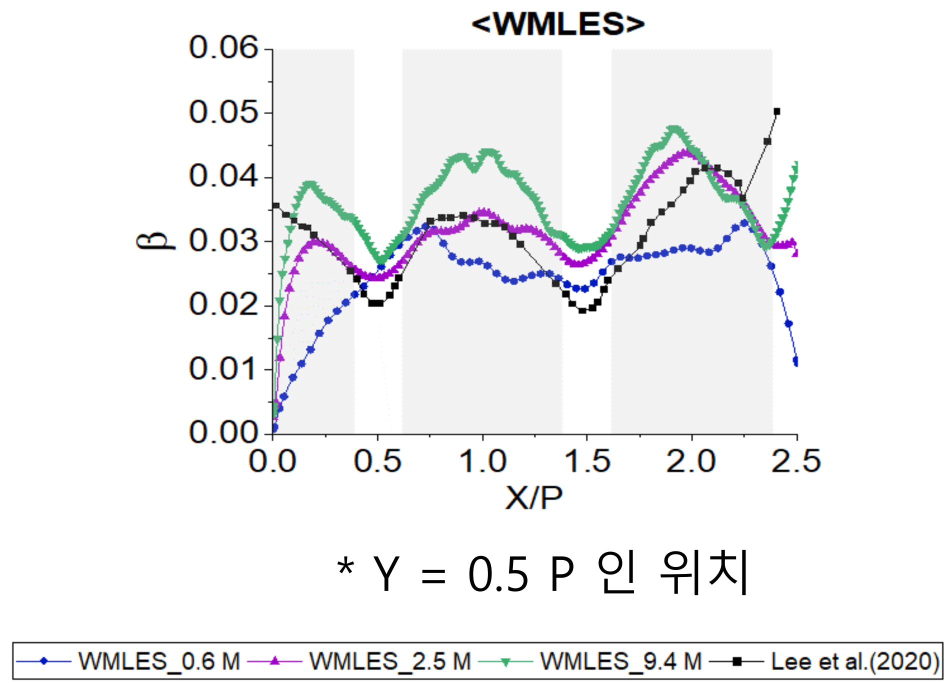
OECD/NEA IBE-2에 대한 개인적 추가 연구





$$CFI_{i,j} = \frac{1}{s_k} \int_{sk} \frac{u_l}{W_{bulk}} ds$$

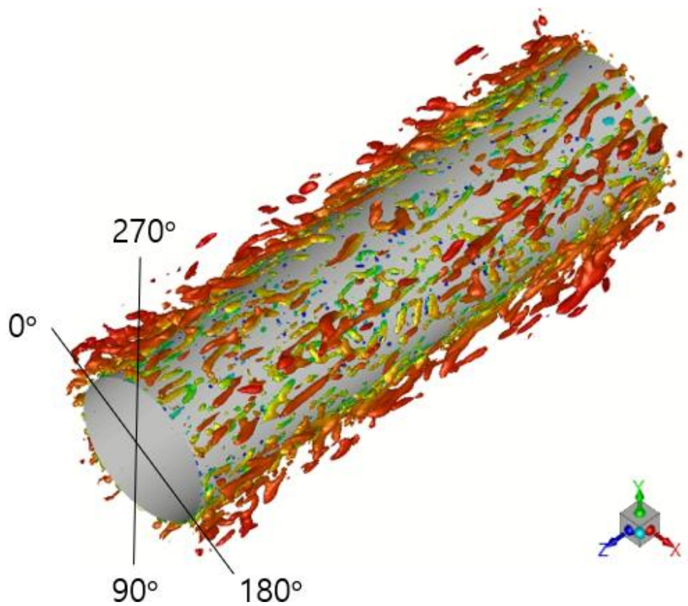
<교차 유동 크기>



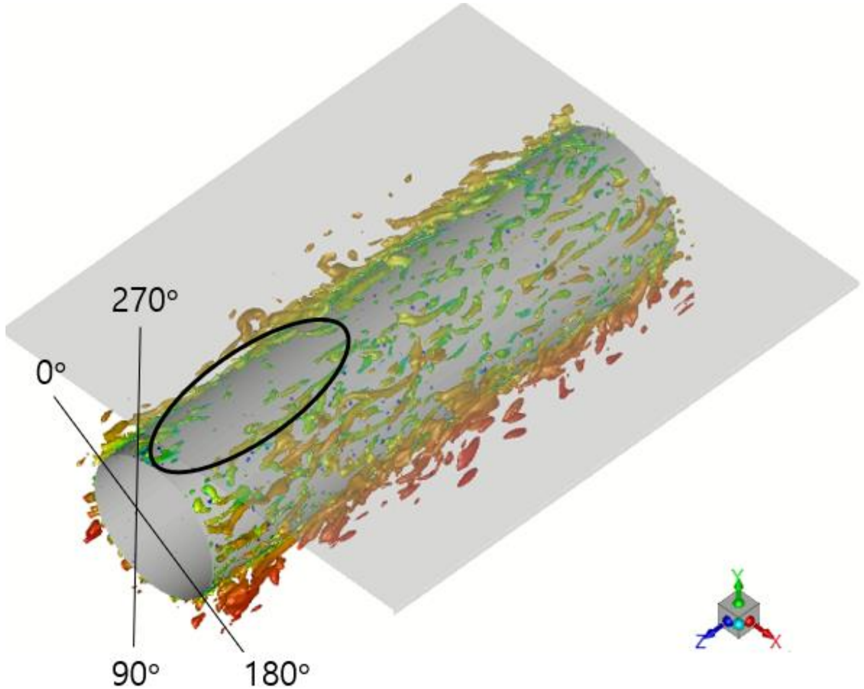
- 혼합계수 모델값을 상당히 정확히 확보할 수 있음.

→ RANS에서는 정확한 RANS값을 얻는 데 한계가 있음.

OECD/NEA IBE-2에 대한 개인적 추가 연구



<3^{rd} Rod>

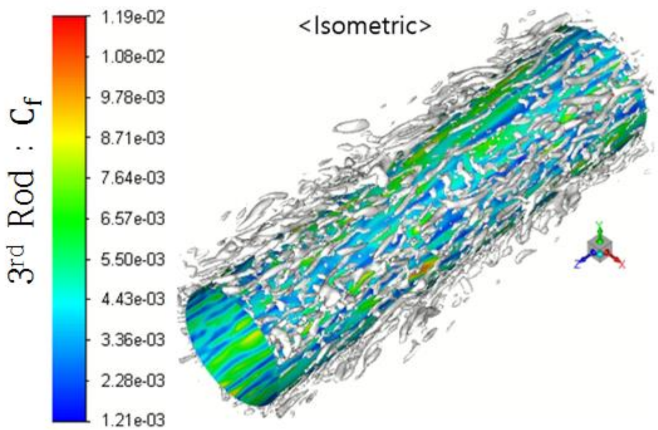


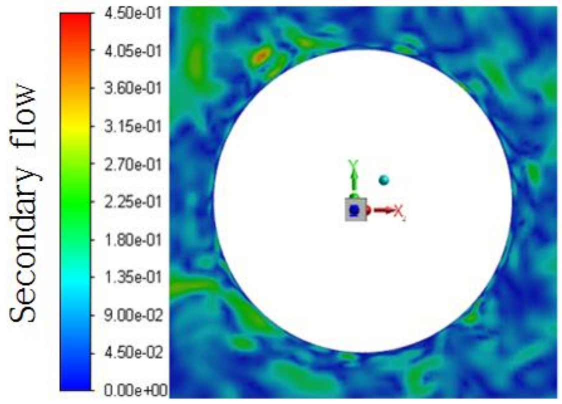
<5^{th} Rod>

- WMLES를 수행할 경우 위와 같이 봉다발 주위 유동구조를 파악할 수 있음.

→ 벽 근처 유동 구조는 벽마찰 또는 벽열전달에 영향을 미침.

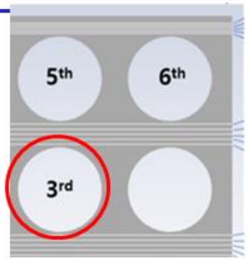
OECD/NEA IBE-2에 대한 개인적 추가 연구

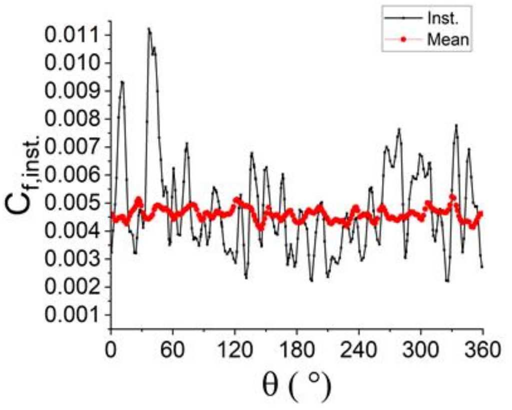


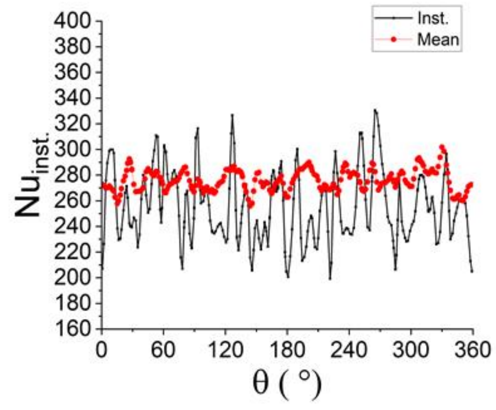


Secondary Flow :

$\sqrt{U^2 + V^2}$ [m/s]





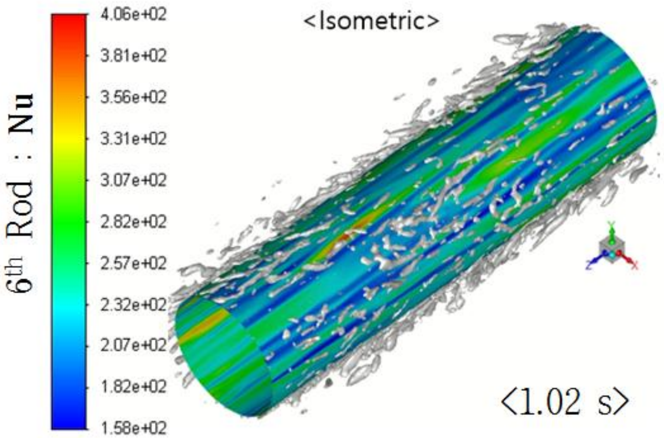


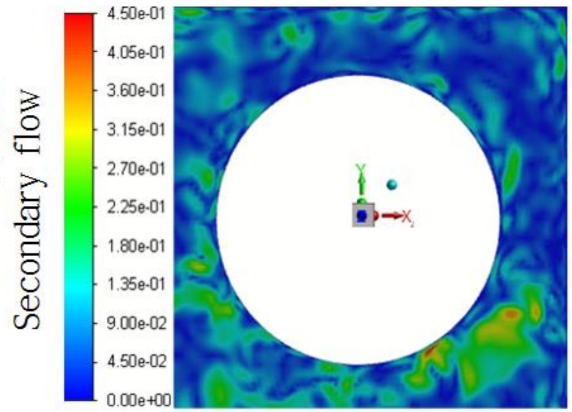
- 약 45°에서 평균 대비 순간 C_f 값이 약 120 % 정도 크게 발생함.

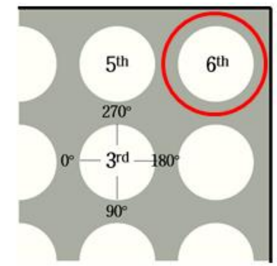
→ 45° 위치의 봉 표면에 비교적 높은 2차 유동이 발생한 것을 고려한다면, 2차 유동이 C_f 에 영향을 미친다고 볼 수 있음.

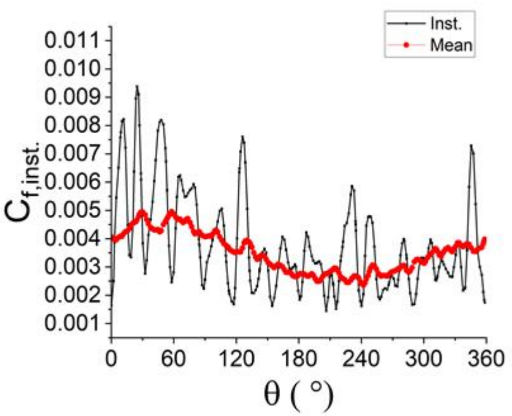
- peak 가 발생하는 위치 이외 : 대체로 순간 C_f 값은 27% 이내, 순간 Nu 수 값은 11% 범위 내 속하였다.

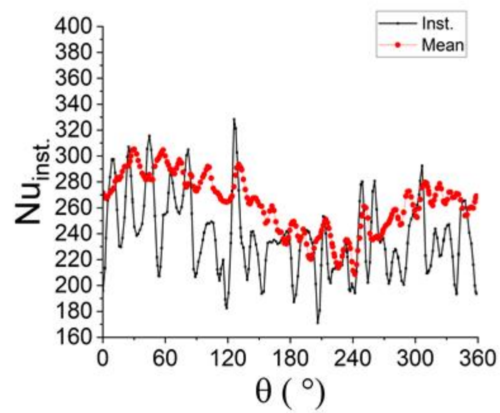
OECD/NEA IBE-2에 대한 개인적 추가 연구











- 5번째 봉보다 더 넓은 범위(180° ~ 300°)에서 2차 유동, 순간 C_f 및 순간 Nu 수 분포 모두 낮게 예측되었음. 이는 덕트와 인접한 영역이며, 유동 구조가 다소 적은 것을 확인할 수 있음.
- 대체로 순간 마찰 계수에 대한 표준편차는 30 %, 순간 Nu 수에 대한 표준편차는 13 % 내외임. 130° 인 위치에서는 순간 마찰 계수가 평균 대비 약 85.7 % 만큼 더 크게 예측하였음.

결론

• OECD/NEA IBE는 원자력 열수력 분야 CFD 국제공동연구임.
→ 2상보다는 단상 난류 현상에 대한 연구가 활발히 진행되는 측면이 있음.

• 한편 IBE 외 프랑스 CEA 등을 주관으로 다른 국제적인 CFD 관련된 연구도 최근 이루어지고 있음. 프랑스 CEA 주관 연구는 2상 비등에 대한 것임.

• 난류해석방법, 격자구성에 따라 개별 IBE 해석 결과 정확도는 상이함.
→ IBE 기간 내에는 엄밀한 격자조밀도 연구를 하기 어려운 관계로 후속 연구를 통해 격자구성 등에 따라 추가연구가 필요할 수도 있음.

• 난류해석방법 중 RANS 방식은 계산 효용성 관점에서 다수 고려되고 있음.
→ LES 와 같은 방식은 평균 유동장에 대한 정확도를 RANS와 비교하여 경우에 따라 향상시키지 못할 수 있음. 다만 RANS를 통해 확인하기 어려운 물리내용을 관찰할 수 있는 장점이 있음.

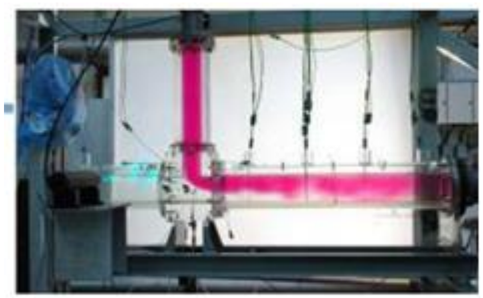
• 난류해석방법에 대한 불확실도 요인이 부정확도 요인 중 하나임.
→ 불확실도 해석에 대한 관심이 존재하나 계산 시간 문제로 활발히 진행되고 있지는 않음.

Next IBE is coming

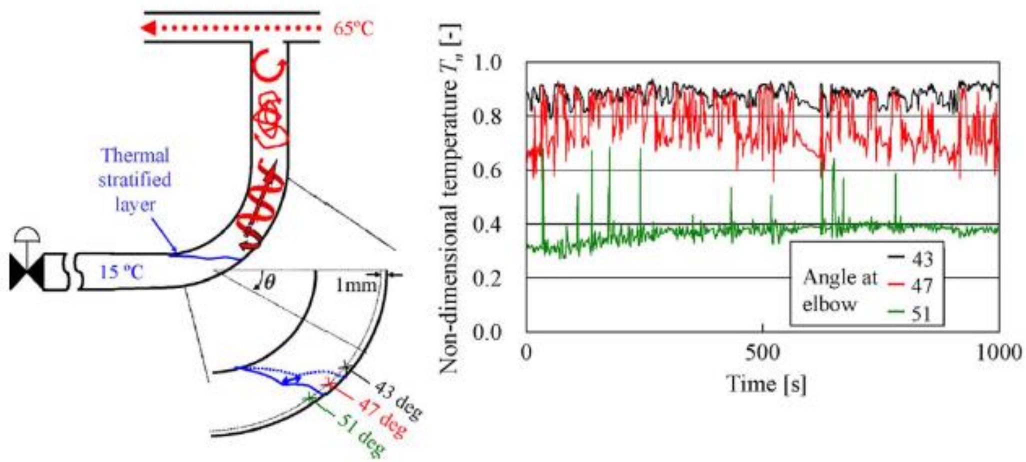
- 다음 IBE 문제는 잠정적인 참여자들에 대한 온라인 투표로 Vattenfall에서 제안한 열피로 문제가 진행될 예정임.

OECD/NEA IBE-1과 유사한 측면이 있으나, 분기관이 곡관이며 열성층이 있는 조건이 상이함.
난류모델 관점에서는 보다 어려운 문제임.

IBE-1: T-junction (2010)



IBE-6 (2022~)



Thank you for your attention!

10th CUPID Workshop

GMG Solver / CUPID Preprocessor

도 성 주

2022년 8월 23일

CONTENTS

Geometric Multi Grid Solver

1

- WHY 'Multi-Grid'?
- GMG Algorithm
- Applications

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%.**
  - Linear solver should be optimized to accelerate CUPID code.

- Requirement for new solver
  - **Faster** than PBICG in large scale simulation
  - Can be applied to **any kind of mesh** (tetrahedral, hexahedral mesh, prism, ...)
  - Efficient for **RV calculation**

Pre-process

Time iteration

Model Calculation

Intermediate Velocity

Pressure equation

Pressure correction

Solver	Complexity
CG/BICG	$O(n^{1.5})$
CG/BICG with Preconditioning	$O(n^{1.4})$
Multi-Grid	$O(n^{1})$

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

WHY 'Multi-Grid'?

[1] A Multigrid Tutorial, 2nd Edition. Briggs W, Henson V, Mccormick S

Jacobi solver

- **One of the simplest iterative linear solver**
- **Computational complexity for convergence is $O(N^2)$**
  - Impossible to use in practical areas
- **Jacobi iteration**

$$Au = f \quad \Leftrightarrow \quad (D + L + U)u = f$$

D: diagonal, L/U: lower/upper diagonal

$$\Leftrightarrow \quad Du = f - (L + U)u$$

$$\Leftrightarrow \quad u = D^{-1}f - D^{-1}(L + U)u$$

$$u^{(n+1)} = D^{-1}f - D^{-1}(L + U)u^{(n)}$$

If it is convergent, the limit is the solution of the linear system.

- **Error propagation**

$$u^{(n+1)} - u = -D^{-1}(L + U)(u^{(n)} - u)$$

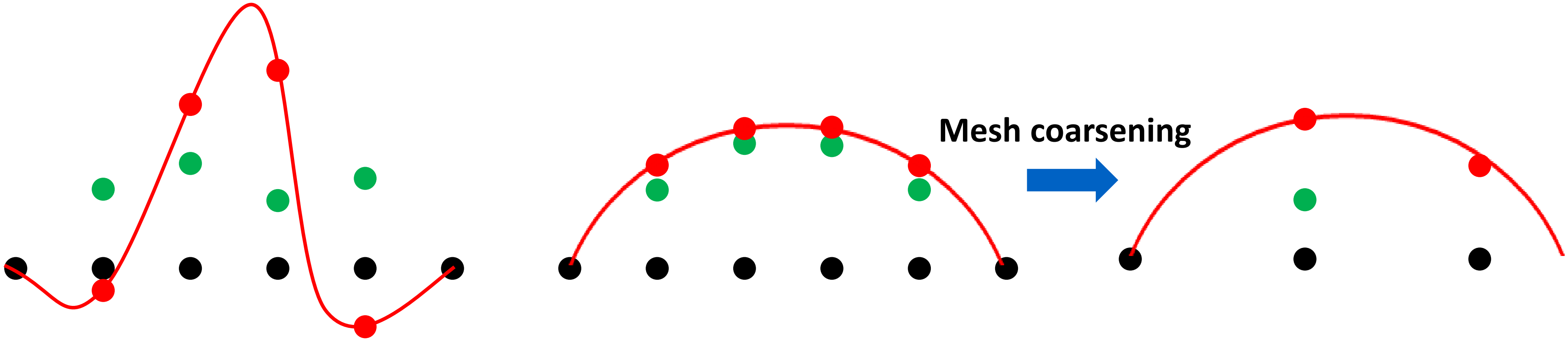
$$e^{(n+1)} = -D^{-1}(L + U)e^{(n)}$$

WHY 'Multi-Grid'?

Jacobi solver

Model equation : $\frac{d^2u}{dx^2} = f \Rightarrow A = \frac{1}{h^2}\begin{bmatrix} -2 & 1 & 0 & 0 \\ 1 & -2 & 1 & 0 \\ \vdots & \ddots & \ddots & \vdots \\ 0 & 1 & -2 & 1 \\ 0 & 0 & 1 & -2 \end{bmatrix}, \quad e_i^{(n+1)} = \frac{1}{2}\left(e_{i-1}^{(n)} + e_{i+1}^{(n)}\right)$

- **High-frequency parts of the error converge quickly, while the low-frequency regions converge very slowly.**

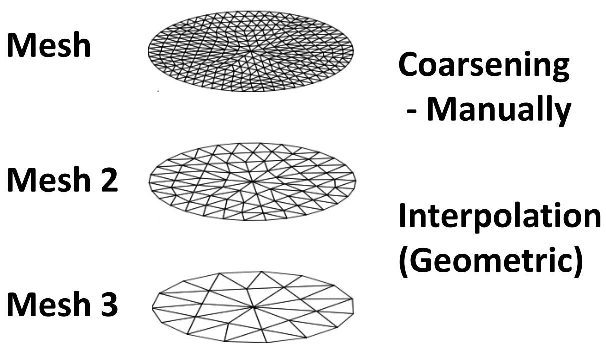


- **Low frequency data can be considered as highly oscillatory data in coarse mesh. → Multi grids are necessary.**

WHAT is GMG?

Classic MG

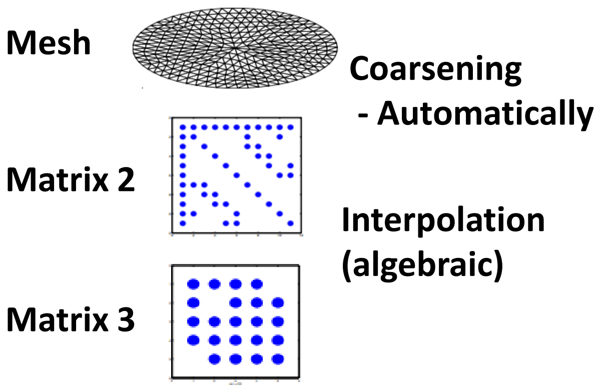
- Coarse grids should be explicitly provided
- Mainly used for structured grids



<MG>

Algebraic MG

- Mesh coarsening is not needed since the coefficients of the matrix are used → Suitable for computation on unstructured grids
- More costly in terms of operator complexity

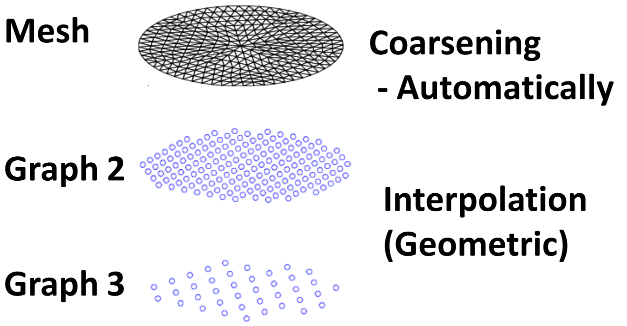


<AMG>

WHAT is GMG?

Geometric MG

- Auto coarsening is equipped
  → applicable to unstructured grids
- Use geometrical interpolation/restriction
  → less costly in terms of operator complexity
- A hybrid method of MG and AMG



<GMG>

GMG algorithm

Elements of MG solver

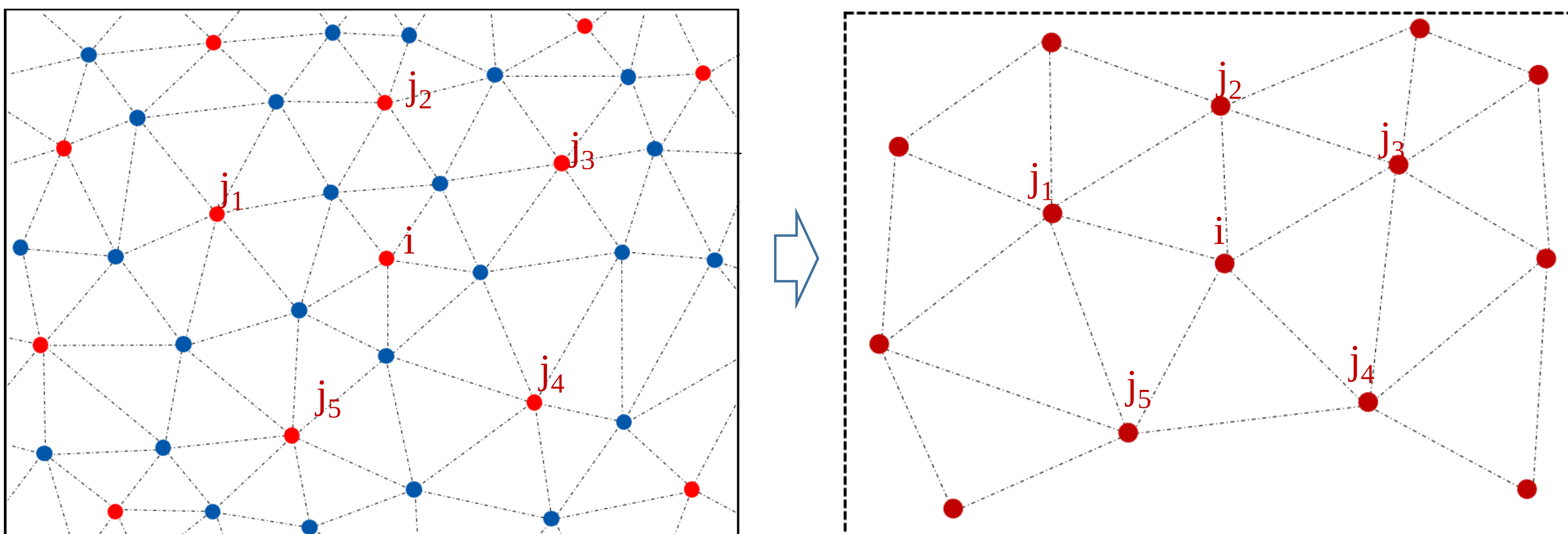
- **Coarse mesh generator**
- **Data transfer between coarse and fine meshes.**
  - Interpolator (coarse → fine)
  - Restrictor (fine → coarse)
- **Smoother**
- **V-cycle iteration**

GMG algorithm

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

Coarse mesh generator

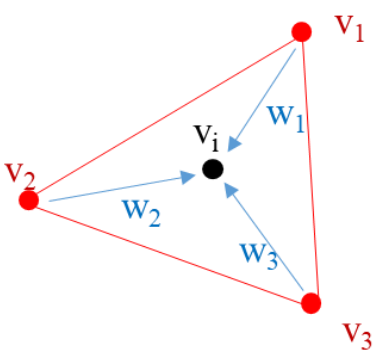
- **Only the finest mesh is required as an input data**
  - Automatic mesh generation on coarse levels
- **Node-coarsening by MIS(Maximum Independent Set [2])**
  - Initially, mark all the nodes in finer mesh '**green**'
  - And then, for each node in a finer mesh:
    - ✓ If the node is **green** add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



GMG algorithm

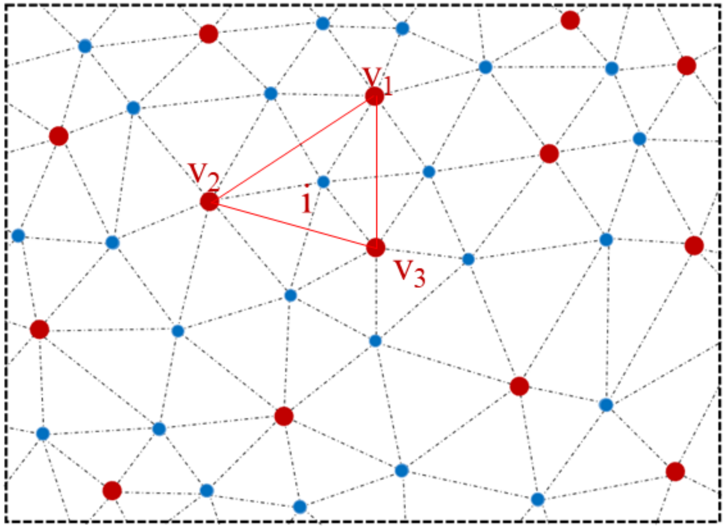
Data transfer between coarse and fine meshes

- Interpolator (coarse → fine)
  - ✓ Inverse distance

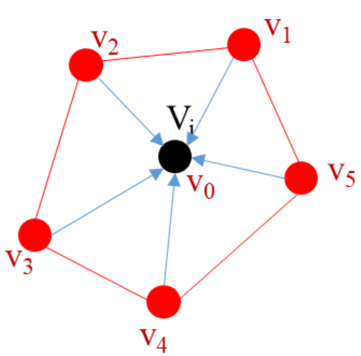


$$v_i = \frac{w_1 v_1 + w_2 v_2 + w_3 v_3}{w_1 + w_2 + w_3}$$

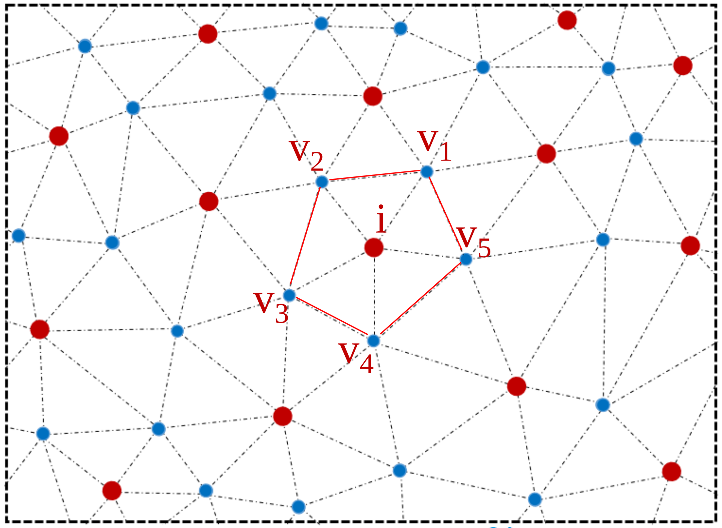
$$w_k = \frac{1}{d(v_i, v_k)}$$



- Restrictor (fine→ coarse)
  - ✓ Injection



$$V_i = v_0$$



GMG algorithm

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

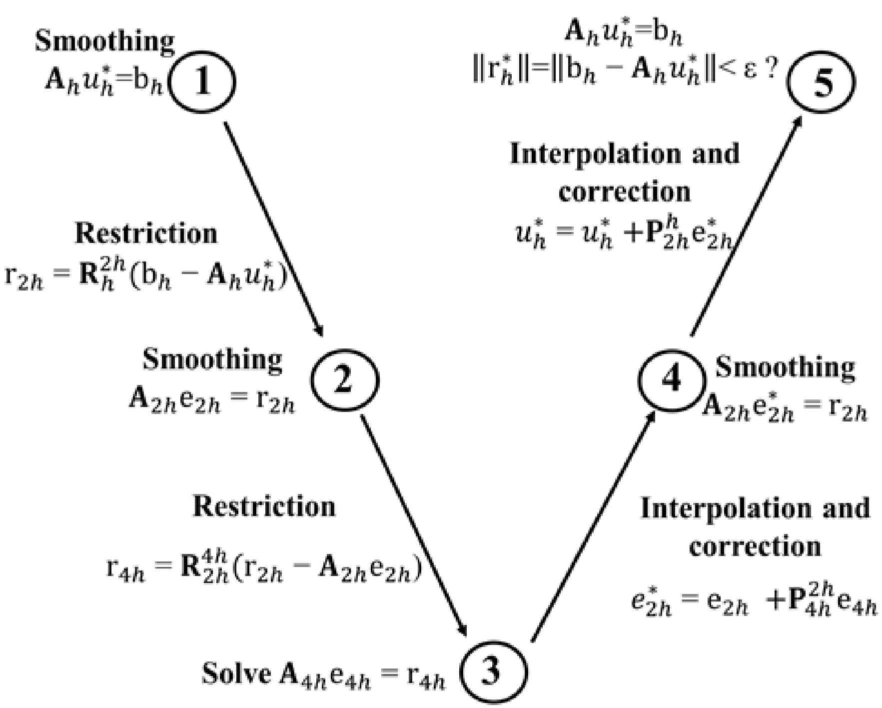
- In CUPID, 2-3 times SOR sweeps are performed on each meshes.

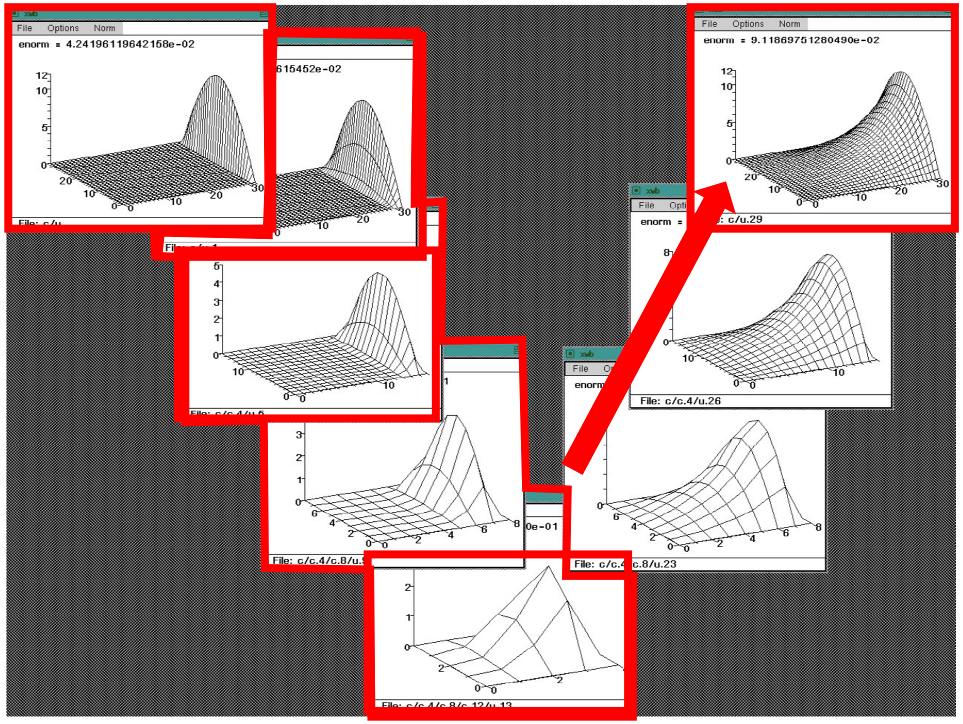
GMG algorithm

[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





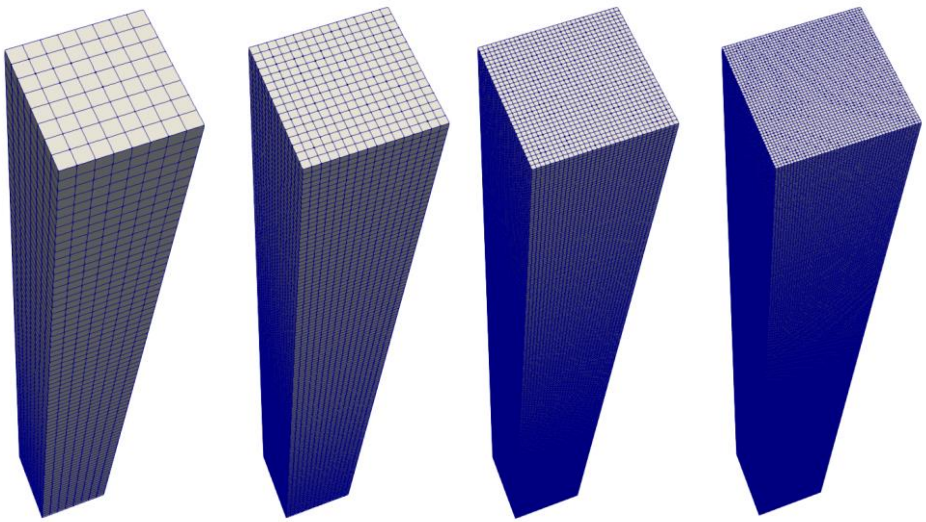
V-cycle workbench [4]

Speedup Test

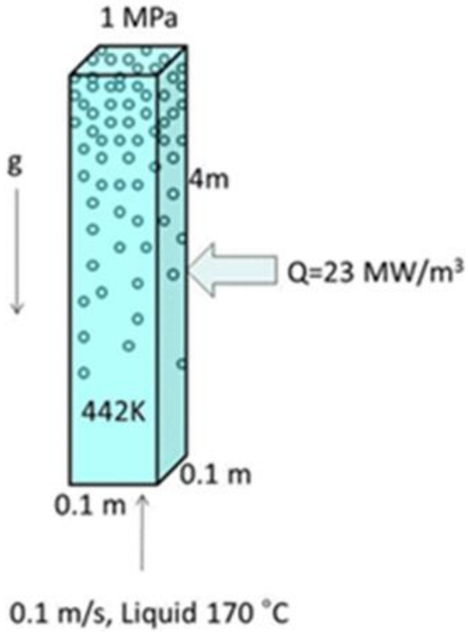
3D boiling test

- 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

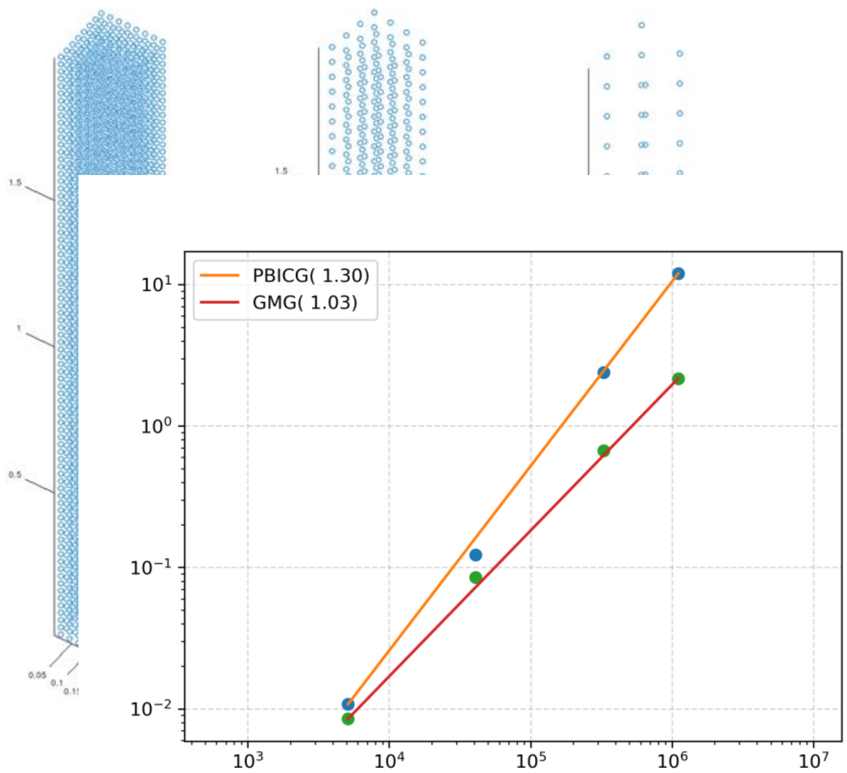


<Test setup>

Speedup Test

3D boiling test

➢ Result of automatic mesh coarsening



➢ 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.

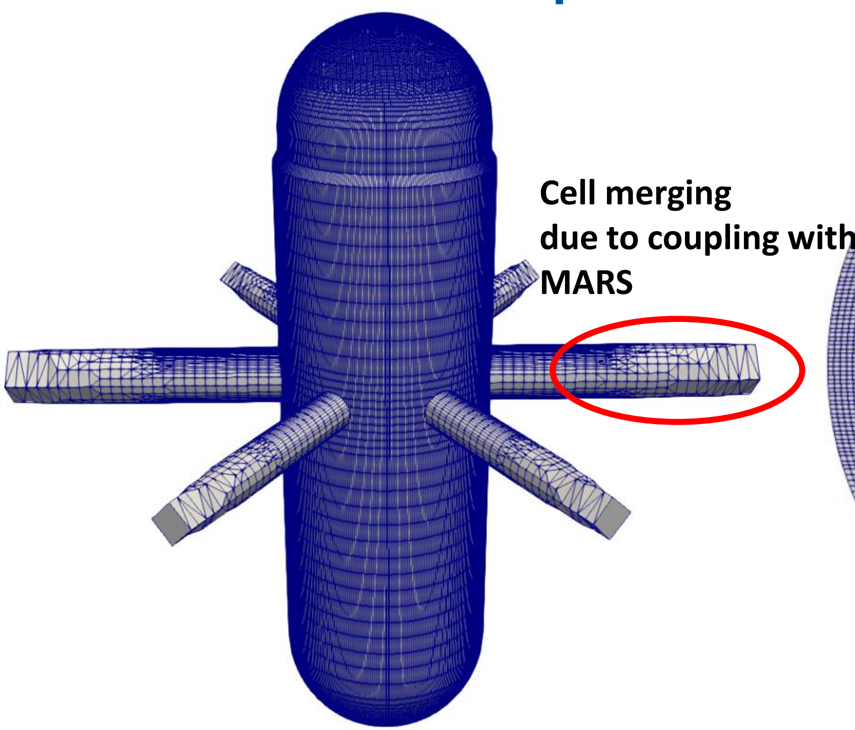
Case	Num. cells	PBICG iteration	GMG iteration	Speed up [times]
Mesh 1	5,120	37	10	0.88
Mesh 2	40,960	63	10	1.44
Mesh 3	327,680	124	10	3.91
Mesh 4	1,105,920	174	10	5.40

Constant !

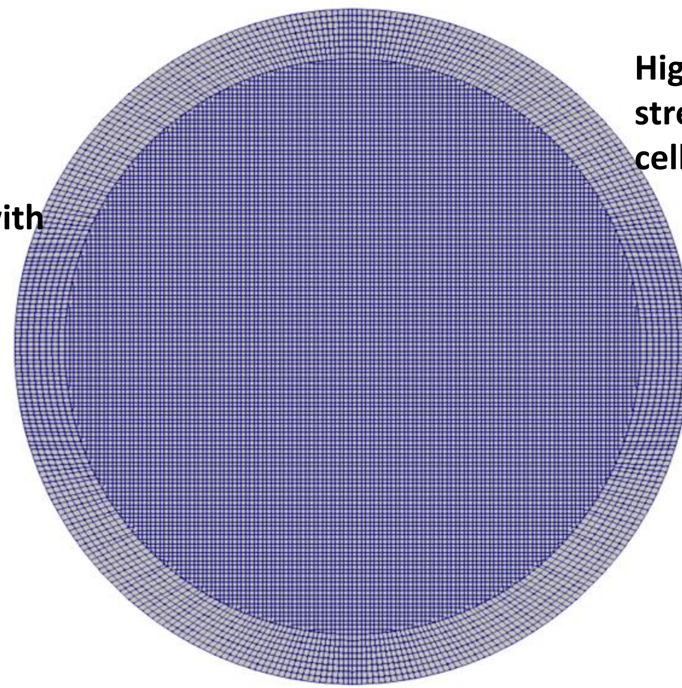
Speedup Test

RV test

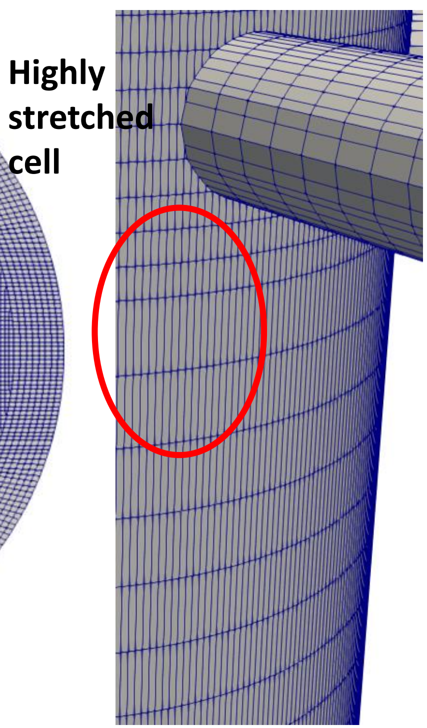
- Polygonal cells / High aspect ratio mesh
- 4 kinds of unstructured meshes are used to evaluate the performance of GMG solver



<RV mesh>



<Cross section>

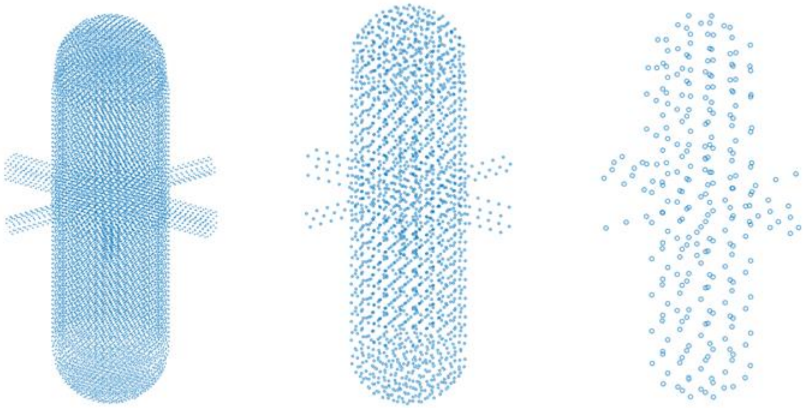


<Junction>

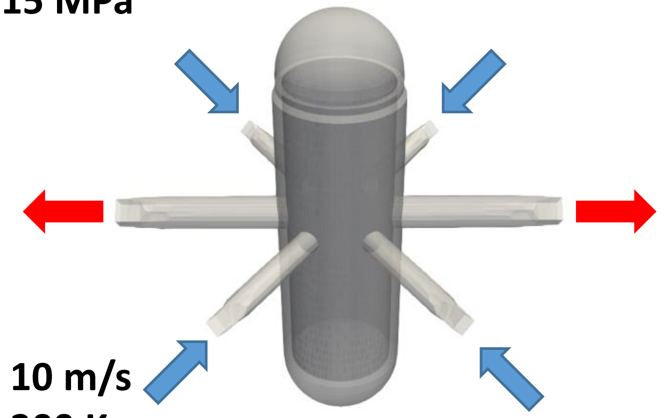
Speedup Test

RV test

Result of automatic mesh coarsening



<Nodes after coarsening>

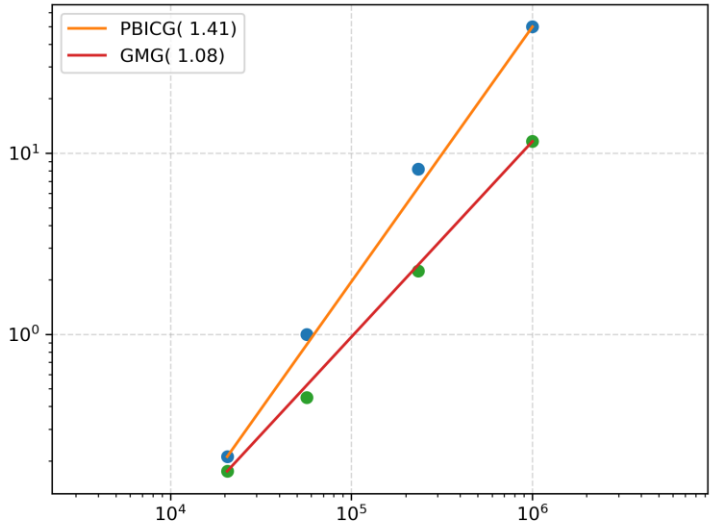


<Test setup>

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

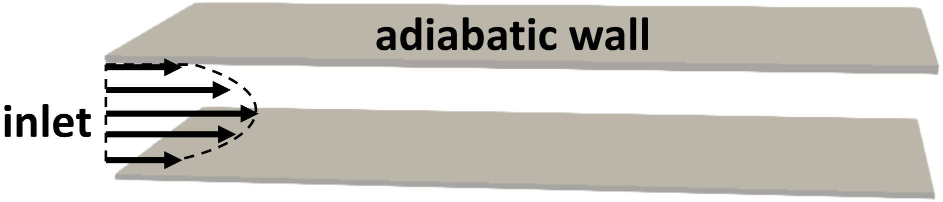
Almost constant



Speedup Test

Single phase channel Flow

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



<Channel configuration>

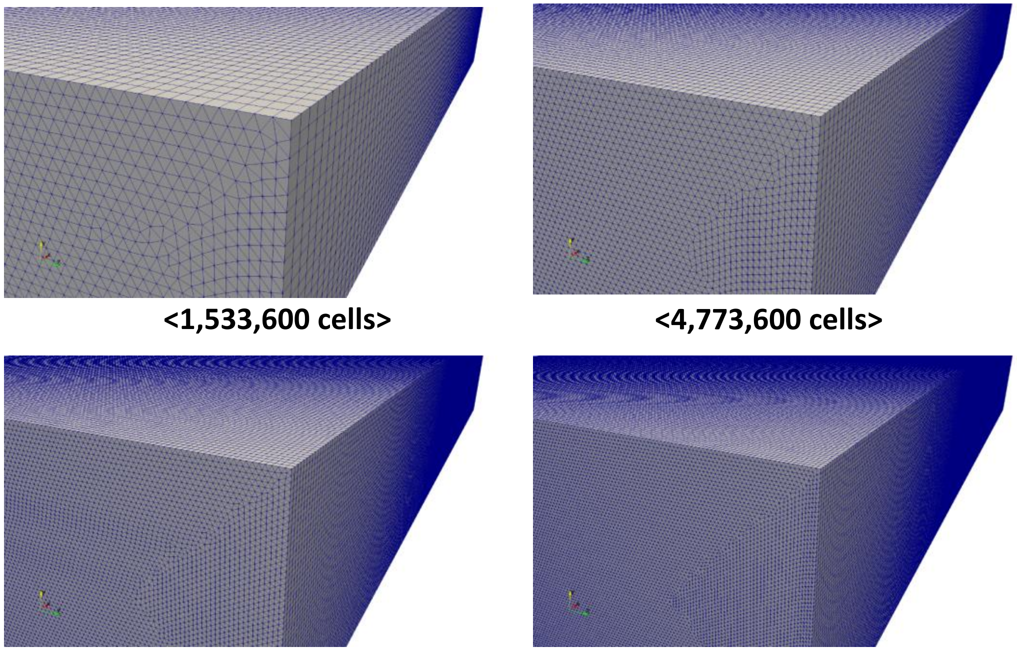
- **Computational setup**
  - Domain

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

  - 5 kinds of unstructured meshes
  - 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>

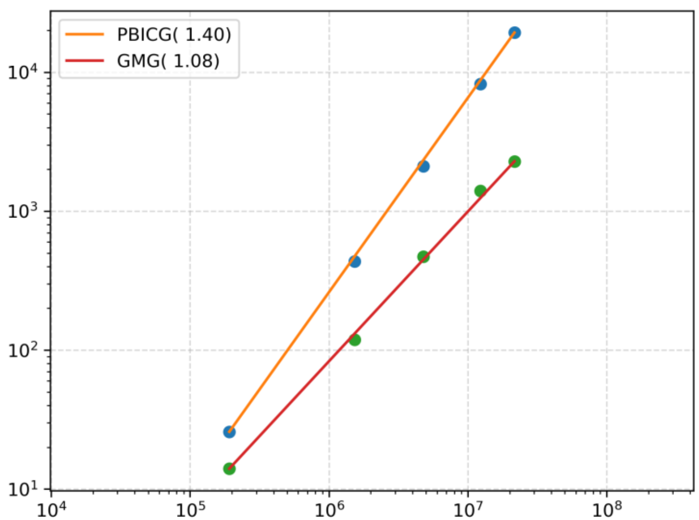
<Computational meshes>

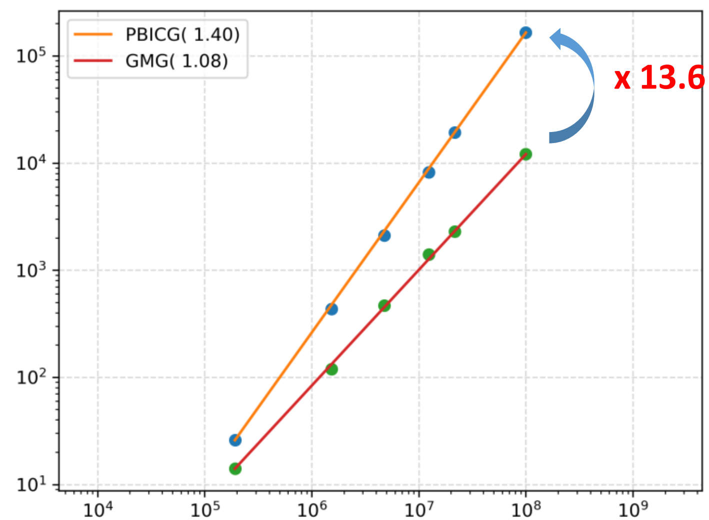
Speedup Test

Comparison of PBICG / GMG

- The number of iteration in GMG is almost *constant* regardless of the number of cells.
- In the case of 100 million cells, GMG is predicted to improve performance by about 13.6 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





CUPID Preprocessor

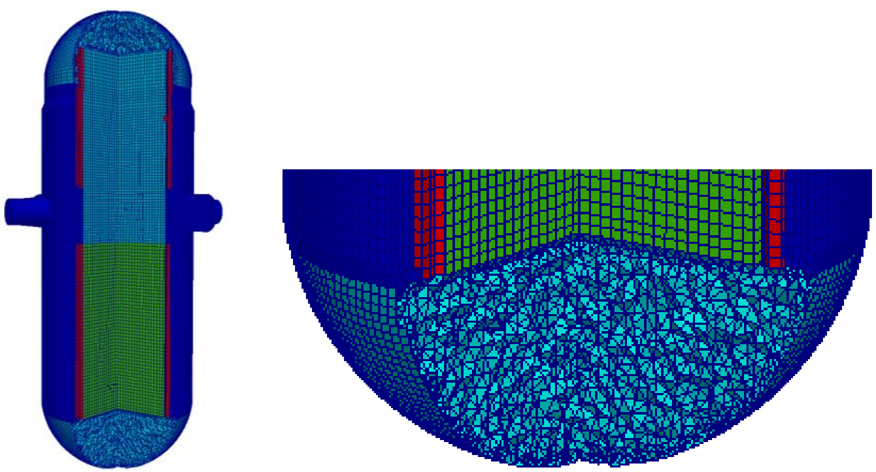
2

- Why 'RV Mesh 3D' ?
- Mesh Generation Algorithm
- Applications (PWRs / SMRs)

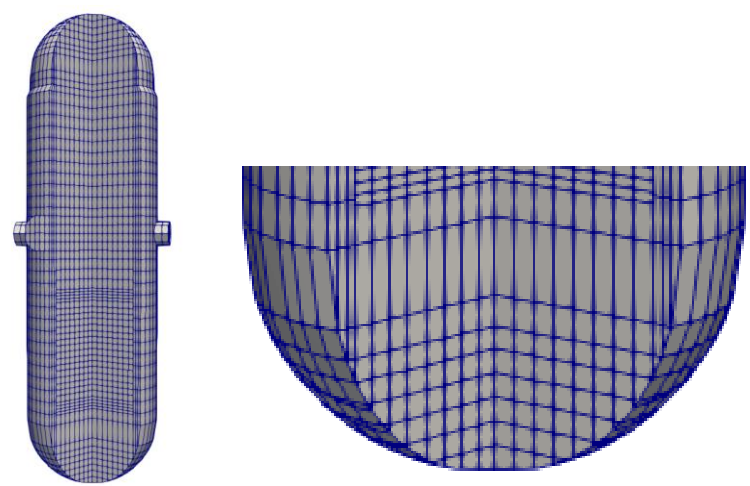
WHY ‘RV Mesh 3D’?

