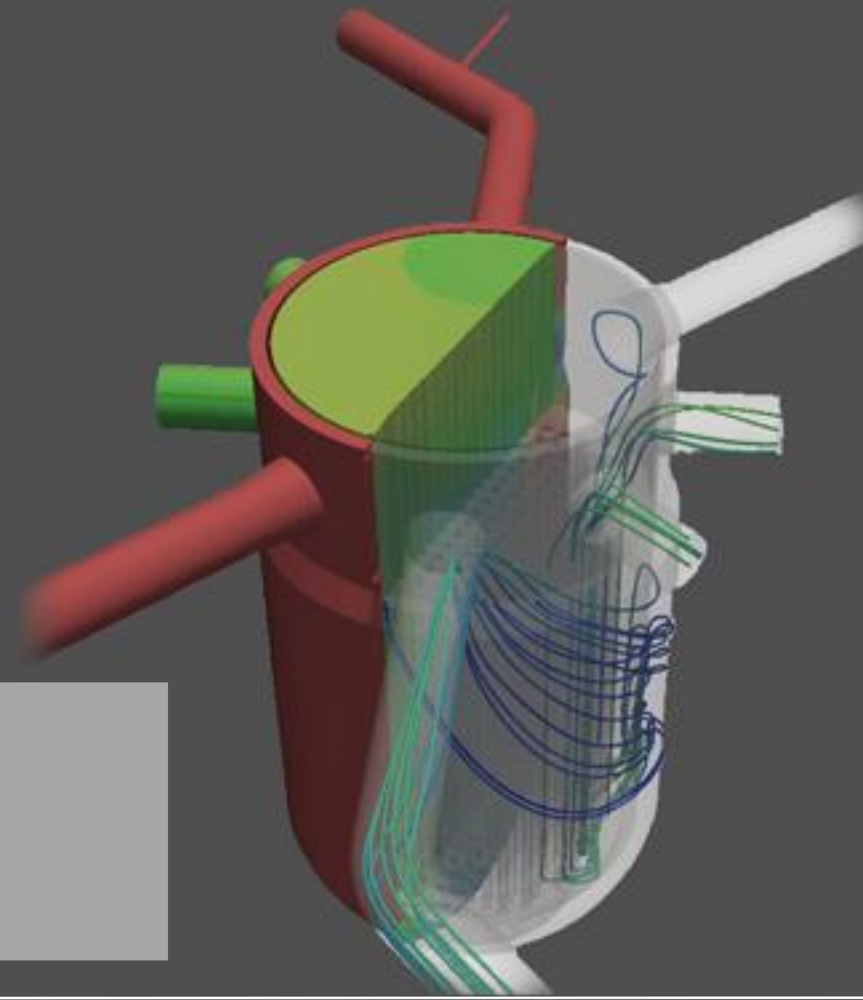


피동안전계통 다차원 열수력 해석

조윤제*, 이승준, 박익규, 이재룡, 도성주, 박소현, 윤한영

2022년 7월 8일

다물리계산과학연구실
한국원자력 연구원

Workshop on Passive Safety Features

CONTENTS

Main Features and Applications of CUPID

1

- Fast and Robust 3D 2-Phase Flow Solver
- Scalable Iterative Solver for a Large-scale Computing
- Multi-scale Simulation
- Multi-physics Simulation
- Etc. (RV, CT, SG, PAFS…)

Fast and Robust 3D 2-Phase Flow Solver

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

Commercial CFD

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

CUPID

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

Models and Correlations for Nuclear Reactors

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

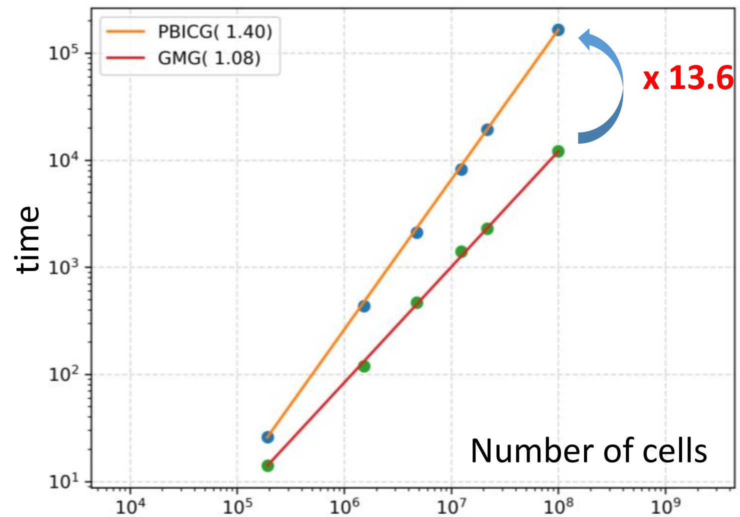
__RV__: Reactor Vessel, __SG__: Steam Generator, __CT__: Containment

Scalable Iterative Solver for a Large-scale Computing

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

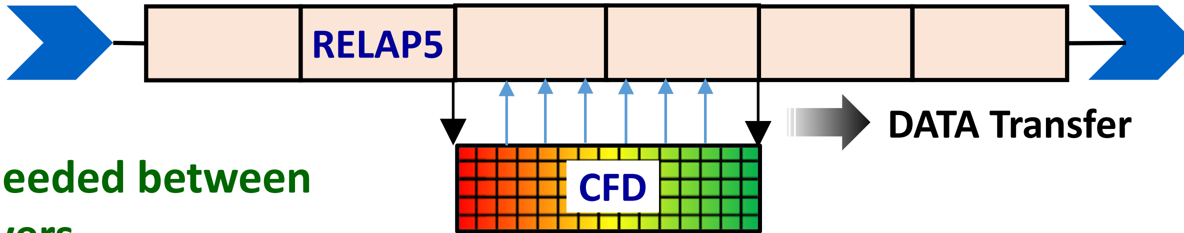
Number of Cells	time_pressure / time_total (%)
191,800	**78.8**
1,533,600	**75.7**
4,773,600	**81.6**
12,357,600	**86.2**
21,683,700	**90.2**
107,968,000	**92.9**

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



Multi-Scale Simulation for a Fast Transient

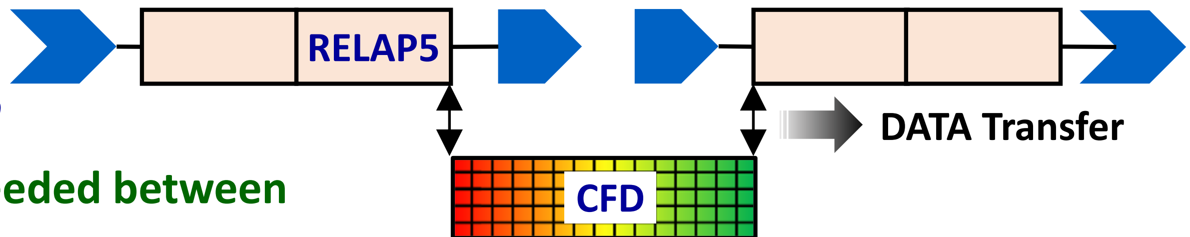
Domain Overlapping



Data Transfer is needed between
Two separate solvers

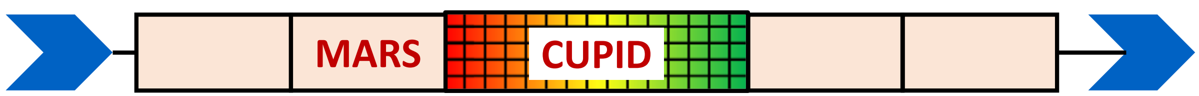
Ex)
RELAP5/
CFX,
FLEUNT,
STAR-CCM+

Domain Decomposition

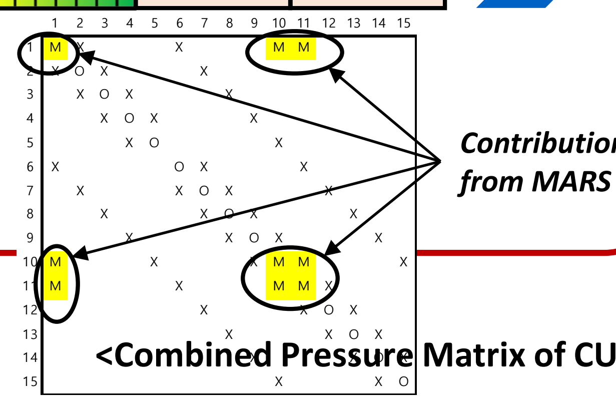


Data Transfer is needed between
Two separate solvers

Single Domain



Single pressure solver matrix
: No need for the Data Transfer
⟶ *Versatile application to transient problems*



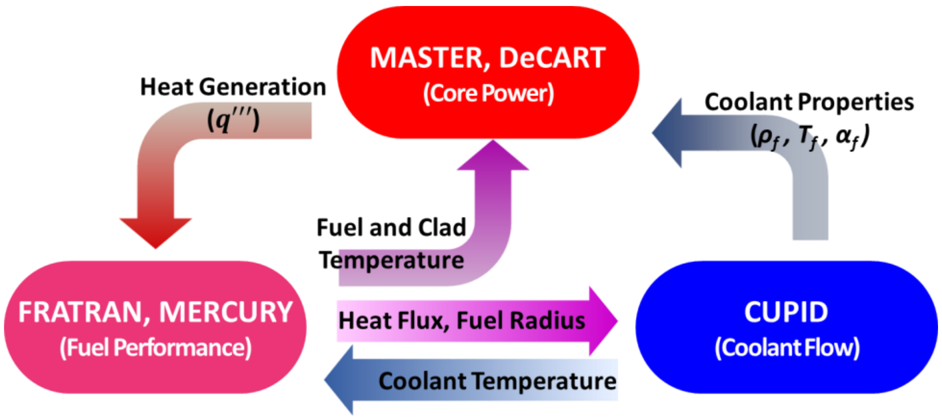
<Combined Pressure Matrix of CUPID>

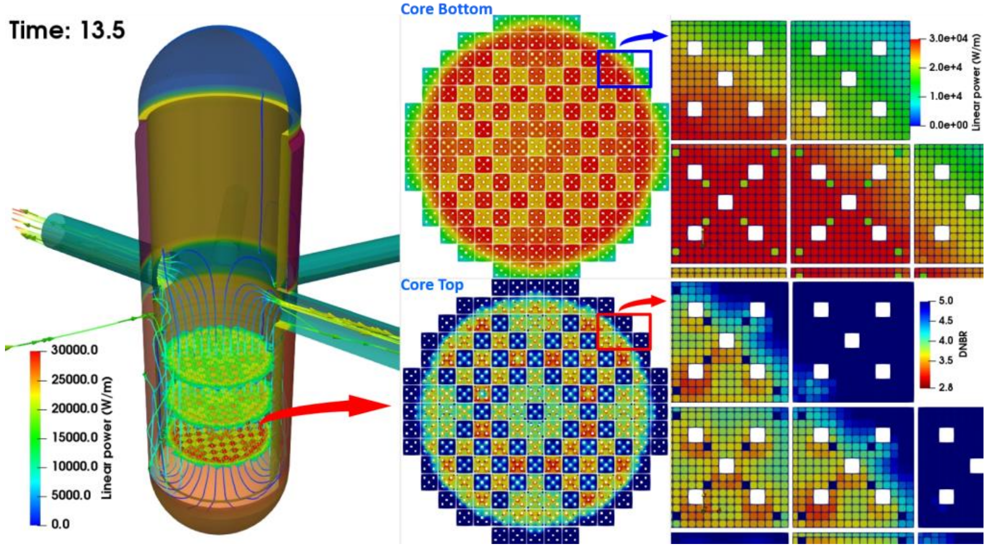
** I.K.Park et al., Annals of Nuclear Energy, 2013.*

Multi-Physics Simulation (TH/NK/FP)

Coupling of Multi-Physics Codes

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



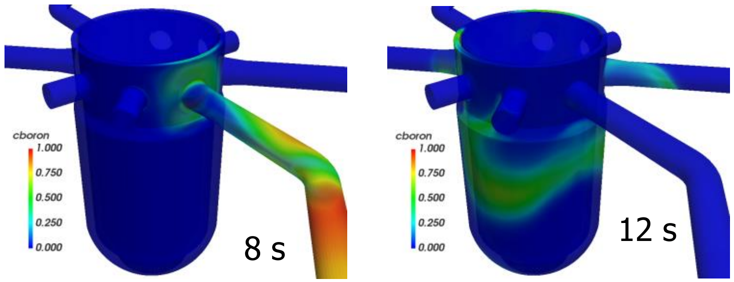


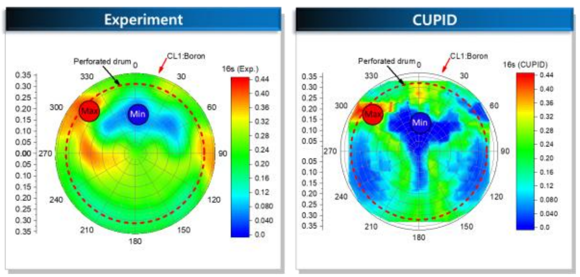
Full Core Pin-wise Simulation of the Steam Line Break Accident of OPR1000

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

Component Analysis in CFD-Scale

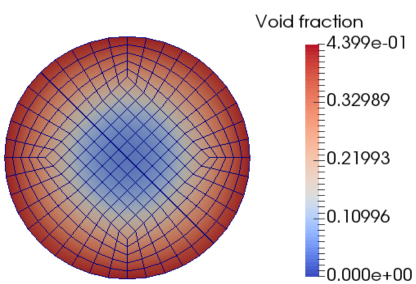
ROCOM (IAEA CRP)

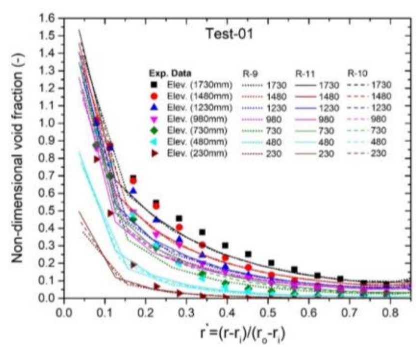




* Y.J.Cho et al., Nuclear Engineering and Design, 2019.

DeBORA Benchmark

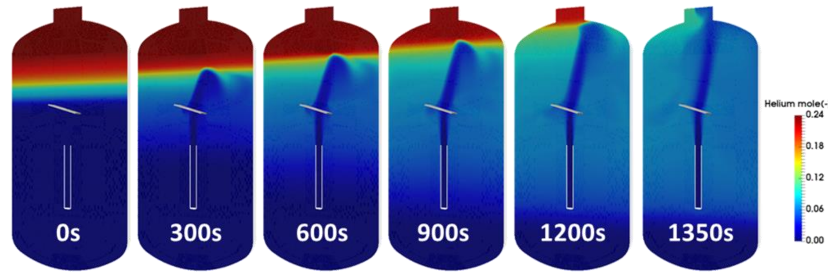




* Y. Alatrash et al., Nuclear Engineering and Technology, 2021.

