SIMULATION OF SLUG-FLOW CONDITION IN VALVES

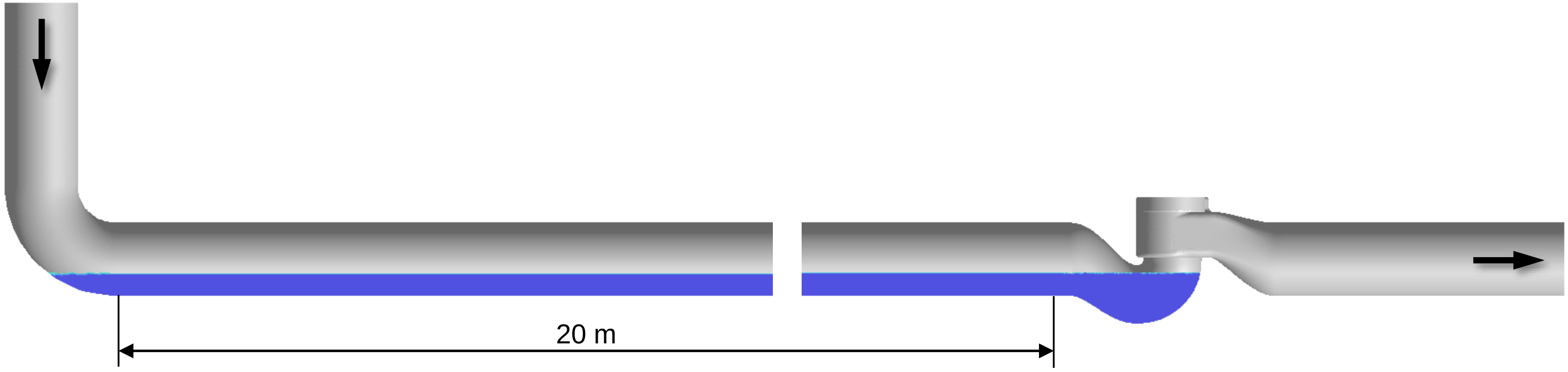
PhD Eng. Răzvan MAHU
PhD Eng. Prof. Florin POPESCU
PhD Eng. Prof. Ion V. ION

Agenda

- Case introduction
- Physical modeling options in 3D CFD
- Numerical modeling and simulation approach
- Numerical results
- Conclusions, Q&A

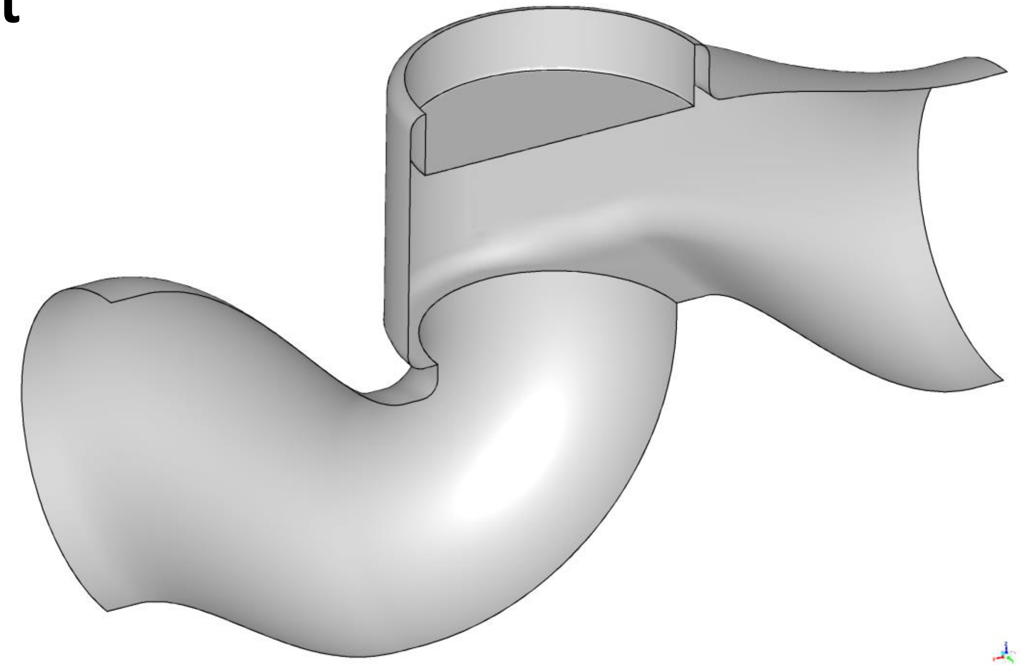
Case Intro

- **A DN200 Valve at the end of a 20 m horizontal Pipe**
- **A 90° bend was placed at the Pipe inlet**
- **Operational conditions:**
  - Inlet Total Pressure = 11 bar
  - Outlet Static Pressure = 1 bar
  - Pipe contains dry steam at 184°C and condensate (0.28 liquid fraction), all initially at rest



Case Intro (2)