- **3D mesh generator dedicated to the reactor 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 PWR / SMR geometries**

It’s hard to apply commercial pre-processors



<Generated by commercial pre-processor>

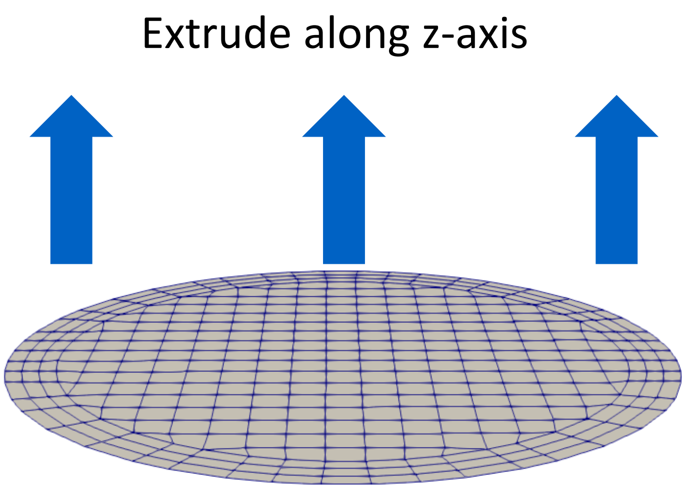


<Generated by RV Mesh 3D>

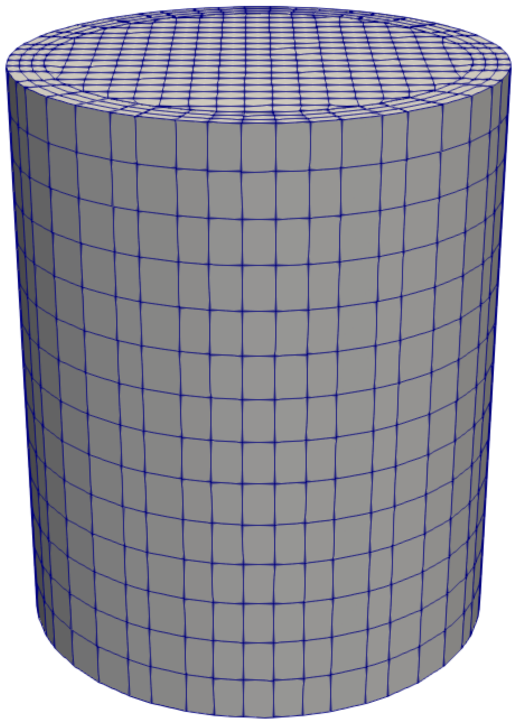
Mesh Generation Algorithm

Plane Extrusion method

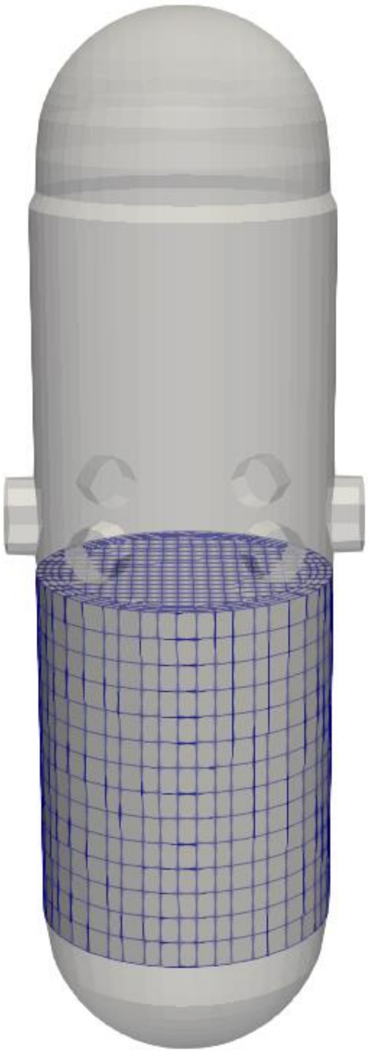
- 2D Mesh generation
- Plane extrusion along z-direction
  - Applicable area : Core / Downcomer(DC) region



<2D plane extrusion>



<3D volume mesh>

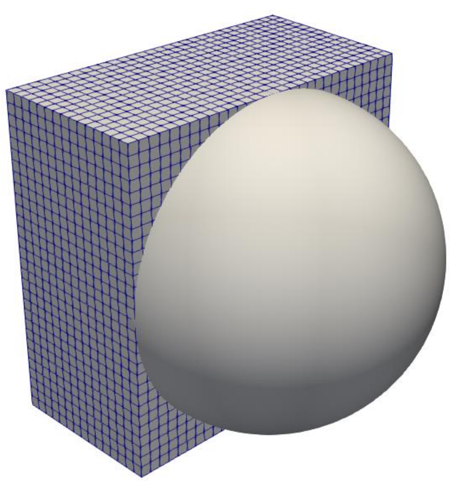


<Core / DC part>

Mesh Generation Algorithm

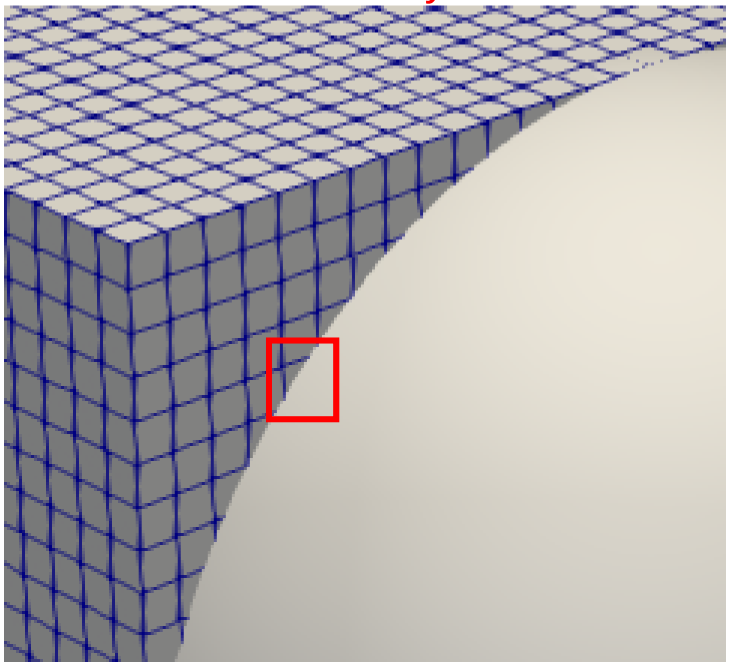
Cut-Cell method

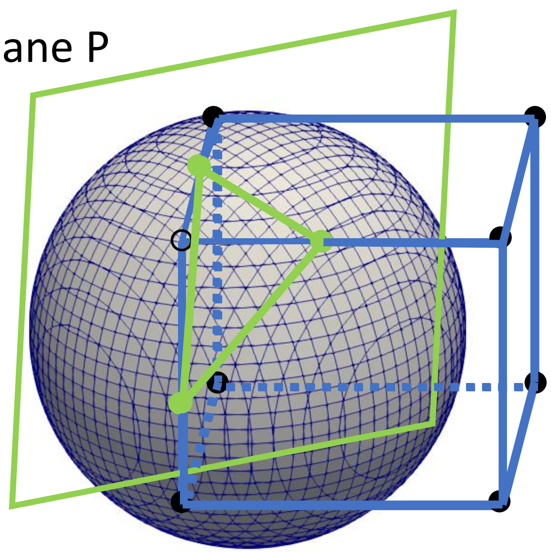
- **Representation of the Curved Surfaces**
  - Applicable area : Upper / lower plenum
- ***Cut-cell method* is applied for the curved faces**
  - ✓ Cut-cell approach uses background Cartesian grid with special treatments being applied to cells which are cut by solid bodies.



<Base mesh>

<Sphere is immersed in base mesh>

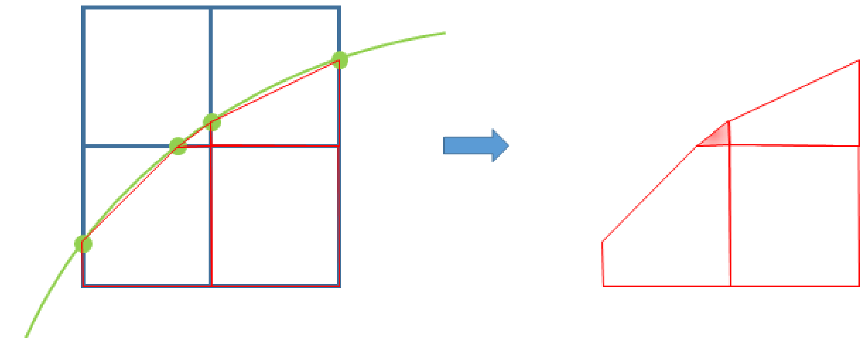




<After cut-cell algorithm>

Mesh Generation Algorithm

- **Cut-cell method may generate small cells**
  - **Small cells cause numerical instability and small time step size.**



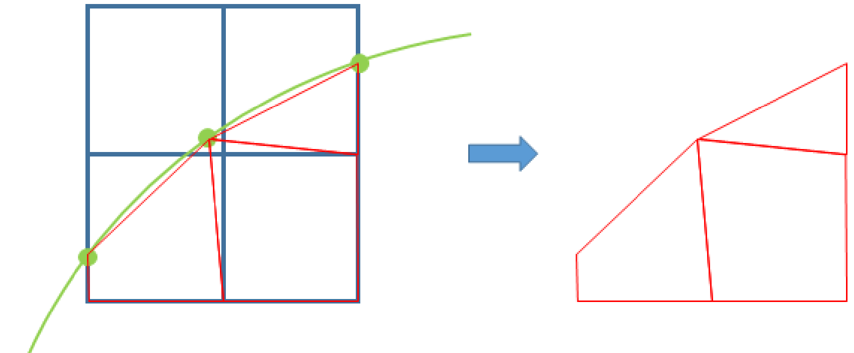
<Cut-cell method>

- **The generation of small cells can be suppressed by transforming the base grid.**

Define threshold value : ε

If $\frac{|x^* - x_l|}{|x_k - x_l|} < \varepsilon$, then move x_l to x^* so that short edge is avoided

<Quality enhancement approach>

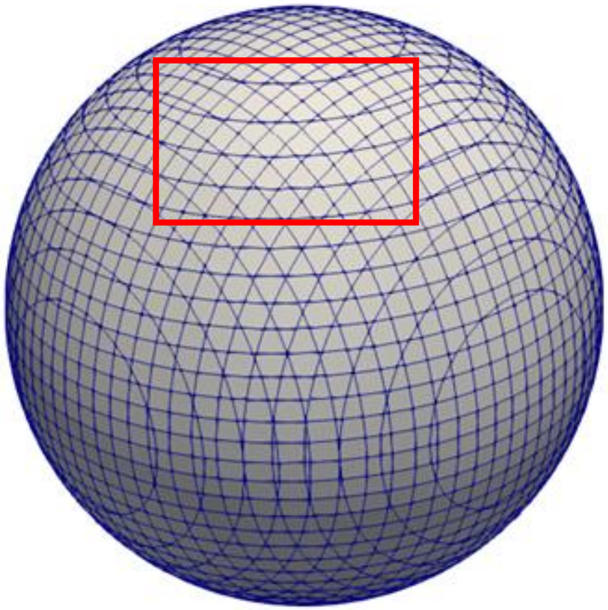


<Cut-cell method with quality enhancement>

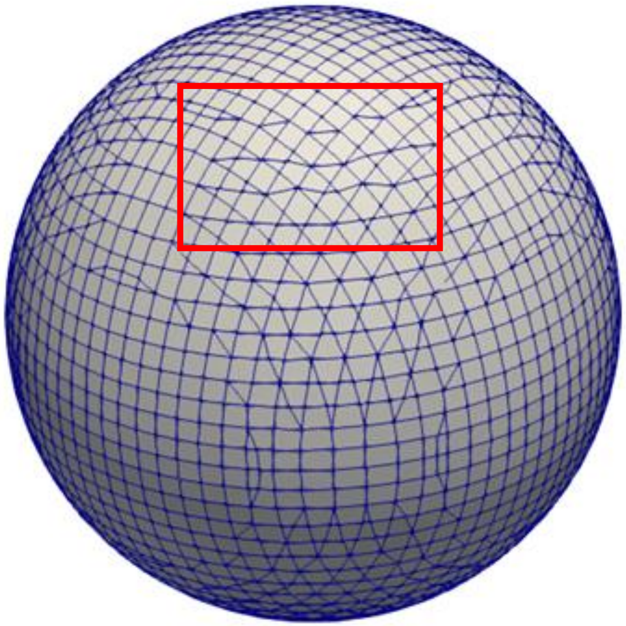
Mesh Generation Algorithm

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 to PWRs

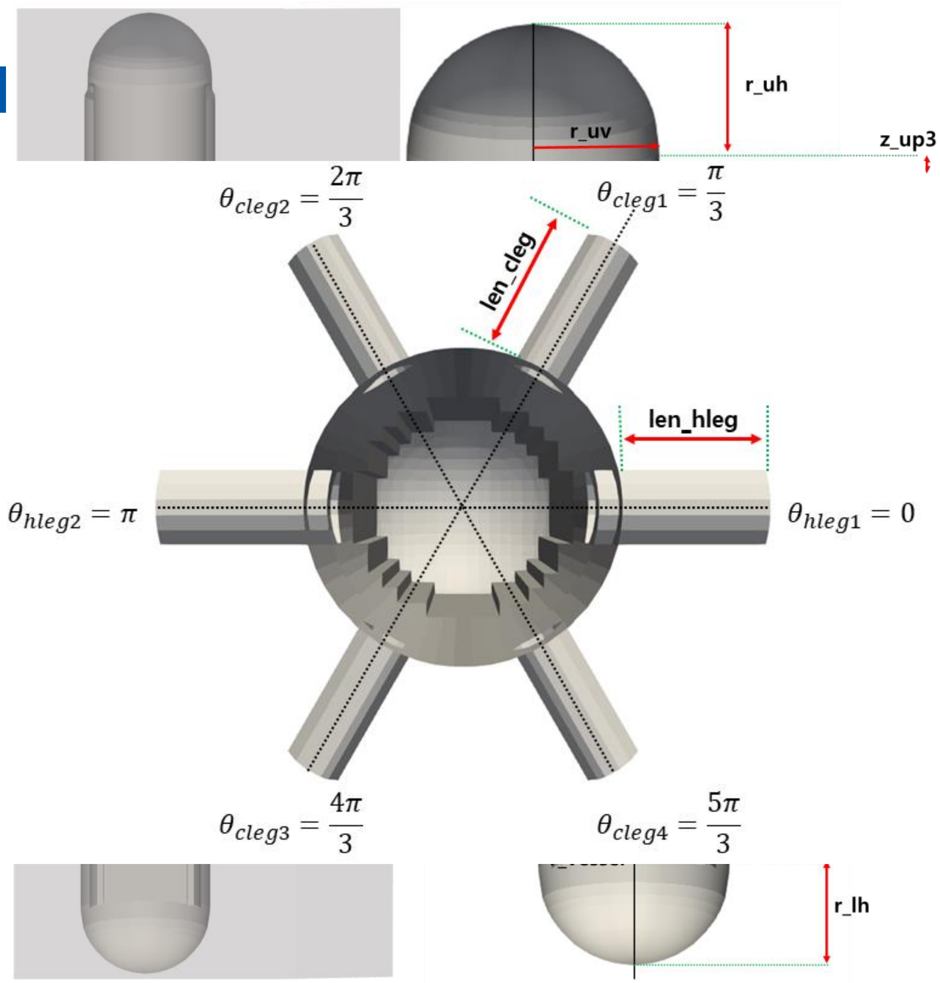
Mesh Generation Procedure

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

Applications to PWRs

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)