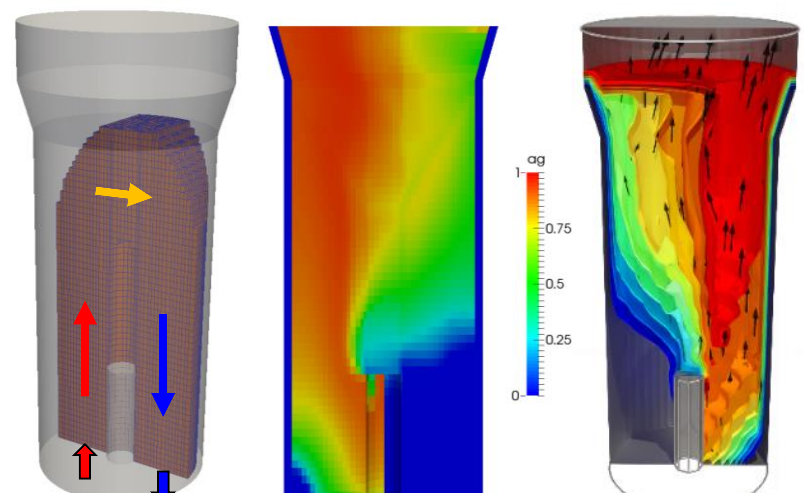
OECD/NEA HYMERES-2

- Helium layer erosion test using PANDA facility of PSI
- Vertical steam jet with obstructions



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

Steam Generator



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

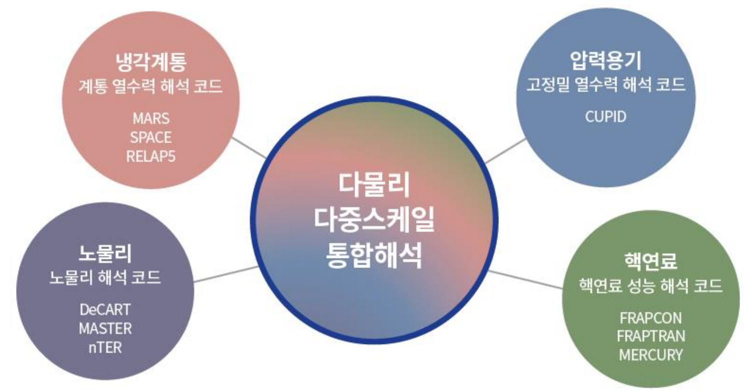
03
모든 것을 마루(MARU) 에 담다
Multi-physics Analysis Platform for Nuclear Reactor SimUlation

MARU는 누구나 쉽게 사용할 수 있는 통합해석 플랫폼입니다.

- 다물리/다중스케일 통합 해석기술 제공
- 계산과학 기술을 활용한 원자로 내부 현상 실시간 구현

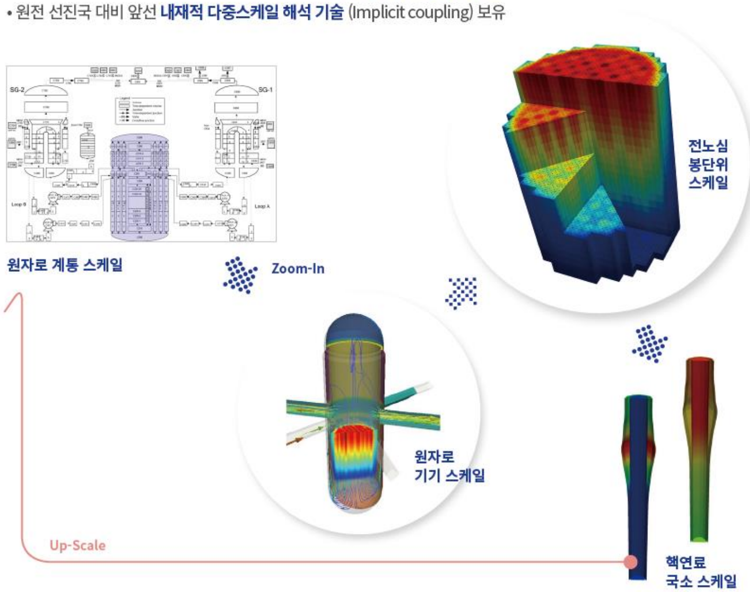
다물리 연계 해석 기술이 탑재 되었습니다.

- 원자로의 열수력 이상유동 / 노물리 동특성 / 핵연료 연소 현상의 상호작용을 정밀 예측 가능
- 전노심 봉단위의 고해상도 다물리 연계 해석 기능 지원



다중 스케일 해석 기술을 제공 합니다.

- 원자로 내의 해석 영역과 모의해야 할 현상에 따라 **계통 / 기기 / 봉단위 / 국소** 스케일 해석이 가능
- 원전 선진국 대비 앞선 **내재적 다중스케일 해석 기술** (Implicit coupling) 보유



비전문가도 쉽게 사용할 수 있습니다.

- MARU 는 원자로 통합 해석을 위한 Turnkey solution입니다.
- 언제, 어디서나, 한 번의 클릭으로 원자로에 따른 격자 생성부터, HPC 기반 다물리/다중스케일 계산 및 후처리 까지 MARU를 통해 수행 가능합니다.

WORK FROM ANYWHERE
MARU

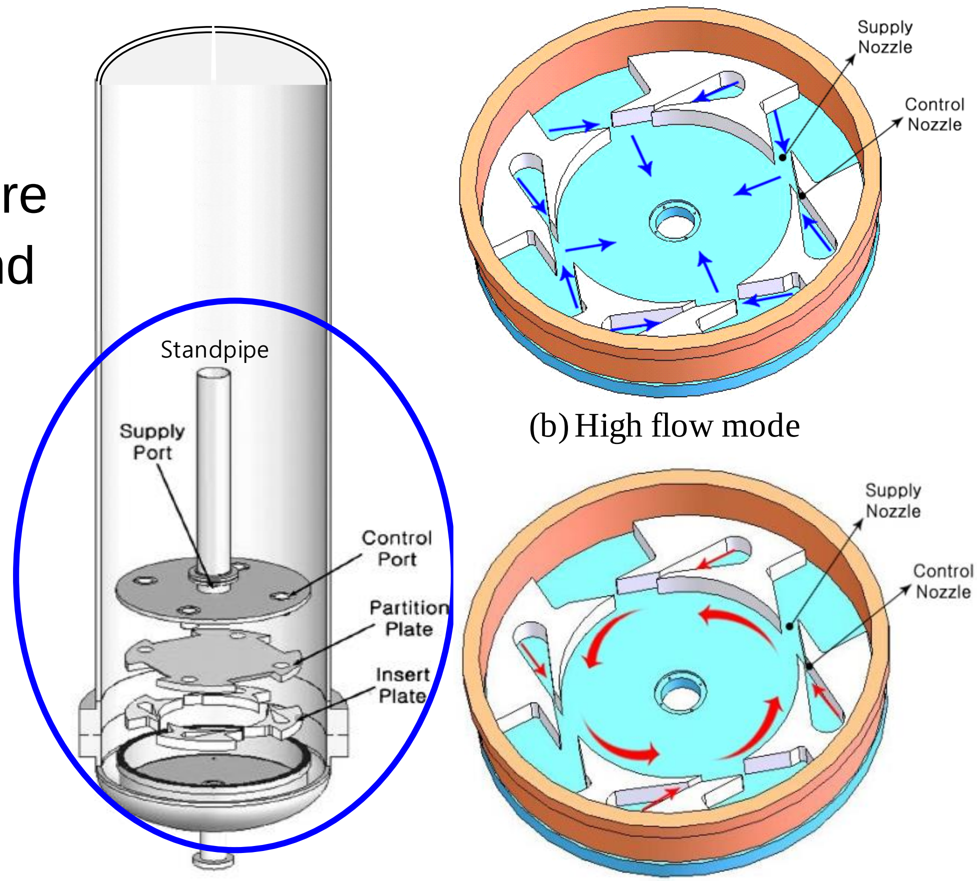
Fluidic Device

2

Concept of Fluidic Device

Advanced SIT of APR1400

- **Efficient use of the ECC water in the SIT**
- **Fluidic Device**
  - Complicated internal structure
  - Standpipe, supply nozzle and control nozzle
  - A vortex flow control device



(a) The internal structure (b) High flow mode (c) Low flow mode

Multi-Scale Analysis

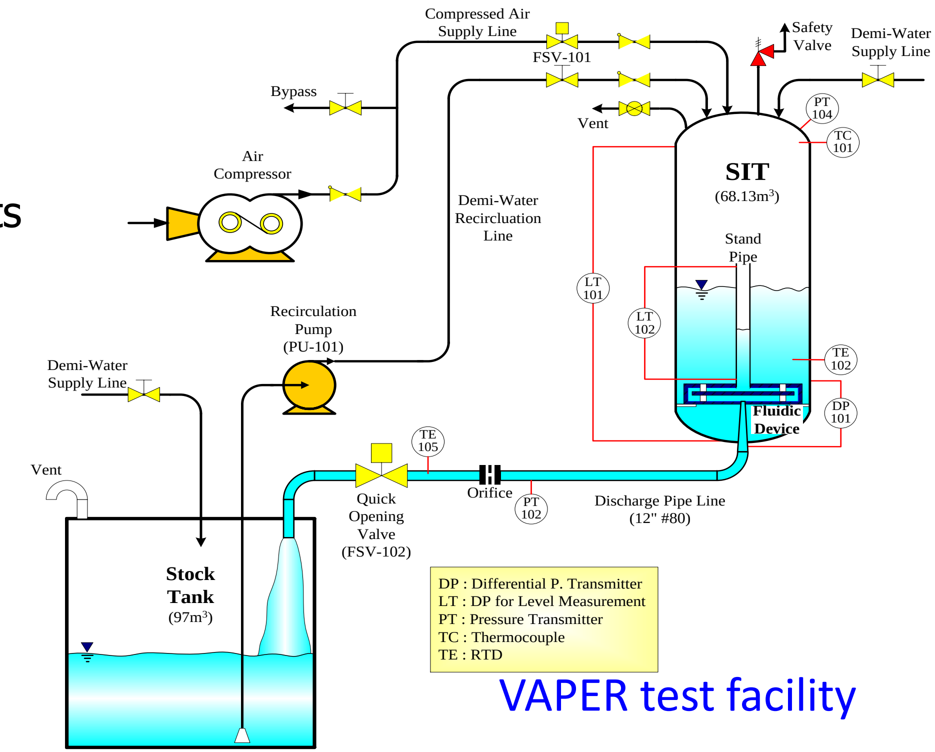
Analysis of VAPER Experiment

MARS (System Scale)

- System scale calculations
  - ✓ Using “accumulator” components
  - ✓ Using “Pipe” components

CUPID (CFD Scale)

- CFD-scale (RANS-scale) local pressure drop calculations for Control/Supply Nozzle



VAPER test facility

CUPID (Component Scale)

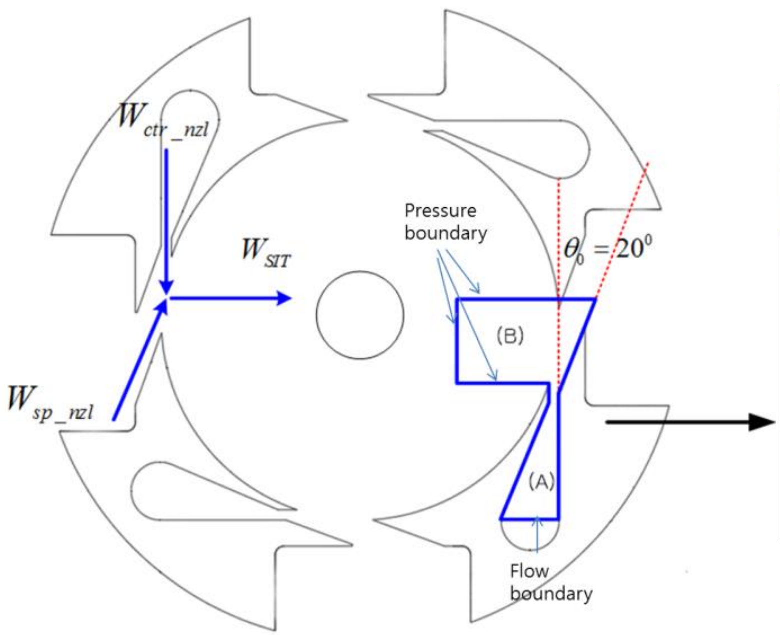
- Flow resistance models
  - ✓ simplifying the flow in the vortex chamber
- Calculation using a component-scale coarse grid

CFD Analysis

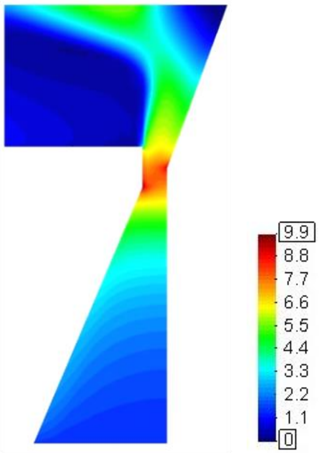
CFD-scale Analysis of control/supply nozzles

➢ $R_{control\_nozzle}$ and R_{supply_nozzle} in CFD scale

- 2D calculations for the supply and control nozzles
- Calculate the pressure drop through the control nozzle



-Inlet mass flow: 40.3 kg/s
-Outlet pressure: 1 bar



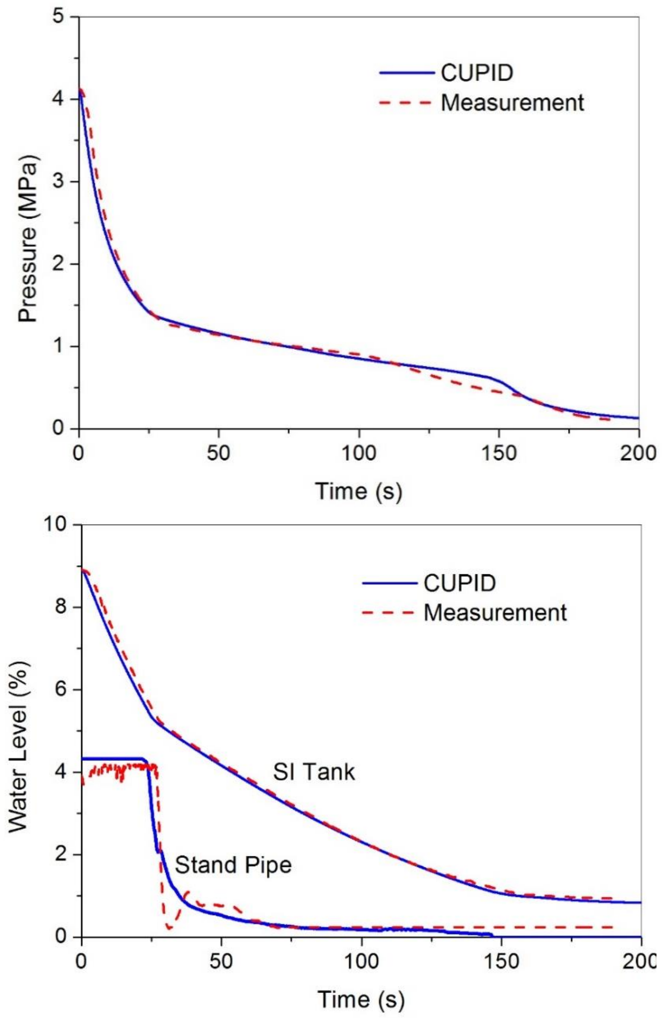
$\Delta P_{control\_nozzle}$ = 0.02 Mpa
→ $K_{control\_nozzle}$ = 4.49

