Validation of Multiphase Flow Modeling in ANSYS CFD

Th. Frank, C. Lifante, A.D. Burns
Head Funded CFD Development
ANSYS Germany

Thomas.Frank@ansys.com
Outline

• Introduction
• MPF model validation for adiabatic air-water flows
• Polydisperse MPF model validation – MUSIG model
• Bartolomej testcase (PWR)
• Lee testcase (BWR)
• Wall boiling with conjugate heat transfer (CHT)
• Summary & Outlook

Courtesy by E. Krepper (FZD)
ANSYS as Part of the German CFD Network in Nuclear Reactor Safety

- FZ Dresden-Rossendorf
- ANSYS Germany
- FZ Karlsruhe
- Becker Technologies
  ThAI test facility
- AREVA
- TU München:
  TD, Nucl. Energy
- Zittau-Görlitz: IPM
- Univ. Applied Sciences
- Univ. Stuttgart:
  Nuclear Energy
- International Guest Visitors

© 2009 ANSYS, Inc. All rights reserved.
Methodology of CFD Model Development & Validation

- Experiment
- phys.-math. Model
- 3d CFD Model
- Validation
- Complex Geometry
- Complex Flow Conditions
- Combination with other Models
Eulerian MPF Modeling - The Particle Model

Mass weighted averaged conservation equations

- Mass, momentum, energy transport equations for each phase

\[
\frac{\partial}{\partial t} \left( \rho_k r_k \right) + \nabla \left( \rho_k r_k U_k \right) = \sum_{l=1}^{N} \Gamma_{kl} \\
\frac{\partial}{\partial t} \left( \rho_k r_k U_k \right) + \nabla \cdot \left( \rho_k r_k U_k U_k \right) = -r_k \nabla P - \nabla \cdot \left( r_k \Pi^k \right) + F_k + I_k
\]

\[
I_k = F_{\Gamma} + F_D + F_L + F_{WL} + F_{TD} + F_{VM}
\]

- turbulence models for each phase (e.g. k-\( \varepsilon \) / k-\( \omega \) SST model, 0-eq. disp. phase turb. Model)
- heat transfer equations for each phase with interfacial transfer closure
- interfacial forces need empirical closure
- high void fraction effects, bubble induced turbulence, etc.
Lift force, Wall lubrication force & turbulent dispersion

**Lift force:**
- due to asymmetric wake and deformed asymmetric particle shape
  - Tomiyama $C_L$ correlation
  \[
  \mathbf{F}_L = C_L r_G \rho_L (\mathbf{U}_L - \mathbf{U}_G) \times \nabla \times \mathbf{U}_L
  \]
  \[
  C_L = C_L (Re_P, Re_V, Eo)
  \]

**Wall lubrication force:**
- surface tension prevents bubbles from approaching solid walls
  - Antal, Tomiyama & Frank W.L.F. models
  \[
  \mathbf{F}_{WL} = -C_{wall} r_G \rho_L \left| \mathbf{U}_{rel} - (\mathbf{U}_{rel} \cdot \mathbf{n}_W) \mathbf{n}_W \right|^2 \mathbf{n}_W
  \]
  \[
  C_{wall} = C_W (Eo, y/d_p)
  \]

**Turbulent dispersion force:**
- turbulent dispersion = action of turb. eddies via interphase drag
  \[
  \mathbf{F}_{TD} = \frac{3}{4} \rho_F \bar{C}_D \frac{v_{lF}}{d_p \sigma_{lF}} (U_F - U_F) r_p \left( \frac{\nabla r_P}{r_P} - \frac{\nabla r_F}{r_F} \right)
  \]
  - FAD model by Burns et al. (ICMF’04)

© 2009 ANSYS, Inc. All rights reserved.
Bubbly Flow Model Validation
FZR MT-Loop and TOPFLOW Database

MT-Loop
CFX Model Validation
MT-Loop & TOPFLOW Test Matrix

- M01 experimental test series on MT-Loop
- evaluation based on air volume fraction profiles at L/D=59.2 (z=3.03m) from the sparger system
- numerically investigated test case conditions

