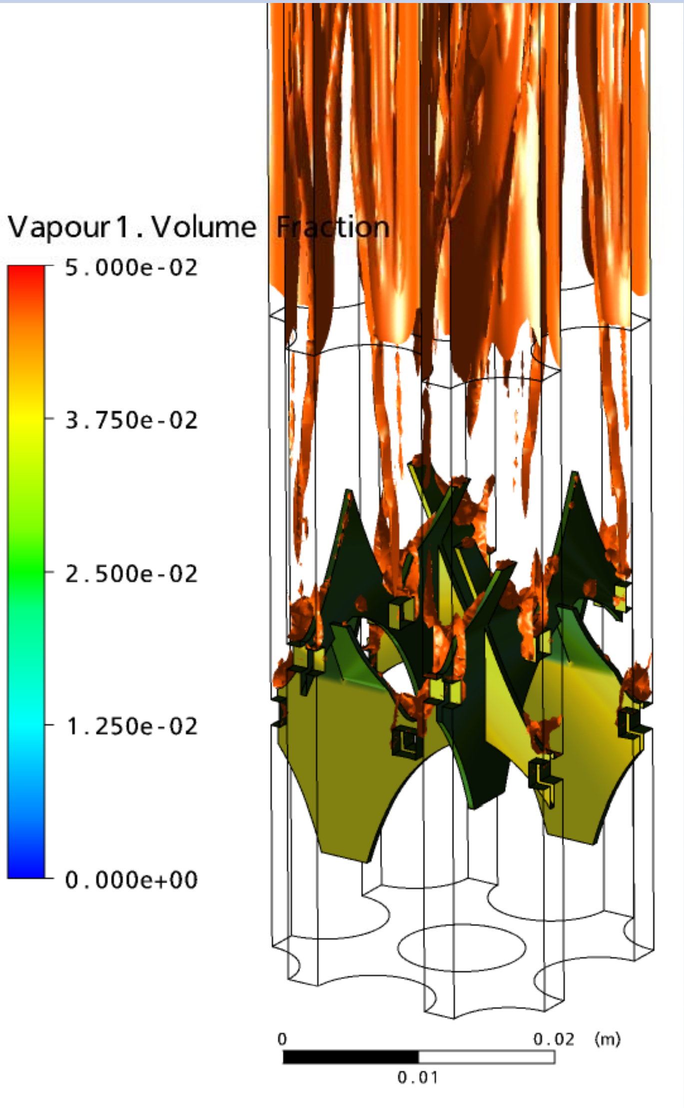
PSBT - CFD Simulation of Subcooled Boiling in Heated Subchannel Geometry using ANSYS CFX 13.0

Th. Frank,
F. Reiterer, C. Lifante
Fluids Validation Manager
ANSYS Germany
Thomas.Frank@ansys.com

Outline

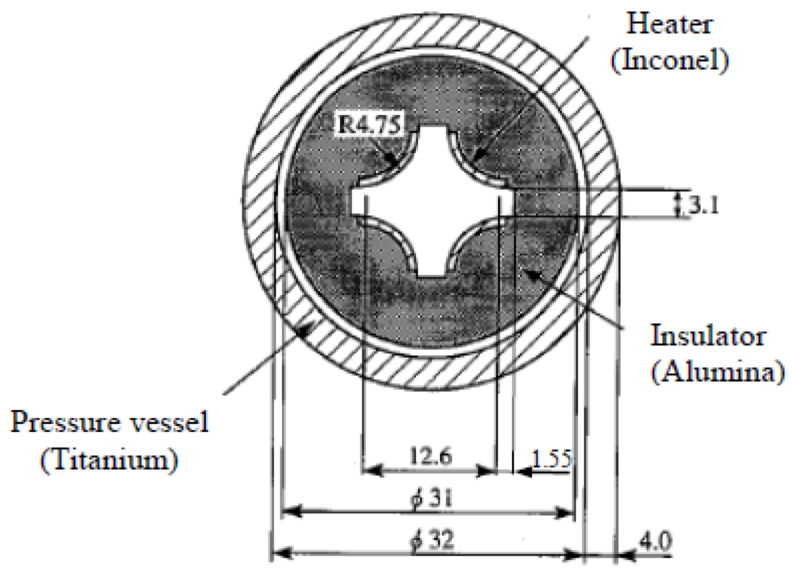
PSBT, Phase I, Exercise 1:

- Testcase matrix
- CFD modeling
  - Eulerian Multiphase Flow
  - Wall boiling
  - Turbulence
- Followed CFD Best Practice
- Obtained results
- Conclusions & Outlook

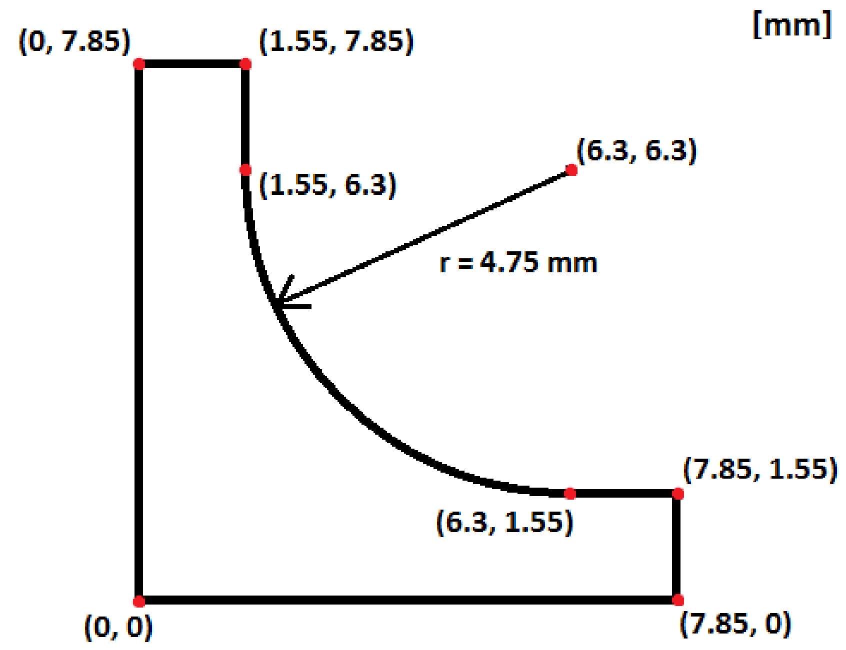


PSBT, Test Section S1 Geometry Horizontal plane (xy-plane)

- PSBT Phase I-1: ANSYS CFX 13.0



S1



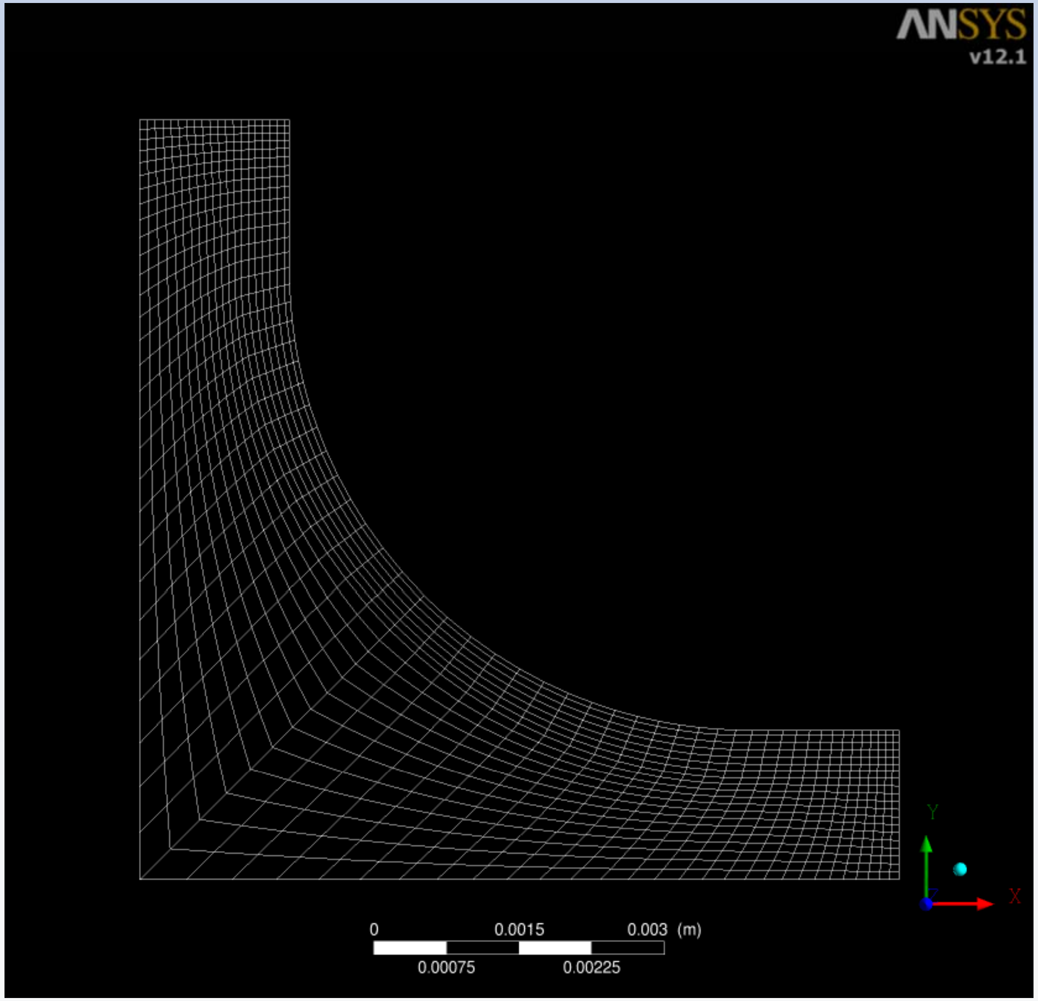
- Electrical heating of four ¼ pipes with 4.75 mm radius (imitating fuel rods)
- Modeling 1/4th of the geometry (1/8th symmetry)

Mesh Hierarchy

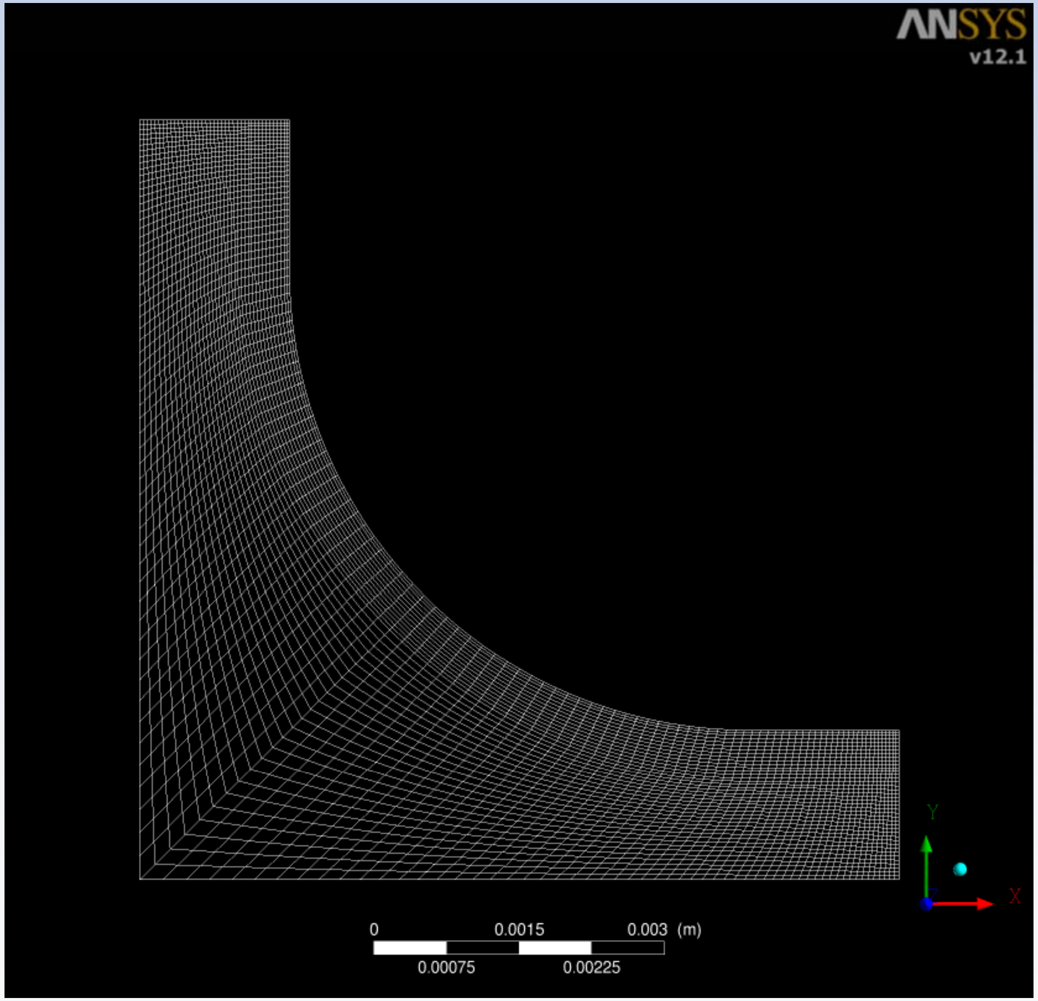
- Hexa meshing using ICEM/CFD Hexa 13.0
- Three consecutively refined levels of meshes
- Refinement by a factor of 2 in all directions
- Near wall zone of controlled mesh element thickness

MESHES:	Mesh 1	Mesh 2	Mesh 3
Size (xy × z resolution)	1160 x 155	4640x310	18560x620
Hexas	$1.8*10^5$	$1.4*10^6$	$1.2*10^7$
Nodes	$1.9*10^5$	$1.5*10^6$	$1.2*10^7$
Min. Angle [°]	42.6186	42.6217	42.6233
Min. Det.	0.890582	0.943188	0.971115
y^+_{max}	~ 161	~81	~41

