Assessment of the Eulerian Two-Fluid Model for disperse and sharp interface flow simulation with application in steam generators

Godino Dario[a], Santiago F. Corzo[a], Norberto M. Nigro[a], Damian E. Ramajo[a]

[a] Research Center for Computational Methods
CIMEC-UNL/CONICET, Santa Fé, Argentina

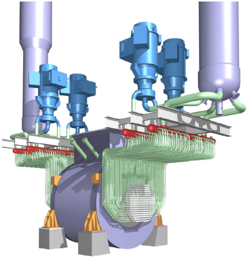
23-10-2019

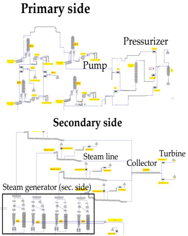
Nuclear New Horizons: Fueling or Future

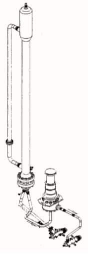
October 21-25, 2019 - Santos, SP, Brazil

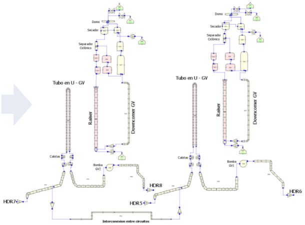
Motivation

In the nuclear industry the **multiphase flows** play a preponderant role. To understand these flows is of great importance for the design, and particularly to ensure the safe operation of the plants







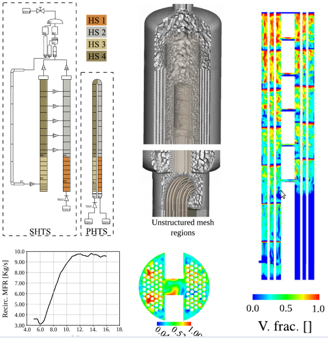


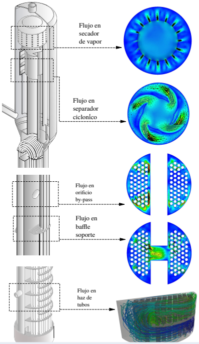
The computational simulation has been mostly from **system codes**, which are widely used for the design and verification of nuclear power plants due to their ability to solve the overall thermal-hydraulics, neutronic and plant control in a simplified way.

Motivation and general objective

MOTIVATION

CFD models can be used in scale steam generators simulation, but not in real-scale installations.





To improve the two-phase flow simulation, some benchmark tests must be performed

Corzo,Godino,Nigro,Ramajo, Thermal hydraulics simulation of the RD-14M steam generator facility, ***Annals of Nuclear Energy***, 2017

Godino,Corzo,Nigro,Ramajo, CFD simulation of the pre-heater of a nuclear facility steam generator using a thermal coupled model, ***Nuclear Engineering and Design***, 2018

Objectives of the present work

Interfacial momentum exchange

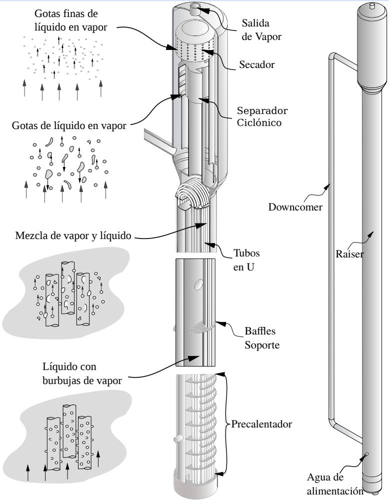
Evaluate the effect of drag, lift, wall lubrication, turbulent dispersion and virtual mass forces on the different regime flows to find a unique set of models able for the all of them.

Turbulence

Assess the more widely used turbulence models for industrial (real-scale) applications.

Blending

Evaluate the use of blending strategy for representing the rheology changes.



