Best practices for the VOF model

On-line FLUENT Training

Outline

- Which problems can be solved with VOF model?
- Settings
  - Multiphase Model
  - Solver Settings:
    - Discretization and URF
    - ITA vs NITA
  - Boundary Conditions
    - open channel
  - Initial conditions: smooth interface
- Example setup: sloshing tank

Which problems can be solved with VOF model?

- VOF is exact multiphase model
  - Based on first principles
  - Solves one set of momentum equations for single “mixture” fluid
  - Turbulence and other models are the same as in single phase flows
  - Becomes impractical if length scale of the interface is comparable to cell length scale

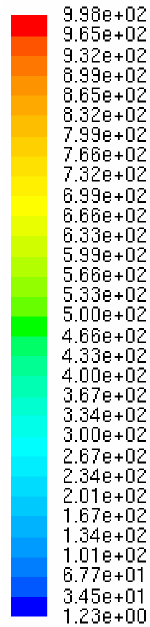
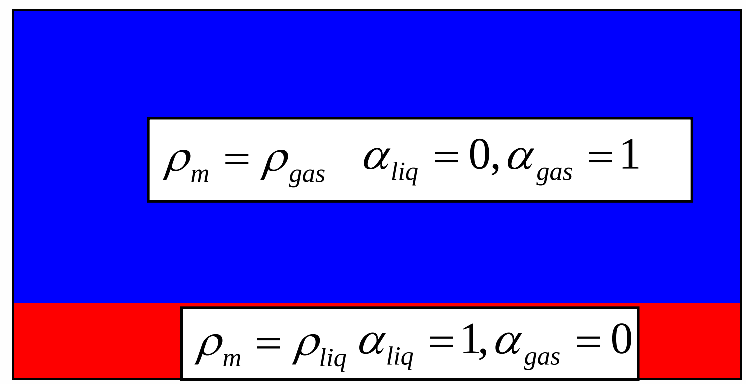


Illustration of mixture density concept

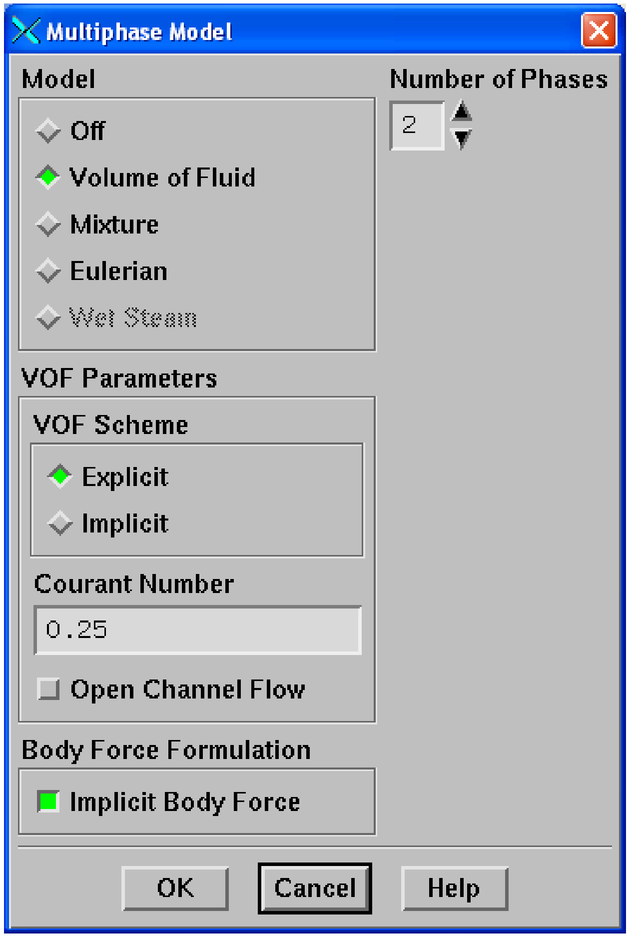


Density of the gas-liquid mixture fluid

$$\rho_m = \alpha_{liq}\rho_{liq} + \alpha_{gas}\rho_{gas}$$

Settings: Multiphase Model

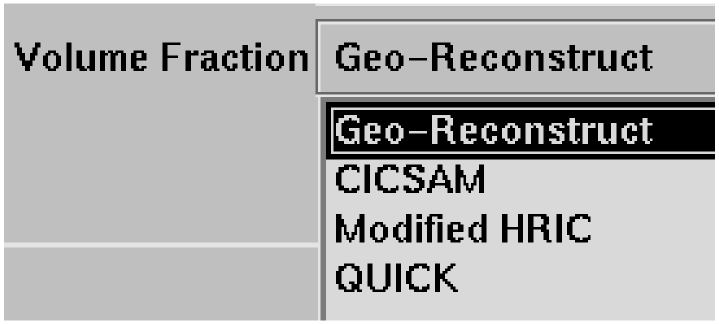
- Most free surface flows are unsteady
  - Explicit vs implicit means how mass conservation equation is discretized in time
  - Explicit method advantages
    - Small CPU effort per time step
    - Accurate interface tracking scheme ensures interface thickness is one cell thick
  - Explicit method disadvantages
    - Restriction in time step size
    - High grid quality requirement for default Geo scheme



Settings: Multiphase Model

- Explicit (cont.) – 4 discretization methods for VOF equation
  - **Geo** – default. Most accurate interface tracking without numerical diffusion.
  - Cases where **Geo-Reconstruct** may fail or require too small time steps include
    - Skewed or high aspect ratio mesh – create high quality mesh in regions of the domain where interface is expected or use **Explicit** scheme can be still used with **Modified HRIC** to minimize numerical diffusion
    - Very large velocity difference between phases (gas assisted atomization) – use implicit scheme with high resolution discretization for VOF (**Modified HRIC**)

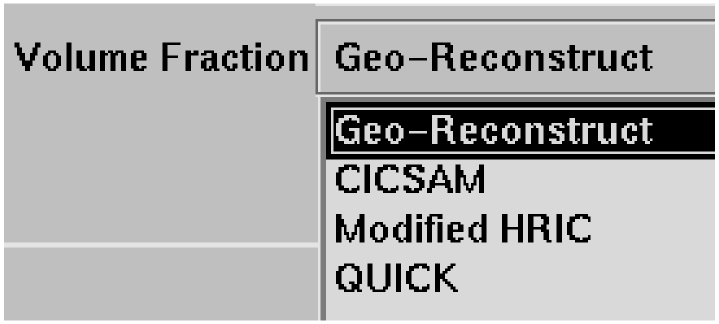
Solve…Controls…Solution



Settings: Multiphase Model

- Explicit (cont.) – 4 discretization methods for VOF equation
  - High viscosity ratio of phase material (molten plastics at low temperature)
  - As the alternative a **CICSAM** scheme can be used – it can handle high viscosity ratio or lower viscosity can be increased so viscosity ratio is below 1e-5
  - **QUICK** scheme is not recommended to use – **Modified HRIC** is preferred