Applications to PWRs

Mesh generation of OPR1000/APR1400

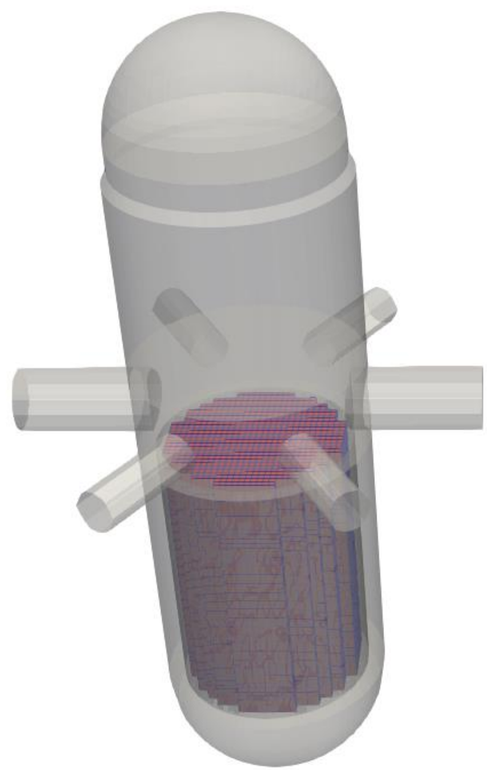
Main geometrical differences

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

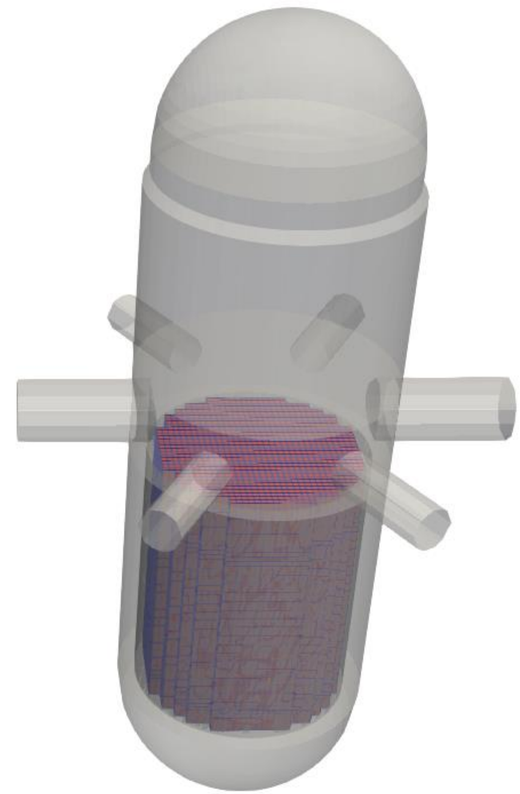
```
!------------------------MASTER nz = 28----------------------------!
&PLANE_2D
  r_core = 1.7526d0 !3.505200d0*0.5d0
  df0 = 0.208740d0
  witdh_dc = 0.3048d0      !...r_vessel=r_core+witdh_dc
  rhlg = 0.6d0  !...'rhlg' is a just some sufficiently large value rather than 'real
  n_layer_dc = 3
  nx_assem = 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>

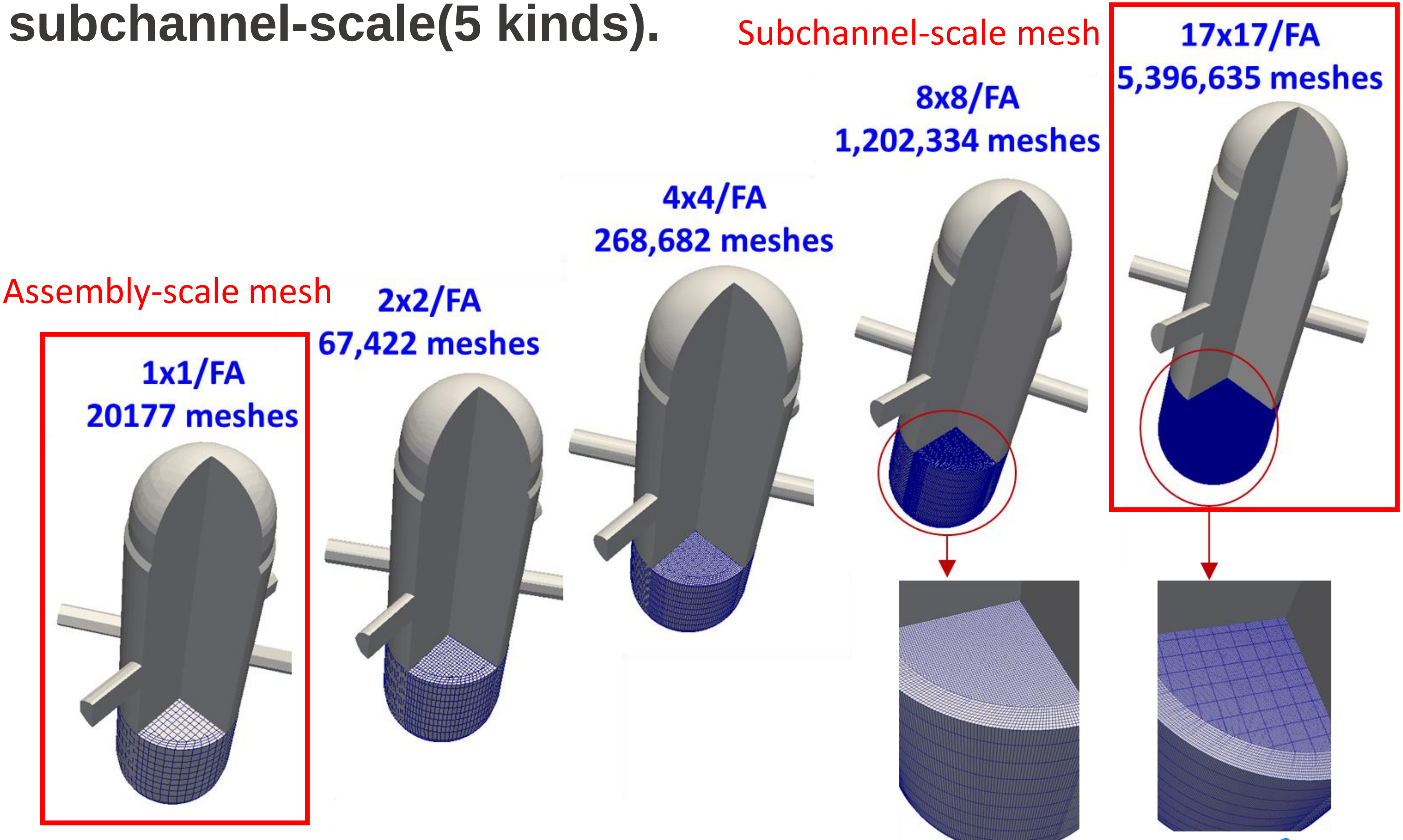


<APR1400>



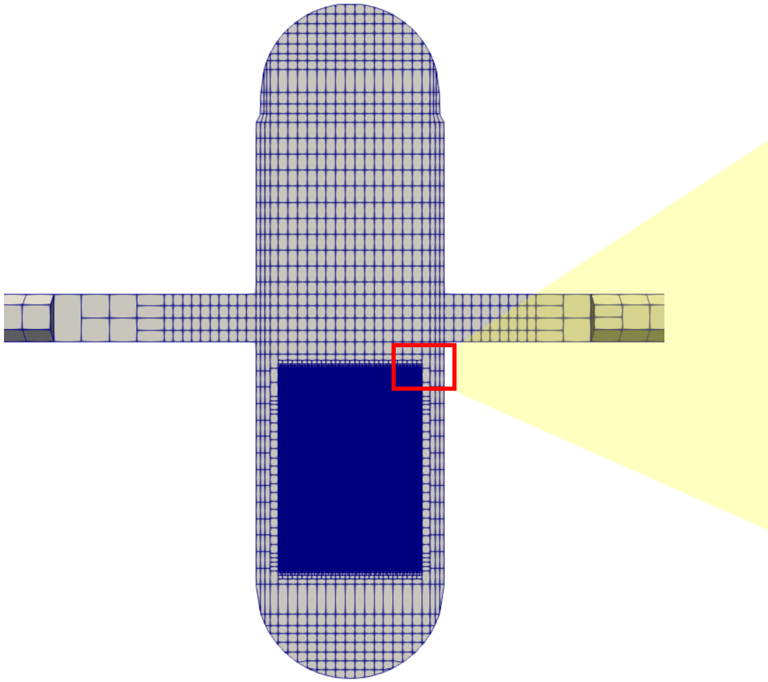
Applications to PWRs

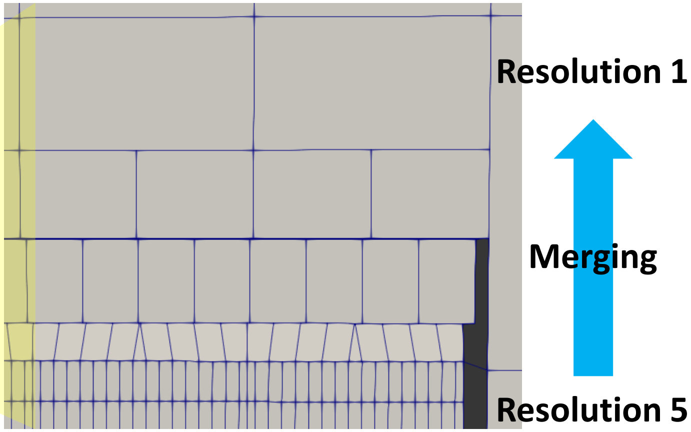
- Mesh resolution can be controlled from assembly-scale to subchannel-scale(5 kinds).



Applications to PWRs

- 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%**.

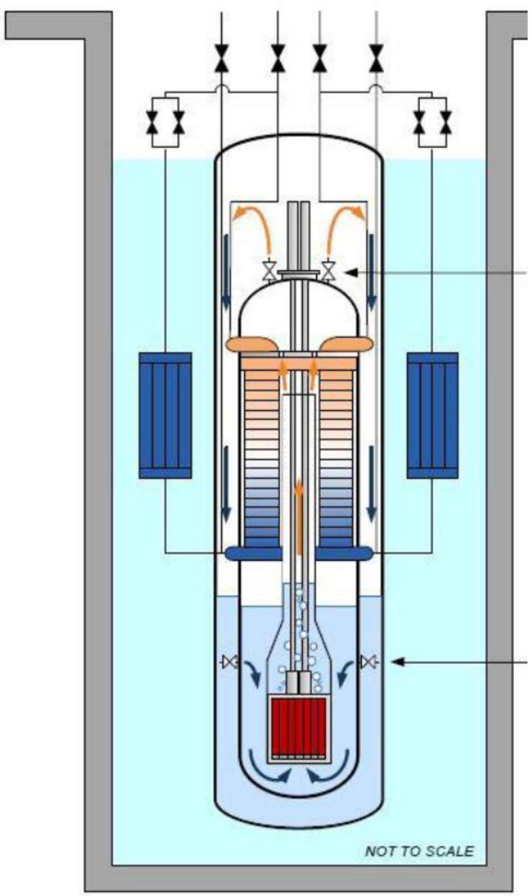




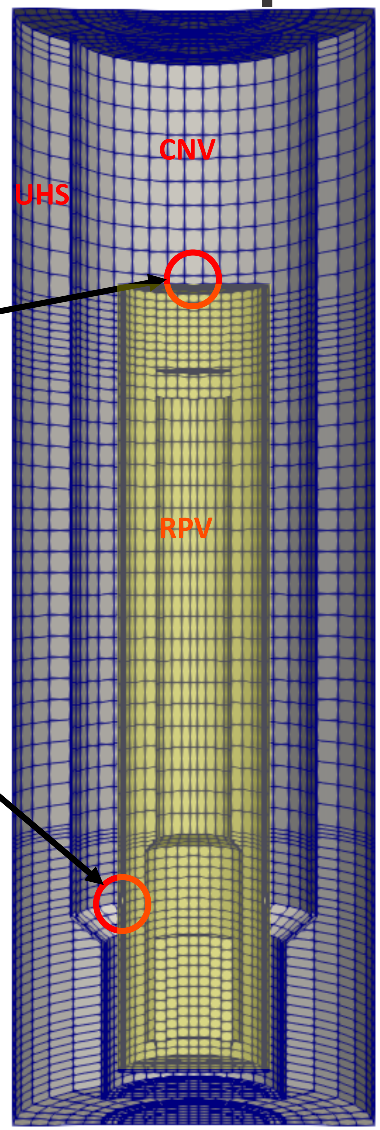
Case	Num. cells
1x1/FA	20,177
2x2/FA	67,422
4x4/FA	268,682
8x8/FA	1,202,334
17x17/FA	5,396,635
Adaptive	1,390,578

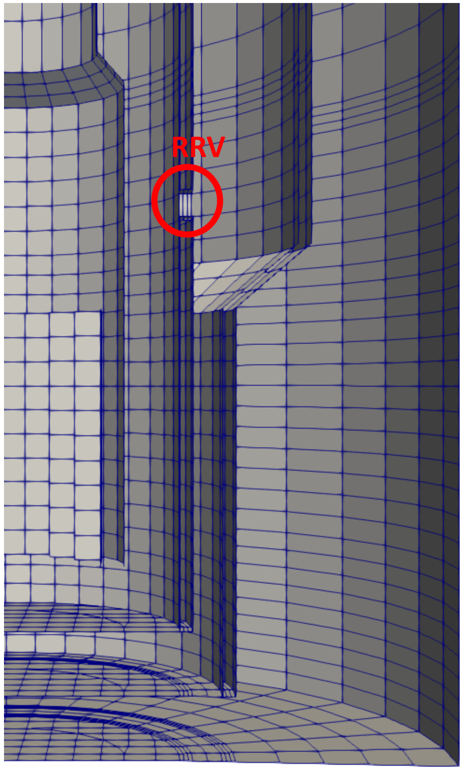
Applications to SMRs - NuScale

» Schematic diagram



» Computational mesh



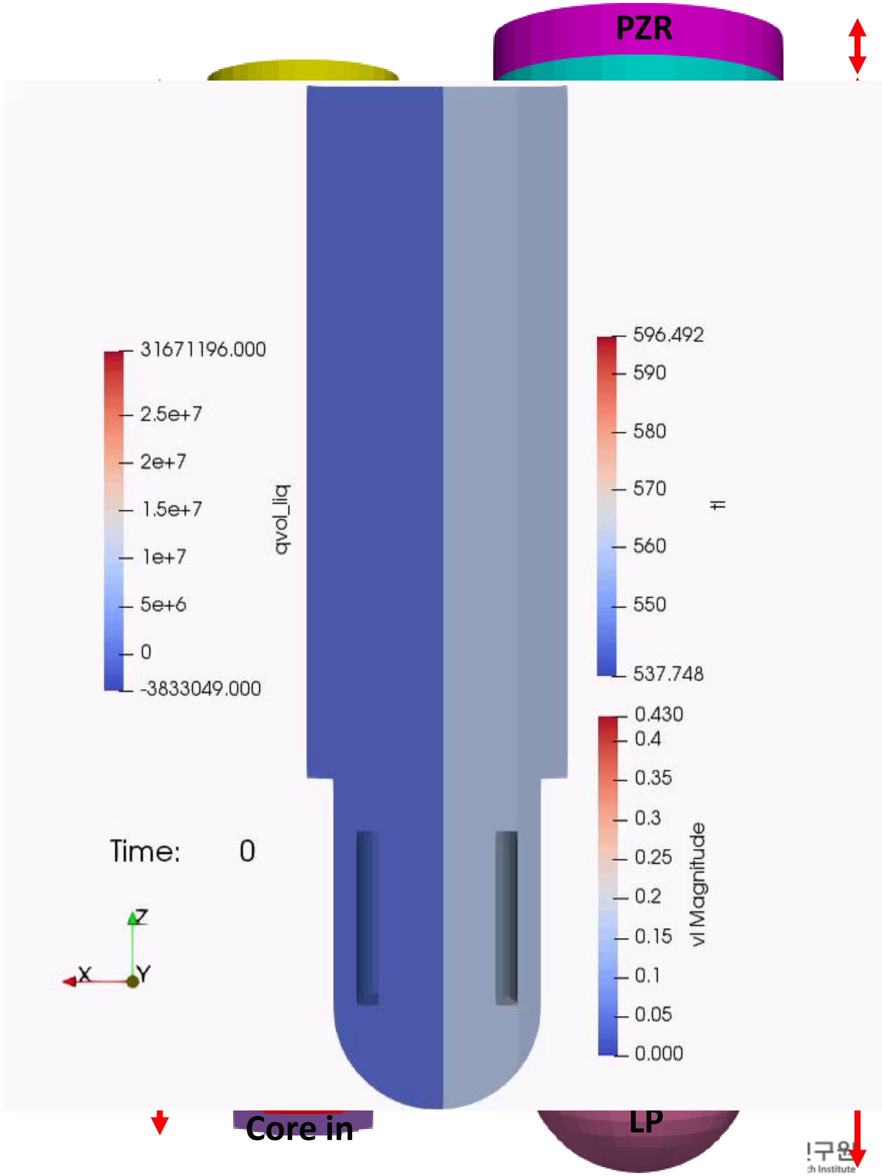


- RVV/RRV flow path is considered to simulate LOCA.
- LOCA simulation is currently in progress.

Applications to SMRs - iSMR

- iSMR design is in progress in Korea.

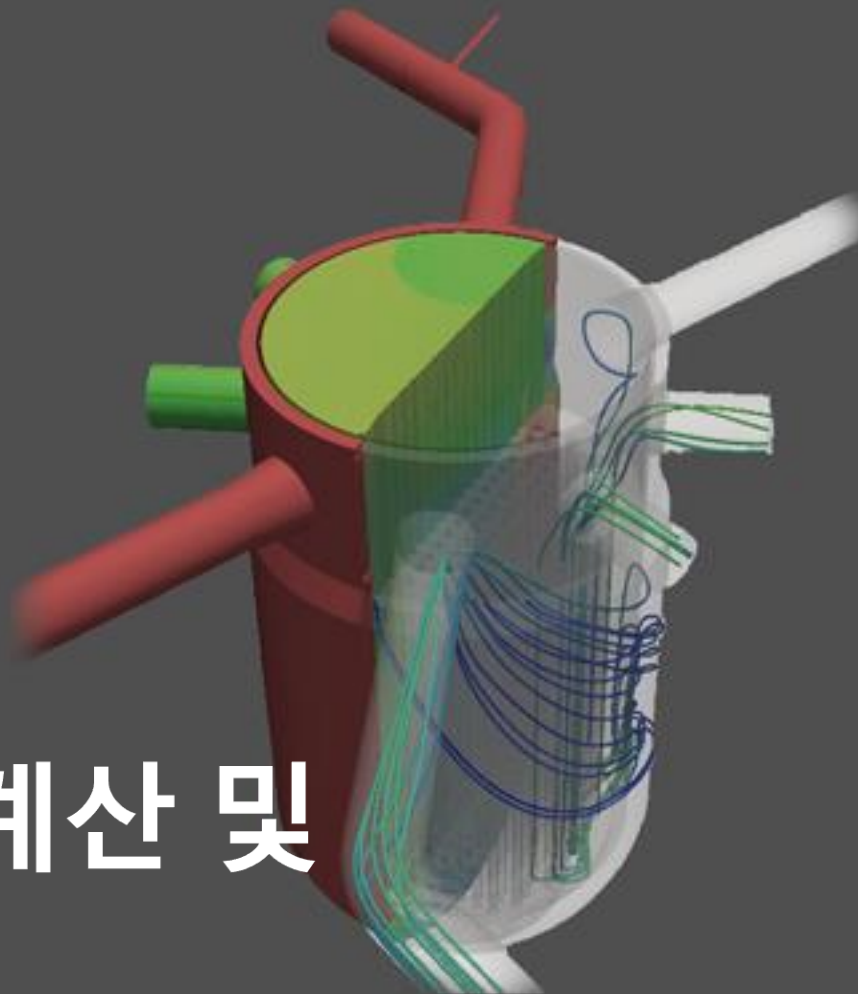
- CUPID and RVMesh3D are used to simulate
  - Natural circulation
  - Steady-state operation with MCP
  - MCP Coastdown



THANK YOU

sjdo@kaeri.re.kr

Korea Atomic Energy Research Institute
KAERI
한국원자력연구원



10th CUPID Workshop

부수로 모델 검증 계산 및 후처리 기법

박 소 현

2022년 8월 23일

CONTENTS

부수로 모델 검증 계산

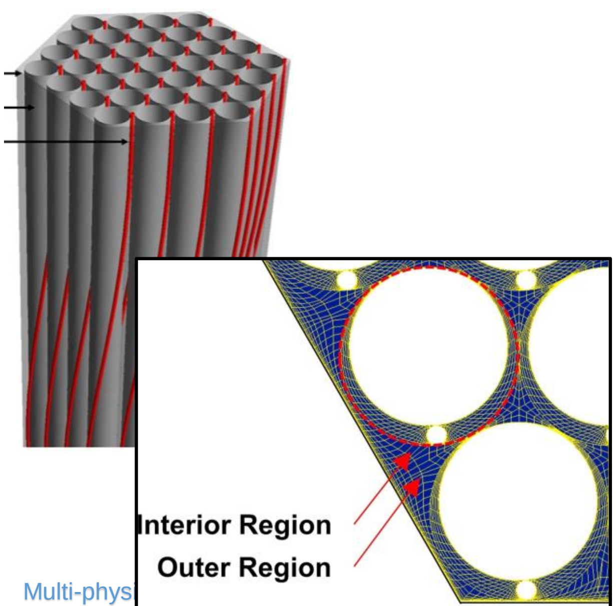
1

- 부수로 모델
- 부수로 모델 검증
- iSMR 적용 계산

Introduction: 부수로 모델

CFD Scale

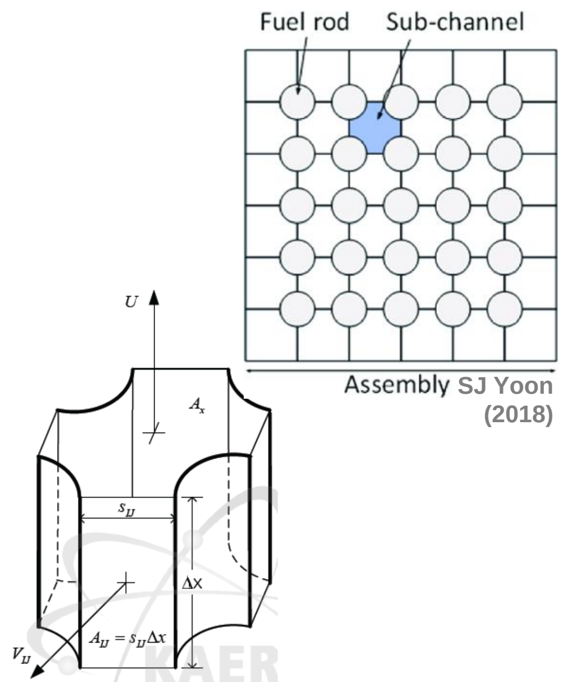
- 정교한 격자
- 다차원 해석
- 복잡, 미세 난류거동
- 단상 유동
- 제한적인 영역
- 높은 계산성능 요구



MS Song (2019)

Sub-channel Scale

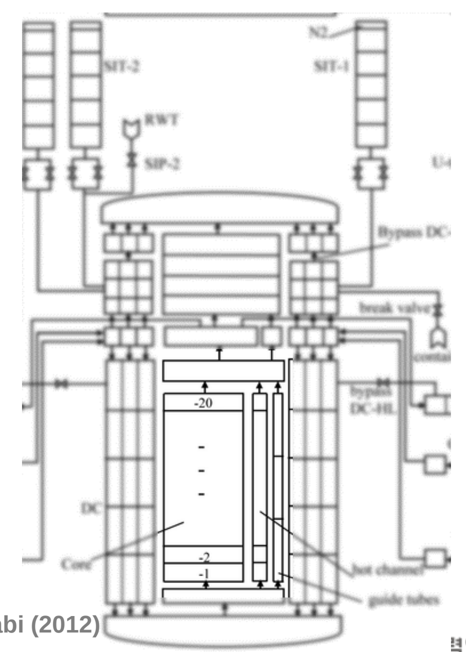
- 다공성 매질 격자
- 다차원 해석
- 상세,국부 거동
- 이상 유동
- 최적안전해석론



SJ Yoon (2018)

System Scale

- Lumped parameter
- 일차원 해석
- 전반적인 거동
- 이상 유동
- 보수적인 방법론



Sharabi (2012)

Introduction: 부수로 모델

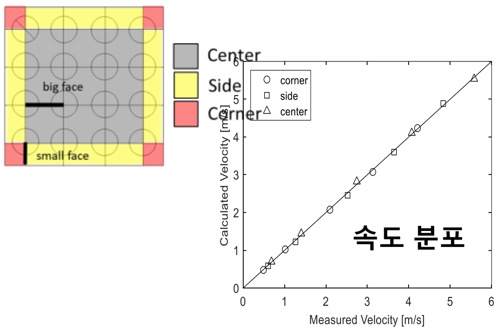
부수로 해석 모델 (Empirical correlation)

- RV model
  - Heat structure
  - Wall flow regime map
  - Interfacial area transport
  - Interfacial drag
  - Interfacial heat transfer
- **Sub-channel model**
  - **Friction loss** : 축방향 압력강하
  - **Form loss** : 횡방향 압력강하
  - **Turbulent mixing and void drift** : 횡방향 단상/이상 난류혼합
  - **Spacer grid**
  - **Mixing vane**

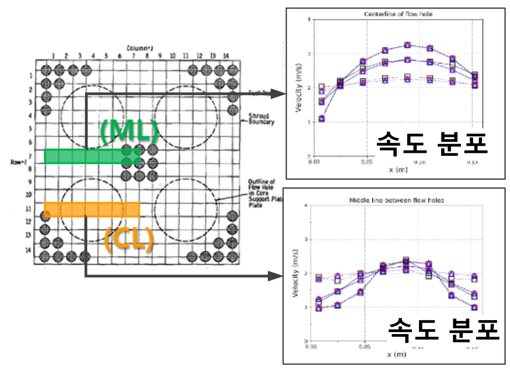
부수로 모델 검증: 단상 유동

단상유동 부수로 실험 검증

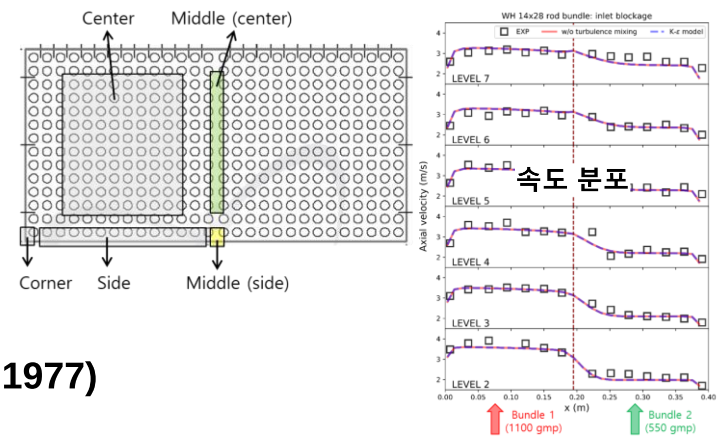
1. CNEN 4x4 (SW,1974)
- 단열 유동
- 압력강하 검증
- 난류혼합 검증



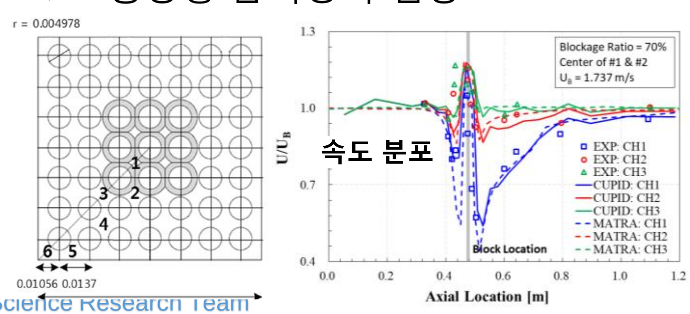
2. CE 15x15 (US,1969)
- 단열 유동
- 비균일 입구 유속
- 횡방향 난류혼합 검증



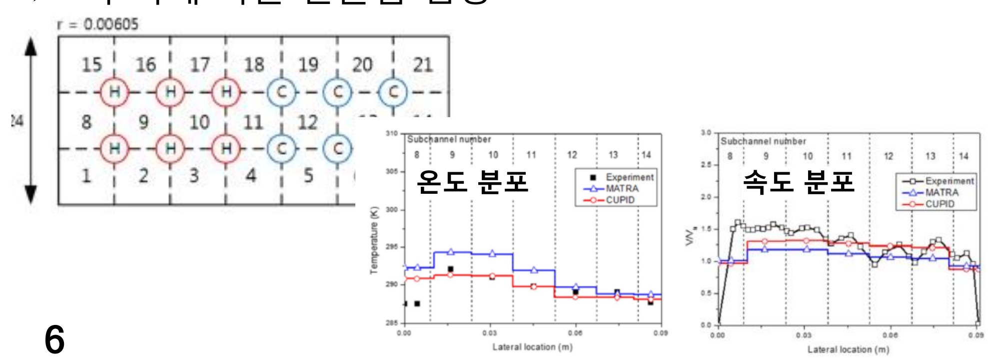
3. WH 14x14 (US,1968)
- 단열 유동
- 부수로 부분 막힘
- 횡방향 난류혼합 검증
- 횡방향 압력강하 검증



4. PNL 7x7 (US,1977)
- 단열 유동
- 부수로 부분 막힘
- 횡방향 압력강하 검증



5. PNNL 2x6 (US,1977)
- 가열 유동
- 부력에 의한 열혼합 검증

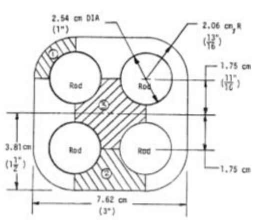


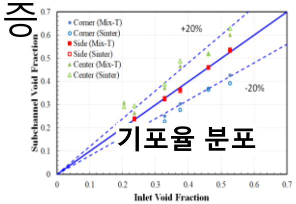
부수로 모델 검증: 이상 유동

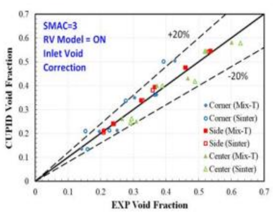
» 이상유동 부수로 실험 검증

1. RPI 2x2 (US,1983)

- ➢ 단열 이상유동
- ➢ 유동맵 검증
- ➢ 난류혼합 검증

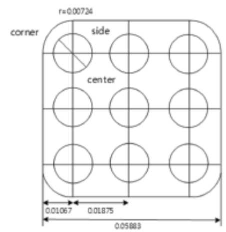


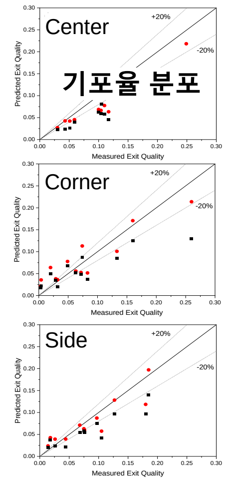




2. GE 3x3 (Canada,1988)

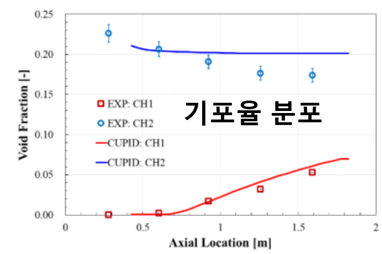
- ➢ 가열 유동
- ➢ 유동맵 검증
- ➢ 난류혼합 검증





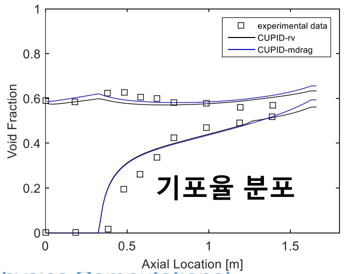
3. Van der Ros (Canada,1988)

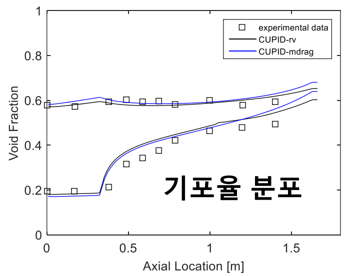
- ➢ 단열 이상유동
- ➢ 비균일 입구조건
- ➢ 유동맵 (BBY), 난류혼합 검증



4. Tapucu (Canada,1988)

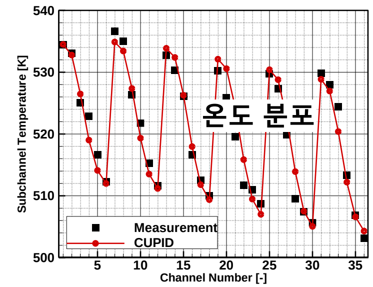
- ➢ 단열 이상유동
- ➢ 유동맵 (ANN), 난류혼합 검증

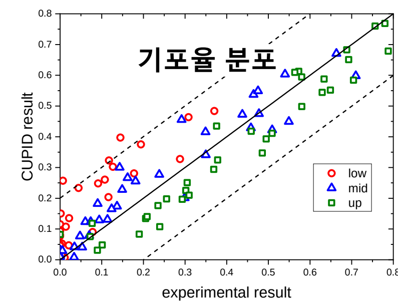




5. PSBT 봉다발 (OECD,2006)

- ➢ 가열 비등 유동
- ➢ Mixing vane 난류혼합 검증





iSMR 적용 계산

부수로 스케일 iSMR 자연대류 거동 해석

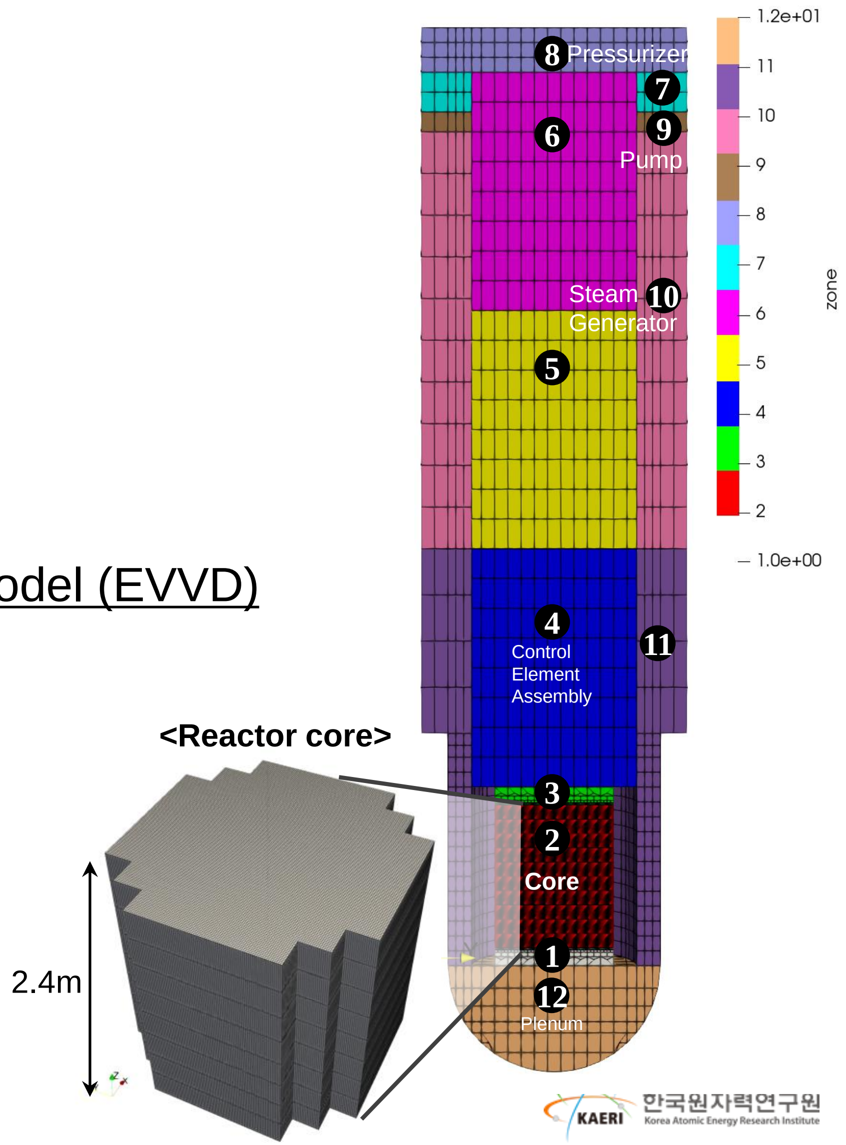
RVmesh 격자 생성: **265,005 cells**

해석 모델

- RV heat structure model
- RV single-phase model
- Sub-channel friction: MATRA model
- Sub-channel form loss: Simple model
- Equal-volume exchange and Void-drift model (EVVD)

해석 조건

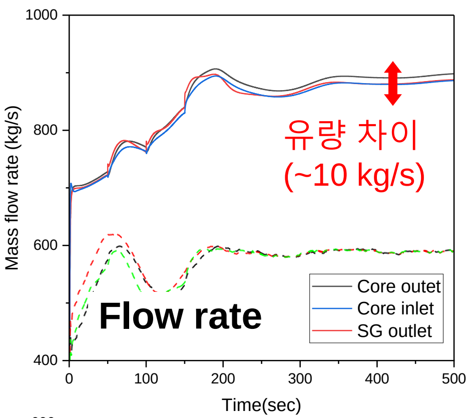
- Full core power (steady-state)
- **Long transient** simulation (~1000 sec)
- No pump, **natural circulation**
- **Buoyancy-driven** flow

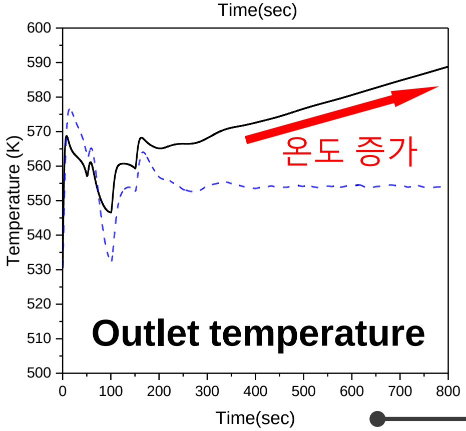


iSMR 적용 계산

정상상태 노심 온도 및 유량 수렴 문제 해결

- 문제: Core inlet-outlet 유량 불일치, 지속적인 노심 온도 증가
- 원인: SMAC3 알고리즘 EVVD 모델 업데이트 오류



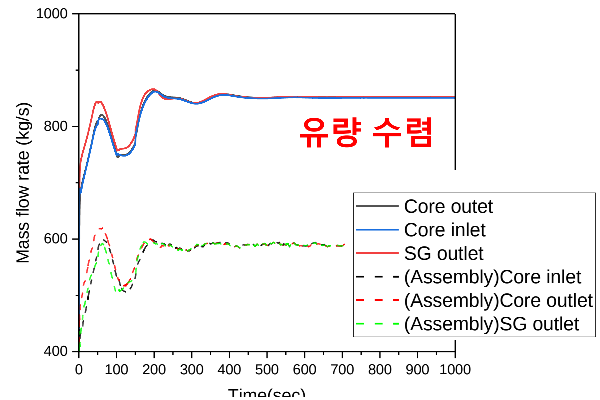


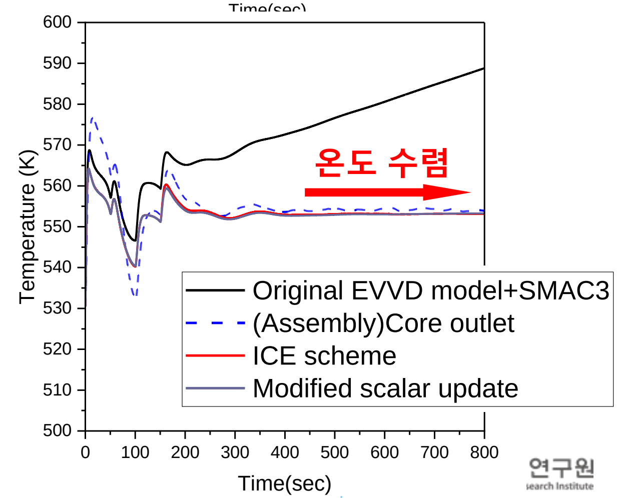
ICE: Energy-coupled scheme

- No problem

SMAC3: Energy-decoupled scheme

- Momentum update
- Continuity update
- Energy update
  - EVVD energy source term
- Pressure update





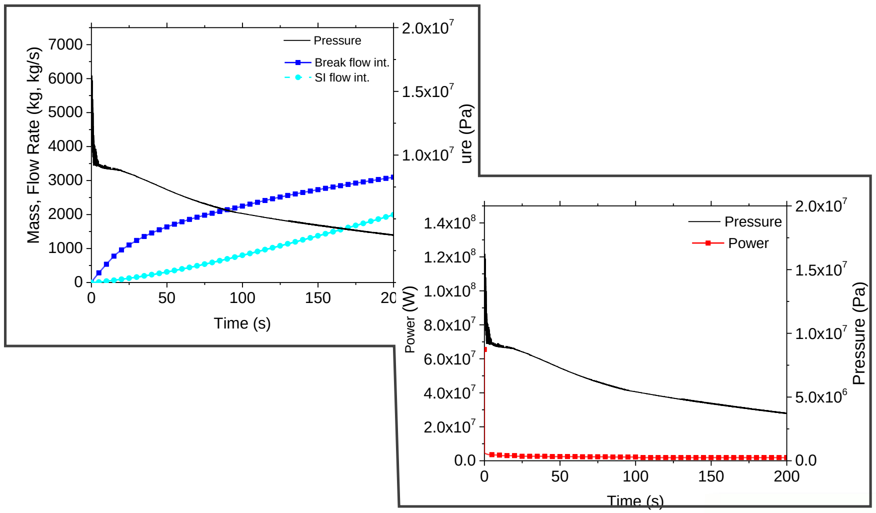
후처리 기법

2

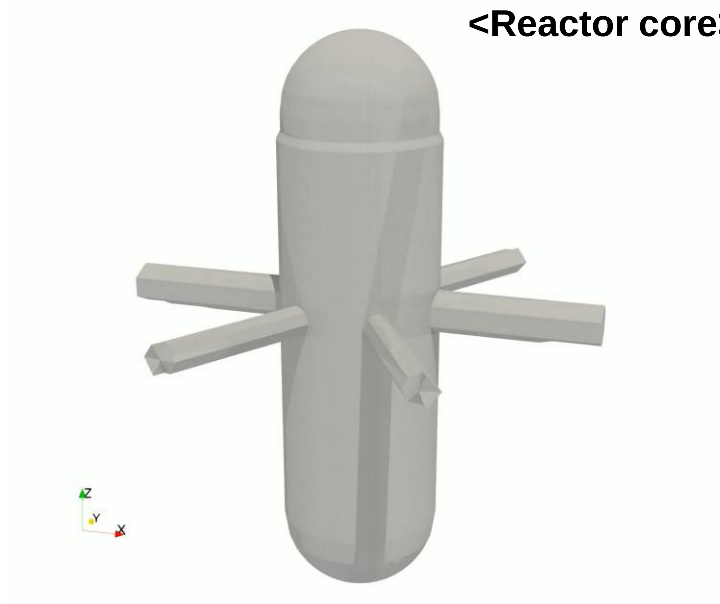
– Paraview

Paraview 후처리 기법

3차원 Rendering 기반 후처리 기법



<Reactor core>



<CUPID-SG>

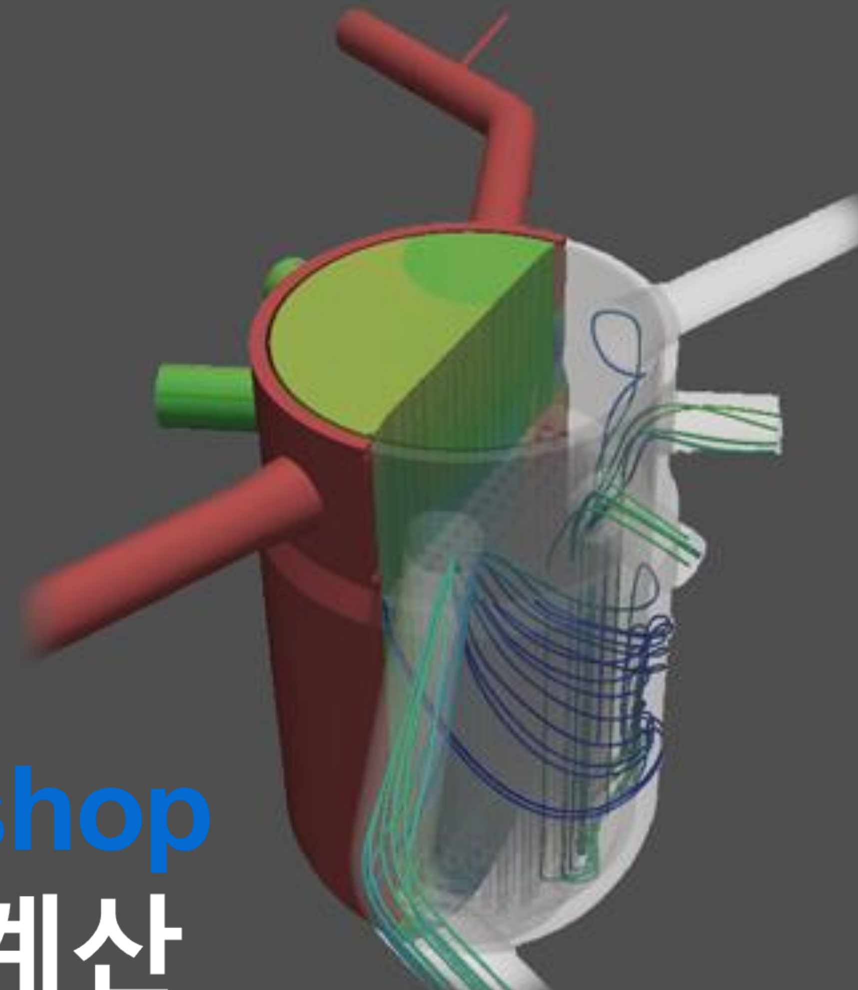
Paraview

- ➢ 오픈소스 프로그램
- ➢ 윈도우/리눅스
- ➢ MPI 지원
- ➢ .vtk, .foam, .pvtu, etc

1. CUPID-SG 후처리
   - Contour
   - Ray traced rendering
   - Make .avi movie
2. CUPID Core 후처리
   - Warp by scalar

THANK YOU

bibirom10@kaeri.re.kr



10th CUPID Workshop

CFD 스케일 검증 계산

조 윤 제

2022년 8월 23일

10th CUPIDERS

CONTENTS

International CFD Benchmark

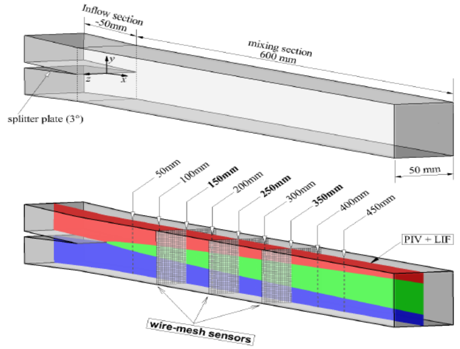
1

- CFD Applications using CUPID
- OECD/NEA Benchmark
- IAEA CRP Benchmark

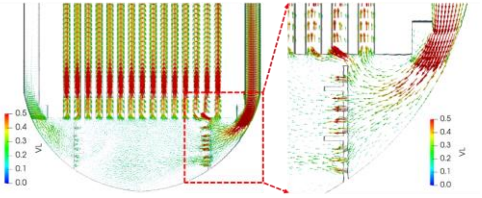
Introduction

CFD-Scale Applications using CUPID

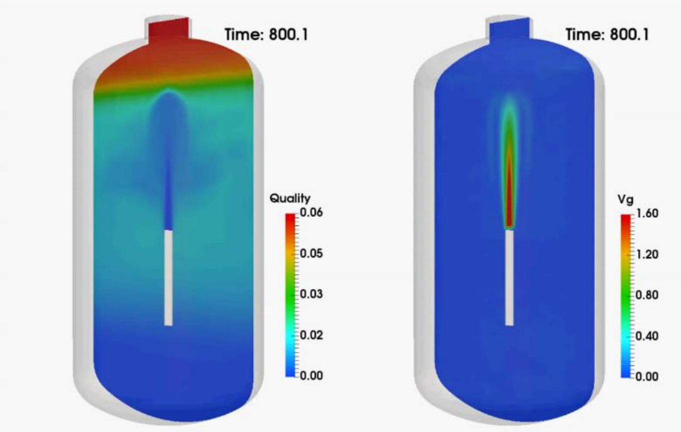
CFD Scale: Downcomer, Lower Plenum, Etc.



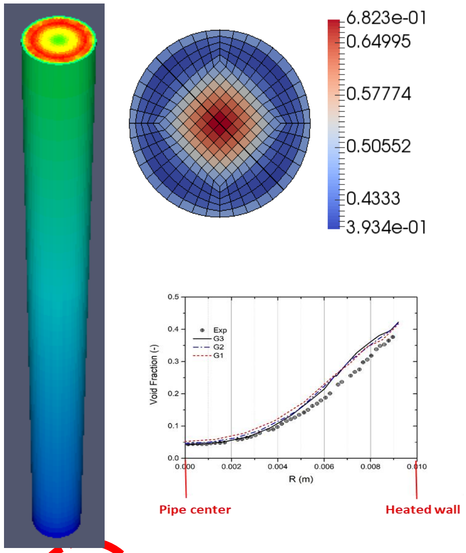
<OECD/NEA IBE: GEMIX>



<IAEA CRP: ROCOM_12>



< OECD/NEA HYMERES-2>



<DEBORA Benchmark>

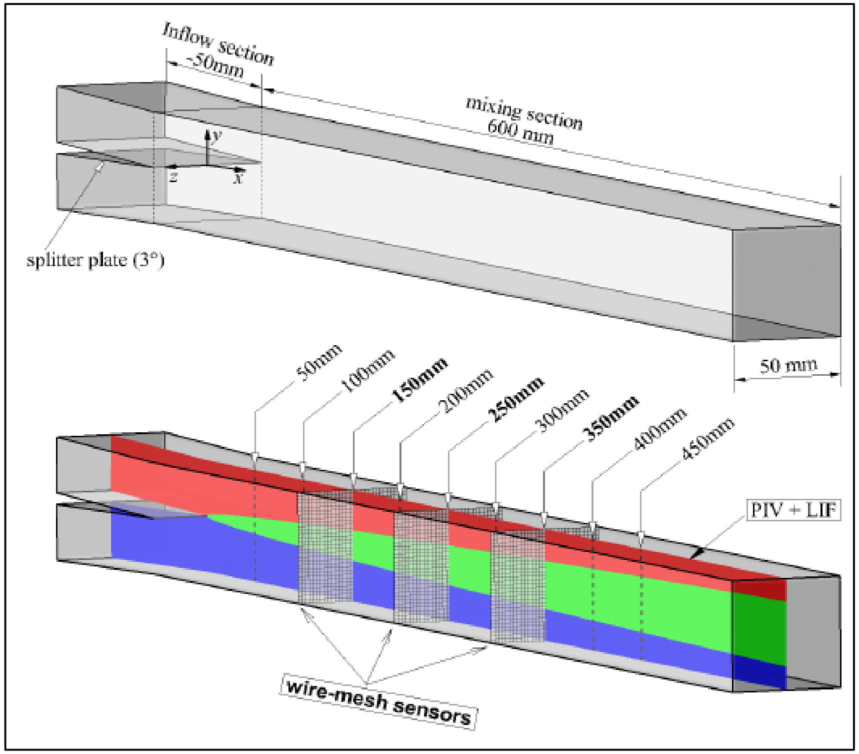
OECD/NEA IBE-4 (GEMIX)

2

- Description of ROCOM
- Boundary Condition
- Model & Numerical Setup
- Turbulence Model in CUPID

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	

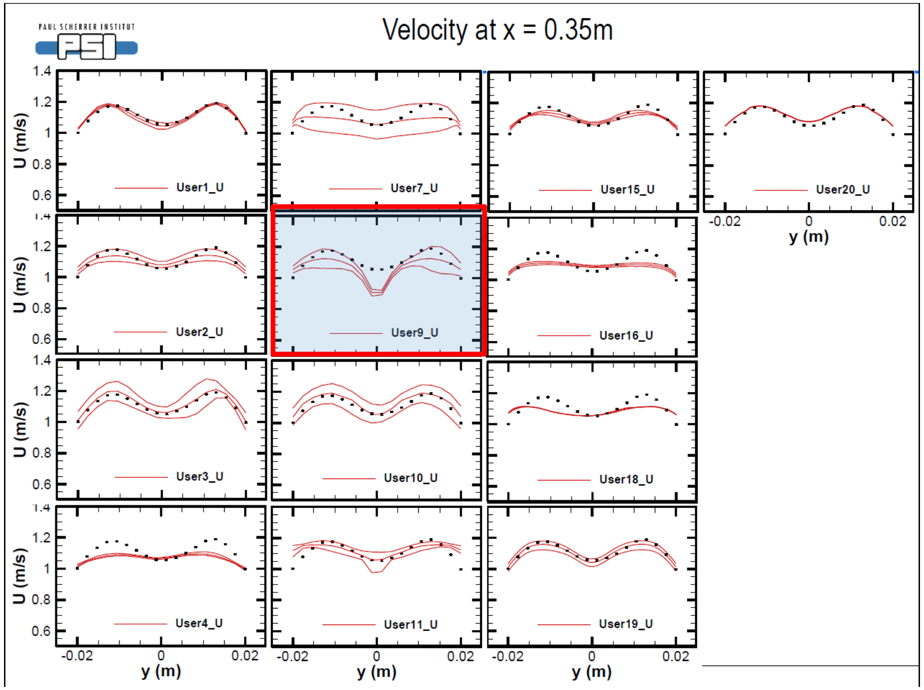
Number of grid

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

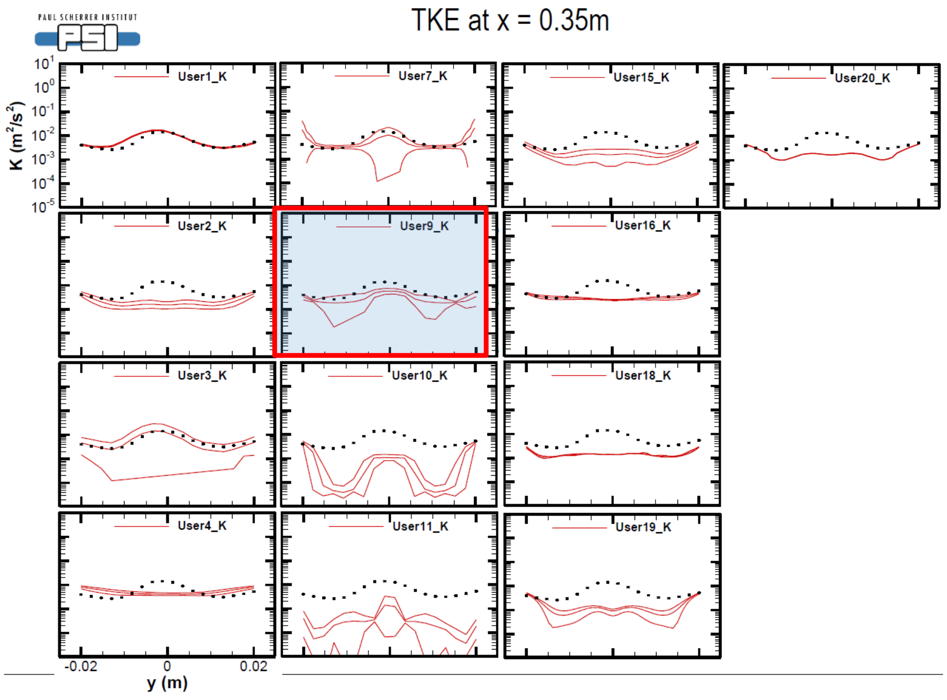
OECD/NEA Benchmark (2/3)

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)

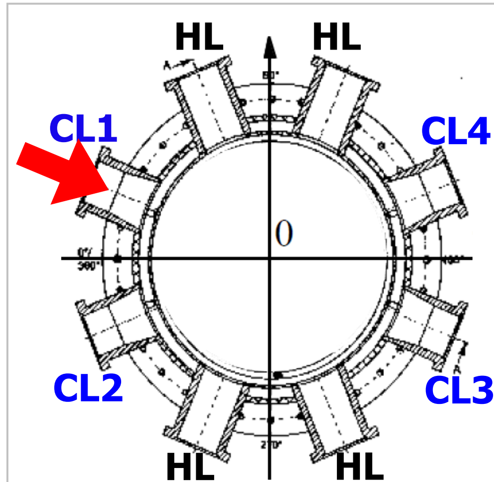
Slug Mixing Experiments

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

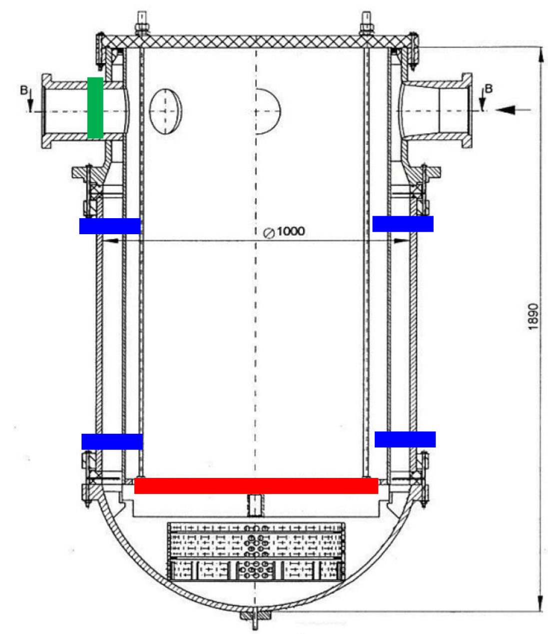
Ramp length	Volumetric flow rate	Slug volume
14 s	185.0 m^3/h	8.0 m^3

- 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

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

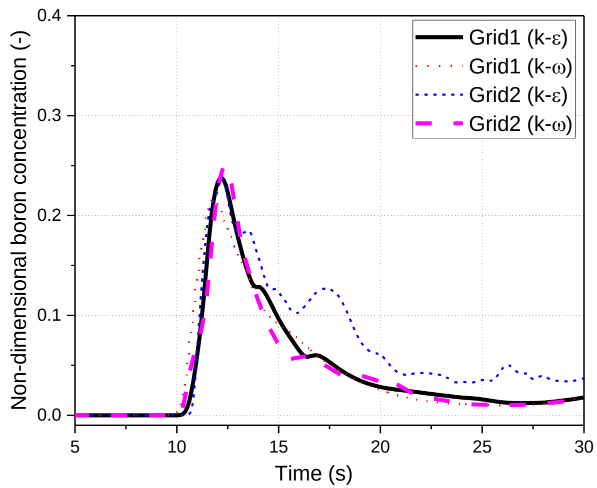
Baseline Calculation Case

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

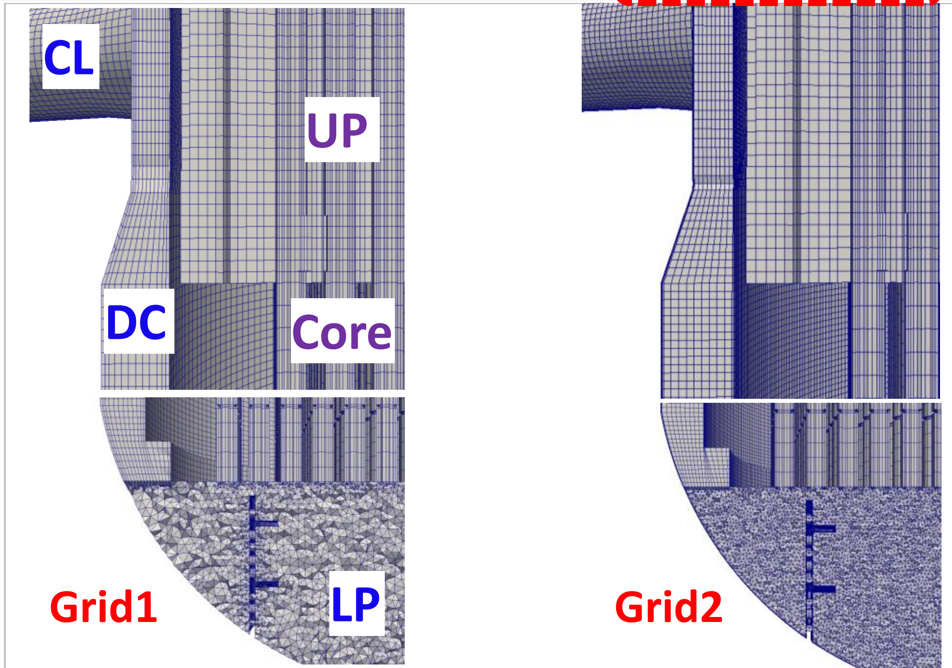
Mesh Sensitivity Test

Additional Grids for Sensitivity Test

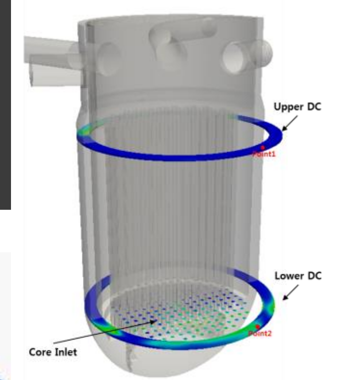
	Grid	Core+UP	CL+DC+LP	Y^+
	Coarse	2.2M (Fixed)	1.15M	$Y^+>300$
Grid1	Reference		2.45M	$30<Y^+<300$
Grid2	Refined		10.21M	$Y^+<5$

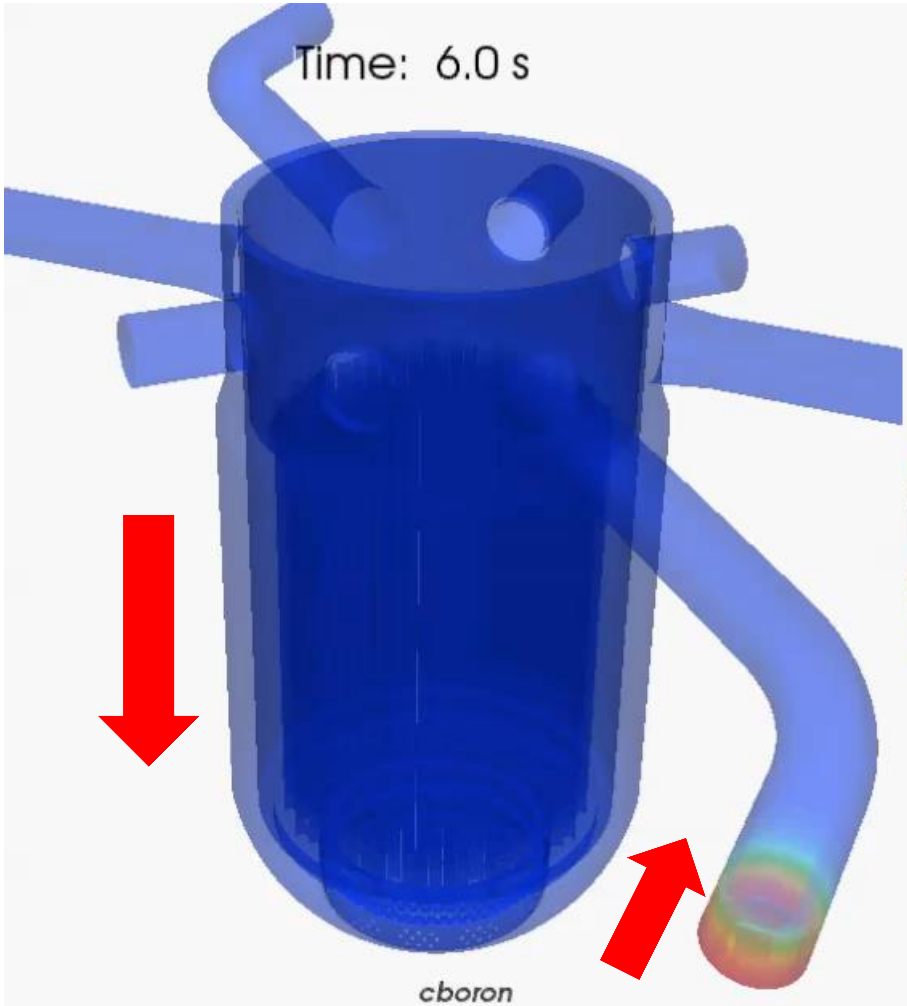


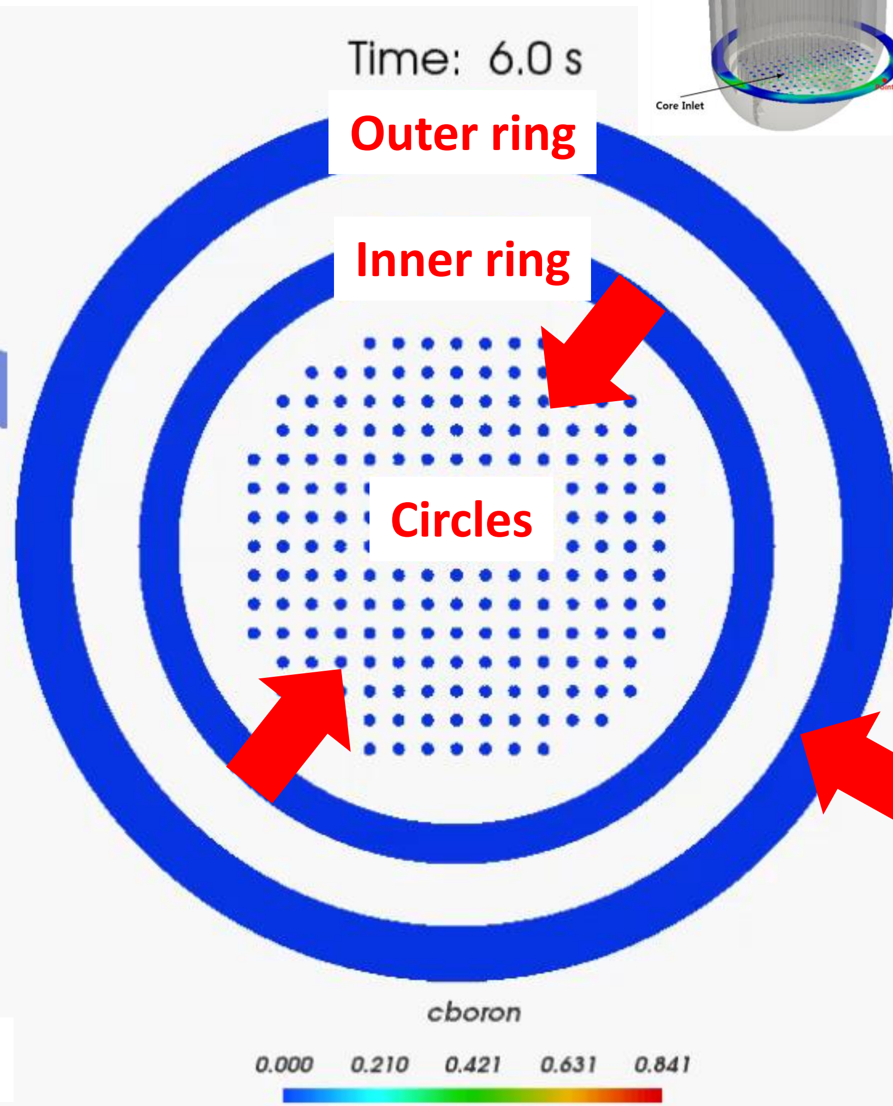
Convergence Test



Overall Mixing Behavior

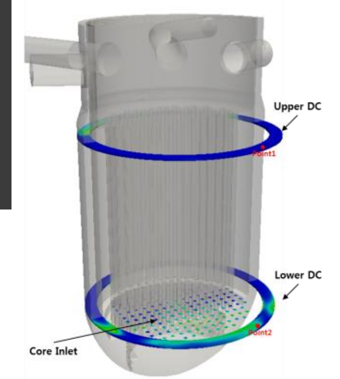




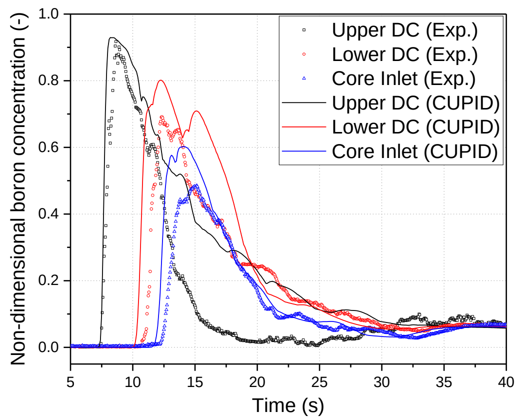


Non-dimensional boron concentration

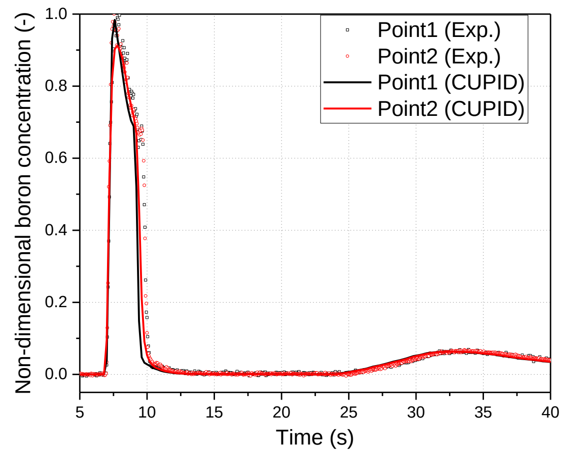
Quantitative Comparisons



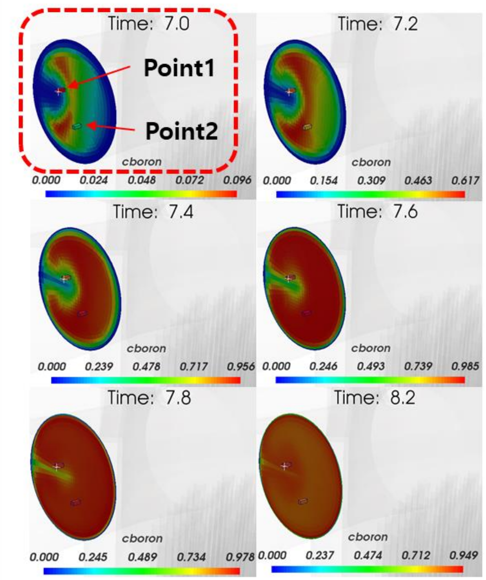
Averaged & Local concentration

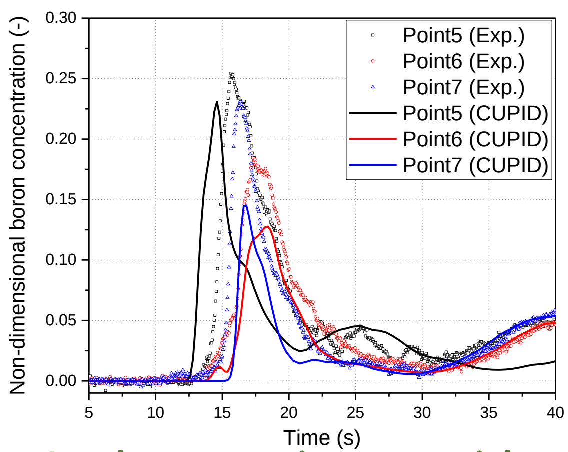


Maximum concentration at upper DC, Lower DC, Core inlet

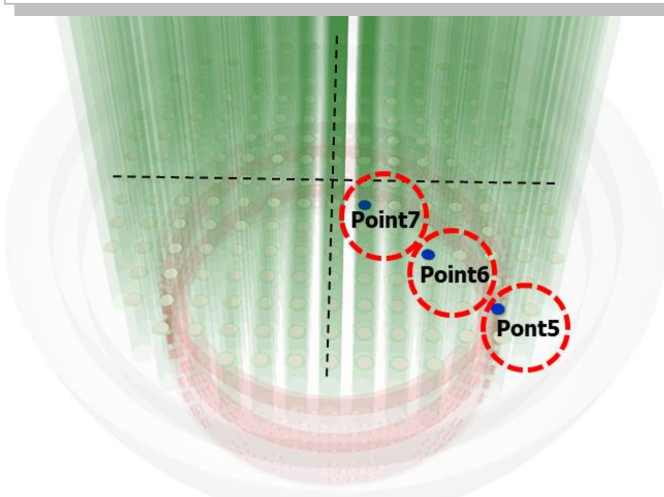


Local concentration at cold leg





Local concentration at core inlet



Synthesis Report (2/2)

IAEA CRP: Boron dilution benchmark

- The first place in three of ranking system

Participant	Code
HZDR	CFX 18
VNIIAES	Star-CCM+
BARC	OpenFOAM
KAERI	CUPID 2.0

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)

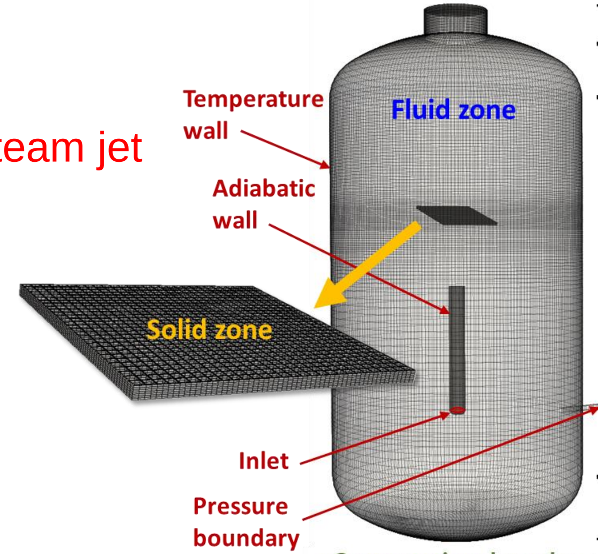
OECD/NEA HYMERES-2

4

Overview of HYMERES-2

HYMERES-2

- **Main objective**
  - To improve the understanding of the containment phenomenology during postulated severe accident with release and distribution of hydrogen
- **HYMERES-2 : 2017.01 – 2021.06**
- **Experimental facility : PANDA (PSI)**
- **Main topics of HYMERES-2**
  - Jet/plume interacting with various obstruction geometries
  - Thermal radiation effects
- **Blind Benchmark (H2P1_10)**
  - 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
- **Computational mesh**
  - Hexahedron mesh
  - 2,384,568 cells



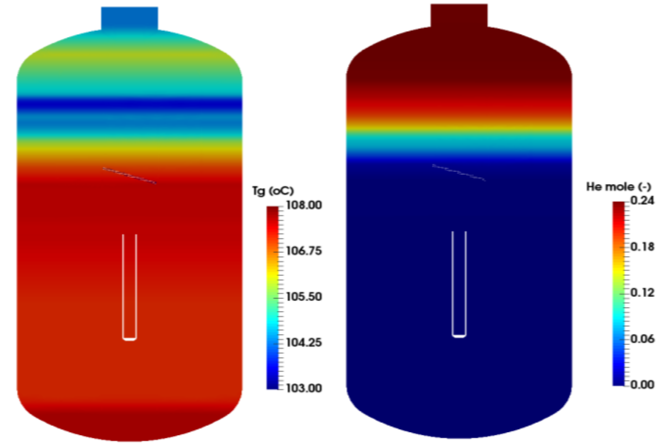
<Computational mesh>

Initial & Boundary Condition

Initial Condition

Helium concentration and gas temperature

- Applying measured experimental data along the height of the central axis at t = 0s



<Initial conditions applying experimental data>
* Left : Gas temperature, * Right : Helium mole fraction

Boundary Condition

Inlet boundary

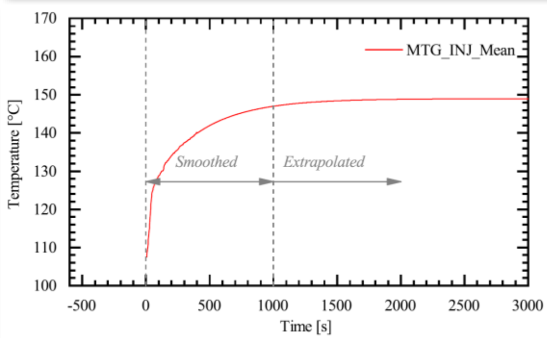
- Applying Exp. data

Pressure boundary

- 1.3bar was applied constantly.

Wall temperature

- No wall condensation
- Upper man-hole
  - ✓ Lid : constant at 101℃
- All other vessel walls
  - ✓ Constant at 108℃



<Steam injection temperature>

Physical Models

Turbulence

- **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*)

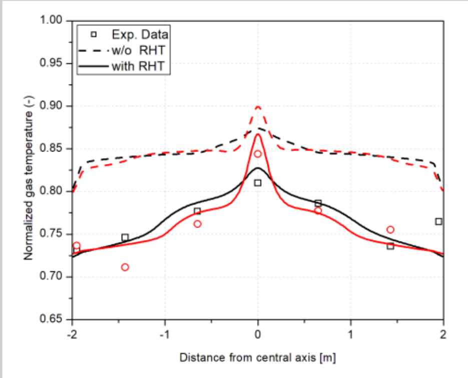
$$\nabla\cdot\left(\frac{1}{3(\kappa+\sigma_s)-A_1\sigma_s}\nabla G\right)-\kappa G+4\kappa\sigma T^4=0$$

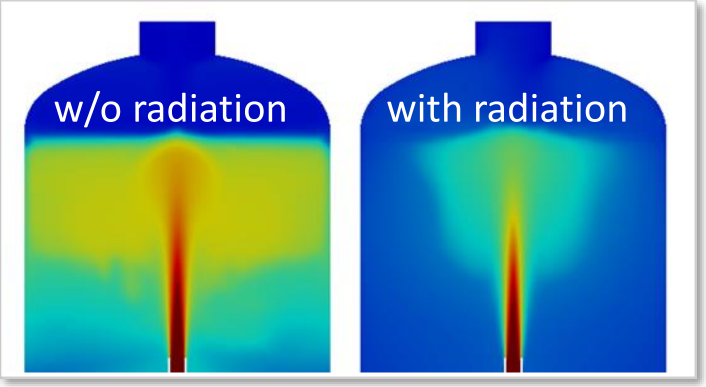
- Radiative heat flux

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

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

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





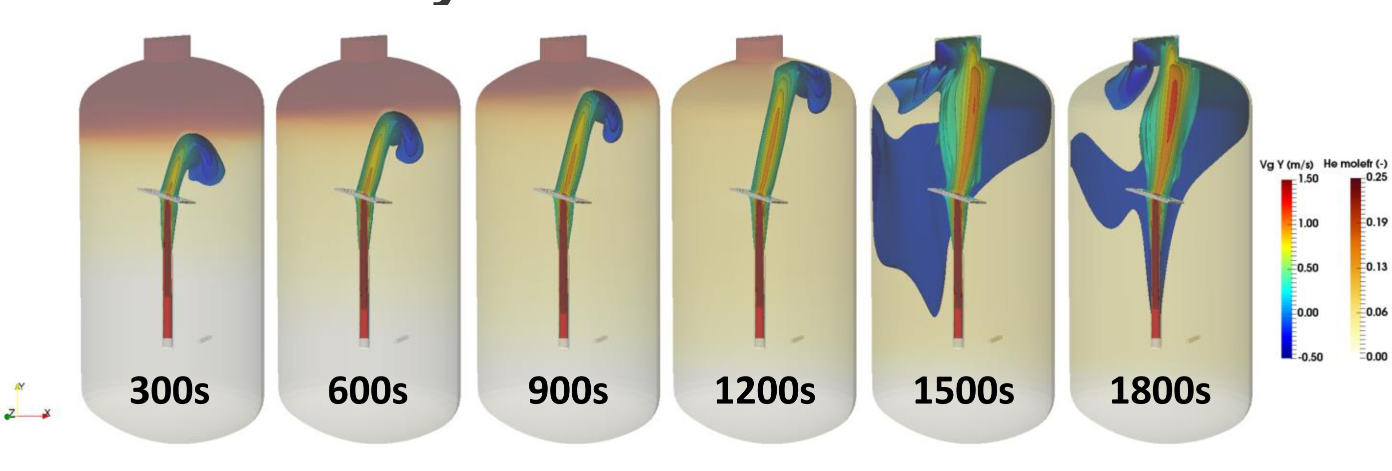