- finely disperse bubbly flow
- bubbly flow with near wall void fraction maximum
- bubbly flow in the transition regime
- bubbly flow with void fraction maximum at pipe center
- bubbly flow with void fraction maximum at pipe center, bimodal
- slug flow
Validation: Bubbly Flows
Turbulent Dispersion Force
Monodispersed Bubbly Flow MT-Loop Test Case FZR-019

FZD-019:

\[ J_L = 1.017 \text{ m/s} \]
\[ J_G = 0.004 \text{ m/s} \]
\[ d_p = 4.8 \text{ mm} \]

Grace drag
Tomiyama lift
T./A./F. Wall L. Force
FAD Turb. Disp.
SST turb. model
Sato model
\[ \Delta t = 0.002 \text{s} \]
2210 Iterations

© 2009 ANSYS, Inc. All rights reserved.
Monodispersed Bubbly Flow
MT-Loop Test Case FZR-052

FZD-052:

\[ \text{J}_L = 1.017 \text{ m/s} \]
\[ \text{J}_G = 0.0151 \text{ m/s} \]
\[ d_p = 4.4 \text{ mm} \]

Grace drag
Tomiyama lift
T./A./F. Wall L. Force
FAD Turb. Disp.
SST turb. model
Sato model
\[ \Delta t = 0.002 \text{s} \]
2400 Iterations
TOPFLOW Test Facility @ FZD

- Wire-mesh sensor
- Movable diaphragm
- Gas injection
- Movable diaphragm
- Movable obstacle
- Halfmoon diaphragm
- Flange 2
- Toothed rack
- Flange 1 with control kit

© 2009 ANSYS, Inc. All rights reserved.
TOPFLOW-074 Test Case
Conditions from Test Matrix

• Selection of test case conditions:

<table>
<thead>
<tr>
<th>Superficial water velocity (m/s)</th>
<th>Superficial gas velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.611</td>
<td>0.0368 0.0574 0.0896 0.14</td>
</tr>
<tr>
<td>1.017</td>
<td>0.074 0.085 0.096 0.107 0.118</td>
</tr>
<tr>
<td>0.405</td>
<td>0.072 0.083 0.094 0.105 0.116</td>
</tr>
<tr>
<td>0.102</td>
<td>0.069 0.080 0.091 0.102 0.113</td>
</tr>
</tbody>
</table>

• TOPFLOW-074 test case was subject of validation in the past
• Superficial velocities: \( J_G = 0.0368 \text{ m/s} \)
  \( J_L = 1.017 \text{ m/s} \)
• Wire-mesh sensor measurements at locations:
  \( z=\pm 10, 15, 20, 40, 80, 160, 250, 520 \text{mm} \)
3d Bubbly Flow Around Obstacle
Water Velocity Comparison

- 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
3d Bubbly Flow Around Obstacle
Air Void Fraction Comparison

- Comparison
  CFD ⇔ Experiment
- Air void fraction distribution in symmetry plane
3d Bubbly Flow Around Obstacle
Air Void Fraction Comparison

1) $z=10\text{mm}$
2) $z=15\text{mm}$
3) $z=20\text{mm}$
4) $z=40\text{mm}$
5) $z=80\text{mm}$
6) $z=160\text{mm}$
7) $z=250\text{mm}$
8) $z=520\text{mm}$

Run: 074
Air-water flow, 1 bar
VOID FRACTION
max = 14.9%
3d Bubbly Flow Around Obstacle Cross-Sectional Air Void Fraction

5) z=80mm

CFX Simulation

TOPFLOW Experiment
• Quantitative data comparison @ cross sections 
  \( z=\pm 10, \pm 15, \pm 20, \pm 40, \pm 80, \pm 160, \pm 250, \pm 520 \text{mm} \):
  - absolute water velocity
  - air volume fraction
**ANSYS CFX ↔ Experimental Data**

**Quantitative Comparison**

- **z=-80mm**
- **y=0mm**

### Graphs

1. **Normalized Air Volume Fraction**
   - **Experiment (z=-80mm)**
   - **CFX Simulation (z=-80mm)**
   - Graph shows the comparison between experimental and simulated data for normalized air volume fraction.