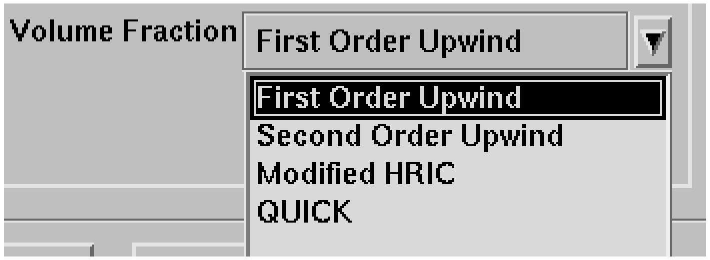
Solve…Controls…Solution



Settings: Multiphase Model

- Implicit – all discretization schemes except Modified HRIC will most likely produce significant numerical diffusion so HRIC should be used
- Implicit scheme also allows for steady state solver – however we recommend to solve even steady state problems with unsteady implicit scheme

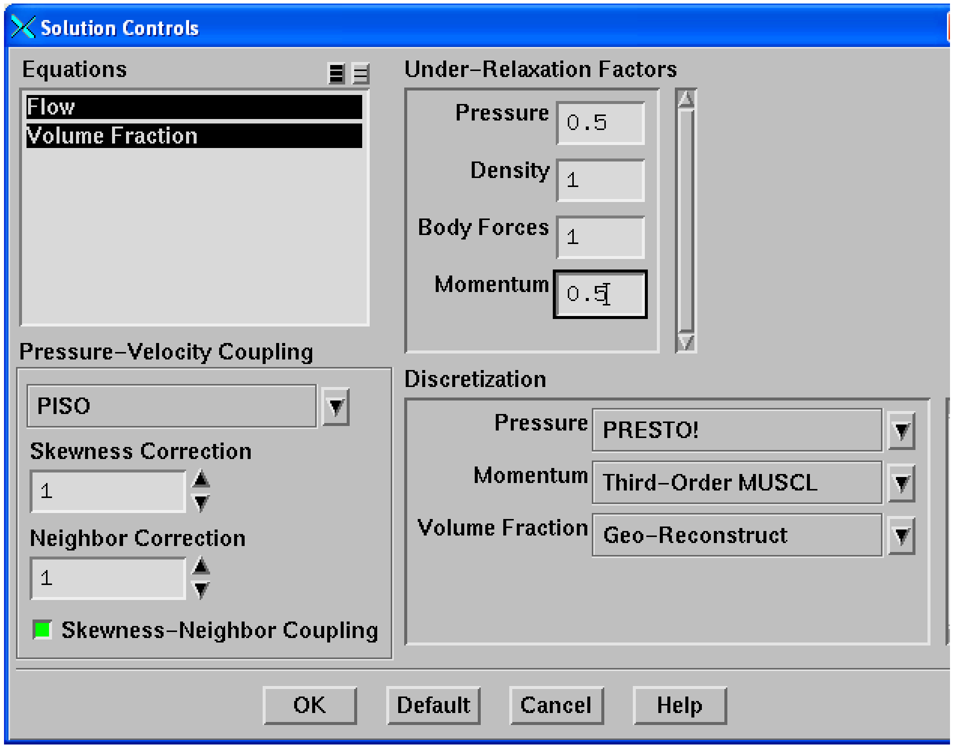
Solve…Controls…Solution



Solver Settings: Discretization and URF

- Default single phase settings should be corrected for both explicit and implicit solvers
- Untoggle **Skewness-Neighbor Coupling** if mesh is skewed
- For compressible VOF problems use **SIMPLE C** instead of **PISO**
- Momentum URF can be increased

Solve…Controls…Solution

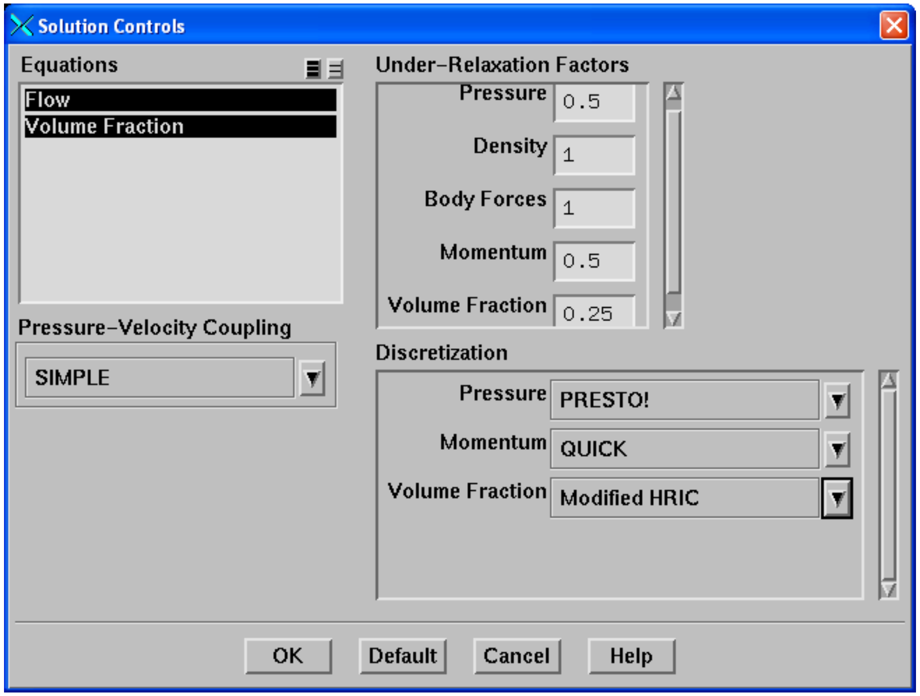


Conservative numerical settings for typical VOF model with good mesh

Solver Settings: Discretization and URF

- Implicit – default singe phase settings may not be correct
- **PRESTO** is recommended pressure discretization