<Comparison of results with and without RHT model>

* Left : Gas temperature profiles

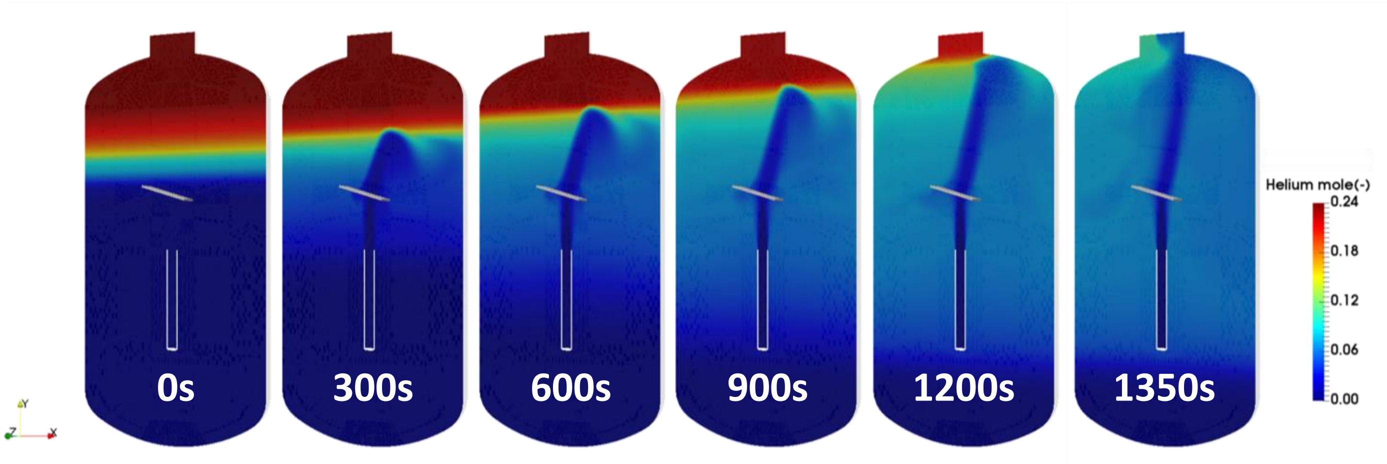
* Right : Distribution of gas temperature

Overall Behavior

Gas velocity



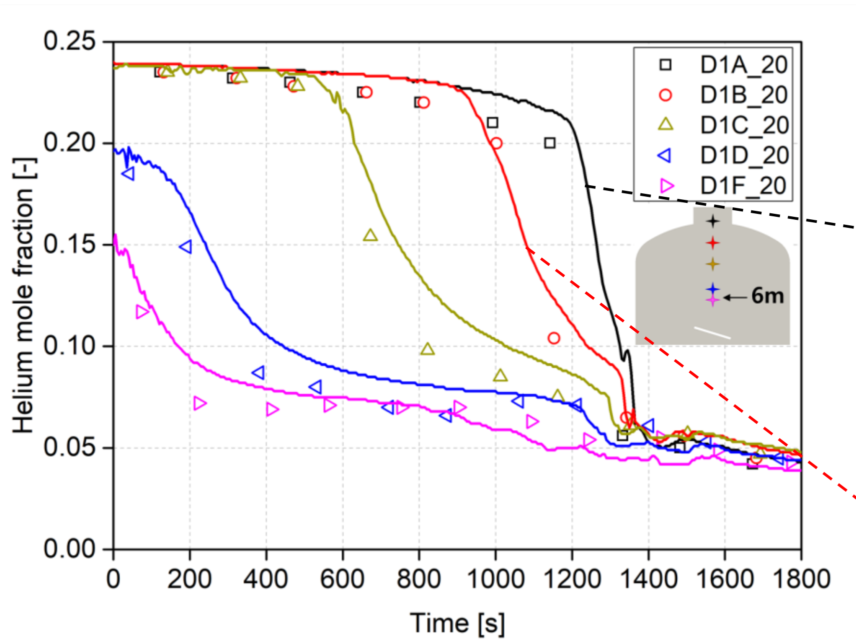
Helium concentration

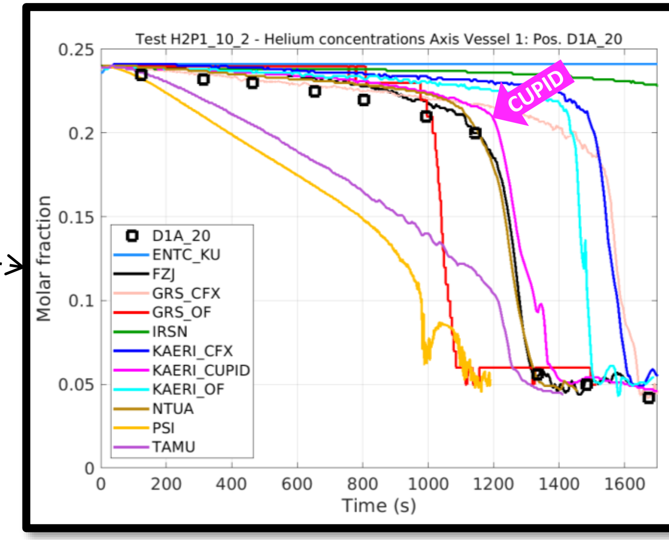


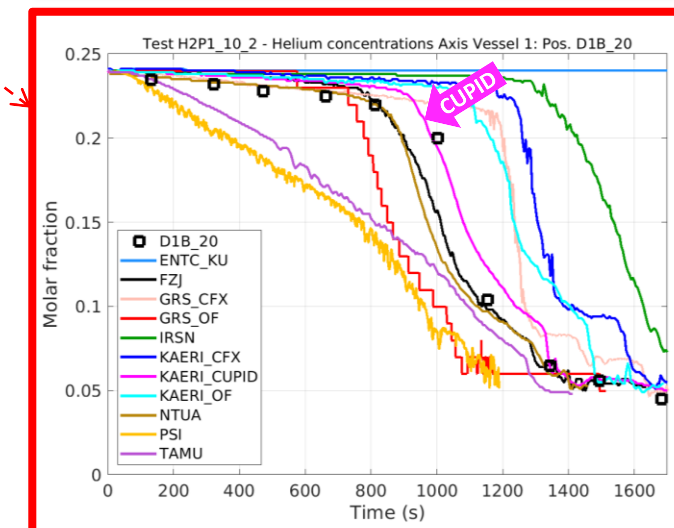
- The jet is inclined slightly to the right in the figure due to the flow obstruction.
- As the jet rises, the stratified helium is gradually eroded.
- After the jet reached the top wall of the vessel, helium was distributed almost uniformly inside the vessel.

Helium Concentration

Evolution of helium concentration over time : Central axis



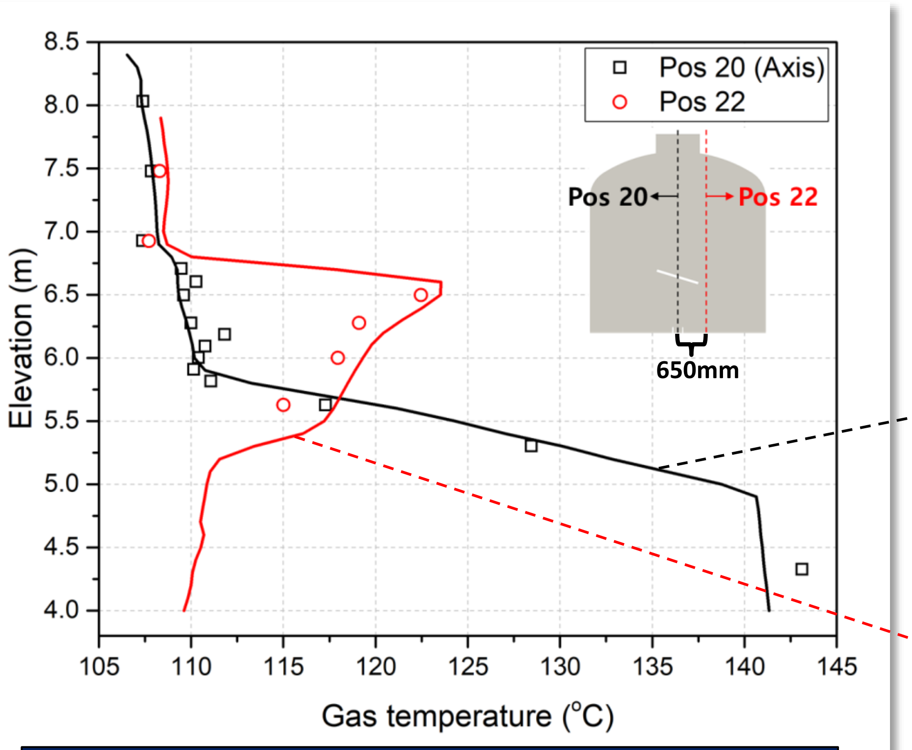




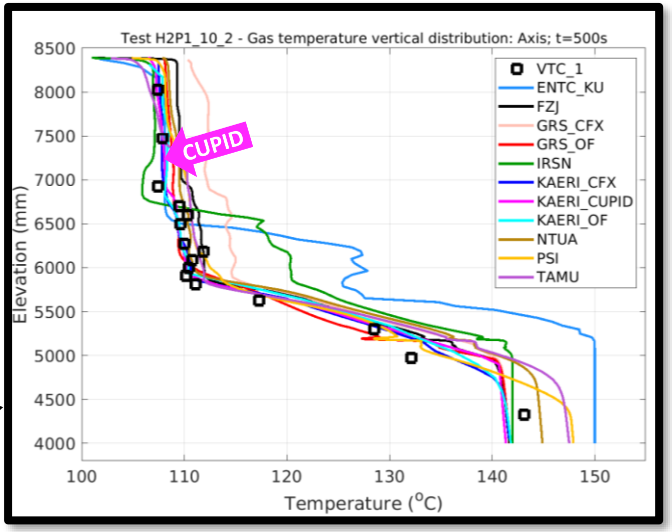
- CUPID predicted fairly well the experimental data in which the helium stratification was completely eroded after 1300 seconds.
- Overall calculation results of CUPID were excellent among the other results.

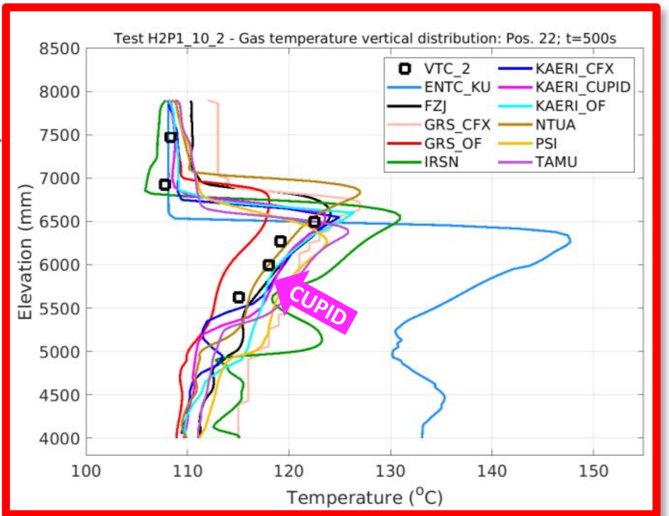
Gas Temperature

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.

DEBORA BENCHMARK (CEA)

5

- Organization and Objectives
- Description of DEBORA Experiment
- Physical Models
- Preliminary 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 more **unified method** for testing and validating CMDF 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)**

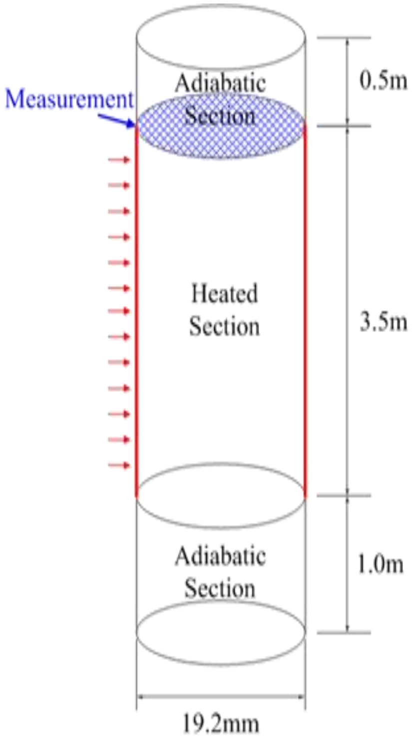
    4 additional cases in blind conditions

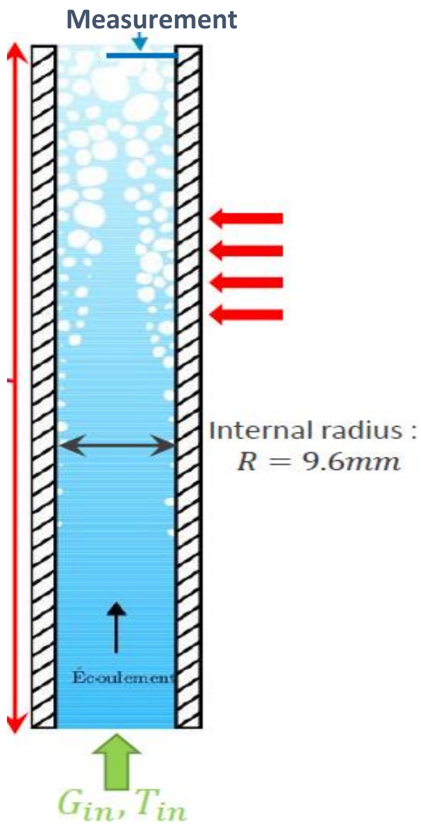
Description of DEBORA Experiment

- DEBORA is **subcooled flow boiling** experiment performed 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**



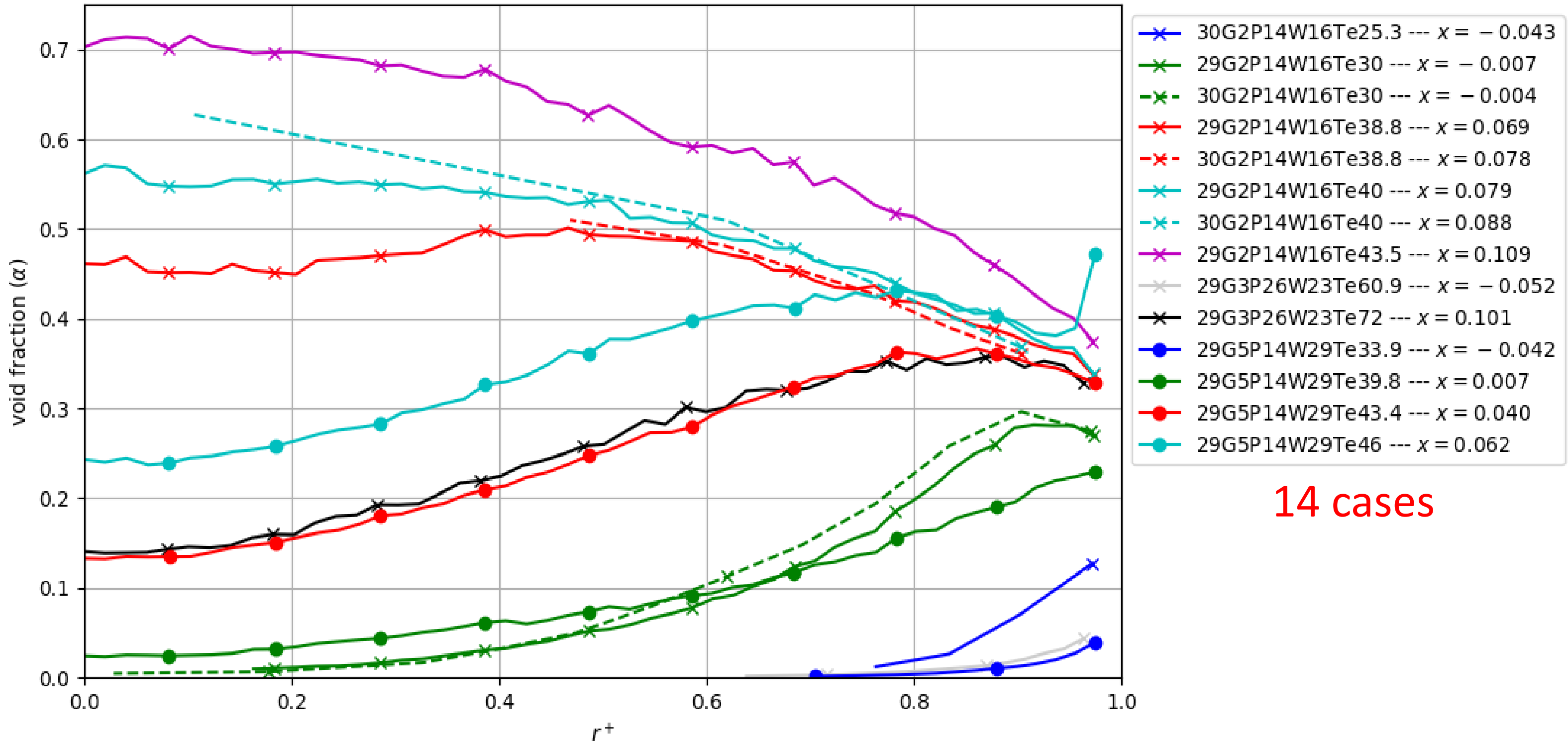


Organization and Objectives

Phase 1

Challenges

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



14 cases

Numerical and Physical Models

Wall Heat Flux Partitioning

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

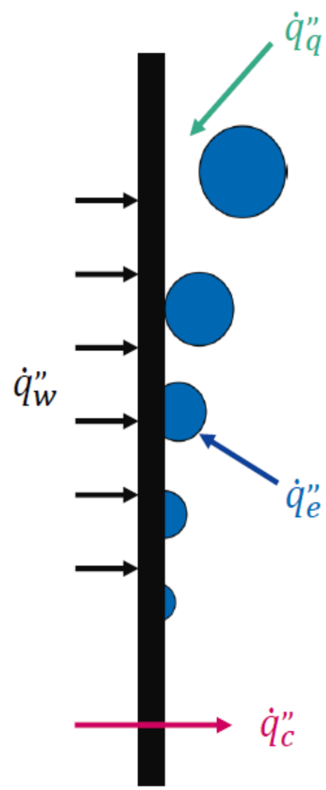
WHFP "RPI-model"

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

Single-phase Convection | Quenching | Evaporatior

Models are needed for

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



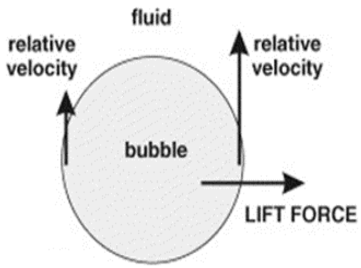
Parameters	Model
Wall Boiling	
Active nucleation site density	Hibiki-Ishii
Bubble departure diameter	Unal
Bubble departure frequency	Cole
Non-Drag forces	
Wall lubrication force	Antal
Bubble lift force	Tomiyama
Turbulence dispersion force	Gosman
Turbulence	Standard *k-ε*
Bubble induced turbulence	Kataoka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

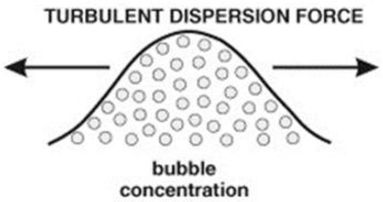
Numerical and Physical Models

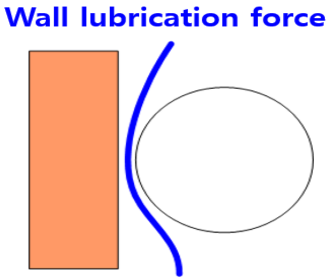
Interfacial Non-drag Forces

- Generated bubbles movement in radial direction is controlled by the momentum interfacial non-drag forces

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

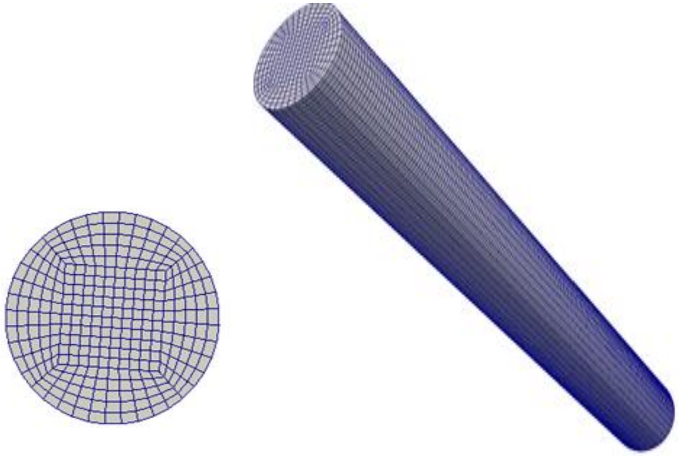






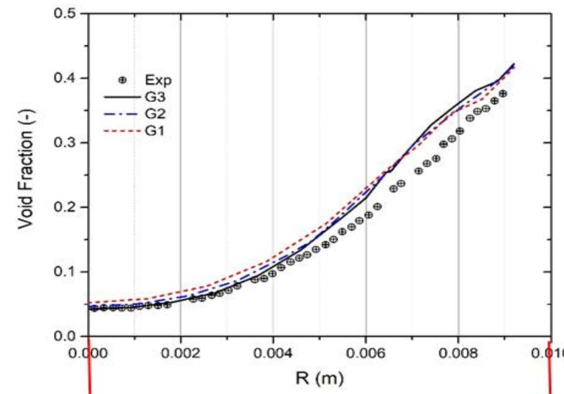
Preliminary Results

Computational Mesh and Test Matrix



<Full representation of the CFD domain>

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



Pipe center

Heated wall

Grid sensitivity

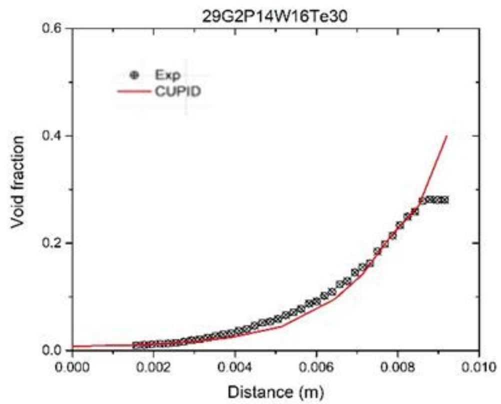
	Pressure (MPa)	Inlet subcooling (K)	Heat flux (W/m²)	Mass flowrate (Kg/m²s)	Test number
DEB 1	1.46	26.2	76240.0	2030	29G2P14W16Te30
DEB 2	1.46	14.4	76260.0	2022	29G2P14W16Te38.8
DEB 3	1.46	16.2	76260.0	2022	29G2P14W16Te40
DEB 4	1.46	12.5	76260.0	2024	29G2P14W16Te43.5
DEB 5	1.46	21.2	135000	5063	29G5P14W29Te33.9
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

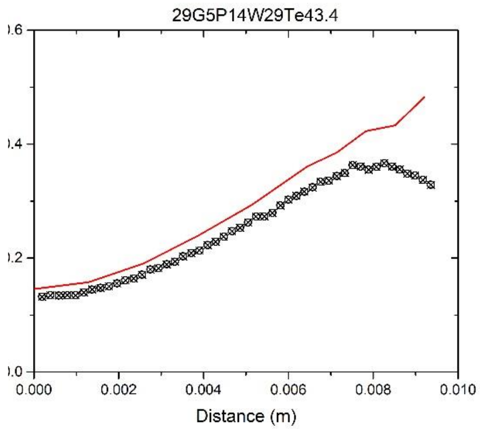
Calculation Results (1)

Wall Peaking Cases

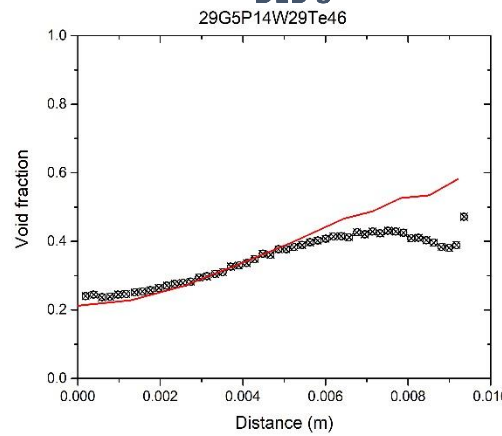
DEB 1



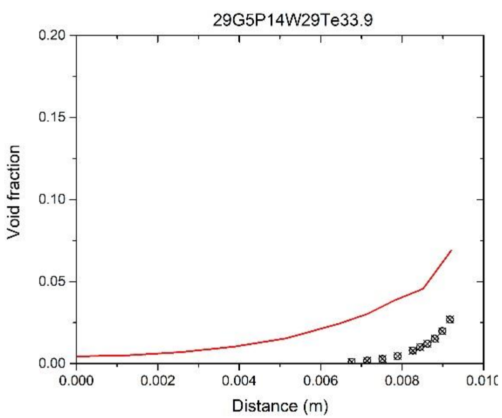
DEB 7



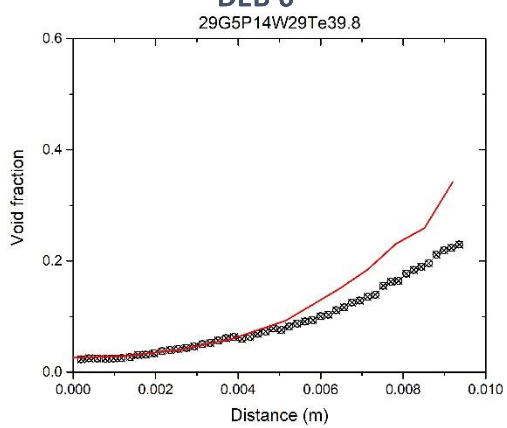
DEB 8



DEB 5

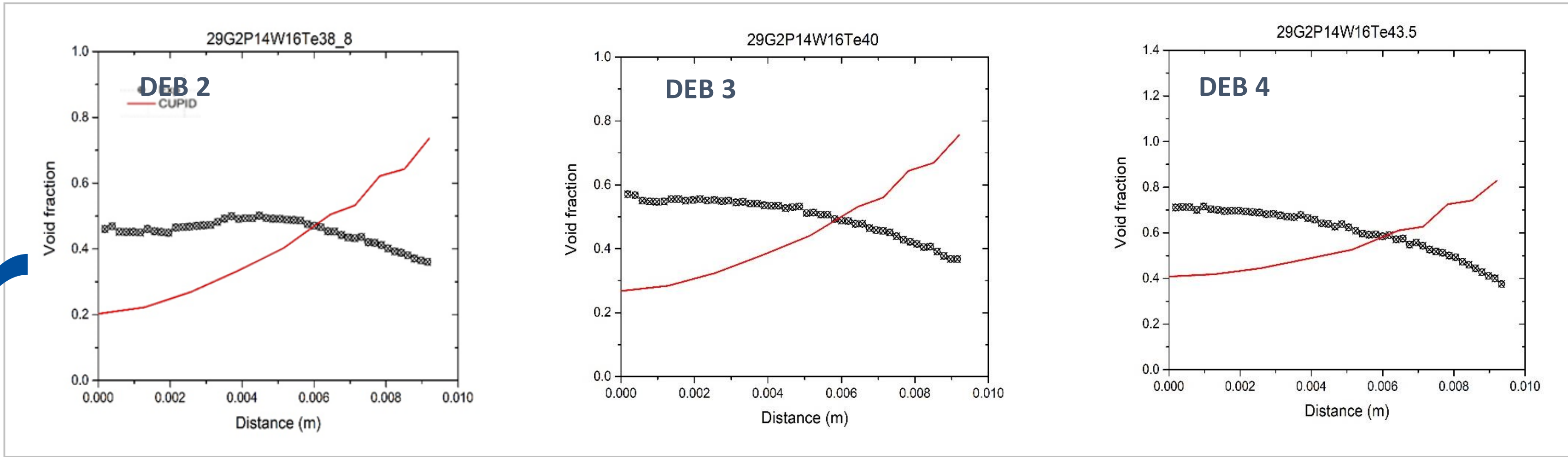


DEB 6

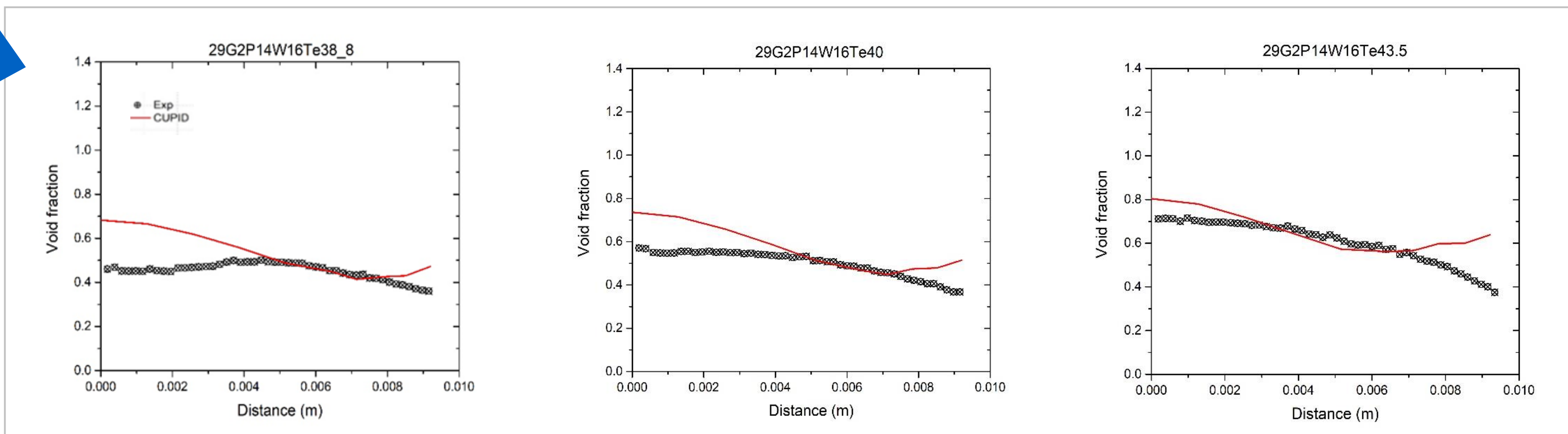


Calculation Results (2)

Core Peaking Cases



Improvement of lift force model

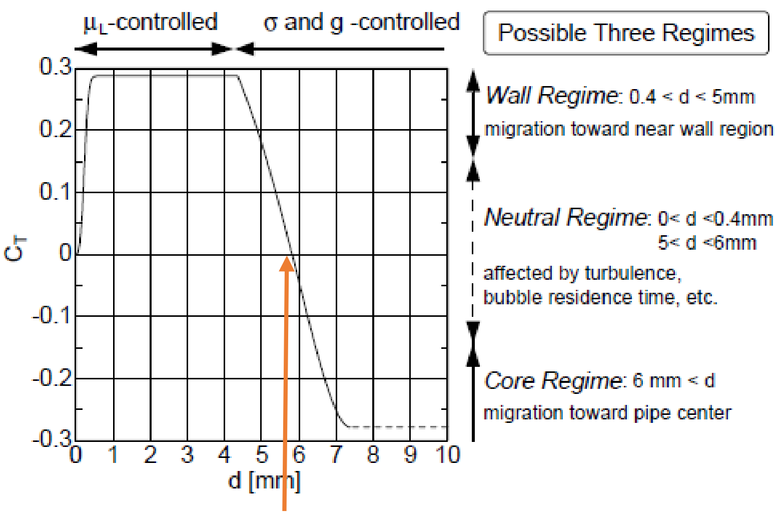


Model Improvement

Modifying Tomiyama model

- Tomiyama model was developed under **atmospheric pressure conditions**
- Change of the lift force coefficient sign occurs at bubble **sizes larger than 5.8 mm**
- At high pressure bubble sizes are smaller, Accordingly to apply the Tomiyama model the threshold for sign change should be modified

Original Tomiyama model



Change of lift force coefficient sign

- **Modified criteria** was set using (Jacob/Boiling) and Reynolds liquid number

$$Ja = \frac{c_{pl}(T_{sub,l})\rho_l}{(h_g^{sat} - h_l)\rho_g}$$

$$Bo = \frac{q}{G(h_g^{sat} - h_l)}$$

$$\mathrm{Re}_l = \frac{D_h \rho_l v_l}{\mu_l}$$

(Jacob/Boiling) < 345 and Reynolds liquid number < 590000
→ lift force coefficient becomes negative

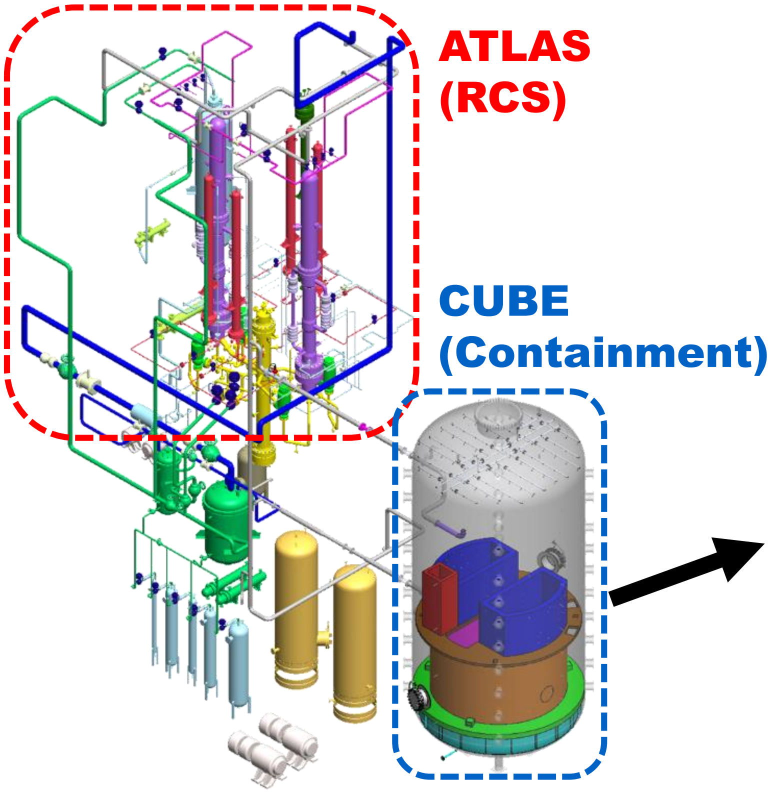
ATLAS-CUBE

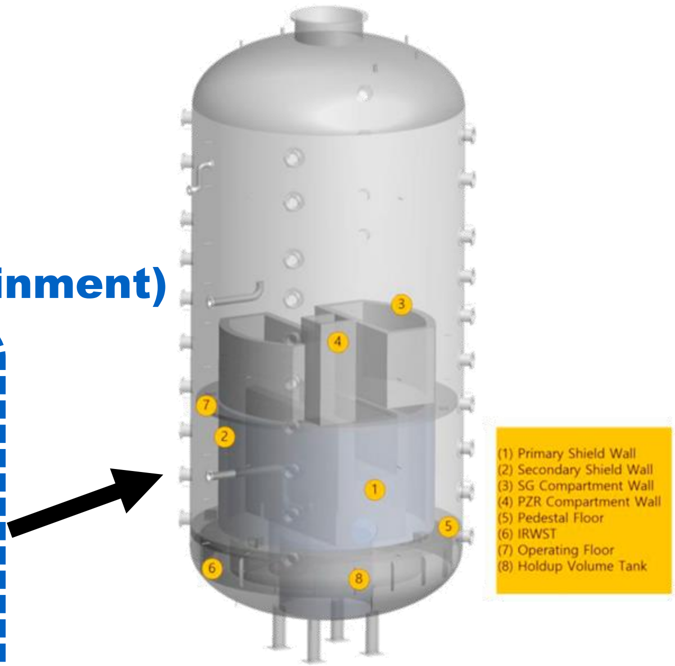
6

Overview of ATLAS-CUBE

Description of Test Facility

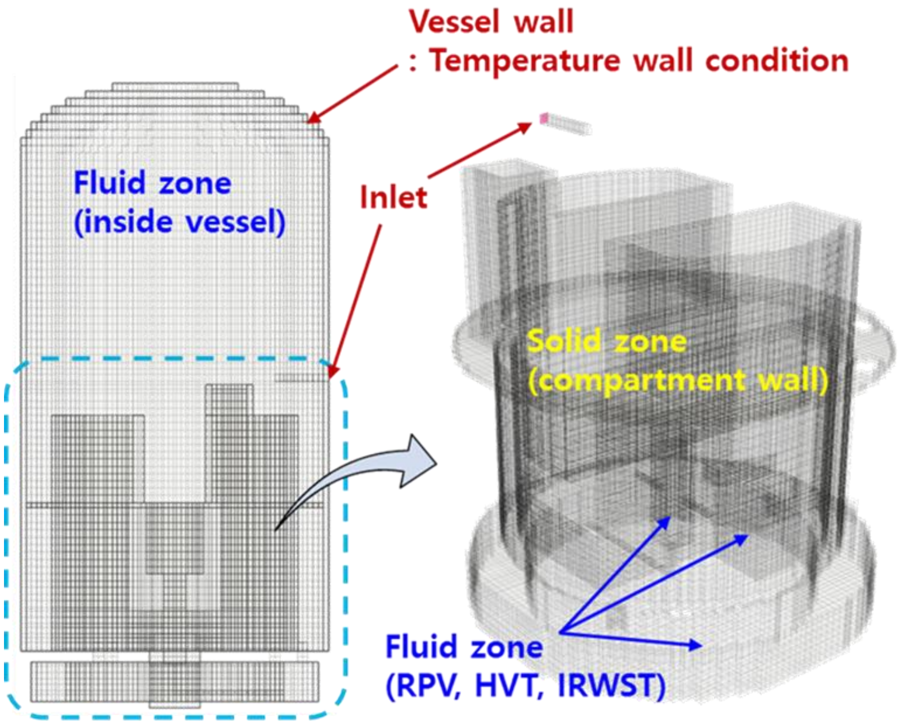
- **Containment Utility for Best-estimate Evaluation (CUBE)**
- **Multi-dimensional behavior of pressure and temperature in the containment with simulating an energy release from ATLAS RCS**

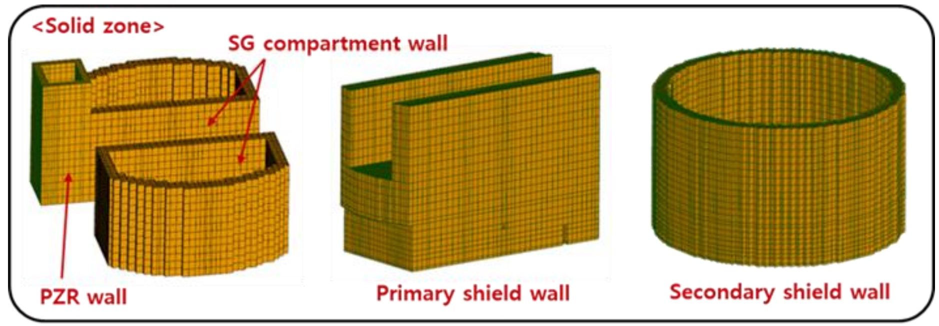


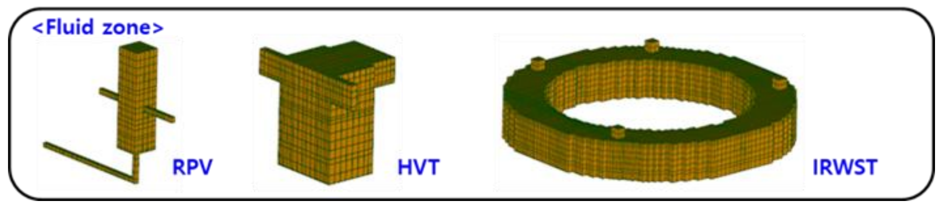


Mesh Generation

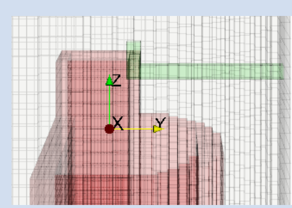
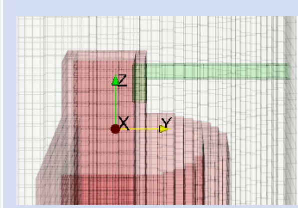
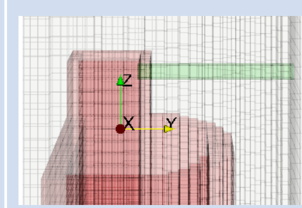
Hexahedral structured mesh







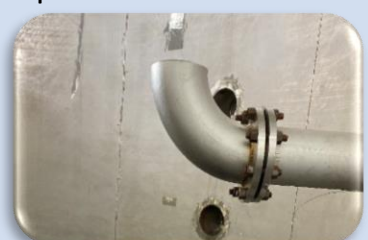
369,446개 셀로 구성된 정렬 격자
- Fluid region: 328,969 cells
- Solid region : 40,477 cells (29.5837m³)

ST2-CT-01	ST2-CT-02	ST2-CT-03
Upward	Downward	Horizontal
		

Test Matrix

Steam Injection Tests

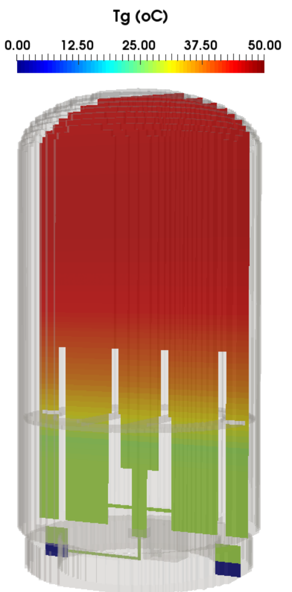
TEST ID	CUBE				RCS				
	Direction of Break	Initial Temperature(°C)			Accident Condition	SI		AFW (kg/s)	Power (%)
		Inner wall	Fluid	Compartments		SIP	SIT		
ST2-CT-01	Upward	50	~40	~30	MSLB	0.2 kg/s of steam supply			
ST2-CT-02	Downward	50	~40	~30	MSLB	0.2 kg/s of steam supply			
ST2-CT-03	Horizontal	50	~40	~30	MSLB	0.2 kg/s of steam supply			

	ST2-CT-01	ST2-CT-02	ST2-CT-03
Steam injection directions (Pipe size : 6 inch)	Upward 	Downward 	Horizontal 

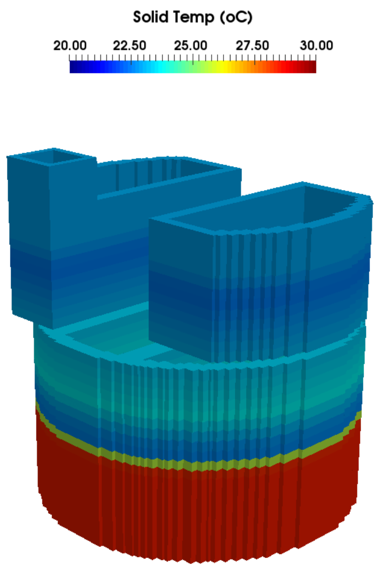
Initial & Boundary Conditions

Initial Condition

Initial conditions (Nominal value)	
Fluid temperature [℃]	40
Vessel wall temperature [℃]	50
Compartment solid temperature [℃]	30
System pressure [MPa]	0.112168
Void fraction	1.0
NC gas (Air) quality	1.0



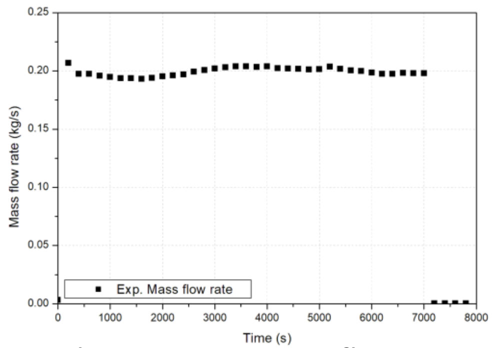
< Initial gas temp.>



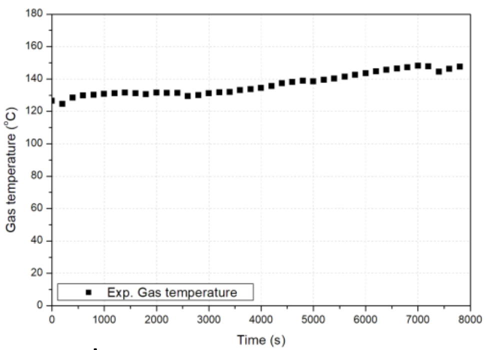
< Initial structure temp.>

Boundary Condition

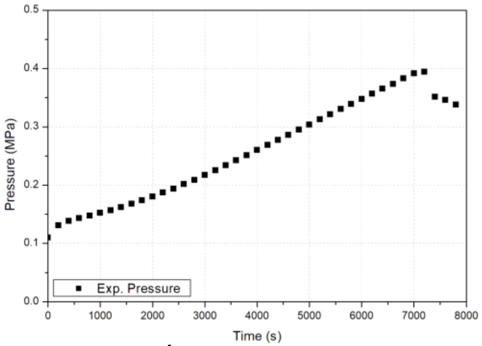
Boundary conditions	
Void fraction	1.0
NC gas (Air) quality	0.0



< Inlet : steam mass flow rate >



< Inlet : steam temperature >



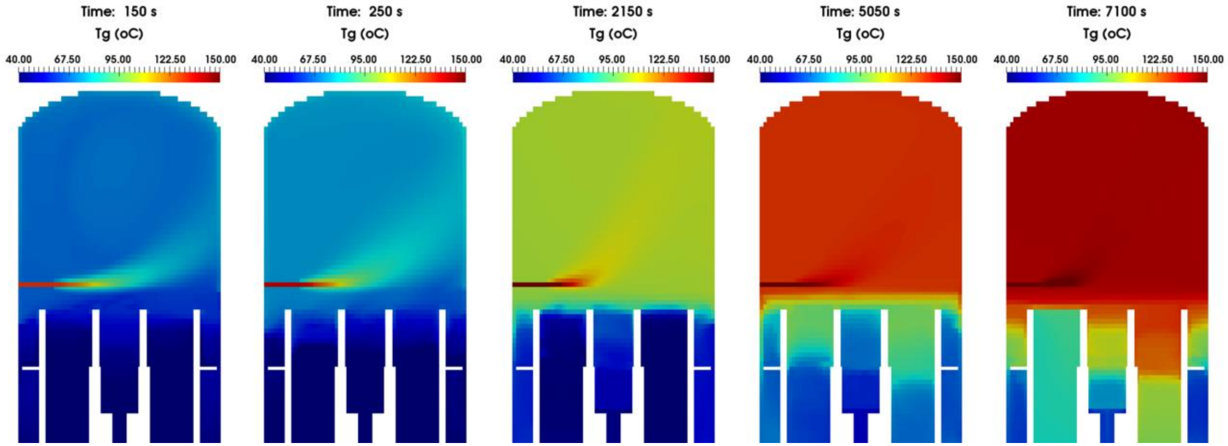
< Inlet : Pressure >

Calculation Setup

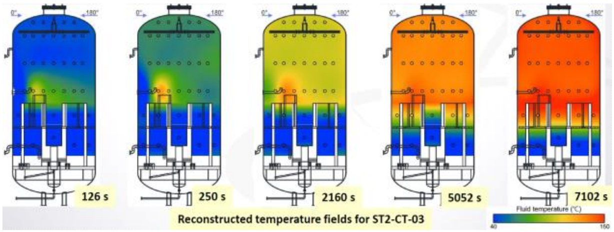
- **CUPID 2.5 version**
- **Turbulence model**
  - **Standard k-ε with turbulence buoyancy effect (k and ε)**
  - **Wall treatment : wall function**
- **Radiative heat transfer model**
  - **P-1 model**
- **Wall condensation model**
  - **Empirical model : Uchida**
- **Computing information**
  - **Simulation time : 8,000s**
  - **Total CPU time : 13hr (30 CPU cores)**

Calculation Results (1)

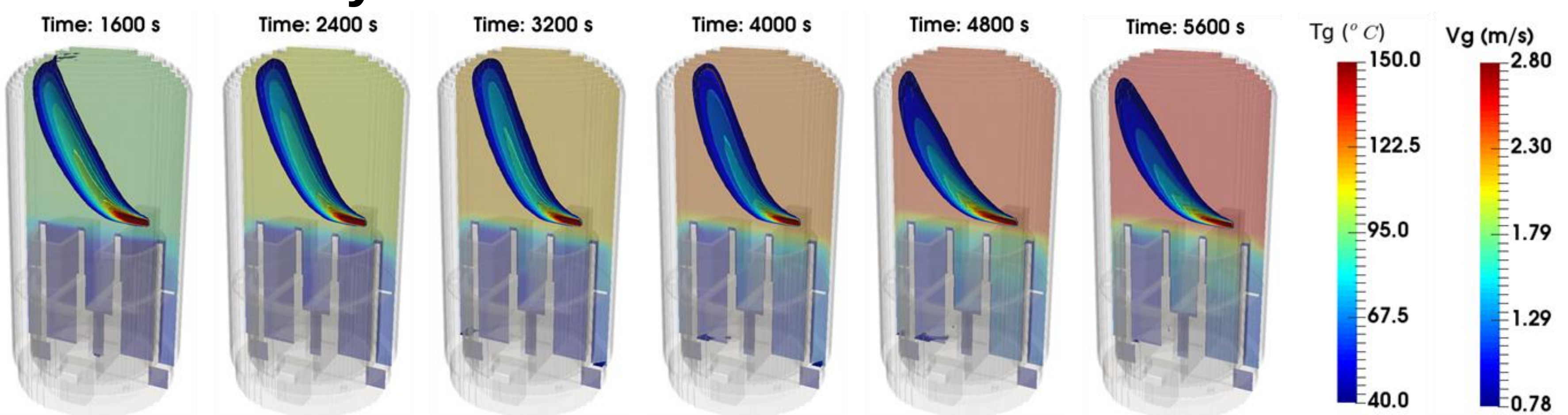
Gas Temperature



< Gas Temperature - Experiment>



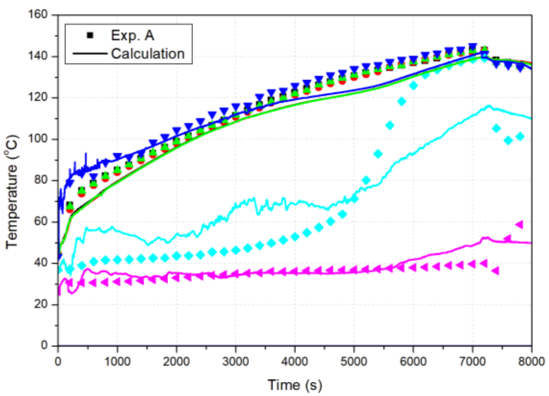
Gas Velocity



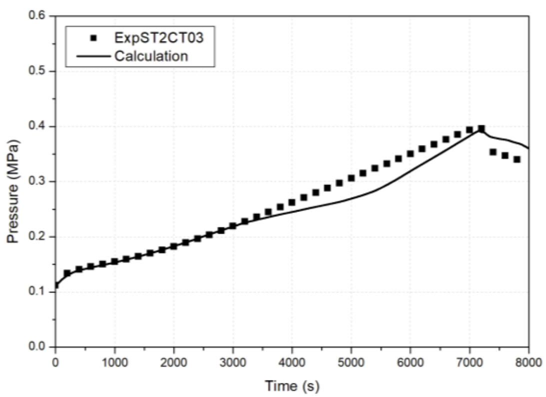
Calculation Results (2)

Prediction of system pressure and gas temperature

ST2-CT-01 (Upward)

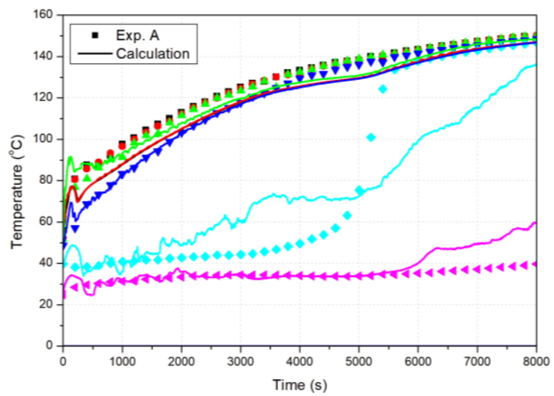


Gas temperature along the elevation

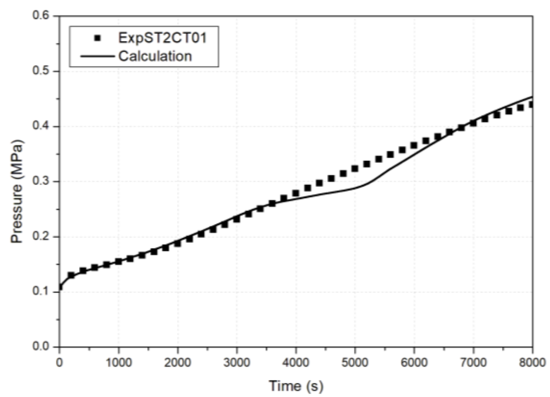


Pressure in containment vessel

ST2-CT-02 (Downward)

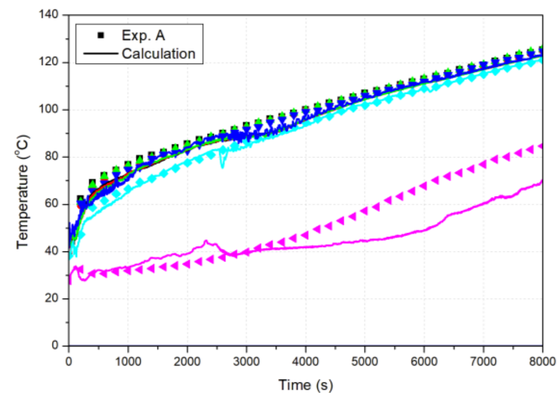


Gas temperature along the elevation

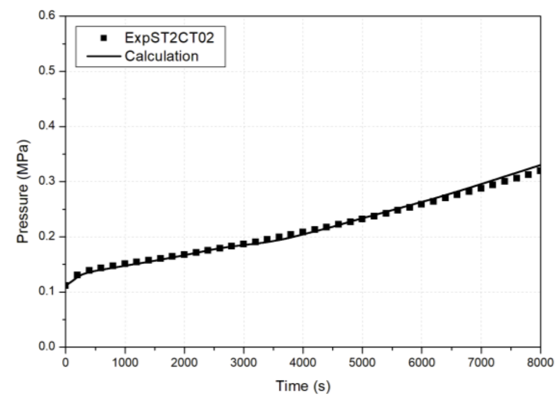


Pressure in containment vessel

ST2-CT-03 (Horizontal)



Gas temperature along the elevation



Pressure in containment vessel

Conclusions

7

Conclusions

Validation of CUPID via international benchmarks

- Radiation model, turbulence model, wall condensation model, and liquid/gas mixing

Summary

OECD/NEA IBE-4
- Turbulence mixing(modified k-ε model) due to the density difference

IAEA CRP
- Turbulence mixing(low Reynolds number k-ε model) in complex geometry
- Diffusion due to the concentration difference

OECD/NEA HYMERES-2
- Thermal stratification with radiation model
- Turbulence mixing in complex geometry

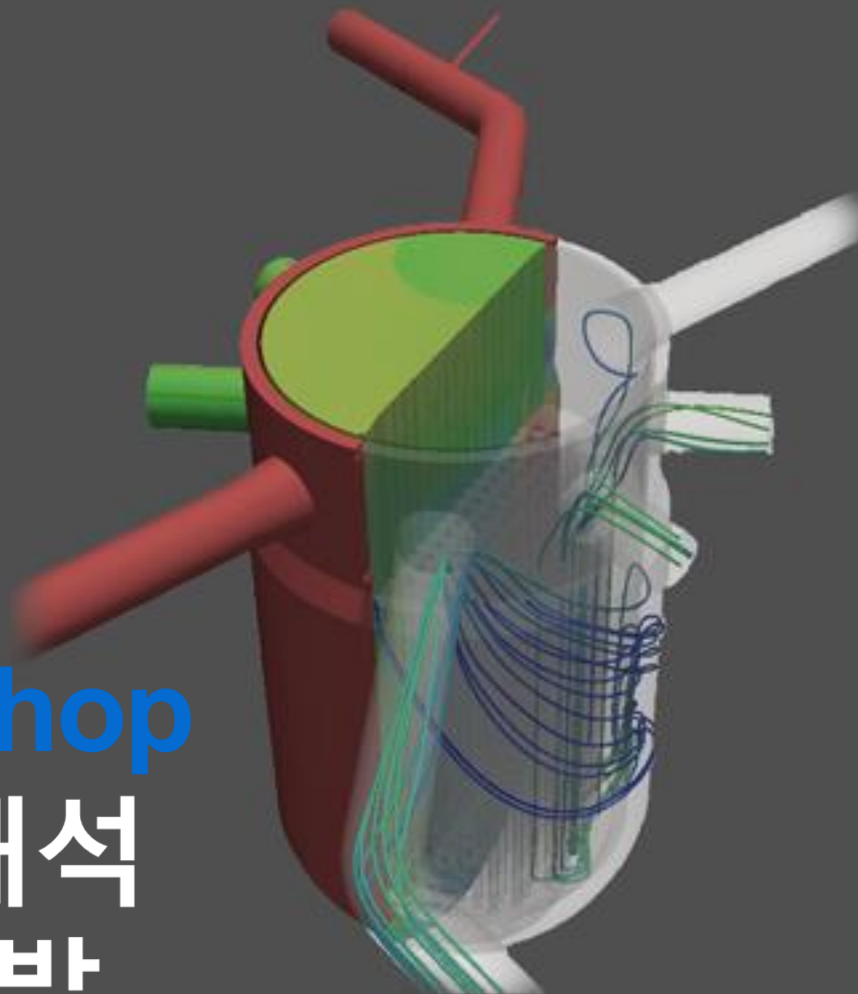
DEBORA Benchmark
- Wall heat flux partitioning model
- Bubble lift force model

ATLAS-CUBE
- Wall condensation model
- Turbulence mixing model

THANK YOU

yjcho@kaeri.re.kr

Korea Atomic Energy Research Institute
KAERI
한국원자력연구원



10th CUPID Workshop

원자로 안전 통합해석 플랫폼 (MARU) 개발

박 익 규

2022년 8월 23일

CONTENTS

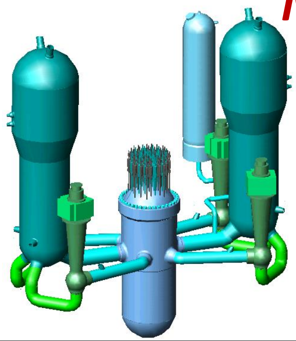
원자로 통합 안전해석

1

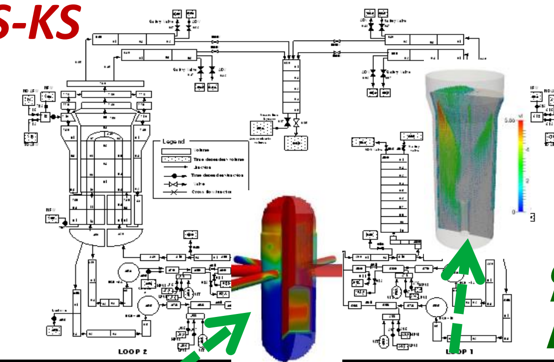
- 다중스케일 연계 해석
- 다물리 연계 해석
- 다물리 다중스케일 원자로 통합 안전해석

다중스케일 원자로 열수력 안전해석 기법

계통 스케일
(거대 장치)



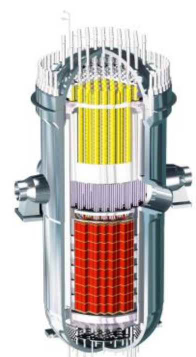
MARS-KS



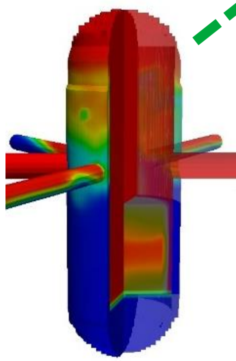
~10^{0} m

연계
다중스케일
계산

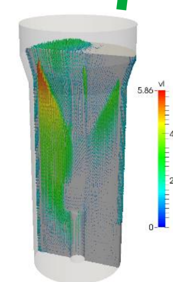
기기 스케일
(CFD-Porous)
(부수로 스케일)



CUPID-RV



CUPID-SG

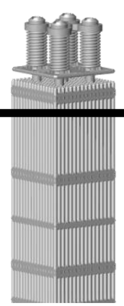


~10^{-3} m

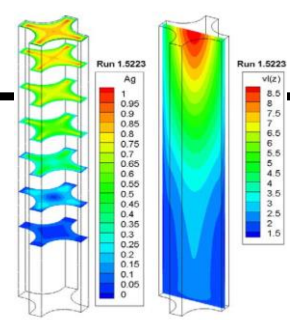
CFD-RANS

CFD-LES

CFD-DNS
(난류 등 세부 현상)



CUPID-CFD

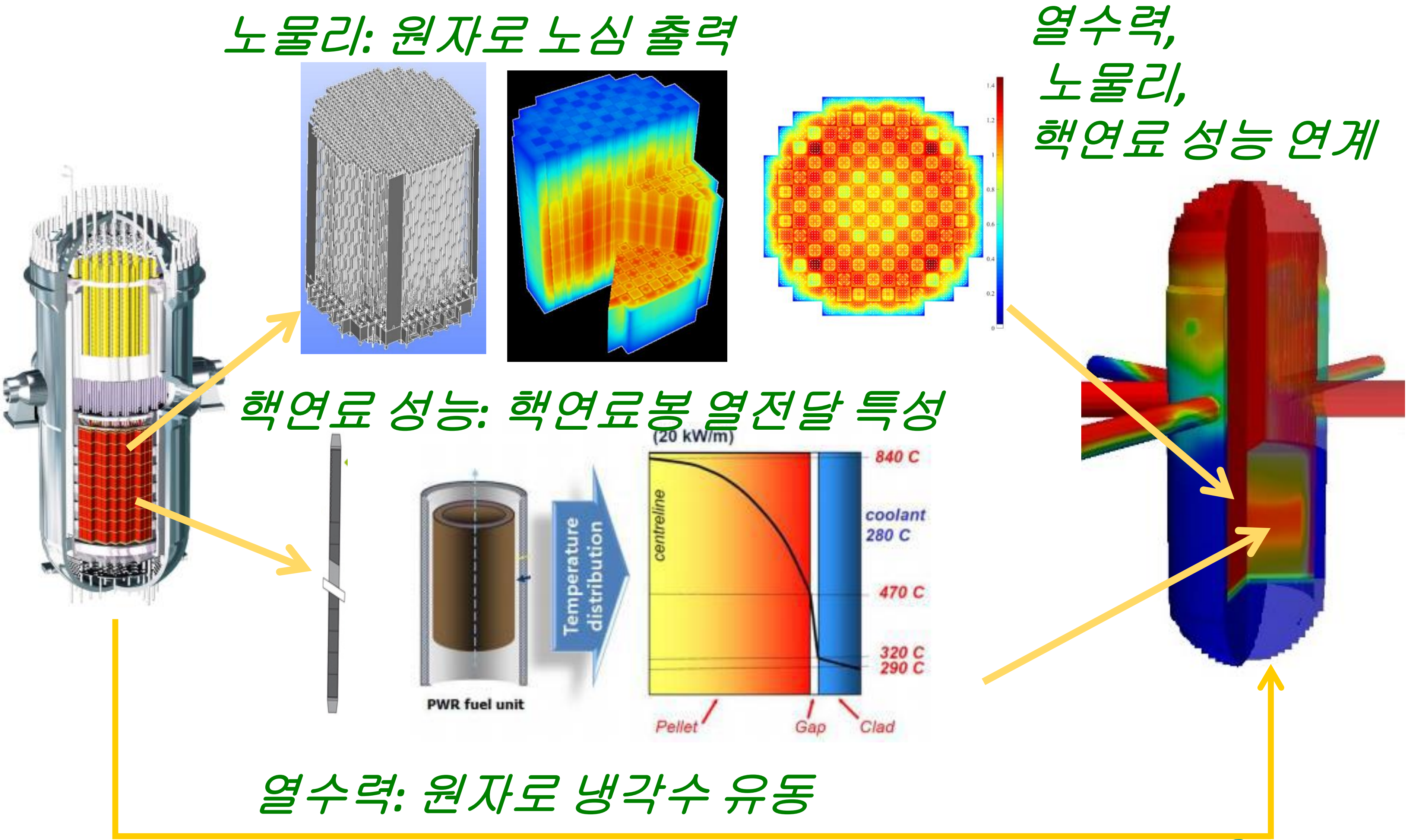


~ 10^{-4} m

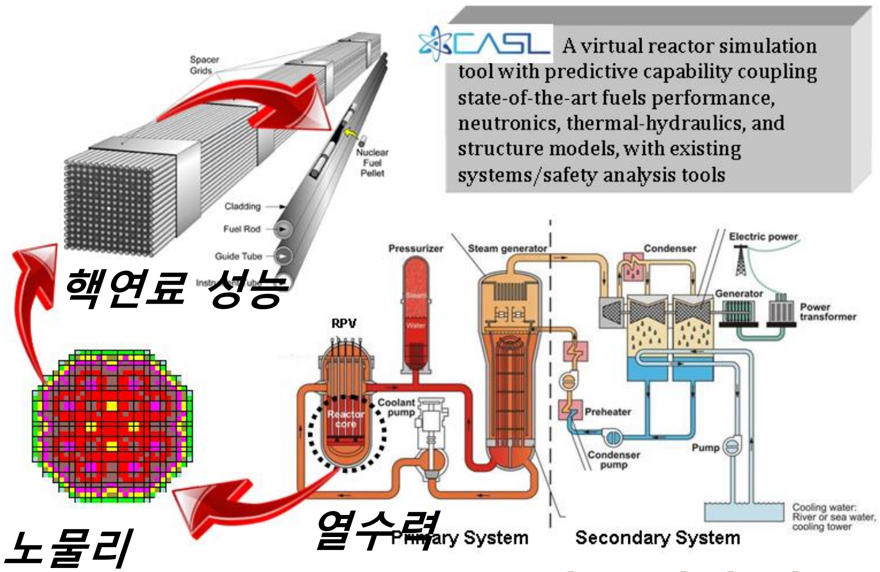
~ 10^{-5} m

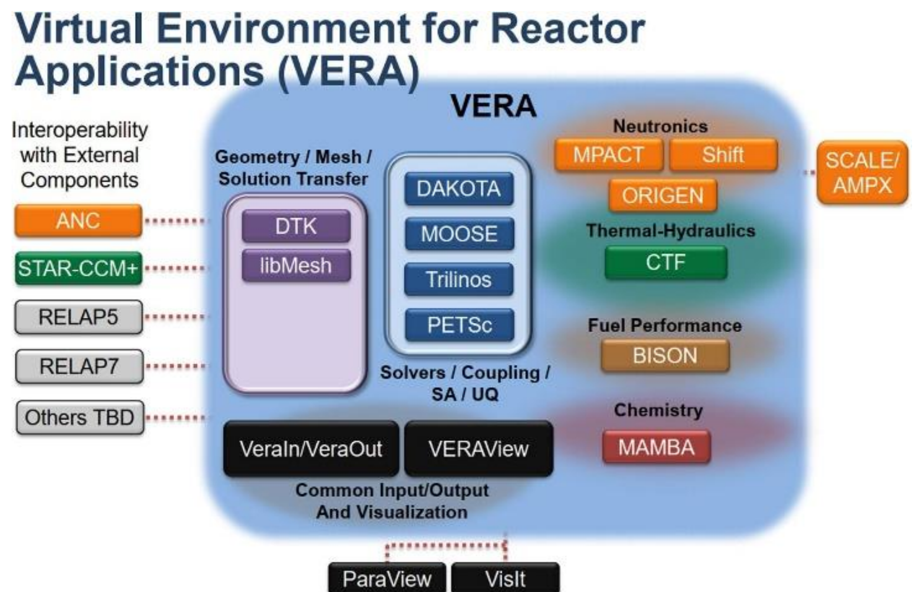
~ 10^{-6} m

원자로 노심 다물리 해석 기법

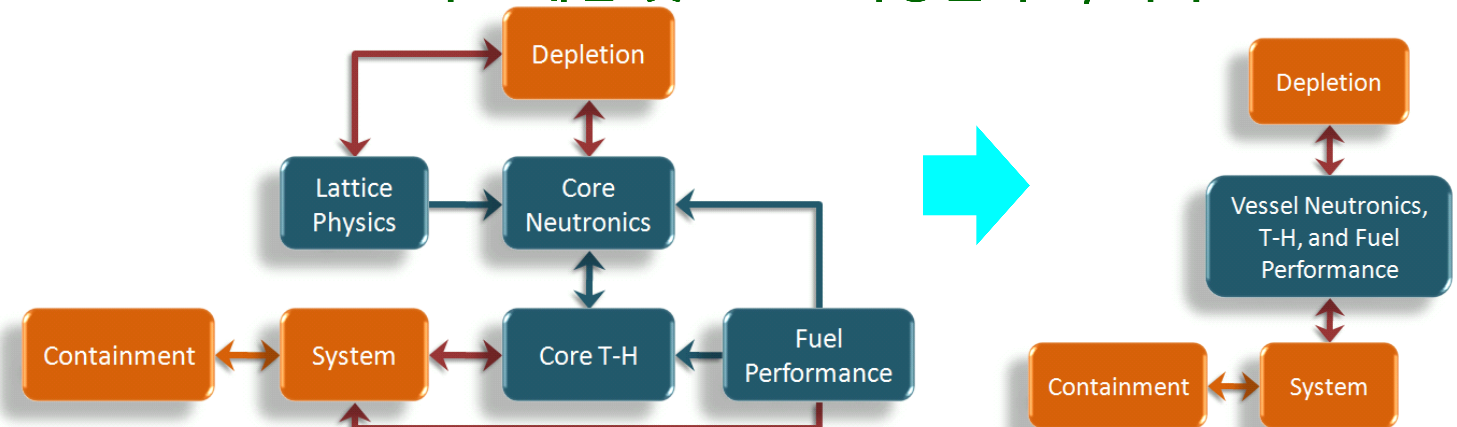


다물리 다중스케일 기법 적용 CASL 프로젝트