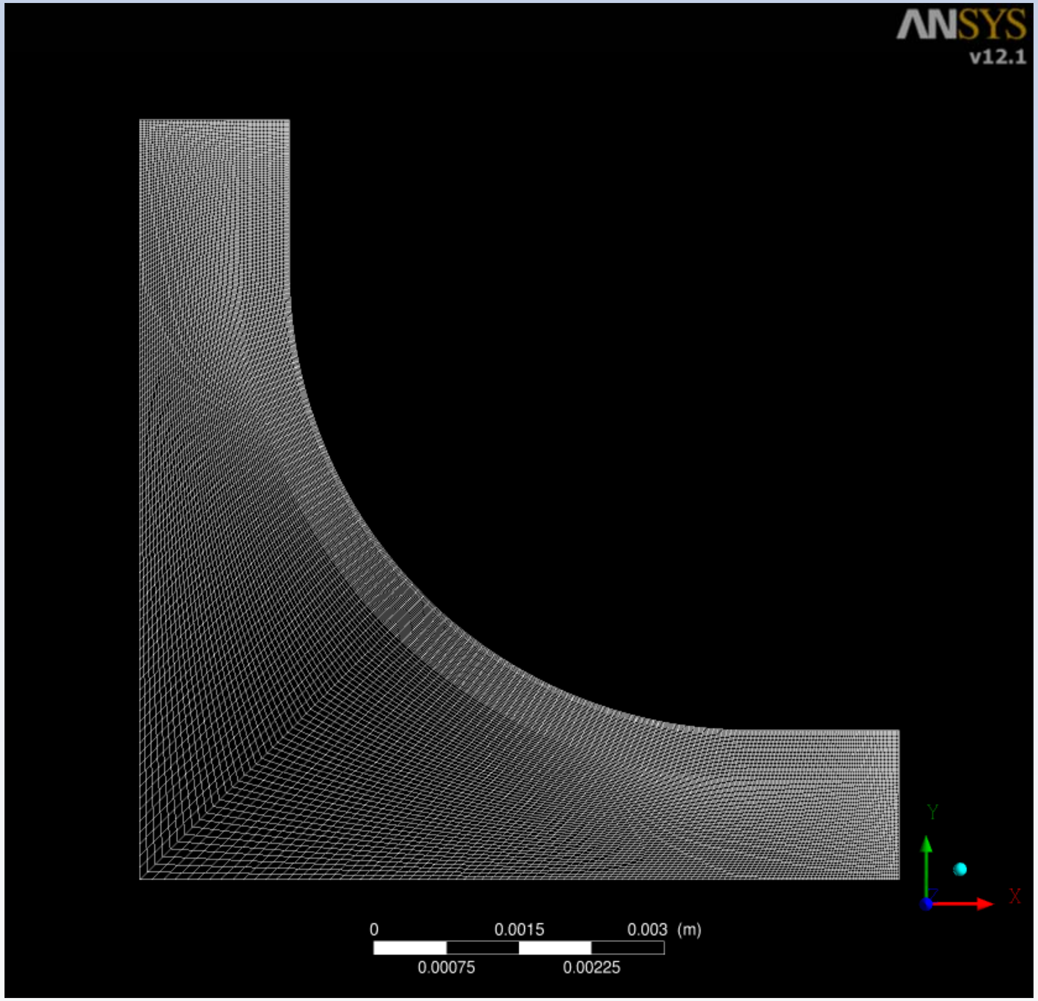
Mesh 1



Mesh 2



Mesh 3



Boundary Conditions (BC's)

Symmetry BC:

- Applied to the 2 inner boundaries of the subchannel segment

Inlet BC:

- Mass flow rate and liquid temperature from PSBT
- Single phase pre-calculation on same geometry (L/D ~99)
  - Mesh 2, only liquid phase, isothermal, $T_{liq}=T_{in}$, different turbulence models
- Turbulence quantities of developed flow:
  - SST: k and Omega
  - EARSM: k and Epsilon
  - BSL RSM: Reynolds Stresses and Omega
- Correction factors to correct the integral mass flux @ Inlet → compensation of mesh discretization error

Boundary Conditions & Initialization

Outlet BC:

- Pressure @ Inlet as specified in PSBT database was used as averaged static pressure @ Outlet
- Neglected contribution of hydrostatics in comparison to total pressure level

Wall BC:

- Prescribed const. wall heat flux (q_{wall}) on heater surface
- all other walls → adiabatic walls

Initialization:

- Pressure @ Domain = p_{ref} (= Pressure @ Outlet)
- u/v/w @ Domain = u/v/w @ Inlet
- Turbulence Quantities are equal to Inlet BC's
- Temperature @ Domain = Temperature @ Inlet = T_{in}
- Steam VF @ Domain = 0

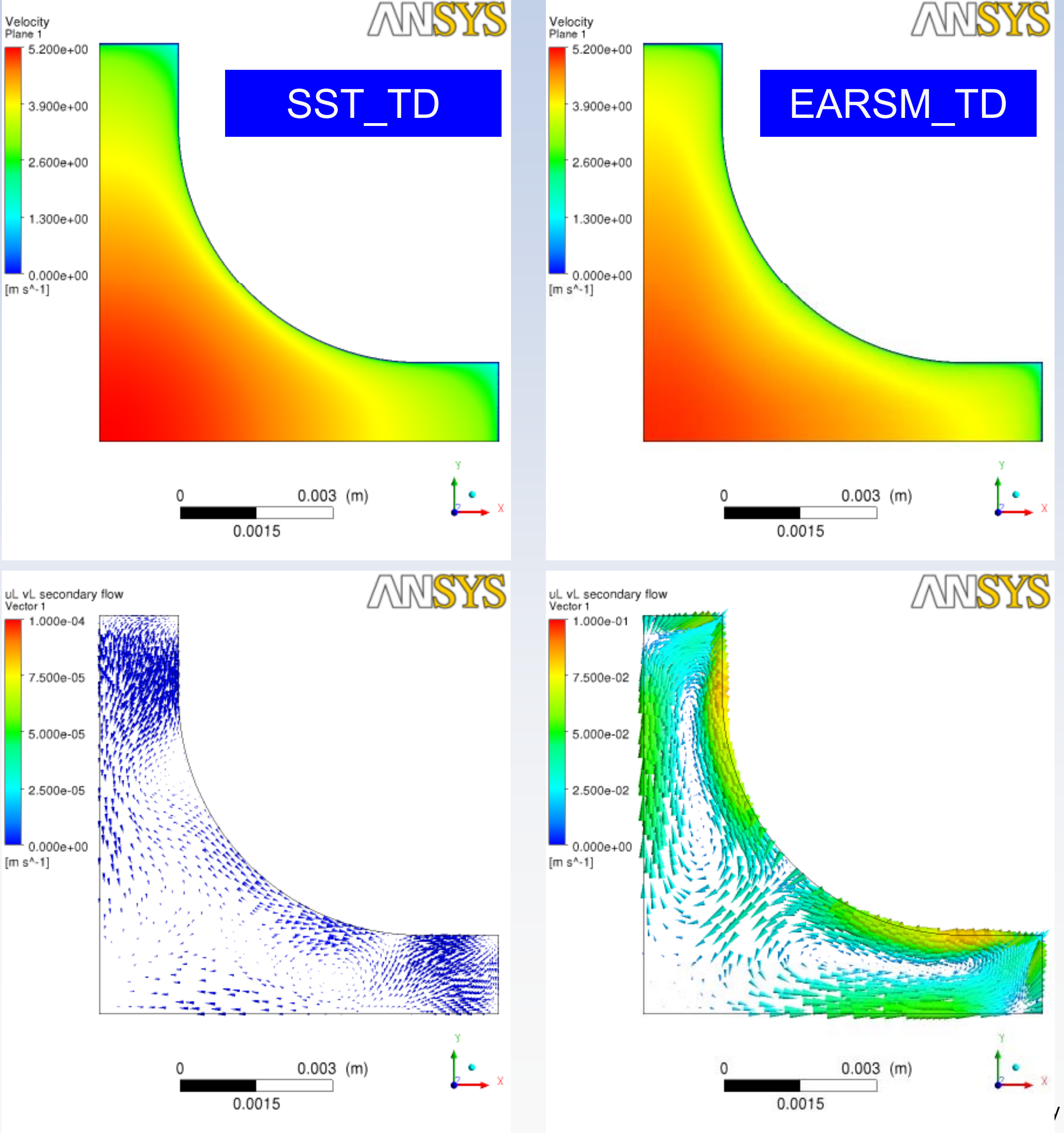
CFD Model Setup

- Eulerian MPF framework, 2-phase flow
- Blending of IAC for higher steam volume fraction
- Turb. model in continuous phase + 0-eq. disp. phase turb. model + Sato bubble induced turbulence
- Continuous phase enthalpy eq.
  + vapour phase set to saturation temperature (IAPWS-IF97)

	SST_TD	SST_ND	EARSM_ND
Turbulence model	SST	SST	EARSM
Drag force	Grace	Grace	Grace
Turb. dispersion force	FAD TD force	FAD TD force	FAD TD force
Lift force	-	Tomiyama	Tomiyama
Wall lubrication force	-	Antal	Antal

Turbulence Model Variation

- SST vs. EARSM
- SST – isotropic turbulence assumption
- EARSM – can predict for anisotropic turbulence and recirculating flows
- cross-sectional recirculation contributes to thermal mixing and redistribution of vapour on heater surface
  → model uncertainty



Flows with Subcooled Boiling (DNB) – RPI-Wall Boiling Model

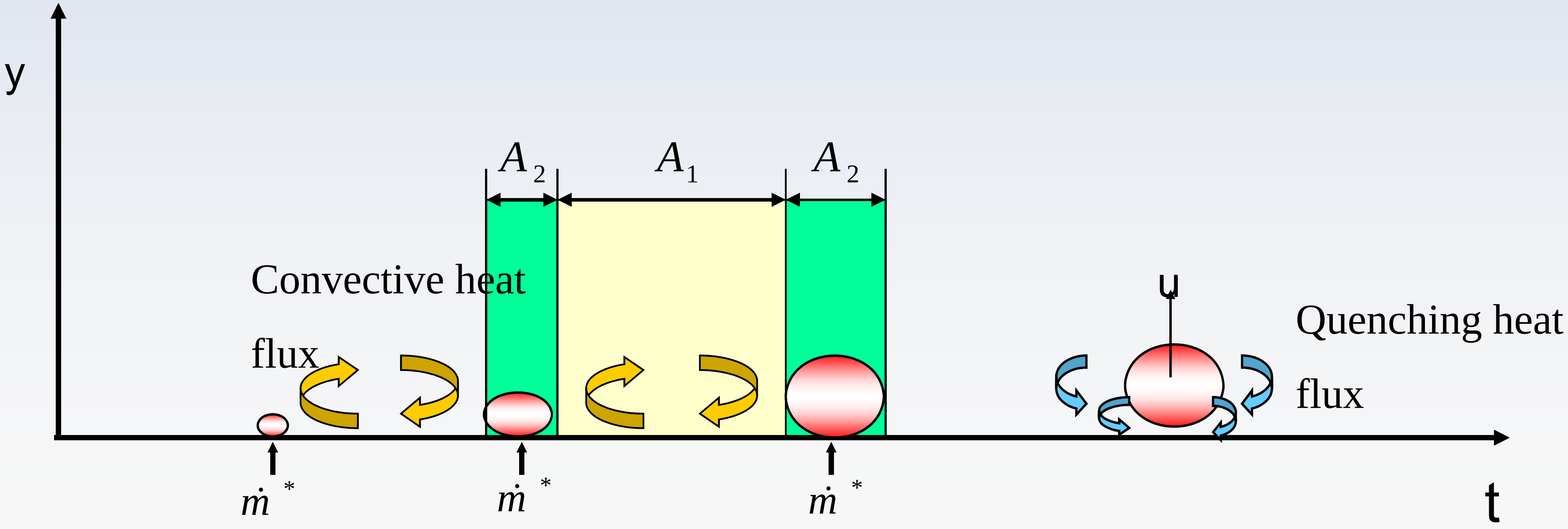
Mechanistic wall heat partioning model:

$$\dot{q}_{Wall} = \dot{q}_F + \dot{q}_Q + \dot{q}_E$$

convective heat flux
$\dot{q}_F = A_1 \cdot h_F \cdot (T_W - T_L)$

quenching heat flux
$\dot{q}_Q = A_2 \cdot h_Q \cdot (T_W - T_L)$

evaporation heat flux
$\dot{q}_E = \dot{m} \cdot (h_G - h_L)$



RPI-Wall Boiling Model – Submodels for Model Closure

Submodels for closure of RPI wall boiling model:

- Nucleation site density: Lemmert & Chawla
- Bubble departure diameter: Tolubinski & Kostanchuk
- Bubble detachment frequency:
  Terminal rise velocity over Departure Diameter
- Bubble waiting time:
  Proportional to Detachment Period
- Quenching heat transfer: Del Valle & Kenning
- Turbulent Wall Function for liquid convective heat transfer coefficient

- Correlation for bulk flow mean bubble diameter required:

→ smoothed Kurul & Podowski correlation via CCL
→ assumed near wall bubble diameter $d_{b,max}$=0.65mm
(consistent d_B value near wall with bubble departure diameter)

CFD Best Practice

Addressed CFD Best Practice:

- Selected case 1.2211 for CFD BPG analysis
- Baseline setup SST_TD used for these ivestigations
- Investigation of required convergence level
- Investigation of integral balances
- Steady-state vs. transient flow behavior
  (transient behavior found for cases: 1.3221, 1.4121, 1.4122, 1.4325, 1.4326)
- Investigation of mesh independency

CFD Best Practice
- CFD Solver Convergence Level -

- Comparison of integral values in dependence on prescribed convergence level:

	MAX RES 10^{-3}	MAX RES 10^{-4}	MAX RES 10^{-5}
Iteration Nr.	377	456	502
r_v @ Domain	0.035568	0.036551	0.036575
p @ Inlet [Pa]	15038339	15038300	15038297
r_v @ Plane 1	0.099402	0.103239	0.103297
T_{Water} @ Plane 1 [K]	608.3125	608.5684	608.5726

→ Continued investigations with N_{Iter}~500 and convergence criterion set to Max RES=10^{-4}

