Motivation

- **Multiphase flow** comes into the picture when argon gas is injected through **UTN** or **stopper rod tip**.

- **Bubble size distribution** is important:
  - Flow pattern is affected by bubbles.
  - Small bubbles could be **captured** on solidified shell.

- Computational model is a valuable tool to understand the phenomenon.
Overall classification of Two phase flow: Continuum

Quasi – multi phase methods

Algebraic-Slip mixture model (Thomas et al., 1994)
Treat bubbles as species that is diffused into continuous phase
\[ C \times 1 + NS \times 1 + SD \times 1 \]

Mixture model (Fluent manual)
Track secondary phase through VF equation and use weighted averages for material & fluid properties (\(=\) mixture property)
\[ C \times 1 + NS \times 1 + VF \times 1 \]

Multi-fluid methods

Eulerian-Eulerian model (Fluent manual)
Treat both fluids as continuous phase
\[ C \times 2 + NS \times 2 \]

Population balance model
- Homogeneous MUSIG (Lo, 1996)
Coalescence and breakup between bubbles are considered by solving Boltzmann equations
\[ C \times (\text{number of bubble sizes} + 1) + NS \times 2 \]
- Inhomogeneous MUSIG (Krepper, 2007)
Allow to have different velocity fields between bubbles
\[ C \times (\text{number of bubble sizes} + 1) \times \text{number of velocity groups}) + NS \times (\text{number of velocity groups}+1) \]

Overall classification of Two phase flow: Discrete

Particle based methods

Discrete phase model (DPM) (Hoomans et al., 1998)
Treat liquid as continuum, but bubbles as particles and track all by Newton’s equation of motion
\[ C \times 1 + NS \times 1 + (N+P) \times (\text{number of bubbles and/or inclusions}) \]

Smooth particle hydrodynamics (SPH)
(Lucy, 1977)
Hybrid model between continuum model and discrete model: solve continuum PDEs for both phases using discrete particles by substituting the spatial derivatives to interpolation functions of neighbor particles.
\[ \text{(C+1)xNS+1+x}\times (\text{number of fluid particles})\times 2\text{phase} \]

Lattice-Boltzmann method (LBM)
(Shan & Chen, 1993)
Solve Boltzmann equations for fluid particle distribution functions of each phase.
\[ B \times 2\text{phase} \]

Dissipative Particle Dynamics (DPD)
(Groot and Warren, 1997)
Track each fluid particles (a particle = a group of molecules) using Newton’s equation of motion.
\[ (P+N)\times1\times\text{number of particles} \times 2\text{phase} \]

Interface capture / tracking methods

Interface capture
- Moving grid method (Liu et al., 2014)
- Moving a single grid line to match interface (Muzafarija & Peric, 1997)
- Moving other grids for good mesh quality (Fluent manual)
- SPINE method (FIDAP manual)

Interface tracking
- Marker and cell (MAC) (Harlow et al., 1965)
Add massless particles as markers on secondary fluid and track them
\[ C \times 1 + NS \times 1 + P \times \text{number of markers} \]
- Surface marker (Chen, 1991)
Add massless particles as markers on interface and track them
\[ C \times 1 + NS \times 1 + P \times \text{number of markers} \]
- Volume of Fluid (Hirt & Nichols, 1981)
An Interface is defined as boundary of volume fraction between 0 and 1
\[ C \times 1 + NS \times 1 + VF \times 1 \]
- Level set method (Osher & Sethian, 1988)
Define interface as interface function and track it through transport equation
\[ C \times 1 + NS \times 1 + I \times 1 \]

Molecular Dynamics (MD) (Alder and Wainwright, 1959)
Track all molecules of each fluid using Newton’s equation of motion
\[ ((P+N)\times1\times\text{number of particles}) \times 2\text{phase} \]
Objectives

- Test **several multiphase models** by benchmarking Dresden experiment (Timmel et al., 2014).
  - 1D pressure energy model (analytical model)
  - Single phase model
  - Eulerian Eulerian model
  - VOF model

- Compare the numerical results with experiment data.
  - Pressure distribution in nozzle
  - Gas pocket shape

- Check pros and cons of each method.