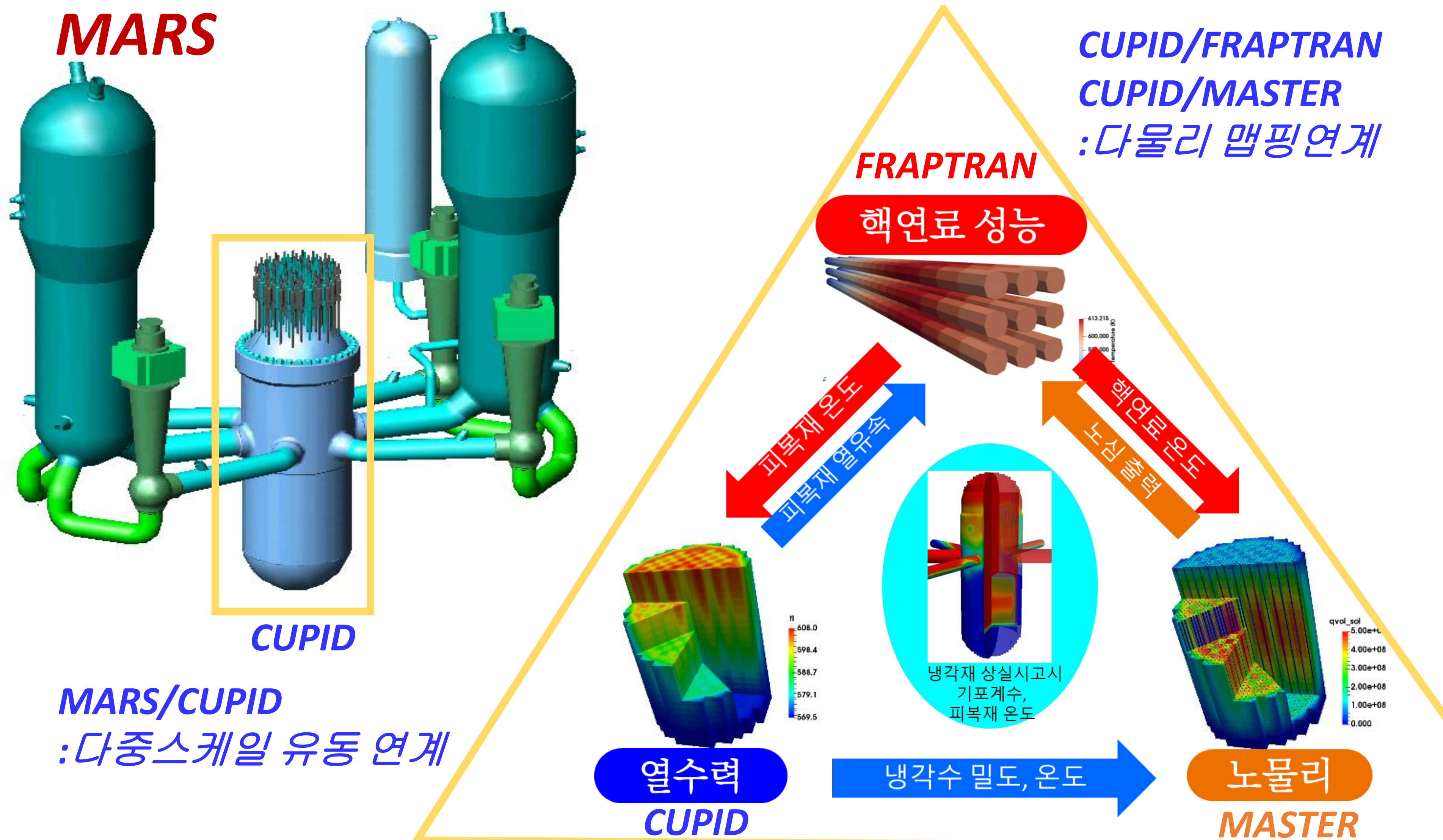


<CASL 프로젝트 개념 및 VERA 가상원자로, 미국 DOE>



<기존 개별 및 CASL-연계 원자로 디자인 업무 수행도>

다물리 다중스케일 원자로 통합 안전해석 체계



다물리 연계 해석 체계

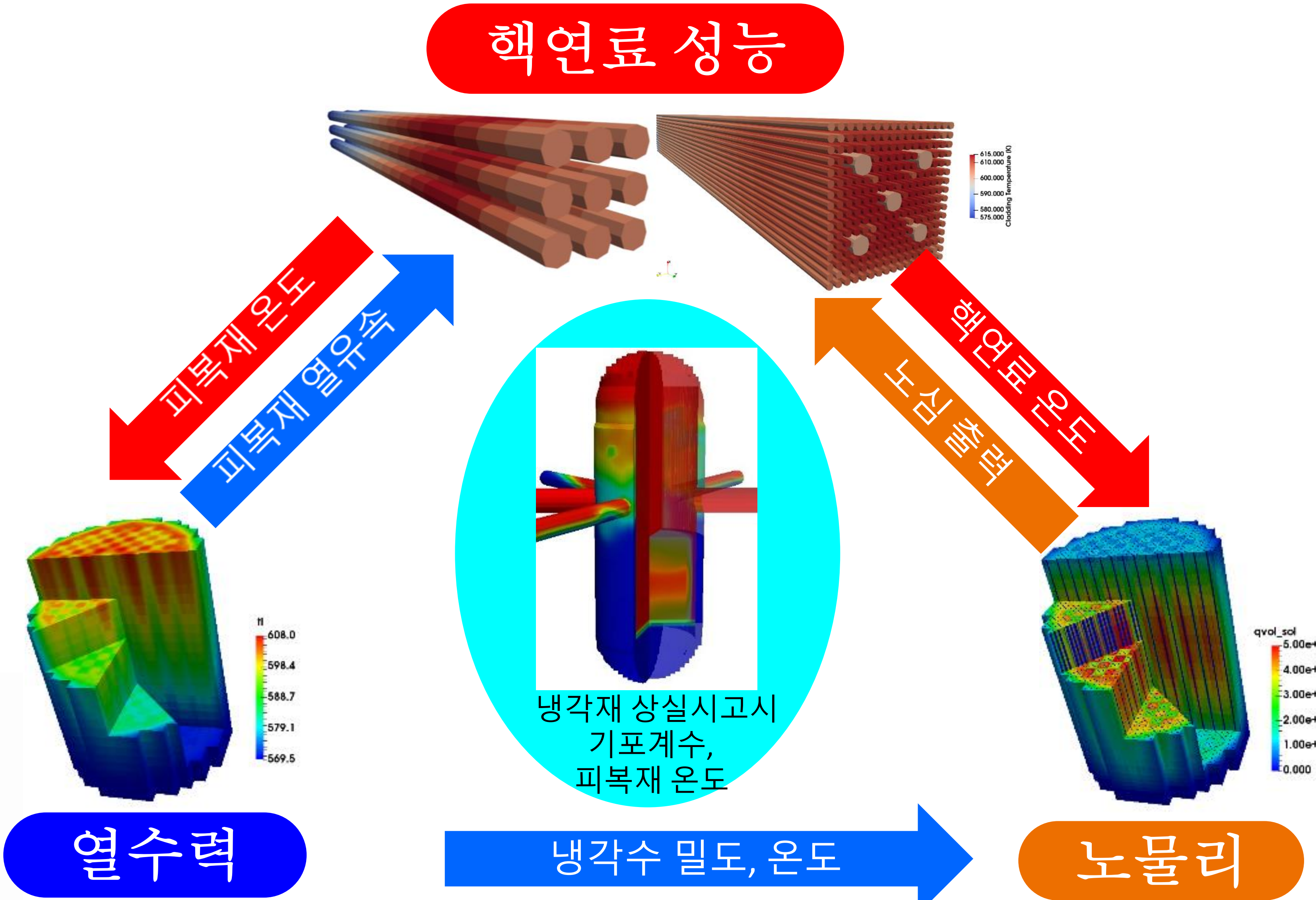
2

– 노물리/열수력 연계 기법

– 핵연료/열수력 연계 기법

– 다물리 연계 검증 계산

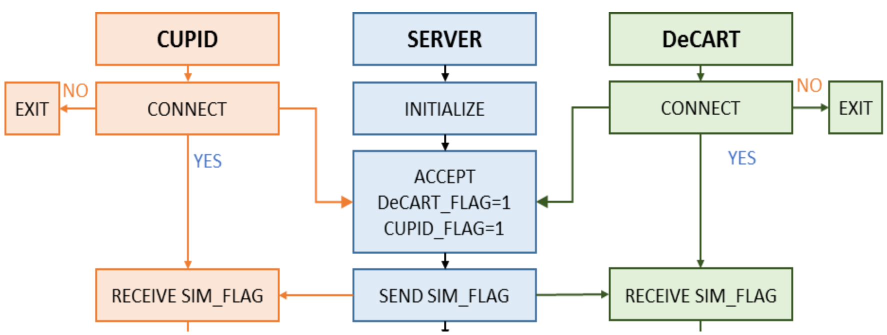
CUPID 기반 다물리 연계 해석 체계



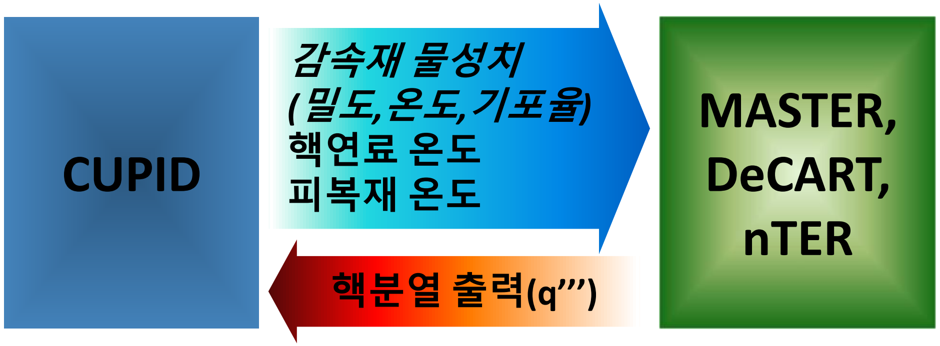
노물리/열수력 연계 기법

노물리 코드: MASTER,DeCART, nTER

- CUPID: 3차원 원자로 노심 열수력 코드
- MASTER: 3차원 중성자 확산해석 코드 (집합체 단위 혹은 봉단위) [KAERI]
- DeCART, nTER: 3차원 중성자 수송 해석 코드(봉단위) [KAERI, SNU]
- CUPID/MASTER: 동적 라이브러리 (DLL)
- CUPID/DeCART,nTER: TCP/IP 소켓 통신



<TCP/IC 소켓 통신>

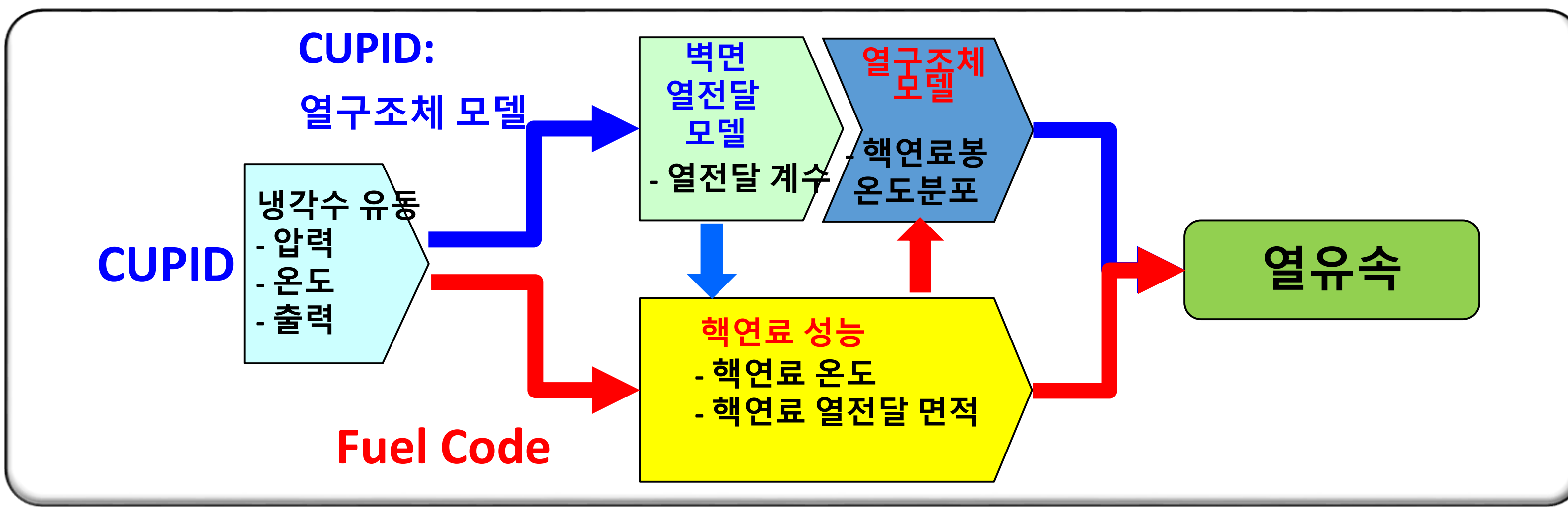


<노물리/열수력 연계 변수>

핵연료/열수력 연계 기법

핵연료 성능해석 코드: FRAPTRAN, MERCURY

- 핵연료 성능 및 핵연료 내부 열전달 현상
  - FRAPTRAN [미국 NRC], MERCURY [KAERI]
- 열수력 코드: 유동 해석, 벽면 열전달 계수
- 핵연료 코드: 핵연료 온도, 핵연료/냉각수 열전달 면적
- 열수력 코드: 핵연료/냉각수 열유속

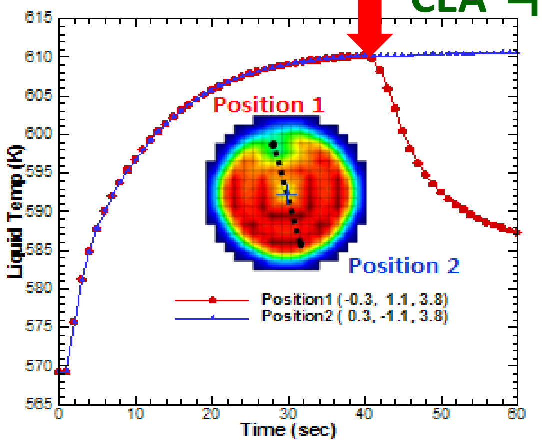


<핵연료/열수력 연계 계산 절차>

노물리/열수력 연계 검증 계산

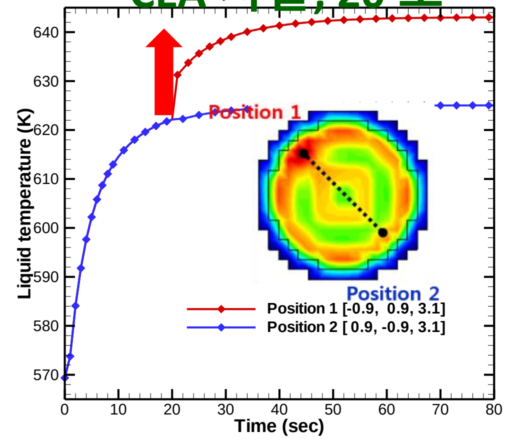
» 제어봉집합체(CEA) 낙하 혹은 사출(CUPID/MASTER)

CEA 낙하, 40초



<CEA 낙하후 냉각수 온도>

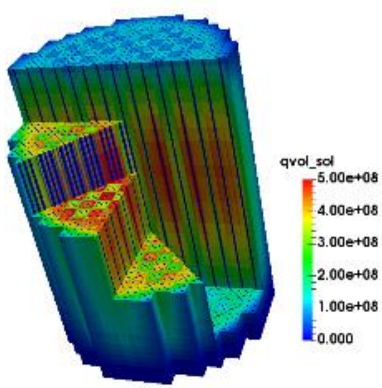
CEA 사출, 20 초



<CEA 사출후 냉각수 온도>

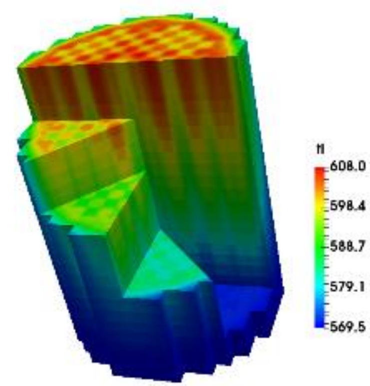
*CEA: Control Element Assembly

» OPR1000 전노심 열출력 연계 계산 (CUPID/DeCART)



<노심 열출력>

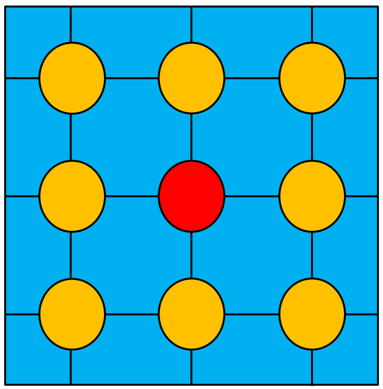
Center-peak 출력분포

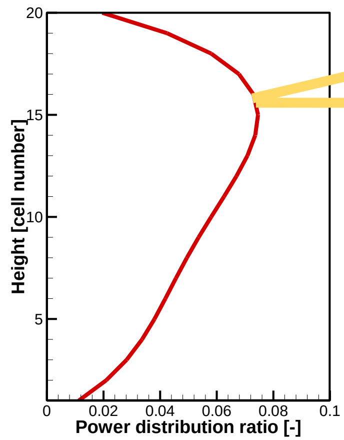


<냉각수 온도>

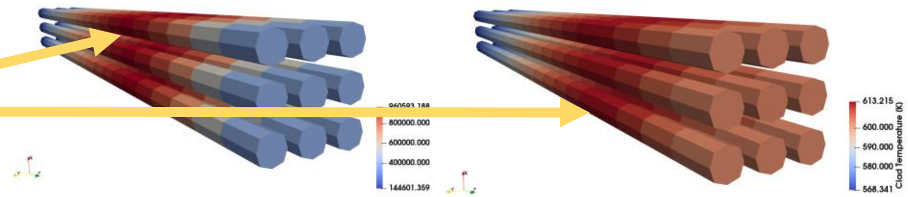
핵연료/열수력 연계 검증 계산

» 3x3 핵연료봉 다발 채널

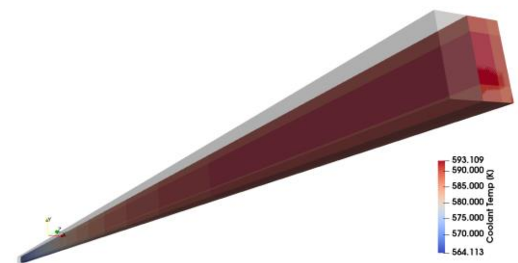




< 축방향 열출력 분포 입력>

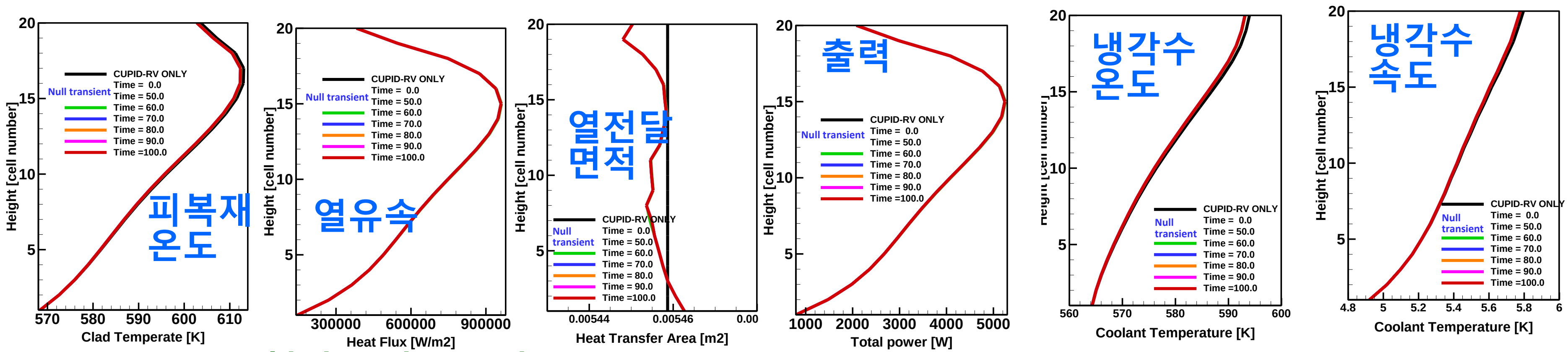


<핵연료봉 열유속 및 핵연료 온도>



<냉각수 온도>

- CUPID 단독 계산과 핵연료 연계 계산 비교



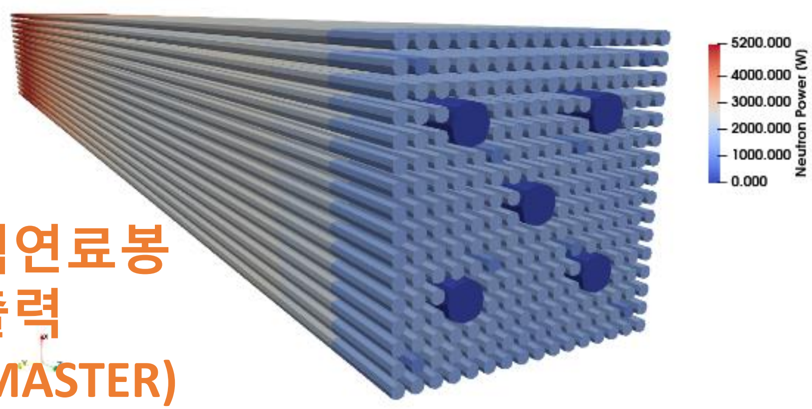
<핵연료 성능 분석>

<냉각수 열수력 거동 분석>

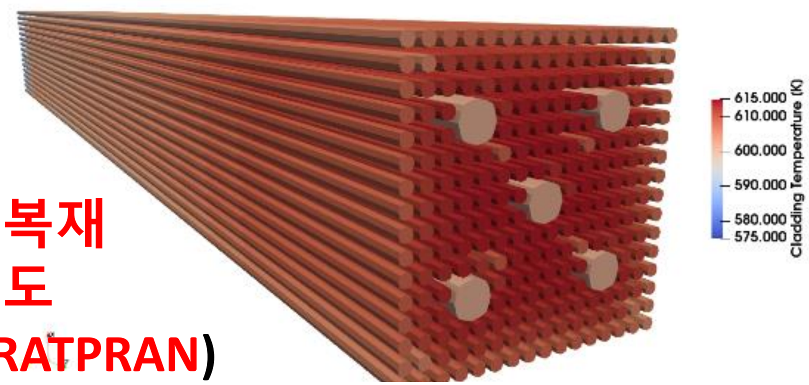
핵연료/노물리/열수력 연계 검증 계산

APR1400 핵연료 집합체 채널

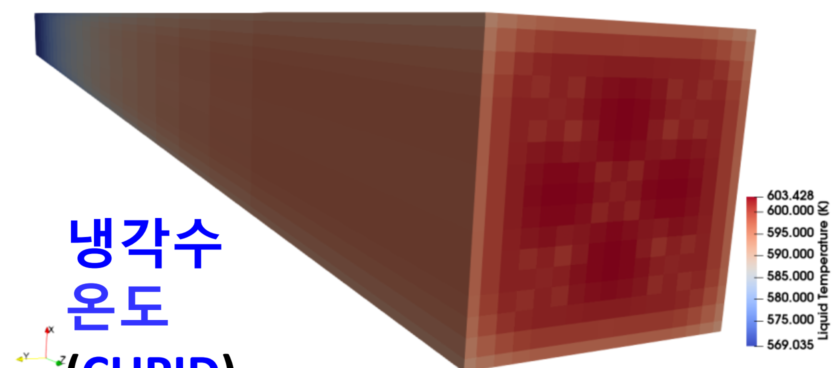
- 16x16 핵연료 봉다발 채널
- 출력: MASTER
- 핵연료 성능: FRAPTRAN
- 원자로 냉각수 유동: CUPID



핵연료봉
출력
(MASTER)



피복재
온도
(FRATPRAN)



냉각수
온도
(CUPID)

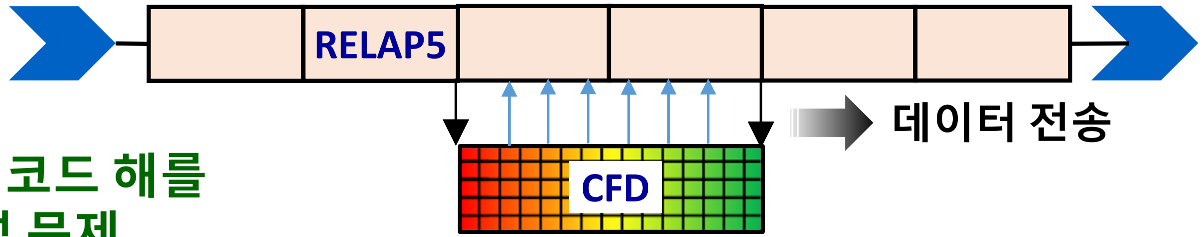
다중스케일 연계 해석 체계

3

- 다중스케일 연계 기법
- CUPID기반 다중스케일 연계 기법
- 다중스케일 연계 검증 계산

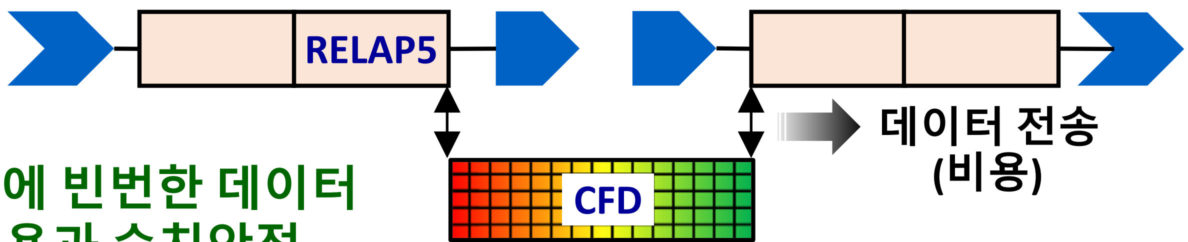
다중스케일 연계 기법

영역 중첩



CFD 코드가 계통 코드 해를 덮어쓴다: 보존성 문제

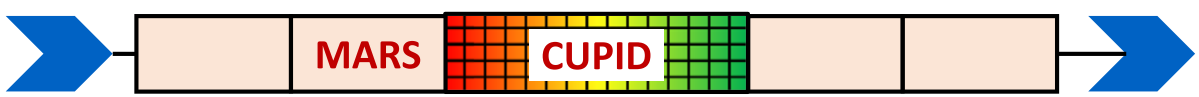
영역 분할



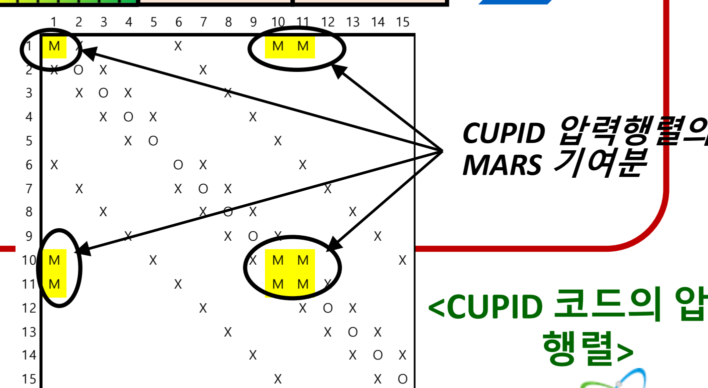
두 코드 솔버 사이에 빈번한 데이터 전송이 요구됨: 비용과 수치안정

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

단일 영역



하나의 압력 솔버 행렬을 사용함
: 데이터 전송이 불필요함.
⟶ *특히, 과도상태 계산에 유리함.*



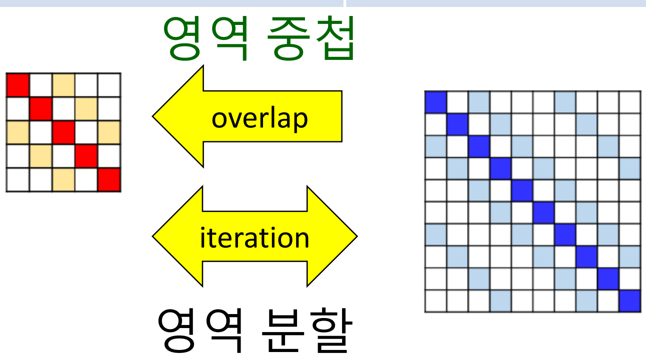
<CUPID 코드의 압력 행렬>

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

다중스케일 연계 기법 특징

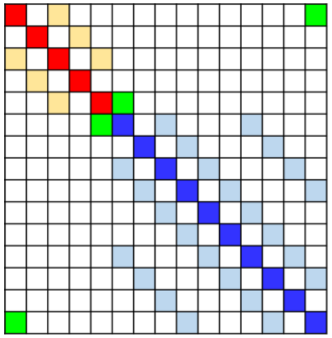
» 다중스케일 연계 기법 비교

연계 기법	특징	한계	적용 예
영역 중첩 (Domain Overlapping)	연계셀 데이터 전송 중첩 영역 해 덮어씀.	매핑 보존성 과도 계산	CATHARE2/TrioCFD SFR 자연대류
영역 분할 (Domain Decomposition)	두 개의 솔버가 연계셀 데이터를 교환하며 반복계산 수행	과도 계산	ATHLET/OpenFOAM ROCOM PKL3 Test 1.1 유동 혼합
단일 영역 (Single Domain: Implicit coupling)	두 개의 영역에 대해 하나의 솔버행렬을 구성함	두 개의 코드에 대해 원시코드가 필요함	MARS/CUPID APR1400 MSLB 사고



단일 영역 및 단일 솔버

-압력행렬 계수 변환

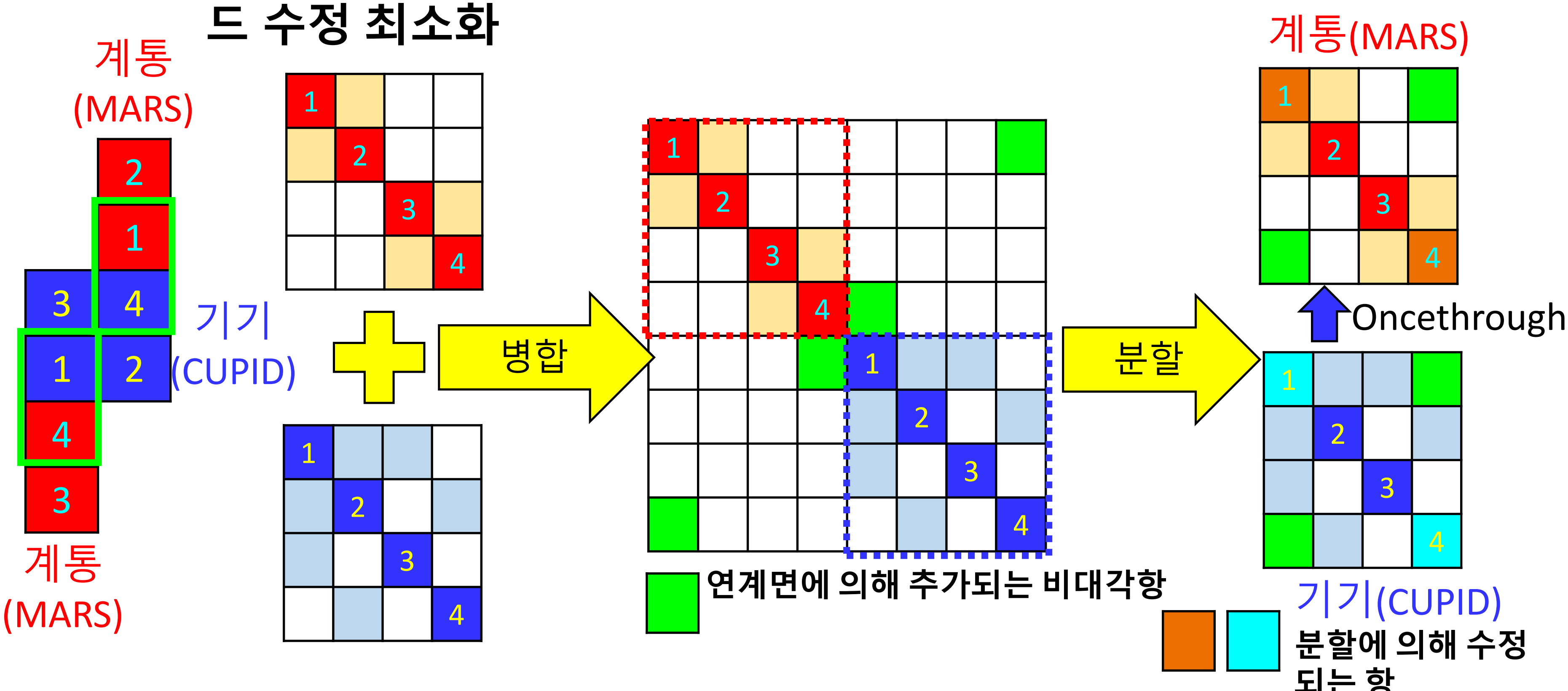


CUPID기반 다중스케일 연계 기법

MARS/CUPID 단일 영역 연계 기법

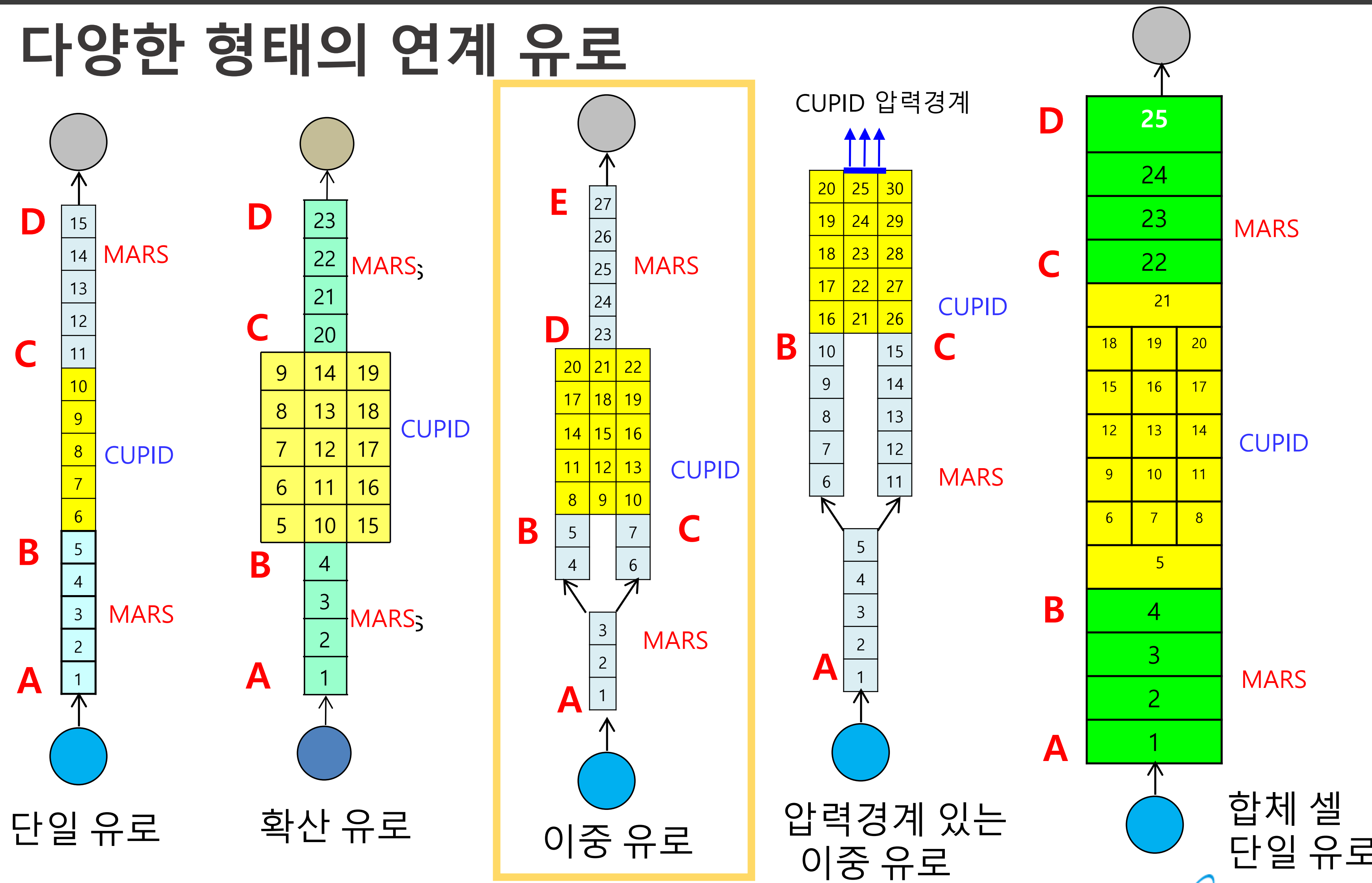
➢ 두 영역을 병합하여 단일 압력 행렬을 구성함.

- 두 코드의 압력 행렬을 병합함.
- *단일 압력 행렬을 두 코드의 솔버에 맞게 분할하는 과정 필요함* → 코드 수정 최소화



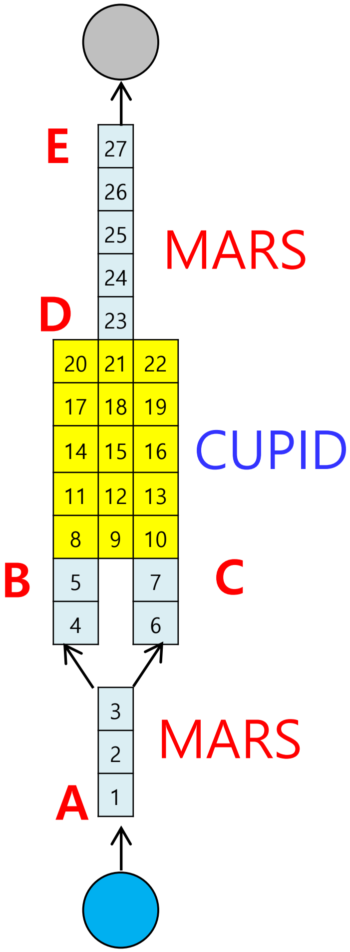
다중스케일 연계 검증 계산 (1/3)

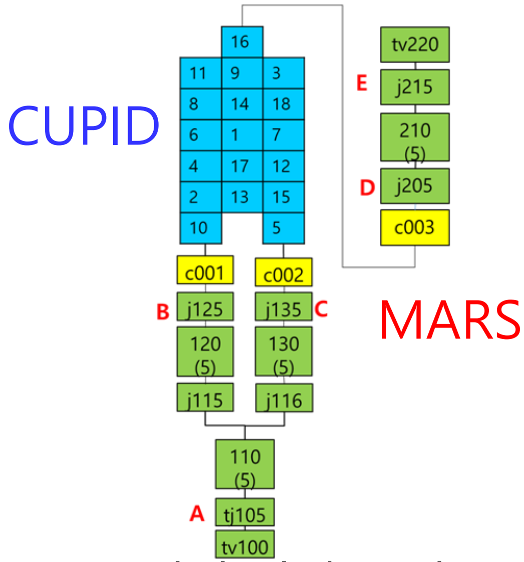
다양한 형태의 연계 유로



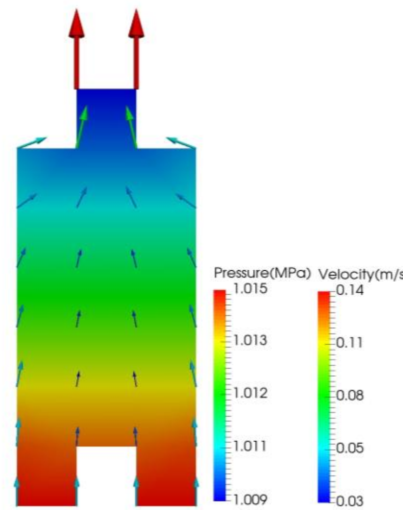
다중스케일 연계 검증 계산 (2/3)

이중 유로 연계 예제 계산 결과

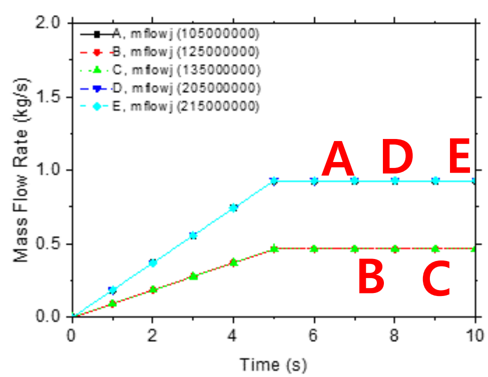




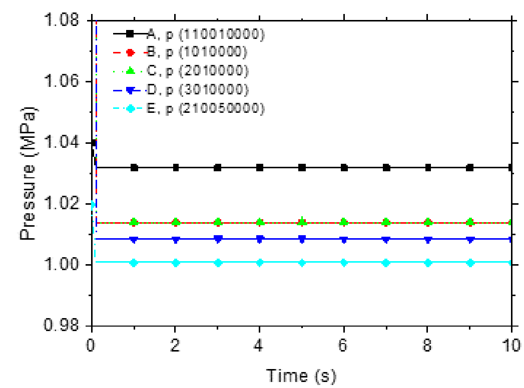
계산 격자 구성도



CUPID 영역에서 압력장 및 속도벡터



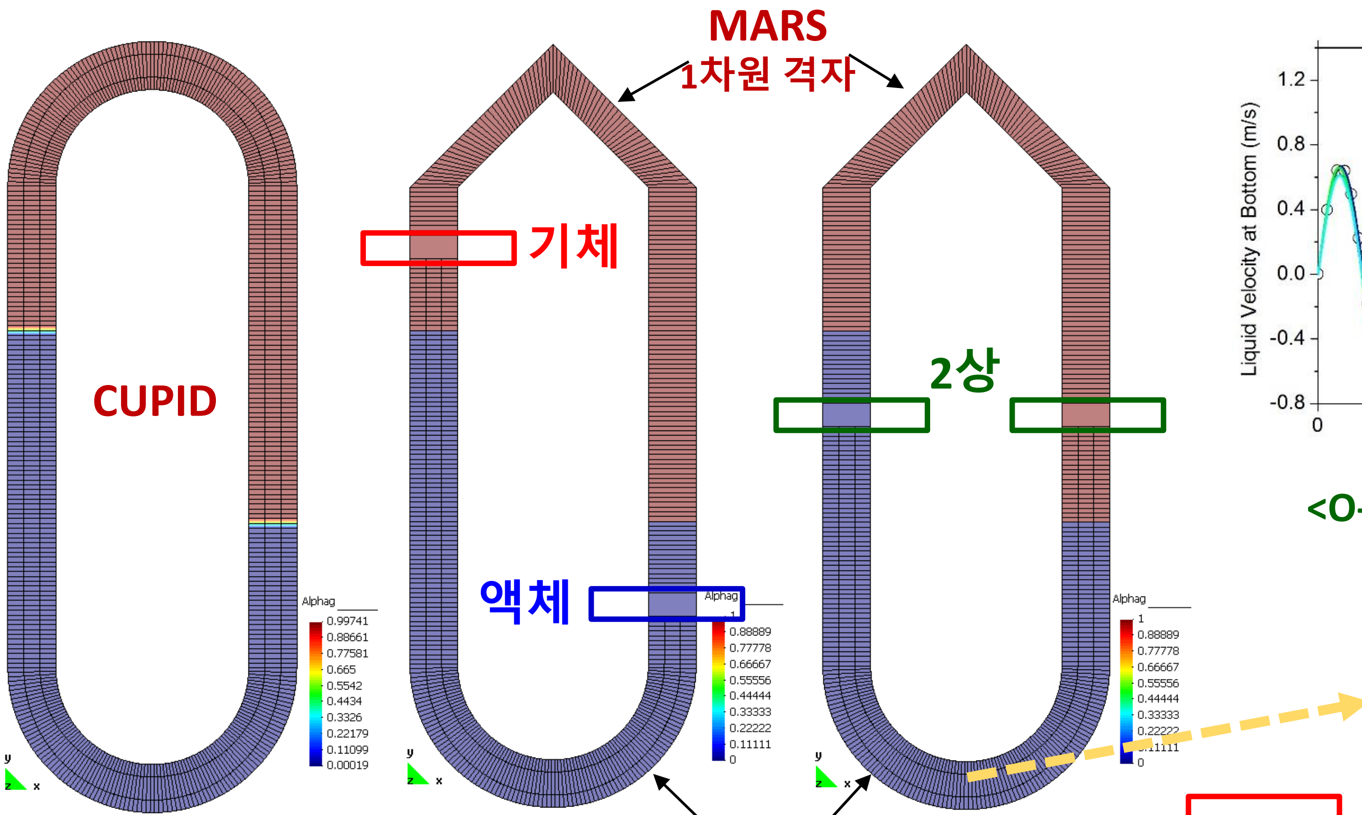
A,B,C,D,E에서 유량



A,B,C,D,E에서 압력

다중스케일 연계 검증 계산 (3/3)

O-튜브 내부 액체 진동

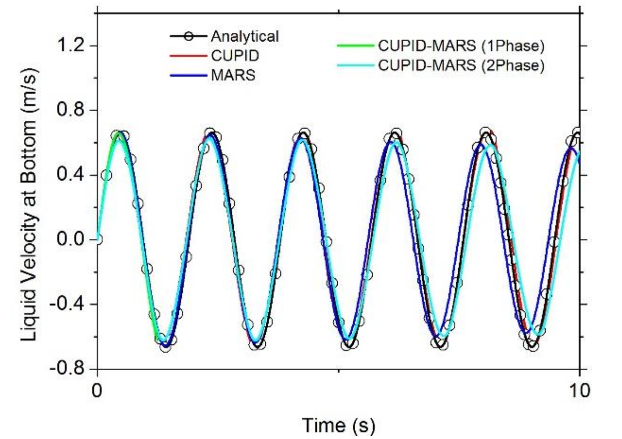


CUPID 단독계산
2차원 격자

MARS/CUPID
단상 연계

MARS/CUPID
2상 연계

기체 연계 격자
액체 연계 격자
2 상 연계 격자



<O-튜브 바닥 액체 속도 비교>

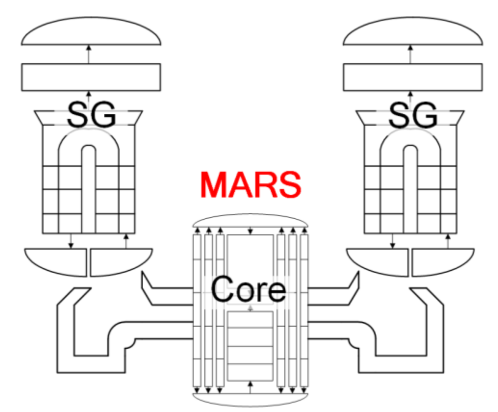
통합 해석 플랫폼 MARU

4

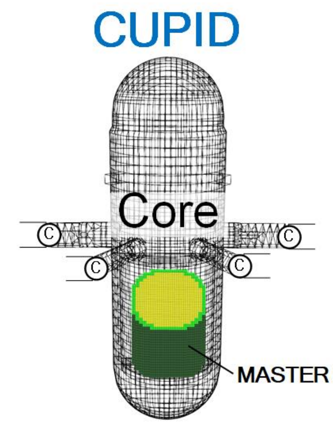
– 원자로 통합 안전해석 계산 절차

– MARU 플랫폼 소개

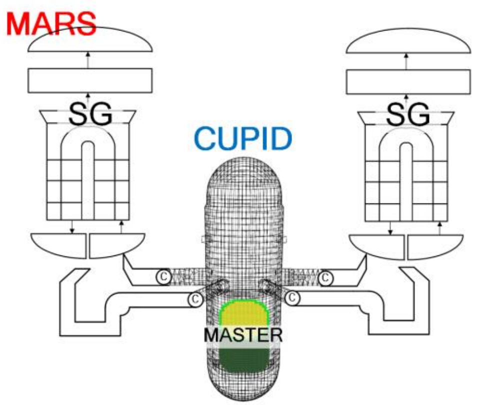
원자로 통합 안전해석 절차 검증 (1/3)



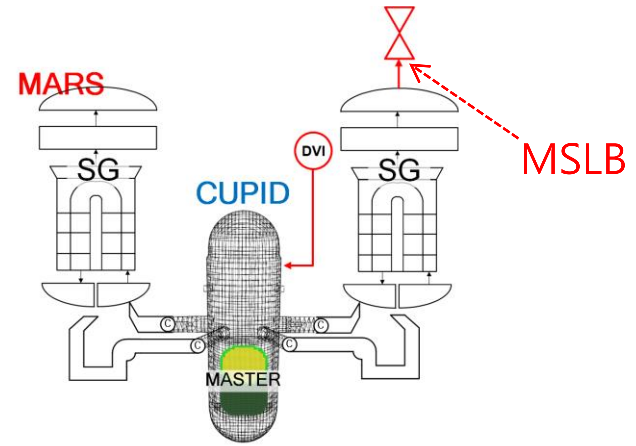
① 1D Reactor System *Steady*



② 3D RPV *Steady*

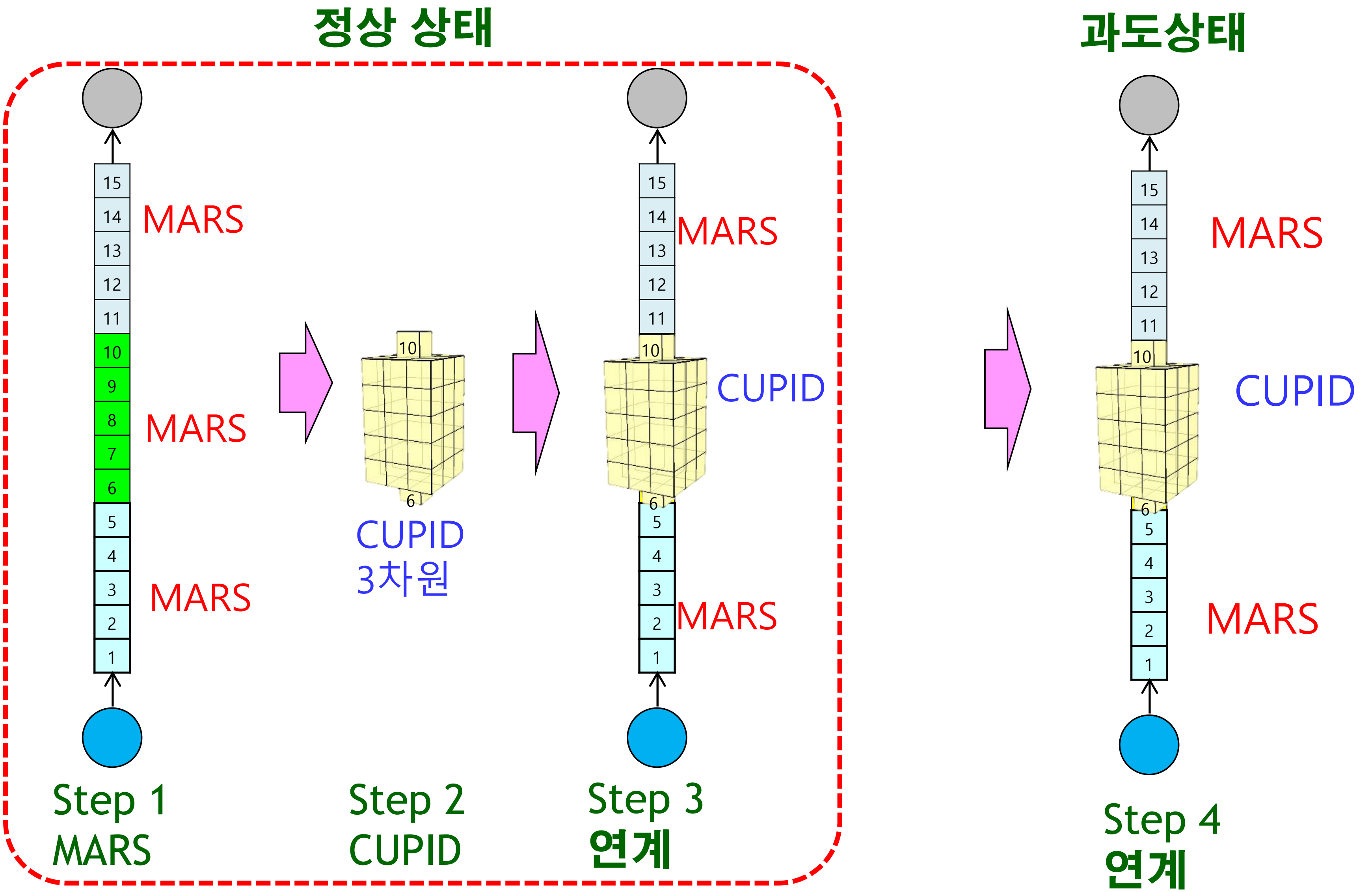


③ 1D/3D Coupled *Steady*



④ 1D/3D Coupled *Transient*

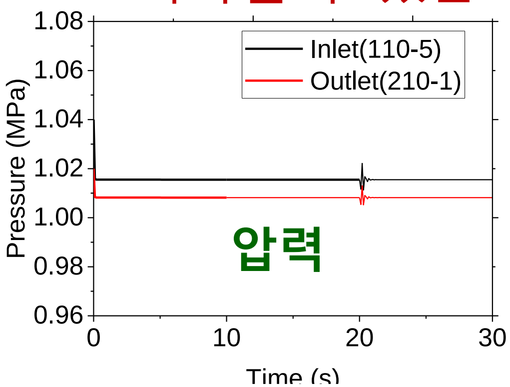
원자로 통합 안전해석 절차 검증 (2/3)

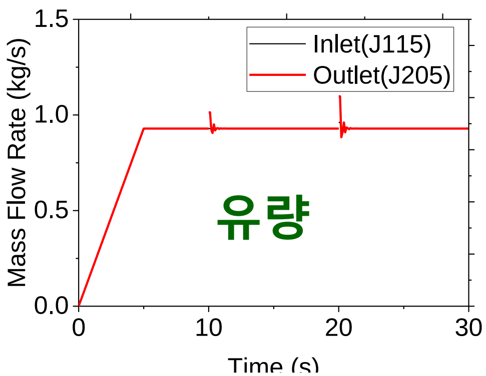


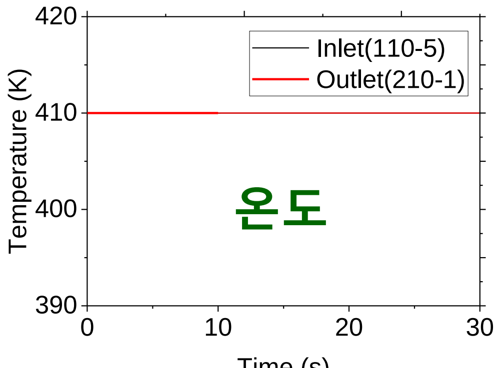
원자로 통합 안전해석 절차 검증 (3/3)

연계 안전해석 절차 검증 계산 결과

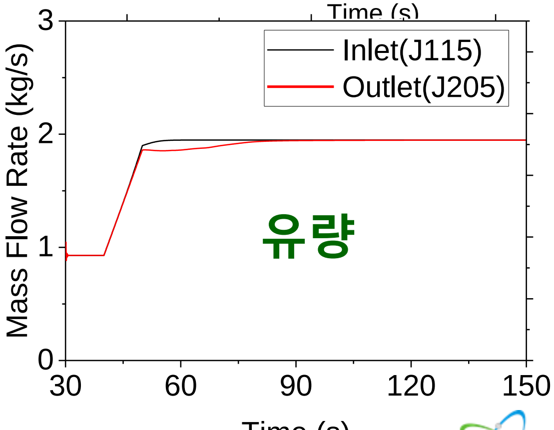
- 정상상태 계산 (1,2,3 단계: 10s,20s,30s)에서 압력, 유량, 온도가 일관된 결과를 보여줌.
  - 1차원 및 3차원 단독 정상상태를 연계하여 연계 정상 상태를 빠르게 획득할 수 있음 .







- 과도상태 계산(4 단계) 은 3단계 연계 정상 상태를 기반으로 수행함.
  - 연계 원자로 안전해석이 제시된 절차를 사용하여 수행할 수 있음.



원자로 통합 안전해석 플랫폼 MARU (1)

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)

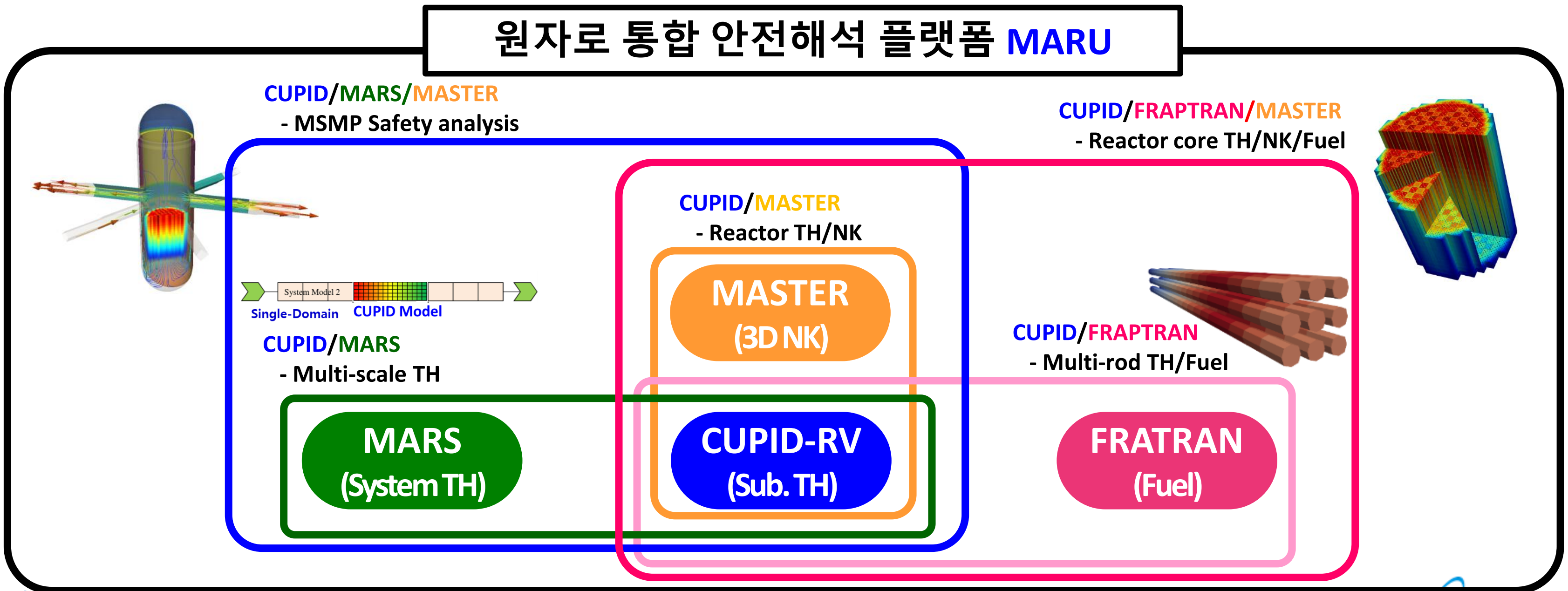
» 통합 안전해석 플랫폼 개발 목표

➢ 개별적 연계 계산 한계 극복

- 개별적 코드 연계부터
- 다물리 다중스케일 통합해석 솔루션 제공

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

➢ 3차원 다물리 다중스케일 안전해석을 위한 플랫폼



원자로 통합 안전해석 플랫폼 MARU (2)

통합해석 플랫폼 사용자 인터페이스

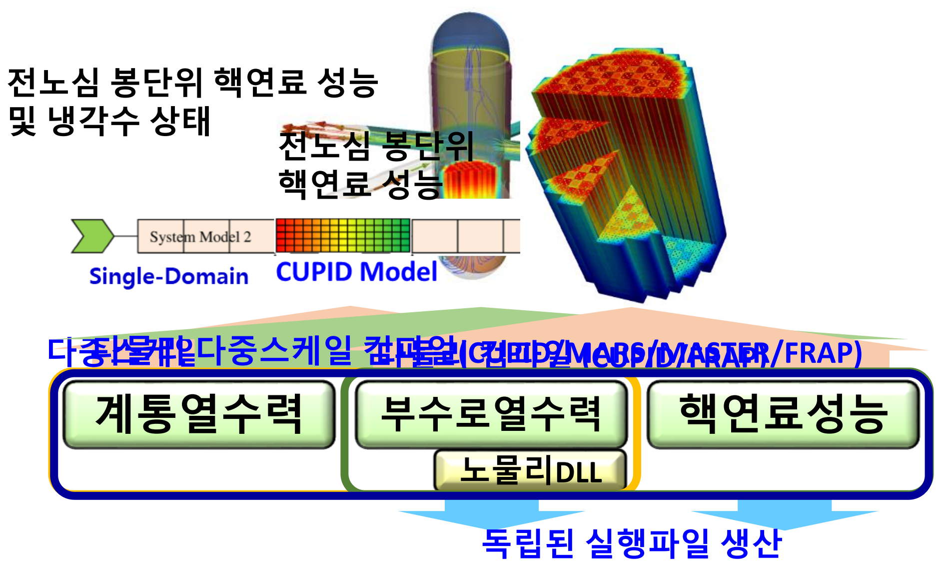
TCP/IP 소켓 통신

- 서버 (리눅스) ←→ 클라이언트 (윈도우즈)

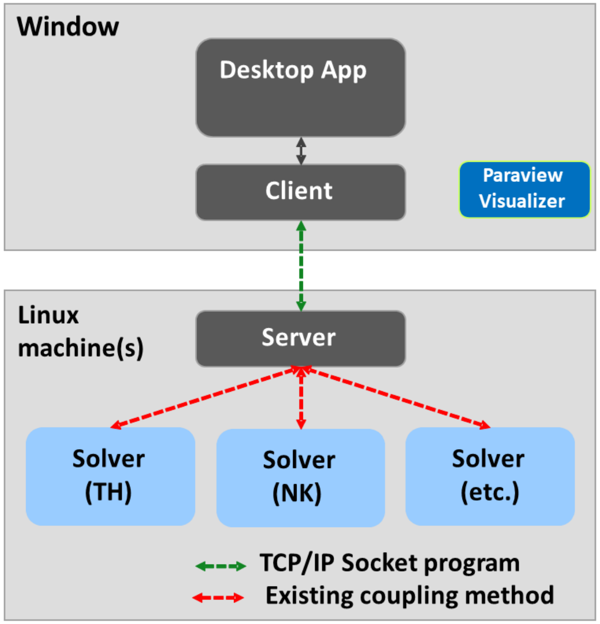
사용자 필요에 따라 소스레벨 컴파일 가능

- 열수력과 핵연료는 동등한 소스레벨 연계
  - ✓ 노물리코드는 DLL 방식

소스 레벨 ➔



MARU 활용 개념도



요약

5

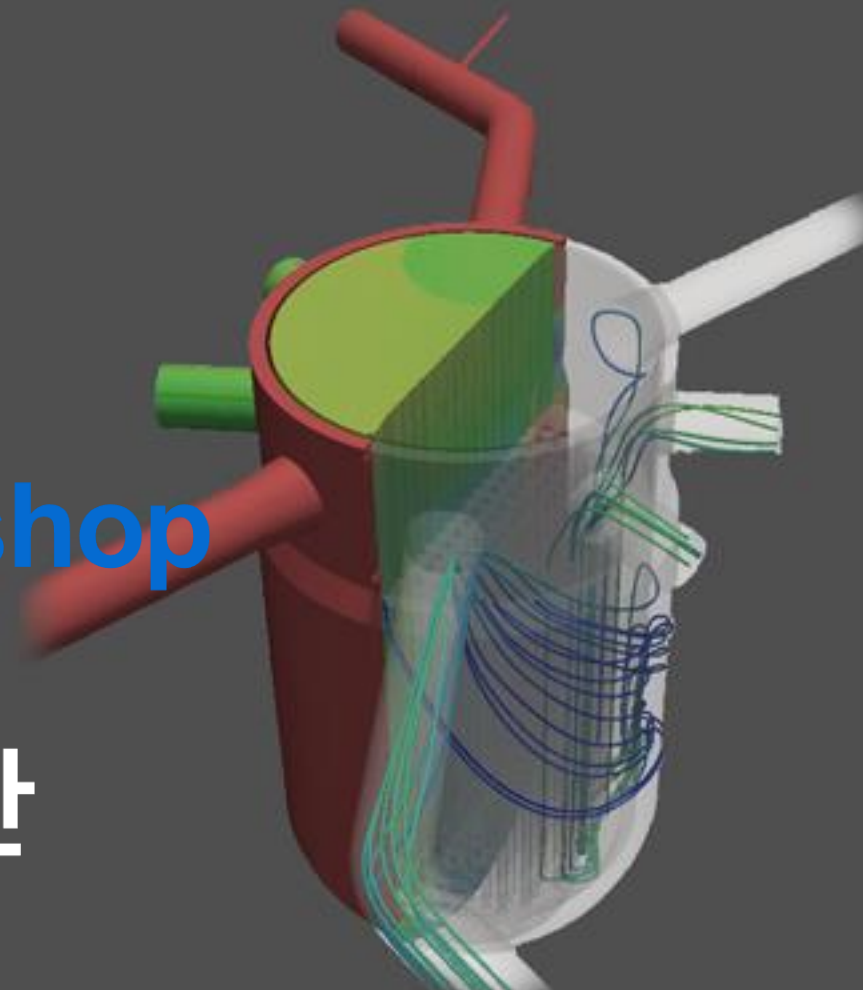
요약

- 노물리 코드 와 핵연료 코드 를 격자 맵핑 기법을 사용하여 CUPID 코드에 다물리 연계함.
  - 노물리 코드: MASTER, DeCART, nTER
  - 핵연료 코드: FRAPTRAN, MERCURY
- 계통 해석 코드와 다물리 연계 3차원 원자로용기 해석 코드를 단일 영역 다중스케일 연계함.
  - 계통해석 코드: MARS-KS, SPACE
  - 데이터의 전송이나 솔버의 반복계산이 불필요함
  - 빠르고 안정적인 과도상태 계산이 가능함.
- 원자로 통합 안전해석 플랫폼 MARU 개발
  - MARU (Multi-physics Analysis Platform for Nuclear Reactor SimUlation)
  - 물리현상별 정밀도별 기관별 다양한 코드 탑재 가능
  - 대용량 계산이 가능한 리눅스기반 클러스터 PC에 설치
  - 윈도우즈 기반 사용자 인터페이스 제공

THANK YOU

gosu@kaeri.re.kr

Korea Atomic Energy Research Institute
KAERI
Since 1959
한국원자력연구원



10th CUPID Workshop

MARU 가동원전 안전해석 적용 계산 (연계검증, SLB)

이 재 룡

2022년 8월 23일

CUPID Workshop

CONTENTS

WHY 3D Safety Analysis ?

1

- 3D Safety Analysis Issues
- High-Fidelity Safety Analysis
  고신뢰도 정밀안전해석
- 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

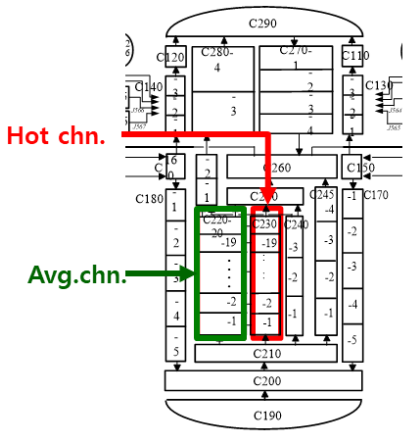
1D nodalization

- 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**

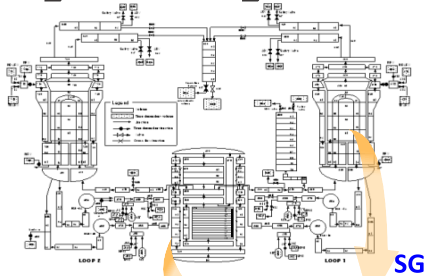


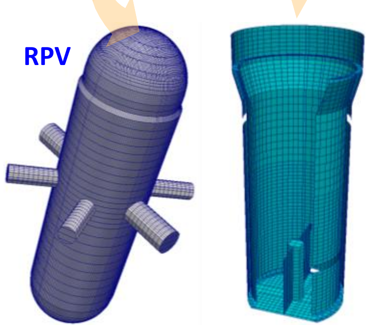
System TH nodal for PWR

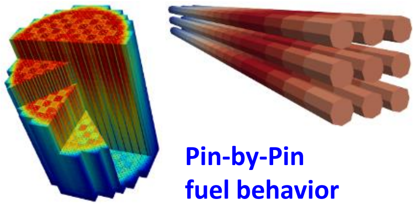
High-Fidelity Safety Analysis 고신뢰도 정밀안전해석

Technologies(기능요구사항) for High-Fidelity Safety Analysis

- **Safety analysis considering *entire RCS***
  - RCS modeling
  - MultiD components as well as OneD nodalization
  - High-resolved components such as RPV and SG
- **3D calculation capability in RPV**
  - High-resolved but practical assessment
  - Fullcore visualization
- **Fuel performance calculation capability**
  - Fullcore-scaled fuel behavior including power distribution
  - Realistic fuel behavior during transient







➔ Multi-Scale and Multi-Physics approach

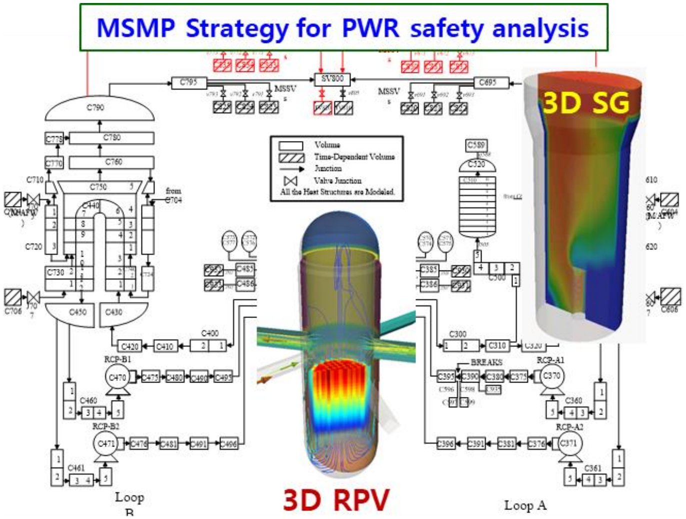
Multi-Scale & Multi-Physics (MSMP) approach

Multi-Scale T/H

- ***3D (subchannel T/H)*** **resolution for region of interest**
  - Reactor pressure vessel(RPV), steam generator (SG)
  - 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)

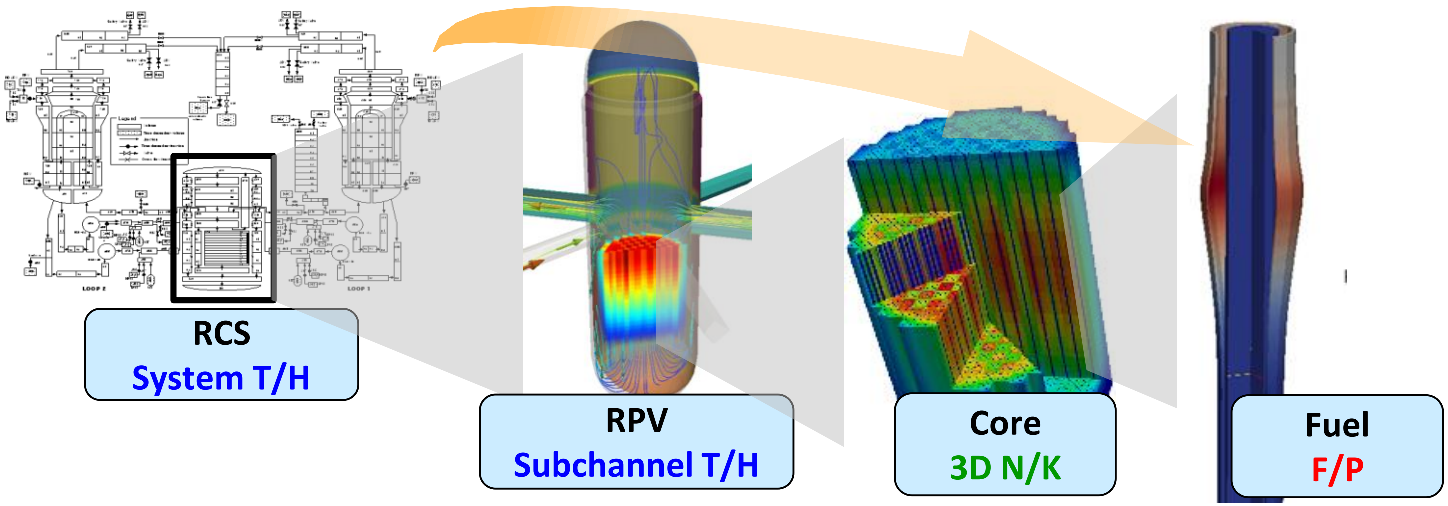
- ***Pin-wise fuel behavior***
  - 3D power distribution
    - ✓ **Neutron kinetics** (N/K) code
  - Realistic fuel rod status
    - ✓ **Fuel performance** (F/P) code



Multi-Scale & Multi-Physics Strategy in MARU

MSMP Simulation Scope in MARU

- 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)

Full core Pin-wise Fuel Performance

2

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

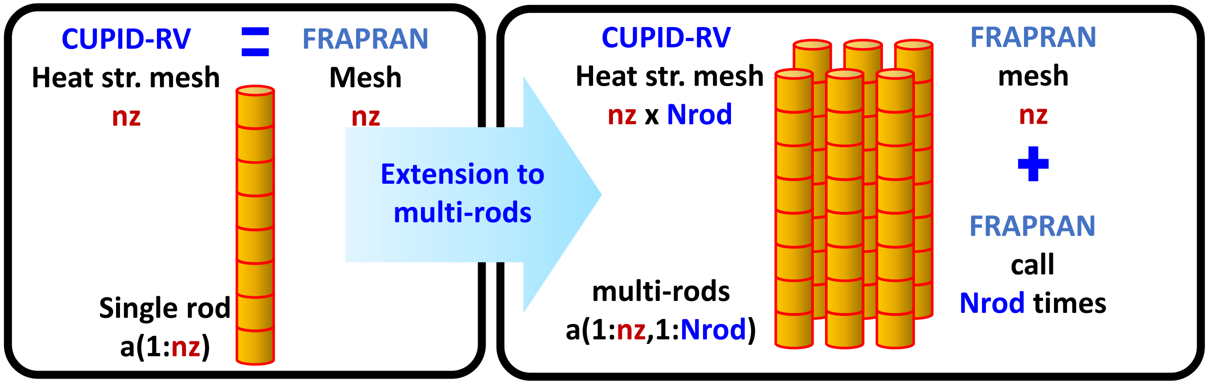
Pin-wise Fuel Performance code Coupling (1/9)

Fuel Performance Code

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

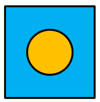
How to Couple for Source Level *Multiple* Fuels

- **Extend fuel code for multiple fuels**
  - Extend coupling variables for multi-rods
  - Call fuel code as many as the number of fuels



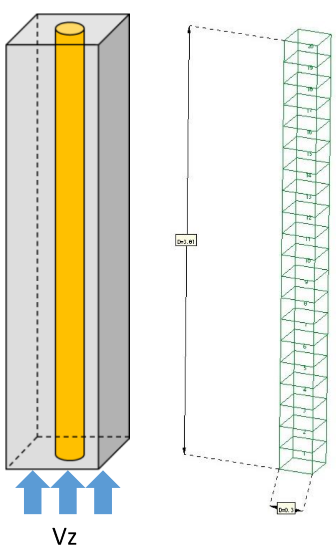
Pin-wise Fuel Performance code Coupling (2/9)

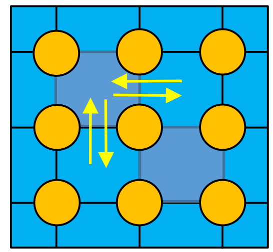
Single vs Multiple

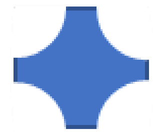


Single fuel rod

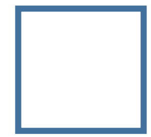
- Cell-to-cell (1:1) mapping
- SPACE-FRAP, MARS-FRAP







Real Subchannel geometry



CUPID computing cell with
- subchannel type
- Porosity
- Hydraulic diameter
- Gap distance
- cell-to-cell pitch

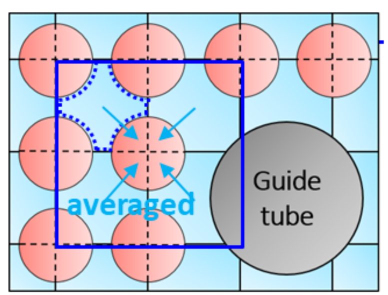
➔ Geometric input

Multiple fuel rods

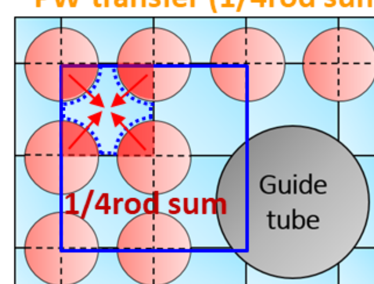
- Subchannel TH mesh (CUPID-FRAP)
- Physical model in subchannel resolution
  - Pressure drop
  - Turbulent mixing
  - Spacer grid

Information between TH subchannel and fuel rod

TH transfer (averaged)



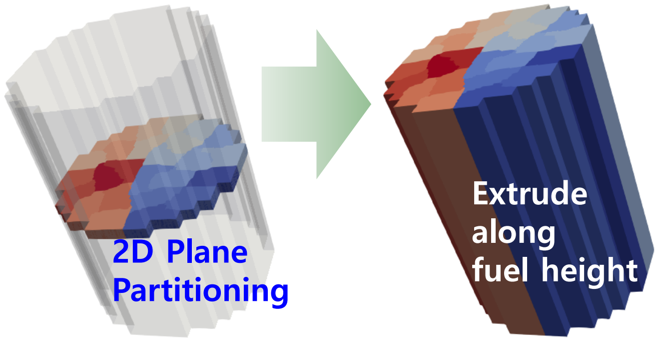
PW transfer (1/4rod sum)



Pin-wise Fuel Performance code Coupling (3/9)

Parallel for Pin-wise Full Core Simulation

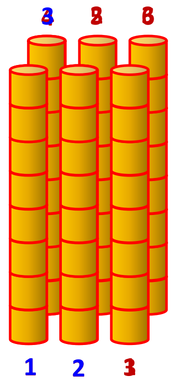
- ***Prerequisite: Single fuel is not partitioned***
- **Domain partitioning by METIS**
  - Partitioned 2D plane
  - Extrude along fuel height



Local Index

Single fuel a(1:nz) → **Multi fuels** a(1:nz,1:nr)

	nz	nr
Before partitioning	20	6
After partitioning	20	3 subdomain1 – 3 rods, subdomain2 – 3 rods,

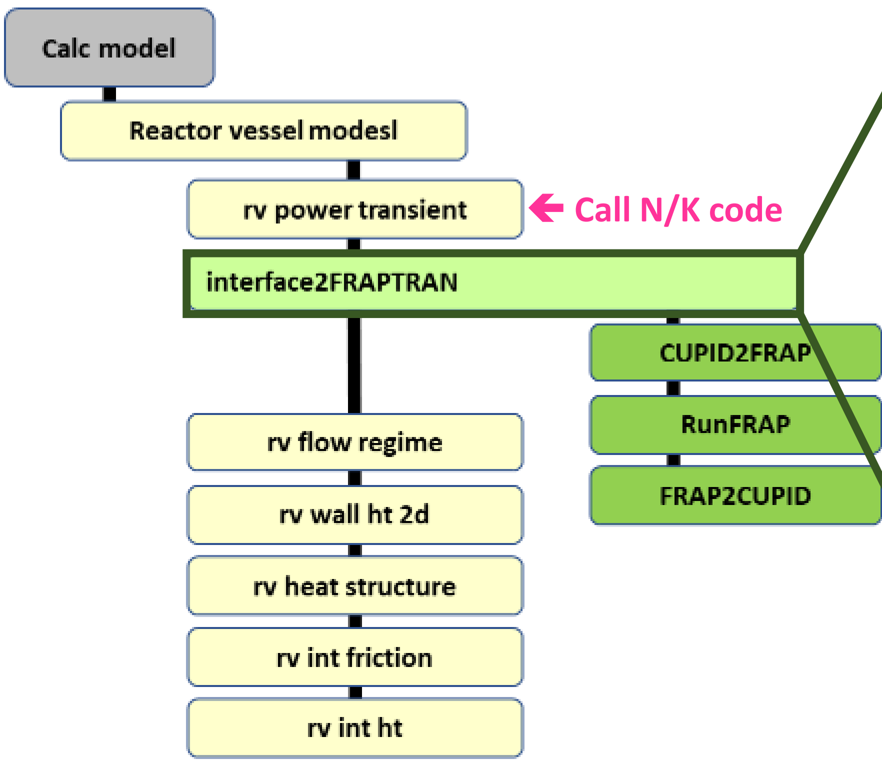


영역분할 전 Rod index
1~6 (전체 수)
- fuel variables: a(1:20, 1:6)

영역분할 후 Rod index
1~3 (Domain 1)
- D1 fuel variables: a(1:20, 1:3)
1~3 (Domain 2)
- D2 fuel variables: a(1:20, 1:3)

Pin-wise Fuel Performance code Coupling (4/9)

Call FRAPTRAN in CUPID



interface2FRAPTRAN.f90 | calc_scalar.f90 | reactor_vessel_model

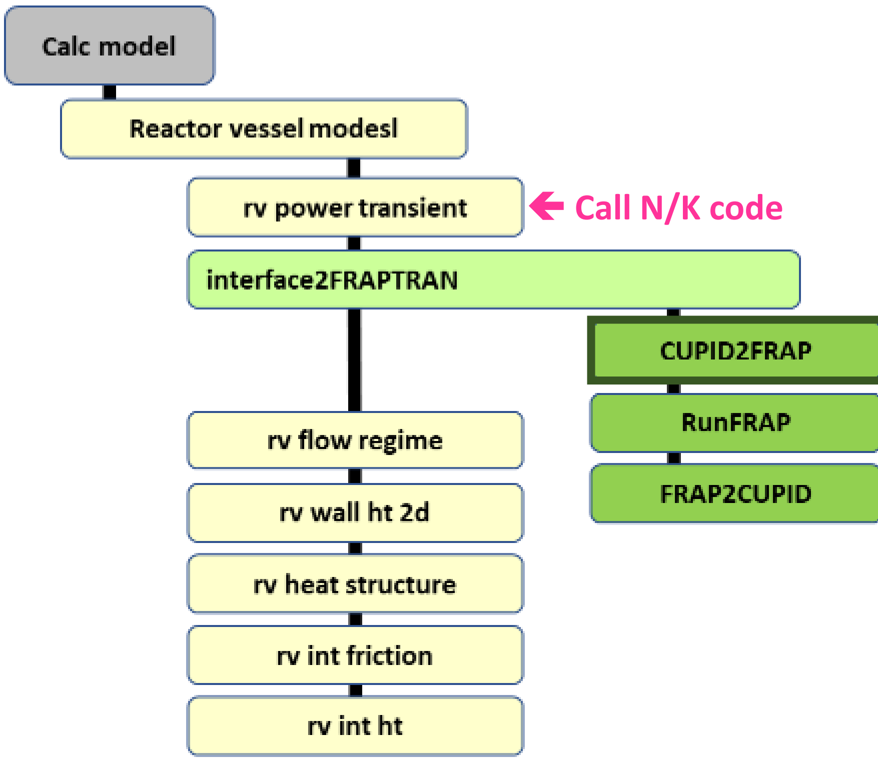
(Global Scope)