Governing equations

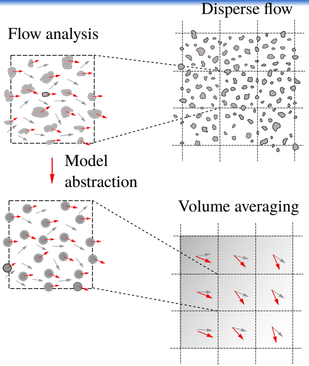
- Continuity equation for α (φ_1 : Continuous phase, φ_2: Disperse phase):

$$\frac{\partial(\alpha_\varphi \rho_\varphi)}{\partial t} + \nabla \cdot (\alpha_\varphi \rho_\varphi \mathbf{U}_\varphi) = \Gamma_\varphi$$

- Conservation momentum equation:

$$\frac{\partial(\alpha_\varphi \rho_\varphi \mathbf{U}_\varphi)}{\partial t} + \nabla \cdot (\alpha_\varphi \rho_\varphi \mathbf{U}_\varphi \mathbf{U}_\varphi) = -\nabla \cdot (\alpha_\varphi (\tau_\varphi + \mathbf{R}_\varphi))$$

$$-\alpha_\varphi \nabla p + \alpha_\varphi \rho_\varphi \mathbf{g} + \mathbf{M}_\varphi + (\Gamma_{2,1} \mathbf{U}_\varphi - \Gamma_{1,2} \mathbf{U}_\varphi)$$



- Energy conservation equation (in terms of total energy)

$$\frac{\partial[\alpha_\varphi \rho_\varphi (h_\varphi + \frac{1}{2}\mathbf{U}_\varphi \mathbf{U}_\varphi)]}{\partial t} + \nabla \cdot [\alpha_\varphi \rho_\varphi \mathbf{U}_\varphi (h_\varphi + \frac{1}{2}\mathbf{U}_\varphi \mathbf{U}_\varphi)] - \alpha_\varphi \left(\frac{\partial p}{\partial t} + \mathbf{U}_\varphi \cdot \nabla p \right) =$$

$$-\nabla \cdot (\alpha_\varphi q_\varphi) + \alpha_\varphi \nabla \cdot (\mathbf{U}_\varphi \tau_\varphi) + (\Gamma_{\varphi,\varphi'} h_\varphi - \Gamma_{\varphi',\varphi} h_\varphi) + \alpha_\varphi \mathbf{U}_\varphi \mathbf{M}_\varphi$$

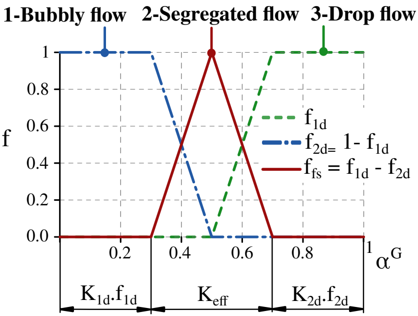
In the momentum equations the **coupling between the two phases** is through the **interfacial force terms** M_φ and the moment exchanged during the **mass transfer** (evaporation).

Blending methodology

Blending methodology is used to locally and run-time switch between three flow regime:

1-Bubbly flow ($\alpha_s < 0{,}3$): Steam bubbles moves in a slow liquid continuous flow (riser). Due to the lower bubble density, the higher bubble deformation, and the higher liquid viscosity, the drag, lift, wall lubrication, turbulent dispersion and virtual mass forces are really significant.

2-Segregated flow ($0{,}3 < \alpha_s < 0{,}7$): Is a transition regime.



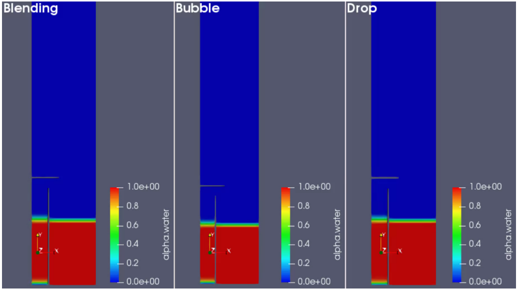
$$K_{eff} = K_{1d}.f_{1d} + K_{2d}.f_{2d} + K_{sf}.f_{sf}$$

3-Drop flow ($\alpha_s > 0{,}7$): Liquid drops are dragged by a fast steam flow (separators and dryers). Due to the larger liquid density and the lower steam viscosity, the lift, wall lubrication, turbulent dispersion and virtual mass forces of minor impact. Drag and inertial forces compete.

