Simulation of Complex Three-Dimensional Bubbly Flows with ANSYS CFX - Model Development & Validation -

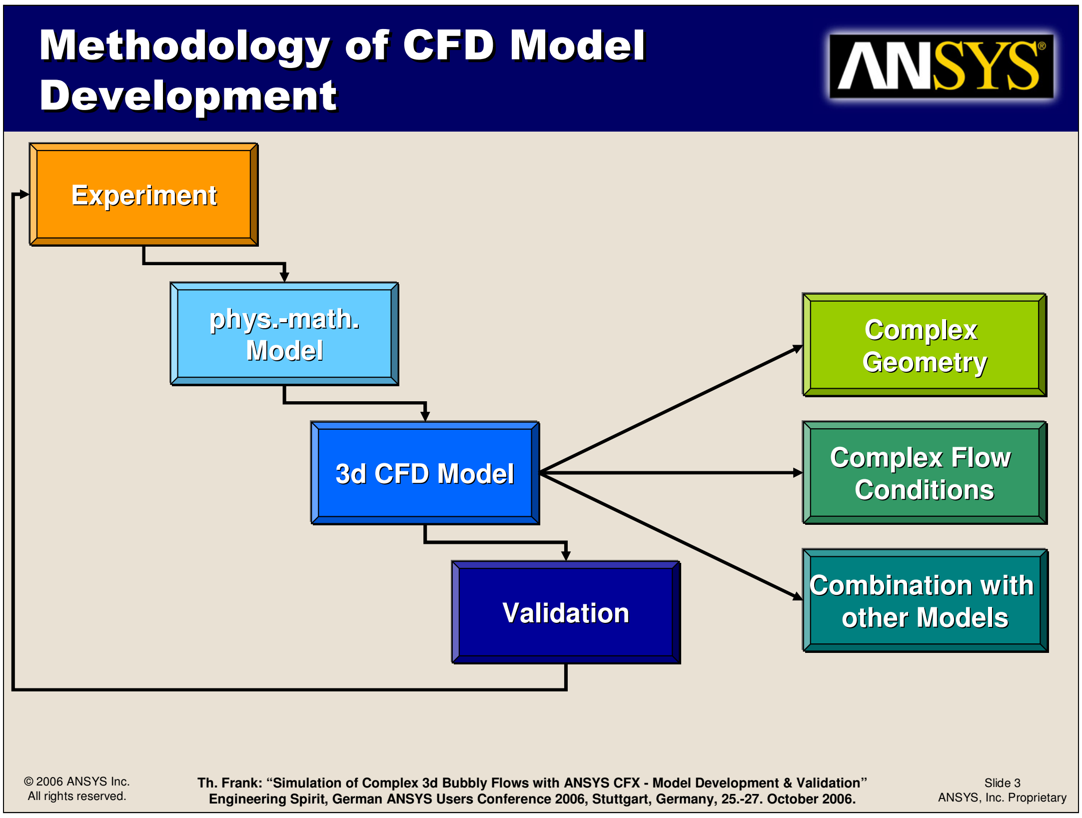
Thomas Frank
ANSYS Germany, Otterfing
Thomas.Frank@ansys.com

Overview

ANSYS

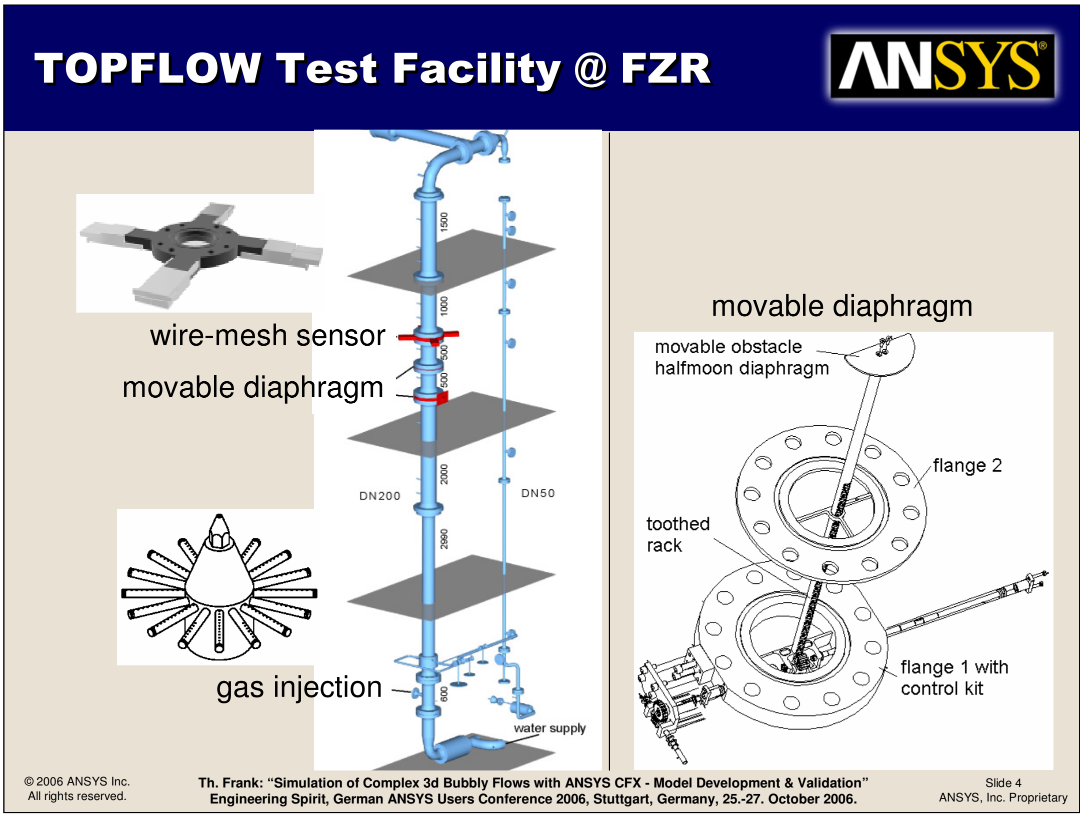
- Introduction
  - Methodology of CFD model development
- Experiments at TOPFLOW test facility @ FZR
- CFX blind test calculations
  - Mesh hierarchy
    → CFD Best Practice Guidelines
  - Setup of the flow physics
    → thermohydraulic CFD models
  - Results of the CFD simulation
- CFD model validation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

After a short introduction the given presentation outlines the ANSYS CFX model development and validation process. The development of new physical models for disperse multiphase flows, there implementation into the ANSYS CFX software and their validation against detailed experimental data will be demonstrated on the example of a complex 3-dimensional bubbly flow around an obstacle in a vertical pipe.