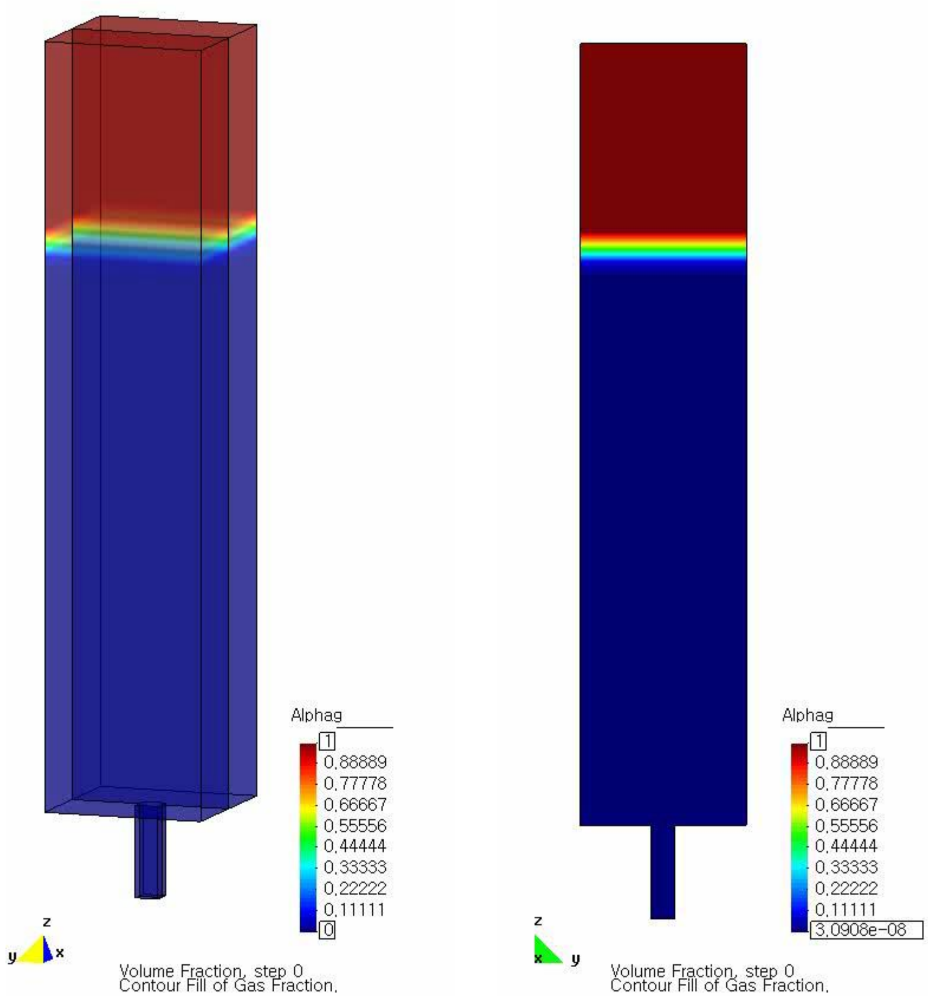
$$\cos\theta_0 \cdot W_{supply\_nozzle} = W_{control\_nozzle}$$

$\theta_0 = 20^o$ → K_{supply_nozzle}

Calculation Results

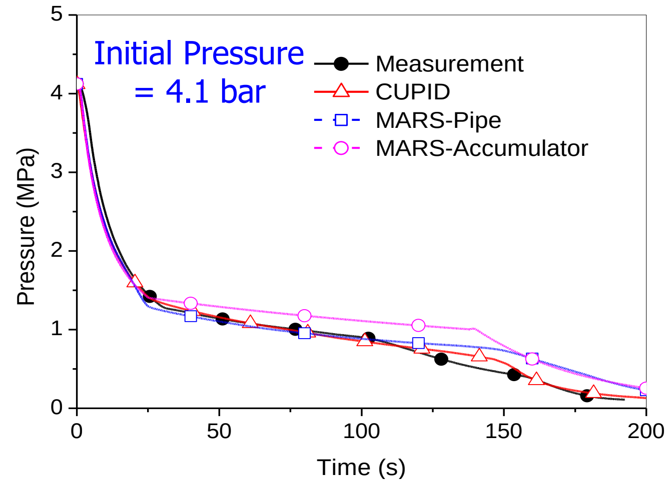
Comparison to VAPER measurement

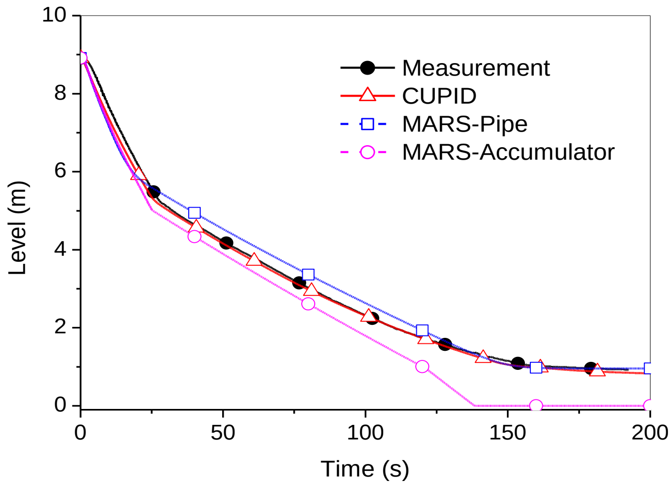


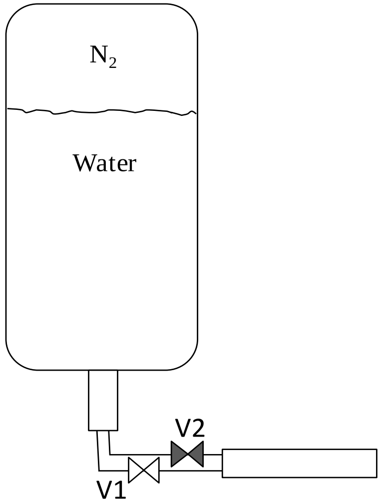


Code-to-Code Validation

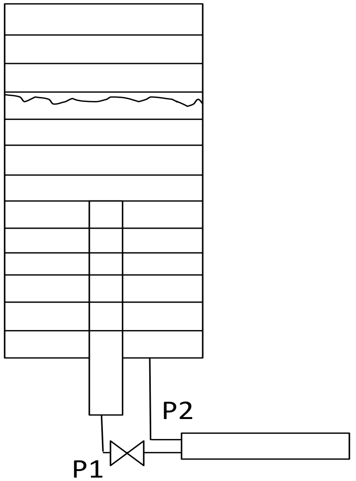
Comparison of MARS and CUPID







MARS (accumulator)



MARS (Pipe)

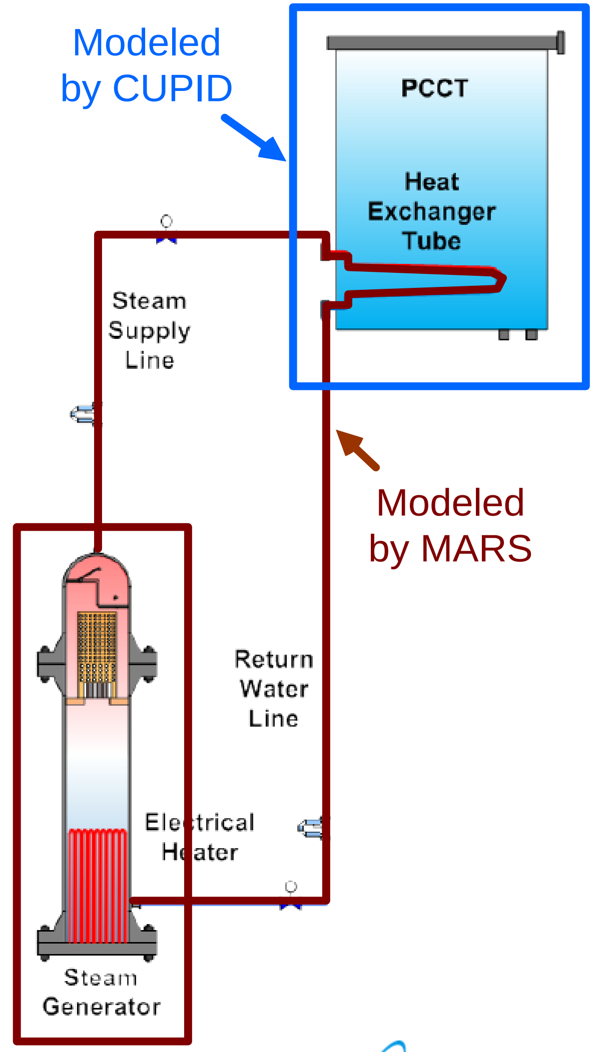
3

APR+ PAFS

Analysis of PASCAL Experiment

Multi-scale T/H analysis

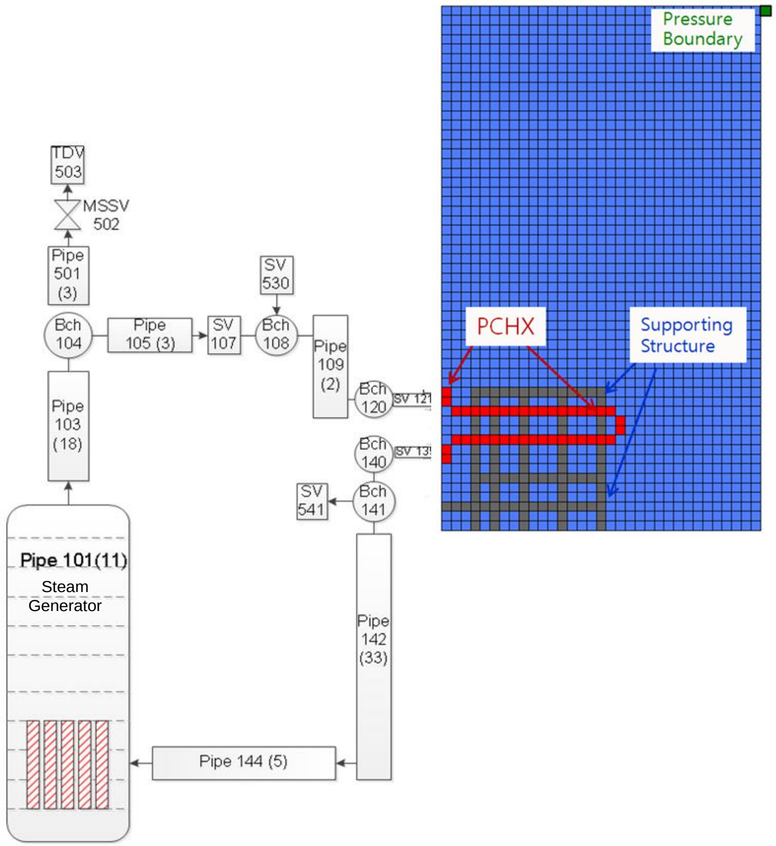
- **Long transient phenomena: ~30,000 s**
- **Coupled MARS-CUPID Simulation**
  - MARS
    - ✓ Steam generator and steam supply line
    - ✓ Two-phase flow in the heat exchanger tube
    - ✓ Water return line
  - CUPID
    - ✓ Two-phase flow in the PCCT
- **Coupling interface**
  - Heat exchanger tube outer wall
  - No flow exchange
  - Heat structure coupling

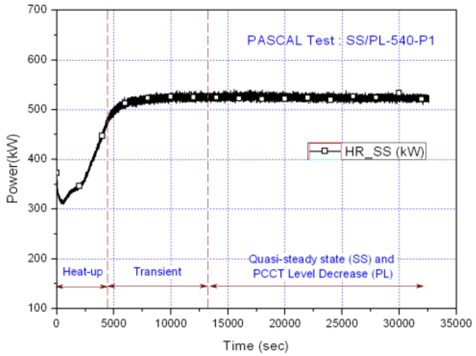


Calculation Conditions

Multi-scale analysis

- **MARS: 1D analysis**
  - 40 nodes for the horizontal U-tube
- **CUPID: 2D analysis**
  - With the porous media model
  - Rectangular mesh (1815, 33x55)
- **Simulation time: 28,800 s**
- **Boundary conditions**
  - Pressure boundary (atmospheric)
  - Steam generator heater power

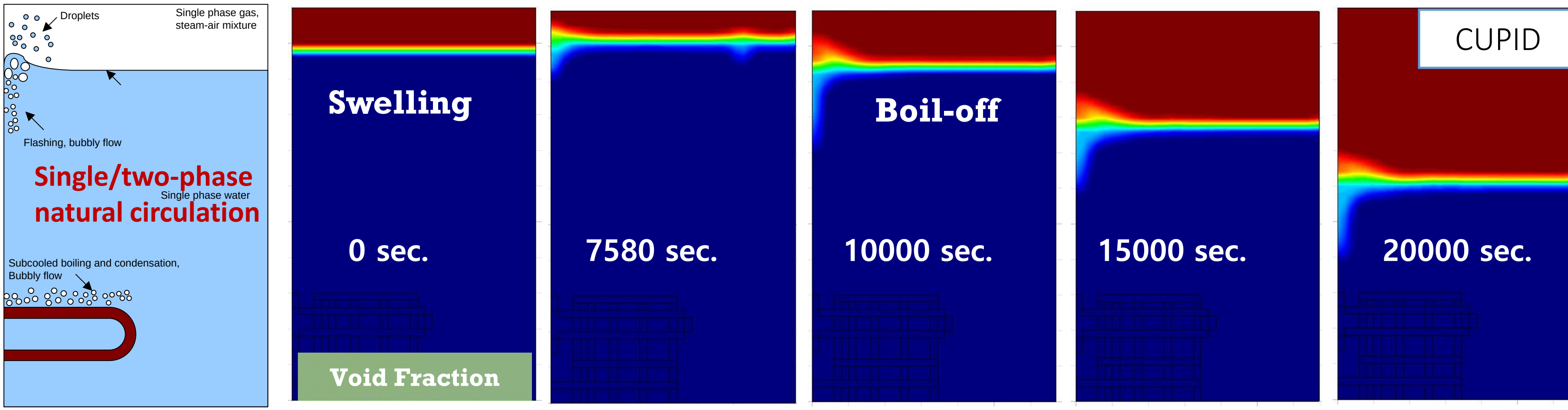




Calculation Results (1/2)