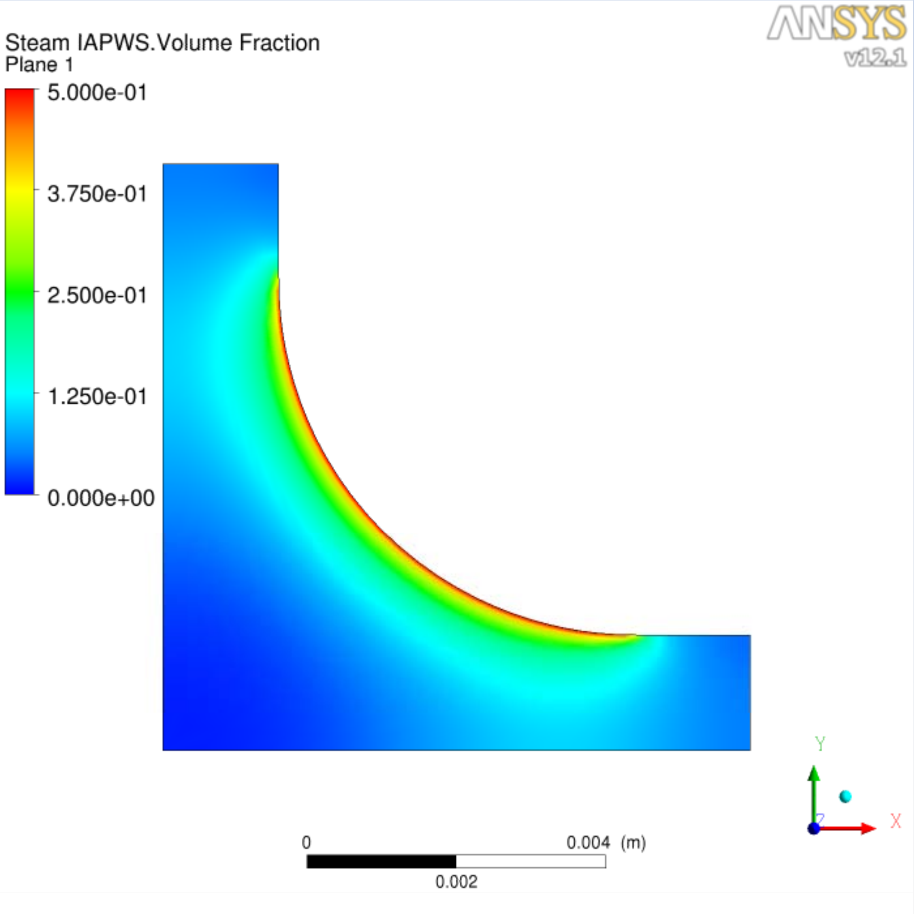
CFD Best Practice
- Mesh Independence -

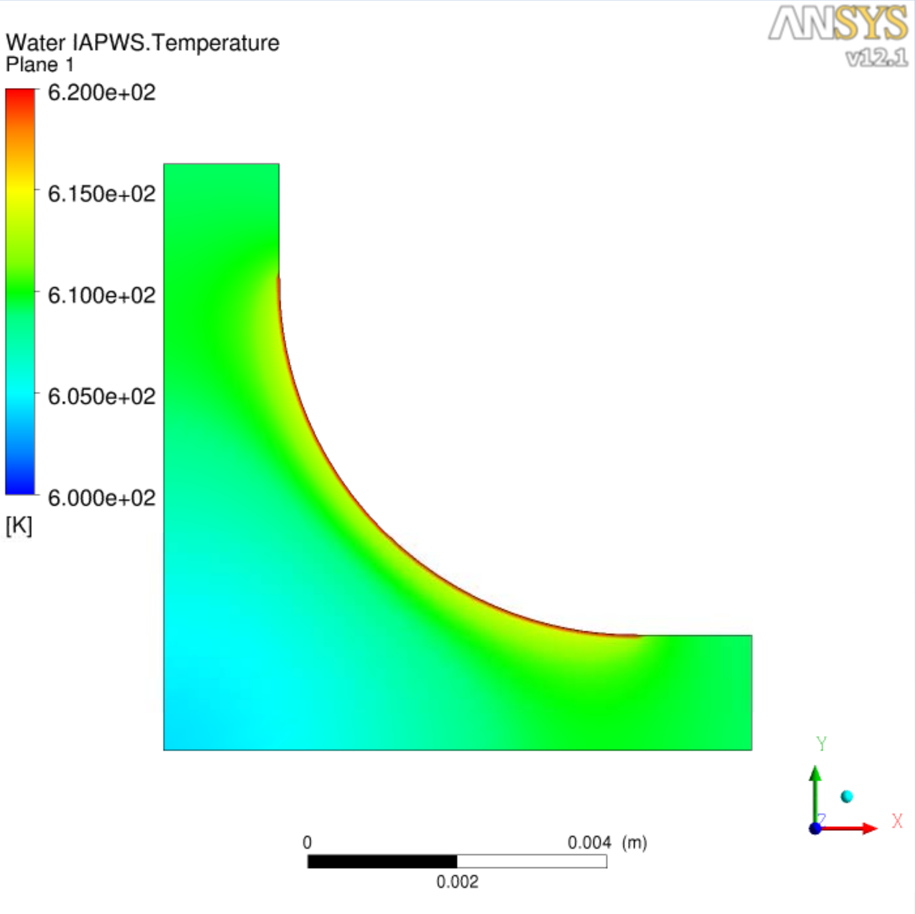
- Comparison of integral values in dependence on mesh resolution:

	Mesh 1	Mesh 2	Mesh 3
r_v @ Domain	0.036577	0.036847	0.036890
p @ Inlet [Pa]	15038297	15037081	15036333
m @ Inlet [kg m^{-2} s^{-1}]	3027.58671	3027.75835	3027.81763
r_v @ Plane 1	0.103303	0.101383	0.099270
T_{Water} @ Plane 1 [K]	608.5729	608.2116	607.6549

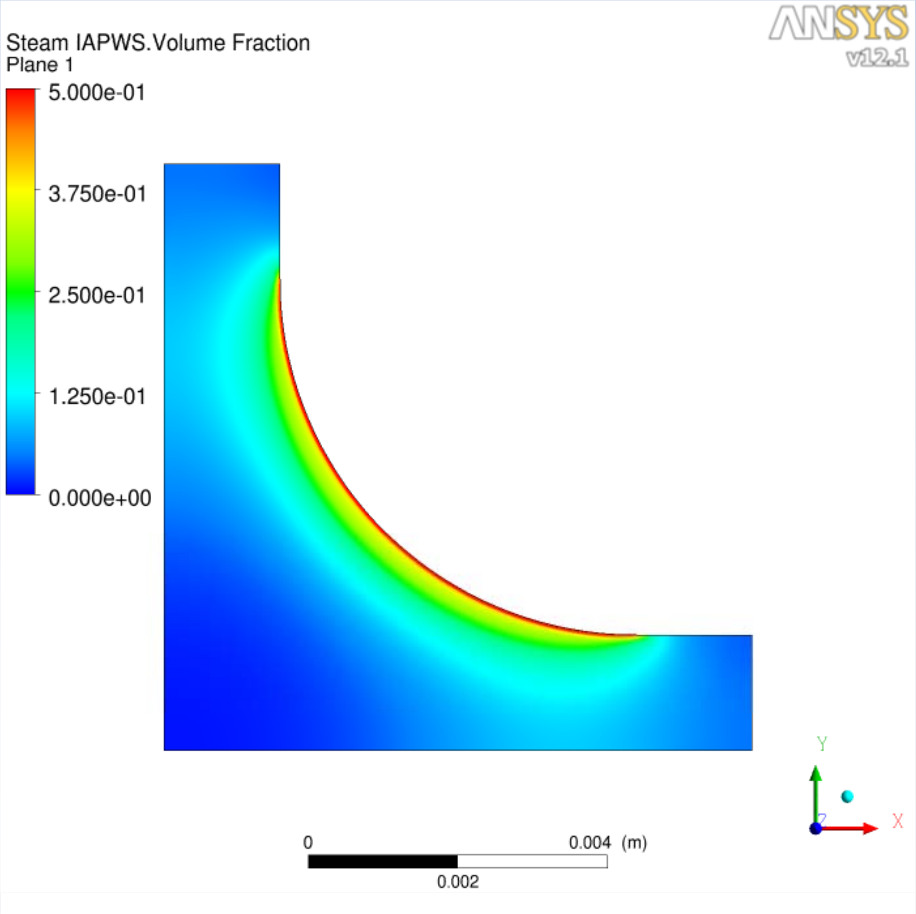
→ Continued investigations on Mesh 2 as a good compromise between accuracy and comp. effort

