Prediction of 3-Dimensional Air-Water Bubbly Flow Around a Movable Orifice

Th. Frank[1], H.-M. Prasser[2]

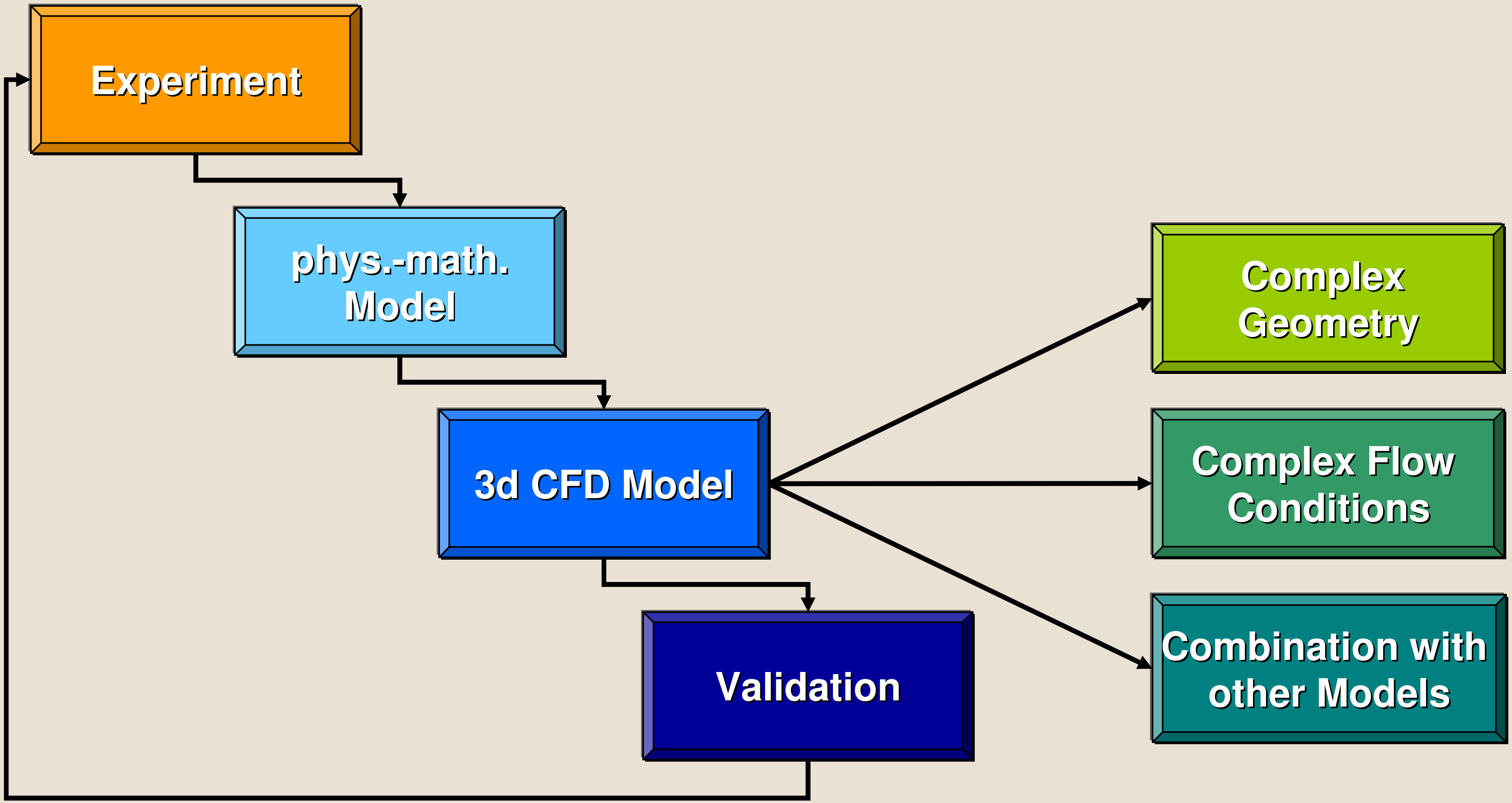
1)ANSYS Germany, 83624 Otterfing
Thomas.Frank@ansys.com

2)ETH Zürich, Switzerland
hprasser@ethz.ch

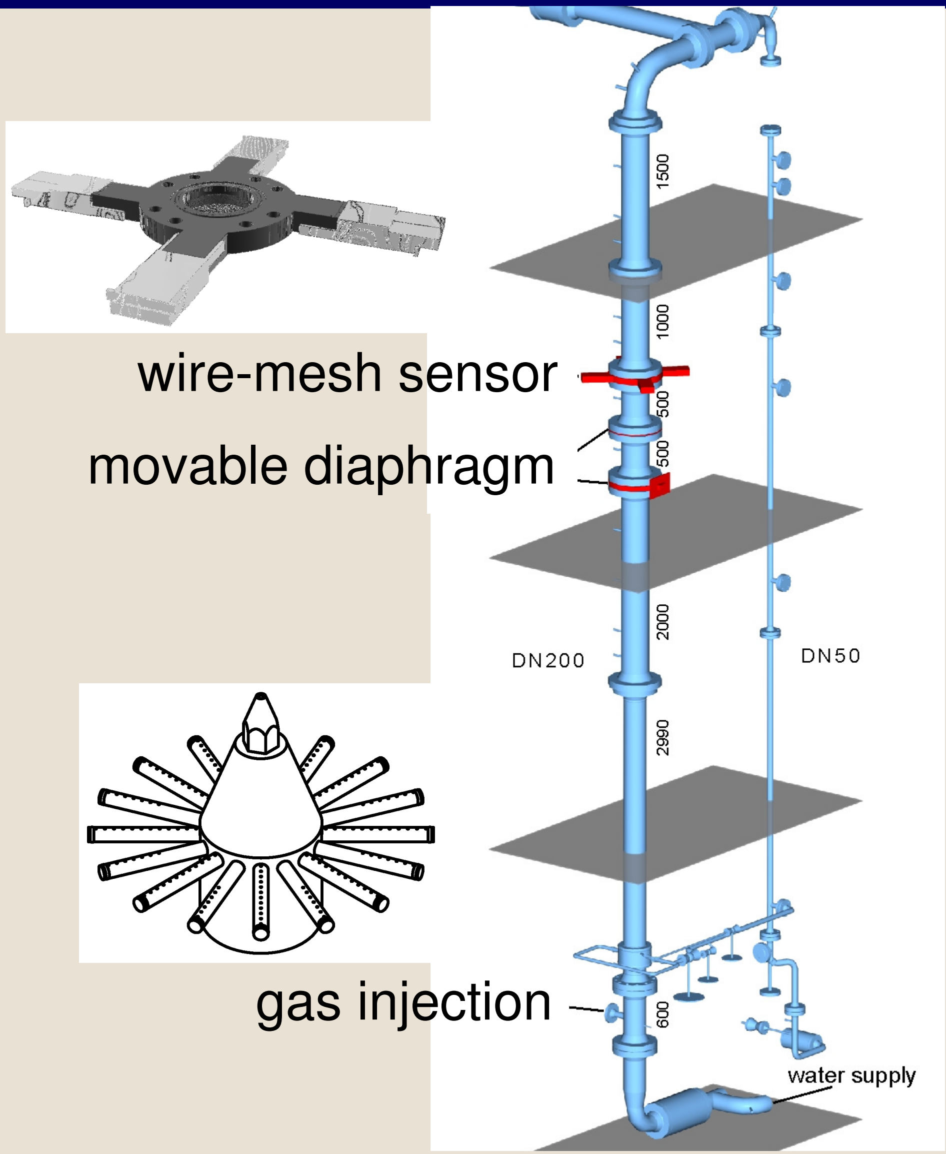
Overview

- Introduction
  - Methodology of CFD model development
- Experiments at TOPFLOW test facility @ FZR
- CFX blind test calculations
  - Mesh hierarchy
    → CFD Best Practice Guidelines
  - Setup of the flow physics
    → thermohydraulic CFD models
  - Results of the CFD simulation
- CFD model validation and comparison to data from complex 3-dimensional flow around obstacle
- Summary & Outlook

