Multi-Phase Flow: Direct Numerical Simulation

Igor Bolotnov
North Carolina State University

Gretar Tryggvason
University of Notre Dame

July 8-10, 2013
Multi-Phase Flow: Direct Numerical Simulation

Igor Bolotnov – North Carolina State University
Gretar Tryggvason – University of Notre Dame
OUTLINE:

• introduction
• types of multiphase flows
• Interface tracking / direct numerical simulation
• the Navier-Stokes equations
• Microscale modeling
• CASL applications
Multi-Phase Flow: Direct Numerical Simulation

Multiphase flows are everywhere:
   Rain, air/ocean interactions, combustion of liquid fuels, boiling in power plants, refrigeration, blood,

Research into multiphase flows usually driven by “big” needs
   Early Steam Generation
   Nuclear Power
   Space Exploration
   Oil Extraction
   Chemical Processes

Many new processes depend on multiphase flows
   Additive manufacturing, carbon sequestration,
Multi-Phase Flow: Direct Numerical Simulation

Examples

Cavitation

Microstructure

Bubbly Flow

Atomization

Splash

School of fish
Multi-Phase Flow: Direct Numerical Simulation

Evolving Heterogeneous Continuum Systems

Systems composed of different phases and materials, separated by a sharp interface whose location changes with time.
Multi-Phase Flow: Direct Numerical Simulation

Multiphase flows are characterized by:

• Systems composed of different phases and materials, separated by a sharp interface whose location changes with time
• The physics is well described by continuum theories
• The systems are sufficiently large so that simulations resolving the smallest and the largest scales are impractical
• There are good reasons to believe that the behavior of the smallest scales is—in some sense—universal
• The goal is to use fully resolved numerical simulations of the small scale behavior to help understand how the large and the small scale motion are coupled and to develop “closure” models
Multi-Phase Flow: Direct Numerical Simulation

Flow in pipes

- Stratified
- Slugs
- Mixed
- Dispersed
What are we looking for?
Multi-Phase Flow: Direct Numerical Simulation

Bubbly flows:

How does the void fraction and the bubble size and shape affect their average rise velocity

How do the bubbles disperse as they rise

Do the bubbles form microstructures as they rise and how do such structures affect rise velocity and dispersion

Does the bubbles size distribution change as the bubbles rise due to coalescence, breakup or size dependent migration

How do bubbles interact with wall and boundaries
Atomization of jets

What is the drop size and distribution, their velocities, and how does it depend on the initial nozzle shape and flow conditions?

How long does it take for the jet to break up and how does it depend on the initial nozzle shape and flow conditions?

What are the basic mechanisms that control the initial breakup and the drop formation and how do they depend on turbulence in the jet and the air flow?

Can we develop models and reduced order descriptions?
Direct Numerical Simulations
Multi-Phase Flow: Direct Numerical Simulation

Why Direct Numerical Simulations?

DNS provide us with full details of the flow in both space and time and allow us to compute any derived quantity.

DNS allow us to turn the various physical processes on and off at will to determine their effect.

DNS allow us to precisely define the initial conditions for each case and determine their effects.

The purpose of DNS is not just to reproduce experiments!

Direct Numerical Simulations:

Fully resolved and verified simulation of a validated system of equations that include non-trivial length and time scales.
Multi-Phase Flow: Direct Numerical Simulation

Bubbles in Vertical Channels
Multi-Phase Flow: Direct Numerical Simulation

- Explosive boiling
- Nucleate boiling
- Solidification
- Atomization
- Drag reduction
- Bubbles in channels
- Thermo-capillary migration
- Rayleigh-Taylor Instability
- Cavitating bubbles
Multi-Phase Flow: Direct Numerical Simulation

Domain size: ~9.8M elements
Liquid flow \( \text{Re}_L = 180 \)

Average Gas Volume Fraction: 1%
Initial number of bubbles: 32

\( \rho_l/\rho_g \sim 860 \)
Multi-Phase Flow: Direct Numerical Simulation

- **Incompressible Navier-Stokes Equations:**
  \[
  \rho(\phi) u_{i,t} + \rho(\phi) u_j u_{i,j} = -p_i + \tau_{ij,j} + f_i
  \]

  \[
  \tau_{ij} = 2\mu(\phi) S_{ij} = \mu(\phi)(u_{i,j} + u_{j,i})
  \]

  \[
  u_{i,j} = 0
  \]

- **Level Set Method - Interface tracking equation:**
  \[
  \frac{D\phi}{Dt} = \frac{\partial\phi}{\partial t} + \vec{u} \cdot \nabla \phi = 0
  \]