2. **Absolute Water Velocity [m/s]**
   - **Experiment (z=-80mm)**
   - **CFX Simulation (z=-80mm)**
   - Graph illustrates the comparison between experimental and simulated data for absolute water velocity.

© 2009 ANSYS, Inc. All rights reserved.
**ANSYS CFX ↔ Experimental Data**

**Quantitative Comparison**

$z = -20\text{mm}$

$y = 0\text{mm}$

![Graphs showing comparison between Experiment (z=-20mm) and CFX Simulation (z=-20mm)](image)

- **Norm. Air Volume Fraction**
- **Absolute Water Velocity**

© 2009 ANSYS, Inc. All rights reserved.

© 2009 ANSYS, Inc. Proprietary
z=20mm
y=0mm

**ANSYS CFX ↔ Experimental Data**

**Quantitative Comparison**

![Graphs showing comparison between experimental and CFX simulation data for normed air volume fraction and absolute water velocity at z=20mm and y=0mm.](image-url)
ANSYS CFX ↔ Experimental Data
Quantitative Comparison

\[ z = 80 \text{mm} \]

\[ y = 0 \text{mm} \]
z=250mm
y=0mm

**Quantitative Comparison**

- **Normalized Air Volume Fraction**
  - **Experiment (z=250mm)**
  - **CFX Simulation (z=250mm)**

- **Absolute Water Velocity**
  - **Experiment (z=250mm)**
  - **CFX Simulation (z=250mm)**

© 2009 ANSYS, Inc. All rights reserved.
Polydispersed Bubbly Flow Caused by Breakup & Coalescence

Transition from disperse bubbly flow to slug flow:

Balance between:

- **coalescence** of bubbles
- turbulent bubble **breakup**

→ bubble size distribution; **polydisperse bubbly flow**

→ counter-current radial motion of small and large bubbles; more than one velocity field

→ new population balance model (inhomogeneous MUSIG)
Inhomogeneous MUSIG Model

- momentum equations are solved for \( N \) gas phases (vel. groups)
- size fraction equations for \( M_i \) bubble size classes in each vel. group
- bubble coalescence and break-up over all \( \Sigma M_i \) MUSIG groups

\[ N(d_P) \]
\[ d_{P,krit} \]
\[ d_P \]

breakup/condensation
coalescence/evaporation

\[ \text{Velocity group mass transfer} \]

Break up
Coalescence

© 2009 ANSYS, Inc. All rights reserved.
Validation of 3x7 Inhomogeneous MUSIG Model on TOPFLOW-074

- good agreement at levels A, L through R
- too fast spreading of the bubble plume from inlet due to too intensive turbulent dispersion
• Basic population balance equations

\[
\frac{dn(m, \bar{r}, t)}{dt} = \frac{\partial}{\partial t} n(m, \bar{r}, t) + \frac{\partial}{\partial \bar{r}} \left( U(m, \bar{r}, t)n(m, \bar{r}, t) \right) + \frac{\partial n(m, \bar{r}, t)}{\partial m} \frac{\partial m(\bar{r}, t)}{\partial t} =
\]

= \left( \frac{m_i}{m_i - m_{i-1}} \right) \Gamma_i - \left( \frac{m_i}{m_{i+1} - m_i} \right) \Gamma_{i+1}

• Size fraction equations

\[
\frac{\partial}{\partial t} \left( \rho_i r_d f_i \right) + \frac{\partial}{\partial x^j} \left( \rho_i r_d U^j f_i \right) = S_{B_B} - S_{D_B} + S_{B_C} - S_{D_C} = S_i
\]

\[ S_i = \left\{ \begin{array}{l}
\frac{m_i}{m_i - m_{i-1}} \Gamma_i - \frac{m_i}{m_{i+1} - m_i} \Gamma_{i+1} \\
\frac{m_i}{m_i - m_{i-1}} \Gamma_{i-1} - \frac{m_i}{m_{i+1} - m_i} \Gamma_i
\end{array} \right. \]