Solve…Controls…Solution

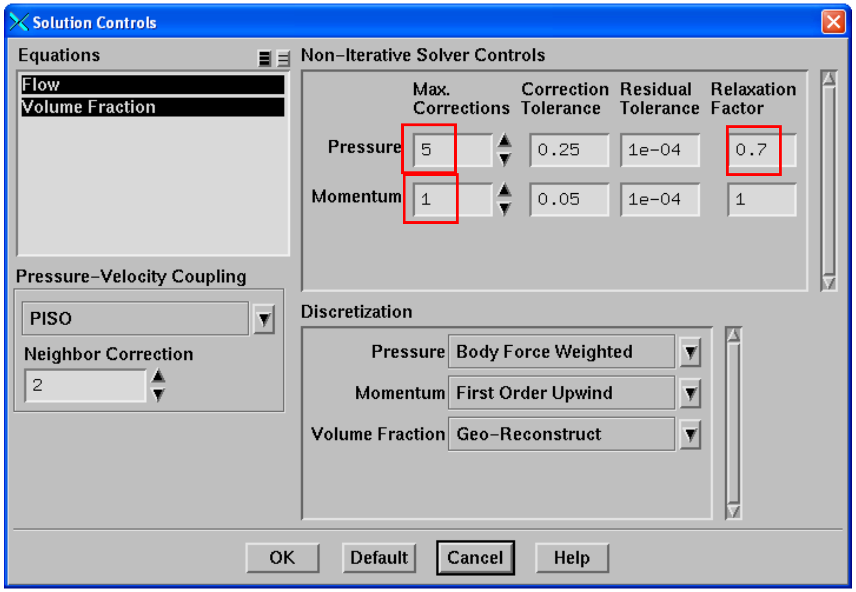


Conservative numerical settings for typical VOF model with good mesh

Solver Settings: ITA vs NITA

- **N**on **I**terative **T**ime **A**dvancement – reduces computational effort per time step in comparison with iterative schemes (SIMPLE, PISO)
- Can be used with VOF model however convergence problems may happen – use variable time step and reduce Max. corrections number for Momentum and reduce Relaxation Factor for pressure

Define…Models…Solver



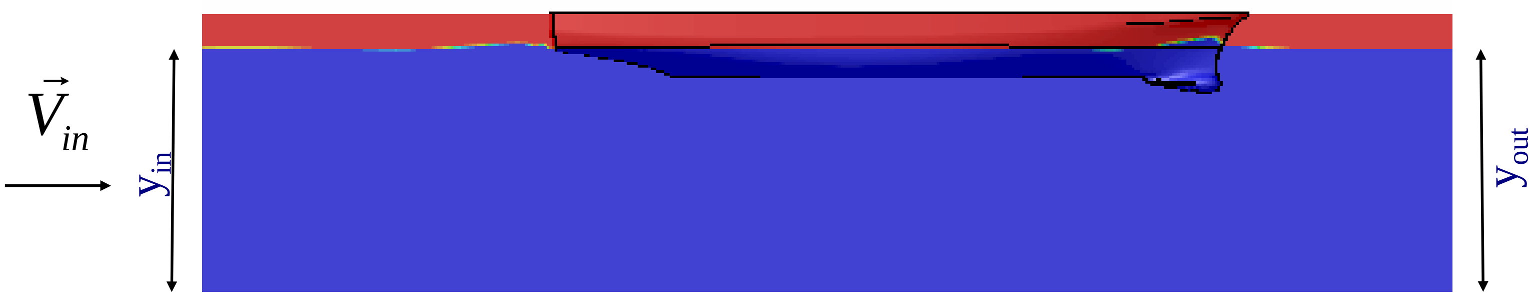
Conservative numerical settings for typical VOF model with good mesh

Boundary conditions

- Volume Fraction needs clear specification of interface location via initial and boundary conditions
  - Initial phase location is specified via marking cells in Adaptation and subsequent patching of Volume Fraction if needed
  - Inlet or outlet BC must be setup such that only one phase can enter or exit domain
    - For velocity inlet, the incoming Volume Fraction of one phase must be 1 and all others must be zero
    - For mass flow inlet, the mass flow rate of only one phase can have a non-zero value
    - For pressure outlet, the back flow value of Volume Fraction of only one phase must be 1, all other Volume Fractions must be zero
    - Outflow BC must never be used in any multiphase model including VOF

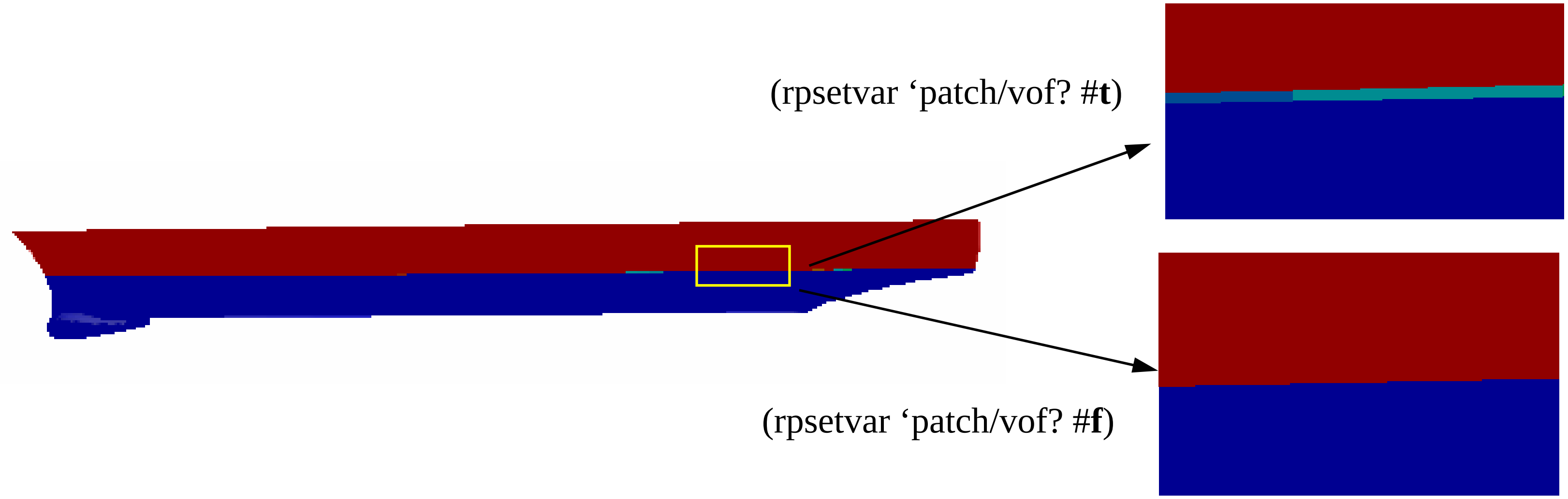
Boundary conditions: Open channel

- Applicable to flows where inertia and gravity are dominant with known depths of the liquid at the inlets or outlets
  - Example – destroyer moving through the sea at depth y_{in} at speed V_{in}
  - Prescribe y_{in} and V_{in} at inlet and y_{out} at the outlet
  - If $Fr = V_{in}*(g\ y_{in})^{-0.5} > 1$, then choice of y_{out} does not matter – flow supercritical, if $Fr<1$, then choice of y_{out} influence solution (subcritical flow)
- One can specify either y_{in} and V_{in} (convection driven) or height difference (gravity driven) between y_{in} and y_{out}