Blending

Blending

- Fluid: water and air
- Bubble/drop size: 3 mm
- Air inlet velocity: 0.4 m/s
- Interfacial models: Drag (Grace/Tomiyama/Marschall), lift(Tomiyama),WLF(Frank), TDF (Burn),VMF (Constant)



Air/liquid vertical column

Video blending

Interfacial moment exchange

Interfacial forces:

$$M_{1,2} = -M_{2,1} = M^D + M^L + M^{WL} + M^{VM} + M^{TD}$$

1. Drag force

$$F_D = -\frac{3}{4} C_D \frac{\alpha_2 \rho_1}{d_2} |\mathbf{U_R}| \mathbf{U_R}$$

2. Lift force

$$F_L = C_L \alpha_2 \rho_1 \mathbf{U_R} \times \nabla \times \mathbf{U_R}$$

3. Wall lubrication force

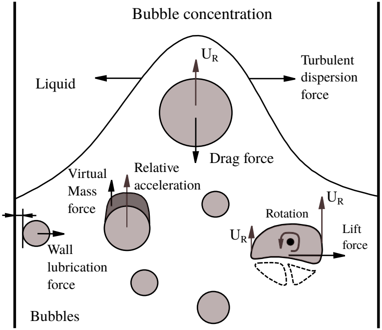
$$F_{WL} = -C_{WL} \alpha_2 \rho_1 |\mathbf{U_R}|^2 \mathbf{n}$$

4. Turbulent dispersion force

$$F_{TD} = -C_{TD} \rho_1 k_1 \nabla \alpha_2$$

5. Virtual mass force

$$F_{VM} = C_{VM} \alpha_2 \rho_1 \left(\frac{DU_2}{dt} - \frac{DU_1}{dt} \right)$$



Drag models:
Grace for bubbles in water
Tomiyama for drops in air
Marschall for continuous-continuous fluid

Interfacial moment exchange

Interfacial forces:

$$M_{1,2} = -M_{2,1} = M^D + M^L + M^{WL} + M^{VM} + M^{TD}$$

1. Drag force

$$F_D = -\frac{3}{4} C_D \frac{\alpha_2 \rho_1}{d_2} |\mathbf{U_R}| \mathbf{U_R}$$

2. Lift force

$$F_L = C_L \alpha_2 \rho_1 \mathbf{U_R} \times \nabla \times \mathbf{U_R}$$

3. Wall lubrication force

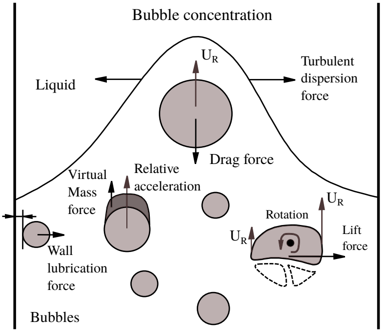
$$F_{WL} = -C_{WL} \alpha_2 \rho_1 |\mathbf{U_R}|^2 \mathbf{n}$$

4. Turbulent dispersion force

$$F_{TD} = -C_{TD} \rho_1 k_1 \nabla \alpha_2$$

5. Virtual mass force

$$F_{VM} = C_{VM} \alpha_2 \rho_1 \left(\frac{DU_2}{dt} - \frac{DU_1}{dt} \right)$$