```fortran
!
!.....Compare problem time between CUPID and FRAPTRAN
!
      IF(time.ge.time_frap)then
         run_frap=.true.
      ELSE
         run_frap=.false.
      ENDIF
!
!.....FRAPTRAN call
!
      IF(run_frap)THEN
!
!........Sent to FRAPTRAN
!
         CALL CUPID2FRAPTRAN(k_nqvol,punit_char2)
!
!........Run FRAPTRAN
!
         DO WHILE(1)
            frapcall=frapcall+1
            IF(init_frp) THEN
               init_frp=.false.
               CALL RunFRAPTRAN_initial
            ELSE
               CALL RunFRAPTRAN
            ENDIF
            time_frap=time_frap+dt_frap
            IF(time_frap.gt.time)EXIT
         ENDDO
!
!........Receive from FRAPTRAN
!
         CALL FRAPTRAN2CUPID
      ENDIF
!
      RETURN
      END SUBROUTINE interface2FRAPTRAN
!
```

Pin-wise Fuel Performance code Coupling (5/9)

» Call FRAPTRAN in CUPID

CUPID→ FRAP 변수	단위	배열
Pressure	Pa	(nz, nr)
Linear power	W/m	(nz, nr)
Heat flux	W/m2	(nz, nr)
Heat transfer coef.	W/m2K	(nz, nr)
Coolant temp	K	(nz, nr)
Max linear power	W/m	(nr)
dt	sec	-



CUPID2FRAPTRAN.f90* | interface2FRAPTRAN.f90 | calc_scalar.f90 | reactor_vessel_models_fraptran.f90

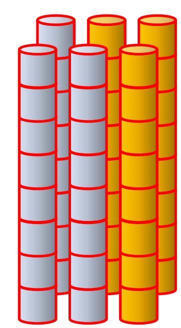
(Global Scope) | CUPID2FRAPTRAN(k, punit_char2)