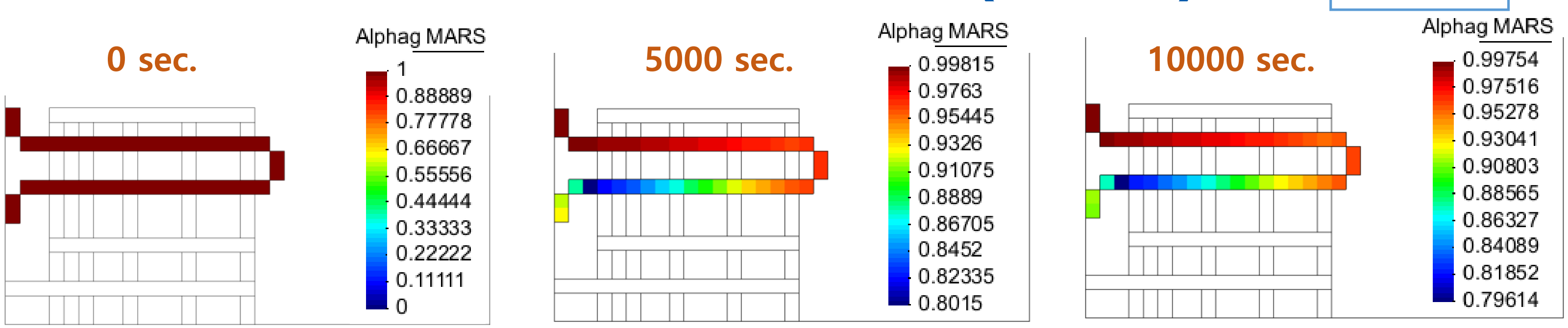
CUPID-MARS Analysis of PASCAL

➢ PCCT boil-off transient (CUPID)



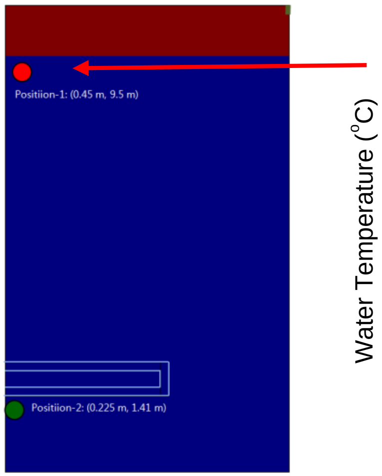
➢ Condensation inside PAFS tubes (MARS)

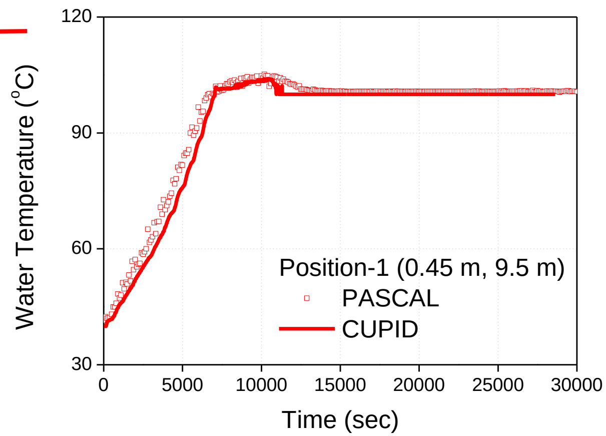
MARS

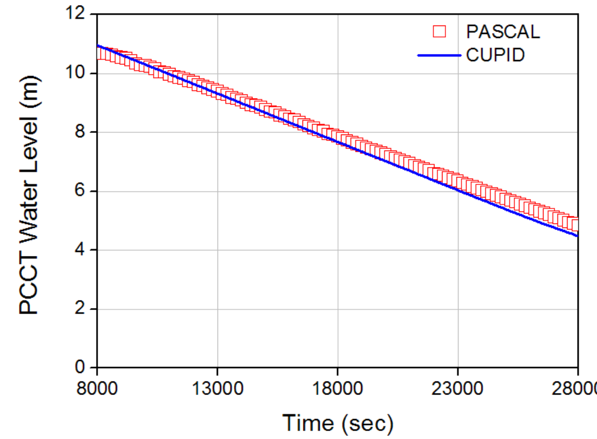


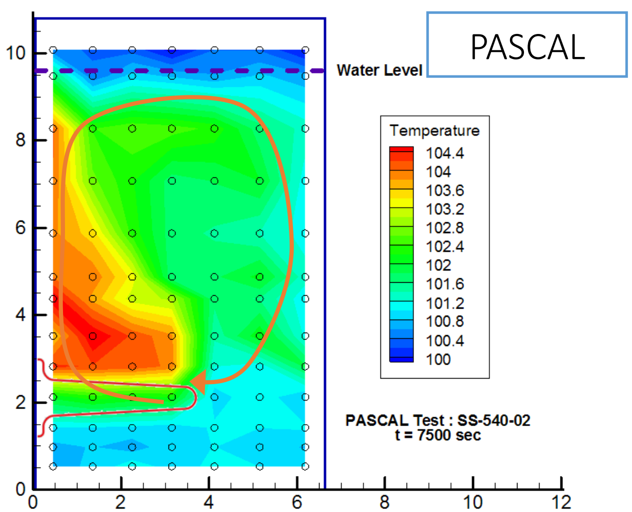
Calculation Results (2/2)

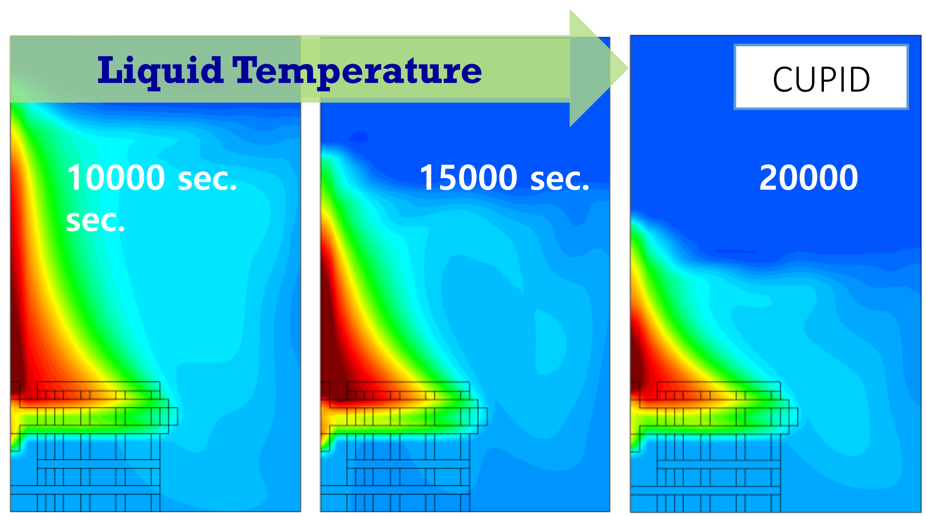
Comparisons to experimental data











Siphon Break in Research Reactor

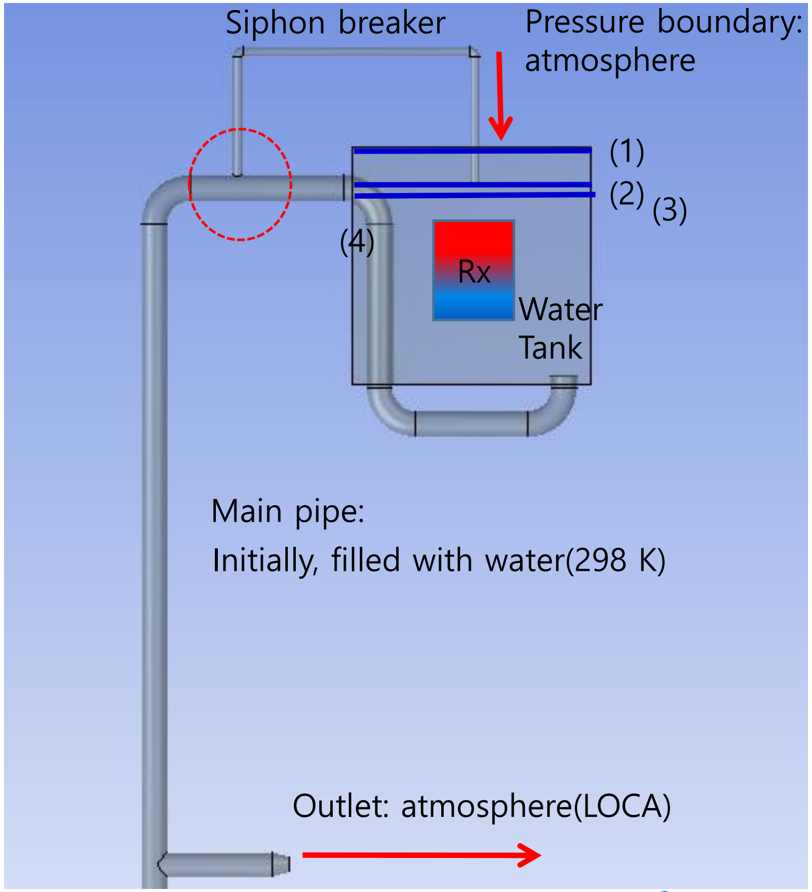
4

Siphon Break in Research Reactor

Schematic of Research Reactor

LOCA at outlet

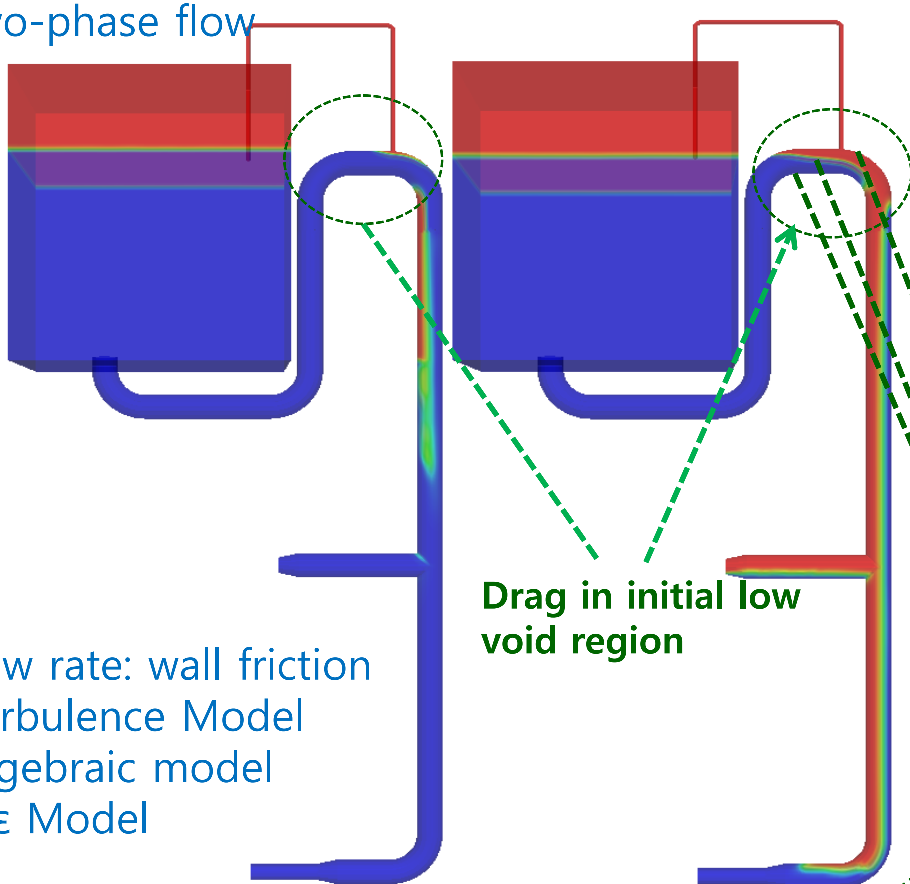
- Water level decreases (1)
- Siphon breaker uncovered (2)
- Air supplied into main pipe through siphon breaker
- Water drain stopped



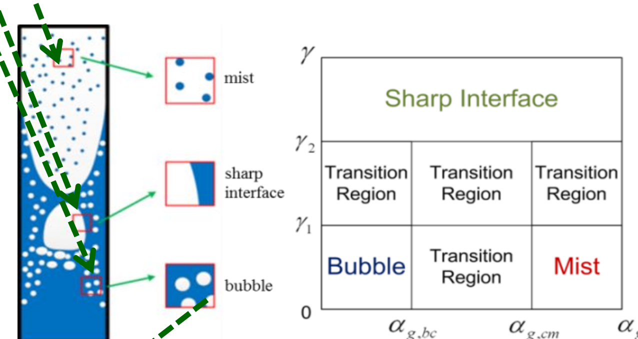
Mechanism of Siphonage Flow and Break

Siphonage Flow Rate and Siphon Break

1) Two-phase flow



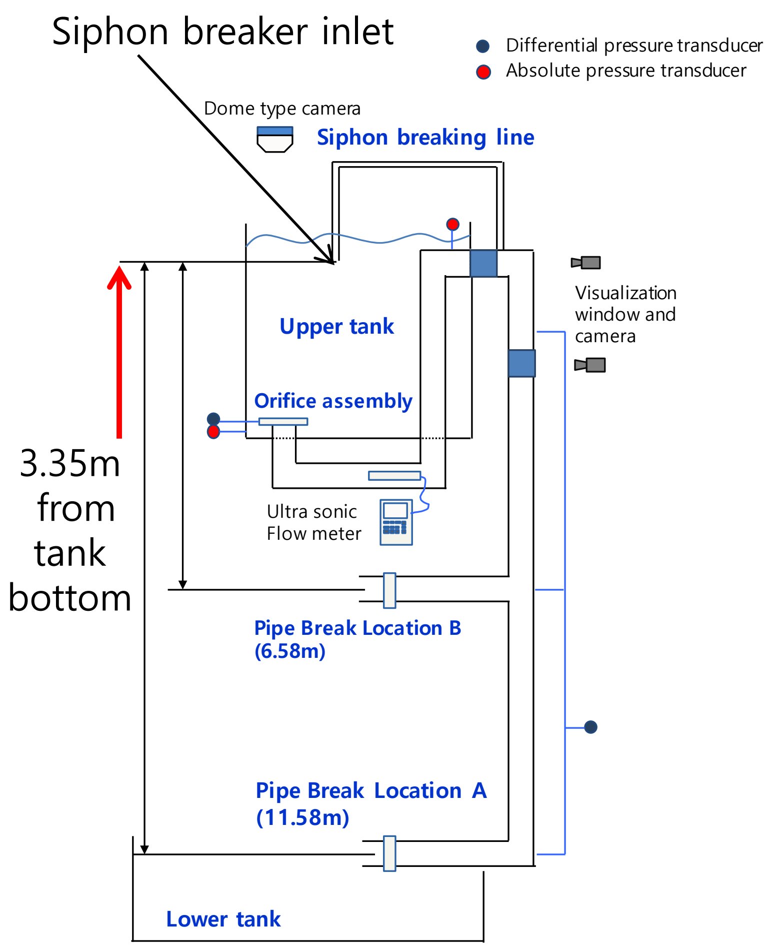
2) Flow rate: wall friction
- -Turbulence Model
- -Algebraic model
- -k-ε Model