Lift models: Constant, Moraga, **Tomiyama**

Interfacial moment exchange

Interfacial forces:

$$M_{1,2} = -M_{2,1} = M^{D} + M^{L} + M^{WL} + M^{VM} + M^{TD}$$

1. Drag force

$$F_D = -\frac{3}{4} C_D \frac{\alpha_2 \rho_1}{d_2} |\mathbf{U_R}| \mathbf{U_R}$$

2. Lift force

$$F_L = C_L \alpha_2 \rho_1 \mathbf{U_R} \times \nabla \times \mathbf{U_R}$$

3. Wall lubrication force

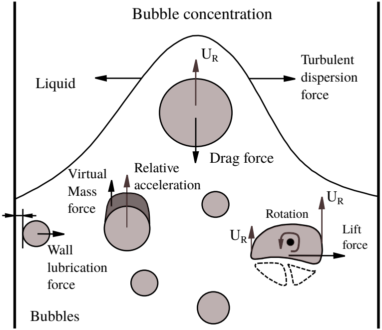
$$F_{WL} = -C_{WL} \alpha_2 \rho_1 |\mathbf{U_R}|^2 \mathbf{n}$$

4. Turbulent dispersion force

$$F_{TD} = -C_{TD} \rho_1 k_1 \nabla \alpha_2$$

5. Virtual mass force

$$F_{VM} = C_{VM} \alpha_2 \rho_1 \left(\frac{DU_2}{dt} - \frac{DU_1}{dt} \right)$$



Wall lubrication models: Antal, **Frank**

Interfacial moment exchange

Interfacial forces:

$$M_{1,2} = -M_{2,1} = M^D + M^L + M^{WL} + M^{VM} + M^{TD}$$

1. Drag force

$$F_D = -\frac{3}{4} C_D \frac{\alpha_2 \rho_1}{d_2} |\mathbf{U_R}| \mathbf{U_R}$$

2. Lift force

$$F_L = C_L \alpha_2 \rho_1 \mathbf{U_R} \times \nabla \times \mathbf{U_R}$$

3. Wall lubrication force

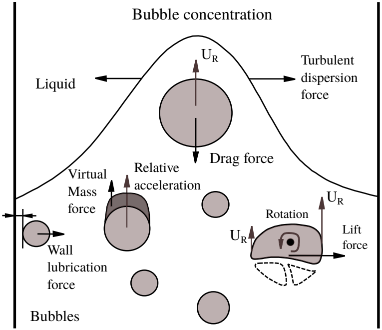
$$F_{WL} = -C_{WL} \alpha_2 \rho_1 |\mathbf{U_R}|^2 \mathbf{n}$$

4. Turbulent dispersion force

$$F_{TD} = -C_{TD} \rho_1 k_1 \nabla \alpha_2$$

5. Virtual mass force

$$F_{VM} = C_{VM} \alpha_2 \rho_1 \left(\frac{DU_2}{dt} - \frac{DU_1}{dt} \right)$$



Turbulent dispersion models: Lopez de Bertodano, Gosman, **Burn**

Interfacial moment exchange

Interfacial forces:

$$M_{1,2} = -M_{2,1} = M^D + M^L + M^{WL} + M^{VM} + M^{TD}$$

1. Drag force

$$F_D = -\frac{3}{4} C_D \frac{\alpha_2 \rho_1}{d_2} |\mathbf{U_R}| \mathbf{U_R}$$

2. Lift force

$$F_L = C_L \alpha_2 \rho_1 \mathbf{U_R} \times \nabla \times \mathbf{U_R}$$

3. Wall lubrication force

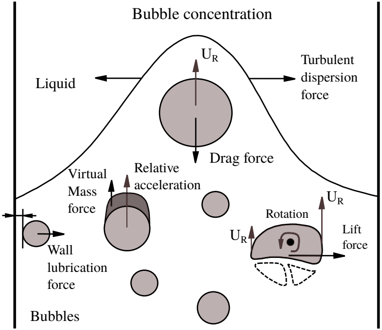
$$F_{WL} = -C_{WL} \alpha_2 \rho_1 |\mathbf{U_R}|^2 \mathbf{n}$$