Initial conditions: smooth interface

- Default initialization of interface through patching creates sharp interface – often leads to divergence
- Smoothing interface can be accomplished by typing the following TUI command before initialization (rpsetvar ‘patch/vof? #t)
- To return back to default sharp initialization type (rpsetvar ‘patch/vof? #f)
- Works only per one initialization register created in ADAPTATION panel

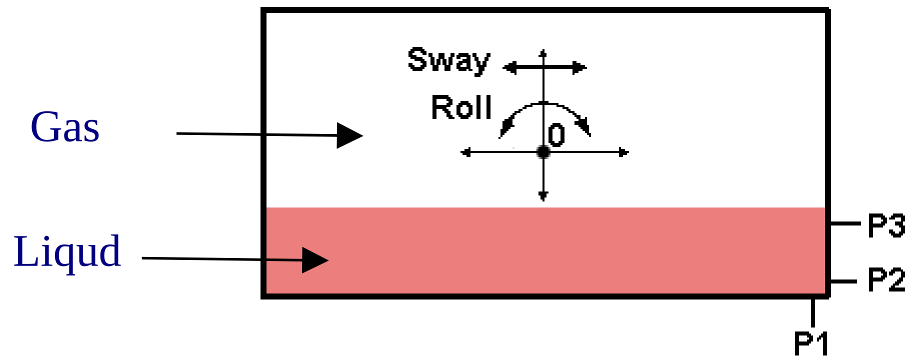


Example setup: sloshing in the tank

- Swaying motion of the partially filled tank is approximated as non-stationary body force using User Defined Function (UDF)

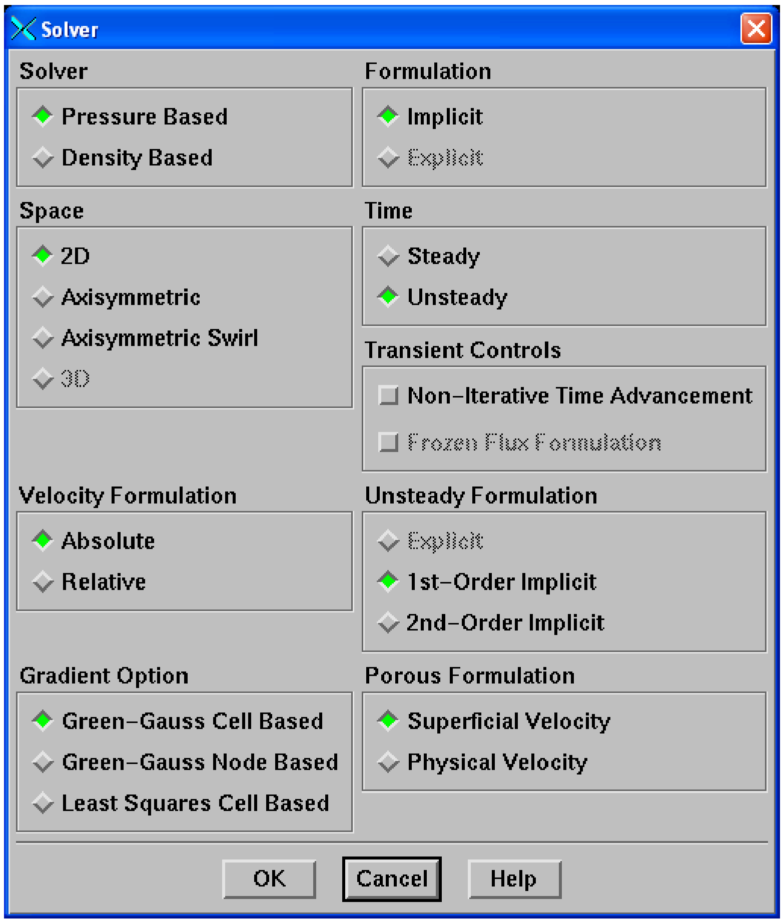
$$F_{sway} = a\rho_m \omega^2 \sin(\omega t)$$

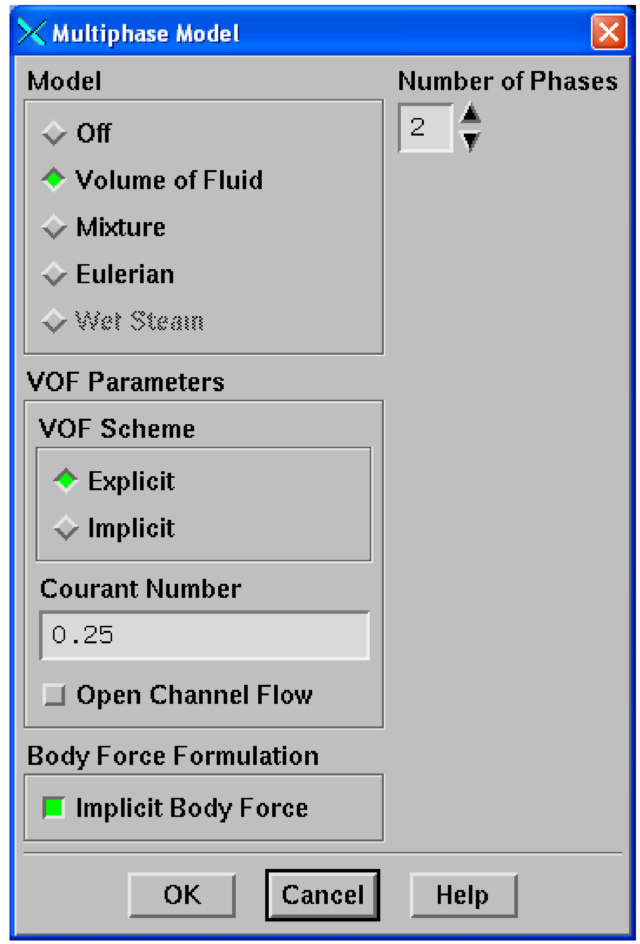
- UDF uses DEFINE_SOURCE Macro to prescribe force per unit of volume in horizontal component of momentum equation