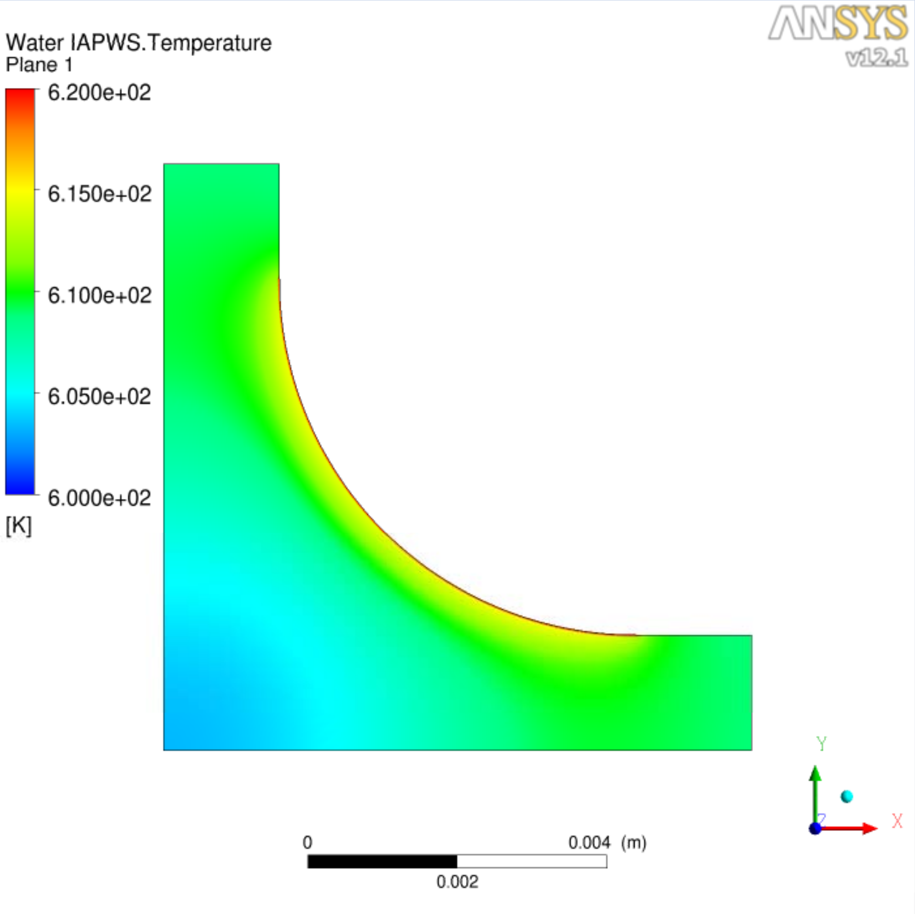
CFD Solver Results: 1.2211, Mesh 1



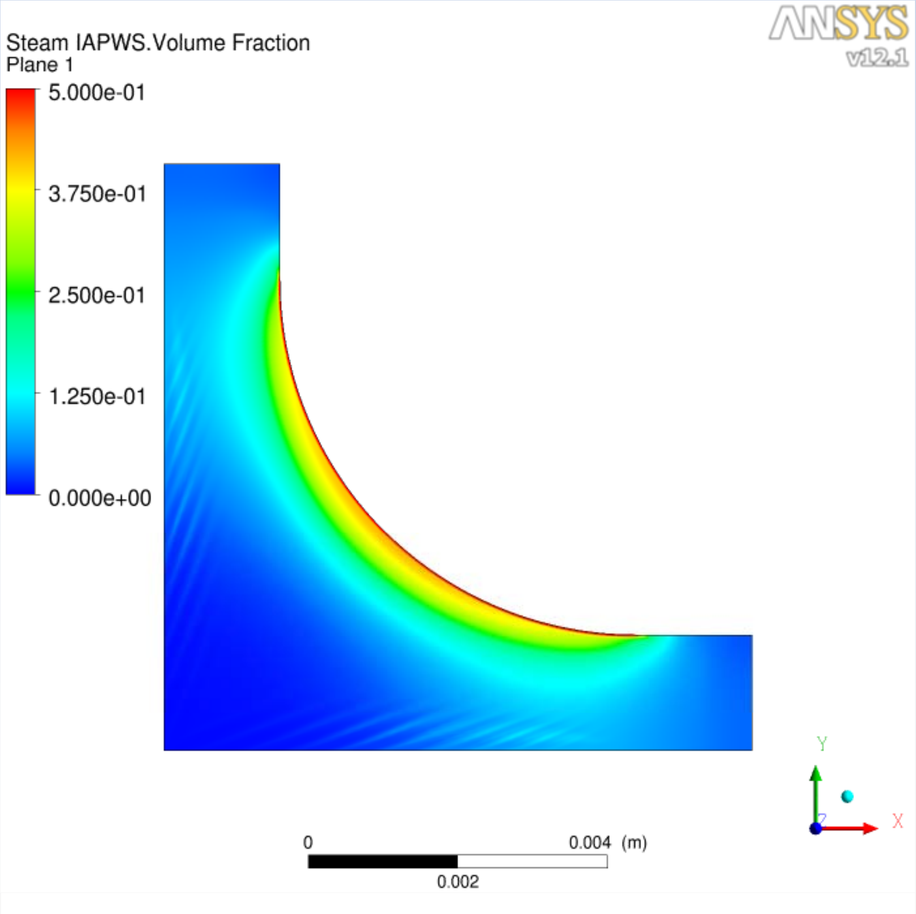


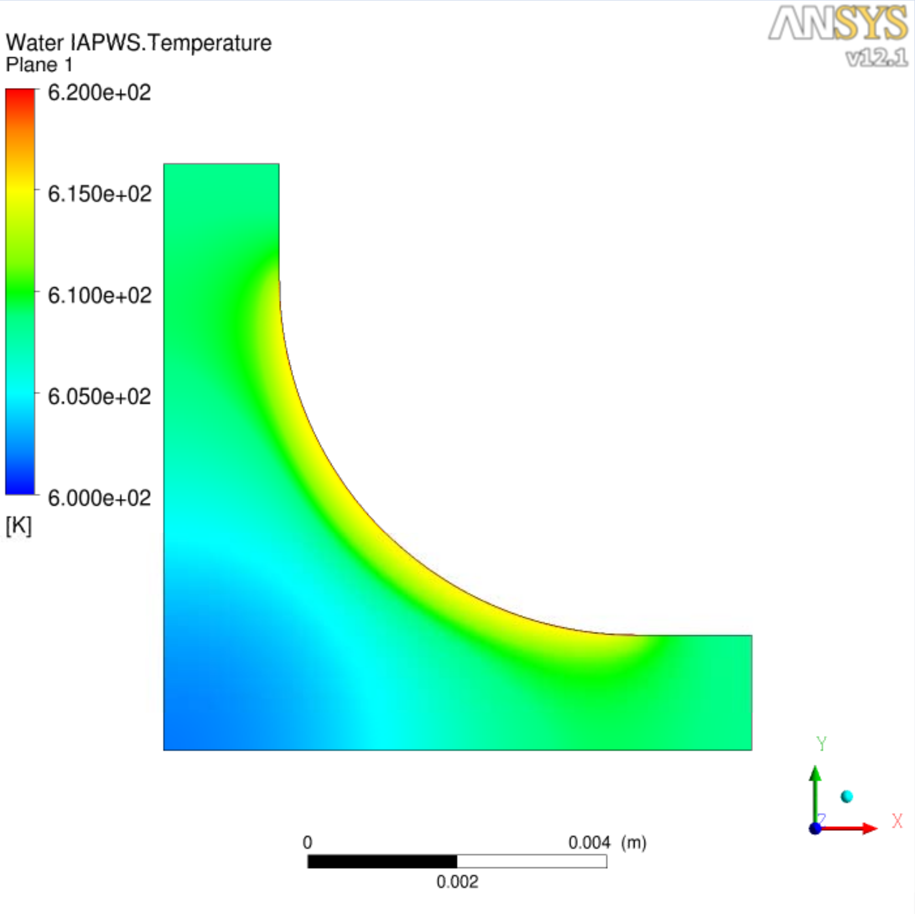
CFD Solver Results: 1.2211, Mesh 2



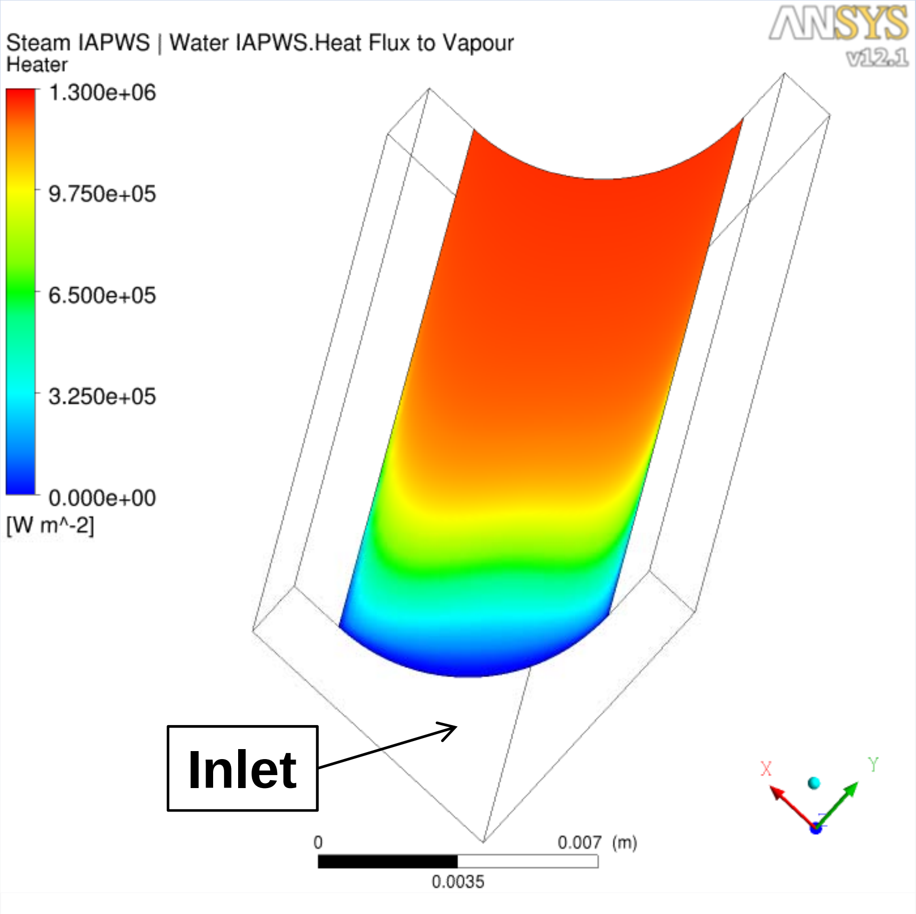


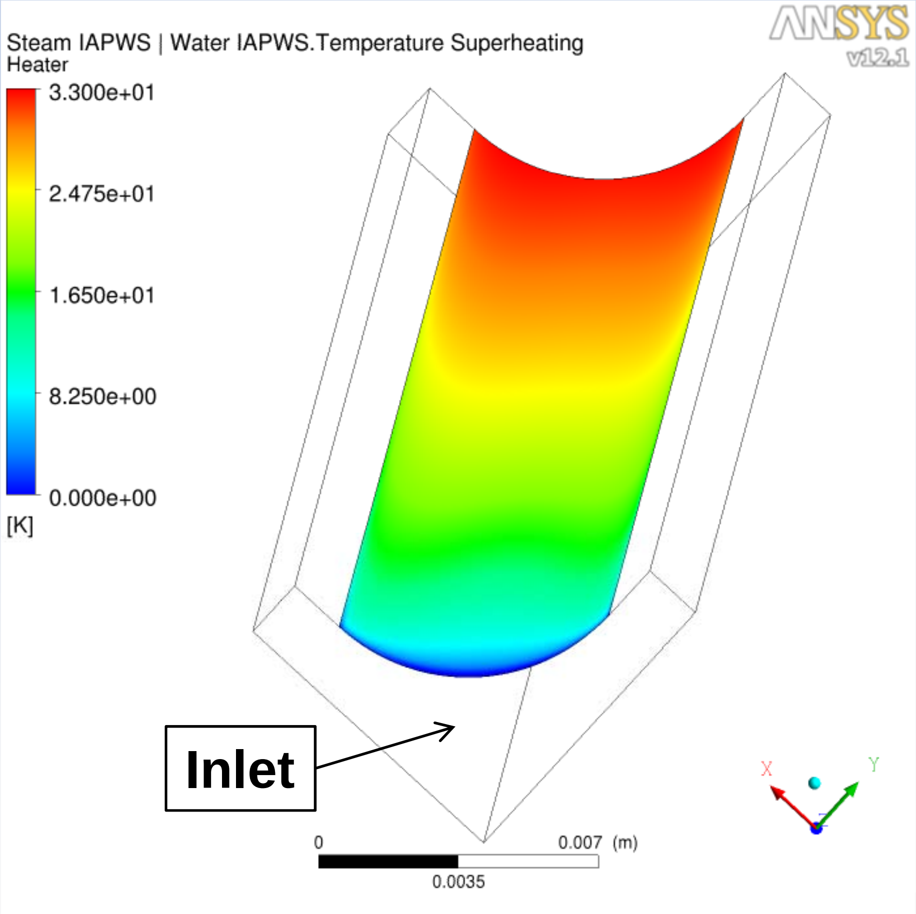
CFD Solver Results: 1.2211, Mesh 3



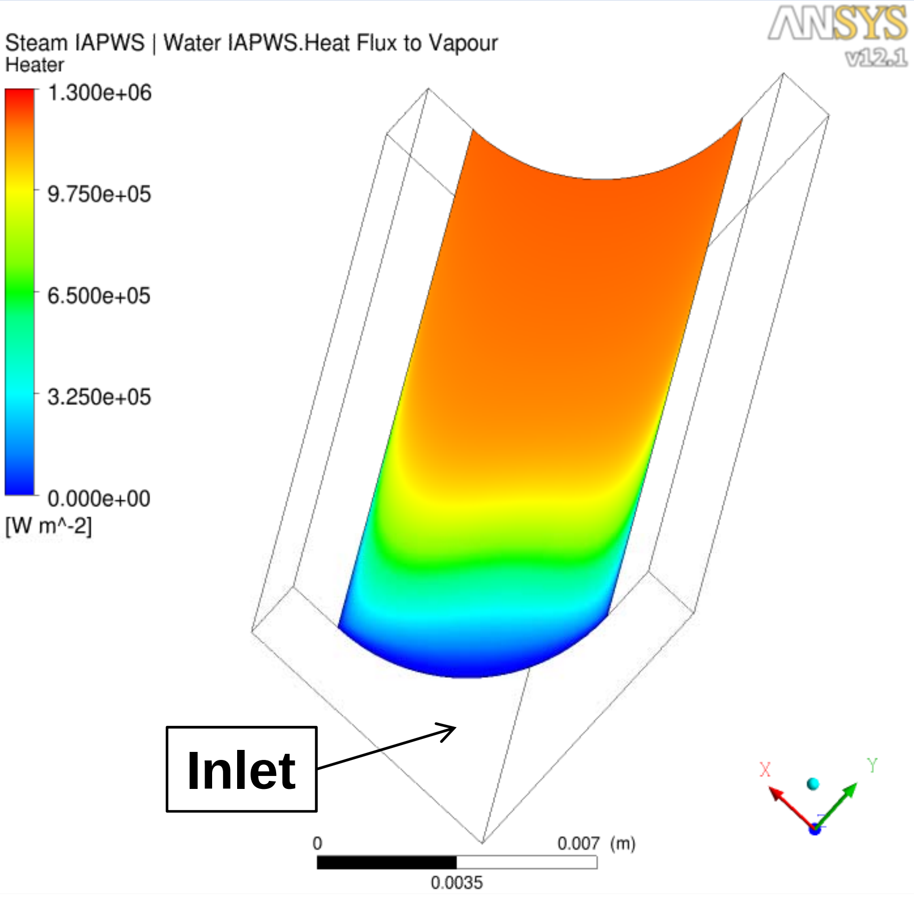


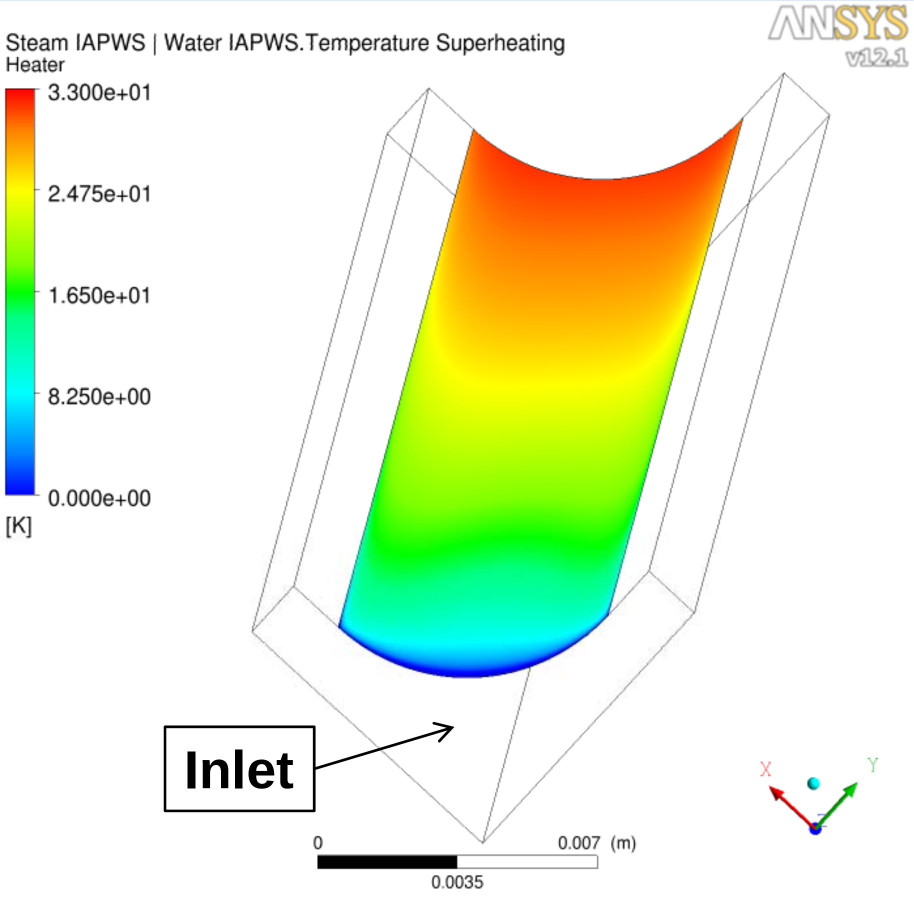
CFD Solver Results:
1.2211, Heater – Mesh 1



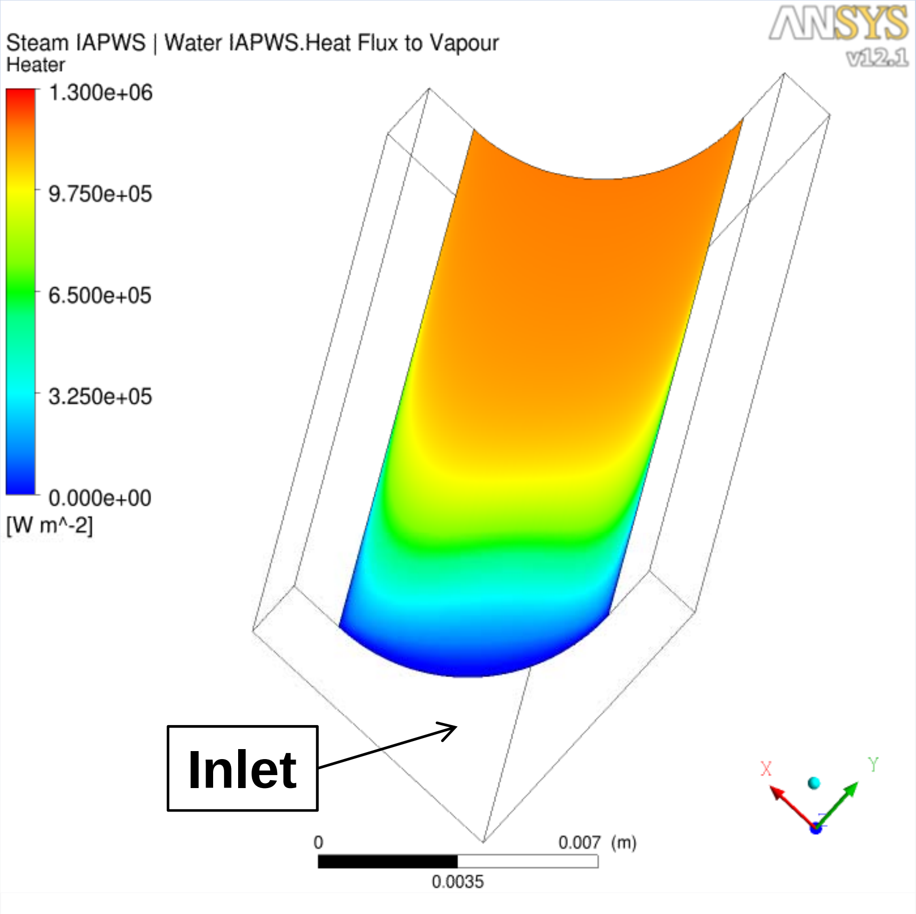


CFD Solver Results: 1.2211, Heater – Mesh 2

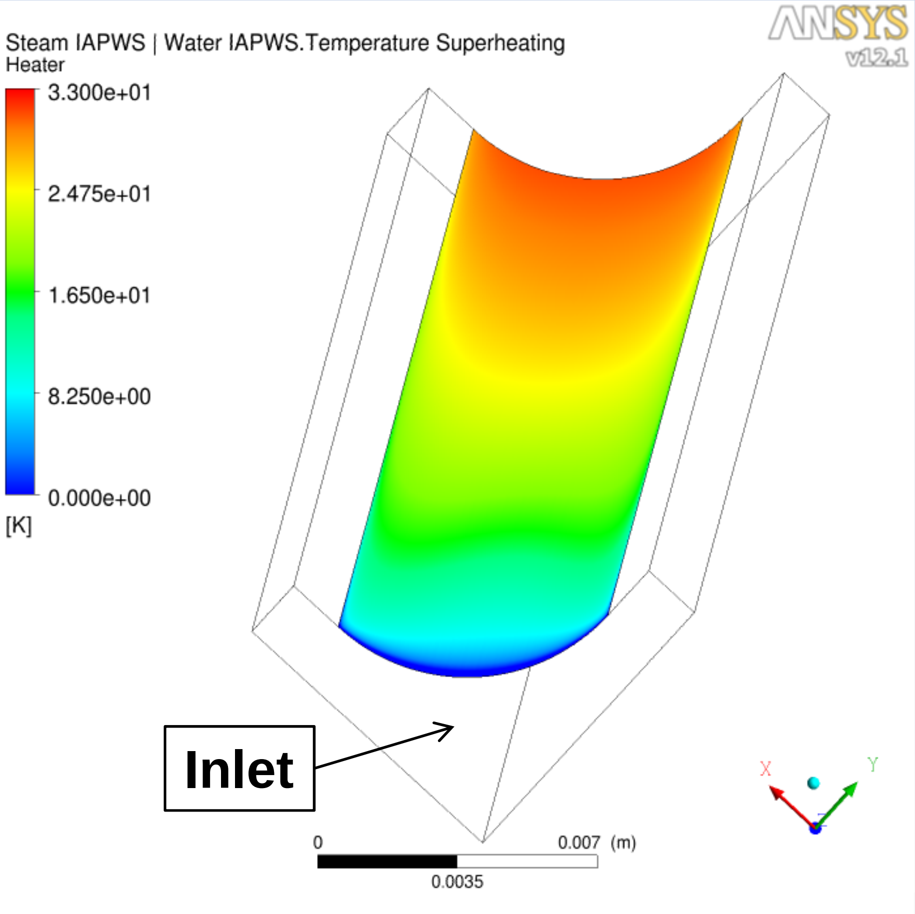




CFD Solver Results: 1.2211, Heater – Mesh 3



Inlet



Inlet

Final PSBT Benchmark Results

Testcase Parameter Variation

- Studied testcase conditions (mandatory testcases):

Testcase	Pressure	Inlet Temp.	Power	Mass Flux	r_v (meas.)
1.2211	150 bar	295.4 °C	1.93 MW m^{-2}	3031 kg m^{-2} s^{-1}	0.038
1.2223	150 bar	319.6 °C	1.50 MW m^{-2}	3031 kg m^{-2} s^{-1}	0.311
1.2237	150 bar	329.6 °C	1.29 MW m^{-2}	3031 kg m^{-2} s^{-1}	0.440
1.4325	100 bar	253.8 °C	1.29 MW m^{-2}	1389 kg m^{-2} s^{-1}	0.335
1.4326	100 bar	268.8 °C	1.30 MW m^{-2}	1389 kg m^{-2} s^{-1}	0.531