- Valve position is “fully opened” at simulation start
- The system pressure is initially equal to Inlet Pressure; a downstream valve is opened, causing a decompression wave to travel upstream; at t = 0 s this wave is considered to have just reached system Outlet
- The simulation must determine the actual flow conditions (i.e. Slugging) and evaluate the corresponding flow properties



Physical Modeling (1)

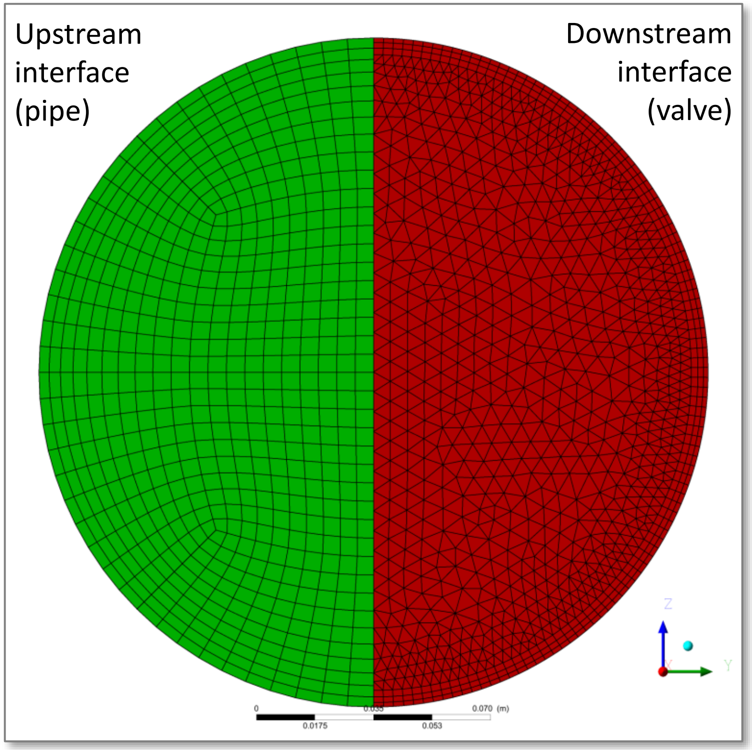
	Pros	Cons
VOF Model	+ Easy to implement and solve + Good interface tracking properties + Very stable	– No phase slip, shared momentum and temperature fields – Interface sharpness and detail strongly depends on mesh resolution
Eulerian Multi-Fluid Model	+ Solves a set of transport eq. for each phase → strong interphase coupling + High accuracy for most flow regimes	– Difficult to implement and solve – Inherently diffuse interface – Large CPU and memory requirements – Low numerical stability
Hybrid Models	+ Have all the advantages of both VOF and Eulerian models	– Computationally intensive – Low numerical stability

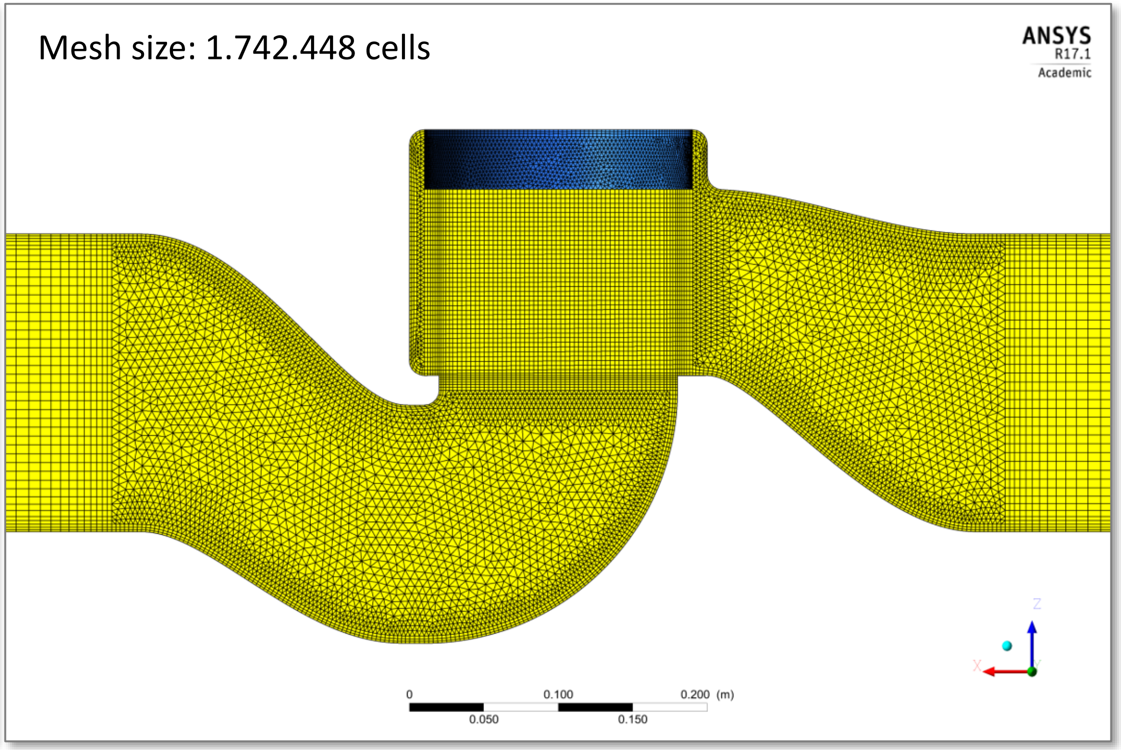
Physical Modeling (3)