Video from Dresden

Geometry of Dresden experiment

- Geometry (Timmel et al., 2014)

![Blueprint of Dresden experiment geometry](image-url)
Operating condition

<table>
<thead>
<tr>
<th>Operating condition</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operating temperature $T_{op}$</td>
<td>room temperature 293 K</td>
</tr>
<tr>
<td>Stopper rod position</td>
<td>9.5 mm</td>
</tr>
<tr>
<td>Tundish level</td>
<td>70 mm</td>
</tr>
<tr>
<td>Galinstan flow rate $Q_I$</td>
<td>115 cm$^3$/s</td>
</tr>
<tr>
<td>Argon gas flow rate $Q_{gas}$</td>
<td>1.7 cm$^3$/s</td>
</tr>
<tr>
<td>Submergence depth $h_{sub}$</td>
<td>92 mm</td>
</tr>
<tr>
<td>Wall roughness</td>
<td>Smooth wall (acrylic)</td>
</tr>
<tr>
<td>Gas volume fraction $\alpha$</td>
<td>1.4 %</td>
</tr>
</tbody>
</table>

Gas volume fraction calculation (Thomas et al., 1994)

$$\alpha = \frac{Q_{gas}H_f}{Q_{gas} + Q_I} \leq 1.4\%$$

Material property

<table>
<thead>
<tr>
<th>Material property</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Galinstan density $\rho_l$</td>
<td>6440 kg/s (~92% of liquid steel)</td>
</tr>
<tr>
<td>Galinstan viscosity $\mu_l$</td>
<td>0.0024 Pas (~40% of liquid steel)</td>
</tr>
<tr>
<td>Galinstan surface tension</td>
<td>0.718 N/m (~58% of liquid steel)</td>
</tr>
<tr>
<td>Contact angle</td>
<td>120 deg (non-wetting) (~80% of liquid steel)</td>
</tr>
<tr>
<td>Argon gas density $\rho_g$</td>
<td>1.6228 kg/s</td>
</tr>
<tr>
<td>Argon gas viscosity $\mu_g$</td>
<td>2.125x $10^{-5}$ Pas</td>
</tr>
</tbody>
</table>

Ref.: Geratherm Medical AG manual (2002)
Karcher et al. (2003)

1. 1D pressure energy model: Pressure distribution

$P_e$ : gage pressure at the point $x$
$\rho_l$ : Galinstan density
$A_{SEN}$ : cross-section area of SEN
$V_{SEN}$ : velocity in SEN
$V_{gap}$ : velocity in stopper rod gap
$z_{ab}$ : distance between two points
$\rho_{elbow}$ : minor loss constant = 0.5
$C_{stopper}$ : stopper rod constant
$H_{SEN}$ : stopper rod position
$Q_l$ : liquid flow rate
$P_{Leiow}$ : pressure loss by elbow
$C_{stopper}$ : stopper rod constant
$H_{SEN}$ : stopper rod position

References:
- Geratherm Medical AG manual (2002)
- Karcher et al. (2003)
- White, 2011
- Liu et al., 2014
2. Single phase flow: Numerical setup

- **Boundary conditions**
  - Galinstan Inlet: mass flow rate BC
  - Galinstan: $m_{in} = 0.7406 \text{ kg/s}$
  - Outlet: constant pressure BC
  - $P = \rho_{G}g\rho_{in} = 5819 \text{ Pa}$
  - Wall: no slip BC + Smooth wall

- **Turbulence model**:
  - Standard $k-\varepsilon$ model
  - The law of the wall for boundary layers

- **Mesh**:
  - 60,000 cells (cell size: ~2mm)

- **Numerical scheme**:
  - Second order Upwind
  - Steady state simulation

---

Fig 4. Boundary conditions of single phase flow simulation

---

2. Single phase flow: Numerical simulation result

- **Pressure [Pa]**
  - YZ Center plane
  - XZ Center plane
  - Velocity [m/s]
    - YZ Center plane
    - XZ Center plane

- **Axial pressure distribution from Tundish to SEN**
  - Height above mold level (mm)
  - Pressure [kPa]

---

Fig 5. Velocity, pressure field and axial pressure distribution of single phase flow model result
2. Single phase flow: Comments

- Three recirculation zones are shown near SEN inlet:
  - Stopper tip, both side walls of SEN inlet.
  - Location matches to gas pocket positions in Dresden experiment.

- Recirculation zone at port is small due to short port length (3mm).

- As expected in 1D pressure energy model, sudden pressure drop happens at SEN inlet by stopper rod.
  - Minimum pressure happens at SEN inlet wall
  - Easiest place for gas accumulation.