for evaporation

for condensation

Bubble number density

Breakup/Coalescence terms

New terms
TOPFLOW Test Facility @FZD

D = 195mm
L = 8m

Courtesy of FZD

© 2009 ANSYS, Inc. All rights reserved.
Condensation Test Case

- \( P = 2 \) [MPa]
- \( J_w = 1.0 \) [m/s]
- \( J_s = 0.54 \) [m/s]
- \( T_s = 214.4 \) [°C]
- \( T_w = 210.5 \) [°C] → \( \Delta T_w = 3.9 \) [K]
- \( D_{inj} = 1 \) [mm]
- Detailed experimental data:
  - Bubble size distribution
  - Radial steam volume fraction distribution

Physical Model Setup

• Standard MUSIG & Extended MUSIG
  – 25 bubble size classes
  – 3 velocity groups:
    0→3 [mm], 3→6 [mm], 6→30 [mm]
  – Arranged in accordance with critical Tomiyama bubble diameter for bubble size dependent lift force
  – Break up model: Luo & Svendsen \( (F_B=0.025) \)
  – Coalescence model: Prince & Blanch \( (F_C=0.05) \)
### TOPFLOW Condensation Testcase

<table>
<thead>
<tr>
<th>Config</th>
<th>Inlet BC</th>
<th>Inlet Position</th>
<th>WLF</th>
<th>TD Force</th>
<th>Heat Transfer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Config 1</td>
<td>$D_{\text{inj}} = 4\text{ mm}$</td>
<td>Source point @ Wall</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Config 2</td>
<td>$D_{\text{inj}} = 4\text{ mm}$</td>
<td>Source point @ 75 mm</td>
<td>$F_{WLF}$</td>
<td>CTD=1.5</td>
<td>$\text{Nu}=2+0.15\text{Re}_{p}^{0.8}\text{Pr}^{0.5}$</td>
</tr>
<tr>
<td>Config 3</td>
<td>$D_{\text{inj}} = 1\text{ mm}$</td>
<td>Source point @ 75 mm</td>
<td>$F_{WLF}$</td>
<td>CTD=1.5</td>
<td>$\text{Nu}=2+0.15\text{Re}_{p}^{0.8}\text{Pr}^{0.5}$</td>
</tr>
</tbody>
</table>
Results: Vertical Averaged Steam Distribution
Results: Radial Steam Distribution

![Graph showing steam distribution with configurations 1, 2, and 3, and experiment levels A and C.](image-url)
Results: Radial Steam Distribution

![Graph showing steam volume fraction vs. radial position for different configurations and experiment levels.](image)
Results: Bubble Size Distribution

\[ \frac{d r_G}{d D_B} \]

Bubbles Diameter [mm]

- Config 1
- Config 2
- Config 3

Experiment Level A

Experiment Level C

© 2009 ANSYS, Inc. All rights reserved.

ANSYS, Inc. Proprietary
Results: Bubble Size Distribution

![Graph showing bubble size distribution with Config 1, Config 2, Config 3, and Experiment Level 1 and Experiment Level L.]
CFD Simulation for Fuel Assemblies in Nuclear Reactors

- Material Properties
- Turbulence
- Multiphase Flow Modeling
- Wall Boiling & Bulk Condensation
- Conjugate Heat Transfer (CHT)
- FSI: Stresses & Deformations
- Validation against Experiments
Multiphase Flow Regimes for Boiling Water Flow

- subcooled flow
- bubbly flow
- slug flow
- annular flow
- spray flow

ONB, OSB

T

T_{\text{sat}}

wall temperature

mean fluid temperature

x

subcooled boiling

nucleate boiling (saturated boiling)
Mechanistic wall heat partitioning model:

\[ \dot{q}_{\text{Wall}} = \dot{q}_F + \dot{q}_Q + \dot{q}_E \]

- **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**
  \[ \dot{q}_E = \dot{m} \cdot (h_G - h_L) \]
Submodels for closure of RPI wall boiling model:

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

**Correlation for bulk flow mean bubble diameter required:**

- e.g. Kurul & Podowski correlation via CCL

**Supported combination of wall boiling & CHT in the solid**

- GGI & 1:1 solid-fluid interfaces
RPI Wall Boiling Model in the ANSYS CFX-Pre 12.0 GUI

**Mass Transfer**
- **Option**: Phase Change
- **Phase Change Model**
  - **Option**: Thermal Phase Change
  - **Saturation Temperature**: SaturTemp
  - **Wall Boiling Model**
    - **Option**: RPI Model

**Heat Transfer**
- **Option**: Two Resistance
- **Liquid Heat Transfer**
  - **Option**: Pratico Marshall
- **Vapour Heat Transfer**
  - **Option**: Zero Resistance

**OK** button highlighted.
The Bartolomej et al. Testcase (1967, 1982)
The Bartolomej Test Case

- **R** = 7.7 mm
- **Z** = 2 m
- **q** = 0.57 MW/m²
- **G_{in}** = 900 kg/(s m²)

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>4.5 MPa</td>
</tr>
<tr>
<td>R</td>
<td>7.7 mm</td>
</tr>
<tr>
<td>G_{in}</td>
<td>900 kg/(s m²)</td>
</tr>
<tr>
<td>\dot{q}</td>
<td>0.57 MW/m²</td>
</tr>
<tr>
<td>Subcooling</td>
<td>58.2 K</td>
</tr>
</tbody>
</table>
Multiphase Flow Model

- **Steam-Water 2-phase flow:**
  - Water: continuous phase
  - Water Steam: disperse bubbles (particle model)

- **Material properties (EOS):**
  - IAPWS-IF97 water - water steam property tables

- **Modified law for interfacial area**
  - Kurul & Podowski type bulk bubble diameter: \( d_B = f(T_{sub}) \)
  - Accounting for higher volume fraction of the steam phase

- **Turbulence Model**
  - SST turbulence model for continuous phase
  - 0-eq. disperse phase turb. model + Sato bubble induced turbulence
Inter-Phase Mass, Momentum and Energy Transfer

• **Mass transfer model**
  – Thermal Phase Change Model (bulk boiling/condensation model)
  – RPI wall boiling model

• **Momentum transfer models**
  – Grace drag
  – FAD turbulent dispersion force
  – Tomiyama lift force
  – Wall lubrication force (none, Antal, Tomiyama)

• **Heat transfer models**
  – **Water:** Thermal Energy
  – **Water Steam:** Saturation temperature
  – Two resistance model
  – Ranz Marshall correlation for bubble heat transfer
Numerical Grids

- Validation on mesh hierarchy with regular refinement factor of 4 (2d meshes)

<table>
<thead>
<tr>
<th>Grid</th>
<th>Grid1</th>
<th>Grid2</th>
<th>Grid3</th>
</tr>
</thead>
<tbody>
<tr>
<td># Nodes (uniform)</td>
<td>20x150</td>
<td>40x300</td>
<td>80x600</td>
</tr>
<tr>
<td>Max y⁺</td>
<td>264</td>
<td>133</td>
<td>69</td>
</tr>
<tr>
<td>Δt [s]</td>
<td>10⁻²</td>
<td>10⁻³</td>
<td>5x10⁻⁴</td>
</tr>
</tbody>
</table>
Grid 2

Liquid Temperature
Plane 1

Gas Volume Fraction
Plane 1

© 2009 ANSYS, Inc. All rights reserved.
Grid 3
Comparison to Experimental Data - Parameter & Model Variation

Influence of wall heat flux:

Influence of wall lubrication force model:
The Lee et al. Testcase
(ICONE-16, 2008)
### Lee et al. (2008) Testcase