- **Model equations (RANS)**
  - Momentum: $$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla\cdot(\rho\vec{v}\vec{v})=-\nabla p+\nabla\cdot\left[\mu\left(\nabla\vec{v}+\nabla\vec{v}^{T}\right)\right]+\rho\vec{g}+\vec{F}$$
  - Energy: $$\frac{\partial}{\partial t}(\rho E)+\nabla\cdot(\vec{v}(\rho E+p))=\nabla\cdot(k_{eff}\nabla T)+S_h$$
  - Volume fraction: $$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}\left(\alpha_q\rho_q\right)+\nabla\cdot\left(\alpha_q\rho_q\vec{v}_q\right)=S_{\alpha_q}+\sum_{p=1}^{n}\xcancel{\left(\dot{m}_{pq}-\dot{m}_{qp}\right)}\right]$$
- **Turbulence modeling**
  - SST k-omega model with:
    - Curvature correction & Compressibility effects
    - Turbulence production limiting & Interface turbulence damping control
- **Material properties**
  - All mixture properties: $\rho=\sum\alpha_q\rho_q$
  - Both phases are treated as compressible fluids!
    - Steam modeled as Ideal Gas: $$\rho=\frac{p_{op}+p}{\frac{R}{M_w}T}$$
    - Condensate modeled using a simplified form of Tait eq.: $$\left(\frac{\rho}{\rho_0}\right)^{n}=\frac{K}{K_0} \qquad K=K_0+n\Delta p \qquad \Delta p=p-p_0 \qquad c=\sqrt{\frac{K}{\rho}}$$

Numerical Modeling & Simulation

- The computational domain was reduced to half using the longitudinal symmetry plane
- A hybrid meshing technique was used, with a pipe-to-valve non-conformal interface to reduce mesh size



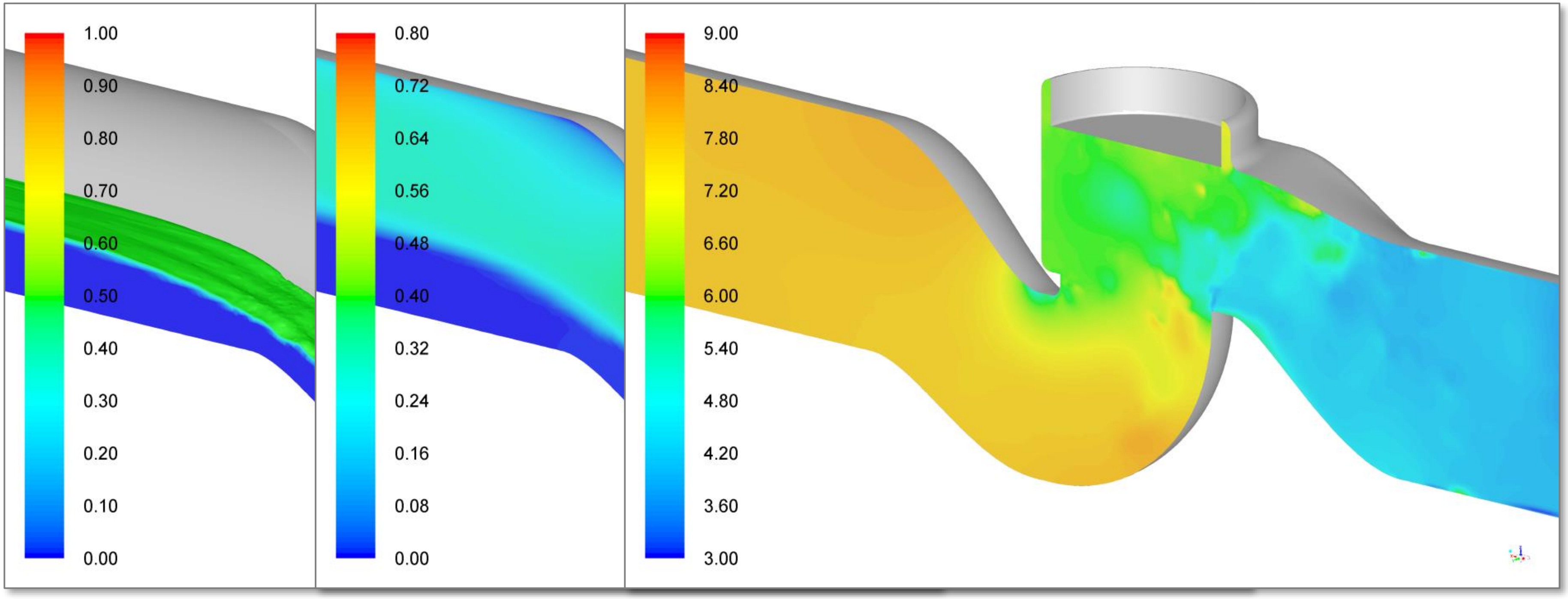


Numerical Modeling & Simulation (2)