3) Siphon break: interfacial drag
- IAC model
- Drag model

Postech Siphon Experiment

Schematic of Test Facility

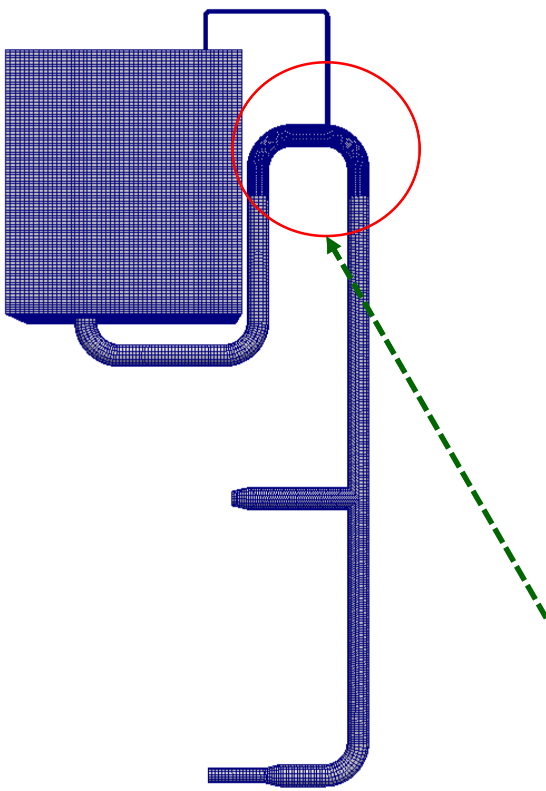


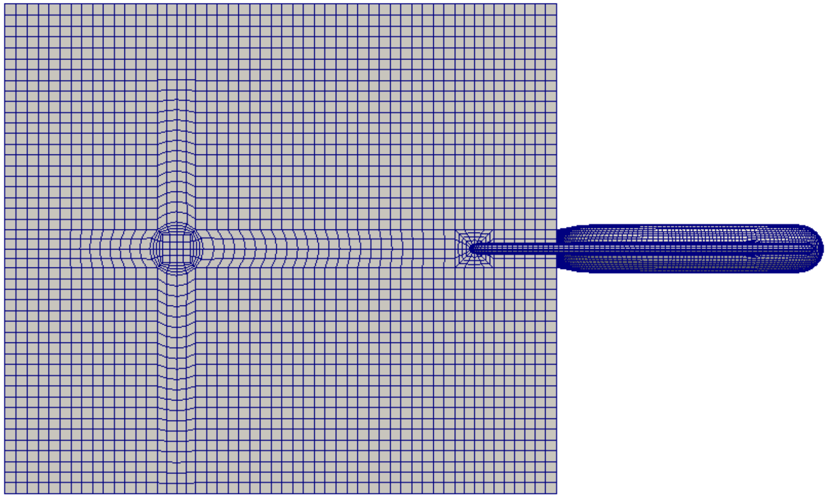
Siphon Break Line Size (inch)	Undershooting Level (cm)	
	Pipe Break Location A (10.58 m)	Pipe Break Location B (6.58 m)
2.5	0.33	0.21
2.0	0.79	0.30
1.5	1.46	0.52
1.0	Fail to break the siphon phenomena	2.05
0.5	Fail to break the siphon phenomena	Fail to break the siphon phenomena

CUPID Calculation Setup

Mesh Generation

- Node, Cell, Face:349207, 328400, 1005120
- Tank shift to the right to save mesh & 14 CPUs were used

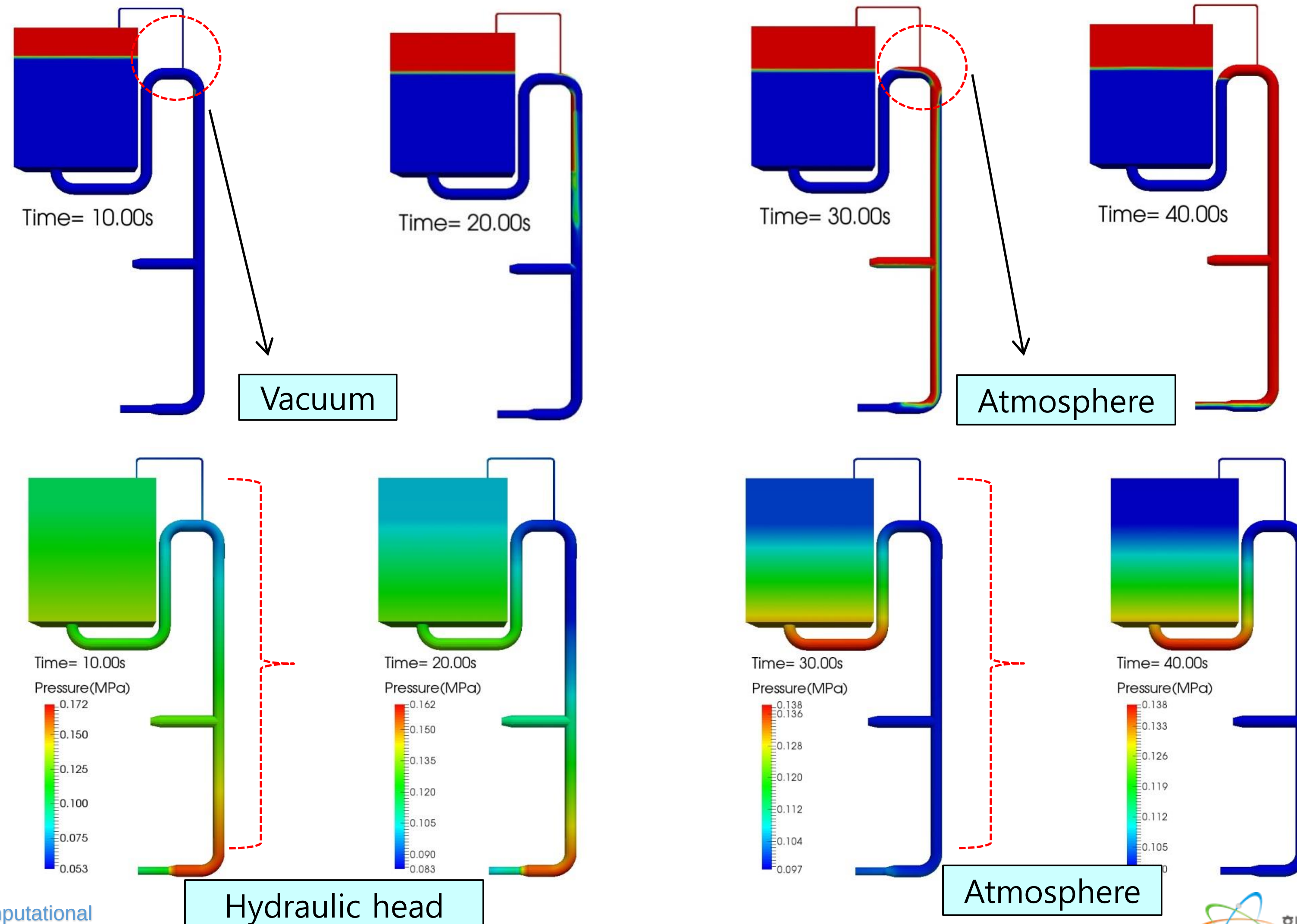




Fine mesh in siphon break region

Calculation Results

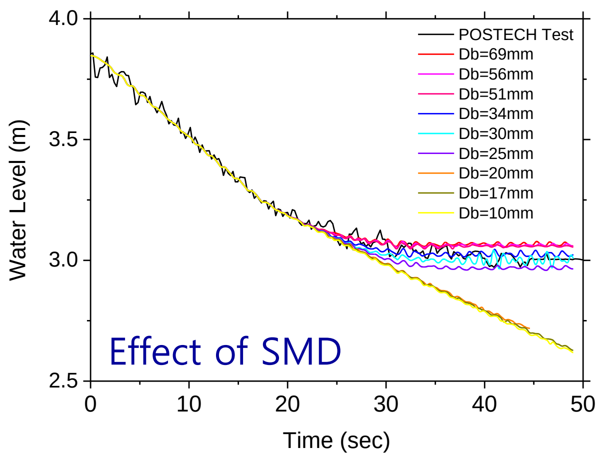
Transient of void fraction and pressure

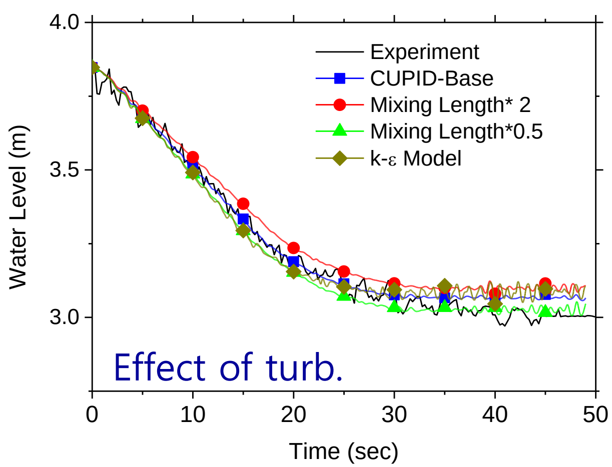


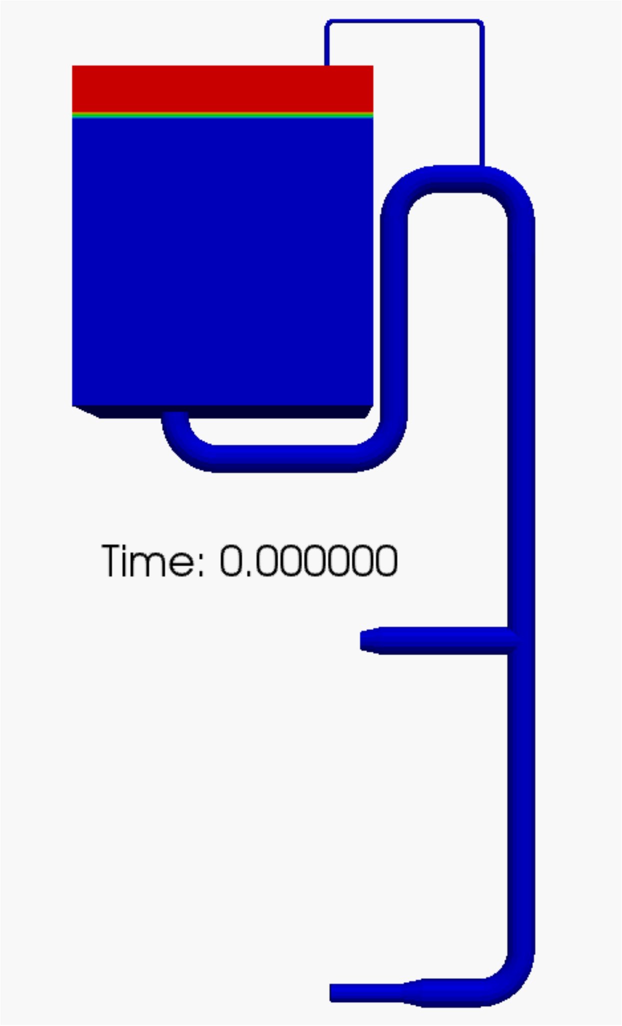
Sensitivity Analysis

Effect of Physical Models

- Interfacial drag & SMD
- Turbulence model







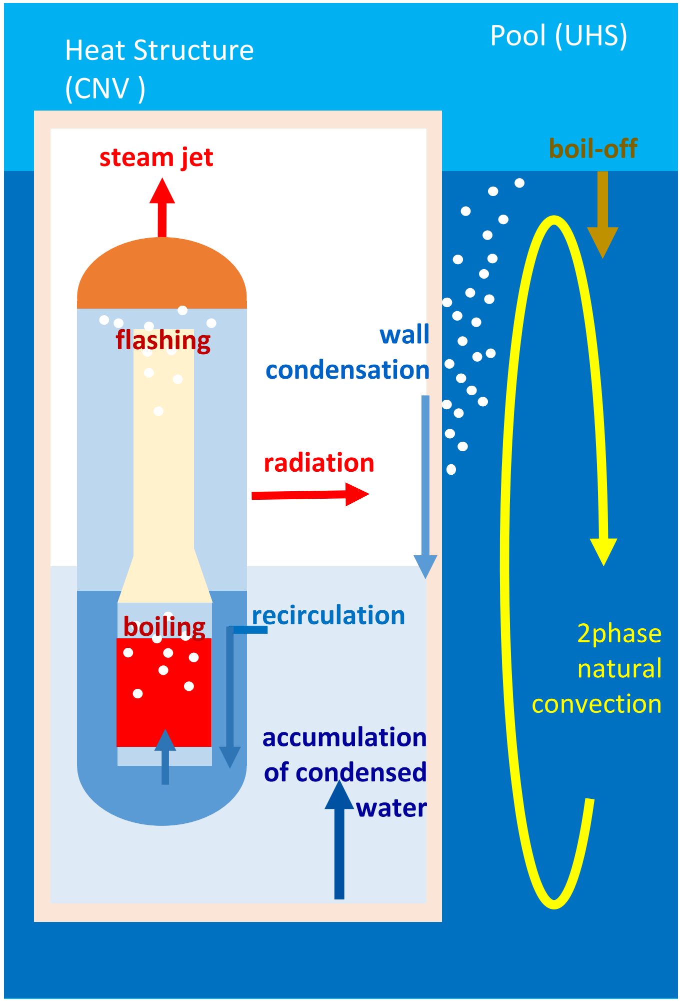
Application to SMR Conceptual Design

5

- Natural Circulation of iSMR
- 3D Full Core Analysis Using Subchannel Model
- Conceptual SMR LOCA Analysis

3D Flows in Two-Phase Natural Circulation

Two-Phase Transient Phenomena under Accident Condition



➢ Reactor Pressure Vessel (RPV) Side
- LOCA (RVV, RRV)
- Boiling and flashing

➢ Containment Vessel (CNV) Side
- Steam condensation on heat structure surface
- Radiation heat transfer

➢ Pool Side
- Two-phase natural convection
- Boil-off

Two Natural Circulation Modes in One System
- **Mode1: Two-phase natural convection in a pool**
- **Mode2: Recirculation via RVV & RRV**

Required Models for iSMR

Component Models

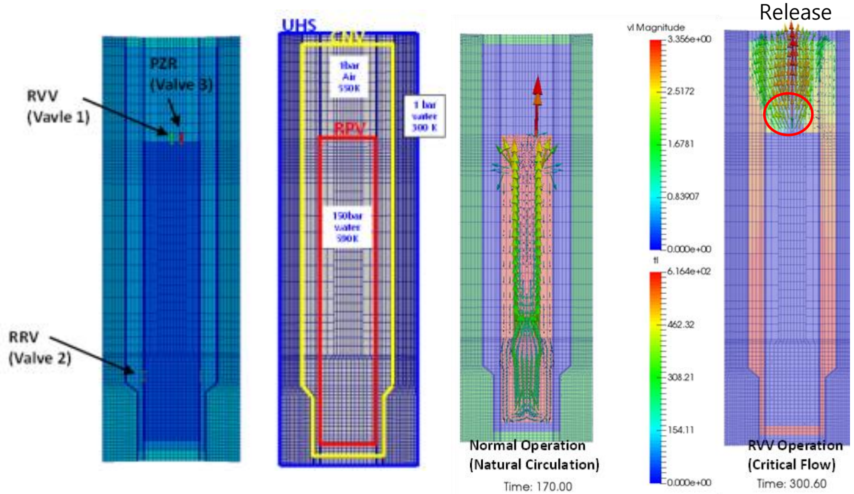
- **Valve(RVV, RRV)**: Blockage, on/off timing control algorithm
- **MCP** : Homologous curve MCP model, flow rate control by RPM
- **Steam generator**: pressure drop model (primary side), heat transfer coefficient model inside/outside helical tube