- CFD simulations carried out on Mesh 1 & Mesh 2 (1.2211 additionally on Mesh 3)

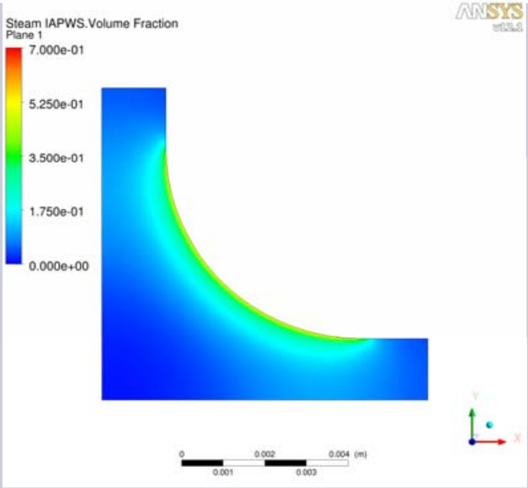
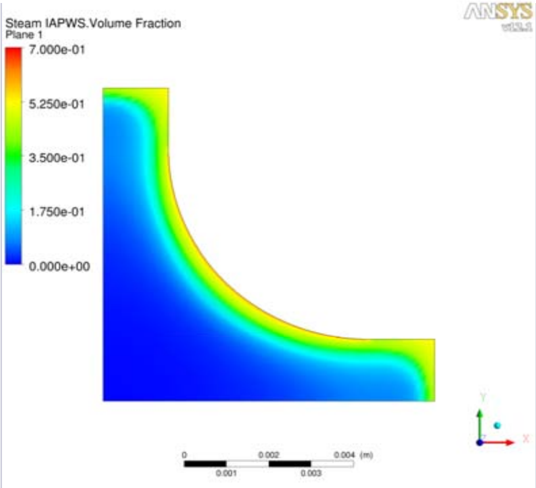
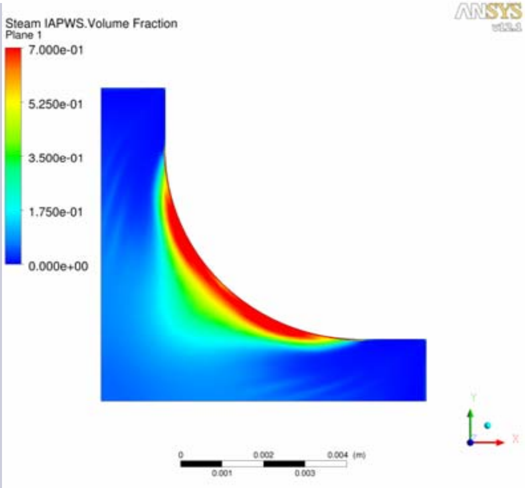
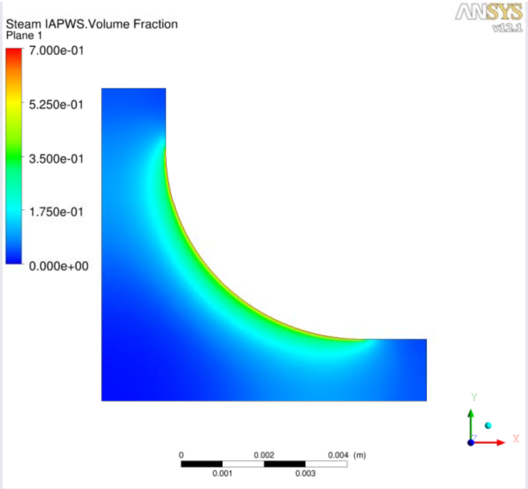
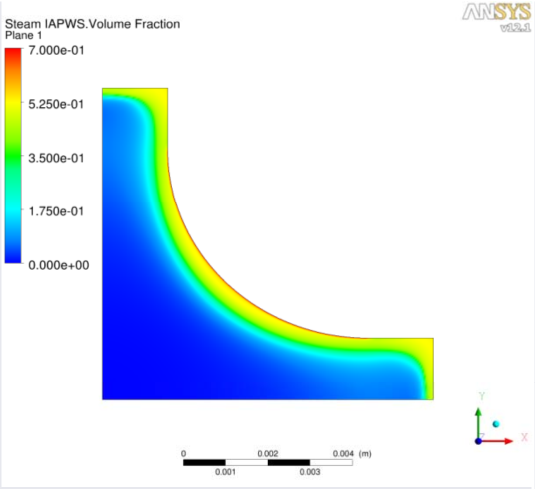
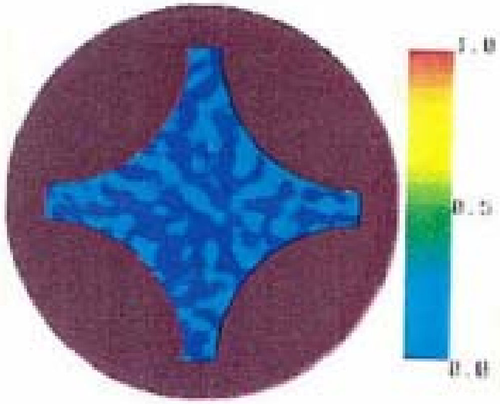
Testcase Parameter Variation

- Additionally studied testcases:

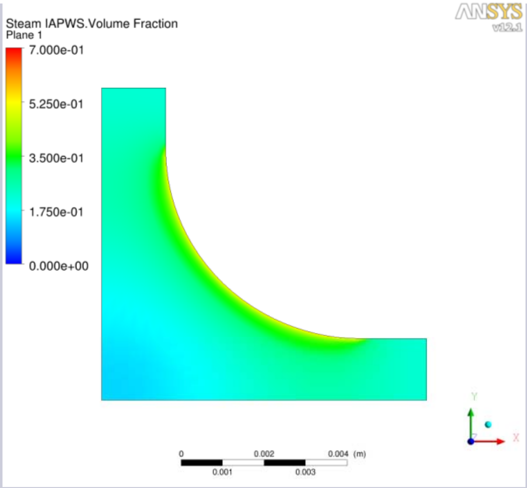
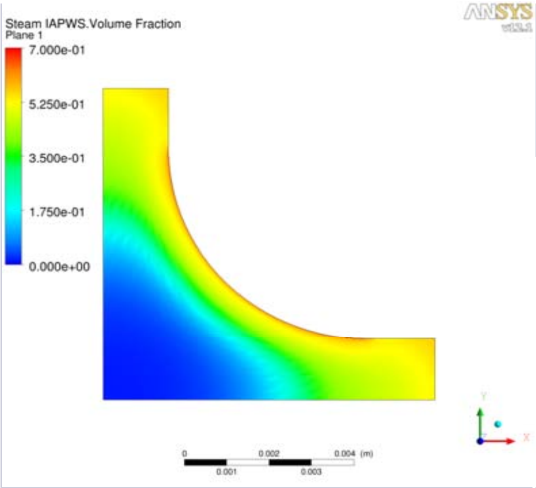
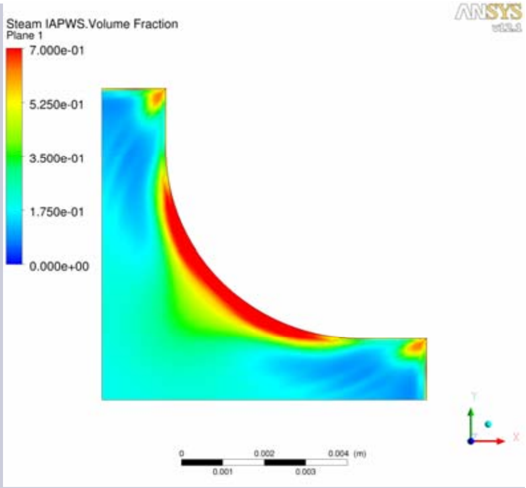
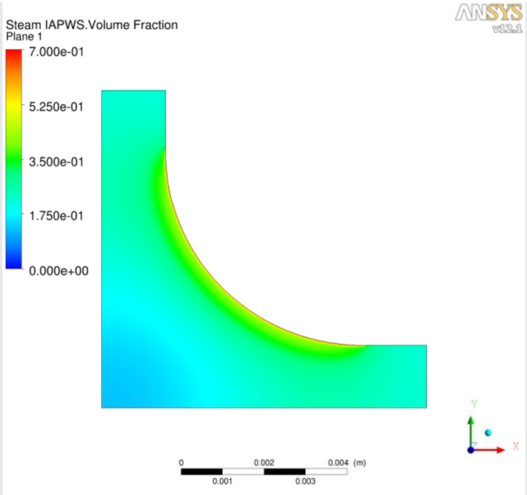
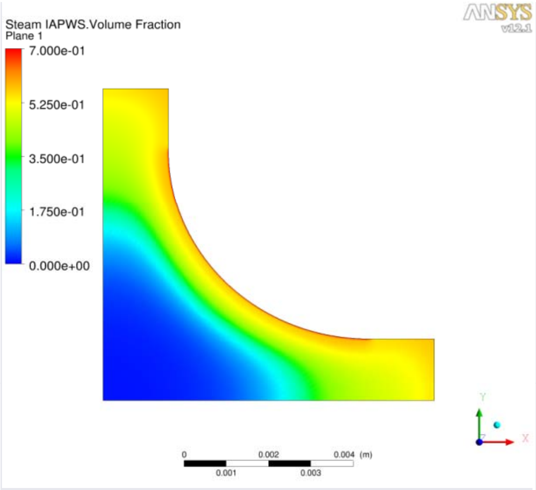
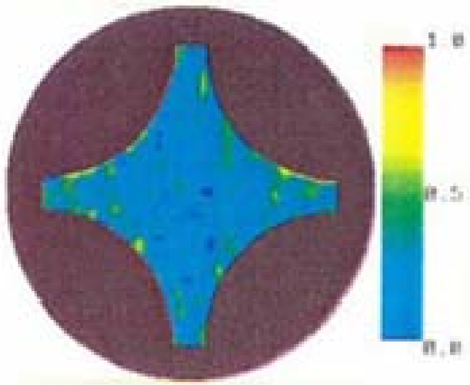
Testcase	Pressure	Inlet Temp.	Power	Mass Flux	r_v (meas.)
1.3221	125 bar	294.4 °C	1.29 MW m^{-2}	3083 kg m^{-2} s^{-1}	0.053
1.3222	125 bar	309.5 °C	1.29 MW m^{-2}	3028 kg m^{-2} s^{-1}	0.357
1.3223	125 bar	319.7 °C	1.30 MW m^{-2}	3083 kg m^{-2} s^{-1}	0.546
1.4121	100 bar	274.1 °C	1.51 MW m^{-2}	3056 kg m^{-2} s^{-1}	0.097
1.4122	100 bar	304.5 °C	1.50 MW m^{-2}	3028 kg m^{-2} s^{-1}	0.636

- Different system pressure
- Systematic variation in liquid subcooling @ inlet
- CFD simulations carried out on Mesh 1 & Mesh 2

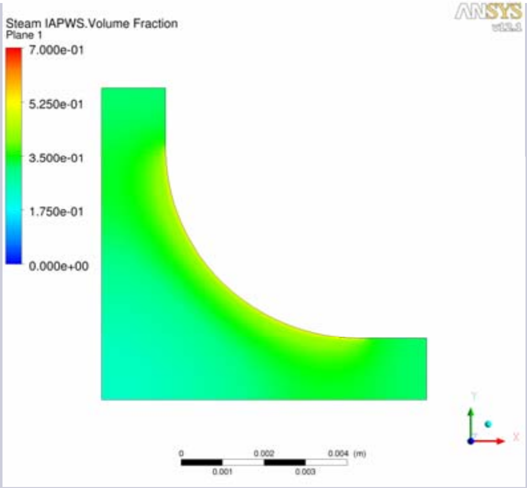
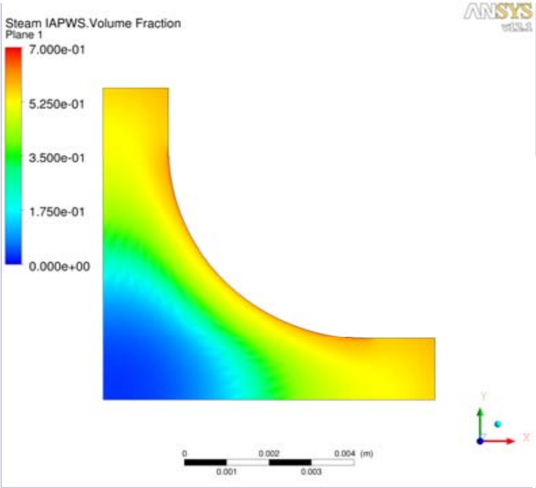
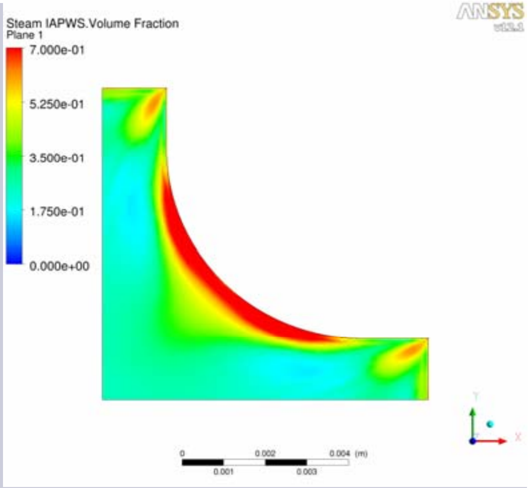
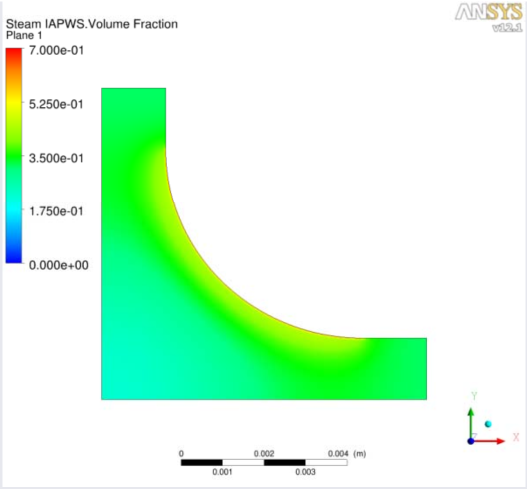
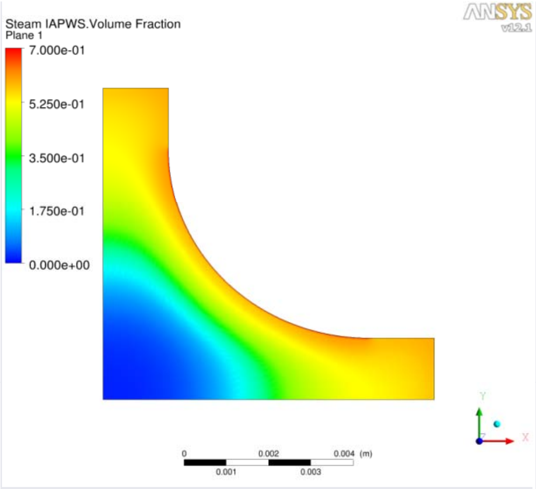
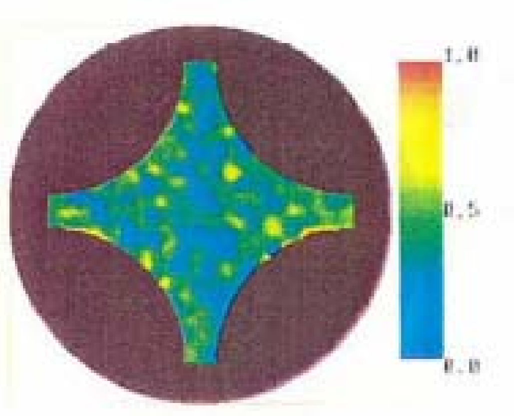
CFD Solver Results: 1.2211