- **Boundary conditions**
  - Pipe inlet: PRESSURE-INLET, $p_{total} = 11$ bar, t = 184°C, no liquid
  - Valve outlet: PRESSURE-OUTLET, $p_{static} = 1$ bar
  - Pipe & valve walls: WALL, adiabatic
  - ZX plane: SYMMETRY
- **Numerical solver**
  - Fully coupled pressure-based solver (pressure, velocity, volume fraction)
- **Spatial discretization**
  - Gradient: Green-Gauss node-based
  - Pressure: PRESTO
  - All other eq.: Second-order Upwind
- **Interface tracking**
  - Implicit Compressive scheme: $\alpha_f=\alpha_d+\beta\nabla\alpha_d\cdot d\vec{r}$ with $\beta = 2$
- **Temporal discretization**
  - Bounded Second-order Upwind

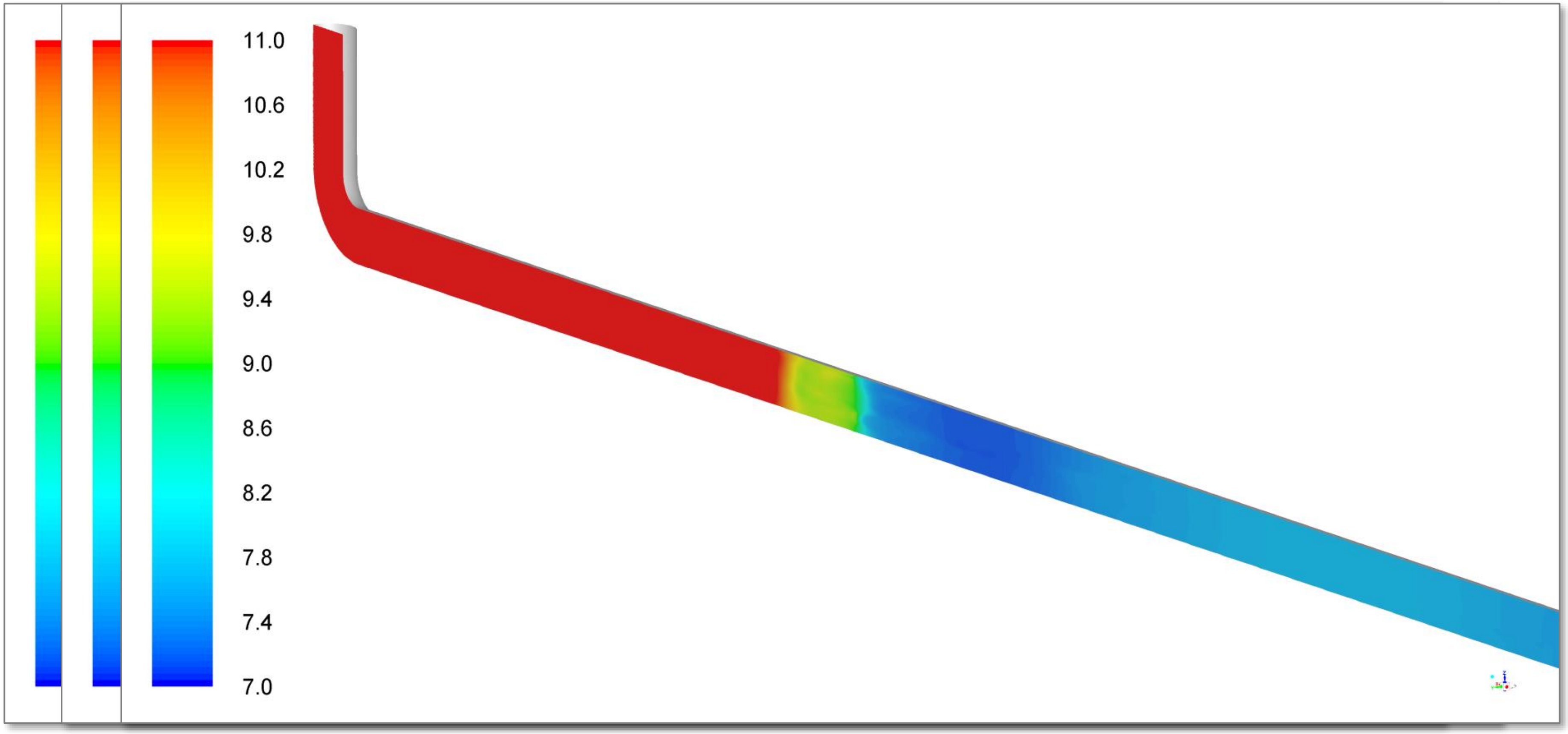
Numerical Results

- **Observations:**
  - Initially, the steam flow is strictly kept under control by the large pressure drop induced by the condensate almost filling the upstream valve branch
  - As the condensate pool is removed by the flowing steam, the total mass flow increases steadily, while the main pressure loss source is the valve itself