4. Turbulent dispersion force

$$F_{TD} = -C_{TD} \rho_1 k_1 \nabla \alpha_2$$

5. Virtual mass force

$$F_{VM} = C_{VM} \alpha_2 \rho_1 \left(\frac{DU_2}{dt} - \frac{DU_1}{dt} \right)$$



Virtual mass models: **Constant**, Lamb

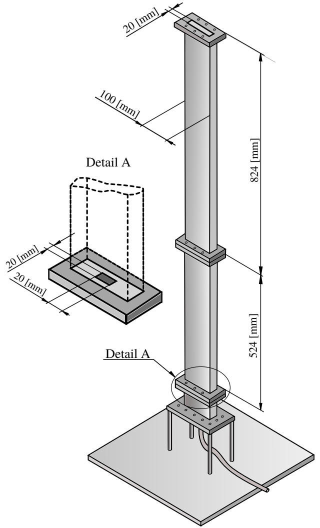
Multiphase benchmarks

Four two-phase benchmarks were considered:

- **Case 1. Bubble plume:** Ascending bubble plume in a vertical rectangular column of water
- **Case 2. TOPFlOW:** Gas-Liquid Flow around an Obstacle in a Vertical Pipe (TOPFLOW experiments)
- **Case 3. Horizontal gas/liquid flow:** Counter-current flow of water and air (HAWAC experiments)
- **Case 4. Horizontal fluid/fluid flow:** Co-current flow

Case 1. Bubble plume

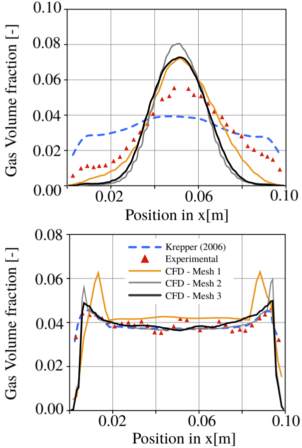
- **Case 1. Bubble plume**
  The test was carried out and simulated by Krepper et al. in 2007[1].
  Air is injected from a sparger at the bottom side of a rectangular water column of 100 mm wide, 20 mm depth and 1448 mm height.
  Small bubbles (1 mm < ϕ_b < 5 mm) ascend swelling the column.
  The void fraction distribution and the average void fraction for several gas velocities are measured.
  In this test the effect of all the interfacial forces becomes significant.



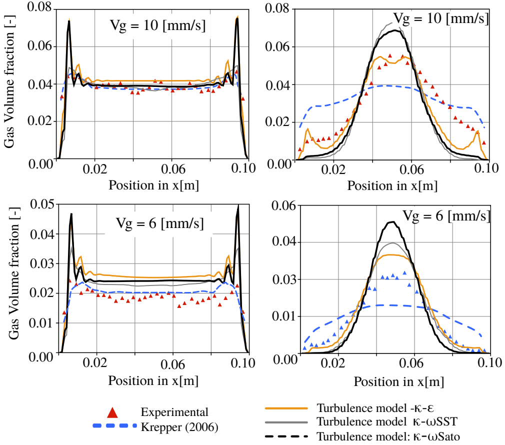
[1] **Krepper**, Vanga, Zaruba, Prasser, and Lopez de Bertodano. Experimental and numerical studies of void fraction distribution in rectangular bubble columns. **Nuclear eng. and design**, 237(4), **2007**.