	SST_TD	SST_NDF	EARSM_ND
Mesh 1			
Mesh 2			Experiment: 

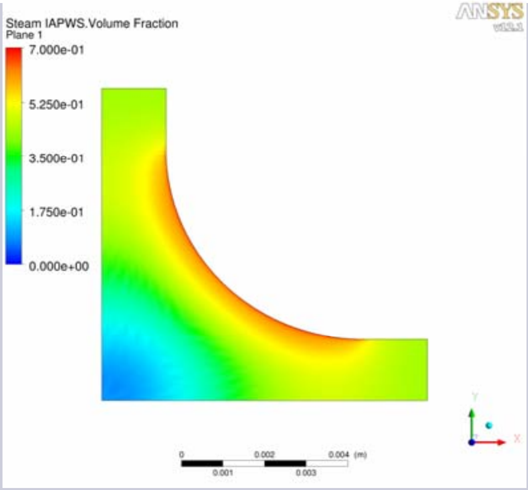
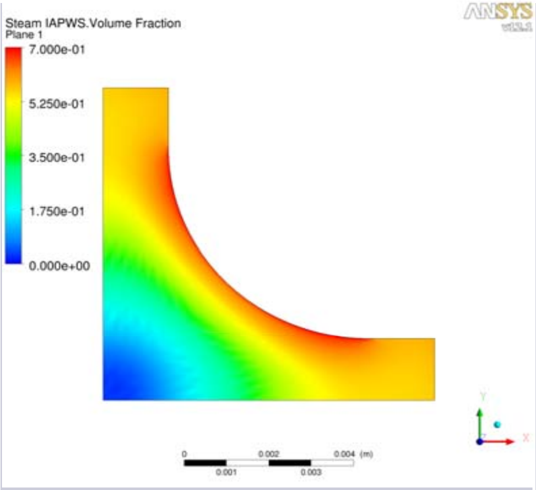
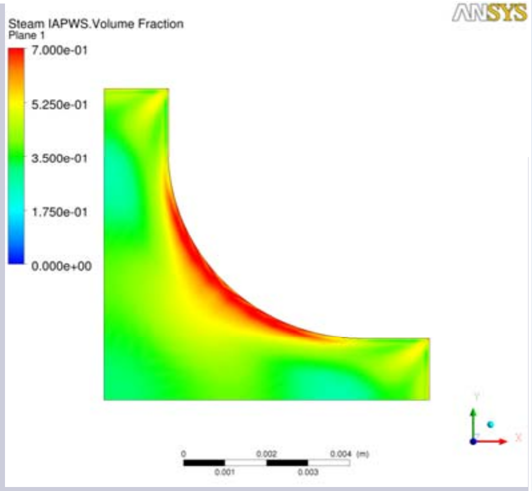
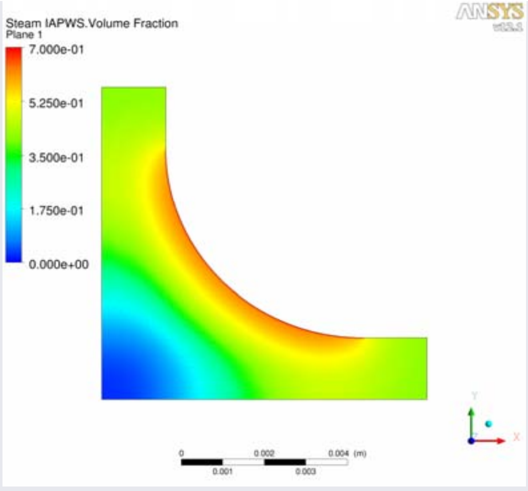
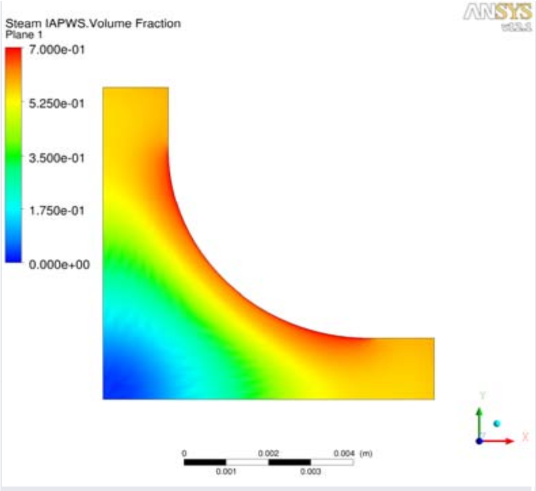
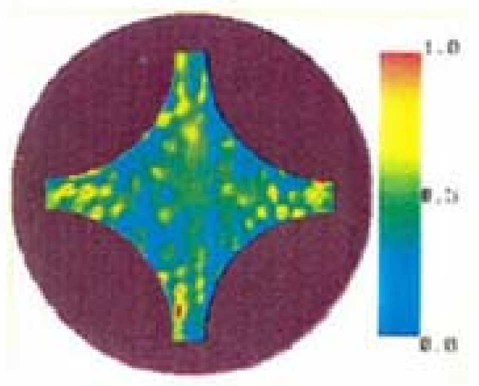
CFD Solver Results: 1.2223

	SST_TD	SST_NDF	EARSM_ND
Mesh 1			
Mesh 2			Experiment: 

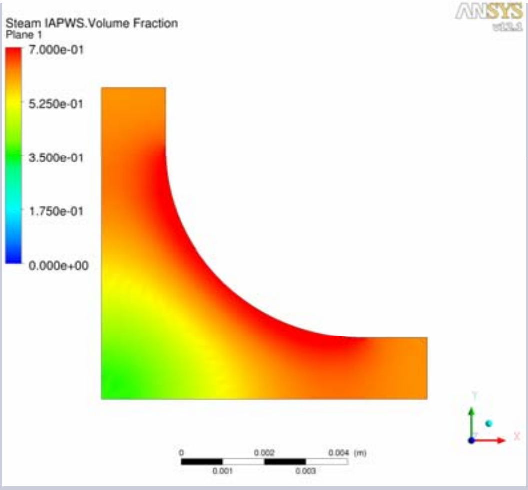
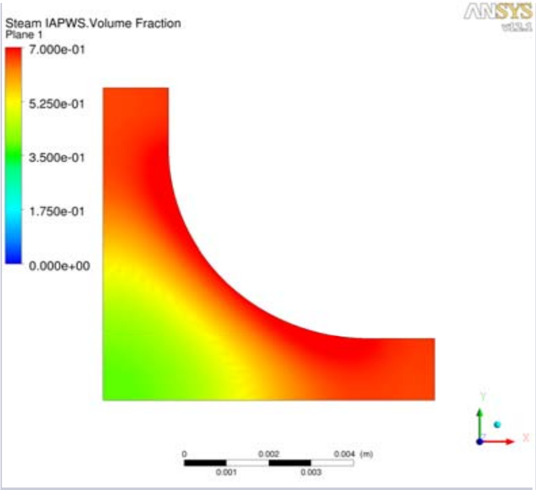
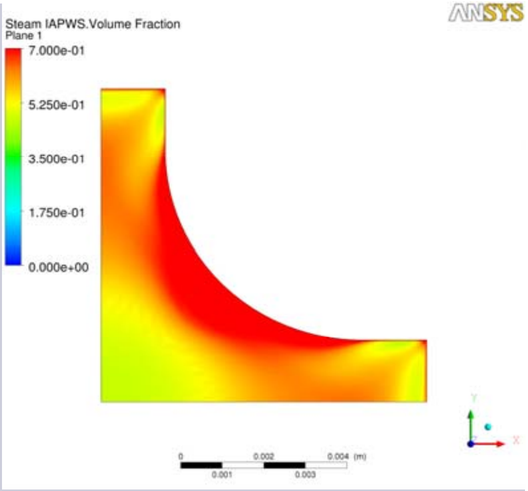
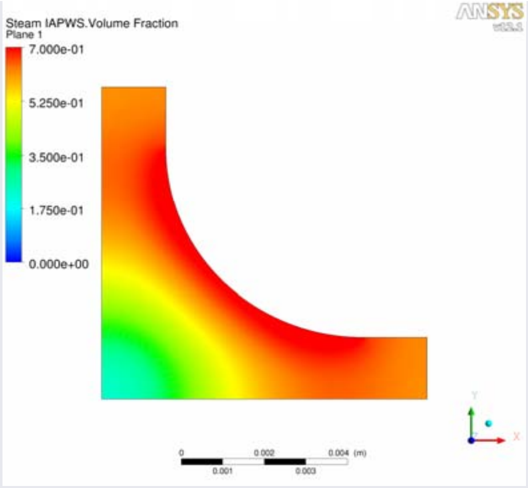
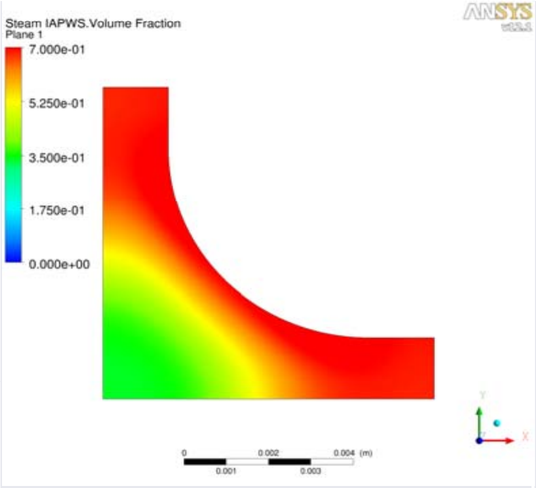
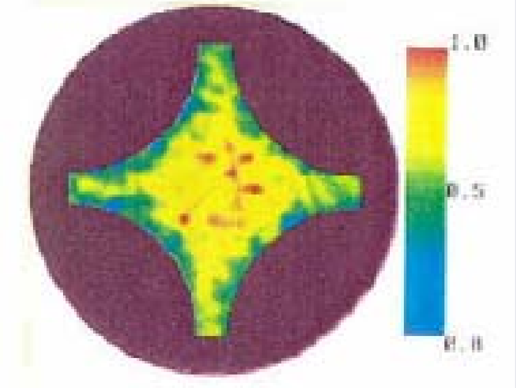
CFD Solver Results: 1.2237

	SST_TD	SST_NDF	EARSM_ND
Mesh 1			
Mesh 2			Experiment: 

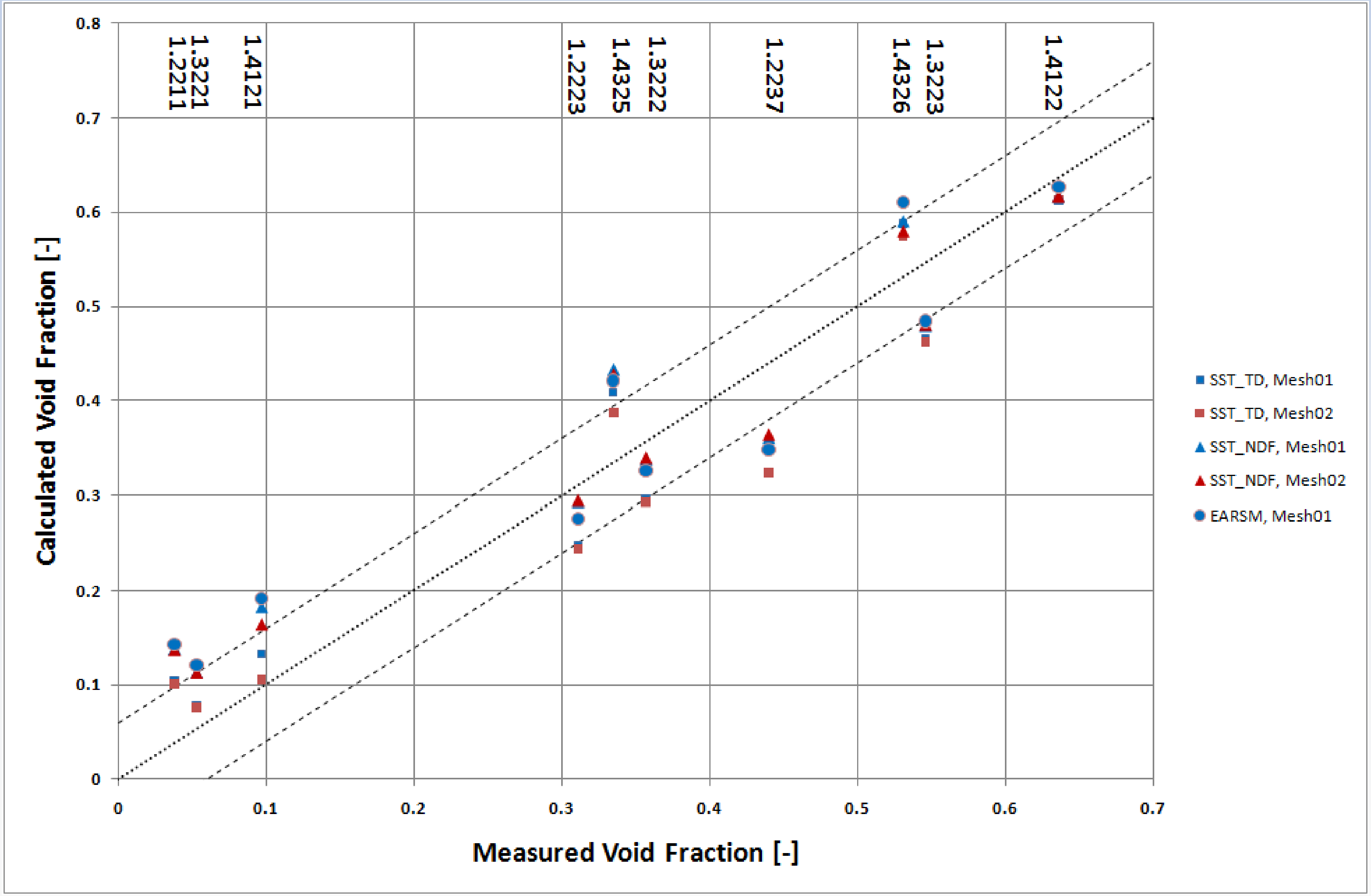
CFD Solver Results: 1.4325

	SST_TD	SST_NDF	EARSM_ND
Mesh 1			
Mesh 2			Experiment: 