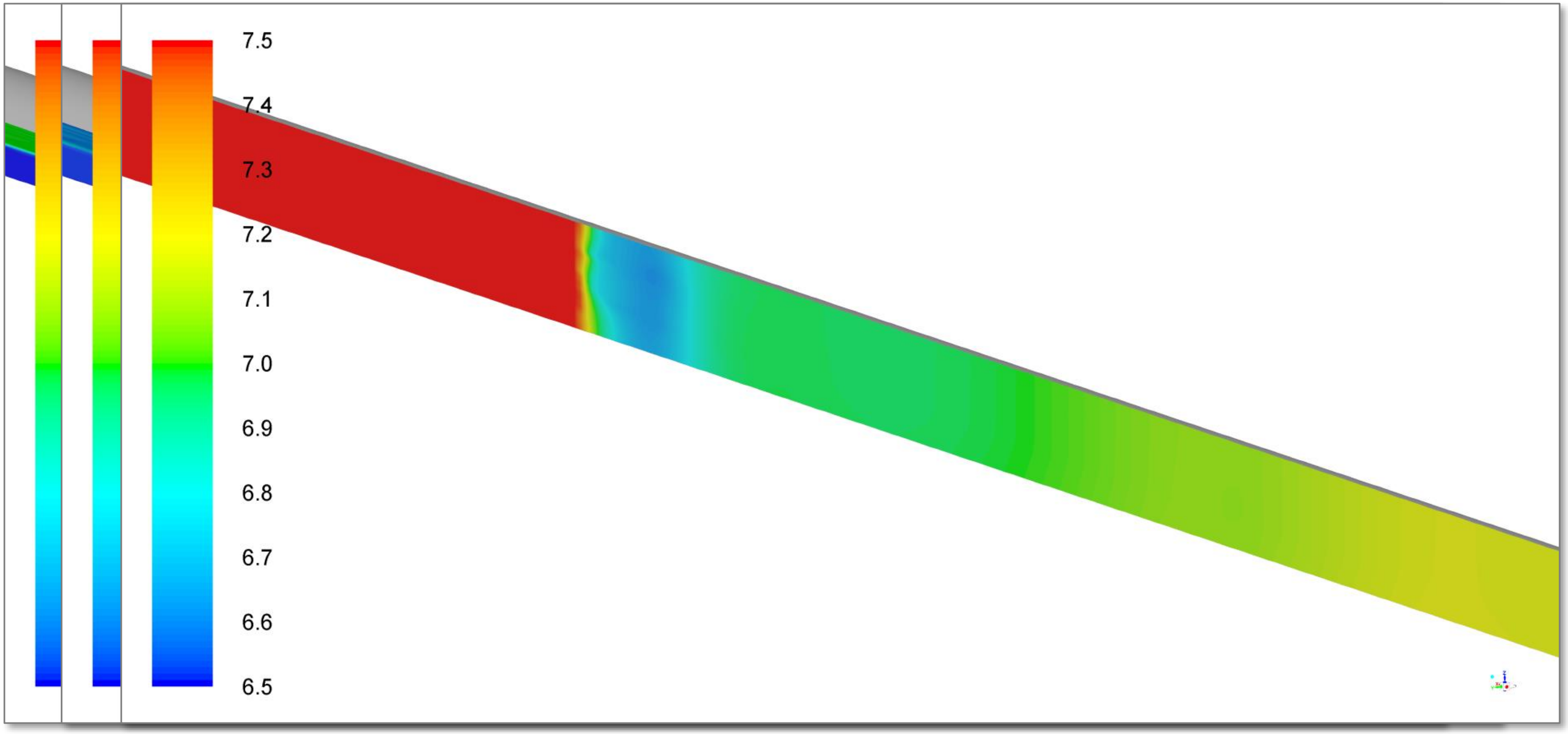
Numerical Results (2)

- **Observations (cont.):**
  - Due to the 90° bend flow conditions, the first slug starts forming directly at the bend-pipe junction