Physical Models

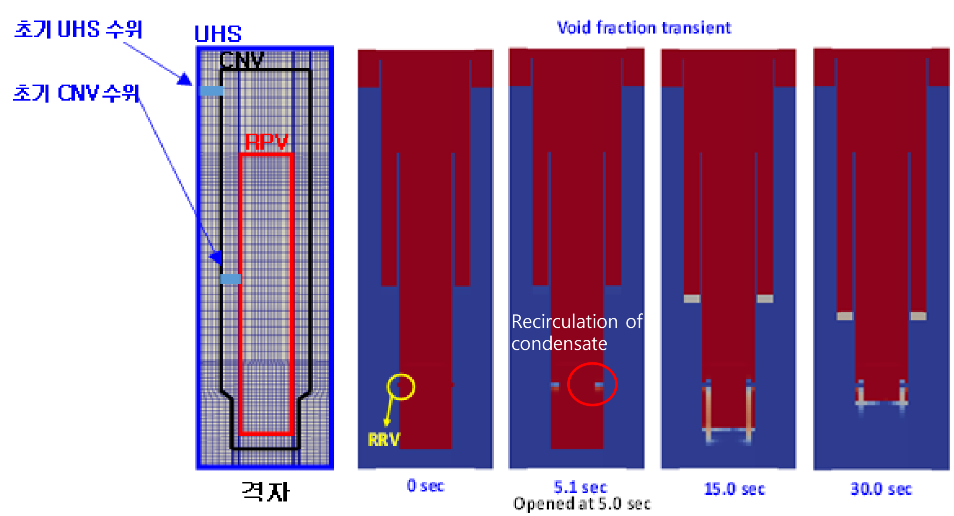
- **Porous model**: flow resistance model outside Rx core
- **Sub-channel model**: pressure drop model, turbulence mixing model
- **Condensation model** in containment vessel
- **Radiation model** in containment vessel
- Critical flow: 2-phase critical flow model based on cell-centered mesh system

Model Development (1/5)

Valve(RVV, RRV) Model

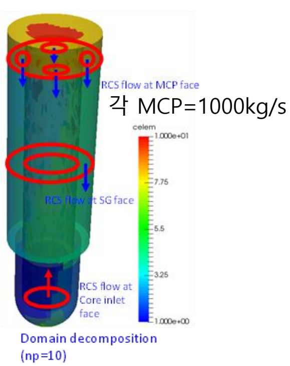


[Test calculation of NuScale RVV]



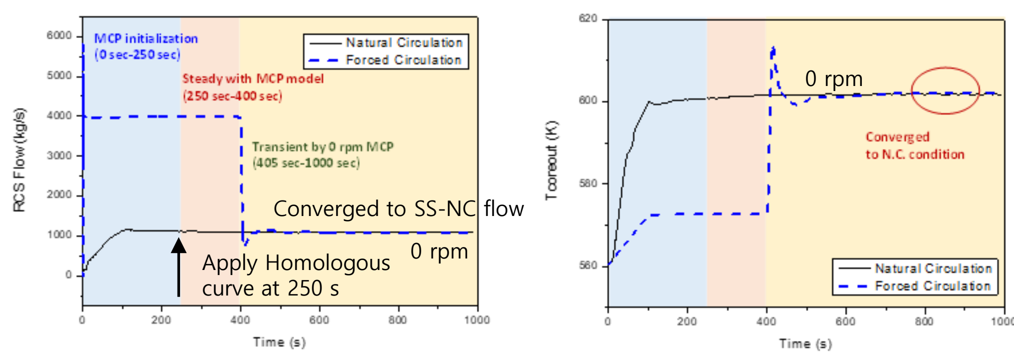
[Test calculation of NuScale RRV]

MCP Model



[Locations 4 MCPs]

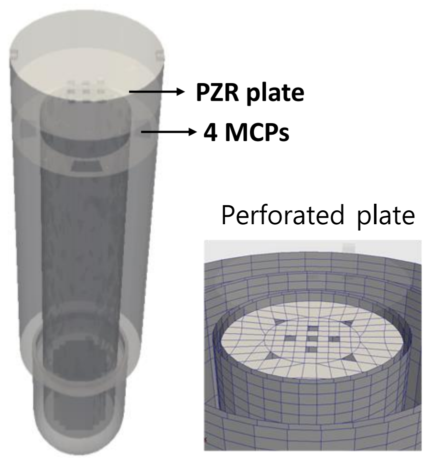
[Flow pattern by MCP]



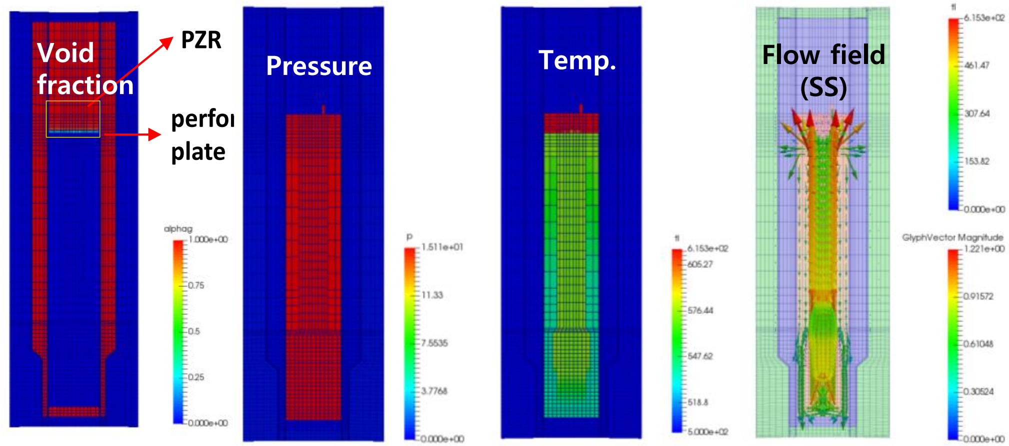
[Transition from forced- to natural- convection by MCP RPM control]

Model Development (2/5)

PZR Model



[PZR plate modeling]



[Steady state calculation with NuScale condition]

Porous Approach-based Inner Structure Model

- Porosity and permeability: flow volume area in governing eqs.
- Three pressure drop models: similar calculation results

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

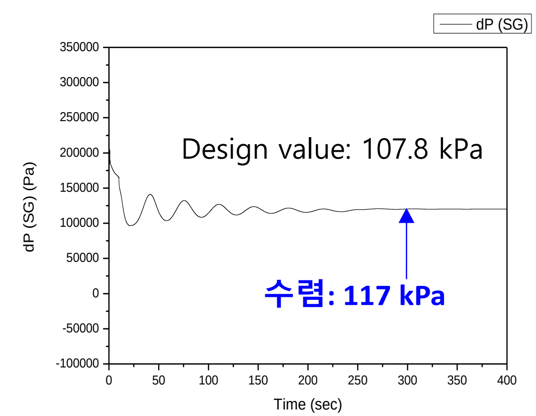
1.SS state-based K-factor

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

2. P-based adaptive K-factor

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

3. Friction-based physical model



[Calculation result with Friction-based model]

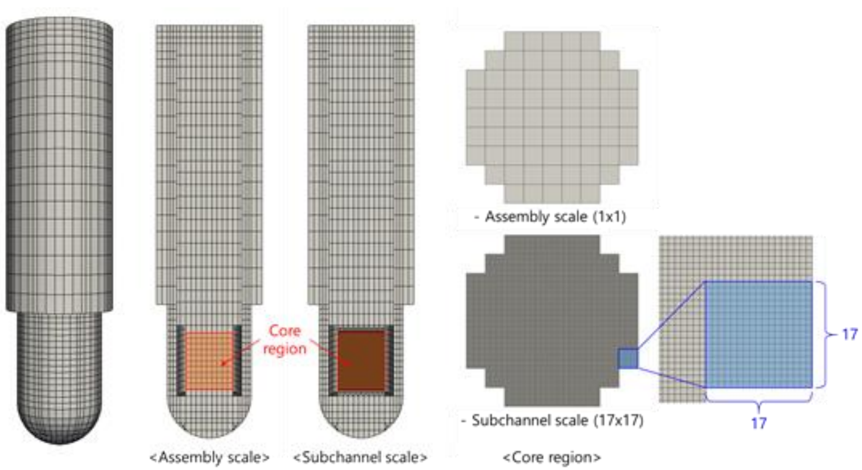
Model Development (3/5)

Reactor Core TH Model

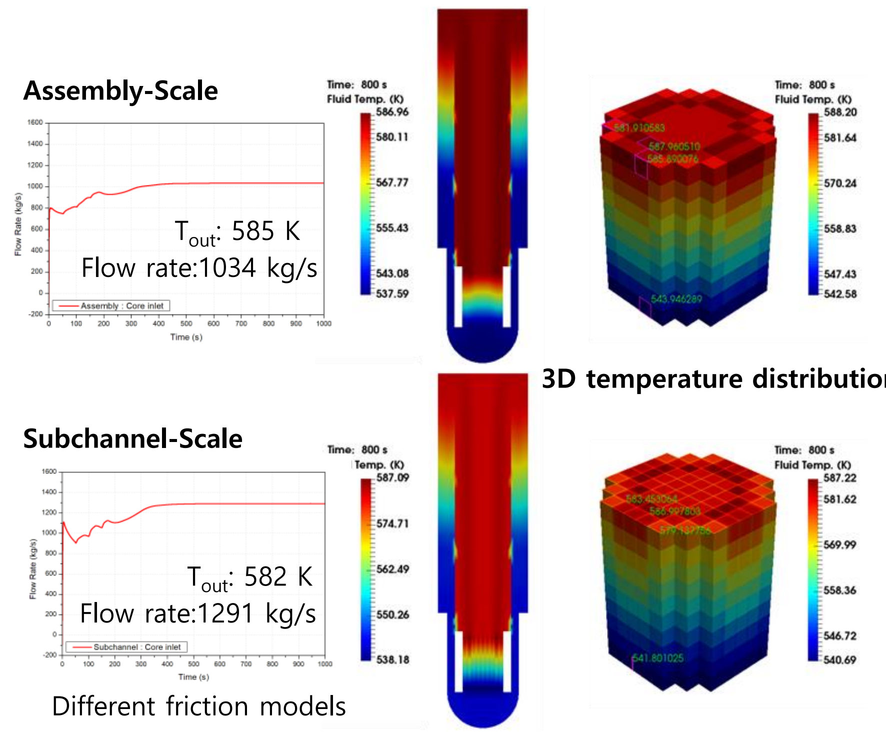
- Assembly-scale: reactor core model package (PWR based)
- Subchannel-scale: reactor core model package (PWR based) + subchannel model (friction & turbulence mixing models)

[Calculation conditions]

Initial condition	Pressure : 150 bar
	Fluid temperature : 560 K
	Fuel rods temperature : 594 K
Volumetric heat	Core : distributed into each channel, Total 259 MW
	SG : simple model, Total 259 MW
Simulation time	1,000s



[Assembly-, Subchannel-scale Full-core meshes]



[Assembly-, Subchannel-scale SS calculation results]

Model Development (4/5)

Condensation model

- ***Colburn-Hougen model, modified shah(KAIST data)***
- ***Effect of NC gas property***
  - *Diffusion coefficient: function of molar weight, atomic diffusion volume*

$$D_{AB} = \frac{0.00266 T^{3/2}}{P M_{AB}^{1/2} \sigma_{AB}^{2} \Omega_D}$$

where D_{AB} = diffusion coefficient, cm^2/s
T = temperature, K
P = pressure, bar
σ_{AB} = characteristic length, Å
Ω_D = diffusion collision integral, dimensionless

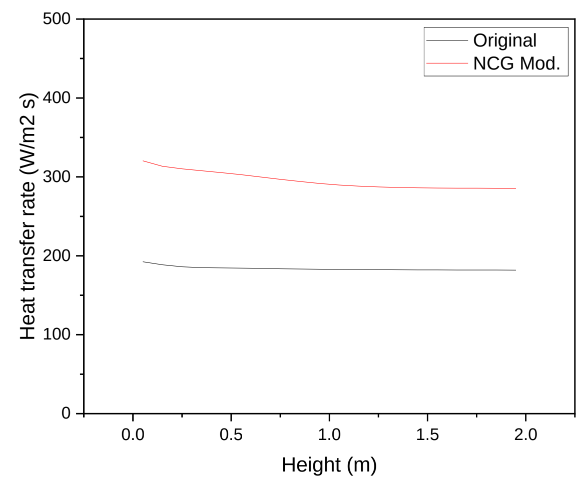
coef_diff(1)	3.416486781592469D-005

Steam+air

coef_diff(1)	7.364200245279183D-005

Steam+air+H_2

Effect of diffusion coefficient on condensation HTC



Model Development (5/5)

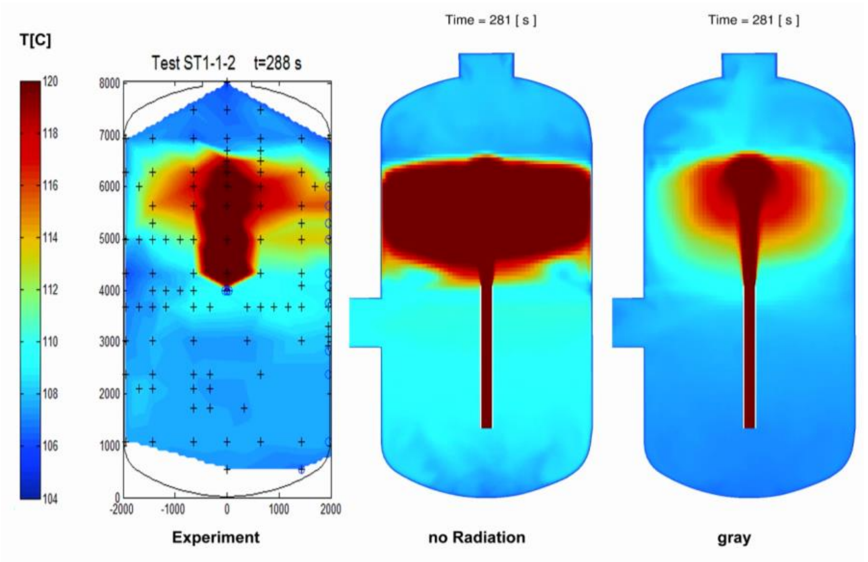
Radiation model

- **Radiation heat transfer in containment vessel (CV)**
- **P1 model**
  - Transport equation of incident radiation (G)

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

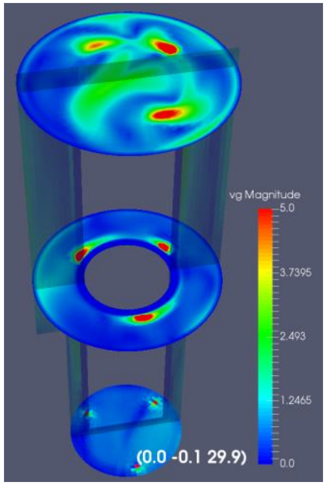
Radiation HT in CV

- **Effect of emissivity**
- **Effect of vacuum**



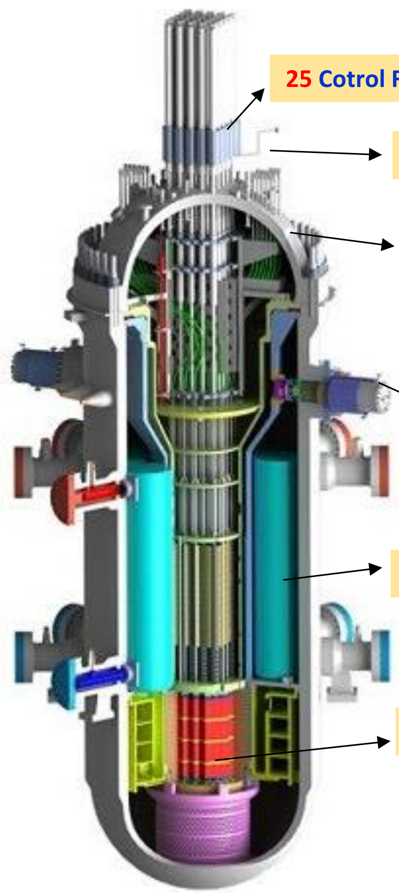
[Effect of radiation model-HYMERES2 project]