CFD Solver Results: 1.4326

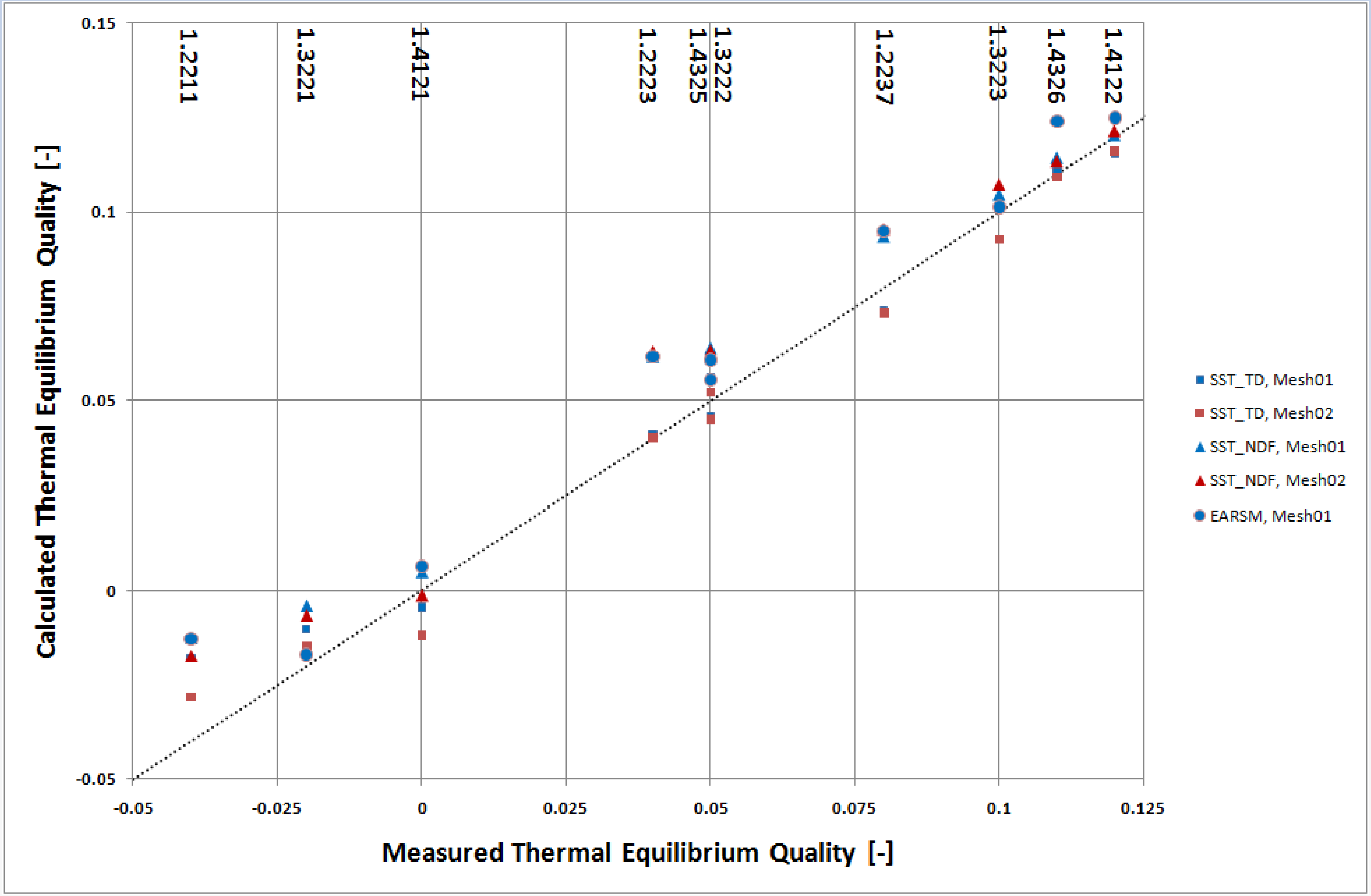
	SST_TD	SST_NDF	EARSM_ND
Mesh 1			
Mesh 2			Experiment: 

CFD Solver Results
Steam VF vs. Data



CFD Solver Results

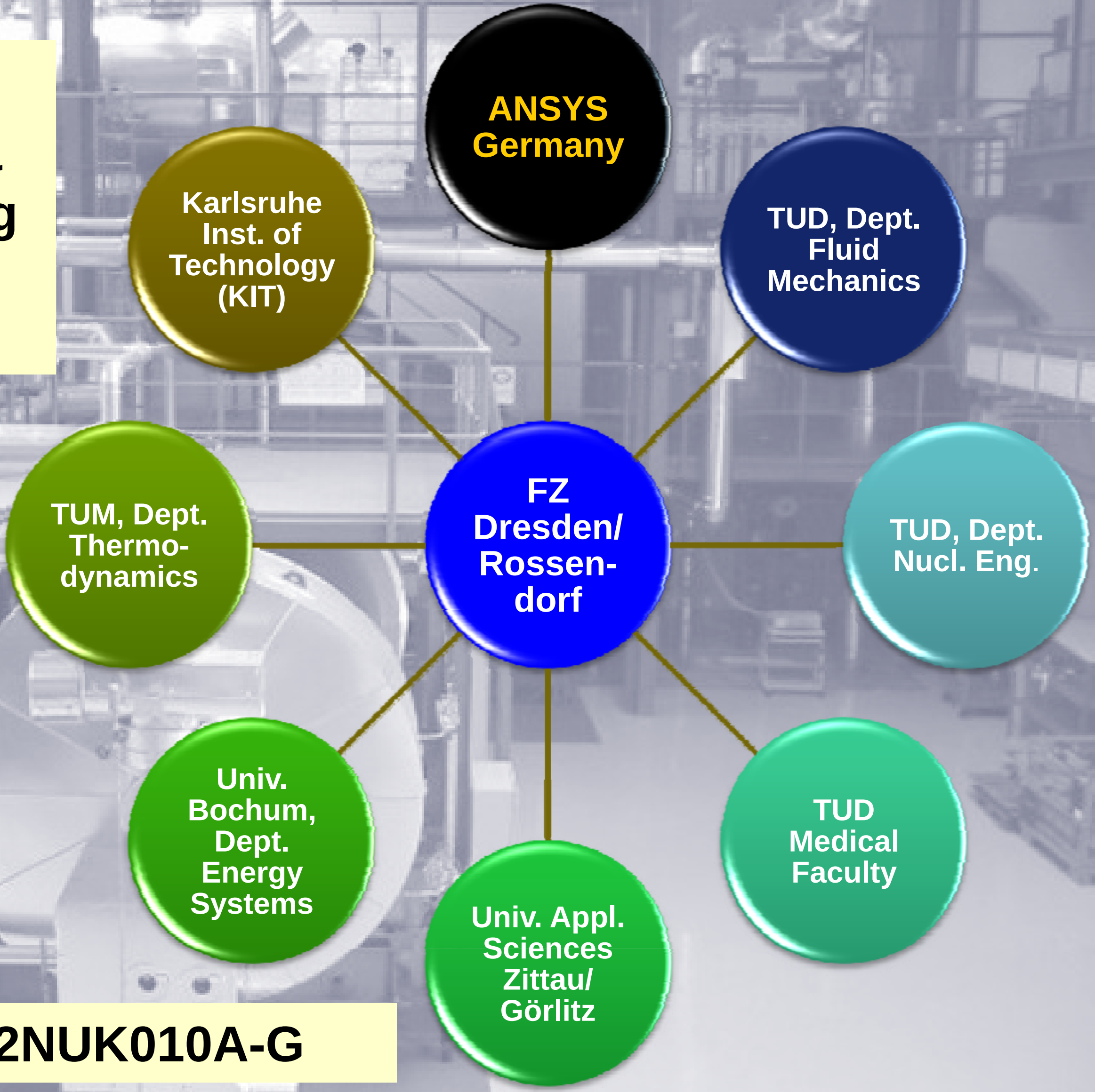
Thermal Equilibrium Quality vs. Data



Modeling, Simulation & Experiments for Boiling Processes in Fuel Assemblies of PWR

R&D Initiative:

“Modeling, Simulation & Experiments for Boiling Processes in Fuel Assemblies of PWR”



ANSYS Germany

Karlsruhe Inst. of Technology (KIT)

TUD, Dept. Fluid Mechanics

TUM, Dept. Thermo-dynamics

FZ Dresden/ Rossen-dorf

TUD, Dept. Nucl. Eng.

Univ. Bochum, Dept. Energy Systems

Univ. Appl. Sciences Zittau/ Görlitz

TUD Medical Faculty

BMBF Project Grant No. 02NUK010A-G

Modeling, Simulation & Experiments for Boiling Processes in Fuel Assemblies of PWR

- **Ultrafast electron beam X-ray CT (ROFEX) of heated rod bundle in titanium pipe on TOPFLOW @ FZD:**

Principle

Setup

High resolution volume fraction & velocity measurements

Images by courtesy of U. Hampel, F. Fischer, FZD

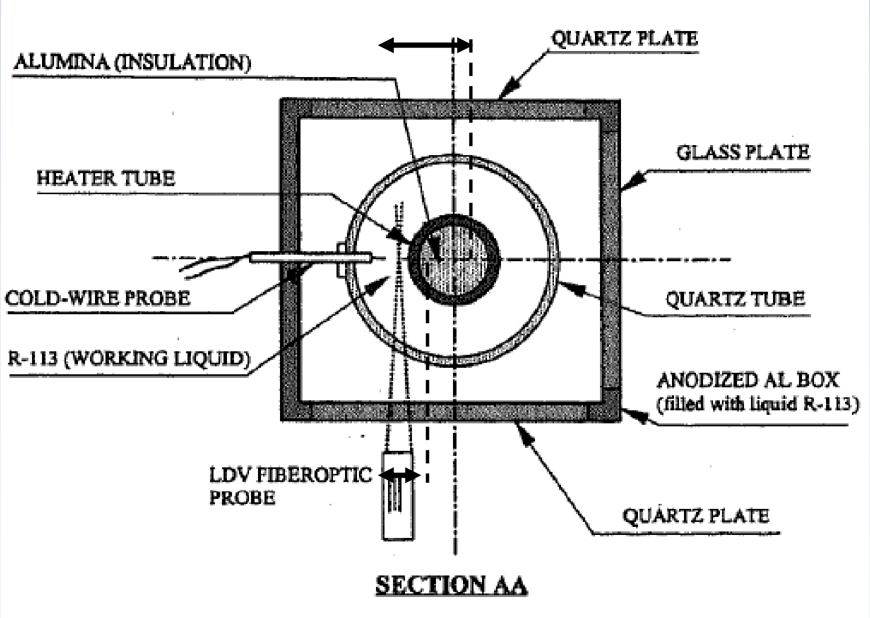
Modeling, Simulation & Experiments for Boiling Processes in Fuel Assemblies of PWR

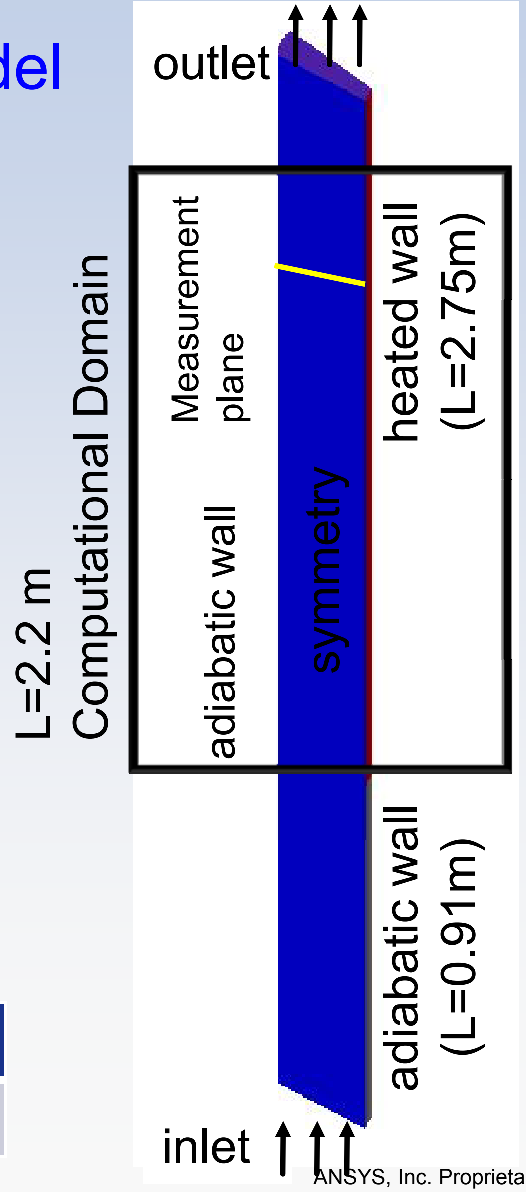
Coupling of RPI wall boiling and MUSIG model