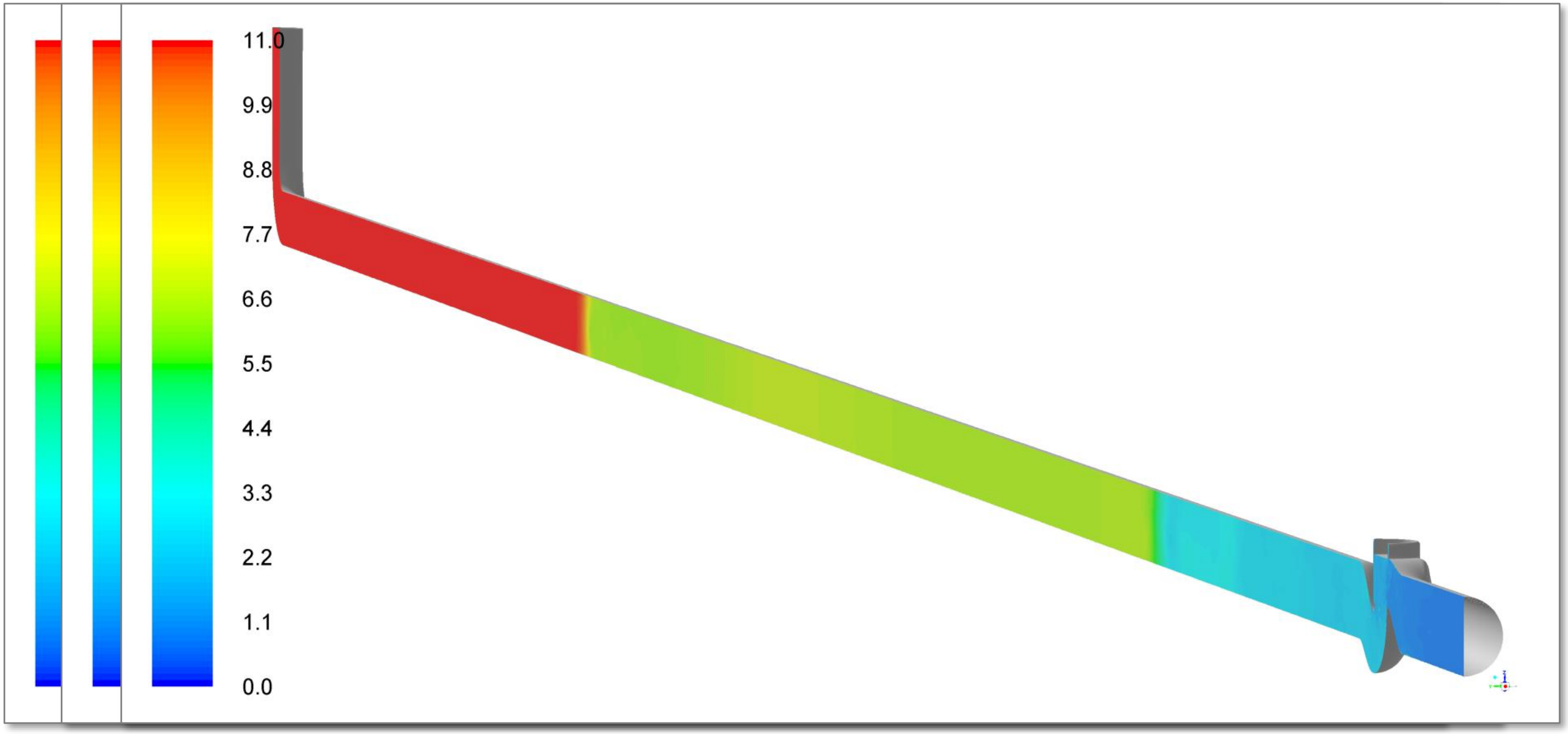
Numerical Results (3)

- **Observations (cont.):**
  - However, completely independently, a second slug spontaneously appears around mid-pipe shortly after