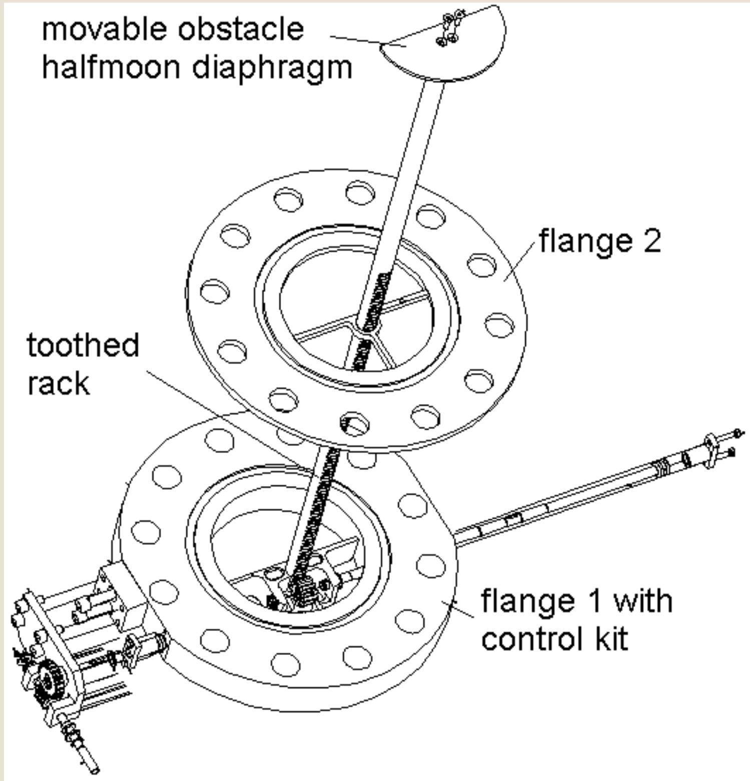
Methodology of CFD Model Development



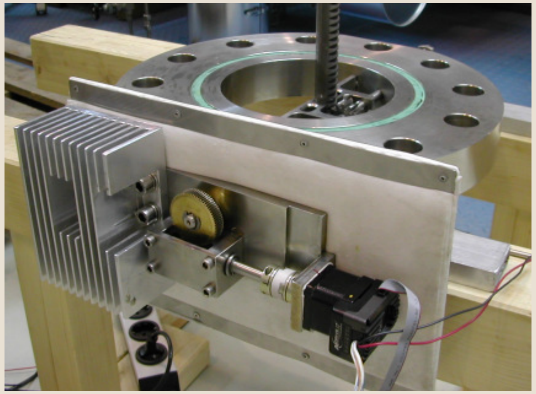
TOPFLOW Test Facility @ FZR

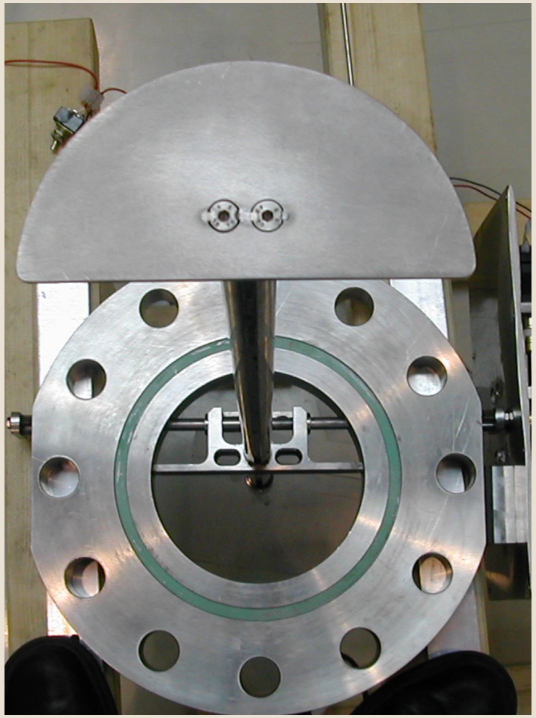


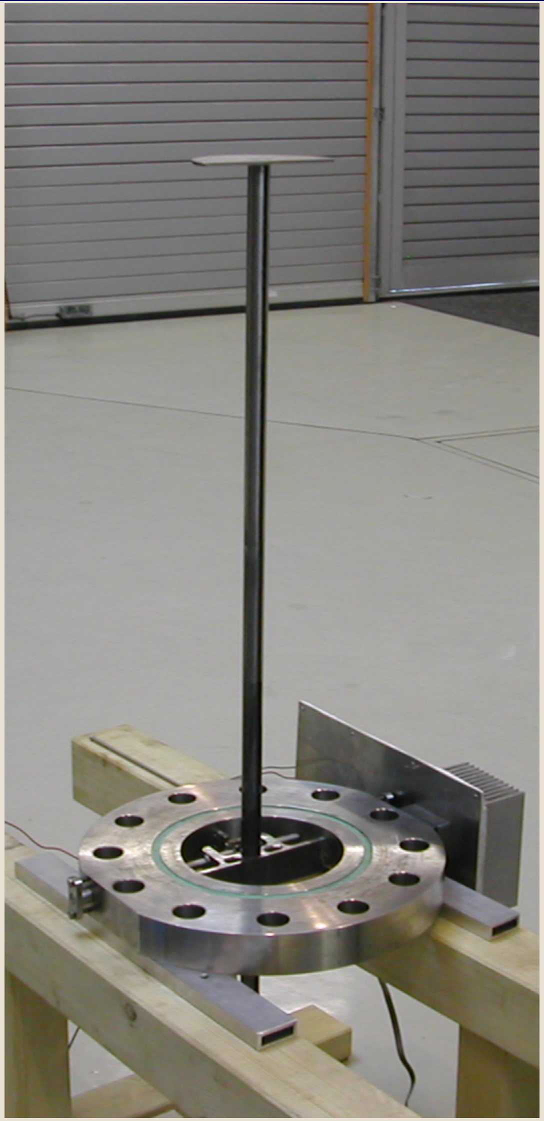
movable diaphragm



The Movable Orifice in TOPFLOW







3-dimensional Bubbly Flow Around Movable Obstacle

ANSYS

Blind test for CFX model application to flow around obstacle:

- 3-dimensional flow; steady state
- Turbulent monodisperse 2-phase flow
- Flow stagnation, recirculation & re-attachment
- Phase separation

Flow geometry and test case conditions:

- CAD data from obstacle geometry from FZR
- 1.5m of TOPFLOW pipe up- and downstream of the obstacle (L≈7.5D, D=198 mm)
- Air-water flow at 1 bar, 25 °C
- Test case conditions of test case TOPFLOW-074

TOPFLOW-074 Test Case Conditions from Test Matrix

- Selection of test case conditions:

superficial gas velocity

superficial water velocity	m/s	0.0368	0.0574	0.0898	0.14	0.219	0.342	0.534	0.835
	1.611	075	086	097	108	119	130	141	152
	1.017	074	085	096	107	118	129	140	151
	0.405	072	083	094	105	116	127	138	149
	0.102	069	080	091	102	113	124	135	146

- **Experiments:**
  - **Air-Water at 1 bar, 25 °C**
  - **Saturated Steam-Water at 65 bar, 280 °C**
  - **some tests at 10, 20, 40 bar**

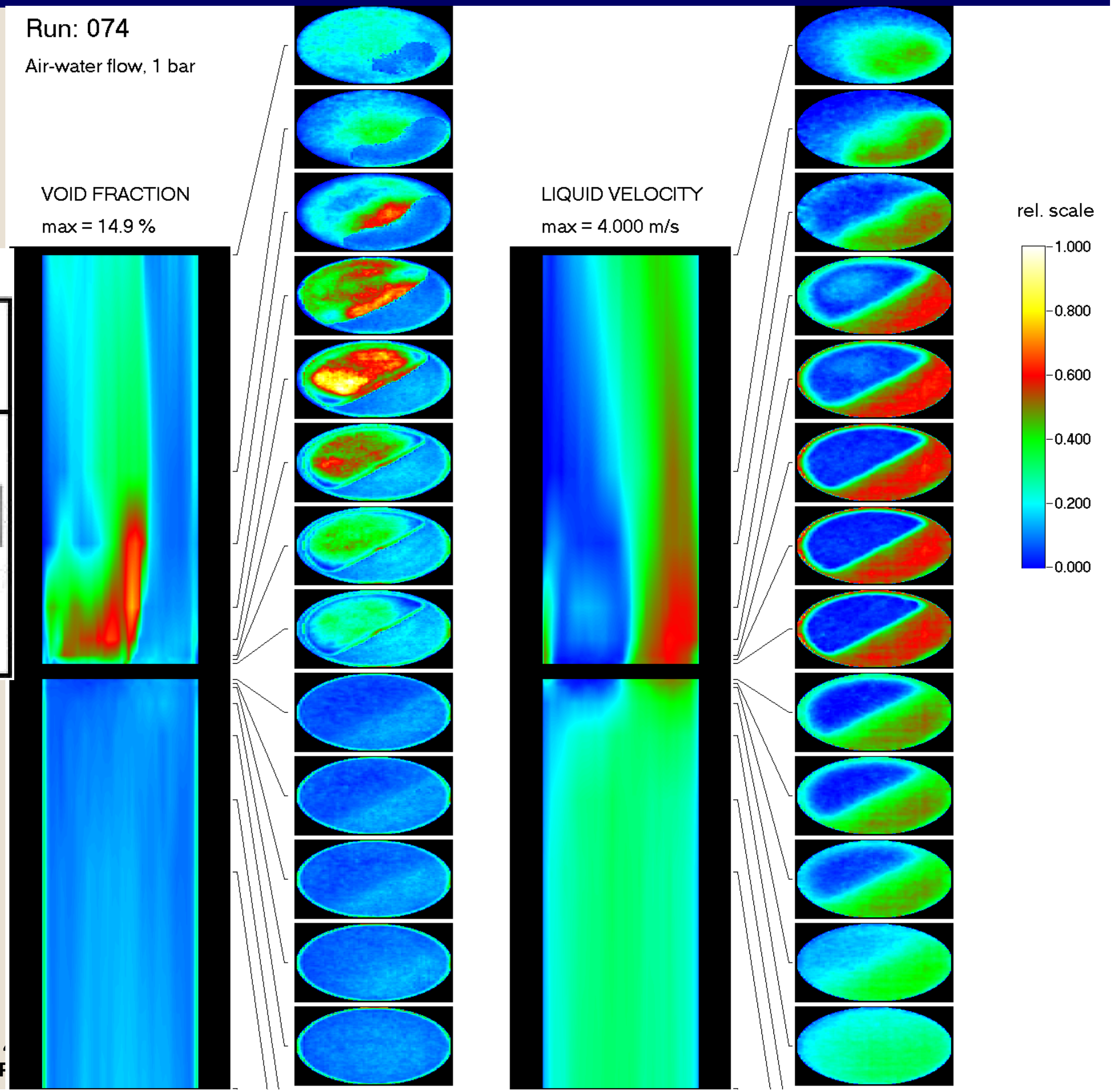
- TOPFLOW-074 subject of validation in the past
- Superficial velocities: J_G=0.0368 m/s
  J_L=1.017 m/s
- Wire-mesh sensor measurements at locations:
  z=±10, 15, 20, 40, 80, 160, 250, 520mm