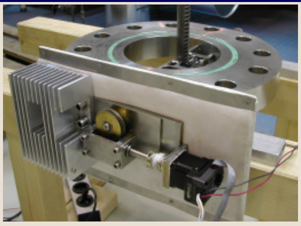
Development of new physical models is mostly based on special and very detailed experimental investigations, which are often carried out in quit simplified geometries. Usually the requirement for optical or mechanical access for sophisticated measurement techniques leads to the demand of geometry simplification. On the basis of these experimental investigations new physical-mathematical models are derived and implemented into CFD software. Also experiments are carried out in simplified geometries or even under quasi-1-dimensional conditions, the model development and implementation is done in a general 3-dimensional framework and special focus is given to model interoperability. Finally the model implementation is thoroughly validated against detailed experimental data.

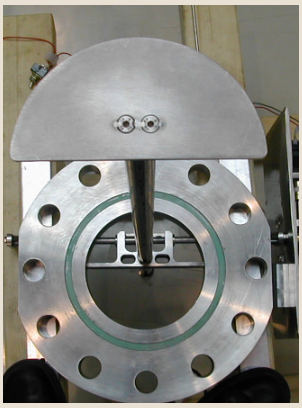
Afterwards the interoperability of different physical models in a general CFD software like ANSYS CFX allows for the application of the new physical models to either much more complex geometries, to complex flow conditions involving more complicated physical processes or even the combination with other physical models, like e.g. different kinds of turbulence models (k-ε, k-ω, RSM, EARSM, LES, DES, SAS), Lagrangian particle tracking, models for chemical reactions or radiation models.

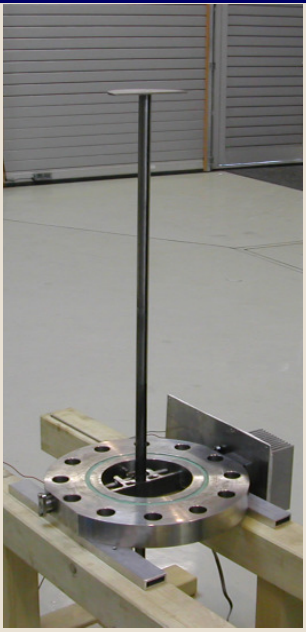


The present investigation on the simulation of complex 3-dimensional bubbly flows is based on experiments, which had been carried out on the TOPFLOW test facility at FZ Rossendorf, Germany. A movable diaphragm (or half-moon shaped orifice) has been installed into the DN 200 vertical pipe of TOPFLOW. The mechanic drive allows for a 500mm displacement of the obstacle in the vertical direction. The wire-mesh sensor technique developed by the FZR is used in order to measure volume fractions, air and water velocities in the air-water dispersed bubbly flow around the obstacle. The wire-mesh sensor is thereby mounted in a fixed location, while the obstacle can be moved up- and downwards in order to allow for different measurement distances between the obstacle and the measurement plane between -500mm and +500mm. A sparger system is used for generation of a monitored and almost constant bubble size distribution.

The Movable Orifice in TOPFLOW







Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

Slide 5

Pictures above show the details of the flanges, the mechanic drive, the support and the half-moon shaped orifice itself, as it was installed into the DN 200 vertical pipe of TOPFLOW. The support and drive of the obstacle allows for its movement over a 500mm distance in axial direction. For measurements downstream of the obstacle (+10mm < z < +500mm) the flanges with the drive and support had been mounted 500mm below the wire-mesh sensors in the vertical pipe of TOPFLOW. In a second measurement series the arrangement of the wire-mesh sensors and the obstacle has been reversed in order to allow for upstream measurements with -500mm < z <10mm.

3-dimensional Bubbly Flow Around Movable Obstacle

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 ($L \approx 7.5D$, D=198 mm)
- Air-water flow at 1 bar, 25 °C
- Test case conditions of test case TOPFLOW-074

Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

Slide 6

Blind 3-dimensional pre-test CFD simulations for the experimental flow conditions have been carried out with ANSYS CFX 10.0 prior to the availability of the experimental data. The reason was to demonstrate and prove the possible generalization of formerly developed and implemented multiphase flow models for disperse bubbly flows. These models where derived from almost 1-dimensional pipe flow experiments, had been generalized for a 3-dimensional model implementation and were implemented into the general CFD solver architecture of ANSYS CFX.

Now the generalized multiphase flow models are applied to a distinct 3-dimensional flow around an obstacle showing flow stagnation, recirculation with complex vortex flow patterns behind the obstacle and re-attachement of the flow to the pipe walls in some distance behind the obstacle. Due to the quit different density of the air and water phases, recirculation behind the obstacle leads furthermore to phase separation to a certain extent.

After the blind pre-calculation of the complex 3-dimensional multiphase flow around the obstacle, the simulation results will be compared to the experimental data.

TOPFLOW-074 Test Case Conditions from Test Matrix

- Selection of test case conditions:

superficial gas velocity (columns) / superficial water velocity (rows)

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**

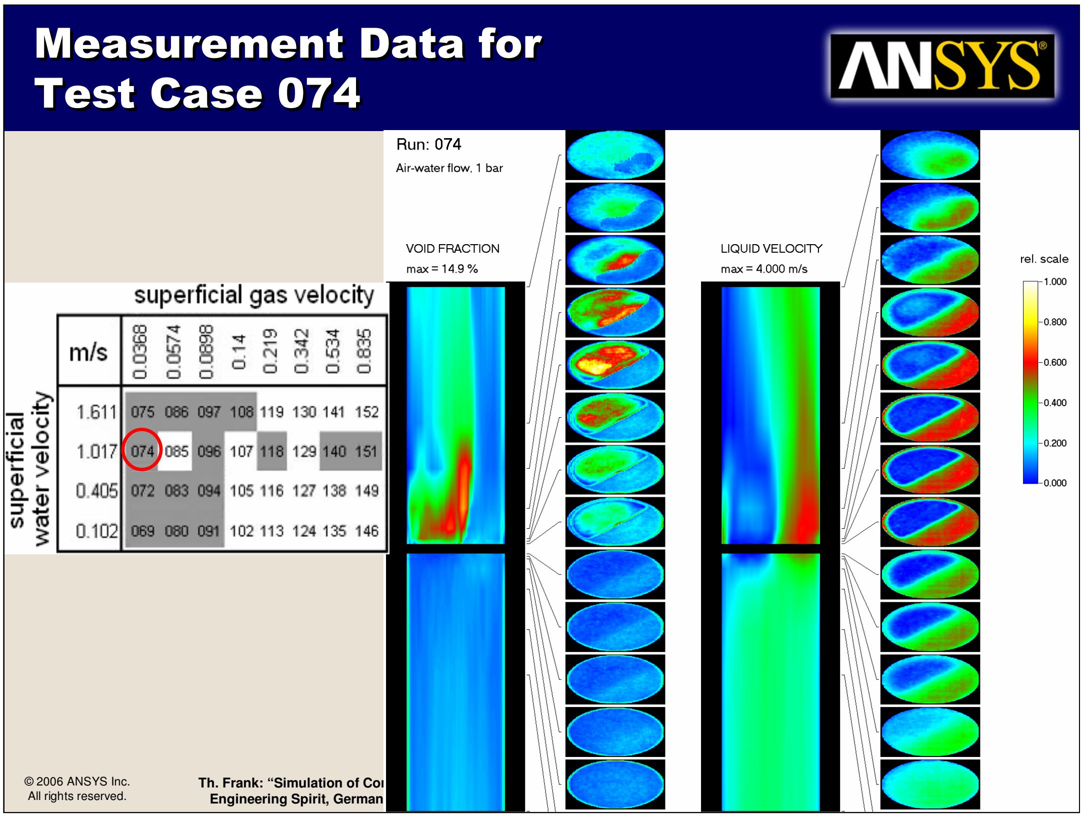
- **TOPFLOW-074 subject of validation in the past**
- **Superficial velocities:** J_G=0.0368 m/s, J_L=1.017 m/s
- **Wire-mesh sensor measurements at locations:**
  z=±10, 15, 20, 40, 80, 160, 250, 520mm

Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

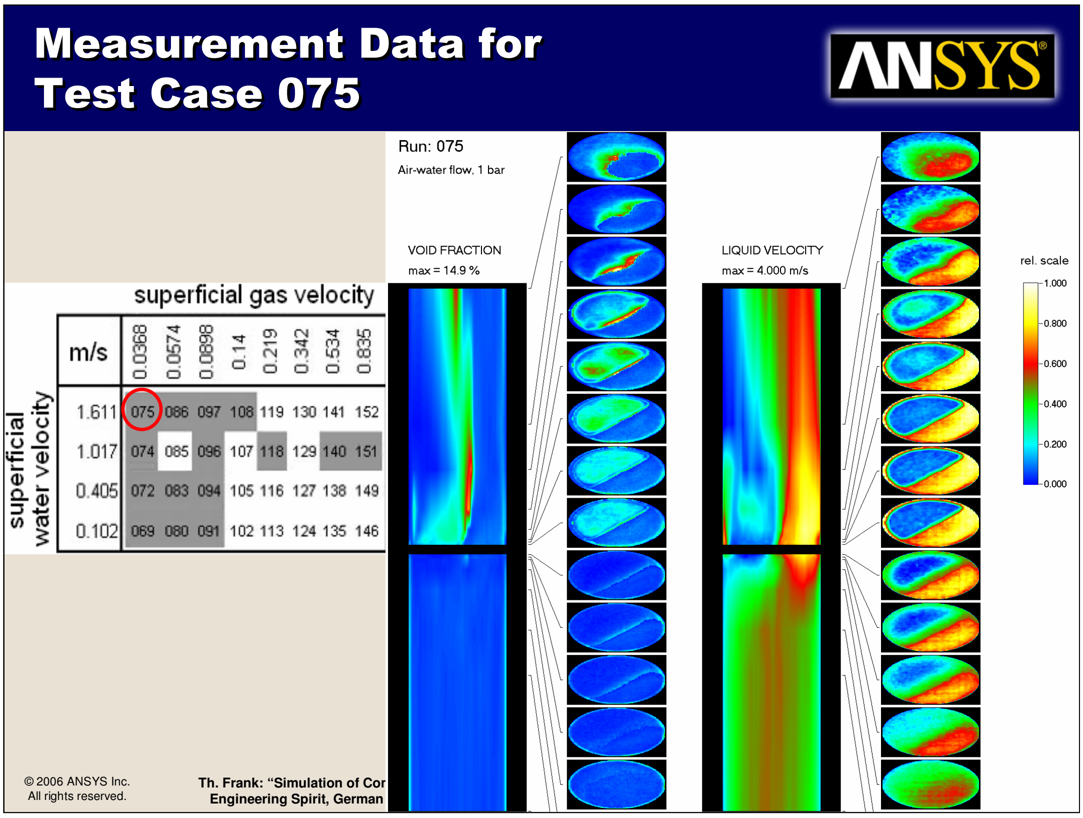
Slide 7
ANSYS, Inc. Proprietary

After the experimental campaign measurement data for the flow around the movable obstacle are now available not only for the initially selected flow conditions of TOPFLOW test case 074, but furthermore for a larger umber of different air and water superficial velocities. In a second series of measurements the flow of multiphase flow mixture of saturated vapor bubbles at 65 bar and 280^{o} C with water has been investigated.

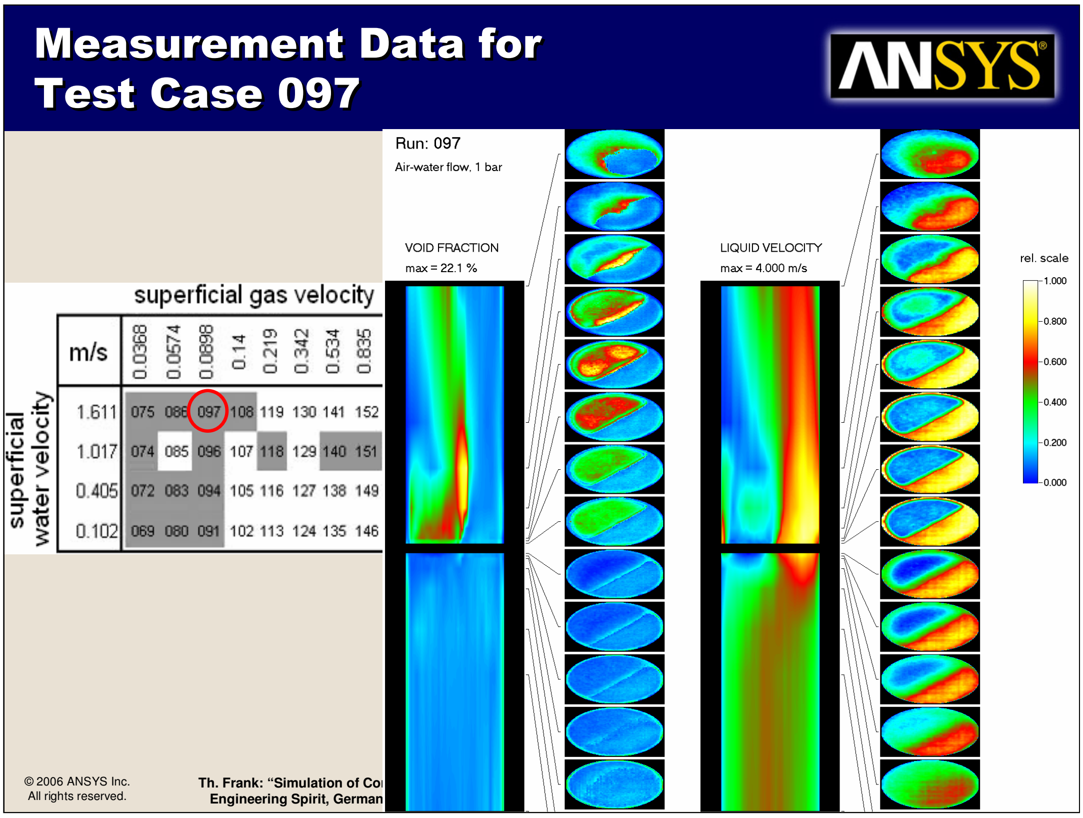
The experiments provide very detailed measurement data at 16 different cross sections of the vertical pipe up- and downstream of the obstacle. Together with the very dense cross-sectional spatial resolution of the 64x64 wire-mesh sensors this results in a 3-dimensional experimental data set for volume fractions, bubble size distributions and air/water velocities, which is excellently suited for validation of 3-dimensional CFD simulations.