Case 1. Bubble plume

Mesh convergence (V_g = 10 mm/s)

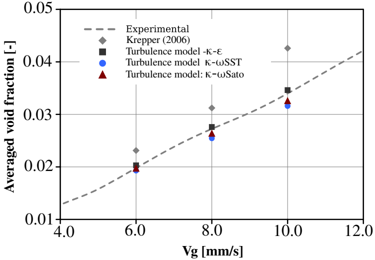


Turbulence modeling (V_g = 6, 10 mm/s)



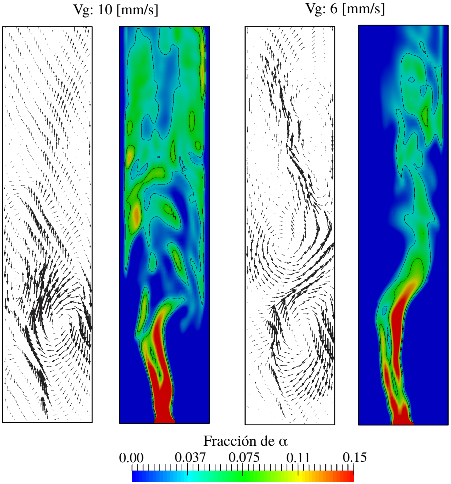
Case 1. Bubble plume (Cont.)

Swelling effect



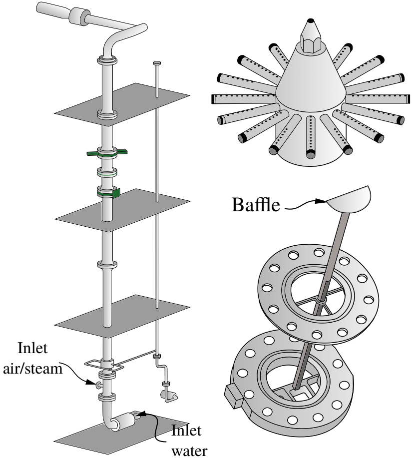
Kreeper[1] overestimated the averaged void fraction. The error increased with V_g up to 26 %.
$\kappa - \omega$ SST model showed the best agreement.

Void fraction and velocity patterns



Case 2. Flow around an obstacle

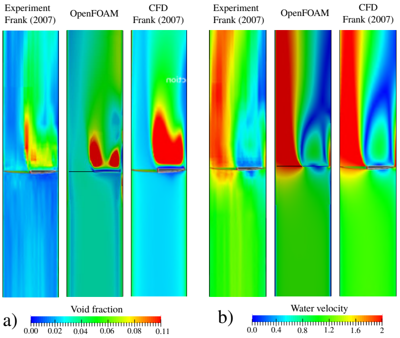
- **Case 2. TOPFlOW:**
  The test was carried out and simulated by Prasser et al. in 2008[2].
  Air is injected from a perforated injector tubes introducing small bubbles (2 mm < ϕ_b < 12 mm) at the bottom side of a circular water column of 195 mm of diameter and 9 m of height.
  A constant water flow is circulated and the air-water mixture pass through an obstacle.
  The void fraction and the phases velocities patterns for several gas/liquid velocities are measured downstream of the obstacle.
  Air accumulation over the obstacle is measured.



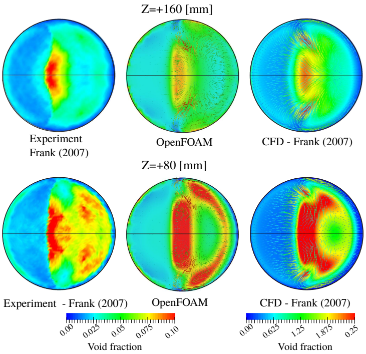
[2] Prasser, Beyer, Frank, Al Issa, Carl, Pietruske, and Schütz. Gas–liquid flow around an obstacle in a vertical pipe. **Nuclear eng. and design**, 238(7), **2008**.