Measurement Data for Test Case 074

superficial gas velocity

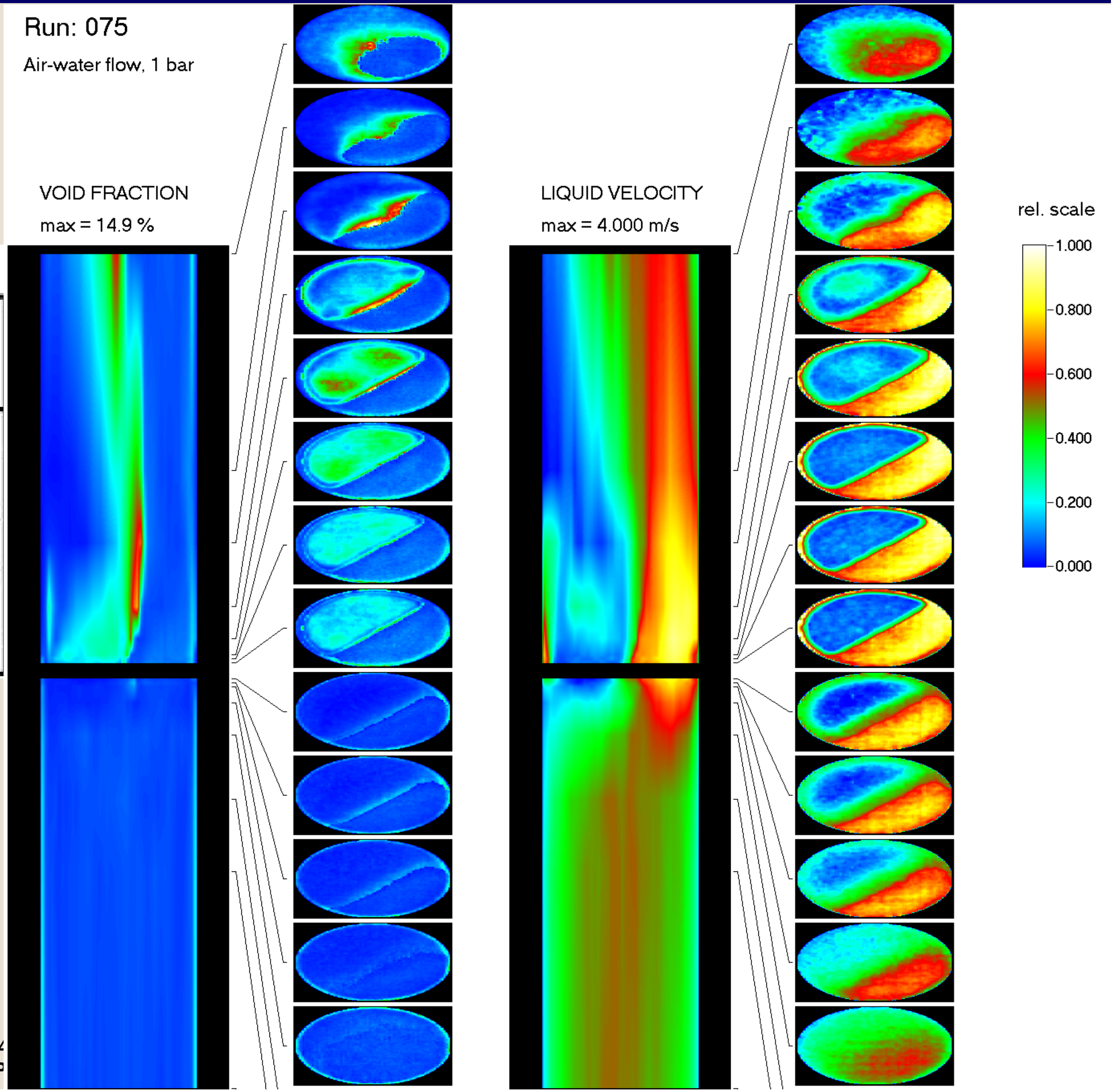
superficial water velocity

m/s	0.0368	0.0574	0.0898	0.14	0.219	0.342	0.534	0.835
1.611	075	086	097	108	119	130	141	152
1.017	074	085	096	107	118	129	140	151
0.405	072	083	094	105	116	127	138	149
0.102	069	080	091	102	113	124	135	146



Measurement Data for Test Case 075

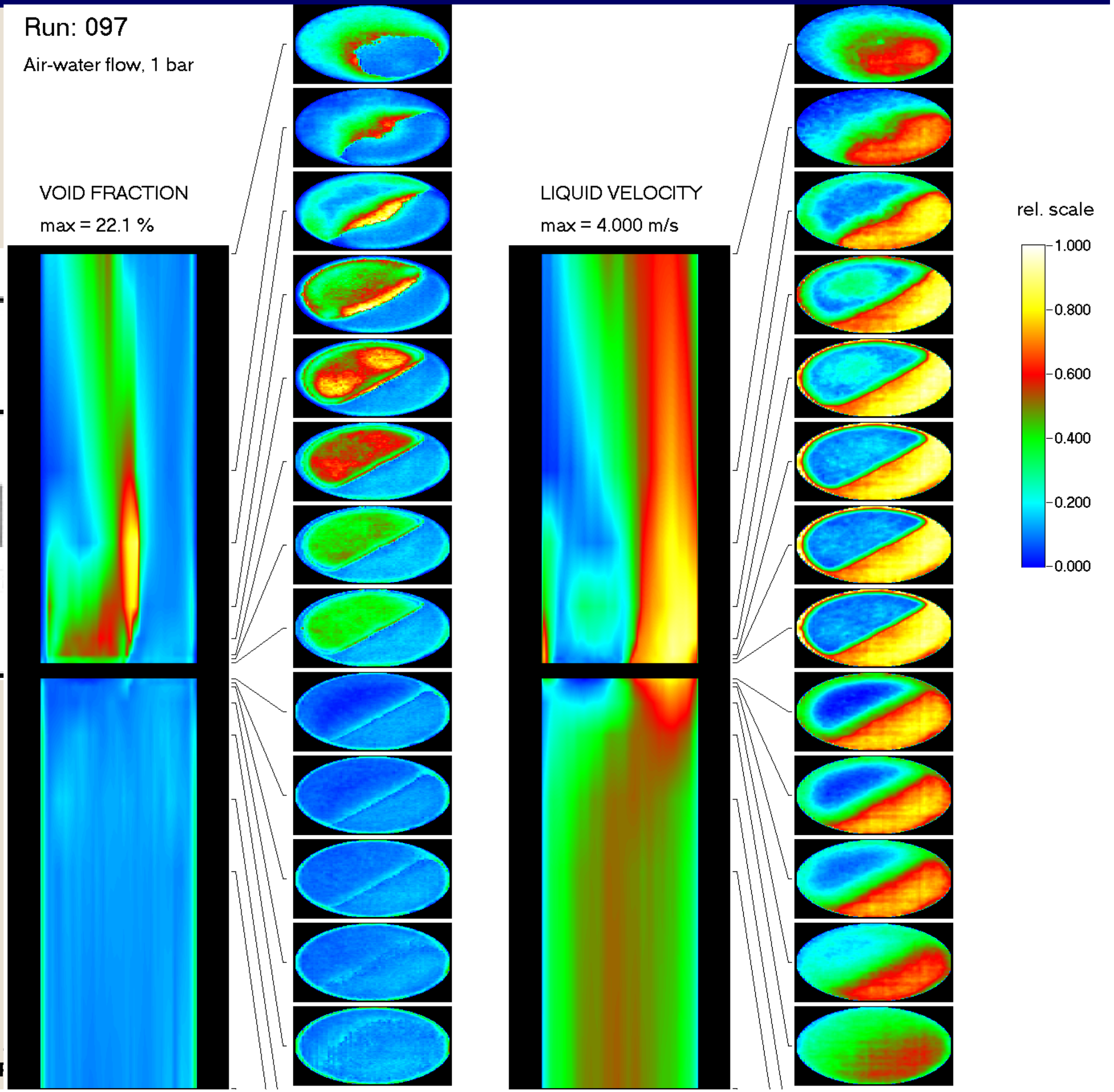
m/s	superficial gas velocity 0.0368	0.0574	0.0898	0.14	0.219	0.342	0.534	0.835
superficial water velocity 1.611	075	086	097	108	119	130	141	152
1.017	074	085	096	107	118	129	140	151
0.405	072	083	094	105	116	127	138	149
0.102	069	080	091	102	113	124	135	146