- **Axially symmetric circular annulus**
- **Radial dimensions**
  - Inner radius of outer tube: $R = 18.75\, \text{mm}$
  - Outer radius of inner tube: $R_0 = 9.5\, \text{mm}$
  - Core radius: $R_C = \frac{3}{4} R_0$
  - Annulus width: $9.25\, \text{mm}$
- **Axial dimensions**
  - Total heating section height: $L_T = 1670\, \text{mm}$
  - Distance between inlet and measuring plane: $L_M = 1610\, \text{mm}$
- **Radial Position: $R_P$**
  - Dimensionless, radial distance from inner tube ($R_P = 0$) to outer tube ($R_P = 1$) across the annulus:
  
  $$R_P = \frac{\left( r - R_0 \right)}{\left( R - R_0 \right)}$$
Geometry and Mesh generation:

**Figure 1:** 25° segment of the geometry

**Figure 2:** 1° simulated segment of the geometry

**Figure 3:** 1° segment with one-layer mesh example

**Figure 4:** Generated one layer mesh
Mesh Hierarchy

<table>
<thead>
<tr>
<th>Mesh Name</th>
<th>Grid 01 (coarse)</th>
<th>Grid 02 (medium)</th>
<th>Grid 03 (fine)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domains (1 = HFO, 2 = CHT) *</td>
<td>1: 6342 2: 12684</td>
<td>1: 24682 2: 49364</td>
<td>1: 97362 2: 194724</td>
</tr>
<tr>
<td>No. of Nodes</td>
<td>1: 6342</td>
<td>1: 24682</td>
<td>1: 97362</td>
</tr>
<tr>
<td></td>
<td>2: 12684</td>
<td>2: 49364</td>
<td>2: 194724</td>
</tr>
<tr>
<td>No. of Elements</td>
<td>1: 20x150</td>
<td>1: 40x300</td>
<td>1: 80x600</td>
</tr>
<tr>
<td>(hexahedra)</td>
<td>2: 40x150</td>
<td>2: 80x300</td>
<td>2: 160x600</td>
</tr>
<tr>
<td>$y^+_{\text{max}}$</td>
<td>Set16 ~84</td>
<td>Set16 ~41</td>
<td>Set16 ~24</td>
</tr>
<tr>
<td>(at 1st node near wall)</td>
<td>Set25 ~88</td>
<td>Set25 ~45</td>
<td>Set25 ~25</td>
</tr>
<tr>
<td>Tstep $\Delta t$ [s]</td>
<td>Set16 0.001</td>
<td>Set16 0.002</td>
<td>Set16 0.0002</td>
</tr>
<tr>
<td></td>
<td>Set25 0.1</td>
<td>Set25 0.0125</td>
<td>Set25 0.0002</td>
</tr>
</tbody>
</table>

© 2009 ANSYS, Inc. All rights reserved.

ANSYS, Inc. Proprietary
Selection of Extreme/Limiting Testcase Conditions

- Concentrating on 2 (out of 12) datasets:
  - Set 25 (least of all steam)
  - Set 16 (most of all steam)

- Parameter comparison

<table>
<thead>
<tr>
<th>Set No.*</th>
<th>$q''$ [kW m$^{-2}$]</th>
<th>G [kg m$^{-2}$s]</th>
<th>$T_{in}$ [$^\circ$C]</th>
<th>$P_{in}$ [kPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>320.4</td>
<td>718.8</td>
<td>83.8</td>
<td>121.1</td>
</tr>
<tr>
<td>25</td>
<td>220.0</td>
<td>1057.2</td>
<td>90.1</td>
<td>134.4</td>
</tr>
</tbody>
</table>
Found that submodels need modifications for BWR conditions
(see also Tu&Yeoh, Anglart et al., Krepper, Koncar):

1. **Bulk bubble diameter (BBD)**
   Kurul & Podowski → $d_{B,max} \sim 1.5\text{mm @ wall}$
   modified $d_B$ law → $d_{B,max} \sim 4.0\text{mm @ wall}$

2. **Bubble departure diameter (BDD)**
   Tolubinski & Kostanchuk → $d_W \sim 0.5\text{mm max.}$
   const. bubble dept. diam. → $d_W = 1\text{mm - 3mm}$

3. **$A_2$ - Wall area fraction influenced by steam bubbles**
   default → 0.5
   increased up to 2.0
BBD & BDD Modifications
Test Matrix Overview