Example setup: sloshing in the tank

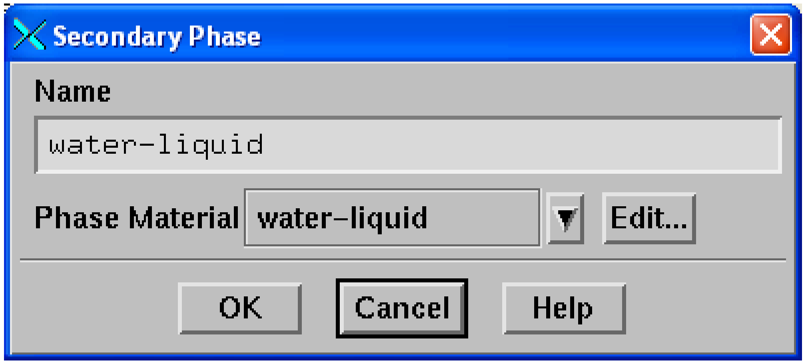
- VOF model and solver definition





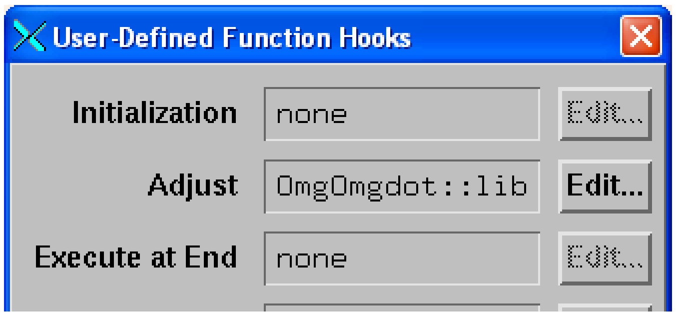
Example setup: sloshing in the tank

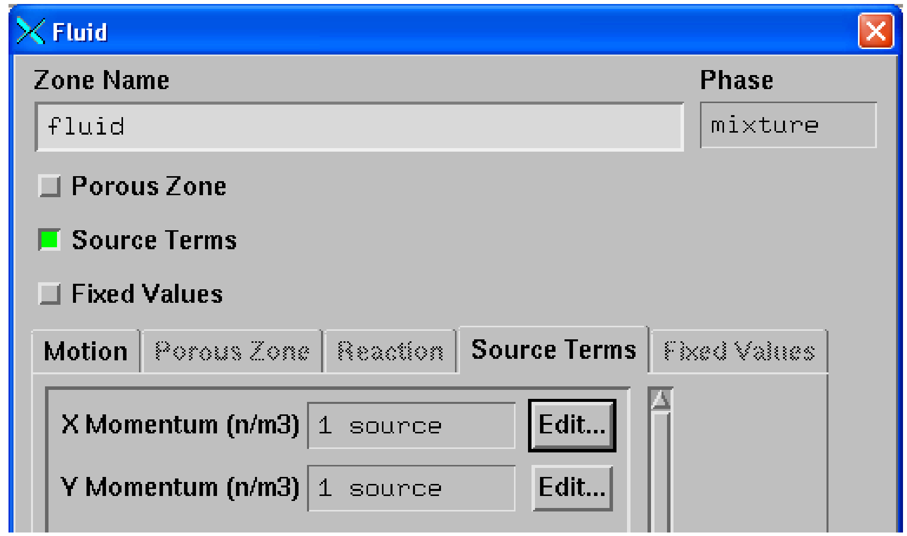
- Definition of phase materials and phases