Measurement Data for Test Case 097

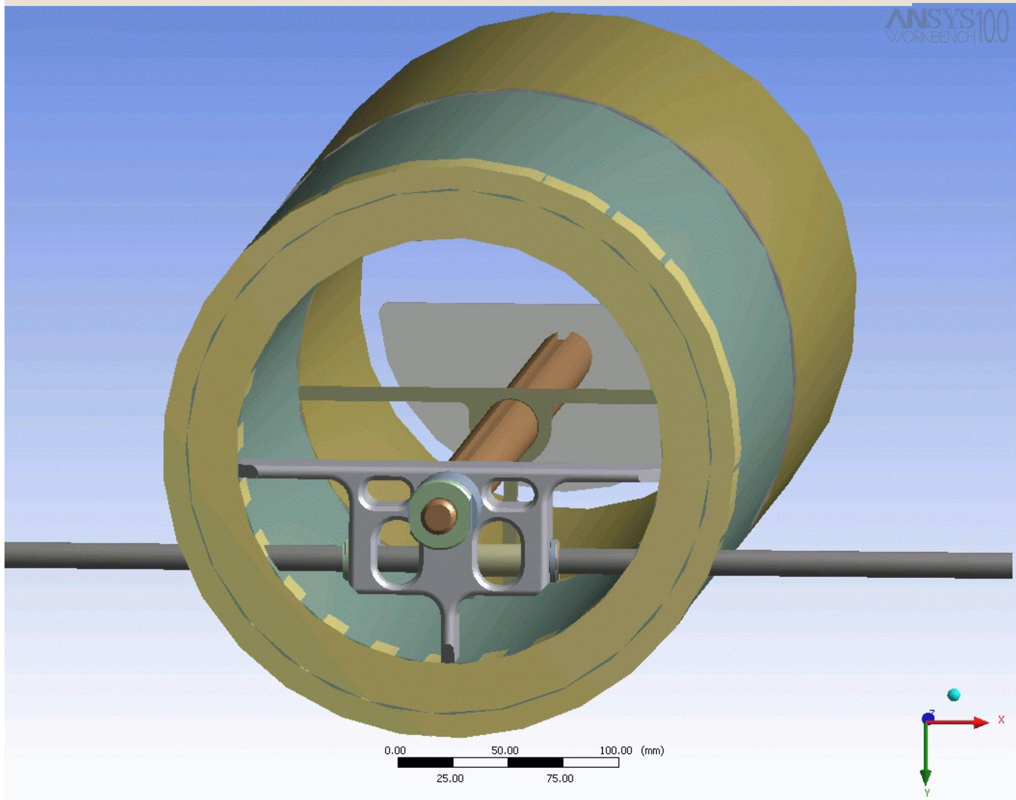
superficial gas velocity

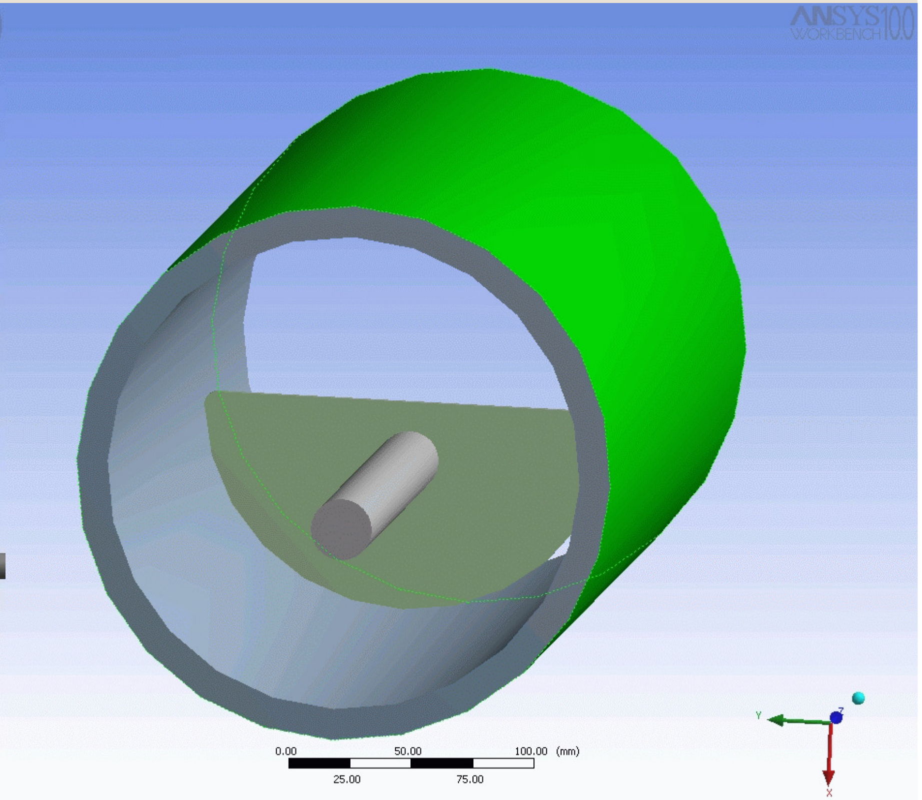
superficial water velocity m/s	0.0368	0.0574	0.0898	0.14	0.219	0.342	0.534	0.835
1.611	075	086	097	108	119	130	141	152
1.017	074	085	096	107	118	129	140	151
0.405	072	083	094	105	116	127	138	149
0.102	069	080	091	102	113	124	135	146



Geometry & Mesh Generation

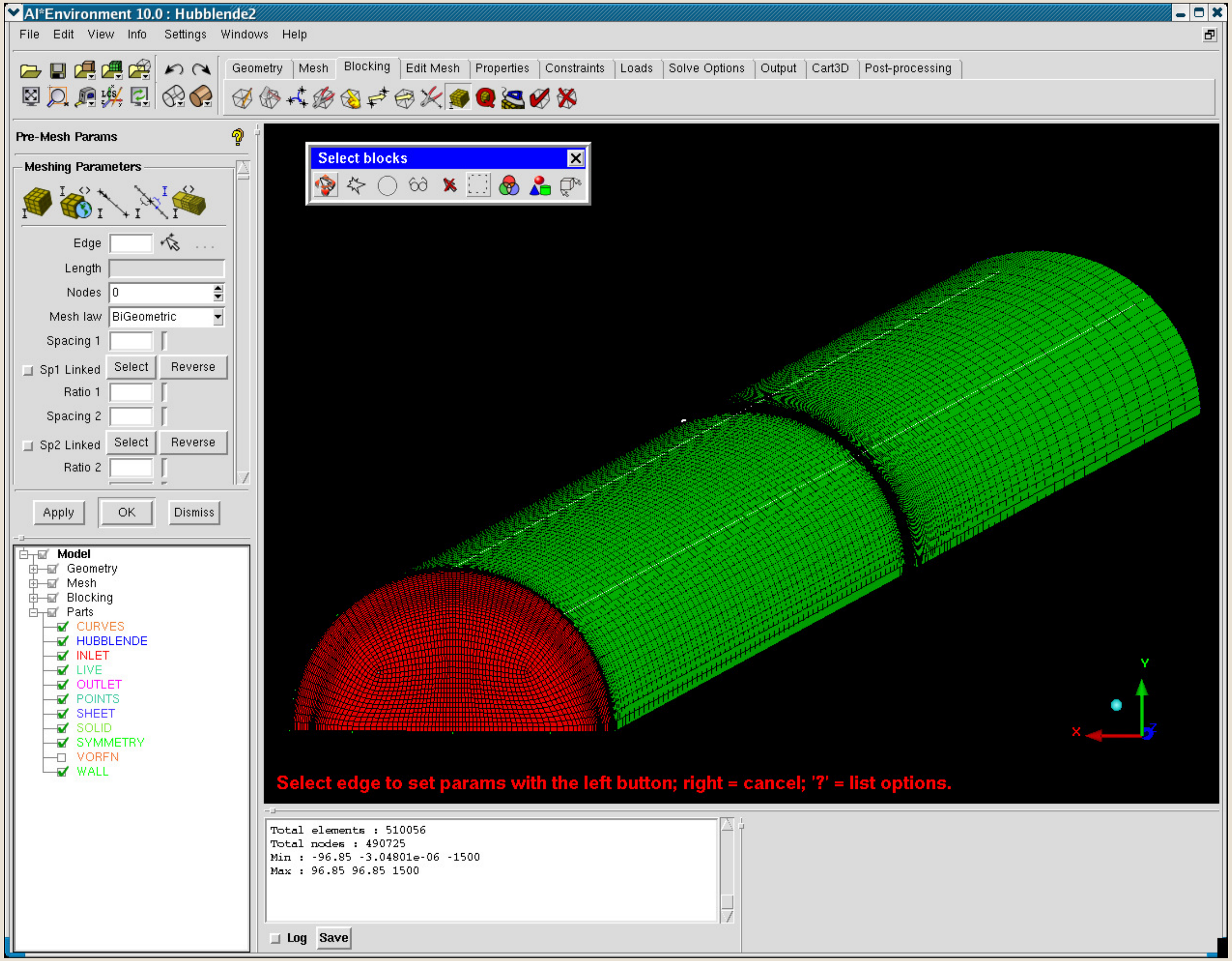
- CAD data import in ANSYS Workbench (ACIS SAT files)
- clean-up of CAD geometry
- neglecting obstacle support and drive
- taking into account axial symmetry