3. Eulerian Eulerian model: Numerical setup

- **Boundary conditions**
  - Galinstan Inlet: mass flow rate BC
  - Galinstan: $m_s = 0.7406 \text{ kg/s}$
  - Argon gas Inlet: mass flow rate BC
  - Argon gas: $m_a = 2.7588 \times 10^{-6} \text{ kg/s}$
  - Outlet: constant pressure BC
  - $P = \rho g h_{sab} = 5810 \text{ Pa}$
  - Wall: no slip BC + Smooth wall

- **Two phase model**:
  - Eulerian Eulerian model is used.
  - Bubble size: $d_{bubble} = 3 \text{ mm}$
  - Drag force: Schiller-Naumann model

- **Turbulence model**:
  - Standard $k-\varepsilon$ model for both phase
  - The law of the wall for boundary layers

- **Mesh**:
  - 60,000 cells (cell size: ~2mm)

- **Numerical scheme**:
  - Transient simulation (URANS)
  - Second order Upwind

Figure 6. Boundary conditions of Eulerian Eulerian model simulation
3. Eulerian Eulerian model: Numerical simulation result

Figure from Timmel et al. (2014) Fig 4(a)
3. Eulerian Eulerian model: Comments

- Eulerian Eulerian two phase model with $k-\varepsilon$ turbulence model is able to capture three gas pockets (stopper tip, both SEN inlet side walls).

- Gas pocket size is determined by recirculation zone size and gas flow rate.
  - Deeper stopper rod position increases recirculation zones (more separation) → bigger gas pocket at stopper tip, thicker and shorter gas pockets at side walls (Timmel et al, 2014)

- Faster than VOF: efficient method if bubble size information is not necessary.

- Cannot resolve small bubble interface: no help to understand bubble size distribution.

---

Pressure distribution comparison

- Two phase flow requires higher tundish level.
- Analytical model results roughly match to single phase and Eulerian Eulerian model.

Fig 9. Comparison of axial pressure distribution from 1D analytical model, single phase model and Eulerian Eulerian model.
4. VOF model
Computational domain

Geometry is slightly different to the Eulerian-Eulerian model case: this geometry is estimated from picture on the paper (Timmel et al., 2014) before getting answer from Dresden. (Especially, 57mm deeper submergence depth)

4. VOF model: Numerical setup

- **Boundary conditions**
  - Galinstan Inlet: mass flow rate BC
  - Galinstan: $\dot{m}_L = 0.7406 \text{ kg/s}$
  - Argon gas Inlet: mass flow rate BC
  - Argon gas: $\dot{m}_g = 2.7588 \times 10^{-6} \text{ kg/s}$
  - Wall: no slip BC + Smooth wall
  - Outlet: outflow BC

- **Two phase model**
  - VOF model is used.
  - Surface tension is included. (continuous surface force model)
  - Explicit + Geometric reconstruction scheme

- **Turbulence model**
  - Filtered URANS (SAS model) is used.

- **Mesh**
  - 1 million cells (cell size: ~1mm)
  - Mesh refinement near SEN inlet

- **Transient simulation**
  - Time step: $10^{-5}$ second
4. VOF model: Numerical simulation result

Fig 11. Velocity, pressure field and axial pressure distribution of VOF model result

University of Illinois at Urbana-Champaign • Metals Processing Simulation Lab • Hyunjin Yang • 19
4. VOF model: Numerical simulation result

3D view

Bubble size: 1-3 mm

Get bigger as it goes down

Projection view from the front

3D view

magnified

Time: 0.31 sec.
4. VOF model: Comments

- VOF two phase model with filtered URANS turbulence model is able to **capture bubble interfaces** in turbulence. (with explicit + geometric reconstruction scheme)

- It shows detachment of small bubbles from gas pocket at stopper tip.

- Requires **finer mesh** (smaller than bubbles) to resolve exact interface shape, and **small time step** to keep Courant number ~1. (current mesh is **not enough** to clearly capture the small bubbles)

- **More calculation time** is required to observe gas pockets at SEN inlet side walls.
  - Gas is filled from stopper tip (in thickness direction), and then expand to width direction → gas captured in recirculation zones at SEN side walls

- Outflow BC is used since constant pressure BC causes instability when bubbles cross the BC.

Conclusions

- **Pressure distribution** of Single phase and Eulerian Eulerian model **match** to1D pressure energy model result.

- **Eulerian Eulerian** model **captures three gas pockets**, but not small bubbles.

- **VOF** model is promising method to **figure out bubble size distribution**.
  - Able to **capture bubble detachment** from gas pockets. (explicit + geometric reconstruction schemes are used for clear interface)

  - High computational cost is required due to fine mesh (smaller than bubbles) & small time step (to keep Courant number ~1).
Acknowledgments

- Continuous Casting Consortium Members (ABB, AK Steel, ArcelorMittal, Baosteel, JFE Steel Corp., Magnesita Refractories, Nippon Steel and Sumitomo Metal Corp., Nucor Steel, Positech/ Posco, SSAB, ANSYS/ Fluent)

- Special thanks to Klaus Timmel for the specific geometry and operating conditions

References


5. FLUENT ANSYS Inc. v14.4-Manual (Lebanon, NH)


10. FIDAP manual (Fluid dynamics international, 1998-1999)