Case 2. Flow around an obstacle

Void fraction - Water Velocity



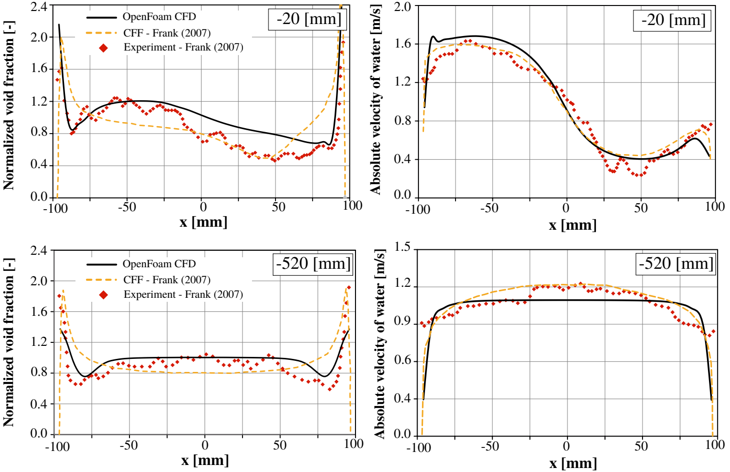
Void fraction



Comparison between Experiments, OpenFOAM and Prasser[2].

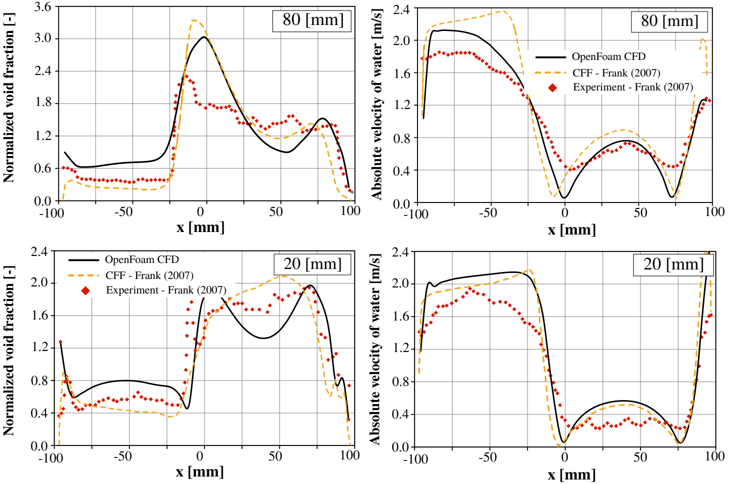
Case 2. Flow around an obstacle (Cont)

Normalized void fraction and axial water velocity axial in symmetry planes. Upstream of the obstacle (z = -20 and -520 mm)



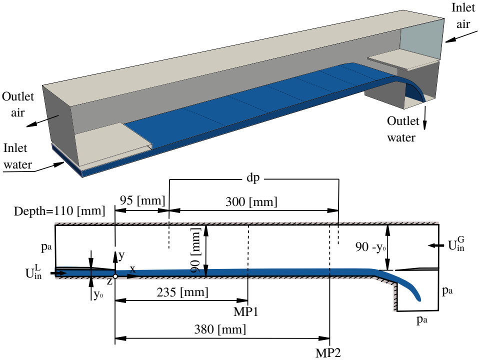
Case 2. Flow around an obstacle (Cont)

Normalized void fraction and axial water velocity axial in symmetry planes.Downstream of the obstacle (z = 80 and 20 mm)



Case 3. Counter-current flow

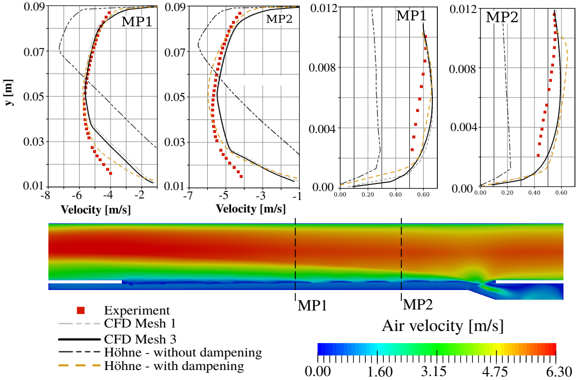
- **Case 3. Horizontal counter-current gas/liquid flow:** The test was performed by Stäbler[3], and numerically reproduced by Wintterle et al.[4], Porombka and Höhne [5]. This consists on a rectangular channel of 583 mm in wide, 110 mm in depth, and 138 mm in height. The water enters at 0.7 m/s from the left side through a 9 mm height section whereas the air flows at 4.44 m/s in counter-current direction.