Geometry & Mesh Generation

- Mesh generated with ICEM- CFD 10.0
- Hexahedral mesh



Mesh Hierarchy

- Mesh hierarchy:

Grid level	No. nodes	No. elements	Yplus@wall
Grid 1	126.532	118.936	18.8,…,173.2
Grid 2	490.725	471.808	0.42,…,53.8
Grid 3	1.908.270	1.861.105	0.19,…,28.6

- mesh refinement by $\sqrt[3]{4}$ ~1.587
- near wall / near obstacle grid refinement
- modified Laplace grid smoothing

Flow Physics Setup

- Eulerian two-phase bubbly flow model:
  - fixed bubble diameter; dependent on hydrostatic pressure (or height)
    d_P=4.8,…5.2mm
  - Grace drag force
  - FAD turbulent dispersion force
  - Tomiyama lift force
  - Frank's generalized wall lubrication force
  - Sato bubble enhanced turbulence model
- Turbulence modelling:
  - cont. phase: SST turbulence model with Menter's modified automatic wall functions
  - disperse phase: zero equation disperse phase turbulence model

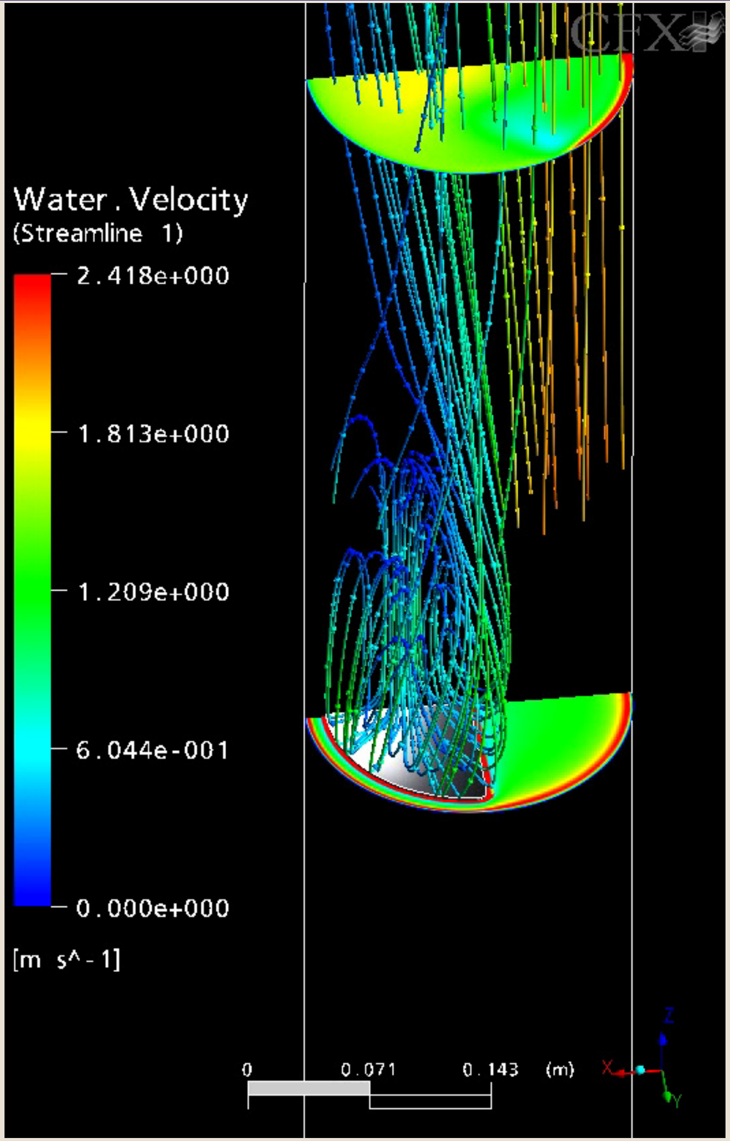
Flow Setup & Boundary Conditions

- Numerical schemes:
  - steady-state simulation
  - High resolution advection scheme
- Convergence criteria:
  - 10^{-3} MAX residuals
- Physical time scale:
  - 0.0005 s

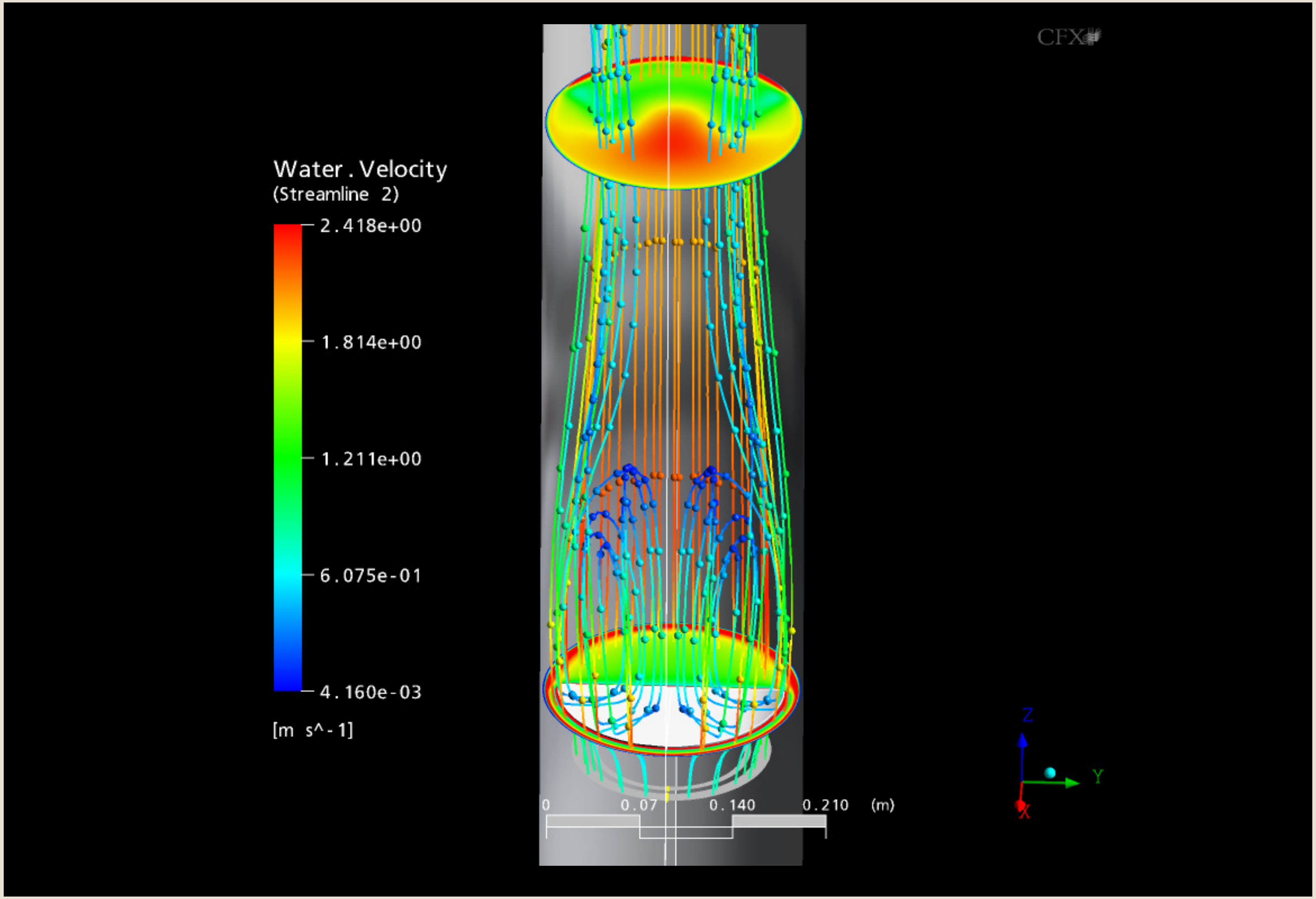
- Initialization:
  - u, v, w, r_G, r_L, k, ω from fully developed pipe flow
- Boundary conditions:
  - Inlet: same as for initialization; fully developed pipe flow profiles
  - Outlet: Average Static Pressure
  - Walls: no slip wall for cont. phase
free slip wall for disp. phase

3d Bubbly Flow Around Obstacle

- Streamlines around the obstacle
  - Vortex system around the edge of the obstacle
  - Velocity component from left to right along the
    vortex core
  - Higher ‚concentration' (residence time) in right vortex core