The picture above shows the air void fraction and water velocity distribution for the test case conditions of TOPFLOW 074. The large image on the left shows the variable distribution in the symmetry plane of the geometry from z=-500mm to z=+500mm. In the column of smaller images on the right you see the variable distribution in the individual measurement cross sections. Lower images show flow conditions upstream the obstacle while upper images show the downstream flow conditions. The latter show clearly the entrainment of air bubbles in the wake behind the obstacle in connection with the small water velocities in the recirculation area.



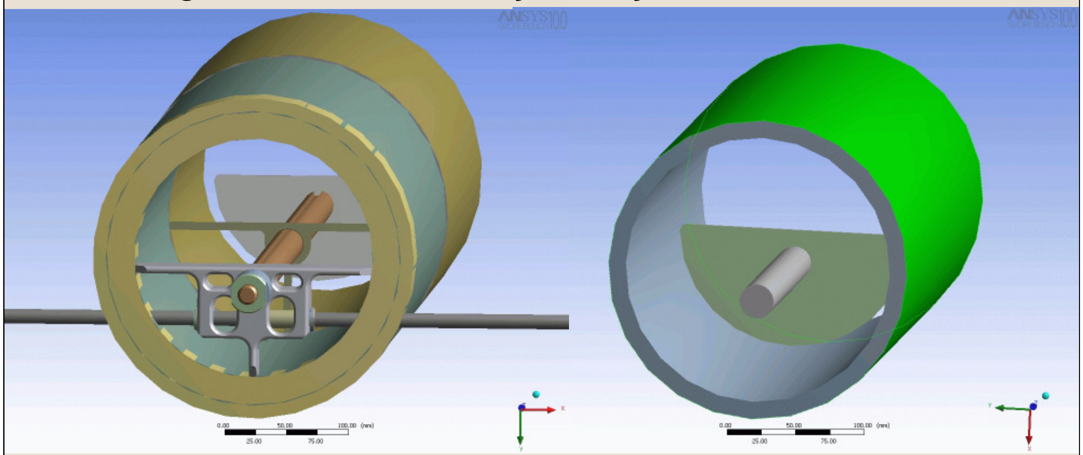
The same for test case conditions of TOPFLOW 075 at higher water velocities. The re-attachement length and the length scale of the recirculation area behind the obstacle is decreased. At the same time the amount of entrained air volume fraction in the vortex core is reduced. The location of the void fraction maximum is shifted almost to the edge of the obstacle.



If for test case 097 the superficial air velocity is increased, the amount of recirculating air volume fraction increases again. Pictures of variable distribution in the cross sections show a sharp separation between regions of high air volume fraction in the vortex system behind the obstacle and low air volume fraction in the free stream besides the obstacle. Water flow is strongly accelerated by the cross sectional obstruction, leading to strong water velocity gradients and strong lift forces exerted on air bubbles.

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



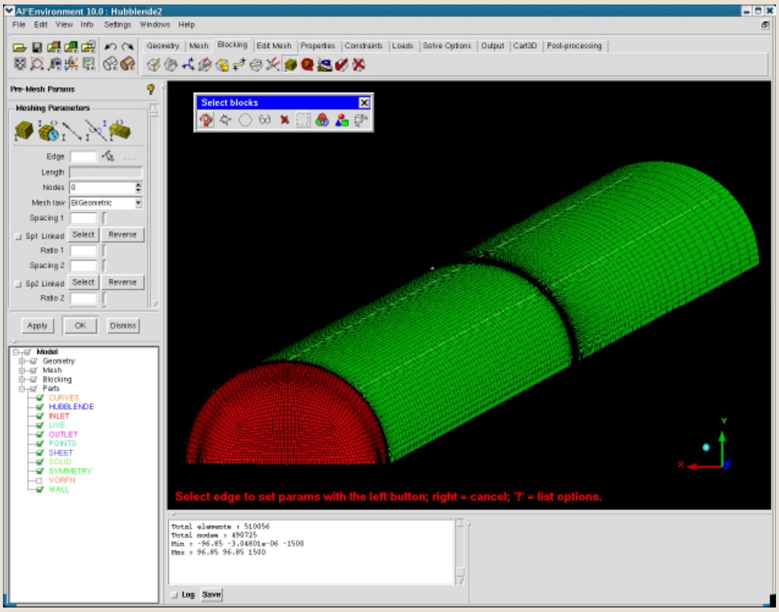
Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

Slide 11

For the blind ANSYS CFX pre-calculations the geometry of the movable orifice with mechanical drive and support structure has been supplied by FZ Rossendorf in ACIS SAT CAD file format, which could be directly imported into ANSYS Workbench. In order to save computational time, the drive and support of the obstacle has been neglected for the numerical simulation and axial 180^{o} symmetry has been assumed.

Geometry & Mesh Generation

- **Mesh generated with ICEM- CFD 10.0**
- **Hexahedral mesh**



Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

ANSYS ICEM-CFD 10.0 has been used to generate a hierarchy of hexahedral meshes of increasing resolution for the intended simulation. The picture shows the mesh refinement in the vicinity of outer pipe walls and in the vicinity of the obstacle surface.

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**

The mesh hierarchy consists of 3 numerical meshes, where the ratio of the total number of mesh elements between them is approx. 4. The Grid 2 has proven to be sufficient to cover most of the large scale turbulent vortex structures behind the obstacle, which could afterwards be observed in the experimental data.

Flow Physics Setup

ANSYS

- **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

Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

Slide 14
ANSYS, Inc. Proprietary

The flow simulation with ANSYS CFX 10.0 has been setup as a monodisperse bubbly flow with an assumed characteristic bubble diameter, as the sparger system was designed to generate an almost constant bubble diameter (or at least a very narrow bubble size distribution). Also it turned out, that the produced characteristic bubble size from the sparger system was slightly dependent on the air and water volume flow rates. The increase of bubble size with vertical coordinate (z) takes into account the slight expansion of bubbles due to the decreasing hydrostatic pressure in the test facility.