```
DO kk=1,ncell_fuel_rod
   ir1=nrod_fuel_rod(kk)
   ir=jjperm_rod(ir1)
   iz=nz_fuel_rod(kk)
```

주요 index	설명	
ncell_fuel_rod	전체 열구조체 격자 수 (CUPID)	nz x nr
nrod_fuel_rod	Rod index (영역분할전, Global)	예. 1~6
jjperm_rod	Rod index (영역분할 후, Local)	예. 1~3
nz_fule_rod	축방향 index	예. 1~20

Multi fuels

a(1:nz,1:nr)

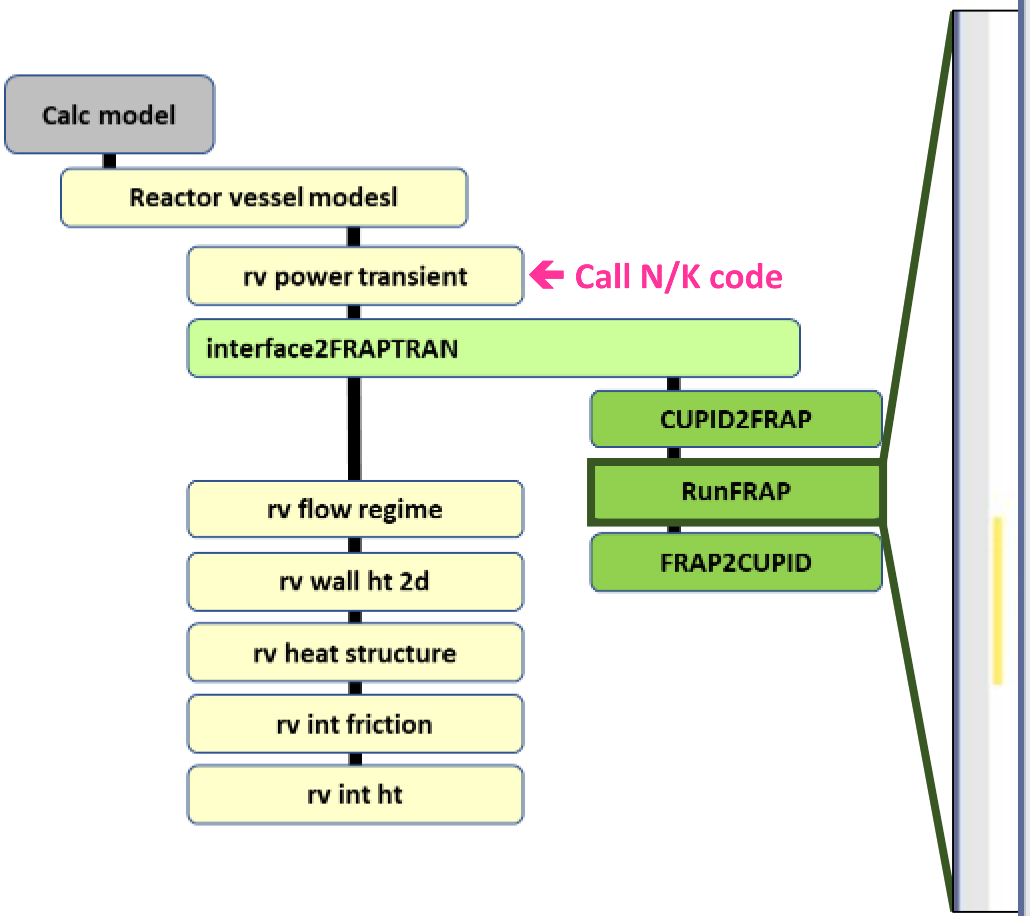


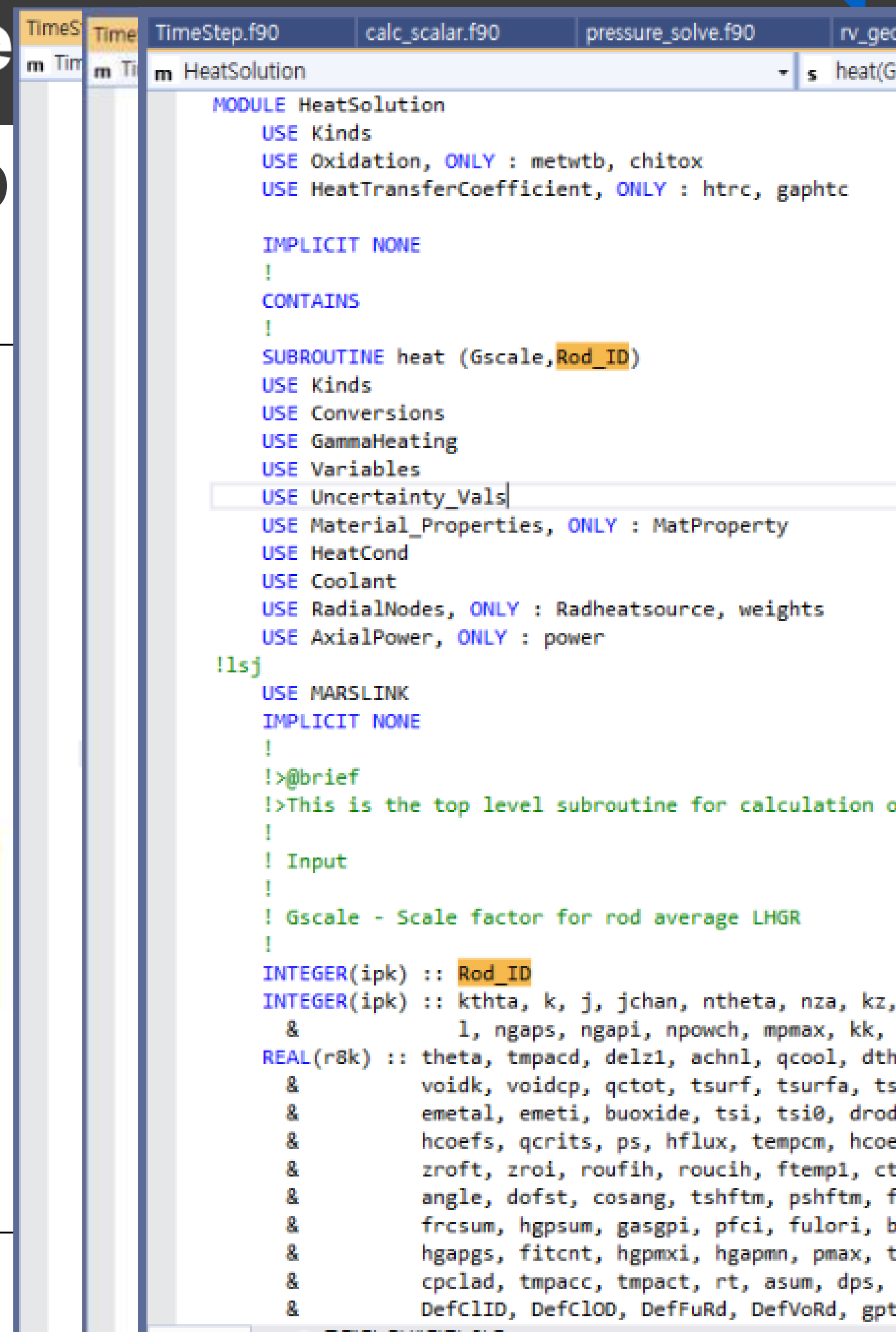
영역분할 후 Rod index
1~3 (영역1)
1~3 (영역2)

영역분할 전 Rod index
1~6 (전체 수)

Pin-wise Fuel Performance

» Call FRAPTRAN in CUPID

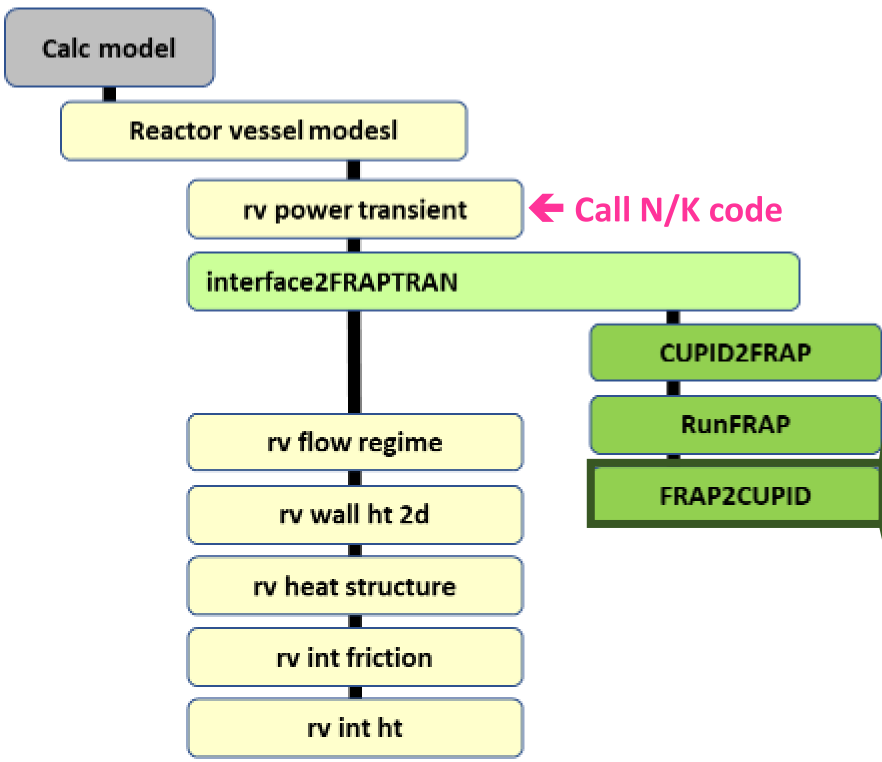





```
MODULE HeatSolution
    USE Kinds
    USE Oxidation, ONLY : metwtb, chitox
    USE HeatTransferCoefficient, ONLY : htrc, gaphtc

    IMPLICIT NONE
    !
    CONTAINS
    !
    SUBROUTINE heat (Gscale,Rod_ID)
    USE Kinds
    USE Conversions
    USE GammaHeating
    USE Variables
    USE Uncertainty_Vals
    USE Material_Properties, ONLY : MatProperty
    USE HeatCond
    USE Coolant
    USE RadialNodes, ONLY : Radheatsource, weights
    USE AxialPower, ONLY : power
!lsj
    USE MARSLINK
    IMPLICIT NONE
    !
    !>@brief
    !>This is the top level subroutine for calculation o
    !
    ! Input
    !
    ! Gscale - Scale factor for rod average LHGR
    !
    INTEGER(ipk) :: Rod_ID
    INTEGER(ipk) :: kthta, k, j, jchan, ntheta, nza, kz,
      &             l, ngaps, ngapi, npowch, mpmax, kk,
    REAL(r8k) :: theta, tmpacd, delz1, achnl, qcool, dth
      &          voidk, voidcp, qctot, tsurf, tsurfa, ts
      &          emetal, emeti, buoxide, tsi, tsi0, drod
      &          hcoefs, qcrits, ps, hflux, tempcm, hcoe
      &          zroft, zroi, roufih, roucih, ftemp1, ct
      &          angle, dofst, cosang, tshftm, pshftm, f
      &          frcsum, hgpsum, gasgpi, pfci, fulori, b
      &          hgapgs, fitcnt, hgpmxi, hgapmn, pmax, t
      &          cpclad, tmpacc, tmpact, rt, asum, dps,
      &          DefClID, DefClOD, DefFuRd, DefVoRd, gpt
```

Pin-wise Fuel Performance code Coupling (7/9)

Call FRAPTRAN in CUPID



FRAPTRAN2CUPID.f90 | TimeStep.f90 | calc_scalar.f90 | pressure_solve.f90 | rv_geo_hts_2

(Global Scope) | FRAPTRAN2CUPID()

```
!
!.....CUPID-FRAPTRAN Coupling Interface
!
      CALL toCUPID(nn,heatflux_F2C    , cladtemp_F2C   , fuelradius_F2C  , &
                   Fuelenthalpy_F2C, HeatTemp_s_F2C , H2GEN_F2C       , &
                   FuelAveTemp_F2C , CladAveTemp_F2C, ECR_F2C         , &
                   HGapAv_F2C      , GapThick_F2C   , HopStrn_F2C     , &
                   Oxit_F2C        , Oxot_F2C       , PelletRadius_F2C, &
                   CladdiRadius_F2C, CldStress_F2C  , RIP_F2C         , &
                   RPT_F2C         , BURSTnode_F2C  , Gappress_F2C      )
!
!.....Copy Fraptran data to CUPID
!
      i=0
      DO Rod_ID=1,N_Rods
         qvol_sum_frap=0.d0
         DO ii=1,nz
            i =i+1

            qflux_F2C(ii,Rod_ID)=heatflux_F2C(i)
            twall_F2C(ii,Rod_ID)=cladtemp_F2C(i)
            tcent_F2C(ii,Rod_ID)=HeatTemp_s_F2C(i) !Fuel center temp (FOR MASTER Coup
            fradi_F2C(ii,Rod_id)=fuelradius_F2C(i)
            gpres_F2C(ii,Rod_id)=Gappress_F2C(i)
            gthik_F2C(ii,Rod_id)=GapThick_F2C(i)

            ht_area_F2C(ii,Rod_ID)=2.d0*pi*fuelradius_F2C(i)*dz_fuel0(ii)
            qvol_sum_frap=qvol_sum_frap+qflux_F2C(ii,Rod_ID)*ht_area_F2C(ii,Rod_ID)


         ENDDO
      ENDDO
!
      RETURN
      END SUBROUTINE FRAPTRAN2CUPID
!
```

Pin-wise Fuel Performance code Coupling (8/9)

Parallel Computing in HPC Environment

➢ SPMD (Single Program, Multiple Data)

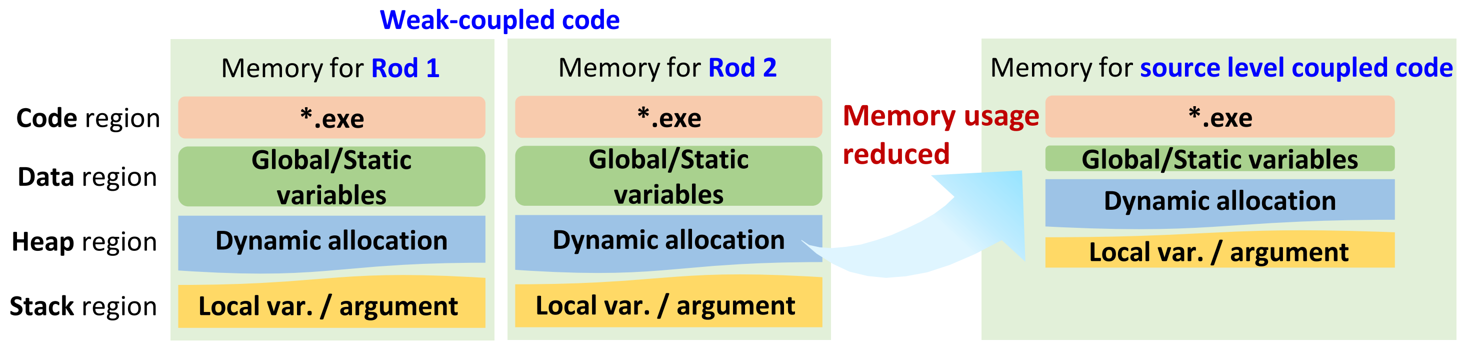
- Source-to-source compilation

용도	명령어	실행파일
CUPID 단독	make –f makefile_CUPID	cupid.x
CUPID/MARS	make –f makefile_MARS	cupid_MARS.x
CUPID/FRAPTRAN	make –f makefile_CFRAPTRAN	cupid_CFRAPTRAN.x
CUPID/MARS/FRAPTRAN	make –f makefile_CUPID_MARS_CFRAPTRAN	cupid_MARS_CFRAPTRAN.x