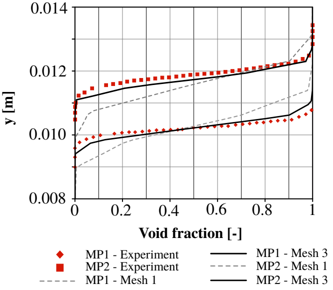
[3] Stäbler. Experimentelle untersuchung und physikalische beschreibung der schichtenströmung in horizontalen kanälen. **2007**.
[4] Wintterle, Laurien, Stäbler, Meyer, and Schulenberg. Experimental and numerical investigation of counter-current stratified flows in horizontal channels. **Nuclear eng. and design**, 238, **2008**.
[5] Porombka and Höhne. Drag and turbulence modelling for free surface flows within the two-fluid euler–euler framework. **Chemical Engineering Science**, 134, **2015**.

Case 3. Counter-current flow

Air velocity - Water velocity



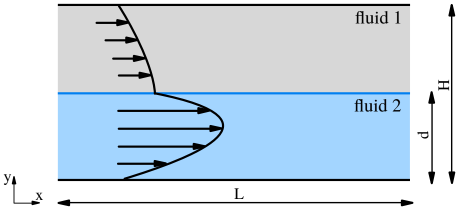
Void fraction along two vertical lines located at the MP_1 and MP_2 axial positions



Comparison between OpenFOAM, experiments [3] and simulations [4,5].

Case 4. Co-current flow

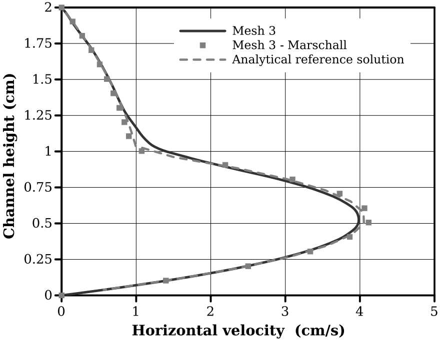
- **Case 4. Horizontal co-current flow liquid/liquid:** The test consists of a 2D rectangular channel of 40 mm in wide and 20 mm in height, where two miscible fluids with the same density of 1 kg/m (non-buoyant) and viscosities of 1,85 $\times$ 10^{-5} Pa.s and 5 $\times$ 10^{-5} flows co-current driven by a imposed pressure difference of 2.1 mPa. This is an academic test with analytic and numeric solutions proposed by Marschall[6].



[6] Marschall. H. Marschall. Towards the numerical simulation of multi-scale two-phase flows. PhD thesis, Technische Universität München, **2011**.

Case 4. Co-current flow

Fluid velocities



A very good agreement was found. Only drag is significant for this test. The segregated model is the only one able to correctly capture the interfacial efforts

Comparison between OpenFOAM and analytic results [6].

Conclusions

- 1- The two-fluid model was assessed against four benchmarks representing flow regime commonly found in steam generators and many industrial processes.
- 2- The four cases were solved with the same computational model and compared with experimental and analytic results founding good agreement for all cases.
- 3- The **linear blending model** was suitable for switching between disperse and segregated flow
- 4- The **segregated model** proposed by Marschall for simple co-current flows was also suitable for capturing the interfacial drag in more complex counter-current flows.
- 5- The κ **-** ω **SST** model showed to be a little better than the k-epsilon realizable and the $\kappa - \omega$-Sato models.

Thank you for your attention!

The authors would like to thank:

- Universidad Nacional del Litoral (CAI+D 2016 PIC 50420150100067LI).
- Agencia Nacional de Promoción Científica y Tecnológica (PICT 2016-2908)
- Consejo Nacional de Investigaciones Científicas y Tecnológicas CONICET.