For the monodisperse air-water two-phase flow the Grace drag correlation, the FAD turbulent dispersion force, the Tomiyama lift force and Frank's generalized wall lubrication force has been taken into account for the disperse bubbly phase. Additionally the bubble enhanced turbulence has been accounted for by the Sato model, while the fluid phase turbulence was predicted by the standard SST-model using automatic wall functions.

Flow Setup & Boundary Conditions

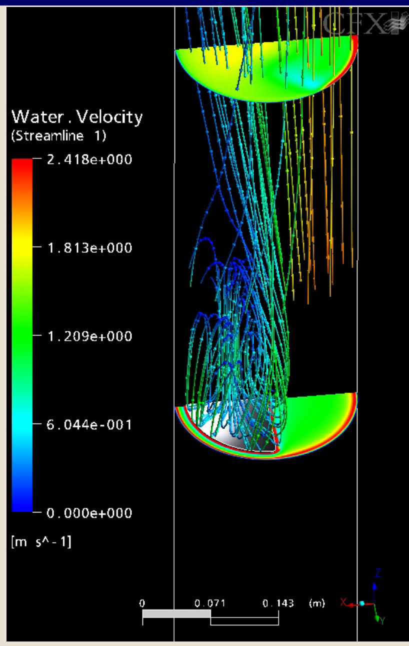
- **Numerical schemes:**
  - steady-state simulation
  - High resolution in space
- **Convergence criteria:**
  - 10^{-3} MAX residuals
- **Physical time scale:**
  - 0.0005 s
- **Initialization:**
  - u, v, w, r_G, r_L, k, ω from fully developed pipe flow
- **Boundary conditions:**
  - Inlet: same as for initialization; fully developed pipe flow profiles
  - Outlet: Average Static Pressure
  - Walls: no slip wall for cont. phase
free slip wall for disp. phase

Blind pre-calculations have been carried out as steady-state simulations using the high resolution advection scheme. Details of the applied convergence criteria, physical time scale and boundary conditions are given on the slide.

3d Bubbly Flow Around Obstacle

ANSYS

- **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



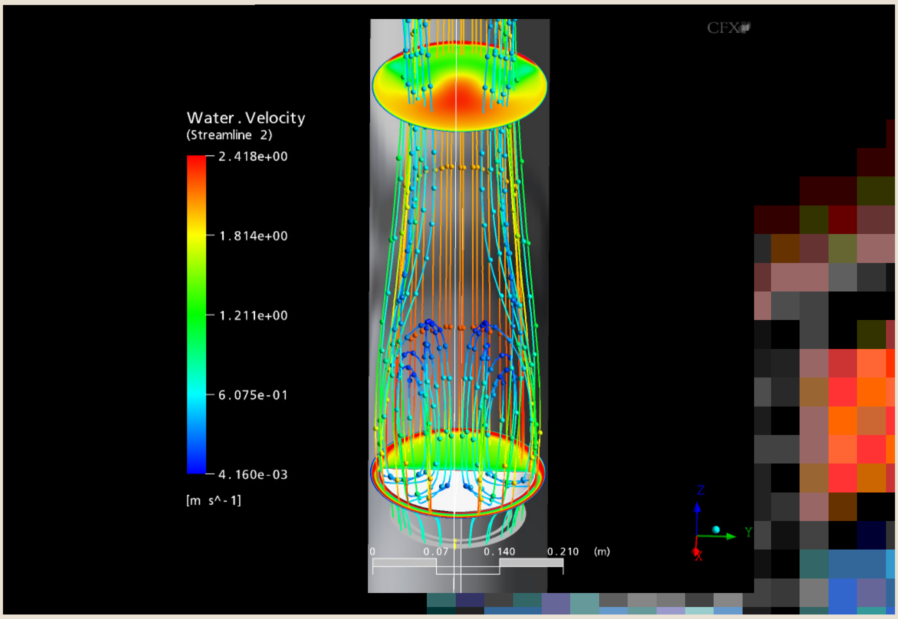
Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

Slide 16

The movie shows an ensemble of streamlines in order to visualize the developing large scale vortex structure behind the obstacle for test case conditions TOPFLOW 074. Due to the small 4mm gap between the pipe wall and the edge of the obstacle a secondary motion along the round edge of the obstacle is observed, leading to the shift of recirculating and entrained bubbles from the left to the straight edge of the obstacle, where they are captured by the strongly accelerated fluid flow and removed from the recirculating vortex core. Higher residence time inside of the vortex core leads to an accumulation of air volume fraction inside the vortex behind the obstacle.

3d Bubbly Flow Around Obstacle

ANSYS



Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

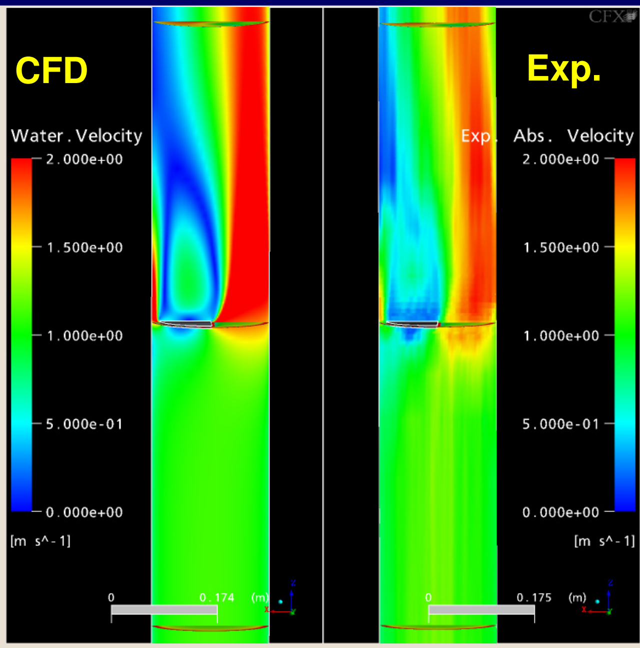
Slide 17

Export of streamline visualization in the CFX Viewer format allows the import of 3-dimensional visualizations into Powerpoint presentations and HTML web sites and provide a deeper inside into the 3-dimensional structure of the vortex system behind the obstacle.