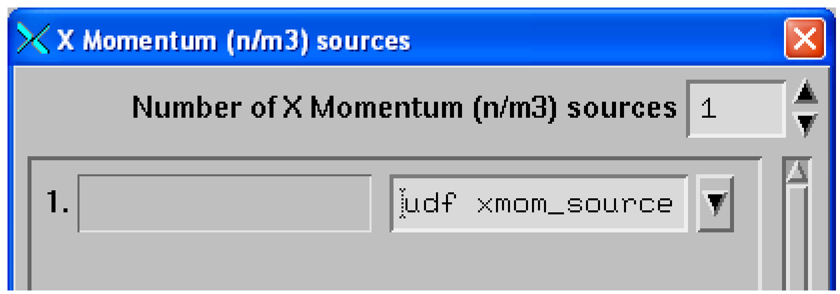


Example setup: sloshing in the tank

- UDF hookup

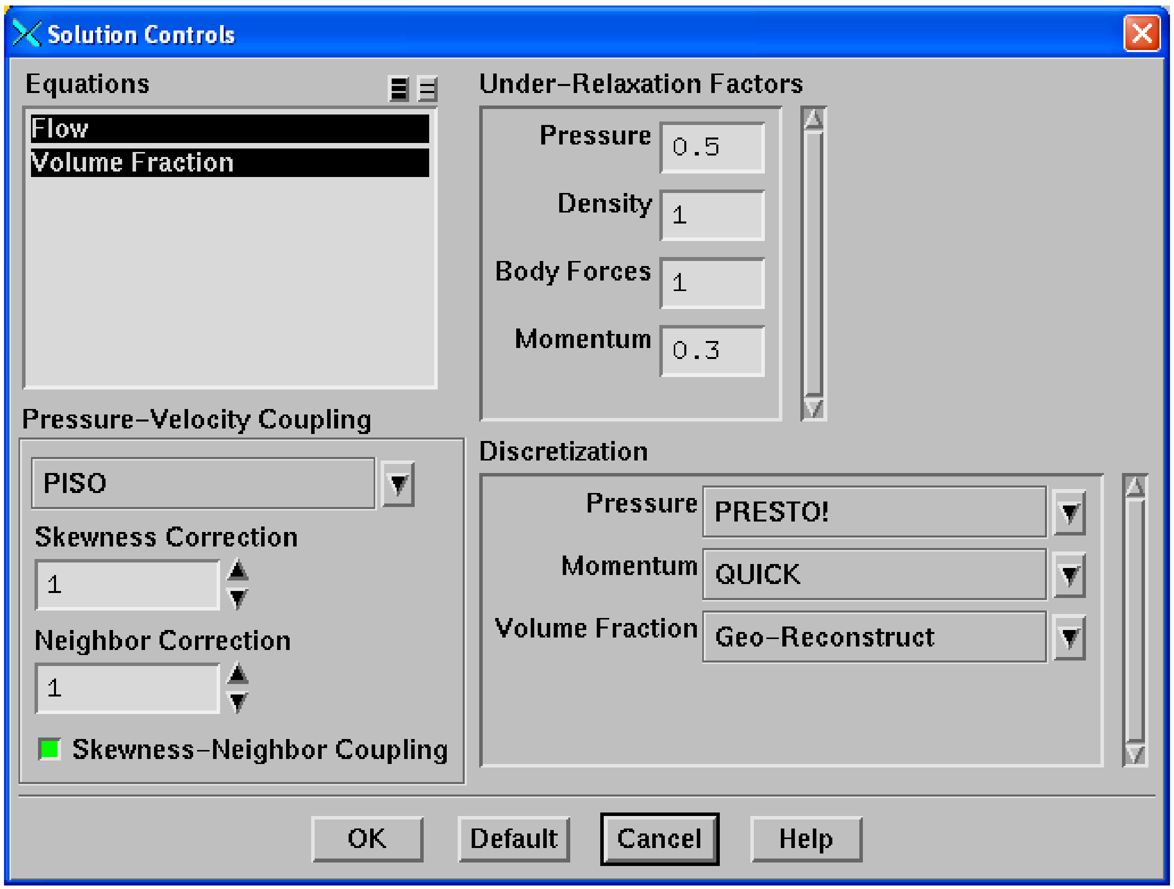






Example setup: sloshing in the tank

- Numerical settings



Example setup: sloshing in the tank

- Choice of time step – order of magnitude estimation
  - Two velocity scales are present
    - gravitational velocity scale

$$\frac{d}{dx}\left(\frac{\rho U_g^2}{2}\right) = \rho g \Rightarrow U_g = \sqrt{2g\Delta x} \cong 3m/s$$

    - oscillation velocity scale

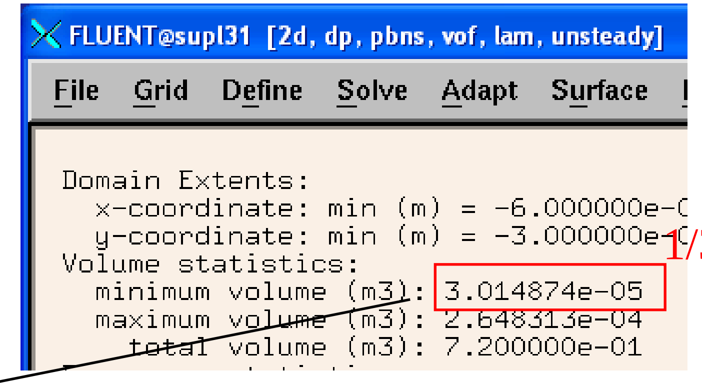
$$U_o = a\frac{\omega}{2\pi} = 0.031m/s$$

    - cell length scale $l = 0.03m$
    - for inertia dominated flows Courant number = 1 so

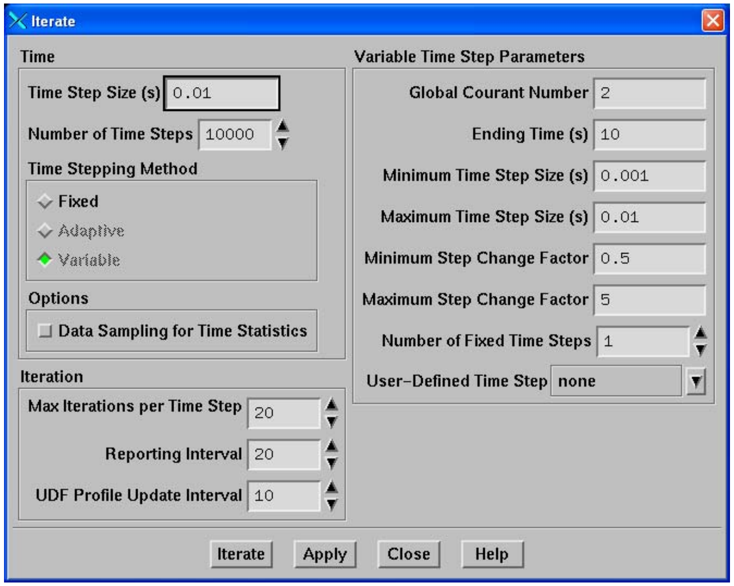
$$\Delta t = \frac{l}{MAX\left(U_g, U_O\right)} \cong 0.01\text{sec}$$