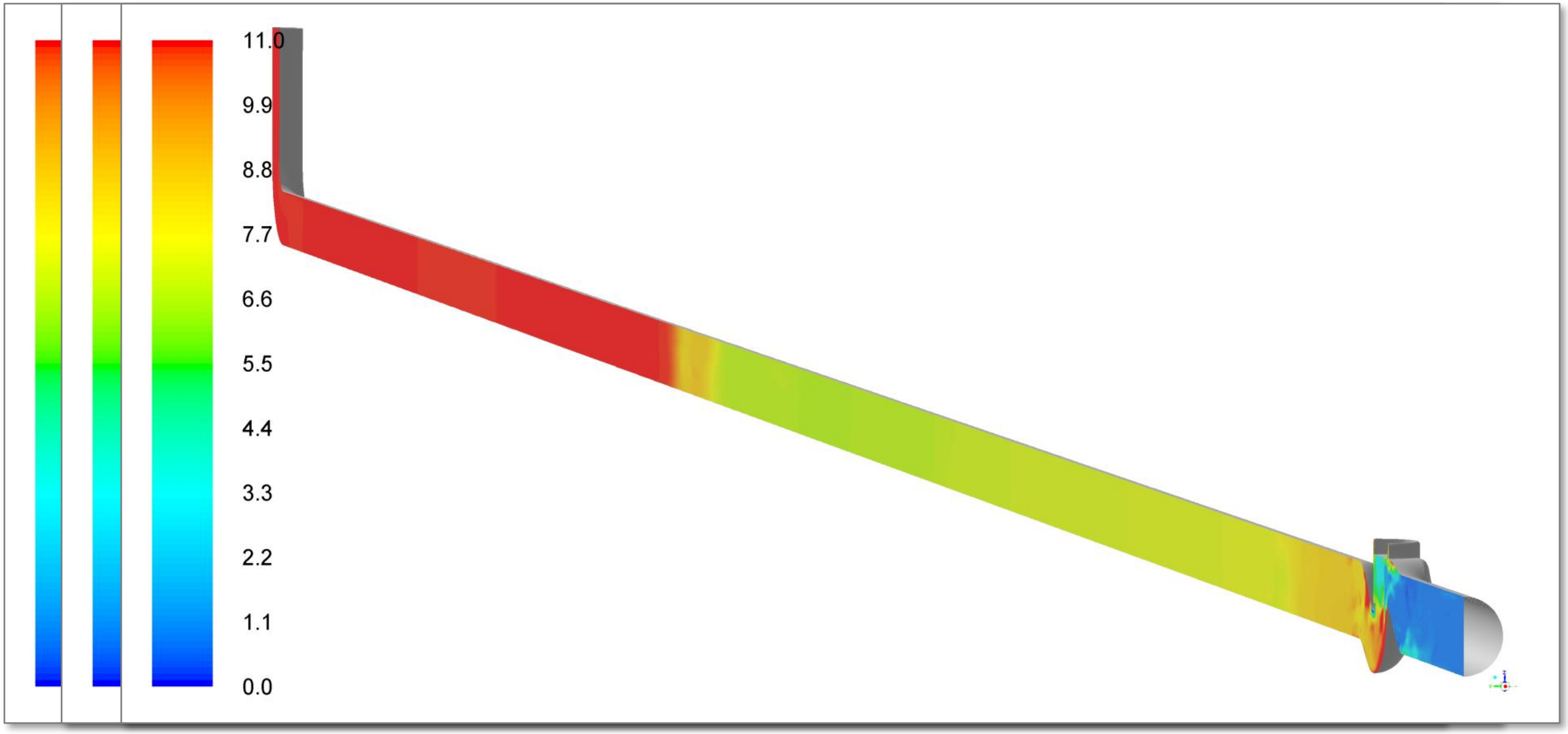
Numerical Results (4)

- **Observations (cont.):**
  - Soon, both slugs develop fully and the second slug starts entering the valve



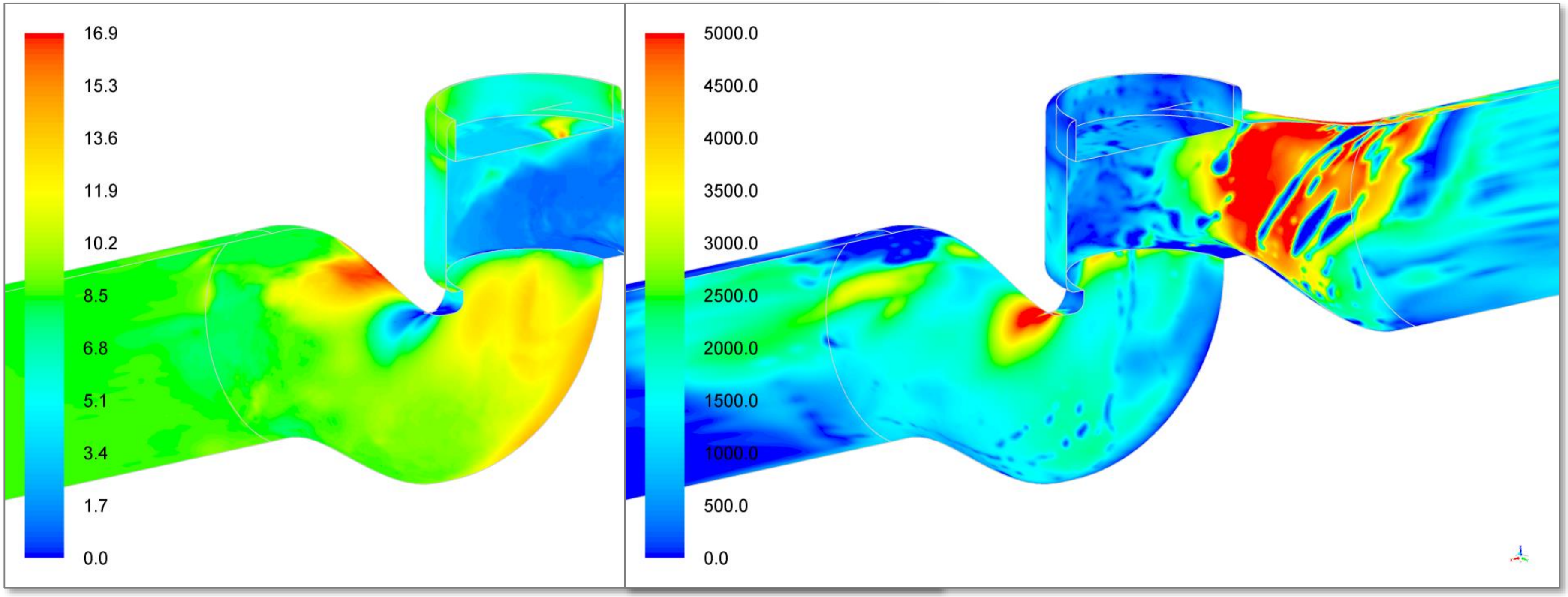
Numerical Results (5)