3d Bubbly Flow Around Obstacle
Water Velocity Comparison

ANSYS

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

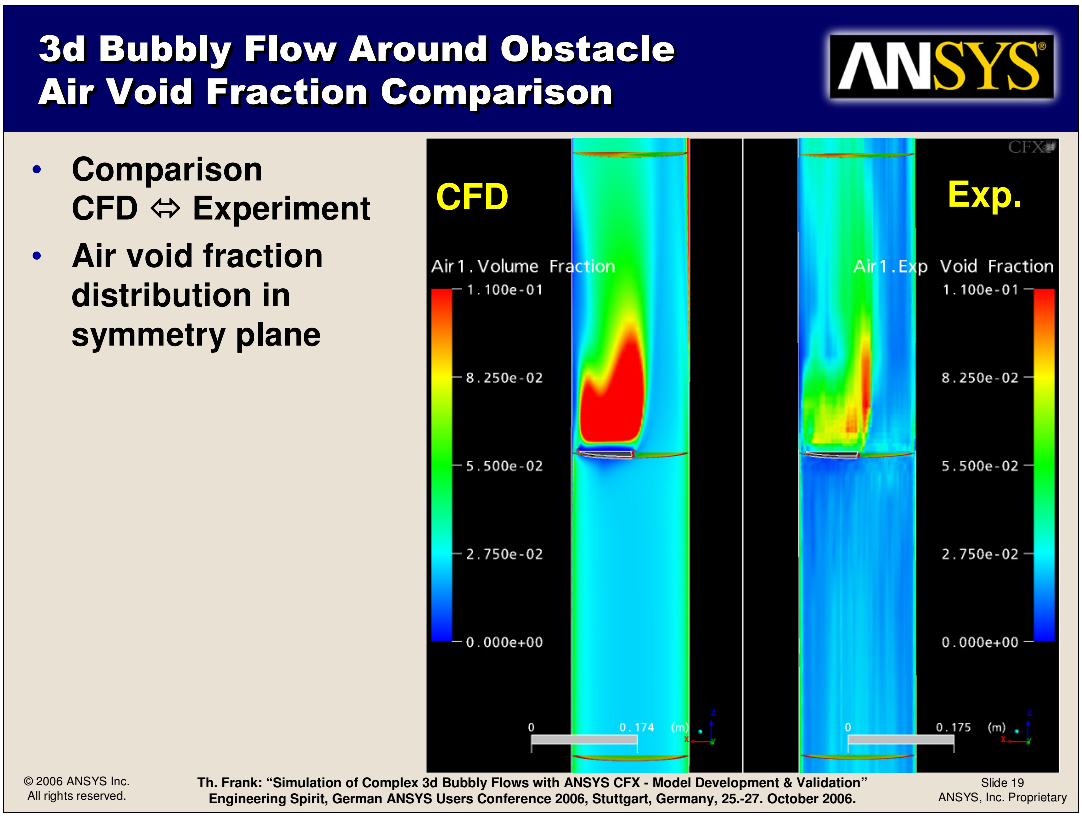


Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

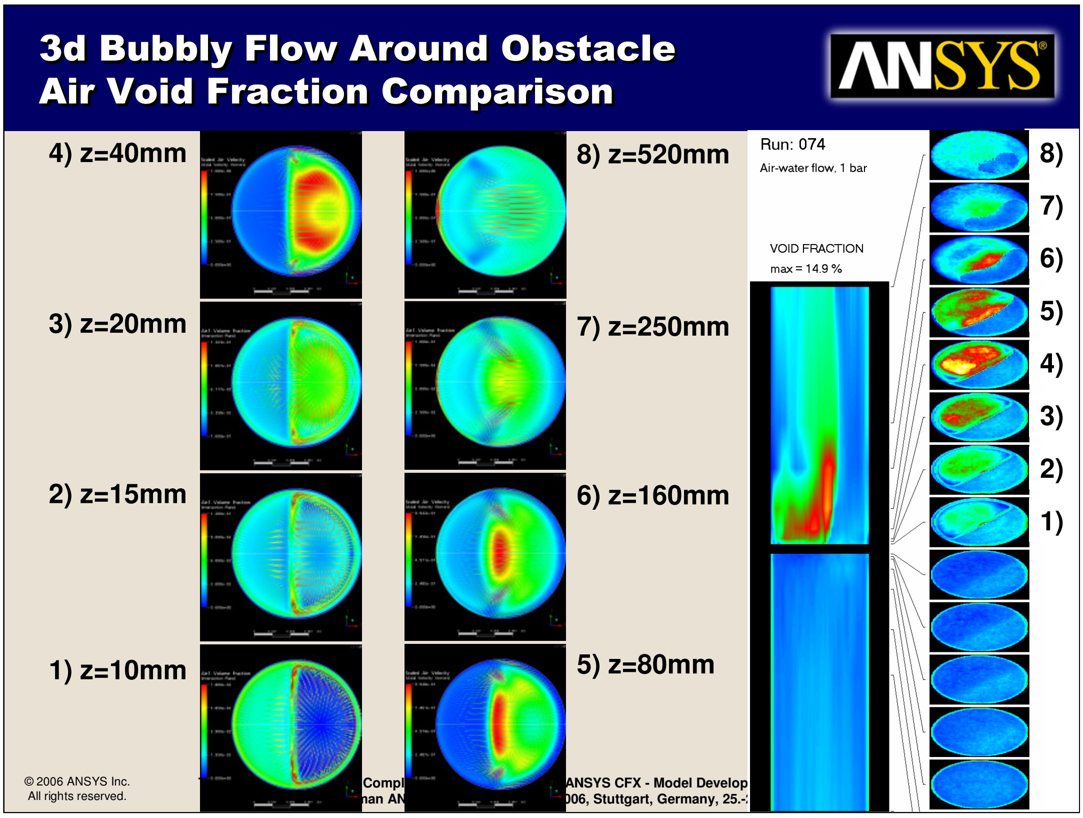
Slide 18

After the experiments had been carried out at FZ Rossendorf, the 3-dimensional data set has been provided for a detailed comparison with the ANSYS CFX simulation. A special data conversion and interpolation program has been written for the wire-mesh sensor data by Prasser & Al Isssa at FZ Rossendorf. This turned out to be necessary (or at least beneficial), since the spatial resolution of the sensor data in z-direction on the one hand side and x-/y-direction otherwise was quit different without pre-processing. Therefore the experimental data were interpolated with Δz=0.01m between the 16 available measurement cross sections as given on slide 7. Afterwards the experimental data were imported as values of additional variables into CFX-Post, which allows to use the same set of flow visualization tools and equal color range for variable value visualizations as in a standard CFD post-processing. In the result the results of the CFX simulation and experimental data can be directly compared in all available detail. As a unique feature streamlines and variable value isosurfaces can be applied to the analysis of the experimental data as well.