    - Variable time step was chosen where maximum time step was 0.01 sec

Grid…Check



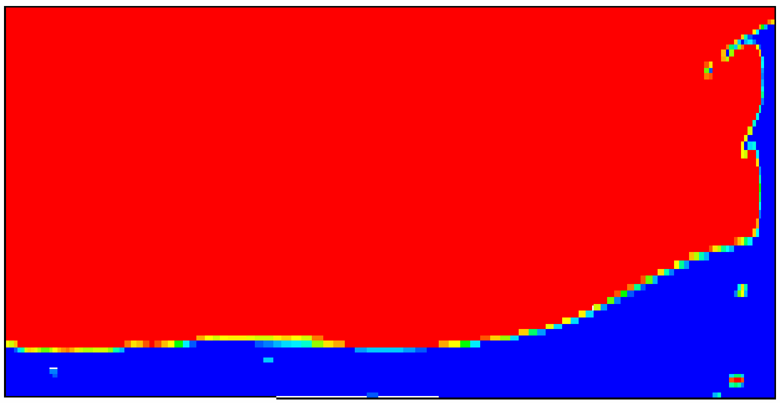
1/3



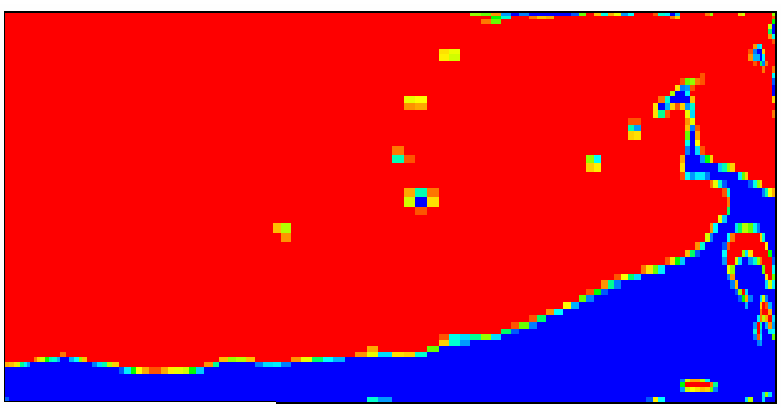
Example setup: sloshing in the tank

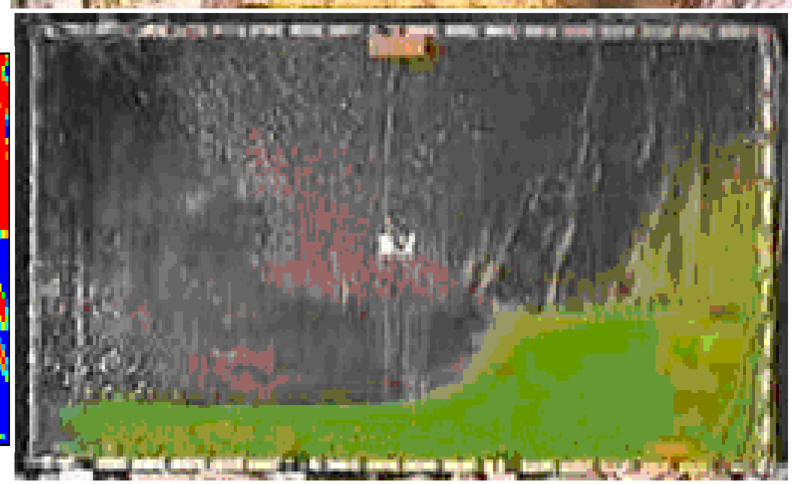
- Results – VOF at different oscillation periods $T = \dfrac{2\pi\omega}{\omega} = 1.94\,\text{sec}$

$t = 2T$



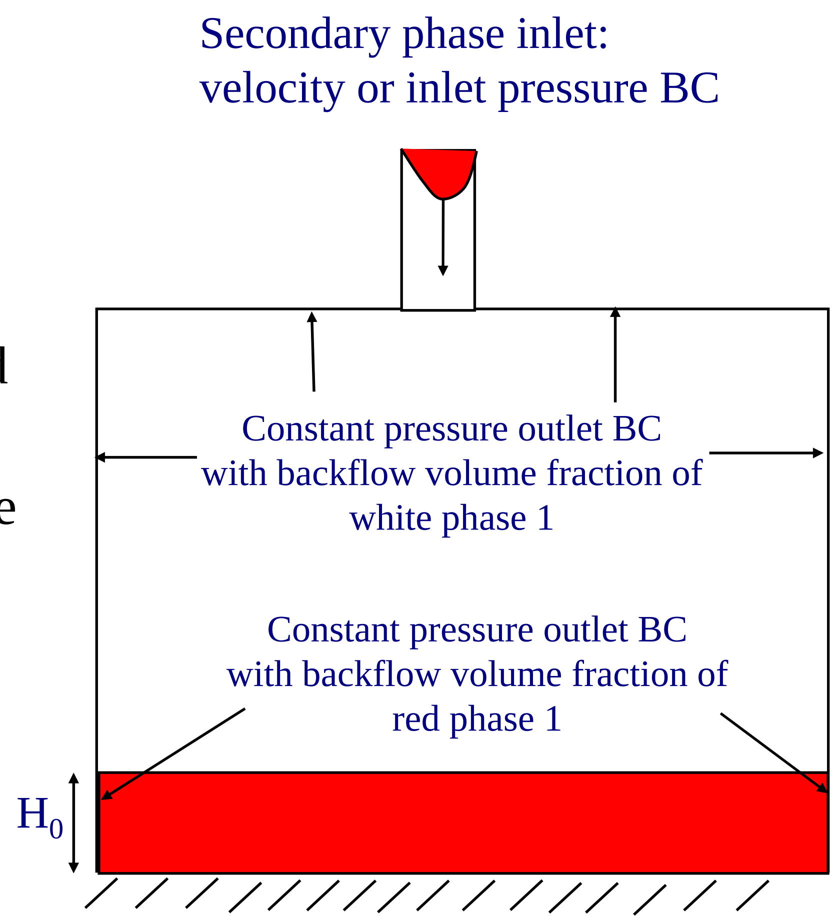
$t = 4T$





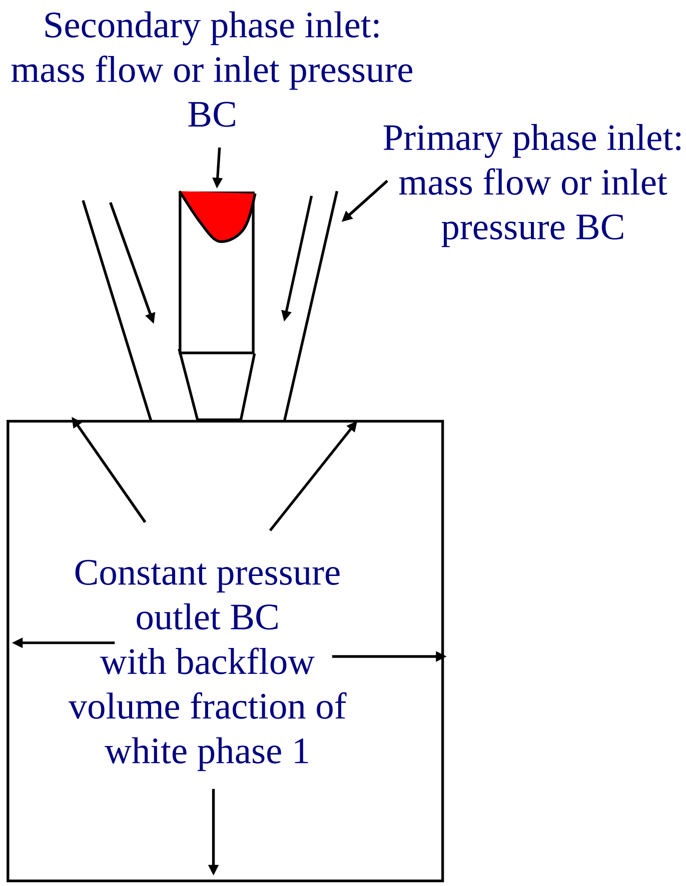
Example setup: injection generic problem