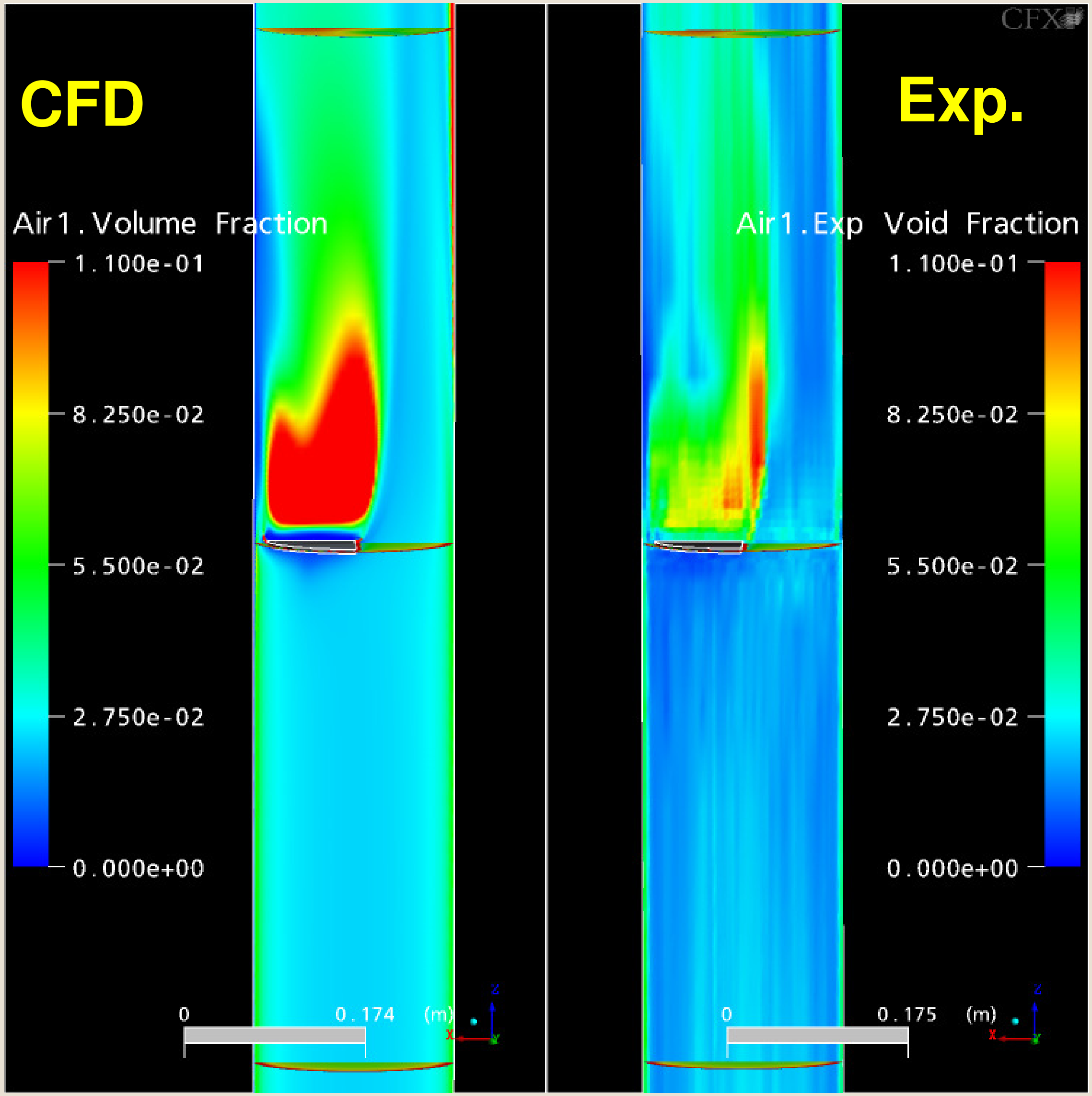


3d Bubbly Flow Around Obstacle



3d Bubbly Flow Around Obstacle
Air Void Fraction Comparison

- Comparison CFD ⇔ Experiment
- Air void fraction distribution in symmetry plane
- Import of exp. data into CFX-Post
- Pre-interpolation of exp. data to Δz=0.01m



3d Bubbly Flow Around Obstacle Air Void Fraction Comparison

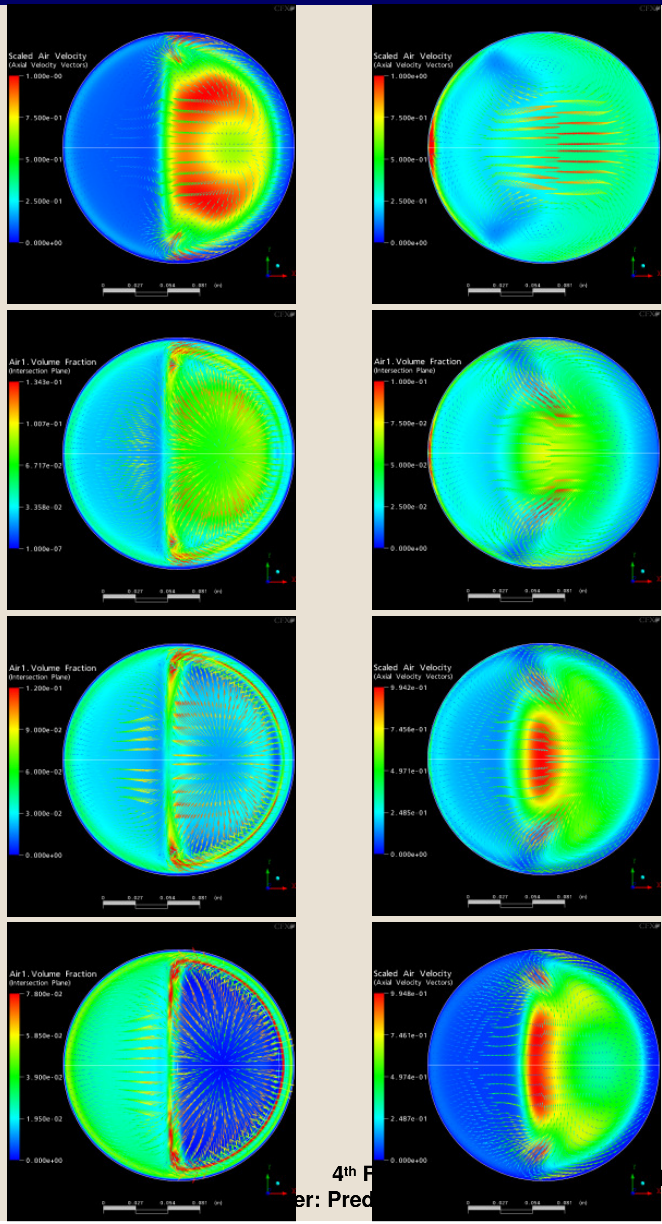
ANSYS

4) z=40mm

3) z=20mm

2) z=15mm

1) z=10mm

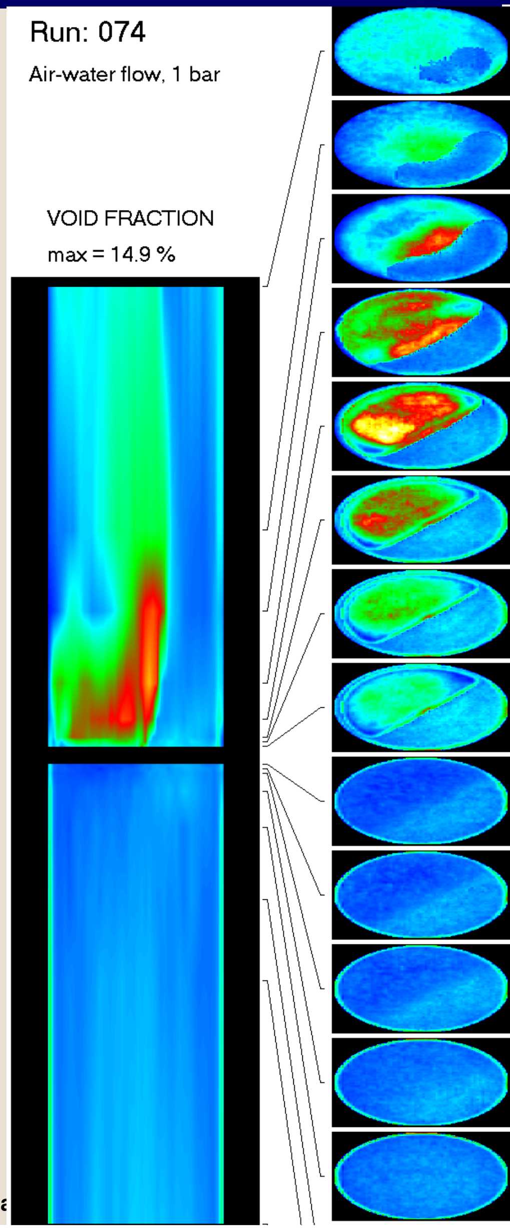


8) z=520mm

7) z=250mm

6) z=160mm

5) z=80mm



8)

7)

6)

5)

4)