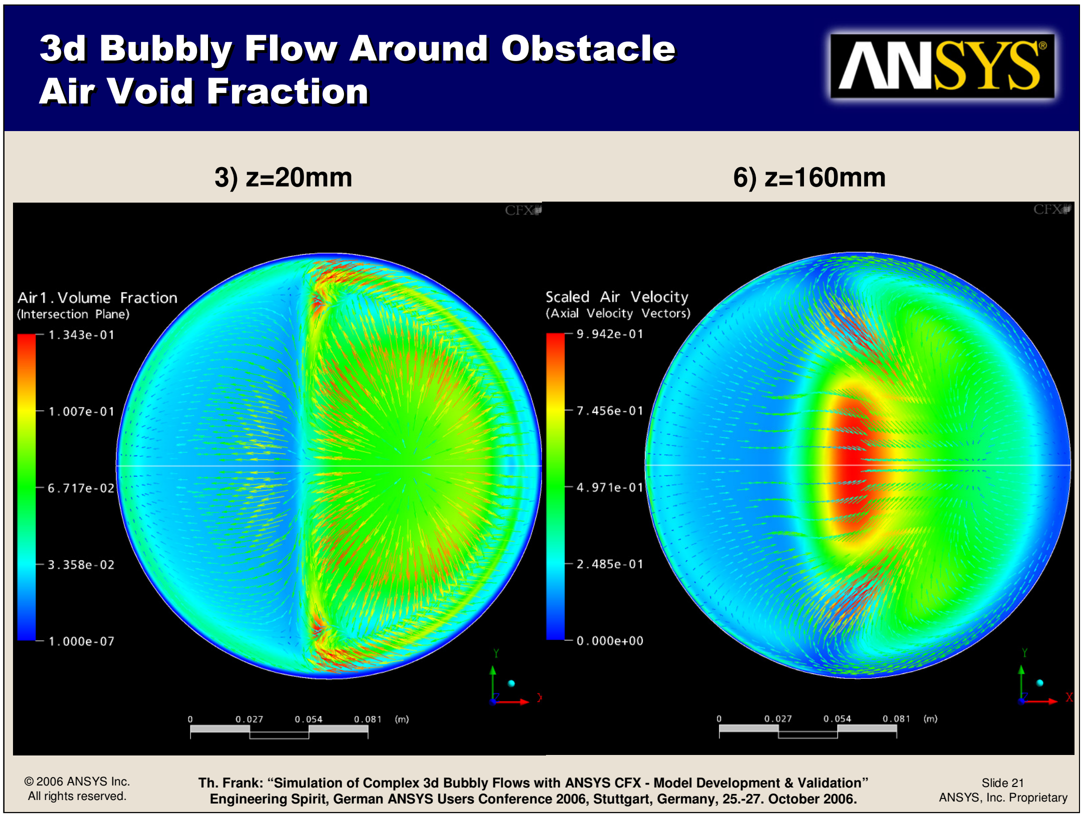
The above picture show the comparison for the absolute water velocity with the CFD result on the left and experimental data from TOPFLOW experiment on the right. Remarkable agreement in the flow structure can be observed. Furthermore the length scale of the recirculation area behind the obstacle, the location of the vortex core (to be seen by the green color directly downstream the obstacle) and the re-attachement length of the downstream flow are in good agreement.



The same method of data comparison has been applied in the above picture to the air volume fraction. Both data sets show the entrainment and accumulation of air volume fraction behind the obstacle in the core of the vortex system, also the experimental data show a slightly lower level of air bubble accumulation. This is explained by bubble coalescence, which takes place in the experiments in the region of high air volume fraction and which was not taken into account by the numerical simulation. The formed larger bubbles are able to escape from the recirculation zone leading to less pronounced maximum void fraction in this area.



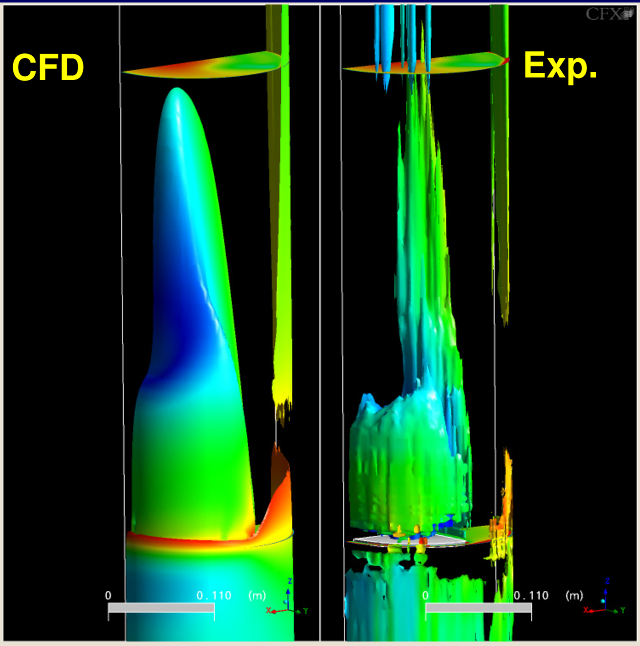
The comparison of the cross-sectional distribution of air void fraction downstream of the obstacle is of special interest, since we can observe strong separation effects and secondary motion in the experimental data. Almost identical patterns can be observed in the CFD simulation, which is mainly addressed to the action of the lateral lift force acting on the bubbles in region of high water velocity gradients. This can be remarkably observed in cross sections 4-6 in the above images, where regions of high air volume fractions can be observed in identical locations and with identical intensity in both the numerical and experimental data set.



For a better view of the driving secondary fluid motion the cross-sectional images from plane 3 and 6 (CFD data) are reproduced in enlargement. Especially on plane 6 the region of high air volume fraction directly corresponds to a strong secondary fluid motion, which is directed inwards towards the center of the vortex system above and behind the obstacle. In contrary on plane 3, which is at z=+20mm downstream of the obstacle, the stagnation point flow on the obstacle surface leads to an almost homogeneous value of the air volume fraction in the right half of the measurement plane 20mm on top of the obstacle surface.