- Explicit scheme with PISO algorithm and Geo scheme
- PRESTO discretization for pressure
- Time step size from the cell length and inlet velocity of red phase
- Surface tension can be turned on if deemed important
- Velocity inlet includes velocity and volume fraction 1 of red phase
- If turbulence and non-isothermal case - make sure that back flow turbulence and temperature values are reasonable



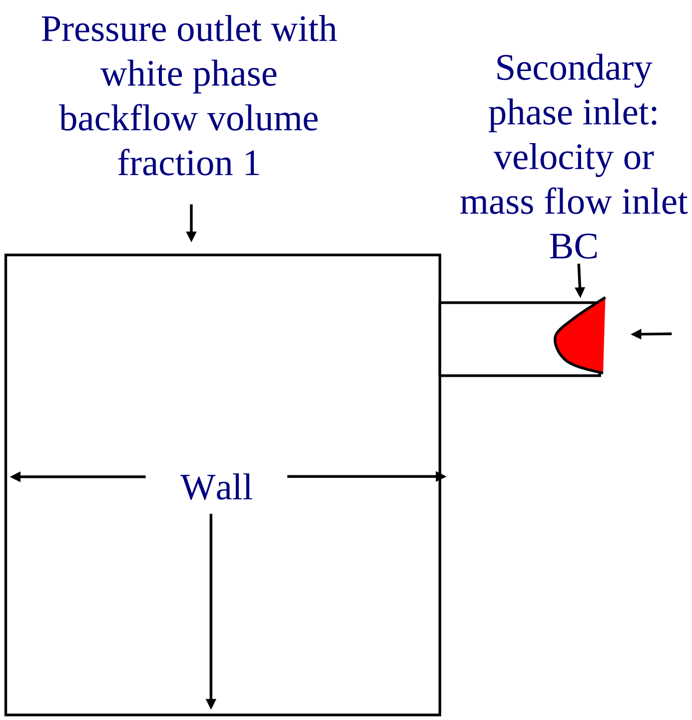
Example setup: injection with compressibility

- White (primary phase) – compressible gas,
- Red – secondary phase with constant density and non-Newtonian
- Implicit scheme with HRIC discretization for volume fraction, PRESTO or STANDARD discretization for pressure
- PISO or SIMPLE pressure-velocity coupling
- Time step from cell length scale and red phase velocity scale
- Turn off solution of energy equation for first 10 time steps of temperature jumps are seen



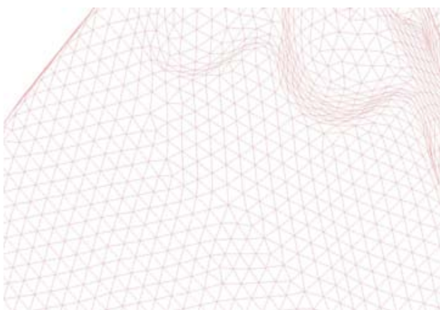
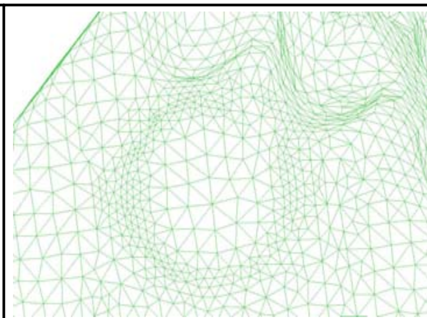
Example setup: tank filling application

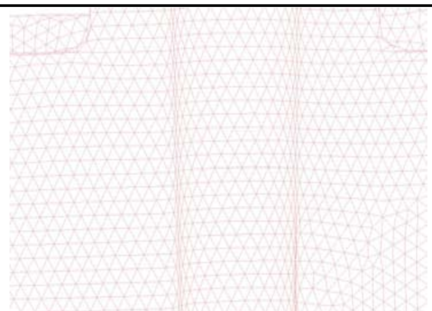
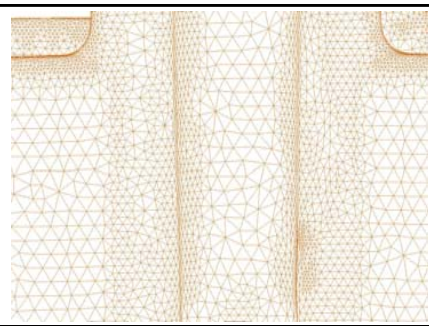
- White (primary phase) – incompressible gas,
- Red – liquid secondary phase with constant density
- Create a uniform mesh and modify the mesh only in regions where mesh refinement is absolutely necessary
- Extrude 1-2 prism layers at the filling inlet. These cells will be part of a separate fluid volume, which is patched during initialization with a volume fraction of 1 (i.e. these cells are filled with filling fluid)

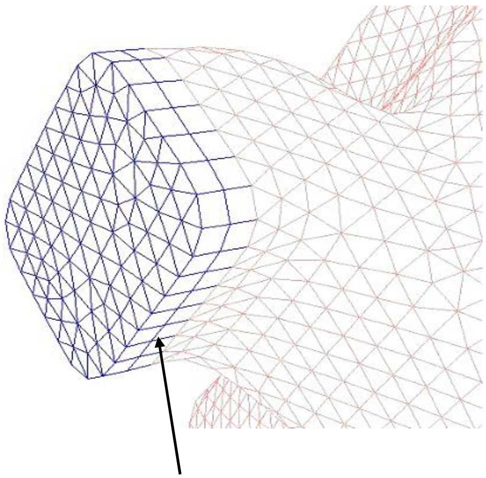


Example setup: tank filling application (ctd.)

- Examples of good and bad mesh practices in real geometry and example if inlet meshing

	
Good	Bad

	
Good	Bad



Inlet prism layer to be filled by secondary phase at initialization

Example setup: tank filling application (ctd.)

- **Geo** reconstruct scheme should be used together with **PISO**
  - 0 iterations for **Skewness Correction**
  - 1 iteration for **Neighbor Correction**
  - Turn **ON Skewness-Neighbor Coupling** option
- The under relaxation factors should have the following values;
  - Pressure 0.7
  - Momentum 0.8
  - Density 1
  - Body Forces 1
- If **NITA** is used set Pressure under relaxation factor to 0.7
- Time step size is usually determined by inlet velocity and mesh scale

Additional training resources

- Several tutorial are available on the Learning CFD center

  http://www.learningcfd.com/login/index.htm

  - Choose a Course
    - FLUENT
      - **Intermediate Tutorials**
        - **Tutorials with a Focus on Multiphase Flow Applications**
      - **Advanced Tutorials**