- Trying to systematically increase Bubble Departure Diameter to investigate its influence on Heat Flux to Vapor ($Q_V$) profile
  - Test series with increasing BDD starting from $d_{W,\text{max}} \approx 0.5$ mm
  - 1 mm; 2 mm; 3 mm
  - $T&K \times 4.0$

<table>
<thead>
<tr>
<th></th>
<th>BDD Tolubinsky &amp; Kostanchuk</th>
<th>BDD User defined $d_W$ [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>K&amp;P</td>
<td>yes</td>
<td>-</td>
</tr>
<tr>
<td>bbdmod01</td>
<td>-</td>
<td>1 = const.</td>
</tr>
<tr>
<td>bbdmod02</td>
<td>-</td>
<td>2 = const.</td>
</tr>
<tr>
<td>bbdmod03</td>
<td>-</td>
<td>3 = const.</td>
</tr>
</tbody>
</table>

© 2009 ANSYS, Inc. All rights reserved.
ANSYS, Inc. Proprietary
Set25 : Bulk Diameter Modification Comparison:
Gas Volume Fraction ($r_G$)

![Graph showing gas volume fraction comparison at z = 1610 mm]
Set25 Bubble Departure Diameter Modification Comparison:
Gas Volume Fraction ($r_G$)

![Graph showing gas volume fraction comparison](image)
Set25 $A_{2F}$ Mod Comparison:
Gas Volume Fraction ($r_G$)

Radial Position ($R_p$) [ ]
Set25 New Grid Comparison:
Gas Volume Fraction ($r_G$)

- $r_G$ vs Radial Position ($R_p$)

Graph showing the gas volume fraction at $z = 1610$ [mm] for different grid models.
The Lee et al. Testcase (ICONE-16, 2008) - Conjugate Heat Transfer -
HFO (Heat Flux Only): Fluid Domain (Annulus) → area specific heat flux boundary condition

CHT (Conjugated Heat Transfer): Fluid Domain (Annulus) + Solid Domain (Non-Heated Rod Shell) + Solid Domain (Heated Rod Core) → volume specific heat source
The RPI Wall Boiling Model: Lee et al. Testcase with CHT

- Specific energy source in solid material, Set25 (equiv. to $q_{\text{Wall}}$):

$$E_{\text{Core}} = 8.23 \times 10^7 \text{ [W/m}^3\text{]}$$

- Temperature and Steam VF distribution in vertical plane
The RPI Wall Boiling Model:
Lee et al. Testcase with CHT

Set25 & CHT: Water temperature monitors $\Delta x_w=1.5\text{mm}$, $\Delta z=83.5\text{mm}$,
The RPI Wall Boiling Model: Lee et al. Testcase with CHT

Set25 & CHT: Grid independence for temperature distribution @ z=1610[mm]
The RPI Wall Boiling Model: Lee et al. Testcase with CHT

Set25 & CHT: Vapour VF distribution @ z=1610[mm]
R&D Initiative:
“Modeling, Simulation & Experiments for Boiling Processes in Fuel Assemblies of PWR”
Modeling, Simulation & Experiments for Boiling Processes in Fuel Assemblies of PWR

- Ultrafast electron beam X-ray CT of fuel rod bundle in titanium pipe on TOPFLOW @ FZD:

Images by courtesy of U. Hampel, FZD
Wall boiling simulation in a 3x3 rod bundle with spacer grid:

Wall superheat $T_{W} - T_{Sat}$
Summary & Outlook

• Overview on ANSYS CFD multiphase flow model development and validation
• Continuous effort in model improvement, R&D
• Emphasis in validation on BPG, comparison to data, geometry & grid independent modeling
• High interoperability of physical models

• Outlook:
  – Ongoing & customer driven CFD model development
  – Research cooperation with Industry & Academia
  – More & more complex MPF phenomena
  – Coupling of wall boiling model to inhomogeneous MUSIG
  – Extension of the wall heat partitioning in wall boiling model

© 2009 ANSYS, Inc. All rights reserved.