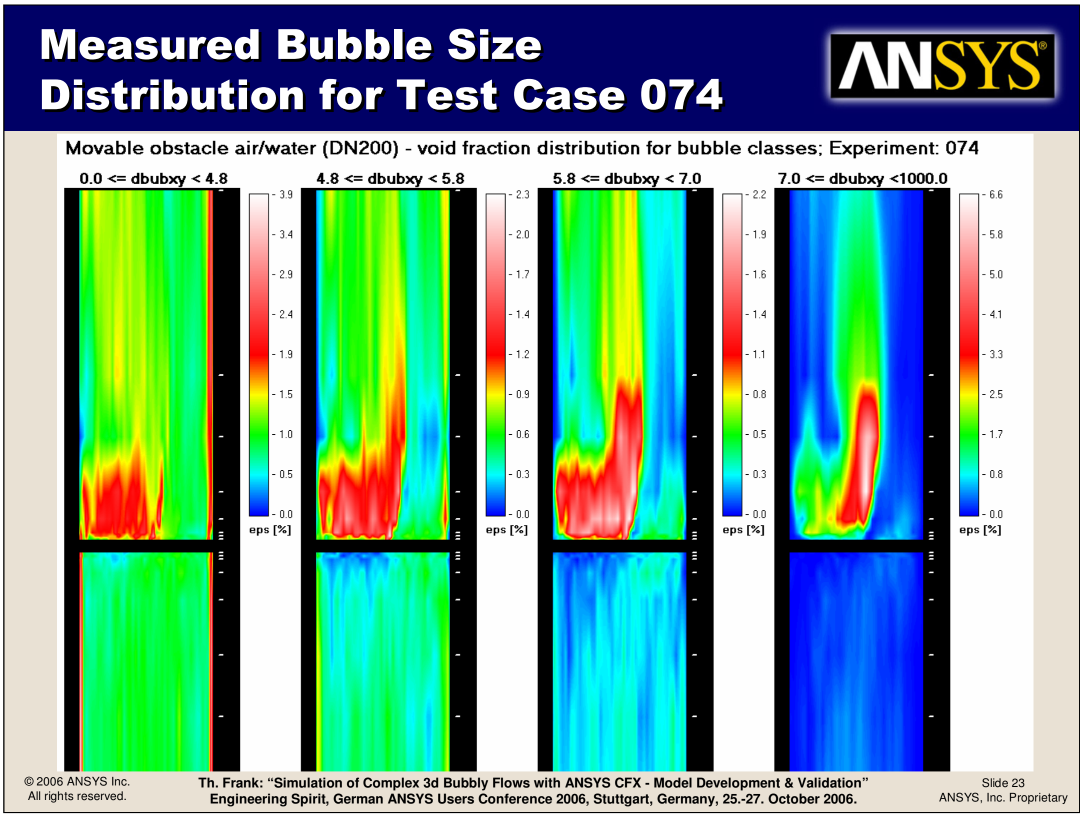
3d Bubbly Flow Around Obstacle
Recirculation Zone Comparison

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



Th. Frank: "Simulation of Complex 3d Bubbly Flows with ANSYS CFX - Model Development & Validation"
Engineering Spirit, German ANSYS Users Conference 2006, Stuttgart, Germany, 25.-27. October 2006.

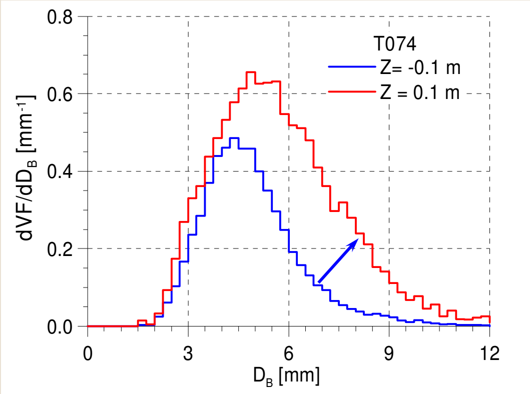
Data import of measurement data into CFX-Post allows for unique post-processing capabilities for experimental data like the plot of an isosurface at air volume fraction of 4%. Due to the larger distance between measurement planes for z>+80mm the representation of the air volume fraction isosurface based on experimental data is not as smooth as for the CFD simulation results. But nevertheless a reasonable good agreement can be observed in terms of its outer shape, its axial and radial length scale. In both figures the isosurface is colored by the local water velocity, which is in good agreement too.



Usage of the wire-mesh sensor for simultaneous bubble size and volume fraction measurements allows the plot of decomposed volume fraction distributions in dependence on the bubble diameters in defined bubble size classes. The above picture shows the air volume fraction distribution for the devision of the overall bubble size spectrum into 4 bubble size classes. In the result we can observe the strong formation of bubbles larger then d_P=5.8mm in the wake region behind the obstacle, where bubble entrainment and accumulation takes place and leads to a general high air volume fraction and strong bubble coalescence. From these data it can be explained, why the air volume fraction in the wake does not reach as high values as for the numerical simulation result, since the larger bubbles are affected by stronger buoyancy forces and additionally by the countercurrently directed lift force (in correspondence with the sign change in the Tomiyama lift force coefficient correlation for bubbles larger then 5.8mm in an air-water two-phase flow system at normal conditions). Due to this two physical mechanisms larger bubbles are able to escape from the entrainment into the recirculating vortex motion and therefore the air volume fraction in theis region is reduced under the experimental conditions.

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 !

Formation of large bubbles in the wake of the obstacle can be observed from the comparison of measured bubble size distributions in front and behind the obstacle, showing a shift of the bubble size spectrum towards larger bubbles. From these facts it seems to be necessary to drop the monodisperse bubbly flow assumption for the CFD simulation and to repeat the numerical simulation with the inhomogeneous MUSIG (Multiple Size Group) model by taking into account bubble breakup and coalescence.

Summary & Outlook

- CFD Model development in tight cooperation between FZ Rossendorf / ETHZ & ANSYS CFX
- Methodology:
  Experiment → Model Development → Validation
- Results of CFX-10 pre-test calculations:
  - Geometry independent 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
- Acknowledgement
  This research by ANSYS Germany and FZ Rossendorf has been funded by German BMWi in the German CFD Network on Nuclear Reactor Safety under contract numbers 150 1271 and 150 1265.

The above slide summarizes the main results of the presented detailed investigation on complex 3-dimensional bubbly flow over a half-moon shaped obstacle in a vertical pipe. The presentation has shown the model development and validation process for new physical-mathematical models in ANSYS CFX. Furthermore a very detailed comparison of a pre-test CFX-10.0 calculation with afterwards obtained experimental data have been presented. The model validation demonstrates the applicability and accuracy of multiphase flow models for complex designs, even if the physical models have been derived under simplified experimental conditions. Good agreement between ANSYS CFX simulation results and experimental data could be obtained. Remaining quantitative deviations could be addressed to the neglected bubble breakup and coalescence processes in regions of higher air volume fraction in the wake of the obstacle.

Future CFD simulations will be carried out using the inhomogeneous MUSIG model of ANSYS CFX by taking into account bubble breakup and coalescence. Further CFD model development in the German CFD Network for Nuclear Reactor Safety will focus on multiphase flows with higher gas content and mass transfer (i.e. vapor-water flows).

TOPFLOW Technical Team

ANSYS

Many thanks to the T³ - TOPFLOW Technical Team

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

Many thanks are addressed to the TOPFLOW Technical Team under the management of H. Carl at FZ Rossendorf, who had carried out the extensive measurement campaign at the TOPFLOW test facility and provided the experimental validation data. Further thanks are expressed to Prof. H.-M. Prasser and the team of the FZR, Institute of Safety Research for the intensive and fruitful research cooperation.

Thank You!