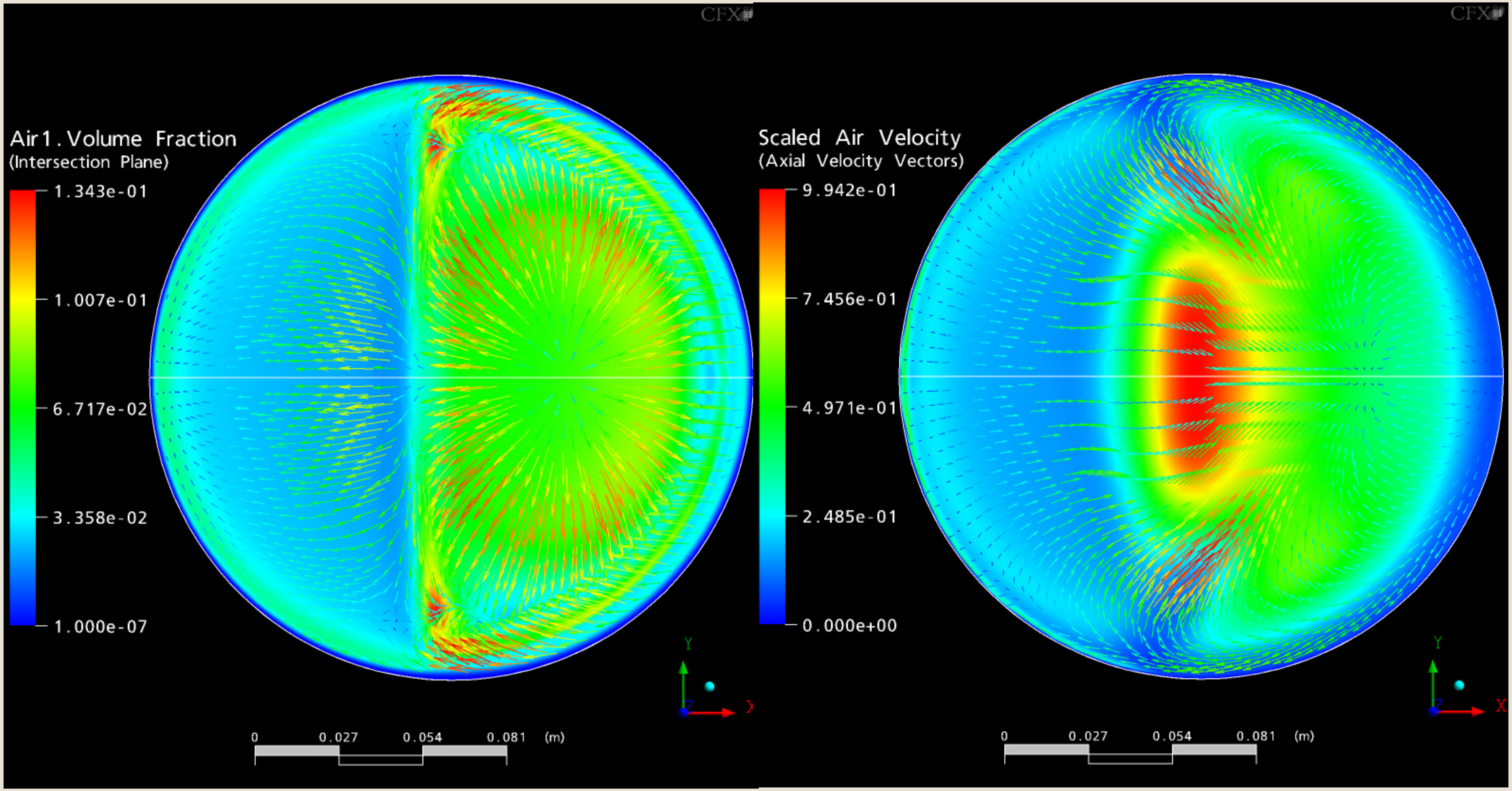
3)

2)

1)

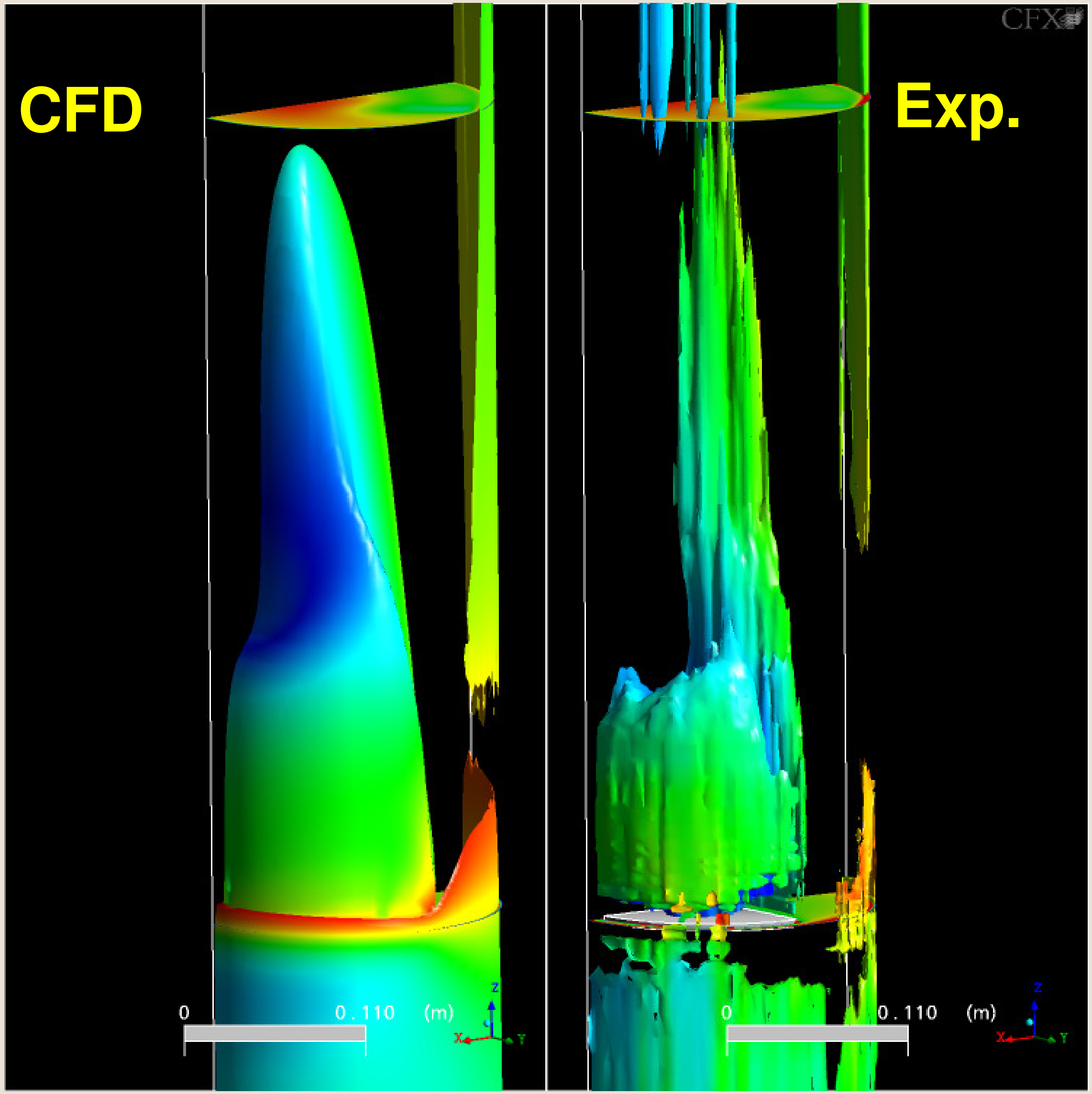
3d Bubbly Flow Around Obstacle
Air Void Fraction

3) z=20mm **6) z=160mm**



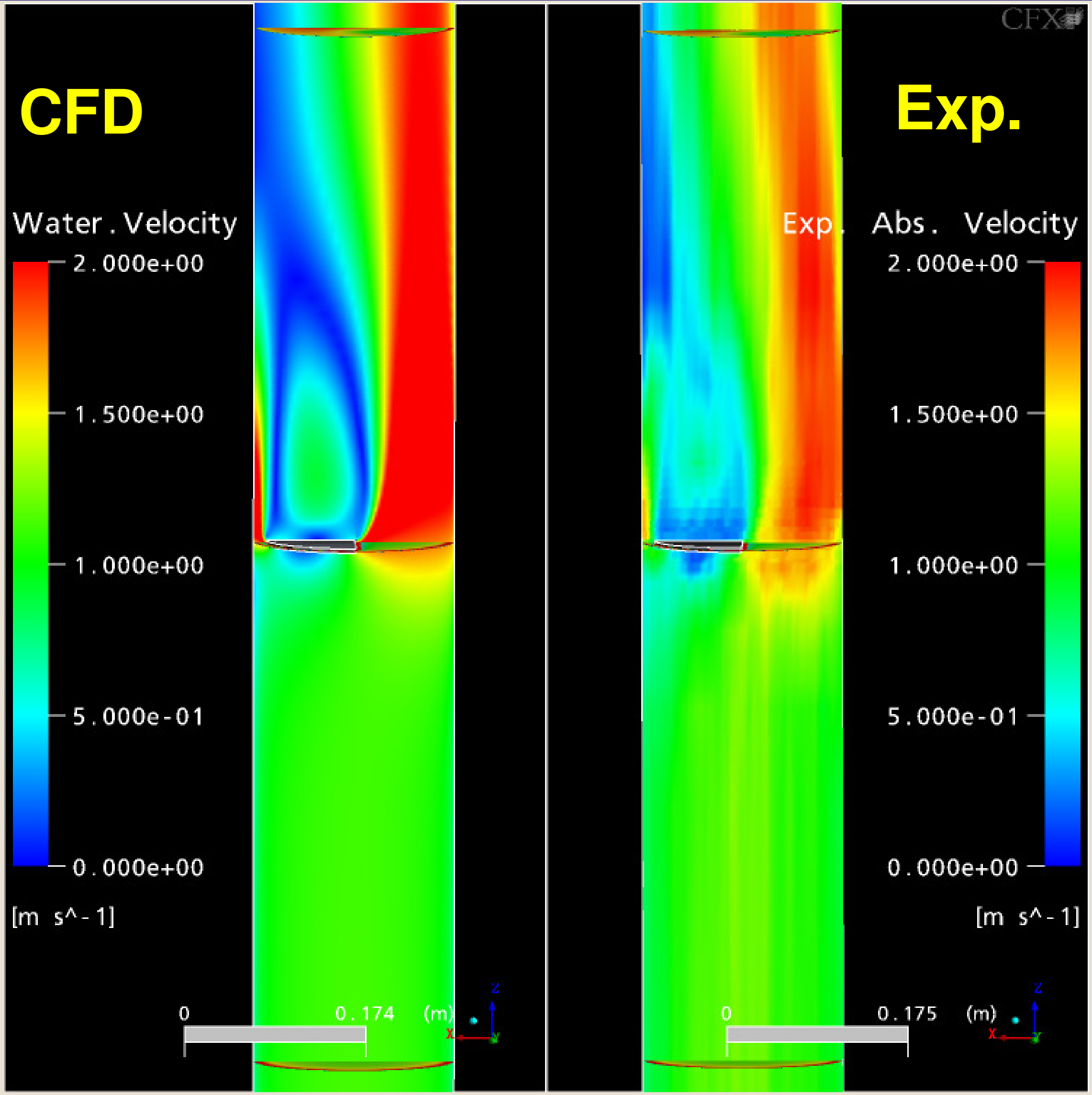
3d Bubbly Flow Around Obstacle Recirculation Zone Comparison