조건	Vacuum	Inerting-Stagnant	Inerting-HVAC
초기 압력	2kPa.abs (압력경계 부여)	1atm (압력경계 부여)	1atm (압력경계 부여)
열손실(Total)	0.109 MW	1.040 MW	0.534 MW (HVAC 제거 열량은 손실에서 제외)
열손실(복사)	0.089 MW	0.083 MW	0.013 MW
내부 기체 온도	392K ~ 538K	419K ~ 441K	327K ~ 349K
RV 외벽 온도	478K ~ 542K	460K ~ 500K	379 ~ 463K



Preliminary Calculation for iSMR Design

Design Specification of SMART

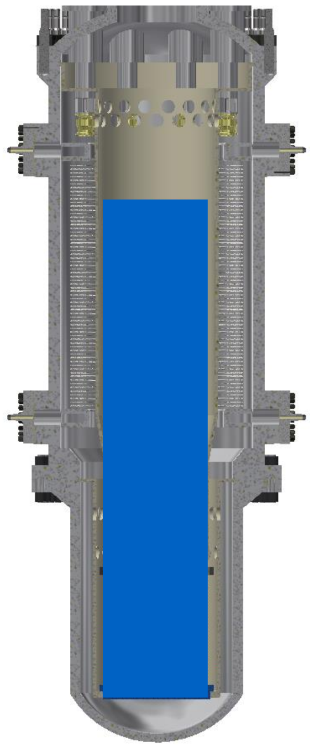


	SMART	NuScale
System Pressure (MPa)	15	13.8
Core Inlet Temperature (K)	568	538
Core Outlet Temperature (K)	596	594
RCS Flow Rate (kg/s)	2090	641
Power(Electric/Thermal)	100/330	60/192
No. of Steam Generators (Type)	8 (heilcal coil cassette)	2 (heical coil)
No. of RCPs	4	0
No. of FAs (Array)	57 (17x17)	37 (17x17)

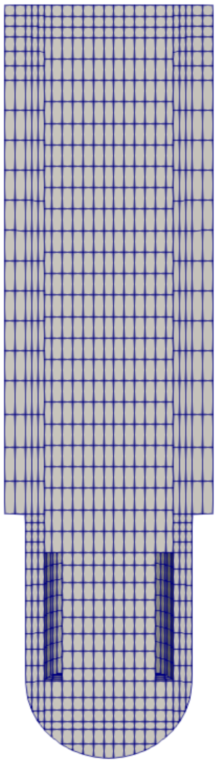
iSMR Mesh Generation

Conceptual iSMR Configuration and Mesh Generation

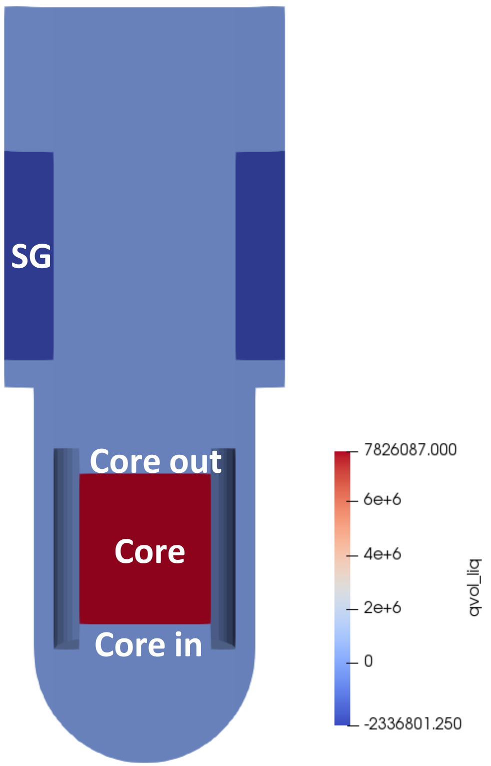
- 15 MPa single-phase water
- Test power=100 MWe, 133MWe, 166 MWe
- Problem time=1000 sec



Conceptual iSMR configuration



The number of cells : 14,153



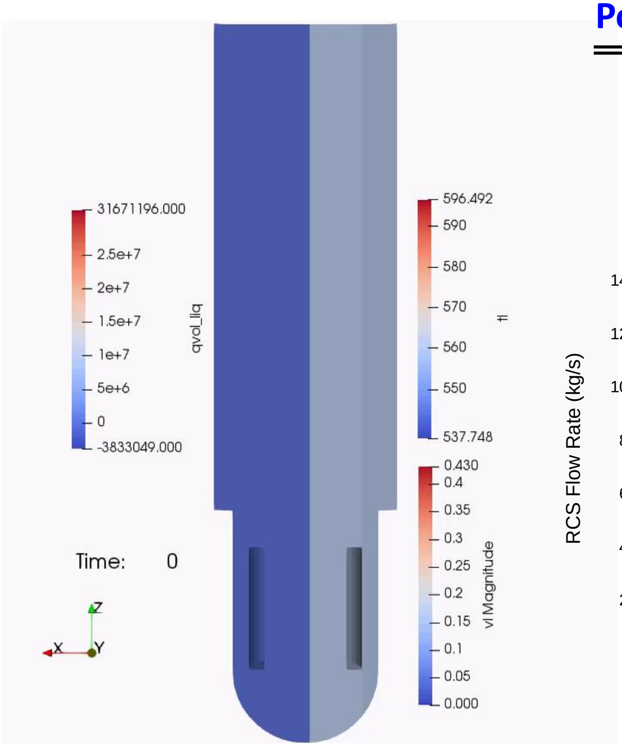
Uniform Heat Source/Sink (Core/SG)

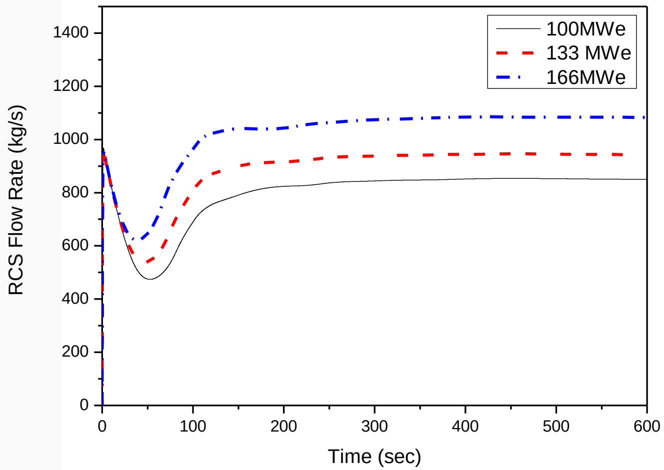
Sensitivity Study for Natural Circulation Flows

Natural Circulation Calculation

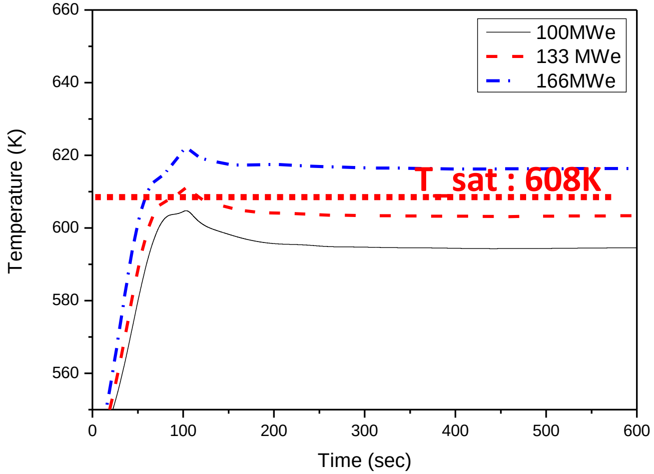
➢ 25% of SMART Pressure drop

Power (MWe)	Flow Rate (kg/s)	Tout (°C)	Tsub (°C)	ΔT (°C)
100	892	319	16	54
133	938	323	12	58
166	1082	338	-3	73





Natural Circulation Flow Rate



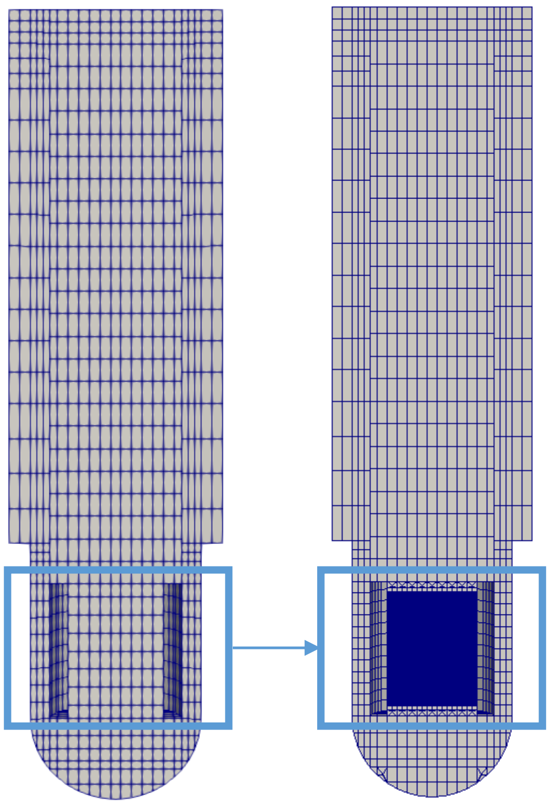
Core Exit Temperature

3D Full Core Mesh Generation for iSMR

Mesh for the Subchannel Scale iSMR

➢ **Full core analysis with local resolution control: nonconformal mesh**

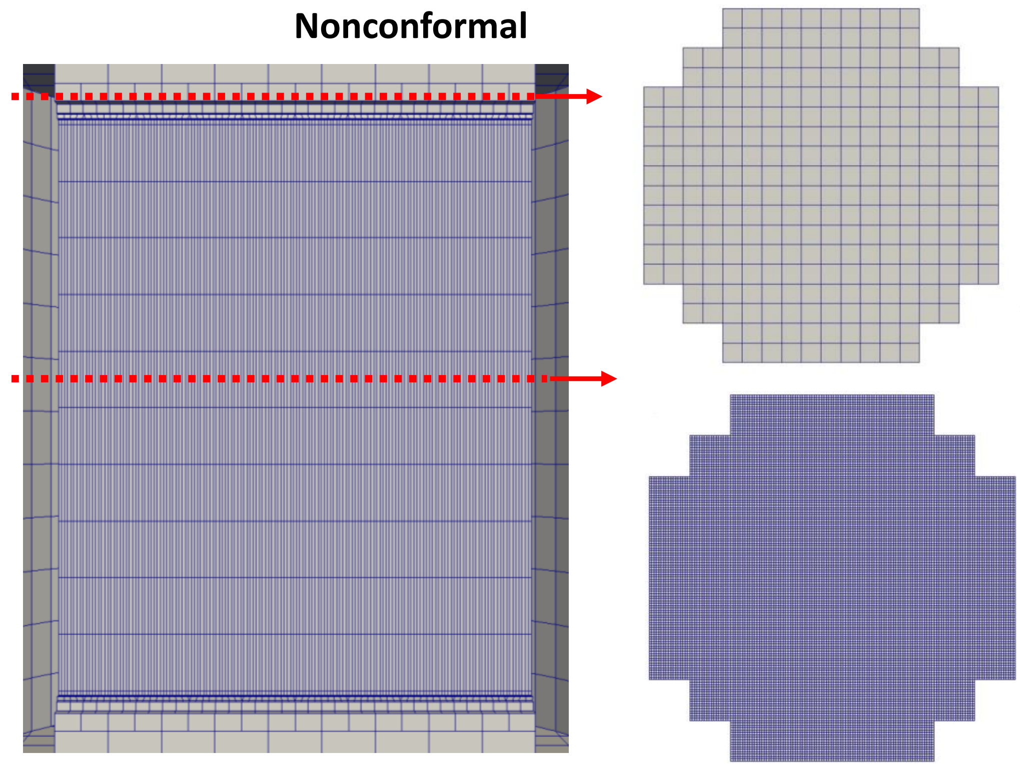
Assembly scale **Subchannel scale**



Number of cells
14,153

Number of cells
265,005

Nonconformal



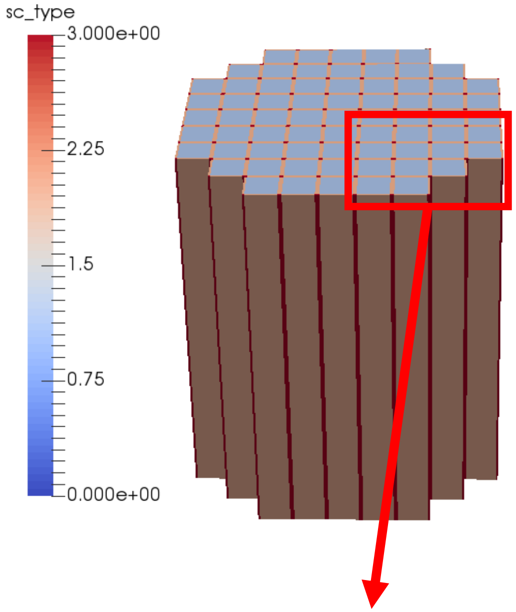
Subchannel mesh for the reactor core

Implementation of Subchannel Model

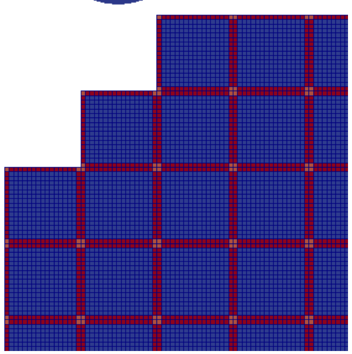
Subchannel Model

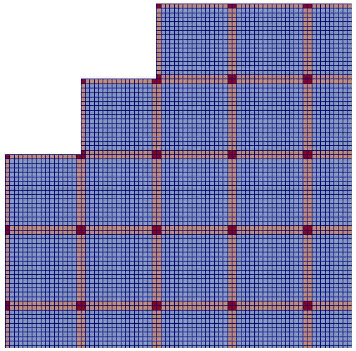
- ➤ **Friction Model from MATRA code**
- ➤ **Turbulence Mixing Model=EVVD**

Model	Note
Friction factor	$\Delta P = -\frac{1}{2}\left(\frac{f}{d_{hy}} + K'\right)\left(\frac{G_k^2}{\rho_k}\right)$
Form loss	$\Delta P_L = -\frac{K_G}{2}\left(\frac{W_{IJ,k}\lvert W_{IJ,k}\rvert}{l_{IJ}\rho_k s_{IJ}}\right)$
Turbulent mixing and void drift	• EM (Equal Mass exchange) • EVVD (Equal Volume exchange and Void Drift)
Grid spacer	$\Delta P = -\frac{K}{2}\left(\frac{G_k^2}{\rho_k}\right)$