- Experiment by Roy et al. (2002)
- Boiling of R-113 in circular heated annulus

r_i = 7.89mm

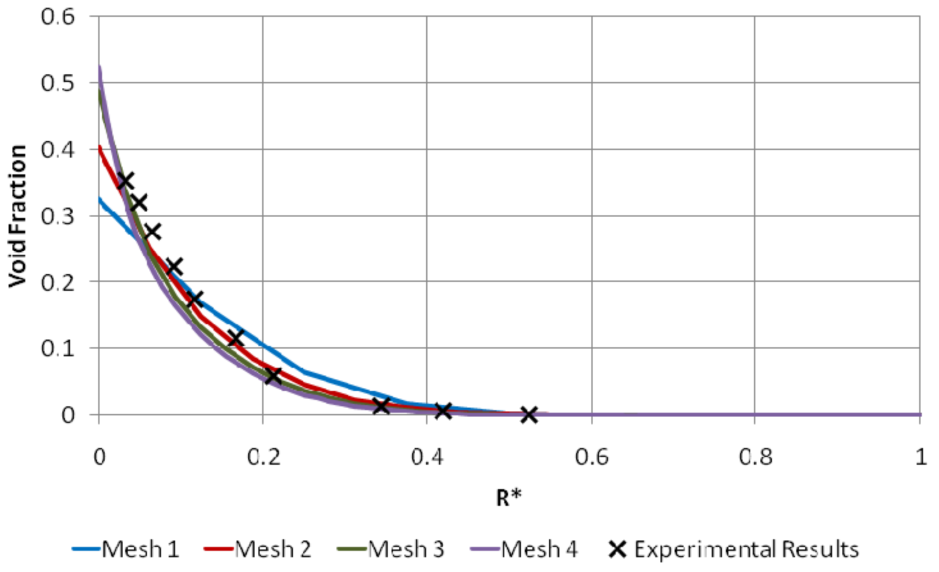
r_o = 19.01mm

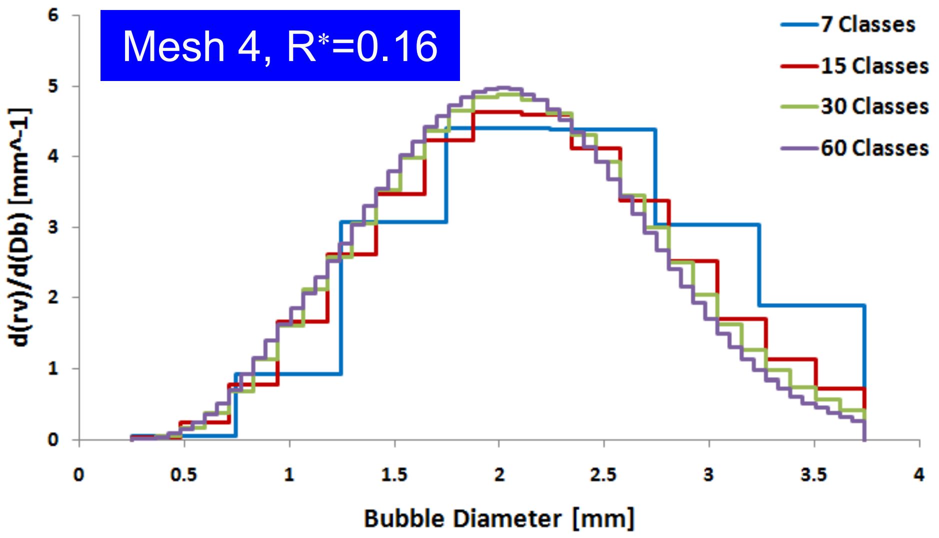


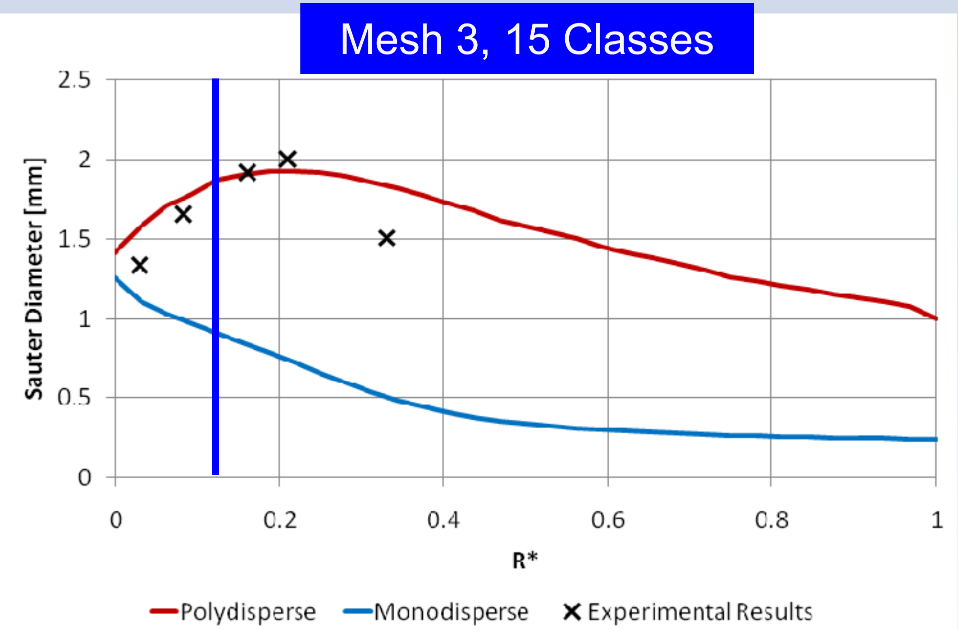


Pressure	Inlet Temp.	Mass Flux	Power
2.69 bar	50.2 C	784 kg m^{-2} s^{-1}	116 kW m^{-2}

Coupling of RPI Wall Boiling & MUSIG Model in ANSYS CFX







Conclusions & Outlook

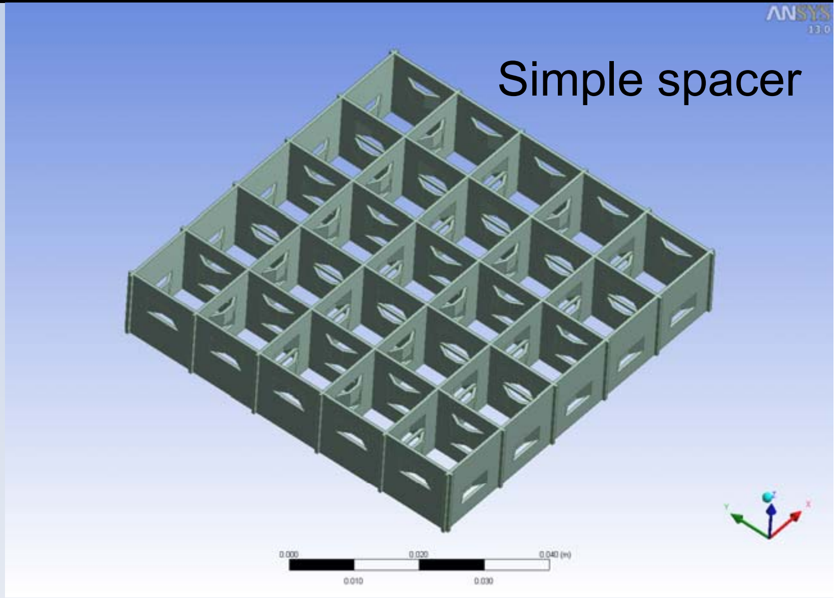
- In general good agreement to data obtained:
  - Broad range of test conditions investigated
  - Trends are correctly covered by ANSYS CFX
  - CFD predicts higher steam VF for low volume fraction cases
  - More difficult convergence for large liquid subcooling
- Improvement of CFD modeling:
  - Uncertainty in MPF turbulence modeling and in MPF flow regime transition modeling (for large steam VF cases)
  - Uncertainty in interfacial momentum transfer modeling close to the heated surface (violation of model assumptions)
  - Provided PSBT data not very suitable for CFD model advancements

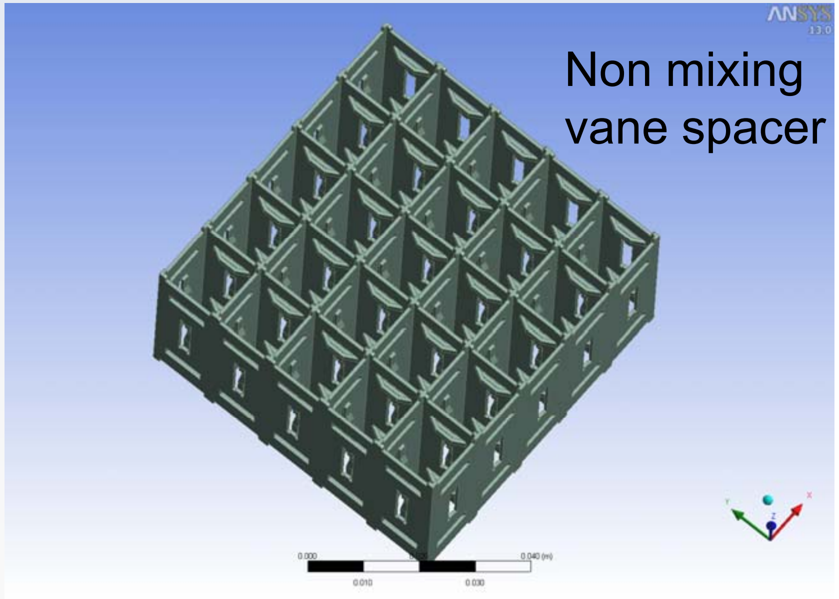
→HZDR boiling and HiRes tomography experiments

→Flow morphology transition, RPI & MUSIG/DQMOM coupling, DNB and CHF model in ANSYS Fluent,…

PSBT Phase I, Exercise 2

- Created CAD model of rod bundle B5 in ANSYS Design Modeler 13.0
  - Simple spacer
  - Non mixing vane spacer
  - Mixing vane spacer

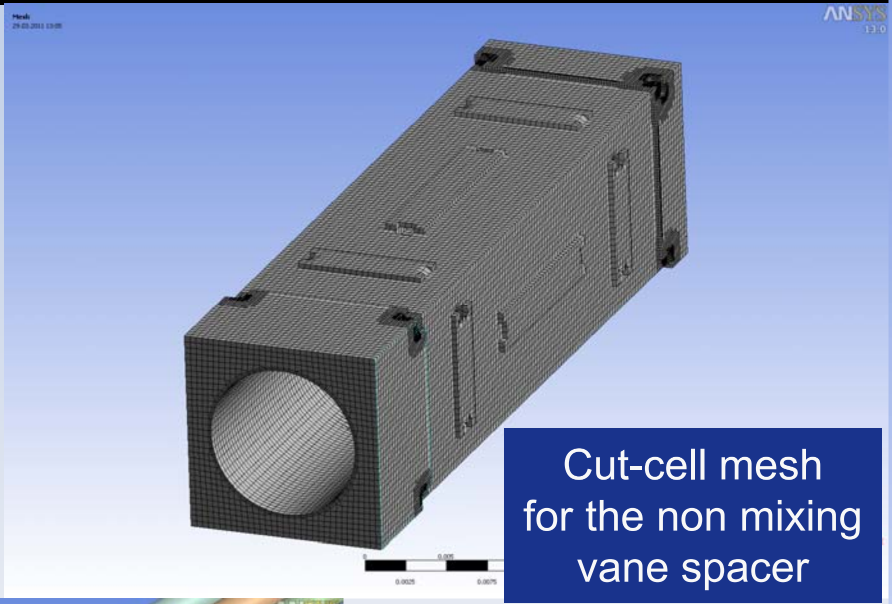




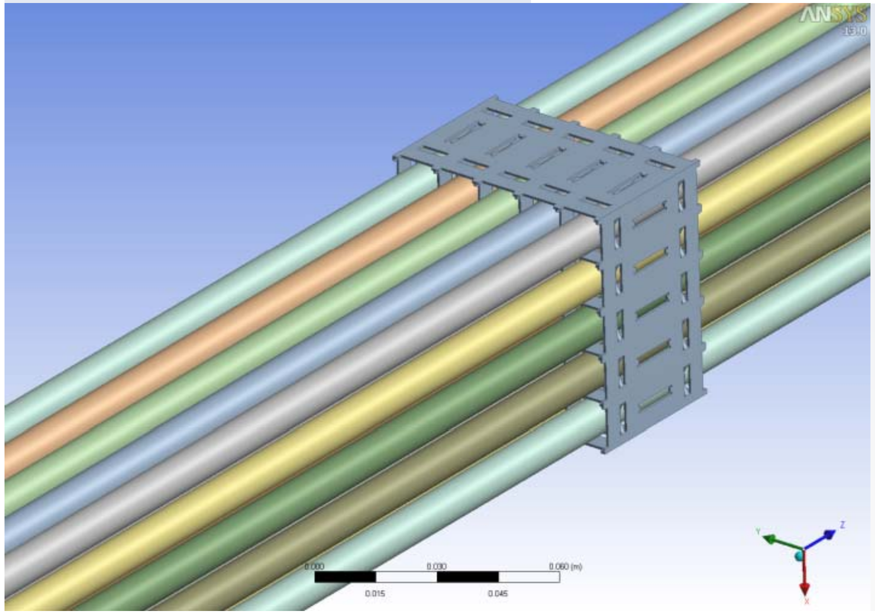
PSBT Phase I, Exercise 2

- 120.000 cut-cell mesh for NMV spacer
- Realizing similar resolution as for exercise 1

→ min. 216 million elements for whole bundle (and still rather coarse)

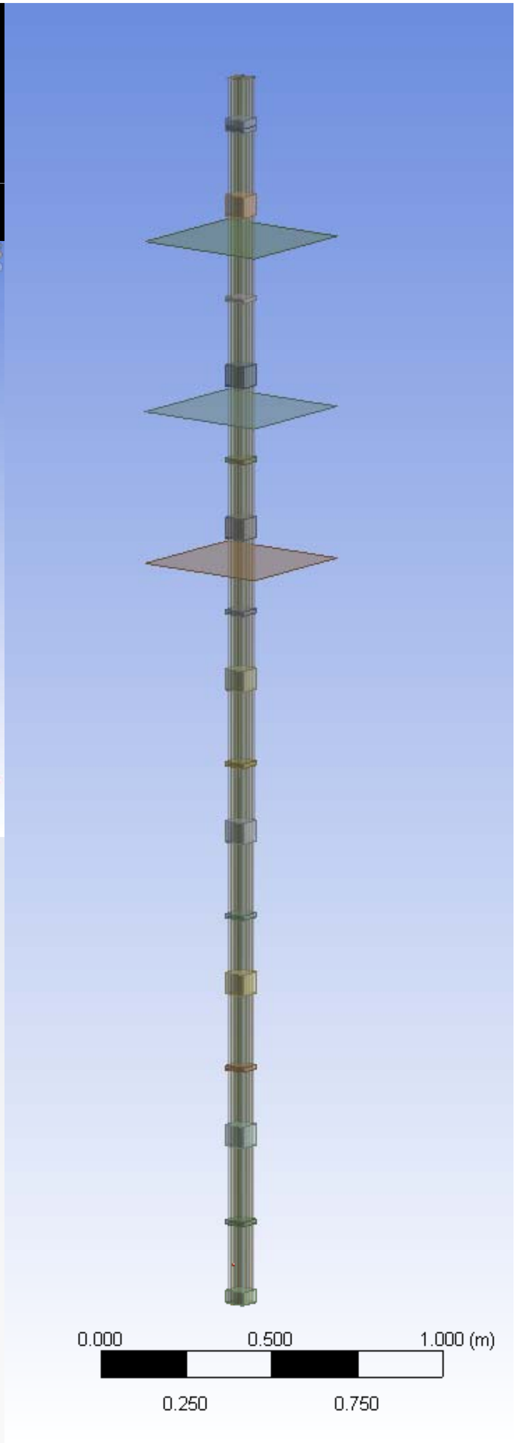


Cut-cell mesh for the non mixing vane spacer



- Central channel + surrounding quarter rods

→ 45 million elements



PSBT Phase I, Exercise 2

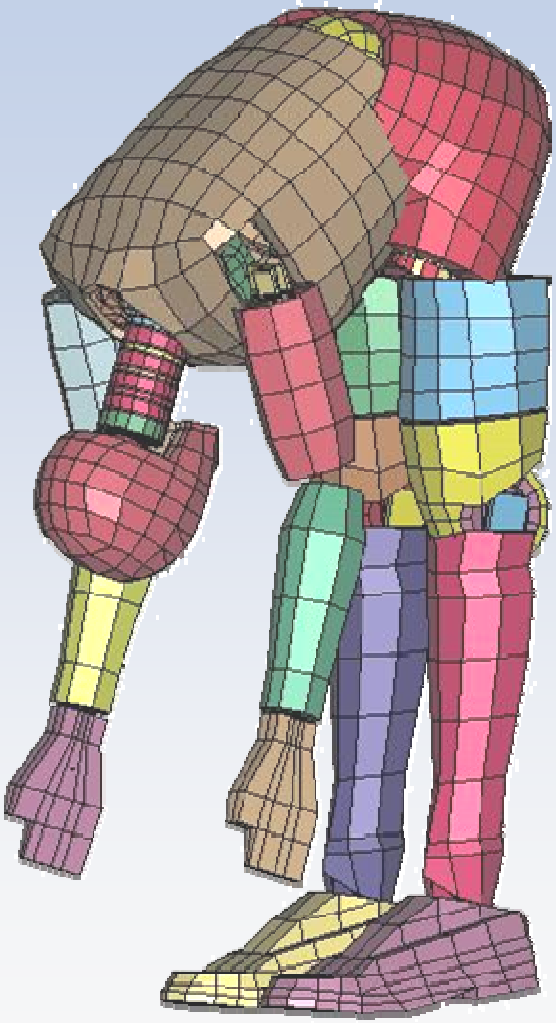
- For CFD the PSBT Phase I, Exercise 2 is more of a meshing benchmark than a multiphase flow modeling benchmark
- Massive HPC resources would be required to compute the benchmark for just a single test condition
  → … and then to compare to <u>3 real numbers</u>
- OECD IBP-2 benchmark based on KAERI MATIS-H experiments are better suited to compare CFD with field measurement data for velocity fields upstream of grid spacers
  → higher benefit for CFD modeling advancement

Acknowledgements

- This research has been supported by the German Ministry of Education and Research (BMBF, *Grant No. 02NUK010G*) in the framework of the R&D funding concept of BMBF *"Basic Research Energy 2020+"*

SPONSORED BY THE

Federal Ministry
of Education
and Research



Thank You!