- **Observations (cont.):**
  - The distance between slugs, their velocity and the pressure drop across the individual slug seems to remain roughly constant over time



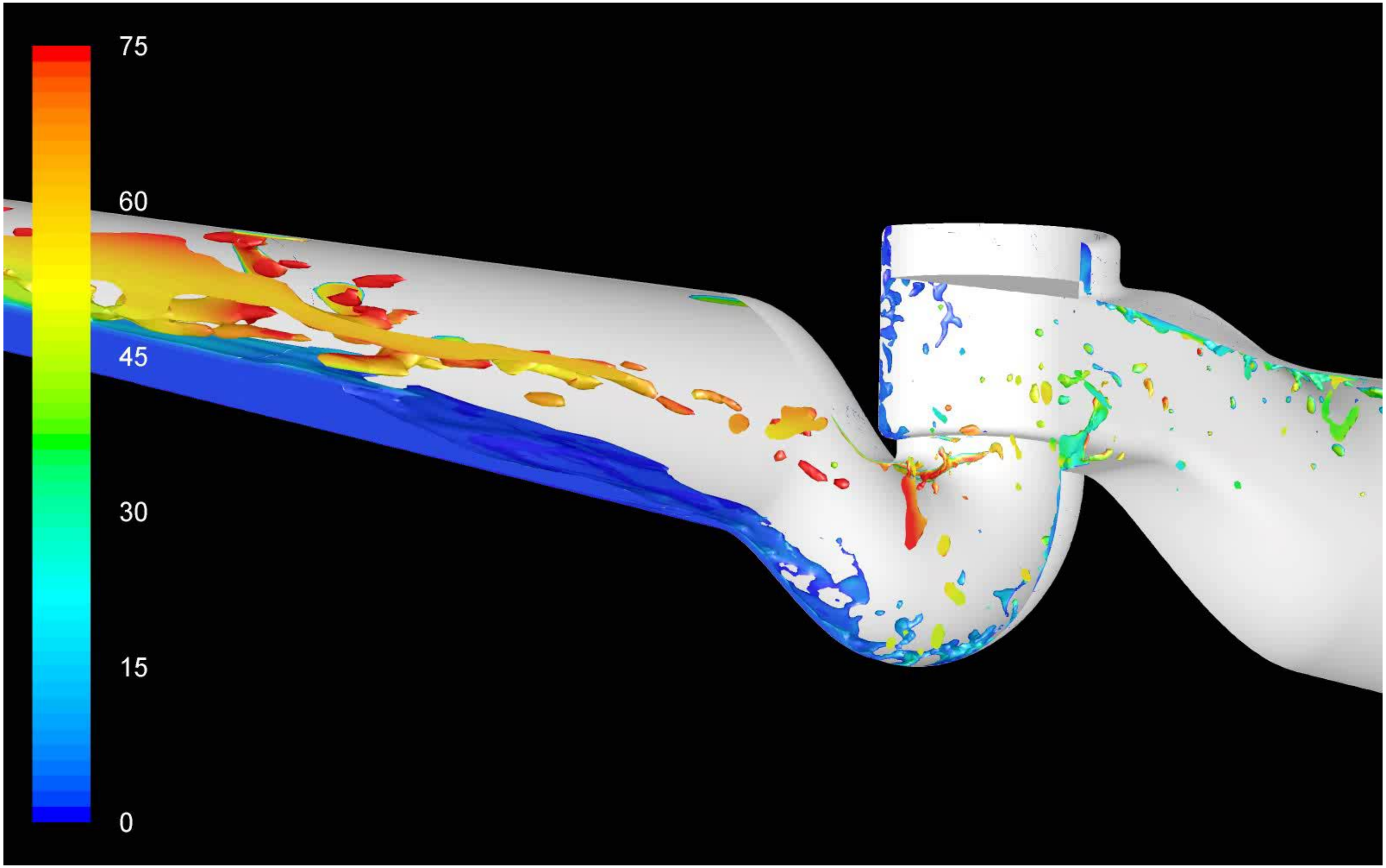
Numerical Results (6)

- **Observations (cont.):**
  - Local regions of very low static pressure are present, suggesting that a mass transfer mechanism should be implemented to capture condensate vaporization
  - Slug impact and break-up on valve walls causes high shear stresses (> 5000 Pa)



Numerical Results (7)

- **The variable time-step size disrupts the fluidity of the animation…**



Conclusions

- The simulation value is rather of a qualitative nature, as no experimental data (or any other source) was available for validation
- Nevertheless, it was only meant as a test designed to assess if in a specific practical situation the slug-flow regime was possible, for which “classic” methods simply could not have been applied
- Following this work, the inherent limitations of the VOF modeling approach would need to be addressed and evaluated by direct comparison to a second model using a Hybrid method