Subchannel type definition



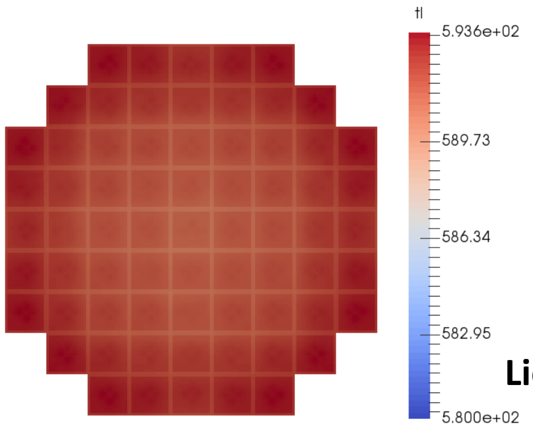


Hydraulic diameter

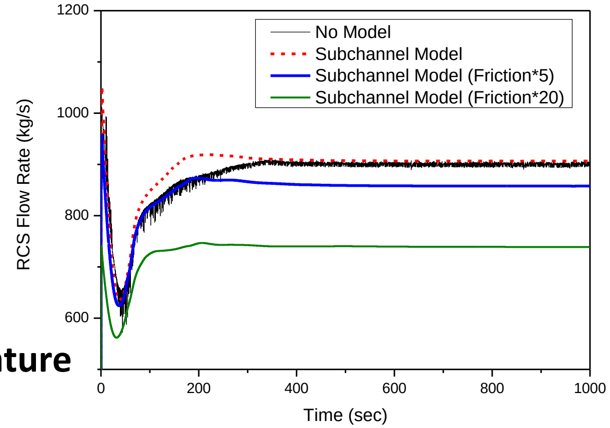
Full Core Analysis with Subchannel Model

Test Calculation of Subchannel Model

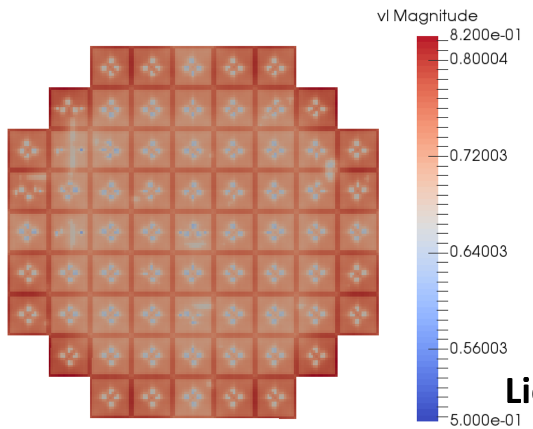
- Uniform heat source at the reactor core
- Turbulence Mixing by EVVD model, Friction by MATRA model



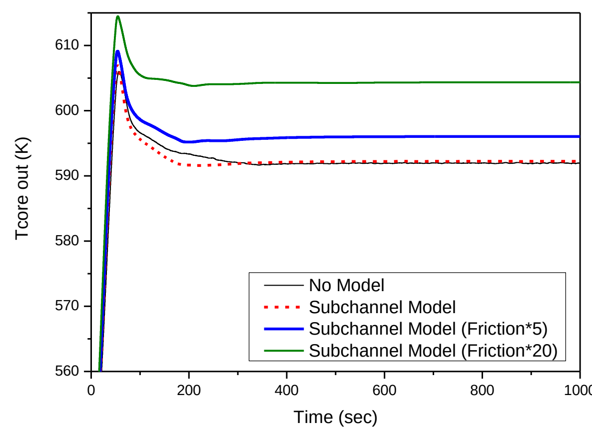
Liquid Temperature



RCS Flow Rate



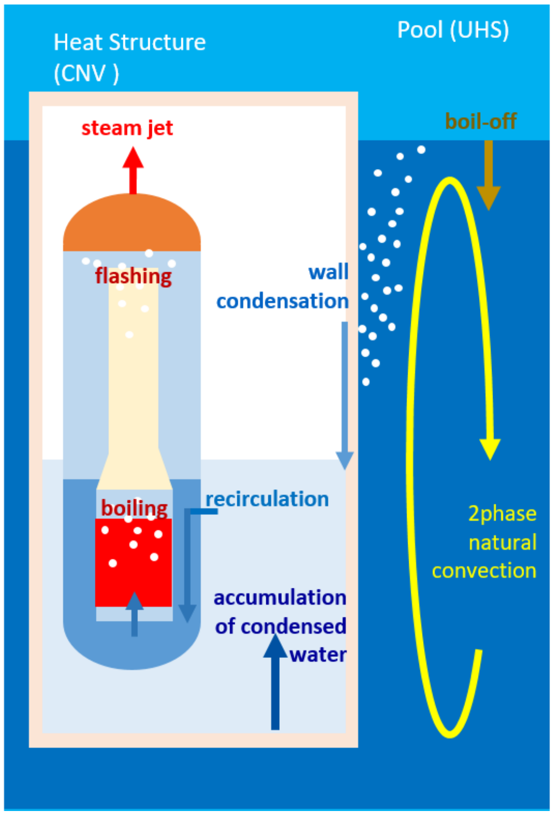
Liquid Velocity



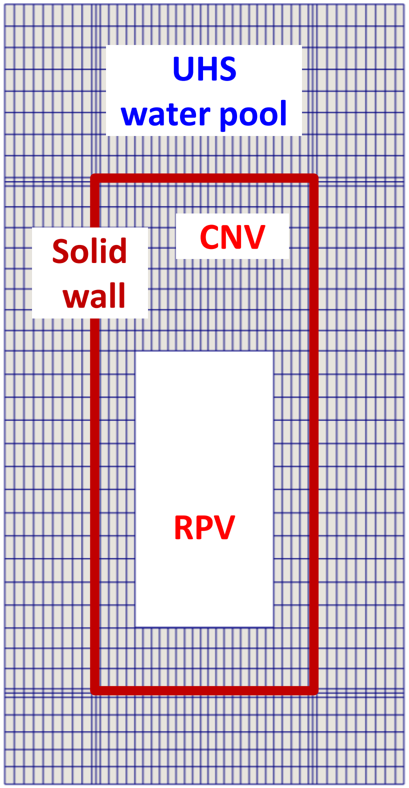
Core Exit Temperature

Conceptual Problem for SMR LOCA Analysis (1/3)

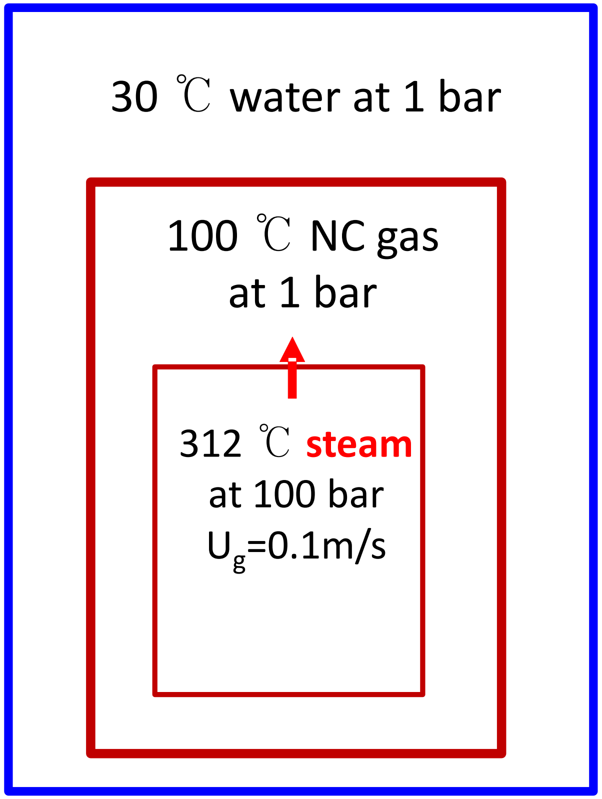
- ***Setup a conceptual problem to verify the CUPID code capability for the application to SMR LOCA analysis***
- ***2D mesh model for RPV, CNV, CNV solid wall, and UHS***



SMR LOCA Phenomena



2d Mesh Model



Initial Conditions

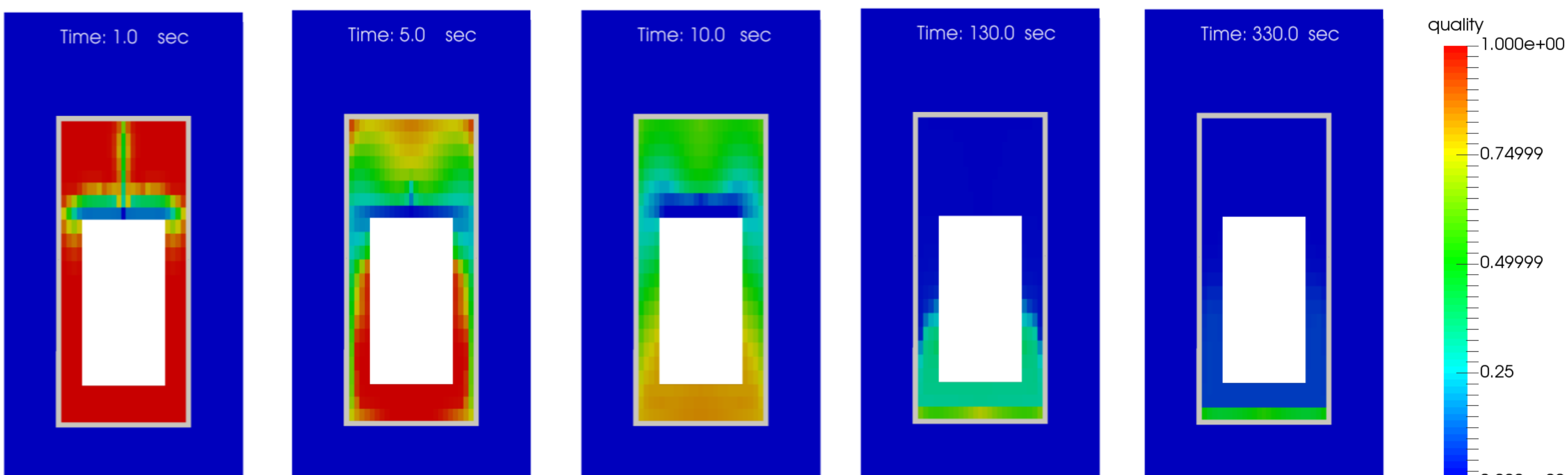
Conceptual Problem for SMR LOCA Analysis (2/3)

Verification of physical models

- Wall condensation in CNV (water level increase)
- Heat conduction of the CNV wall
- Natural circulation in UHS
- Boil-off in UHS (water level decease)

Verification of numerical stability for

- Simultaneous calculation of the two fluid regions separated by a solid wall

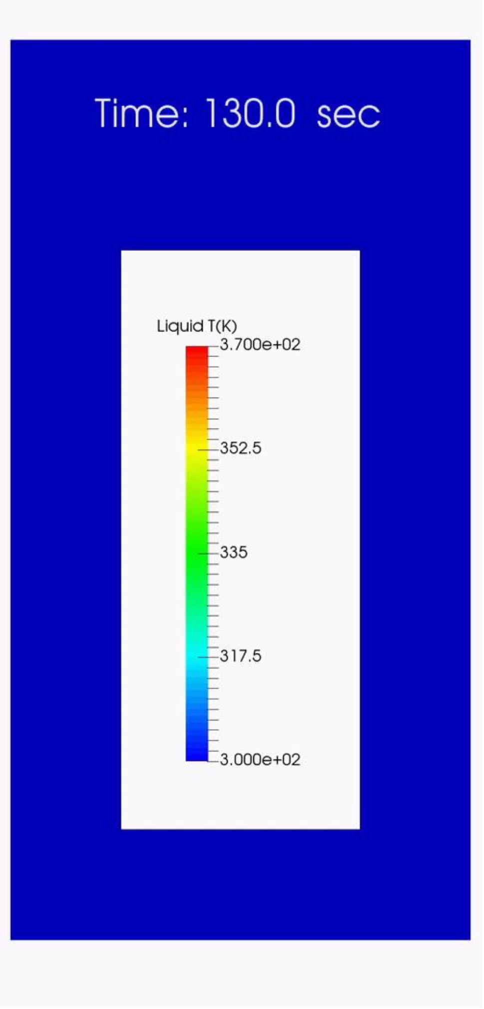


Non-condensable gas transient in CNV up to 330 s

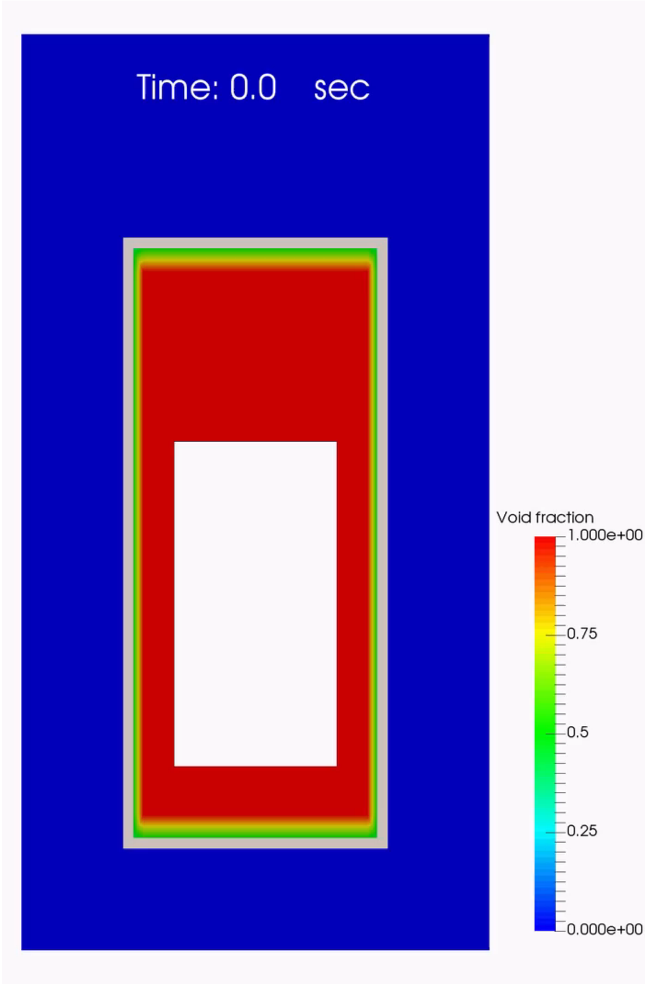
Conceptual Problem for SMR LOCA Analysis (3/3)

- 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



Summary

6

Summary

Analysis of passive safety features using CUPID

- Fluidic Device in APR+
- Siphon Breaker in Research Reactor
- Passive Auxiliary Feedwater System in APR+
- RDV/RVV in NuSCALE

Future Works

- LOCA Analysis of iSMR
  - EDV/EVV
  - Passive Auxiliary Feedwater System
  - Air Circulation in reactor building

THANK YOU

yjcho@kaeri.re.kr