- Gas void fraction isosurface at 4%
- Colored by fluid velocity

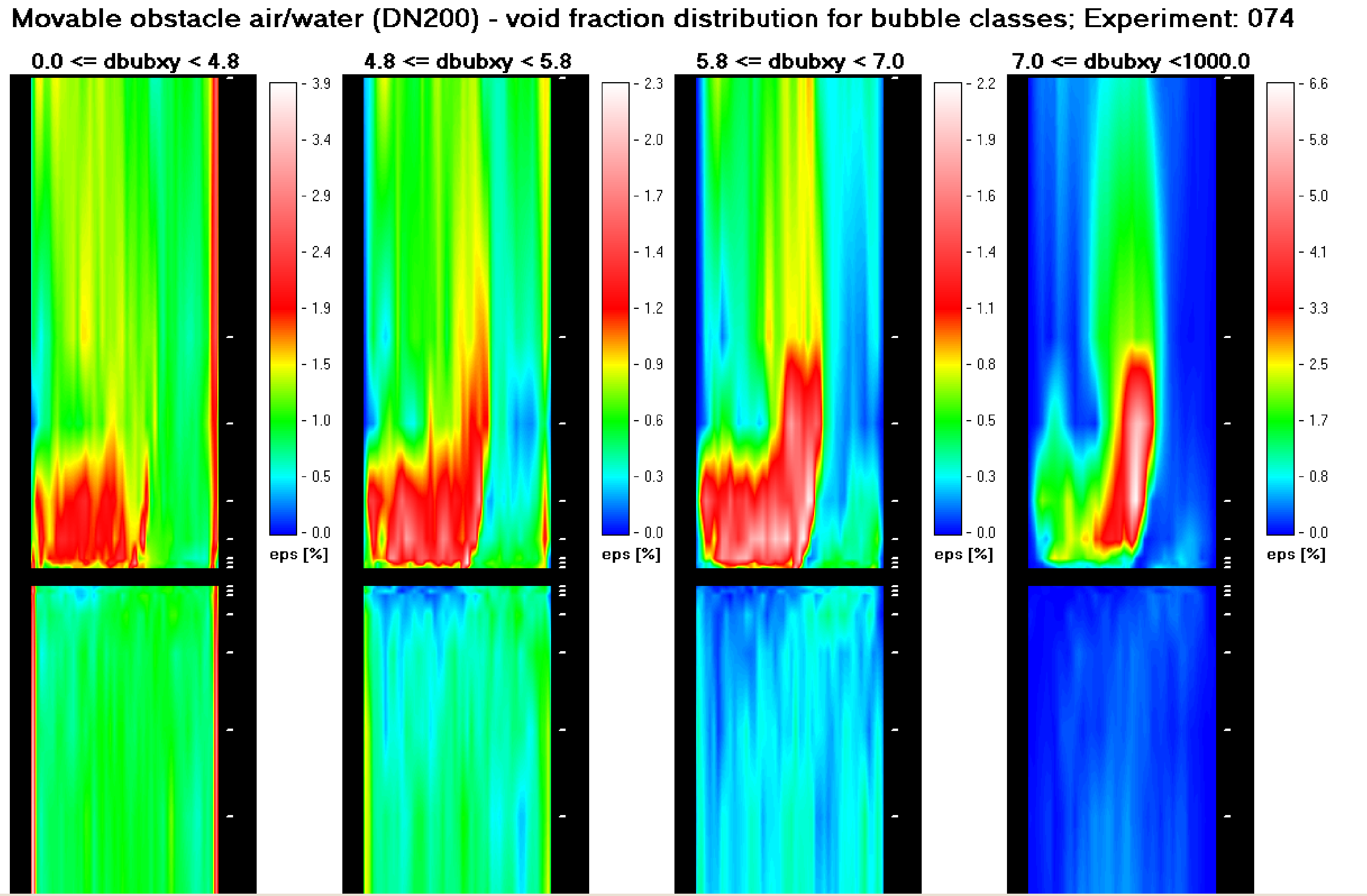


3d Bubbly Flow Around Obstacle
Water Velocity Comparison

- Comparison
  CFD ⇔ Experiment
- Absolute water velocity distribution in symmetry plane

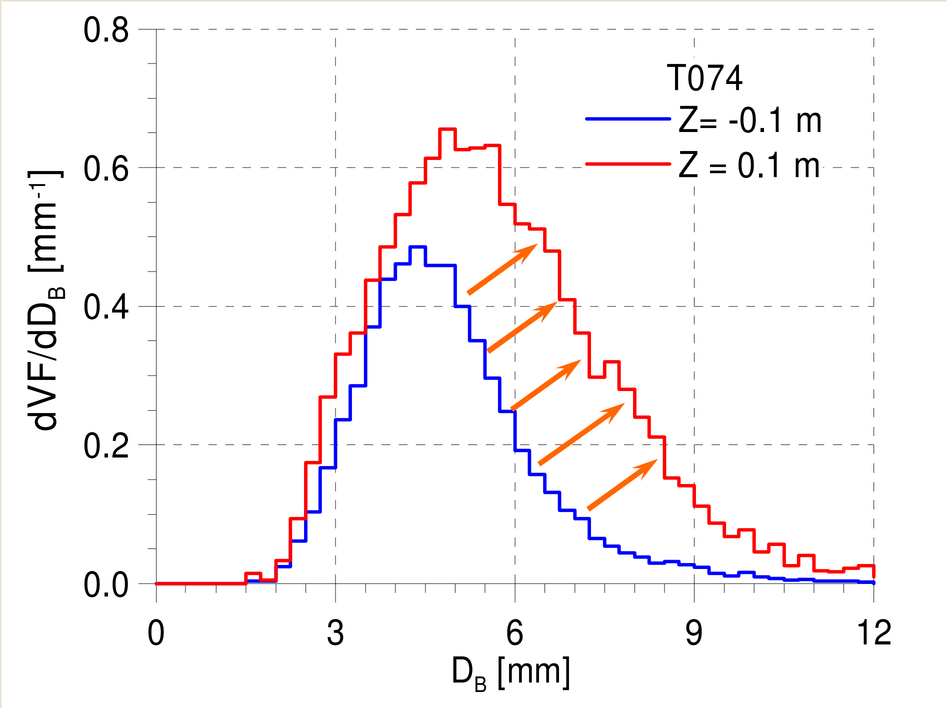


Measured Bubble Size Distribution for Test Case 074



Measured Bubble Size Distribution for Test Case 074

TOPFLOW movable orifice test case 074



- experiment shows formation of larger bubbles behind the obstacle
  → smaller accumulated air void fraction in recirculation zone

  → necessary to account for polydisperse bubbly flow !

Summary & Outlook

ANSYS

- CFD Model development in tight cooperation between FZ Rossendorf / ETHZ & ANSYS CFX
- Methodology:
  Experiment → Model Development → Validation → Application
- Results of CFX-10 pre-test calculations:
  - Geometry <u>independent</u> modeling
  - Good qualitative agreement
  - Quantitative deviations arise from assumption of monodisperse bubbly flow → recalculation with inhomog. MUSIG model
  - Models applicable to complex design & NRS studies
- Outlook:
  Further CFD model development towards flows with higher gas content, evaporation & condensation, bubble size distributions

TOPFLOW Technical Team

Many thanks to the **T**^{3} – the **T**OPFLOW **T**echnical **T**eam

- Head: H. Carl

- Team:
  M. Beyer,
  S. al Issa,
  K. Lindner,
  H. Pietruske,
  H. Rußig,
  P. Schütz,
  M. Tamme,
  Ch. Vallee,
  S. Weichelt

ANSYS

ETH
Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

CFX

Forschungszentrum
Rossendorf
Institute of Safety Research

ICEM CFD

Thank You!