```
(base) [ali@eddy_e001 cupidmarsfraptranl]$ ll
total 86628
-rwxrwxr-x  1 ali ali       89 Apr  5 14:13 1.rm
drwxrwxr-x  4 ali ali       46 Feb  9 18:37 CFRAPTRAN
-rw-rw-r--  1 ali ali      163 Feb  9 18:37 cfraptran_dir0.in
-rw-rw-r--  1 ali ali      190 Feb  9 18:37 cfraptran_dir.in
-rwxrwxr-x  1 ali ali     2693 Feb  9 18:37 chmod_x.sh
drwxrwxr-x 17 ali ali     4096 Feb  9 18:37 CUPID_4244
-rwxrwxr-x  1 ali ali 22846688 Apr  6 13:43 cupid_CFRAPTRAN.x
-rw-rw-r--  1 ali ali     2803 Feb  9 18:37 cupid_dir.in
-rwxrwxr-x  1 ali ali 32760792 Apr  5 14:09 cupid_MARS_CFRAPTRAN.x
-rwxrwxr-x  1 ali ali 32456416 Mar 11 15:54 cupid_MARS_CFRAPTRAN.x.O
-rw-rw-r--  1 ali ali   580718 Mar 10 15:13 libmpitrace.a
-rw-rw-r--  1 ali ali      478 Feb  9 18:37 makefile_CFRAPTRAN
-rw-rw-r--  1 ali ali     2935 Feb  9 18:37 makefile_CUPID
-rw-rw-r--  1 ali ali      717 Mar  9 17:00 makefile_CUPID_MARS_CFRAPTRAN
-rw-rw-r--  1 ali ali     3756 Apr  6 14:48 makefile.in
-rw-rw-r--  1 ali ali      602 Feb  9 18:37 makefile_MARS
drwxrwxr-x  9 ali ali      156 Feb  9 18:37 MARS_4238
-rw-rw-r--  1 ali ali      161 Feb  9 18:37 mars_dir.in
drwxrwxr-x  2 ali ali       10 Apr  6 14:48 modules
-rw-rw-r--  1 ali ali      333 Mar  9 16:55 outdta
drwxrwxr-x 16 ali ali     4096 Feb  9 18:37 run_vv
(base) [ali@eddy_e001 cupidmarsfraptranl]$
```

makefile 스크립트

- Efficient memory usage for massive multiple fuels calculation.



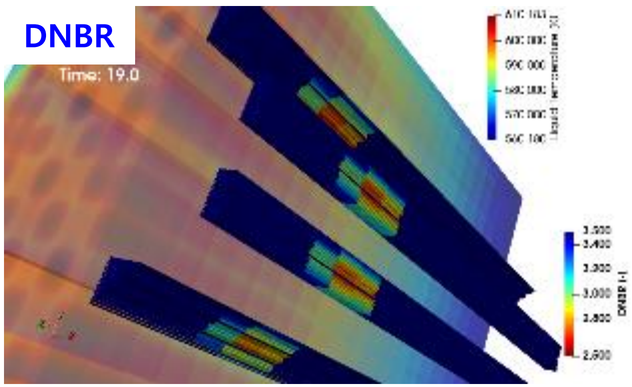
Pin-wise Fuel Performance code Coupling (9/9)

Visualization of fuel calculation results

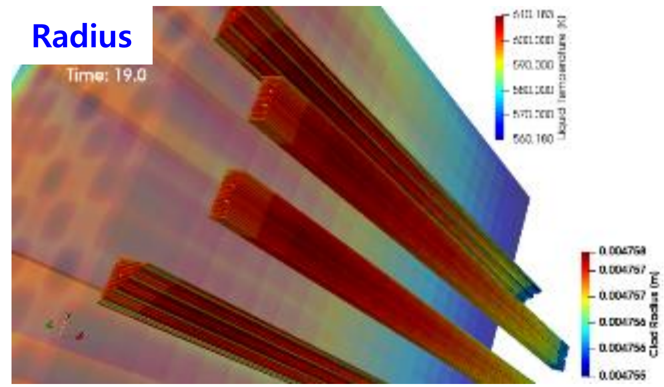
- FRAPTRAN major results
- Binary vtk data generation

vtk (Visualization ToolKit)
: Open source software system for 3D computer graphics, image processing, visualization, etc.

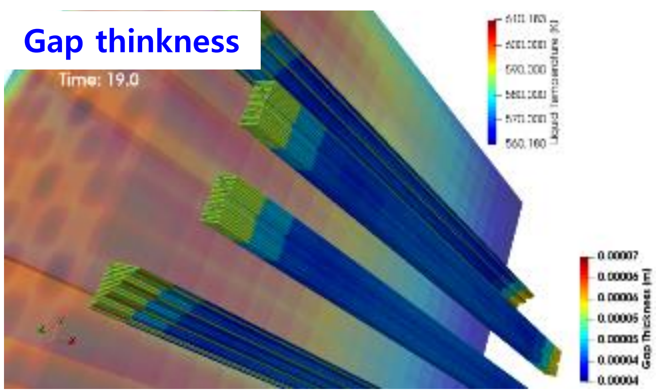
DNBR



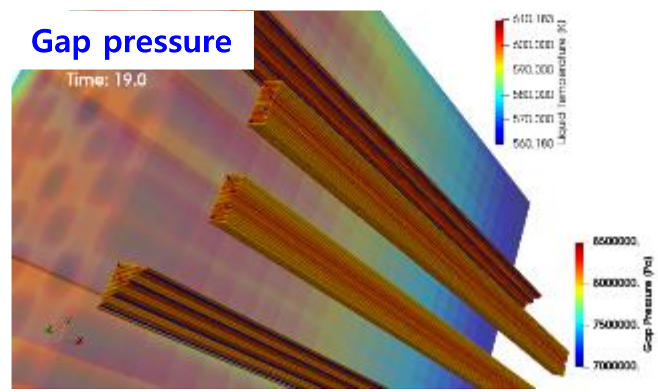
Radius



Gap thinkness



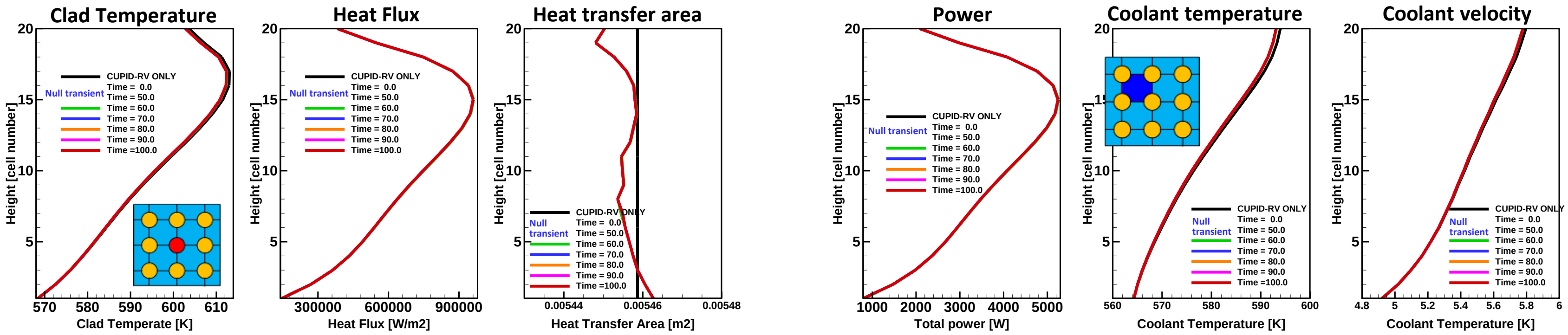
Gap pressure



Evaluation of Pin-wise Fuel Performance (1/3)

Verification of CUPID-RV/FRAPTRAN(1)

➢ LWR 3x3 steady state

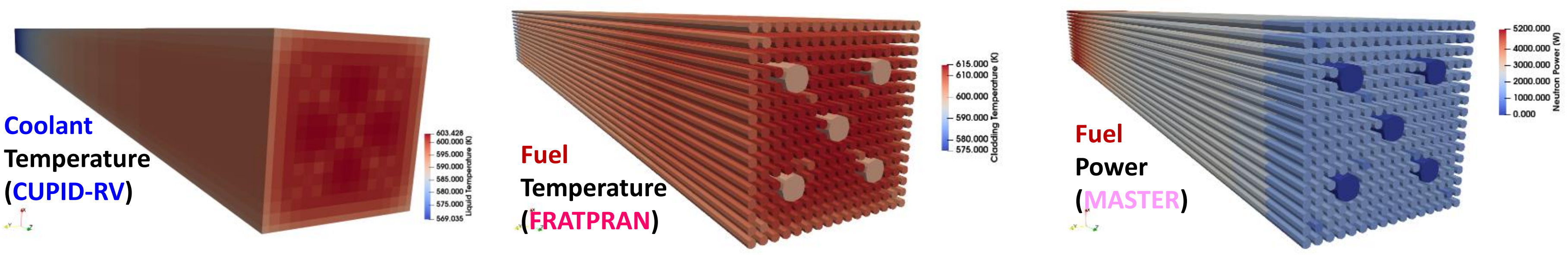


Fuel results by FRAPTRAN

Coolant results by CUPID-RV

➢ LWR Unit Assembly (16x16) steady state

- Fuel power – N/K coupling (MASTER code)



Evaluation of Pin-wise Fuel Performance (2/3)

Performance

➢ Computation performance with CUPID-RV/FRAPTRAN

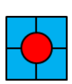
Rod type	Mesh Ratio	CUPID	CUPID/FRAPTRAN (FRAP dt = 0.05)		Ratio
		Total time	Total time	Time of FRAPTRAN	Ratio [%]
Single rod	1	13.08	15.91	2.56 [1.0]	**20**
1 FA (16x16)	72	1274.17	1932.54	666.19 [260.2times against **1x1**]	50
2 FA	2 [.vs. 1FA]	3373	4900.71	1380.35 [2.08times against **1FA**]	45
4 FA	4 [.vs. 1FA]	8012.16	10536.9	2742.81 [4.11times against **1FA**]	**31**
9 FA	9 [.vs. 1FA]	18893.85	24174.62	6066.43 [9.11times against **1FA**]	**28**
16FA	16 [.vs. 1FA]	34732.58	45299.78	11275.77 [16.9times against **1FA**]	**30**

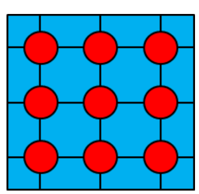
Number of mesh (smaller than 1FA)

➔ Relatively **larger portion** of FRAPTRAN calculation

	CUPID		FRAPTRAN	
	#. Mesh	Mesh Ratio	#. Mesh	Mesh Ratio
Single rod	**4**	-	**1**	-
3x3	16	**4**	9	**9**
16x16	289	**72**	256	**256**

Mesh of Single rod
-CUPID: 4
-FRAP: 1





Mesh of 3x3 rods
-CUPID: 16 [4-times]
-FRAP: 9 [9-times]

Number of mesh (larger than 1FA)

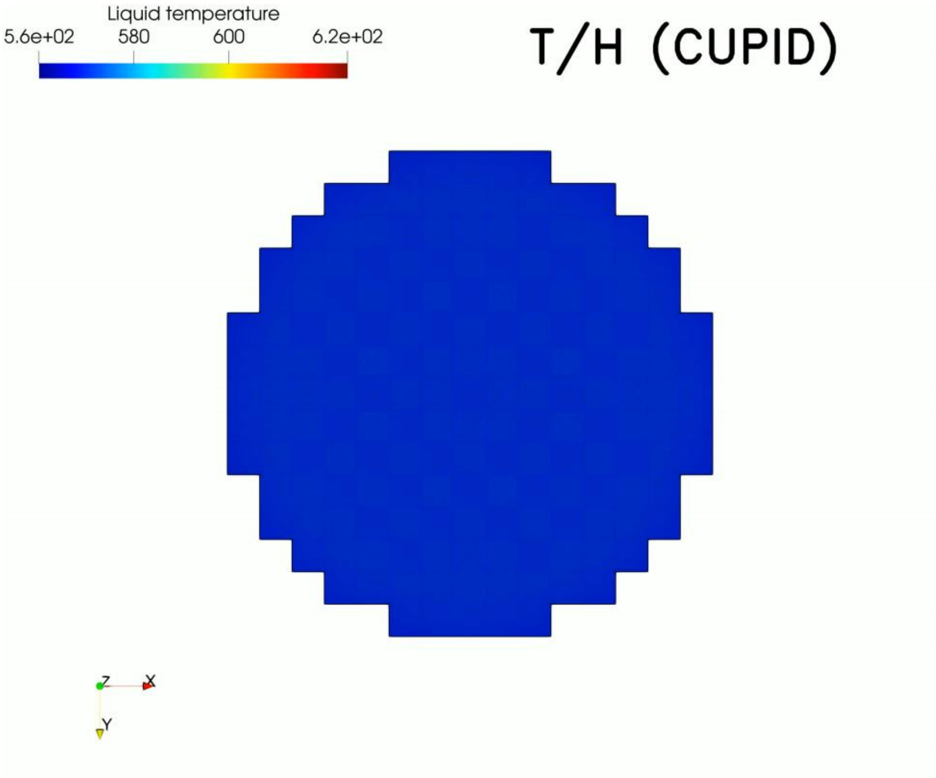
➔ Mesh ratio increases LINEARLY
➔ FRAPTRAN portion is about **30%**
➔ CUPID portion is relatively larger (due to Pressure matrix solving)

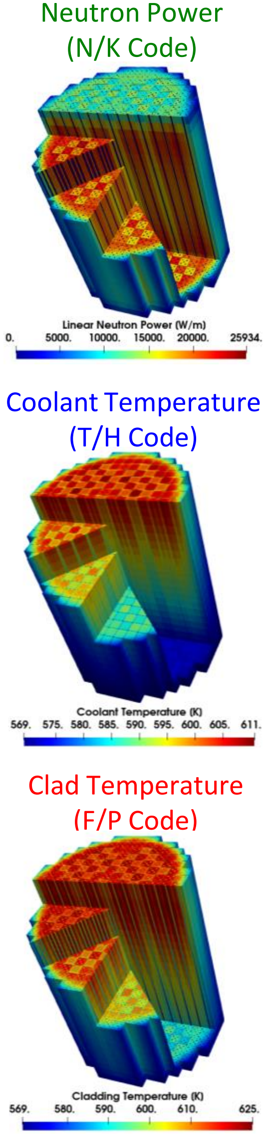
	CUPID		FRAPTRAN	
	#. Mesh	Mesh Ratio	#. Mesh	Mesh Ratio
1FA (16x16)	289	-	**256**	-
2FA	289x2	**2**	256x2	**2**
4FA	289x4	**4**	256x4	**4**
n-FA	289xn	**n**	256xn	**n**

Evaluation of Pin-wise Fuel Performance (3/3)

Verification(2) – LWR Full Core

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





MSMP Safety Analysis of a PWR

3

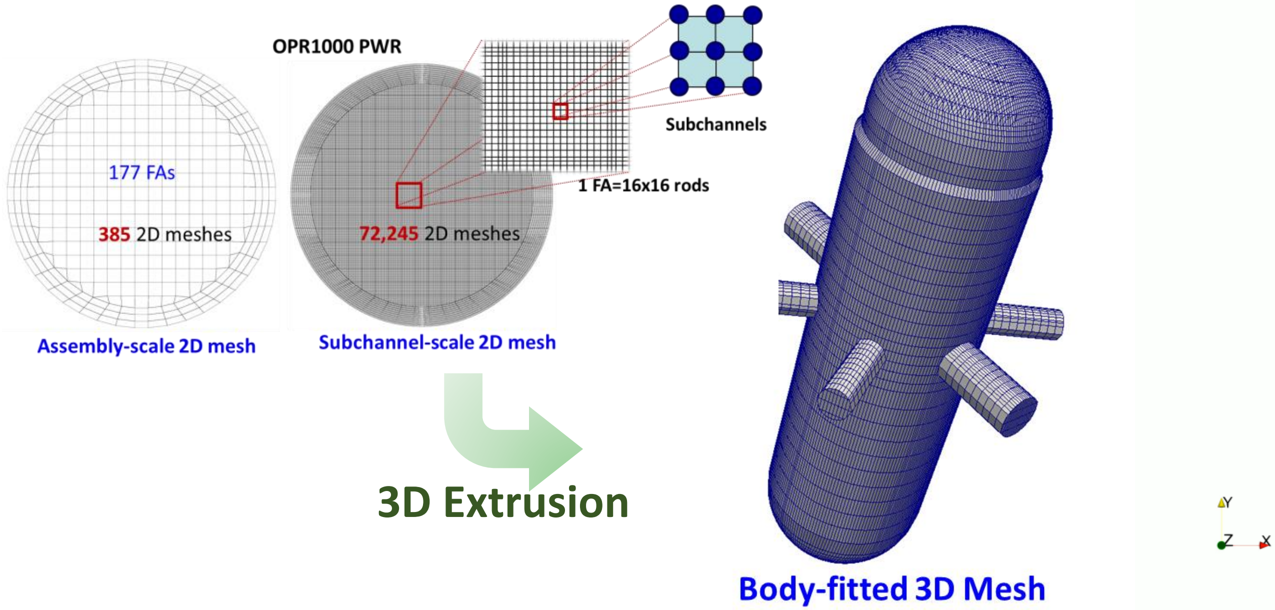
- MSMP Simulation of OPR1000 SLB Accident
- Improvement of Safety Margin (DNBR)

3D Reactor Pressure Vessel Modeling

Subchannel-scale RPV Computational Geometry

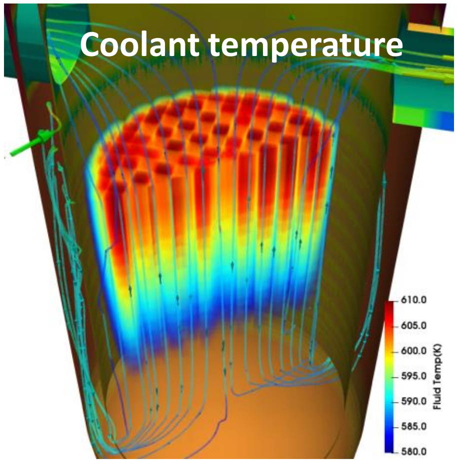
Body-fitted RPV mesh

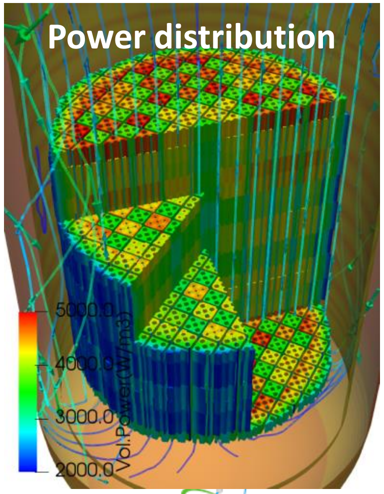
- **In-house** RPV mesh generator (***RVMesh3D***)
- Reactor core, downcomer, upper/lower plenum, and hot/cold leg
- Practical number of meshes (Currently **1.3M**)
- **Subchannel T/H resolution** for core region



MSMP Simulation of OPR1000 SLB Accident

Steady State (End of Cycle Full Power)





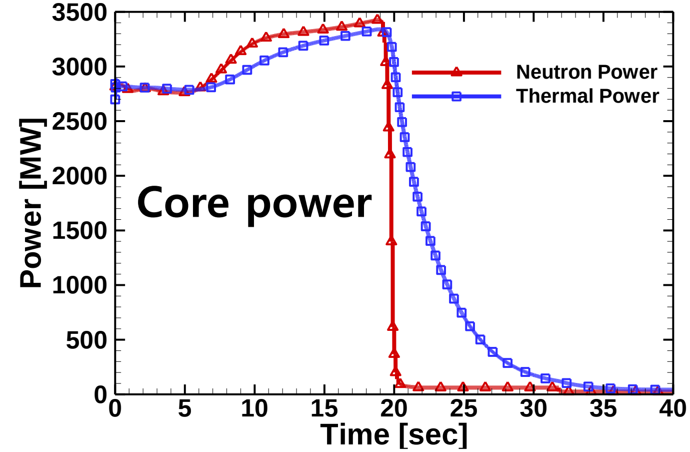
MSMP Simulation of OPR1000 SLB Accident

Sequence of Events and Major Parameters

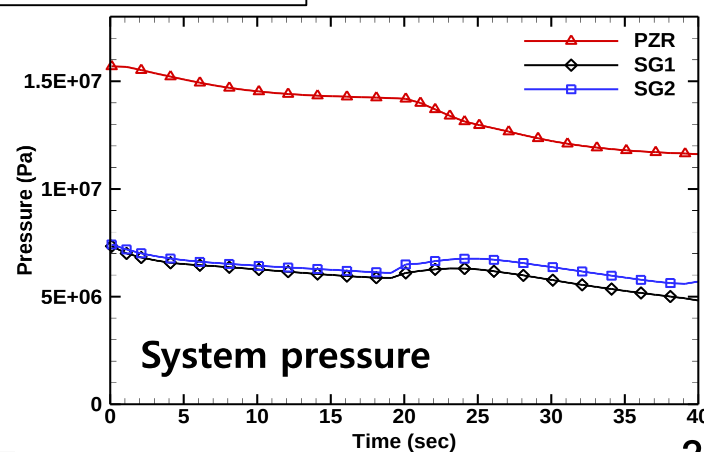
Sequence of events

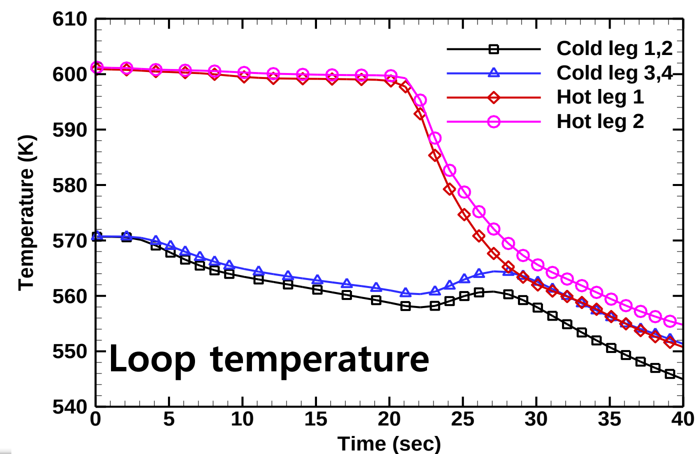
Time(sec)	Event	Setpoint
0.0	Steam line break occurs	
18.4	**Overpower trip setpoint reached**	**121 %**
18.6	Turbine Trip	
19.1	Rod begins to drop	
34.0	Low SG1 setpoint reached	5.44 MPa
35.1	MSIV1 closed	

MSMP result



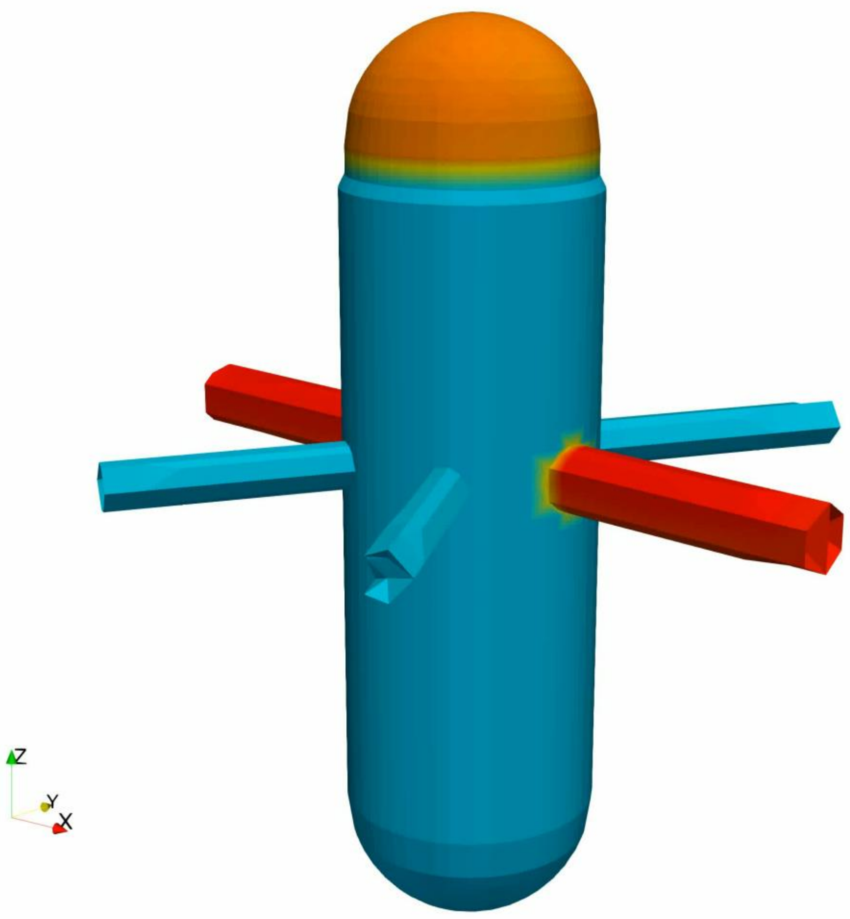
System TH result





MSMP Simulation of OPR1000 SLB Accident

Power & DNBR Distribution

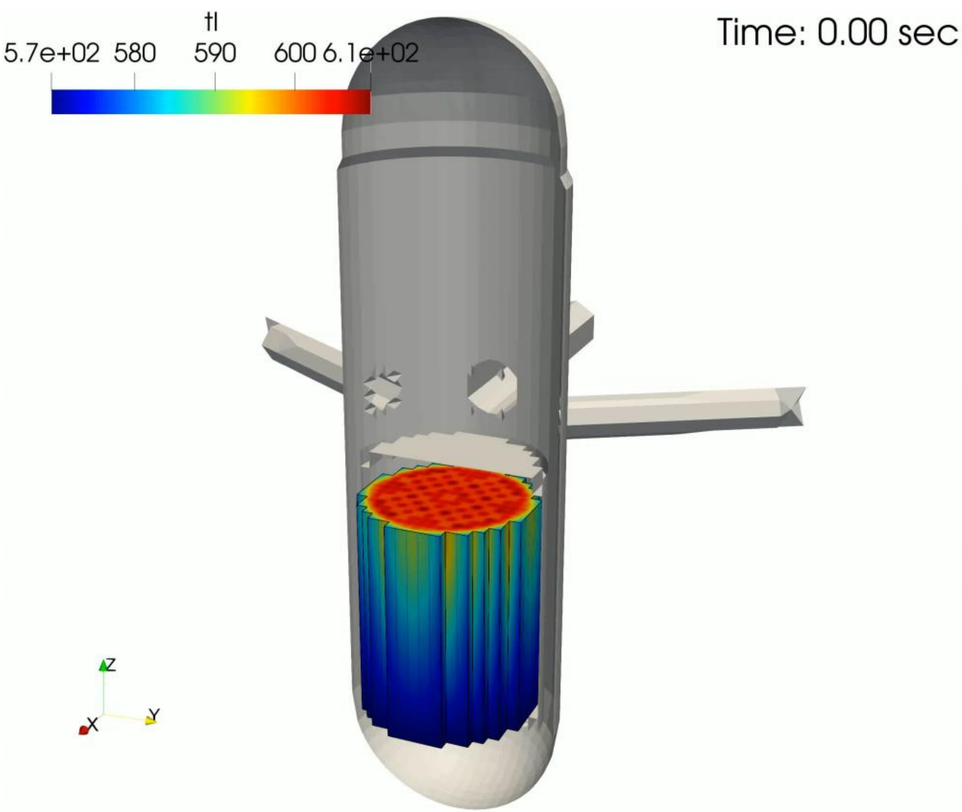


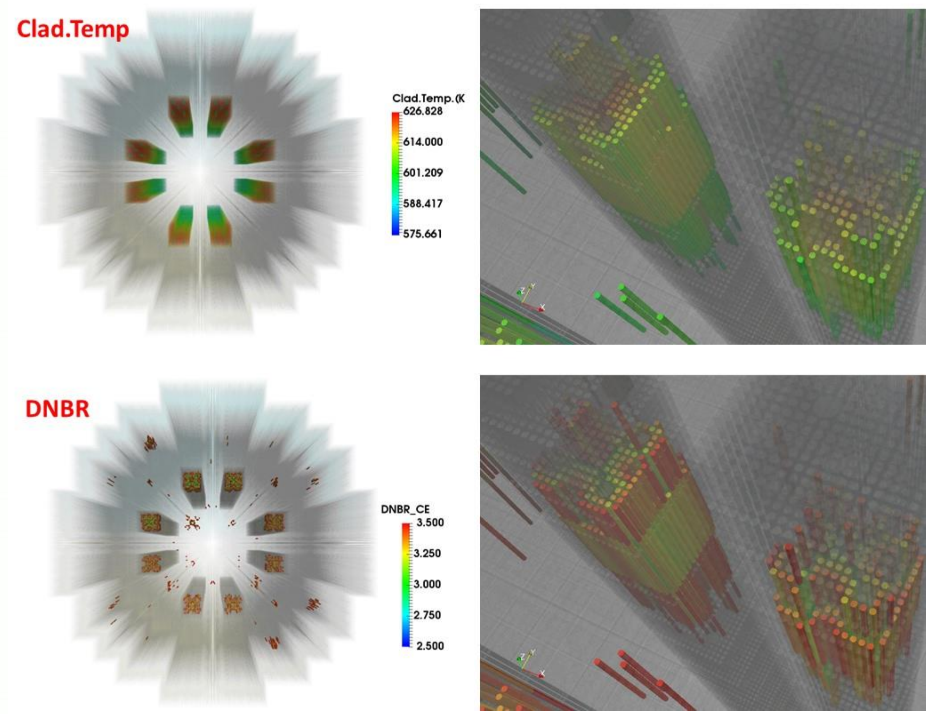
Sequence of events

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

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

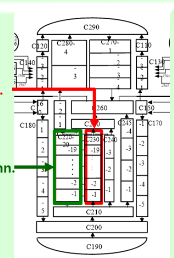
Methodology	MDNBR
1D System-scale TH	2.020
MSMP (w/o Turb. Mixing, w/o FRAPTRAN)	2.331
MSMP (w/i Turb. Mixing, w/o FRAPTRAN)	2.615
MSMP (w/i Turb. Mixing, w/i FRAPTRAN)	2.563

MSMP (+16%)

Phy.model (+11%)

Conservative Fuel model (-2%)

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

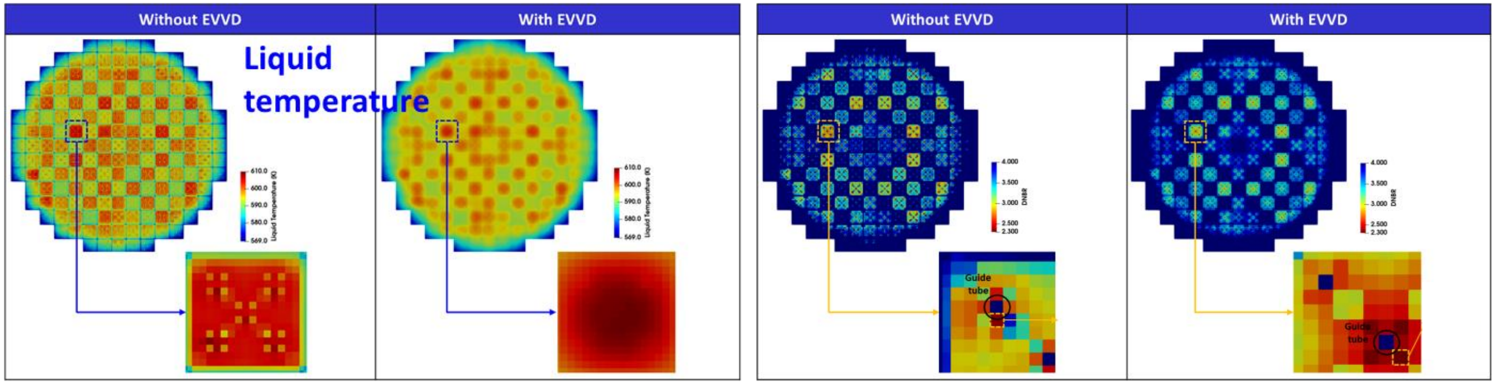


System TH nodal for PWR

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: Non-identical geometric parameters**
  - Various subchannel information
- **Key parameter 2: Realistic fuel power (with N/K code)**
  - Power output of fuel assembly (pin-wise power)
- **Key parameter 3: 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**



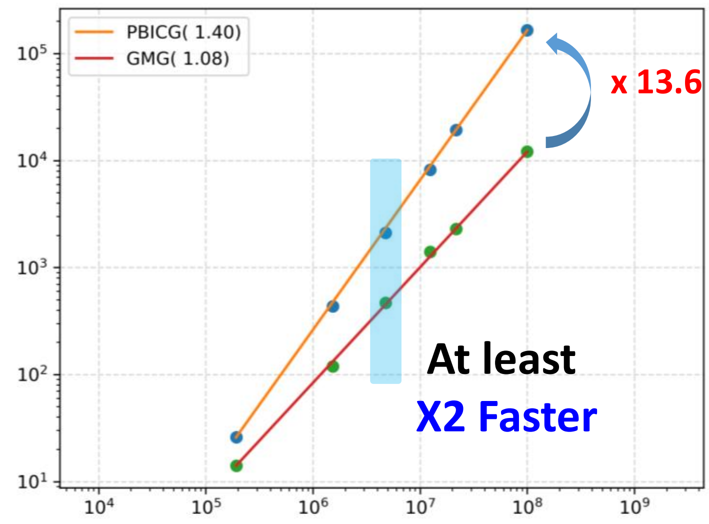
Performance of Pin-wise F/P in SLB Accident

Performance of F/P-Coupled Code

SLB accident safety analysis

- 33% increase of computing time
- Needs to optimize (On-going)
- **Improve performance (using Parallel MG)**

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**



Summary

4

Summary

Summary

High-Fidelity Safety Analysis for Regulation

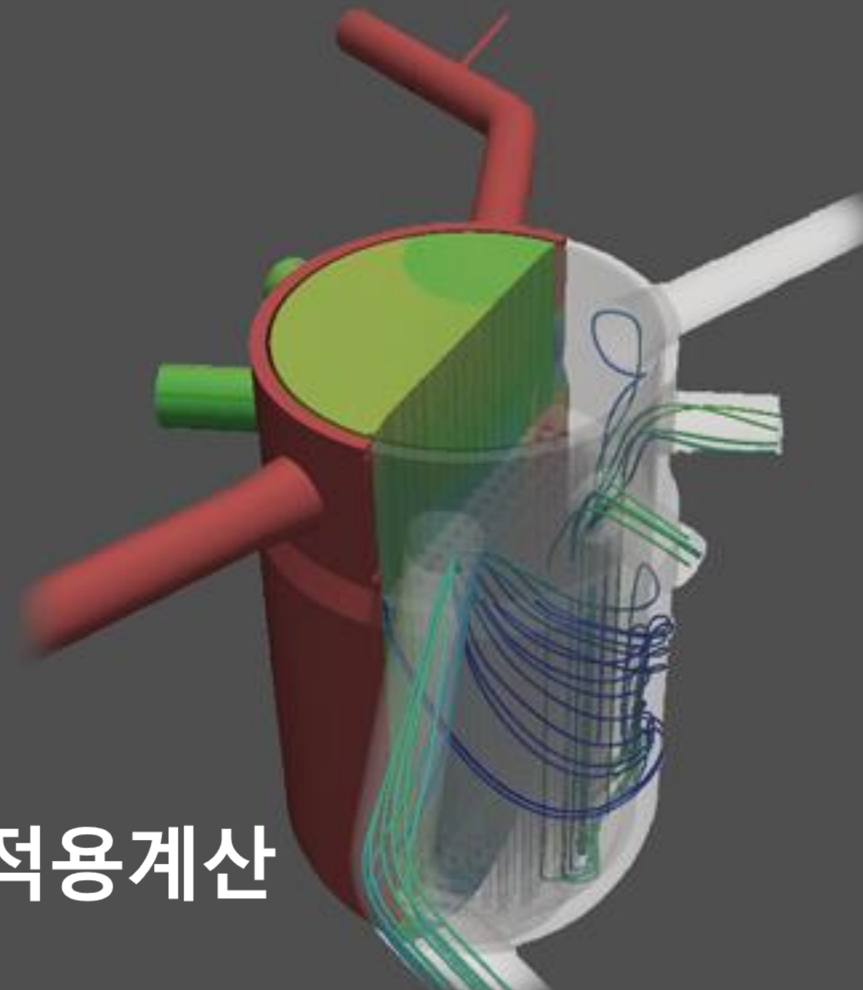
- **MSMP (Multi-scale & Multi-Physics) approach**
- **Precise and practical 3D simulation capability**
  - Subchannel-scaled resolution of CUPID-RV
- **Platform for T/H & N/K & F/P code**

Necessity for 3D MSMP Simulation - SLB

- **Detailed Visualization inside of RPV**
  - **3D** Visualization of **full core fuel power** distribution
- **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
- **Enhancement of safety margin**
  - Realistic **MDNBR evaluation**
  - **Improvement of MDNBR** designed by 1D safety analysis

THANK YOU

jrlee@kaeri.re.kr



10th CUPIDERS

MARU 혁신원전 안전해석 적용계산 (iSMR)

Seung Jun Lee

August 23, 2022

CUPIDERS Workshop

CONTENTS

Introduction to SMR Analysis

1

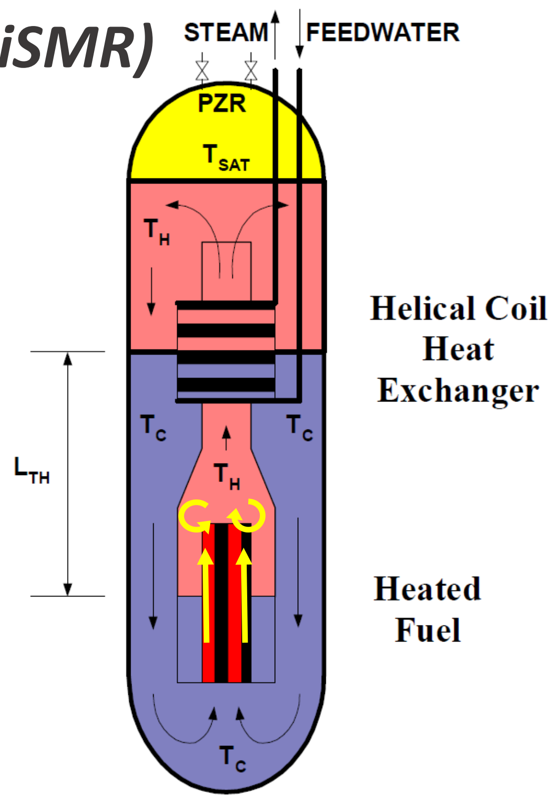
– SMR and 3D Analysis Technology

– SMR 해석기술 개발 전략

SMR and 3D Anlaysis Technology

- ***Flow instability in low flow rate (startup transient)***
  - **Inherently less stable due to nonlinear nature of NC oscillatory flow**
- ***Mixed Convection: RCP + natural circulation (iSMR)***
  - **Asymetry flow in the core (local RCP off)**
  - **Forced-to-natural circulation transition flow**
- ***Power transient by load follow operation***
  - **Local eddy flow at core outlet (power transition)**

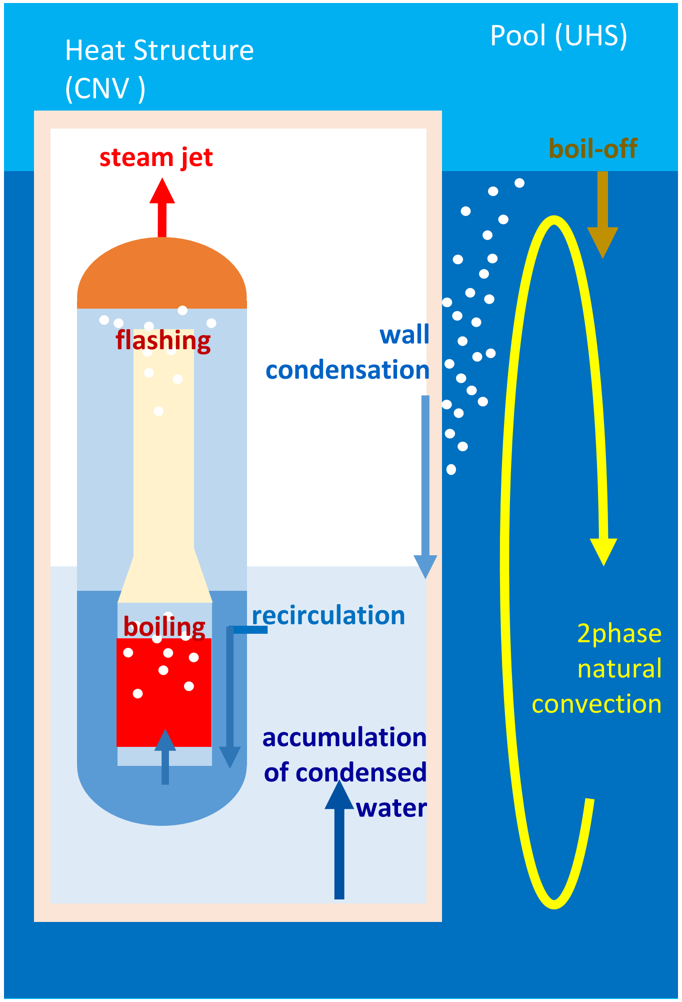
Single-phase natural circulation within an integral reactor*



* IAEA, Natural circulation in water cooled nuclear reactor power plants, IAEA-TECDOC-1474, 2005.

3D Flows in Two-Phase Natural Circulation

Two-Phase Transient Phenomena under Accident Condition



➢Reactor Pressure Vessel (RPV) Side

- Boiling in the core and downcomer
- Flashing when the vent vale opens

➢Containment Vessel (CNV) Side

- Steam condensation on heat structure surface
- Accumulation of the condensed water

➢Pool Side

- Two-phase natural convection
- Boil-off

SMR 해석기술개발 전략

혁신형 SMR 3D 열수력 해석기술 개발

기기스케일 요소기술개발	노심, 증기발생기 해석 연계	사고 시나리오 분석
가상원자로 구현을 위한 주요 기기모델 개발	3차원 열수력 및 노물리 코드 연계 기술개발	가상원자로 기반 혁신형 SMR 성능 분석
혁신형 SMR 기기스케일 해석 기술 - 혁신형 SMR 가상원자로 기능요건 개발 - 혁신형 SMR 3차원 격자기 개발 - 혁신형 SMR 3차원 열수력 RV 모델 개발 및 검증 - 혁신형 SMR 기기모델 개발 (MCP, 가압기 모델) - 증기발생기 모델 개발 (1차계통) - 병렬 해석 알고리즘 개발	**혁신형 SMR 기기스케일 해석 기술** - 전열관 열전달 모델 개발 - 급수관/증기관 모델 개발 **3차원 열수력 및 노물리 코드 연계 기술** - 혁신형 SMR 정상/과도상태 해석 - 열수력-노물리 연계기술 - 다물리 연계를 통한 혁신형 SMR 정상/과도상태 해석	**혁신형 SMR 가상원자로 기술** - 혁신형 SMR 가상원자로 시나리오 선정 및 입력작성 - 가상원자로 기반 시나리오 적용 해석 - 3차원 해석 병렬 후처리 기술개발 - 가상원자로 기반 혁신형 SMR 성능 평가
혁신형 SMR 3차원 열수력 해석 기술 개발 →	3차원 열수력-노물리 연계 기술 개발 →	가상원자로 기반 혁신형 SMR 성능 평가

설계 협력

3D Mesh and Component Models for SMRs

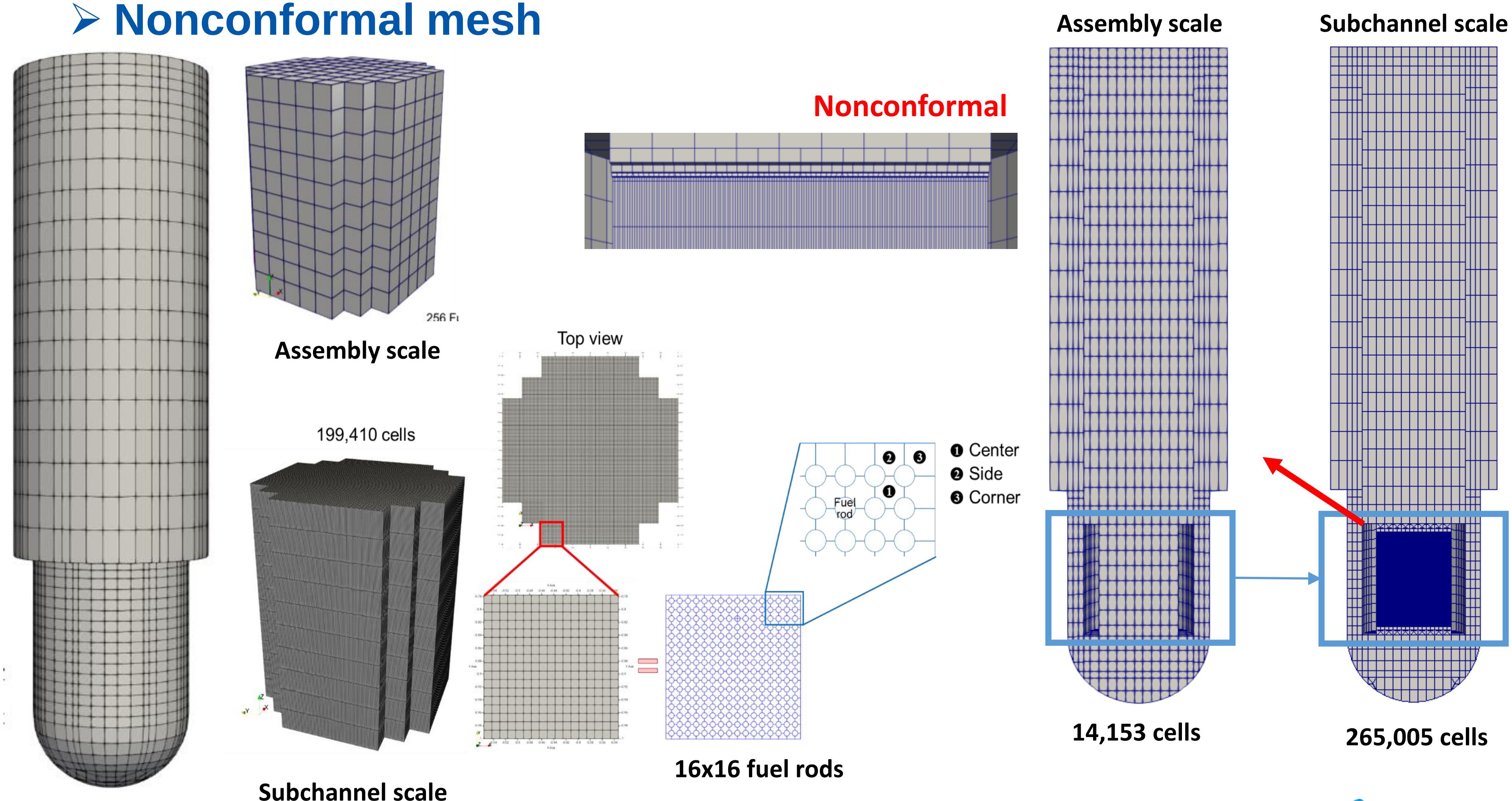
2

- 3D Mesh Generation
- MCP and Valve Model
- Pressurizer Model
- Internal Structure Model: Heat Conduction and Pressure Drop
- Steam Generator and Nuclear Reactor Core

3D Mesh Generation

3D Mesh generation (assembly and subchannel-scale)

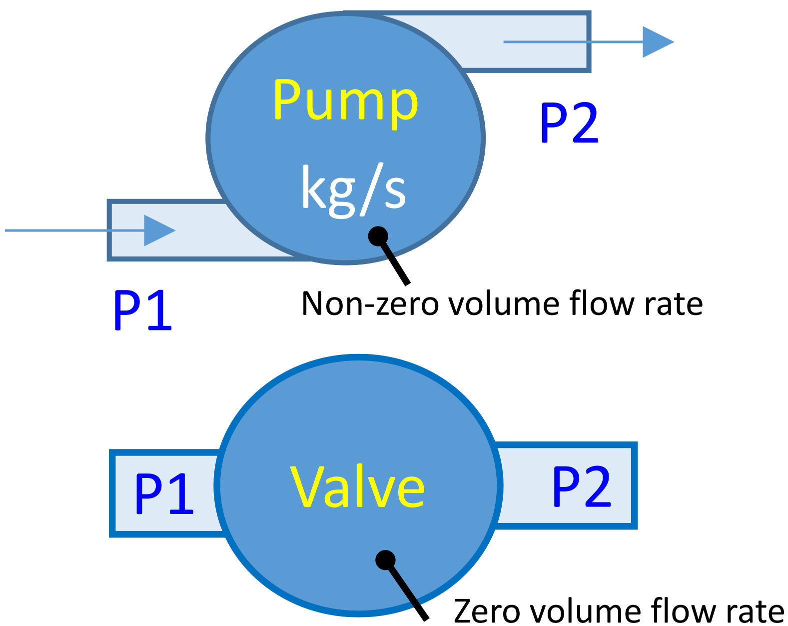
- 주요 영역 분할
- Nonconformal mesh



MCP and Valve Model

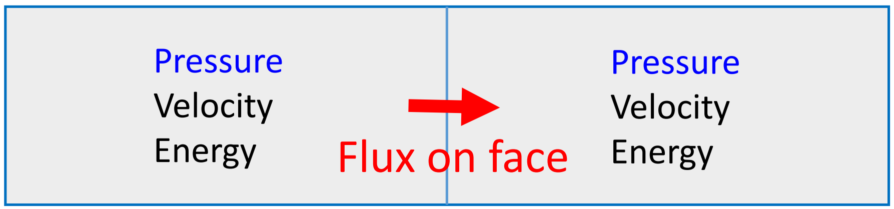
Flux Boundary Condition Model

- 펌프와 밸브의 공통 사항: 유량 제어, 압력장 단절
- 단일 격자 적용 유동 경계조건의 변화
- MCP, Valve 모델
- 추가 적용: Critical Flow 모델

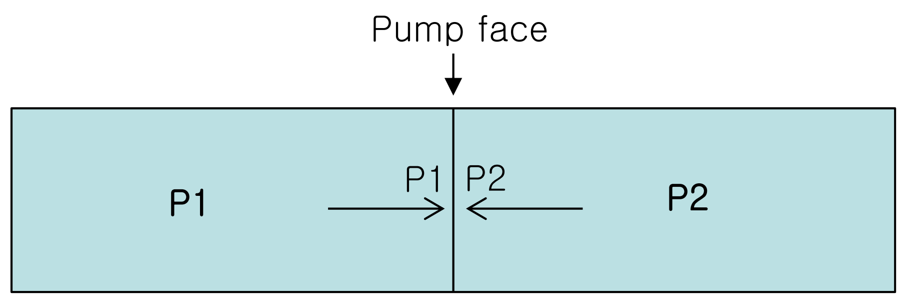


펌프와 밸브의 압력, 유동 특성

Collocated grid based finite volume approach



압력변화(∇P)는 cell face에서의 유량과 연계
Flux 재정의시 압력변화(∇P) 수정 필요



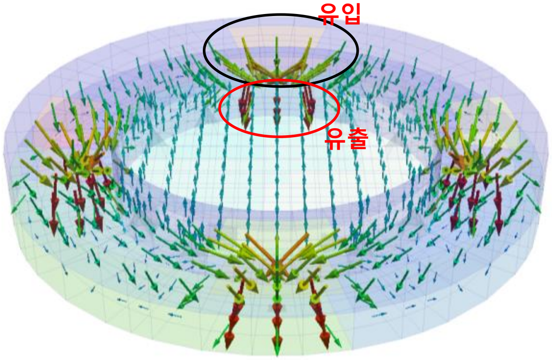
Asymmetric summation of ∇p_i^{nc}

Neumann BC at FluxBC face

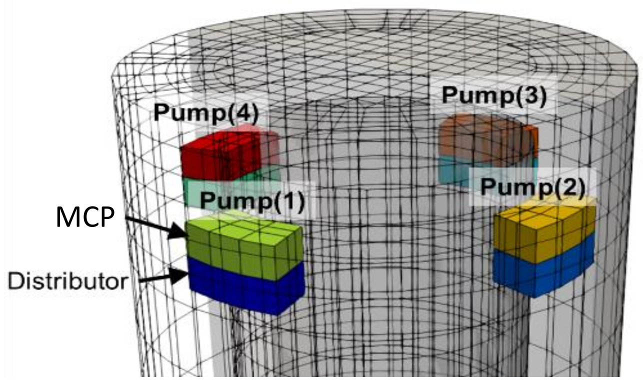
Validation of MCP Model

Mass Balance and MCP RPM Control

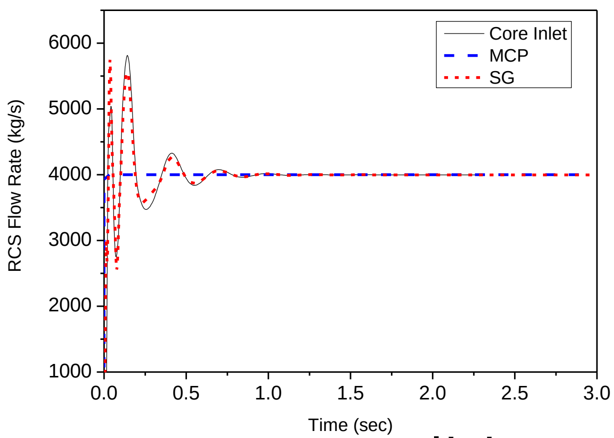
- **4 MCPs (1 MCP mass flow rate=1000 kg/s)**
- **RPM control 3600 rpm to zero at 400 sec**



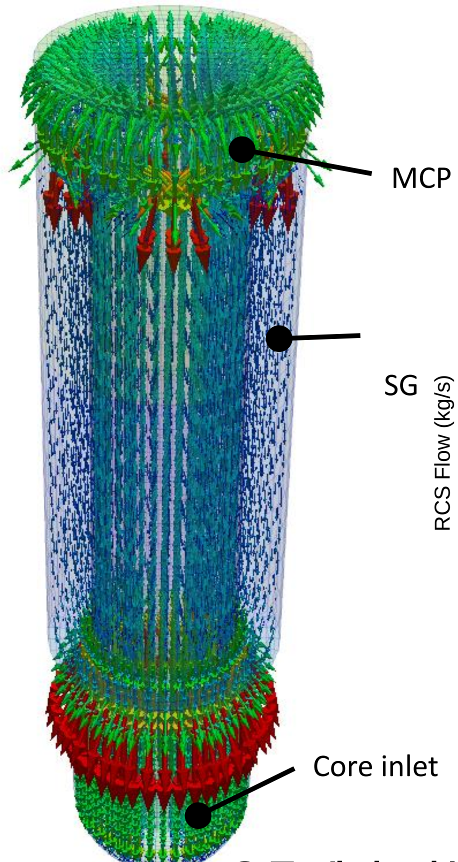
유동 벡터 (MCP, Distributor)



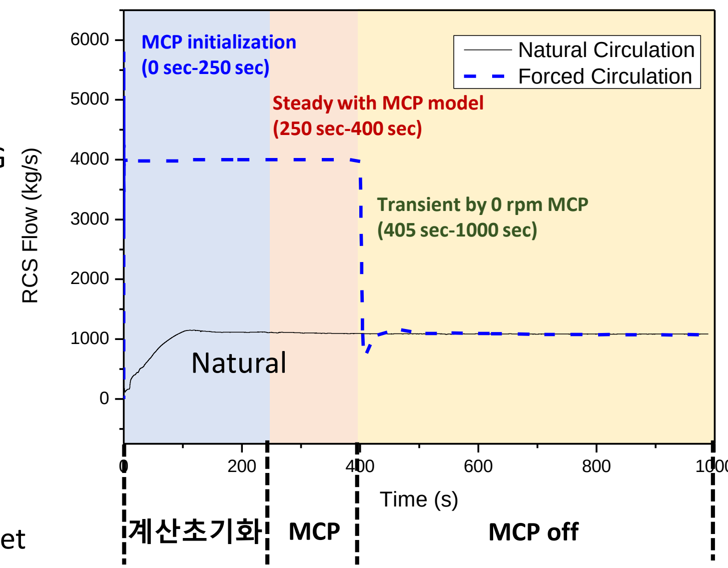
MCP와 Distributor



RCS Flow Rate 확인



유동 벡터 (전체)

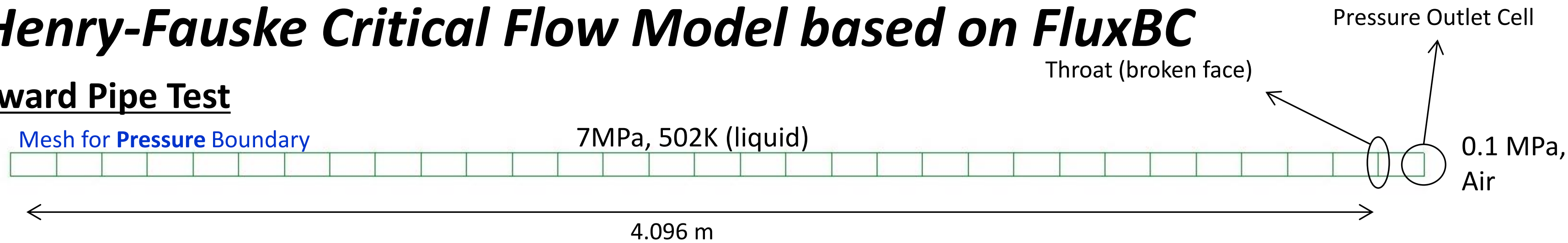


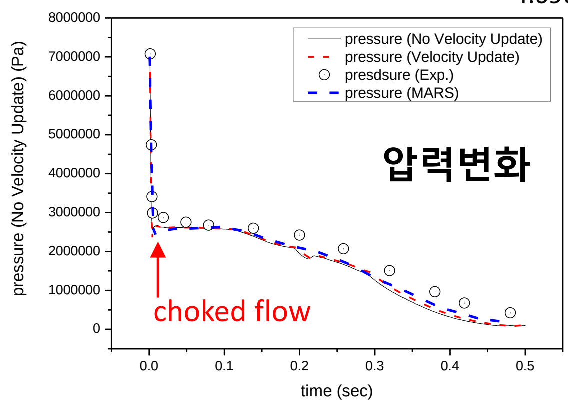
MCP RPM 제어

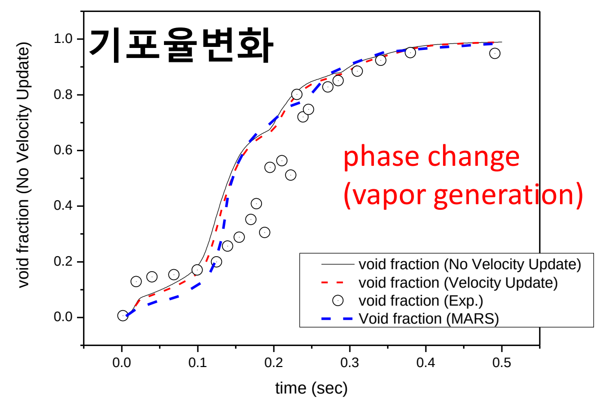
Validation of Critical Flow, Valve Model

Henry-Fauske Critical Flow Model based on FluxBC

Edward Pipe Test

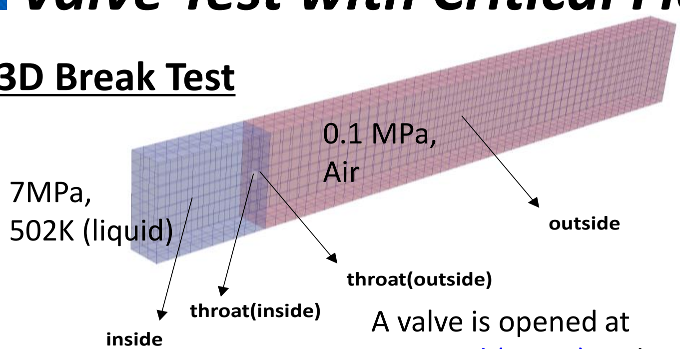




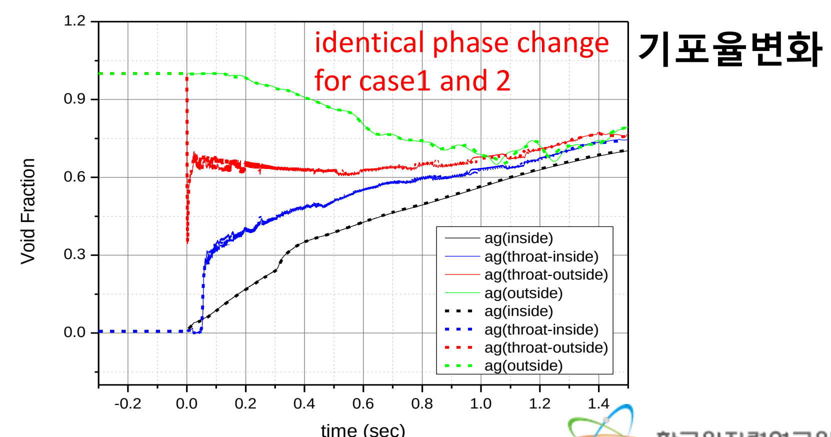


Valve Test with Critical Flow Model

3D Break Test

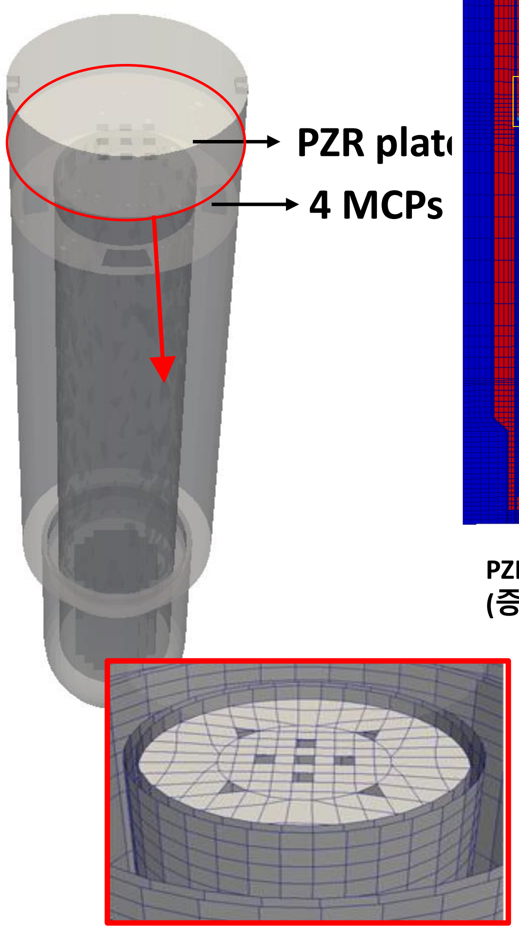


A valve is opened at
0 second (case1) and 0.3 seconds (case2).

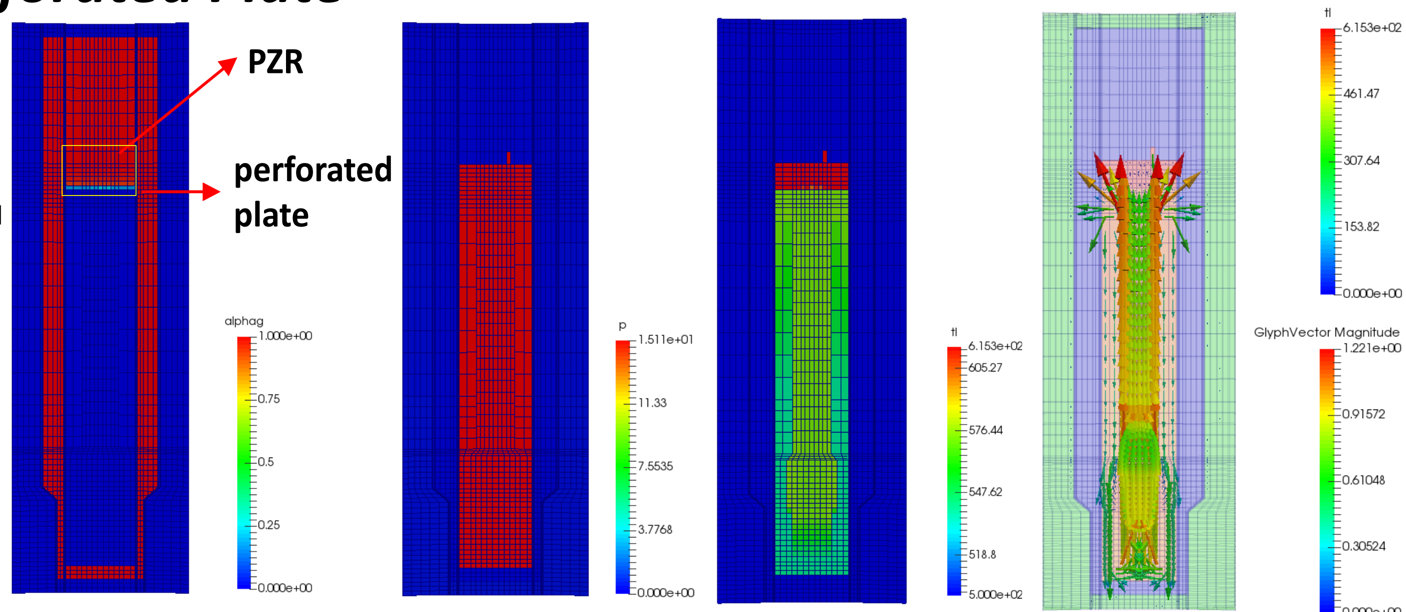


Pressurizer Model

PZR with a Perforated Plate



다공판 모의 (임의형상)

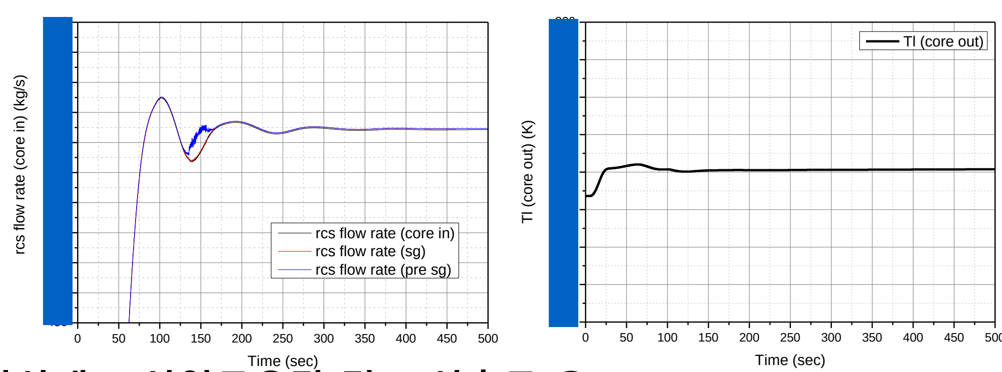


PZR 영역 설정 (증기 100%)

압력 (150 bar, 정상상태)

액체온도 (정상상태)

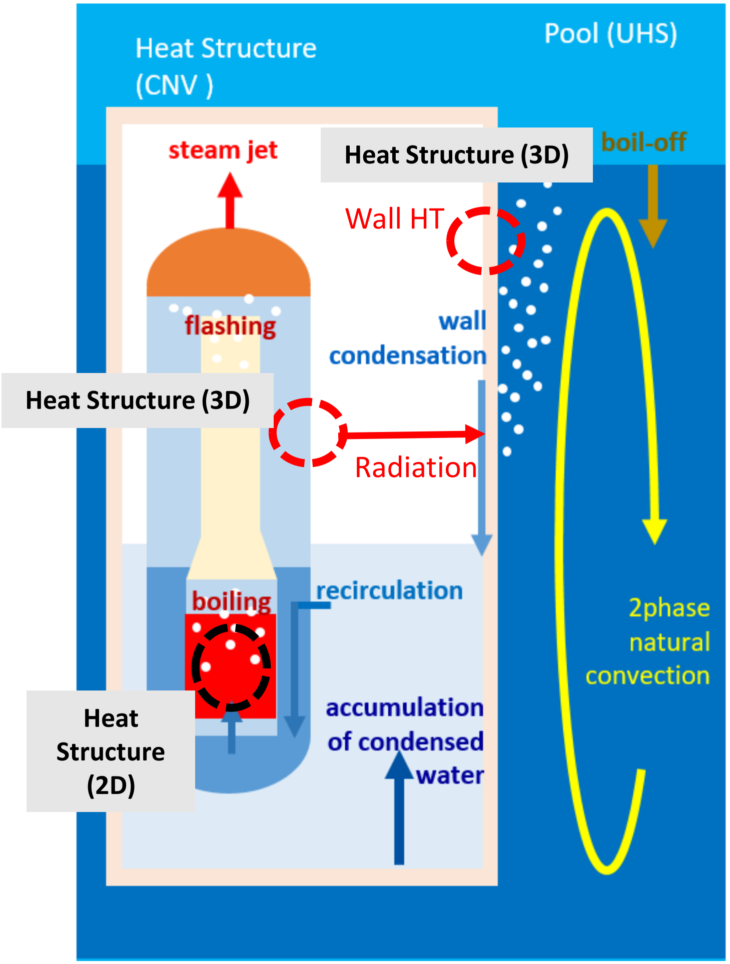
액체속도 (정상상태)



정상상태 노심입구유량 및 노심출구 온도

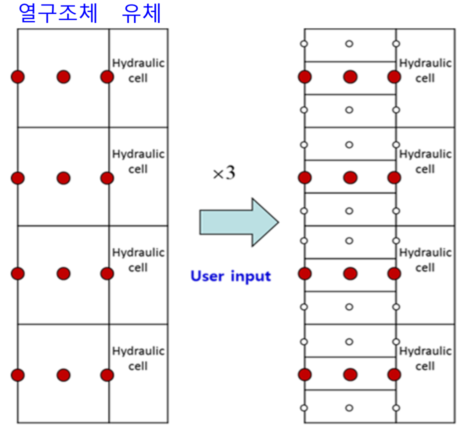
Internal Structure Model: Heat Conduction

Heat Structure Analysis

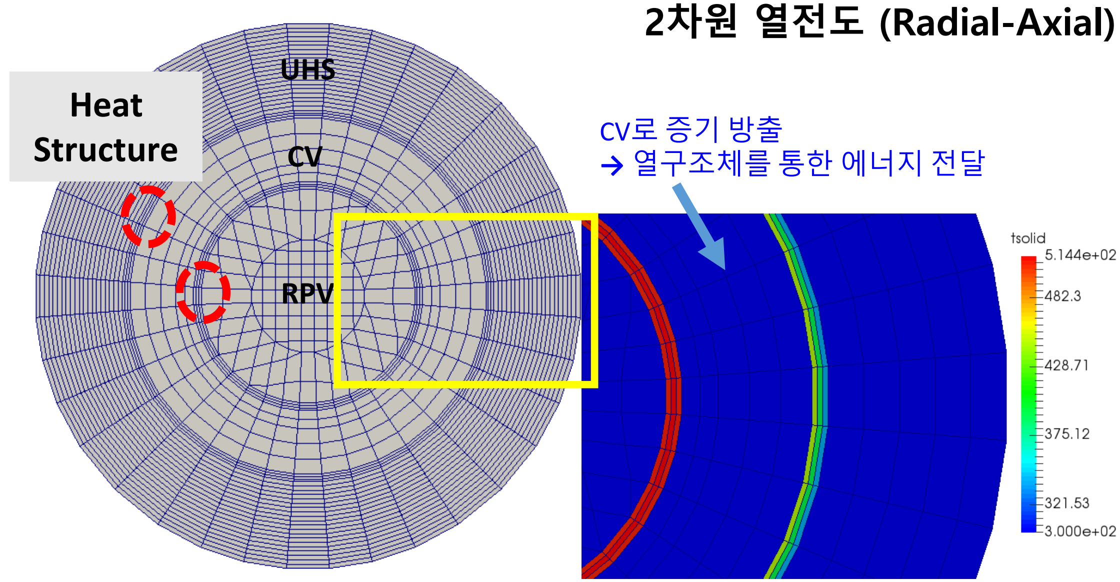


$$\rho C_p \frac{\partial T}{\partial t} = \nabla \cdot \left(k \nabla T \right) + \ddot{q}$$

RPV, CV 구조물
3차원 열전도



노심 핵연료 열구조체
2차원 열전도 (Radial-Axial)



3차원 열전도 적용
iSMR 열구조체 계산 (LOCA 조건)

Internal Structure Model: Pressure Drop

Friction Model for the Porosity-based Inner Structure in SMR

- **Porous medium approach for component-scale resolution**
- **Pressure drop modeling**
  - CFD, experiment, friction model

$$F_{wk} = \frac{dp/dz}{V_{steady}^2} V$$

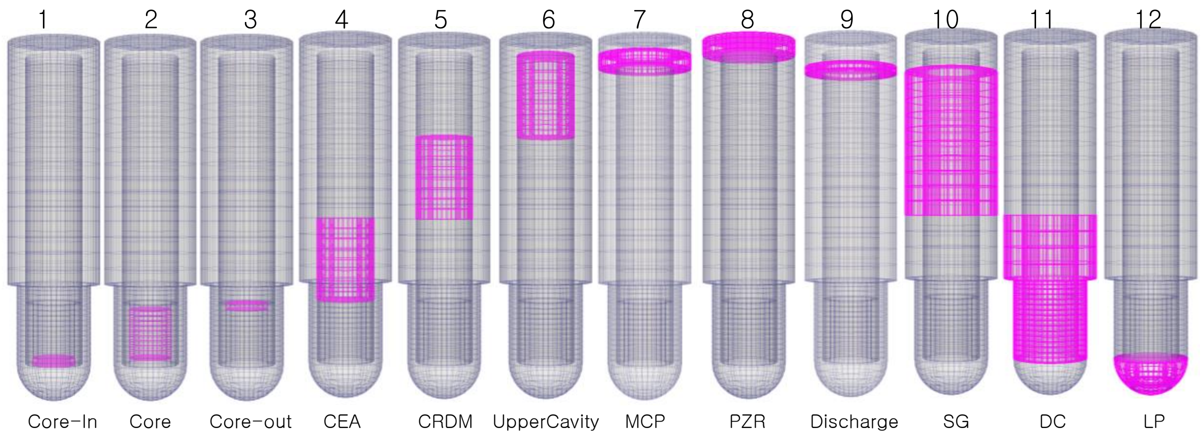
1) S.State-based K-factor

$$F_{wk} = \frac{(dp + dp_{control})/dz}{V_{steady}^2} V$$

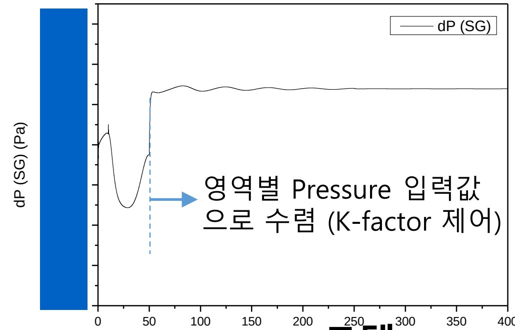
2) Adaptive K-factor

$$F_{ix} = f_{cH} C_{AH} C_{VH} A_{SH} \rho u_{ix} |u_{ix}|$$

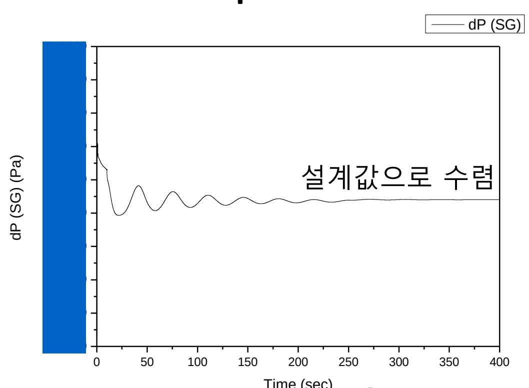
3) Friction-based physical model



영역별 다공도 및 압력강하 모델 적용



Adaptive 모델

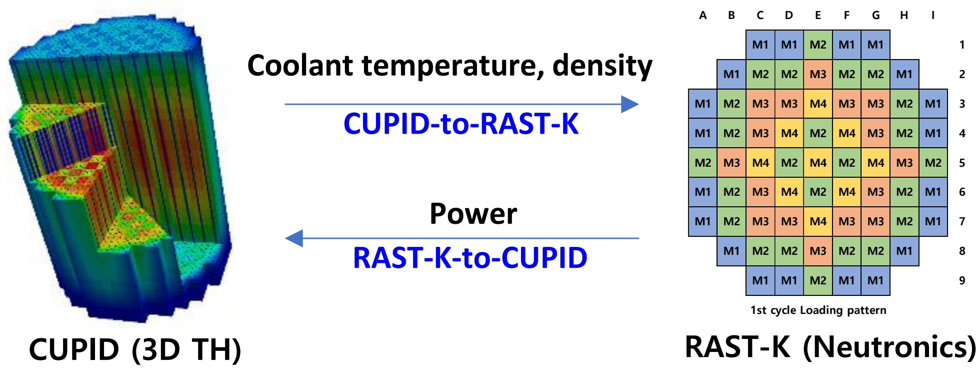


Physical 모델

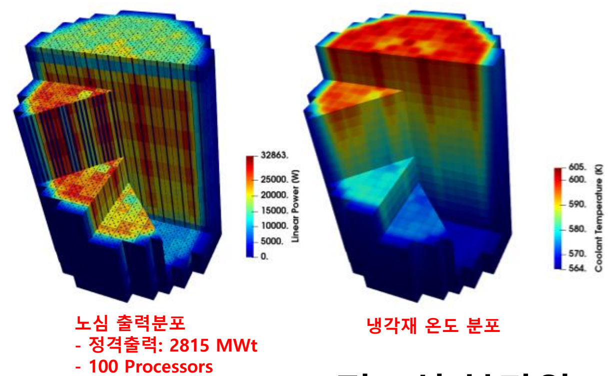
Steam Generator and Nuclear Reactor Core

Multi-Scale, Multi-Physics Code Coupling (진행중)

- **MARU 기반 socket 통신 연계기술**
- **Nuclear Reactor Core (현재 DLL, so 연계)**
  - CUPID-Neutronics 코드 연계 (Diffusion 코드: RAST-K, MASTER)

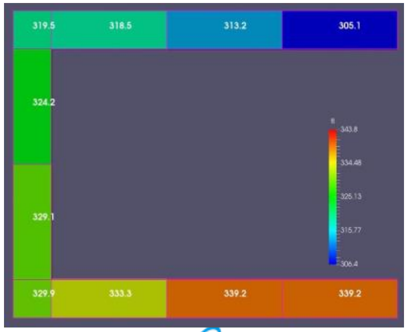


CUPID-RAST-K 연계 개념도



CUPID-RAST-K 전노심 봉단위 정상상태 예비해석 (OPR1000)

- **Stream Generator (현재 socket 통신 연계)**
  - CUPID-CUPID-SG 코드 연계
  - CUPID-SG (가칭) for helical coil SG



CUPID3D-CUPID1D 열구조체 연계 예비해석 (1D 결과)

Application to SMR Analysis

3

- Radiation Model
- Condensation Model
- Natural Circulation of NuScale
- 3D Full Core Analysis Using Subchannel Model
- Conceptual SMR LOCA Anlysis

Radiation Model

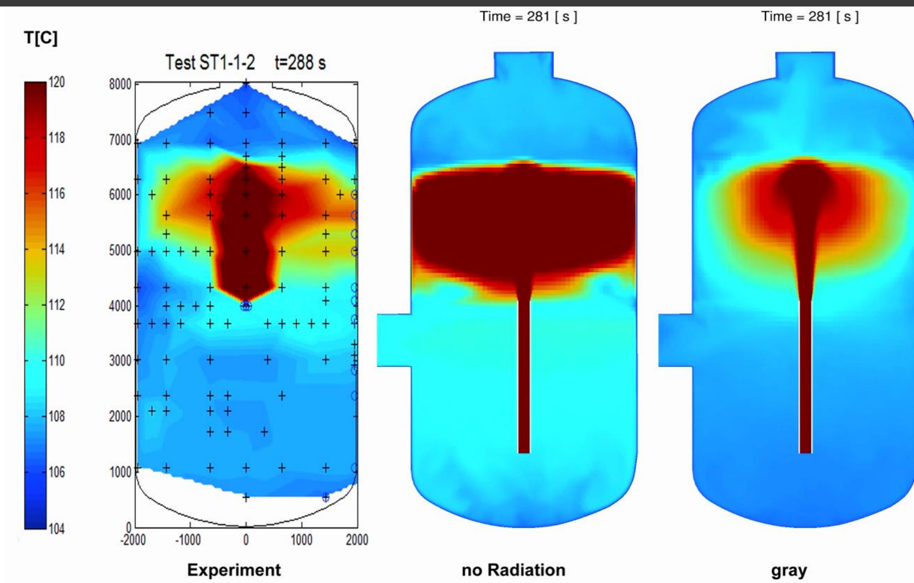
Two Radiation Models

Radiation HT in CV

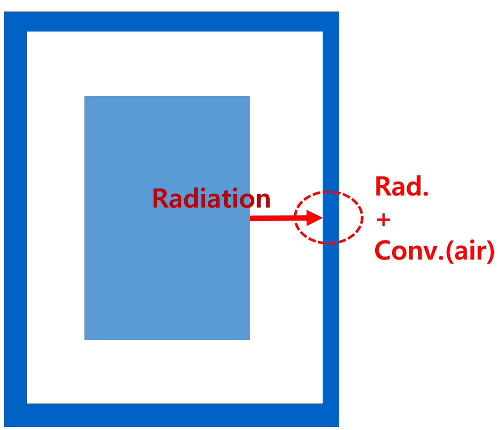
- Radiation from steam
- P1 model

Radiation HT Test

- Radiation from heat structure
- Effect of vacuum



Effect of radiation model-HYMERES2 project



Vacuum and air-filled CV (정상상태)

	CV-Vacuum	CV-Air inerting
Initial Pressure	2kPa.abs	1atm
Heat Loss(Total)	0.109 MW	1.040 MW
Heat Loss (Rad.)	0.089 MW	0.083 MW

Condensation Model

벽면응축모델 적용

Uchida 응축 모델

$$h_{Uchida} = 380\left(\frac{W_s}{1-W_s}\right)^{0.7}$$

W_s : 증기질량분율

Colburn-Hougen 응축 모델

$$h_c\left(T_{vi} - T_w\right) = h_m i_{fgb} \rho_{vb} \ln\left(\frac{1 - p_{vi}/p}{1 - p_{vb}/p}\right)$$

h_m : 질량전달계수

Heat and mass transfer analogy 모델 (HMTA)

$$\Gamma_{wall} = m'' \frac{A_{cell}}{V_{cell}} = -h_m \ln\left(\frac{1-W_{s,w}}{1-W_{s,\infty}}\right) \frac{A_{cell}}{V_{cell}}$$

$W_{s,w}$: 응축벽면의 증기질량분율

$W_{s,\infty}$: 벌크영역의 증기질량분율

Resolved boundary layer approach 모델 (RBLA)

$$\Gamma_{wall} = m'' \frac{A_{cell}}{V_{cell}} = -\frac{1}{\left(W_s - 1\right)} \rho D \frac{\partial W_s}{\partial n} \frac{A_{cell}}{V_{cell}}$$

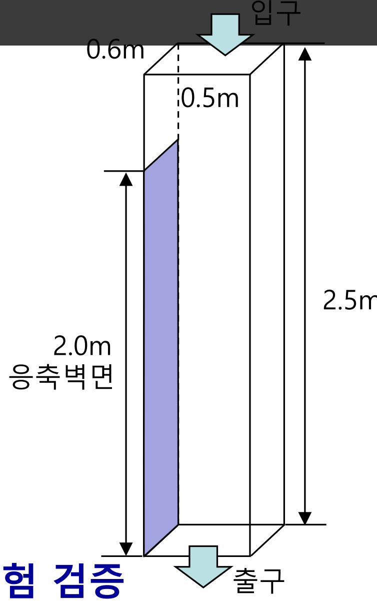
D : 2혼합기체 확산계수

Validation of Condensation Model

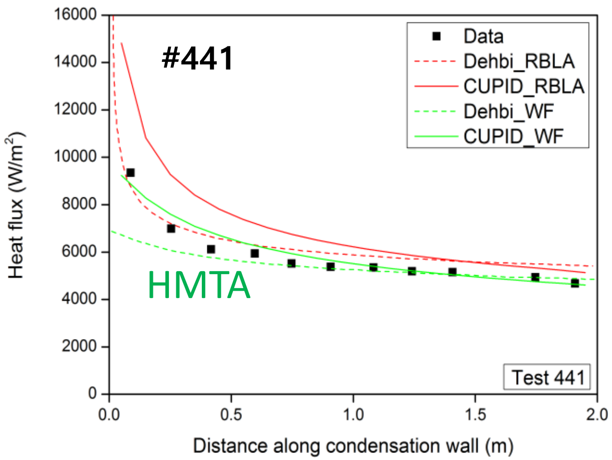
벽면응축모델 검증 계산

- HMTA: 강제대류 조건 기반 모델
- RBLA: y+~5 의 격자기반

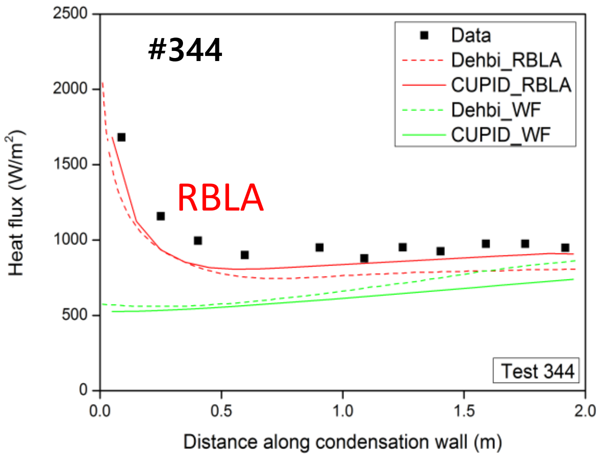
Case	Convective Heat Trans.	Velocity [m/s]	Pressure [bar]	Gas T [K]	Wall T [K]	Quality
P0441	Forced	3.0	1.02	353.23	307.4	0.767
P0344	Natural	0.3	1.21	344.03	322	0.864



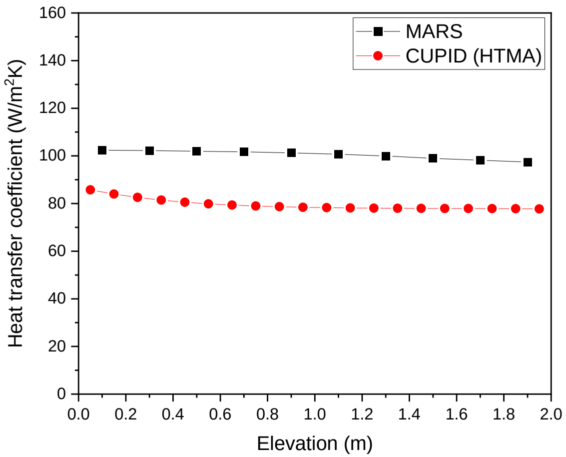
COPAIN 실험 검증



강제대류 조건 결과



자연대류 조건 결과

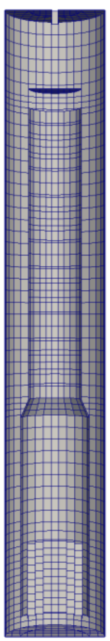


MARS (C-H) 와 비교

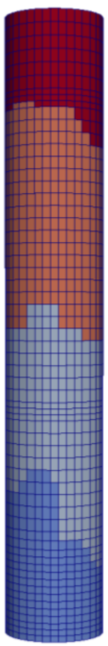
Natural Circulation of NuScale Reactor

Analysis Conditions:

- Power=100 MWth (Uniform heat source and sink: Core and SG)
- Problem time=100 sec (wall clock time=180.3 sec)

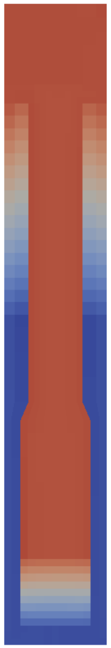


Wall BC

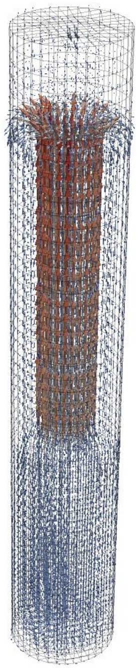


6 MPI Processors

(9624 Cells)

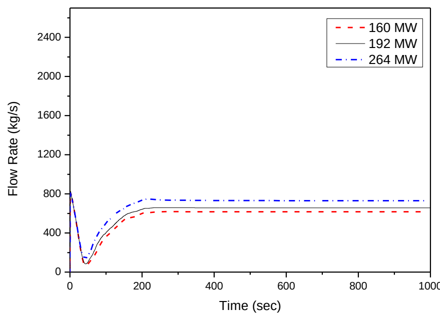


Liquid Temperature

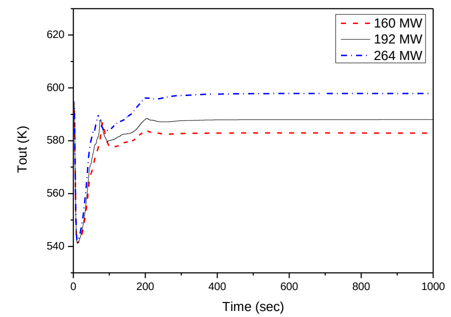


Liquid Velocity

Power (MWe)	Flow Rate (kg/s)	Tsub (K)
50	617.3	25.1
60	656.2	20.1
77	730.5	10.1



Natural Circulation Flow Rate



Core Exit Temperature

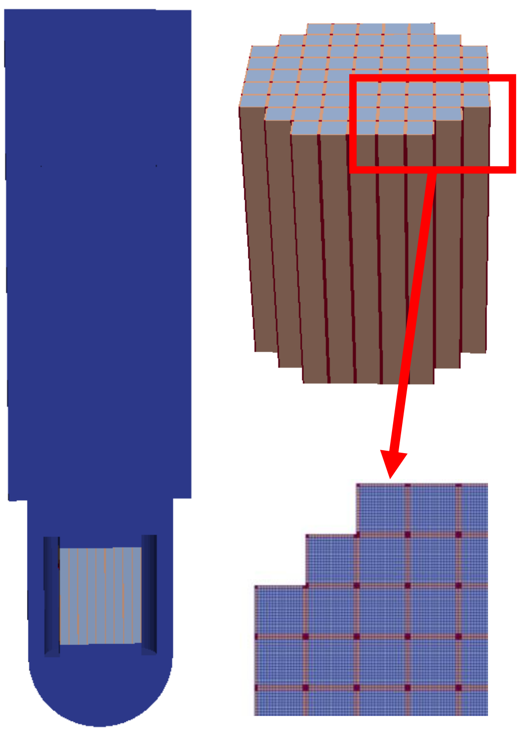
(Ref) Pressure=13.80 MPa (Tsat=608 K)
Flow rate=641.5 kg/s
$T_{core,in}$=538.15 K
$T_{core,out}$=594.15 K

Difference with design value	
Flow Rate	2.29 %
$T_{core,out}$	1.05 %

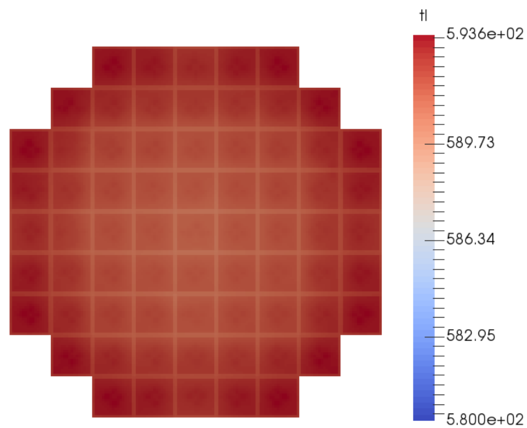
3D Full Core Analysis Using Subchannel Model

Subchannel model test

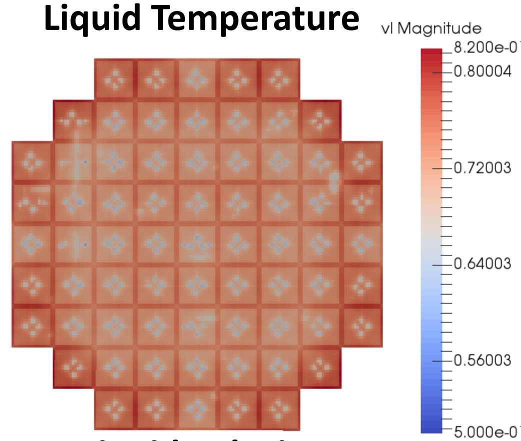
- Natural Circulation in iSMR
- Turbulence Mixing by EVVD model, Friction by MATRA model



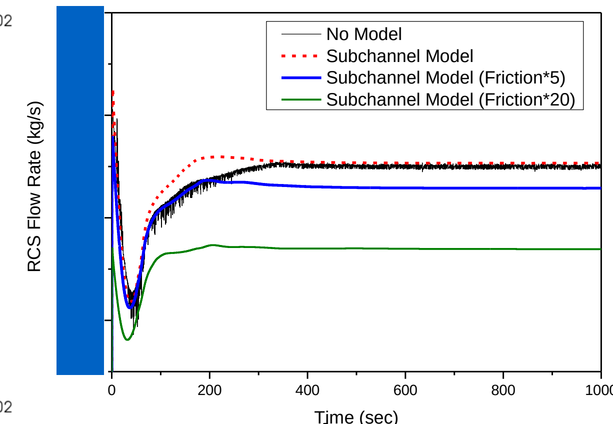
3D subchannel mesh
for a conceptual iSMR



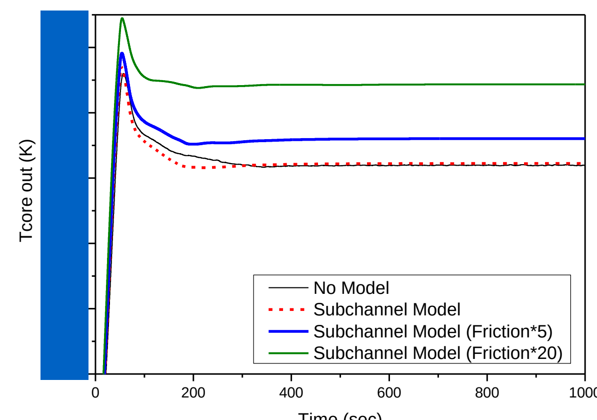
Liquid Temperature



Liquid Velocity



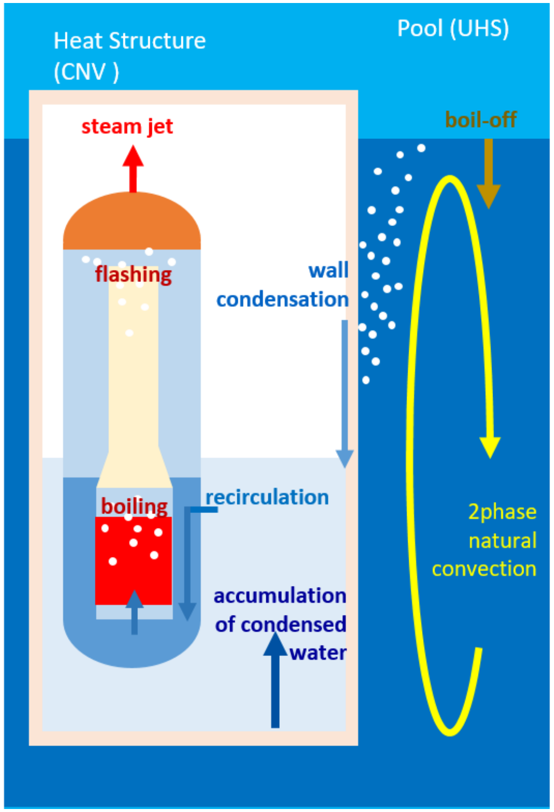
RCS Flow Rate



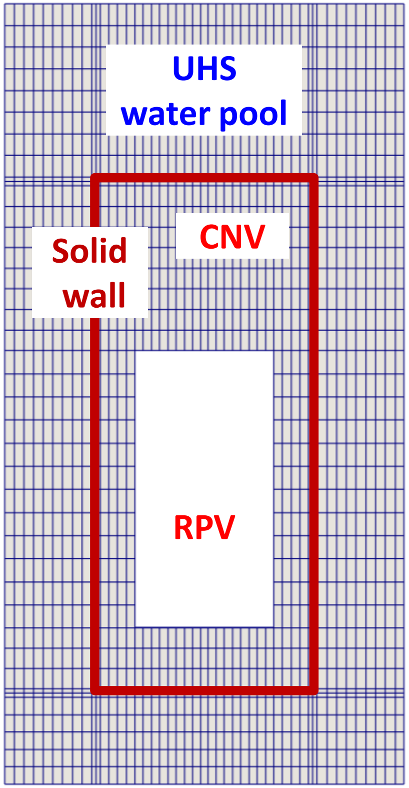
Core Exit Temperature

Conceptual Problem for SMR LOCA Analysis

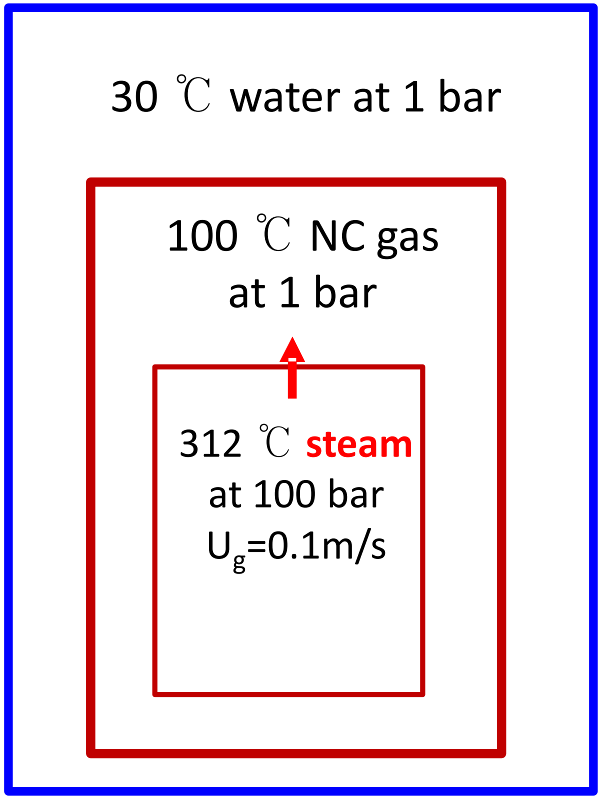
- ***Setup a conceptual problem to verify the CUPID code capability for the application to SMR LOCA analysis***
- ***2D mesh model for RPV, CNV, CNV solid wall, and UHS***



SMR LOCA Phenomena



2d Mesh Model

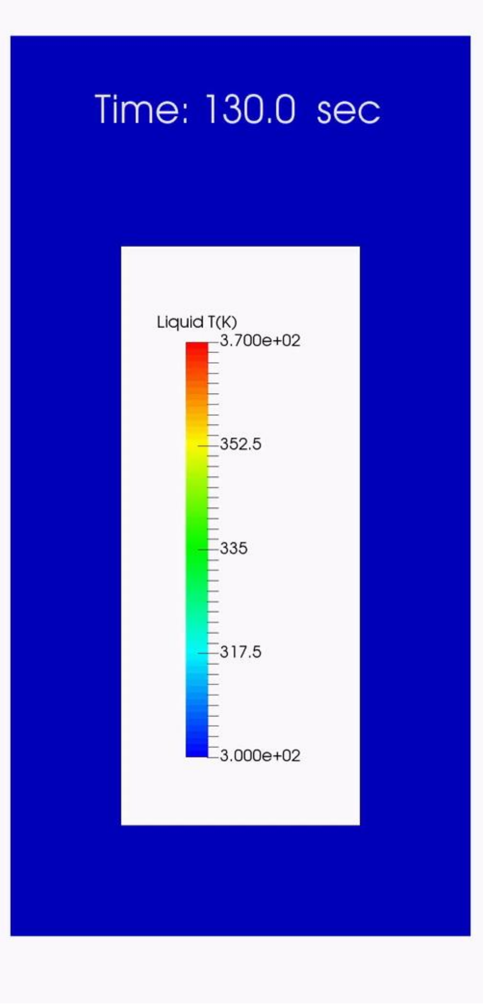


Initial Conditions

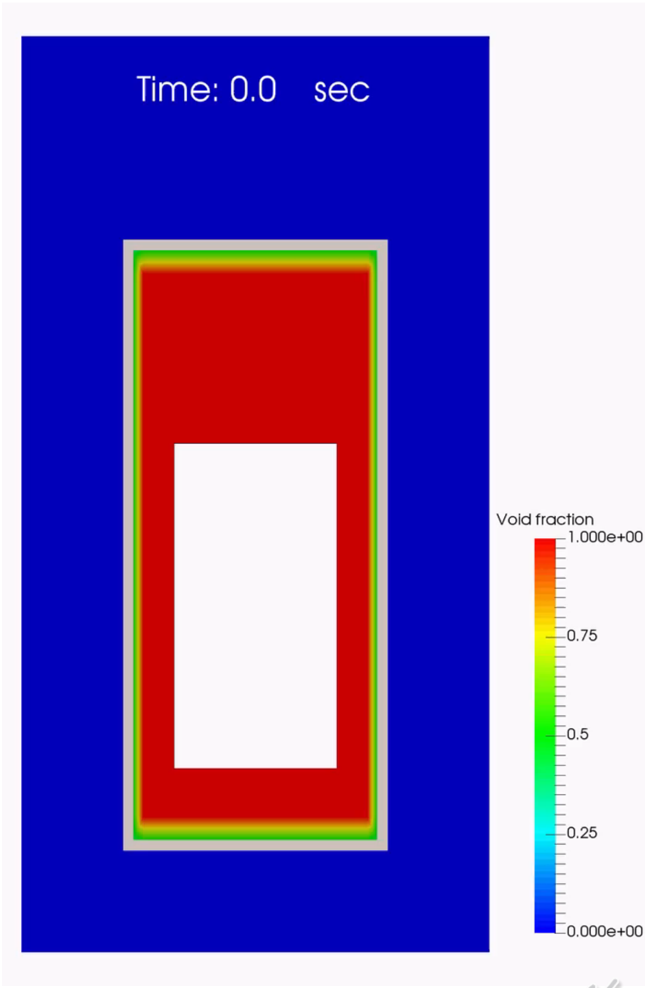
Conceptual Problem for SMR LOCA Analysis

- ***10^5 seconds (27.7 hours) of long transient was successfully simulated***
  - **Water lever increase in CNV due to condensation**
  - **Water lever decease in UHS due to boil-off**
- ***Numerical stability***
  - **Simulation took 4300 seconds with 4 CPUs**
  - **Practical application to Full 3D analysis is achievable**

Liquid temperature in UHS



Gas volume fraction

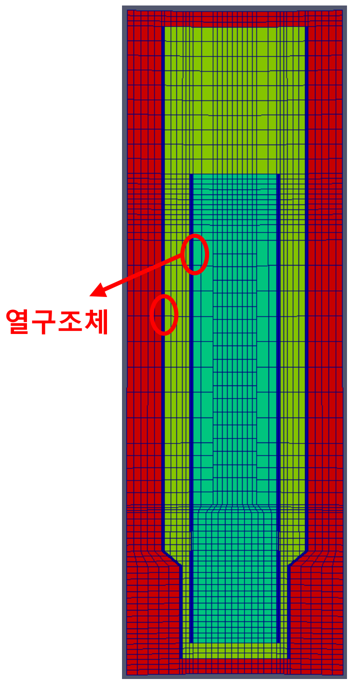


NuScale 3D LOCA 해석: 정상상태

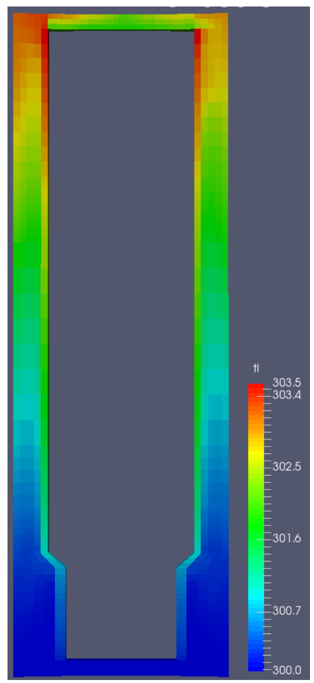
Initial & Boundary Condition

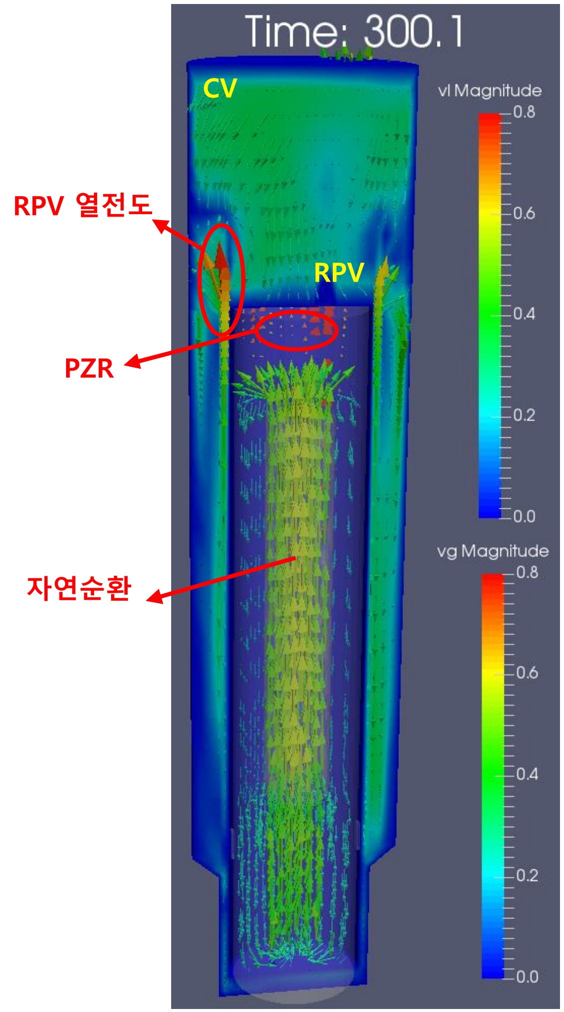
- *Pressure=15.0 MPa*
- *Power=192 MW*
- *Tcore,in=538.15 K*
- *Decay heat curve: ANS-73*

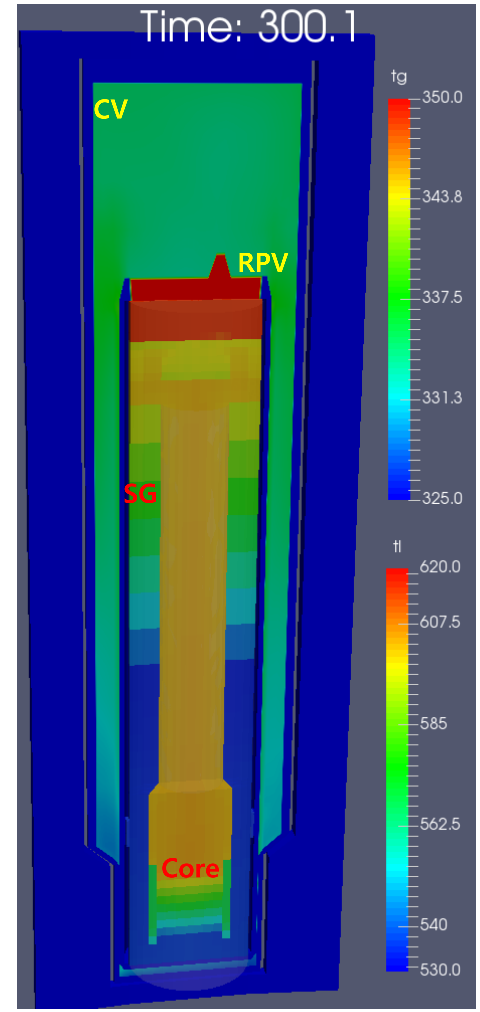
Grid (48k)



Water Temp. in Pool



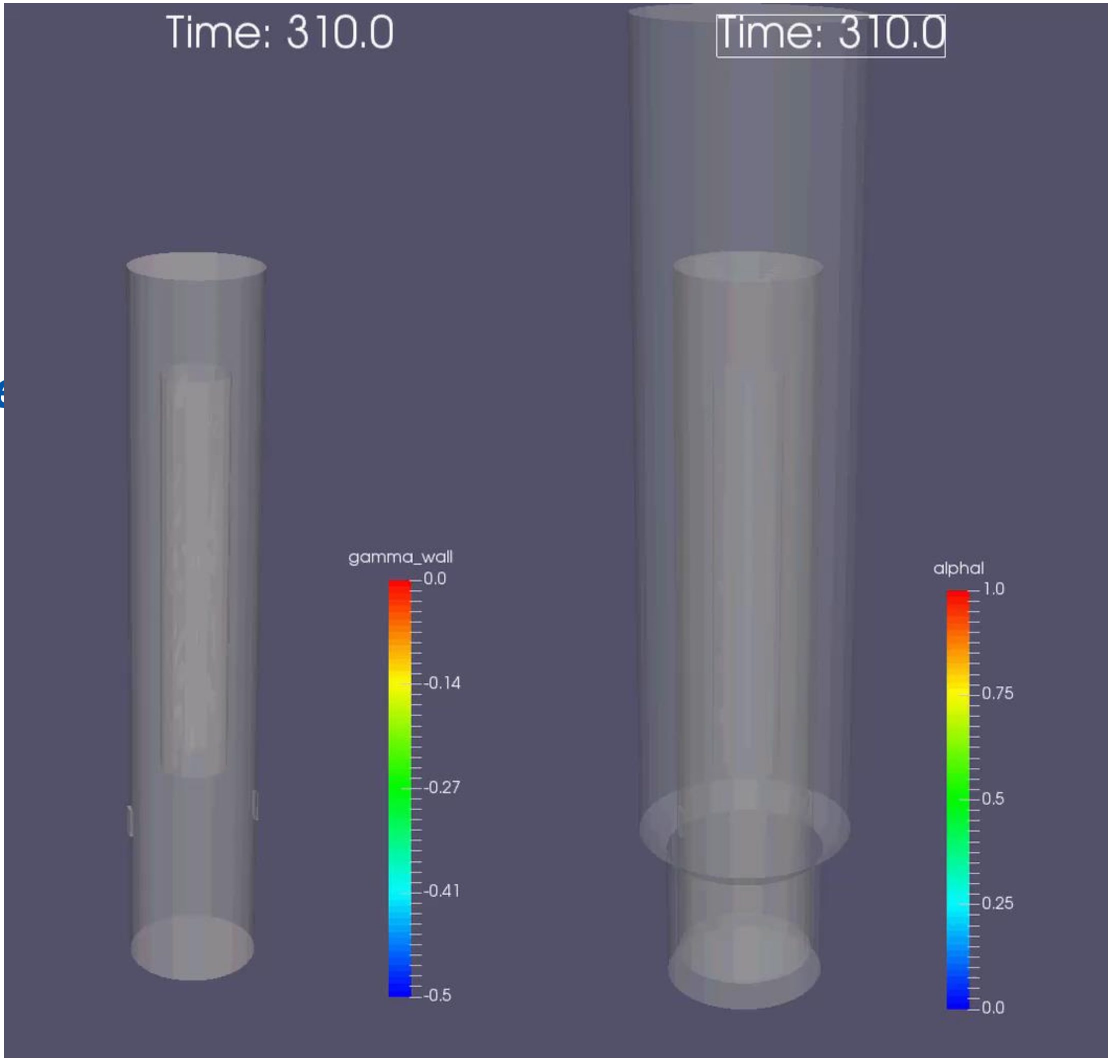




NuScale 3D LOCA 해석: 벽면응축

Condensation

- *Wall condensation at side wall of CV*
- *Upper wall of CV is uncovered*
- *Accumulated condensate at lower part of CV*
- *Water level decreases after RRV opening*



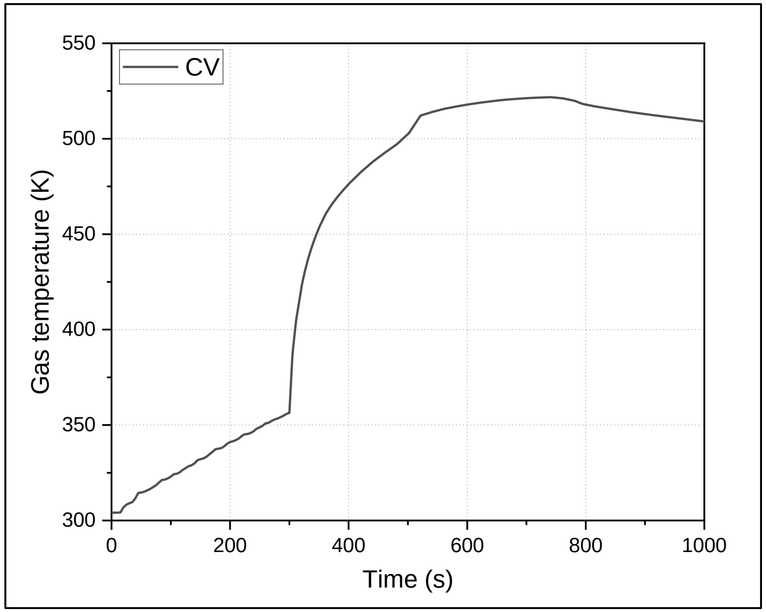
Condensation Rate (kg/s) **Liquid Fraction**

NuScale 3D LOCA 해석: RVV 개방

RVV Actuation

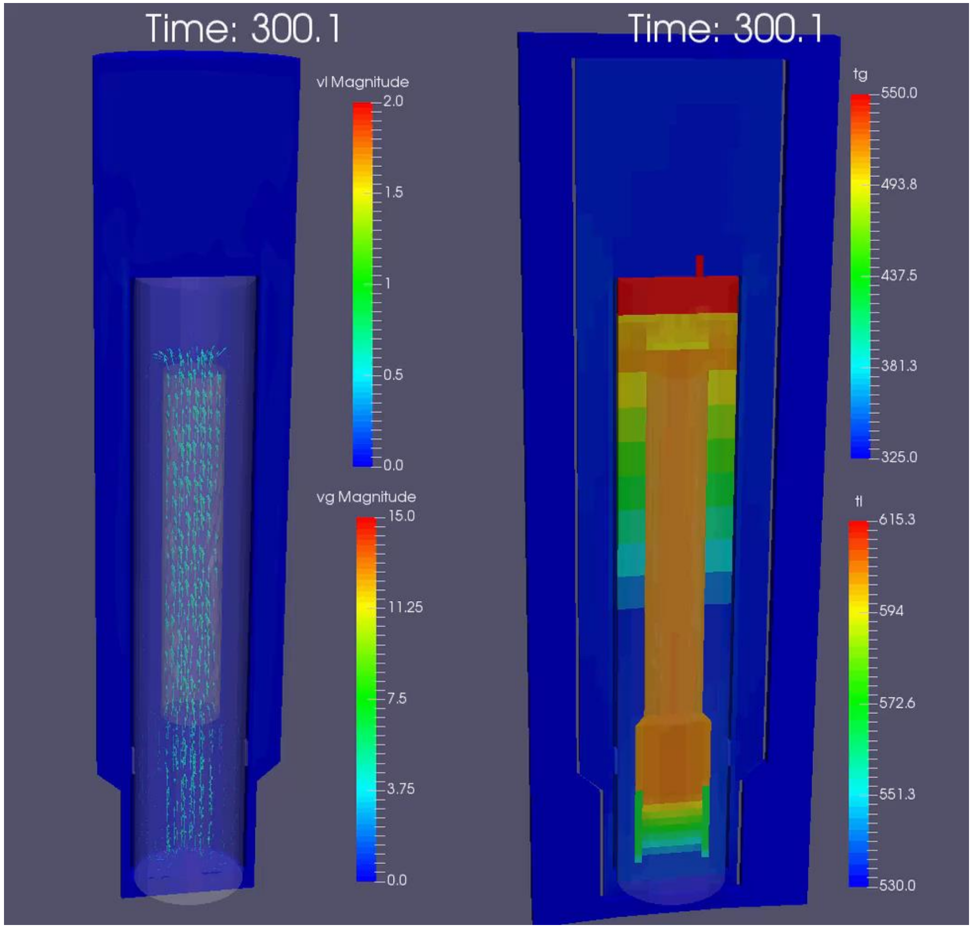
- *Opens at 300 sec. (forced)*
- *Henry-Fauske critical flow model*

Gas temperature in upper CV



Gas & Liquid Velocity

Gas & Liquid Temp.



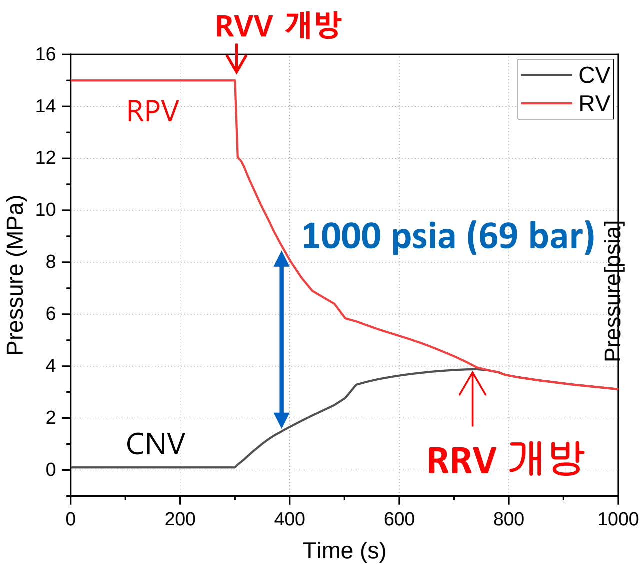
Condensation Rate (kg/s)

Liquid Fraction

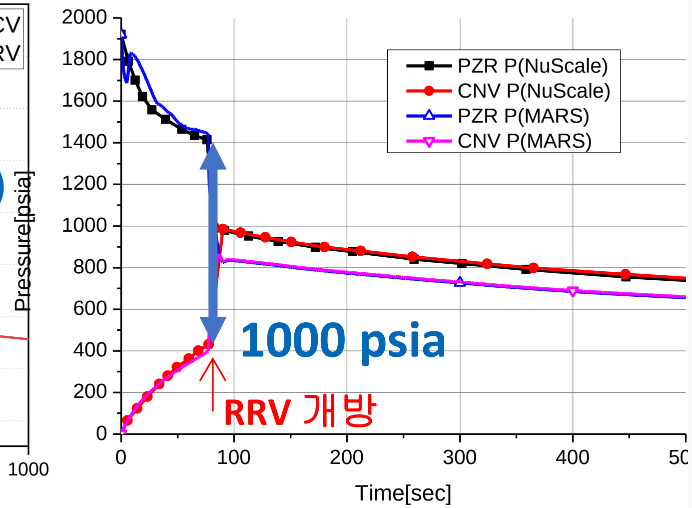
NuScale 3D LOCA 해석: RRV 개방

RRV Actuation

- *Open at 780s after pressure equalizaiton*
- *Recover the water level in the active core*



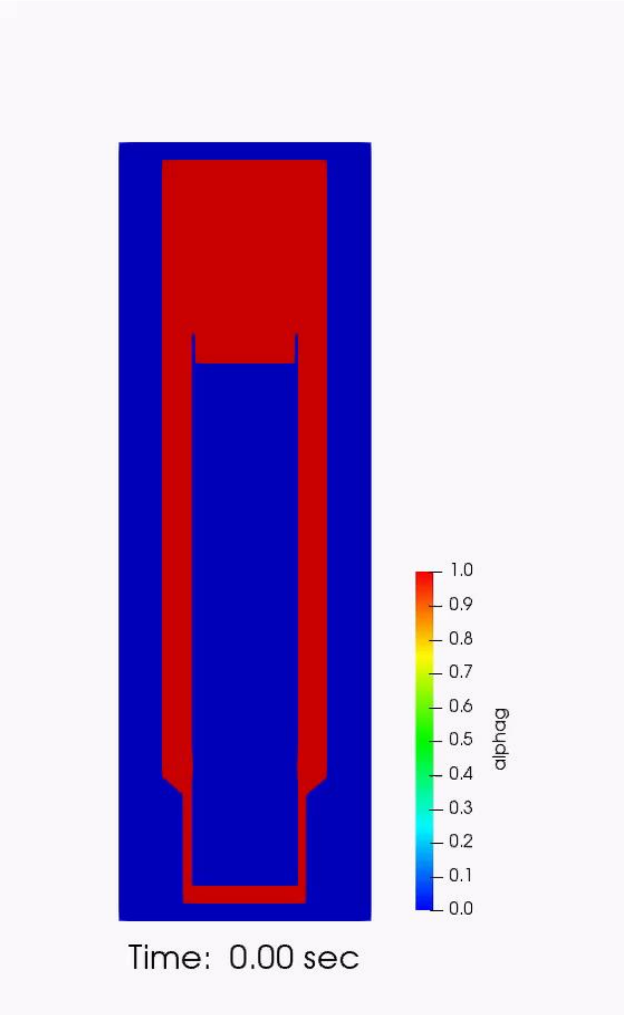
CUPID



NuScale & MARS*

Pressures in RV & CV

Liquid Fraction



* 임상규, 혁신형 SMR 개념설계를 위한 RCS-격납용기 연계 SB-LOCA 예비해석, 2021.02.24.

Summary and Future Works

4

Summary and Future Works

- 3차원 기기모델 개발
  - 3차원 격자생성기: Assembly 부터 Subchannel-scale 격자
  - MCP 모델
  - Valve 모델
  - PZR 형상 모의
  - 물리 모델: 내부구조물 압력강하, 부수로 모델, 2상유동 물리모델
- iSMR 3차원 사고조건 예비해석
  - 3차원 iSMR 정상상태 해석
  - RVV 개방 (공기 100% CV)
  - 열구조체 (RPV-CV 와 CV-UHS) 열전달
  - CV 벽면응축
  - RRV 개방에 따른 응축수 유입
  - 수조내 벽면열전달과 자연대류현상
- 노물리 코드와 증기발생기 코드 연계한 iSMR 3차원 정상/과도상태 해석
- MARU 플랫폼 기반 사고 시나리오 해석

경청해 주셔서 감사합니다.

cosinesj@kaeri.re.kr