- **Fluid properties distribution:**
  \[
  \rho(\phi) = \rho_1 H_\varepsilon(\phi) + \rho_2 (1 - H_\varepsilon(\phi))
  \]

  \[
  \mu(\phi) = \mu_1 H_\varepsilon(\phi) + \mu_2 (1 - H_\varepsilon(\phi))
  \]

  \[
  H_\varepsilon(\phi) = \begin{cases}
  0 & \text{, } \phi < -\varepsilon \\
  \frac{1}{2} \left[1 + \frac{\phi}{\varepsilon} + \frac{1}{\pi} \sin \left(\frac{\pi \phi}{\varepsilon}\right)\right] & \text{, } |\phi| < \varepsilon \\
  1 & \text{, } \phi > \varepsilon
  \end{cases}
  \]
Averaging and Analysis of the DNS Data

Mean velocity:
\[ U_i^k(t) = \frac{1}{\alpha_k N_e} \sum_{m=1}^{N_e} \left( \frac{1}{N_w} \sum_{j=1}^{N_w} X_k u_m^i(t + t_j) \right) \]

Turbulent kinetic energy:
\[ k^k(t) = \frac{1}{\alpha_k N_e} \sum_{m=1}^{N_e} \left( \frac{1}{N_w} \sum_{j=1}^{N_w} X_k \sum_{i=1}^{3} \frac{1}{2} \left( u_m^i(t + t_j) \right)^2 \right) \]

Turbulent shear stress:
\[ \tau_{xy}(t) = \frac{1}{\alpha_k N_e} \sum_{m=1}^{N_e} \left( \frac{1}{N_w} \sum_{j=1}^{N_w} X_k u_m^{ix}(t + t_j) u_m^{iy}(t + t_j) \right) \]

Turbulent dissipation rate:
\[ \varepsilon^k(t) = \nu \frac{1}{\alpha_k N_e} \sum_{m=1}^{N_e} \left( \frac{1}{N_w} \sum_{j=1}^{N_w} \sum_{i=1}^{3} \sum_{k=1}^{3} X_k \left( \frac{\partial u_m^i(t + t_j)}{\partial x_k} \right)^2 \right) \]

Local void fraction:
\[ \alpha_k(t) = \frac{1}{N_e} \sum_{m=1}^{N_e} \left( \frac{1}{N_w} \sum_{j=1}^{N_w} X_k(t + t_j) \right) \]

\( k \): field (liquid/gas)  
\( i \): velocity component (1..3)  
\( j \): current time window  
\( m \): current ensemble run  
\( X_k \) is the phase indicator function for field-\( k \)  
\( u_m^i(t + t_j) \) is the fluctuation of velocity component-\( i \) computed during the ensemble run \( m \) at the time instant \( t + t_j \); \( N_e \) is the number of ensemble runs, \( N_w \) is the number of velocity samples in each window, \( t \) is the current time \( \left( j - N_w/2 \right) \Delta t \) is the local window time, and \( \Delta t \) is the time step
## DNS Cases Overview

<table>
<thead>
<tr>
<th>Case:</th>
<th>Single-phase flow</th>
<th>Group 1, (\frac{\rho_i}{\rho_g} = 858.3)</th>
<th>Group 2, (\frac{\rho_i}{\rho_g} = 12.3)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Multiple bubbles</td>
<td>Large bubble</td>
</tr>
<tr>
<td>Case number:</td>
<td>1</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>Typical simulation time instant:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Data averaging interval over all ensemble runs, bulk time units:</td>
<td>115</td>
<td>88.7</td>
<td>17.45</td>
</tr>
</tbody>
</table>
Mean Liquid Velocity and Void Fraction

- Higher density ratio (cases 2 and 3) allows the bubble’s buoyancy force to accelerate the liquid through the drag (top left)
- Observed relative velocity is nearly the same since due to the specified density difference we expect only a 4.2% (bottom left)
- Multiple bubbles concentrate near the wall (cases 2 and 4) while the large bubbles stay near the center line (bottom right)
Informing the Development of multiphase CFD models using Bubbly Turbulent Flow Interface Tracking Results

- Perform new bubbly flow simulations in a turbulent channel with 1% void fraction at Re$_\tau$ = 400
- Statistically analyze the simulation results
- Perform multiphase CFD simulations using NPHASE-CMFD and demonstrate that informed model modifications provide improvements in the results

**Motivation**: Demonstrate that ITM simulations can meaningfully inform the development of multiphase CFD models.
## Multi-Phase Flow: Direct Numerical Simulation

### Results: bubbly flow ITM

<table>
<thead>
<tr>
<th>Direction</th>
<th>Boundaries:</th>
<th>Number of nodes:</th>
<th>Resolution:</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>0.0 … 2(\pi)h</td>
<td>587</td>
<td>(\Delta x^+ = 4.3)</td>
</tr>
<tr>
<td>y</td>
<td>-1.0h … 1.0h</td>
<td>187</td>
<td>(\Delta y^+ = 4.3)</td>
</tr>
<tr>
<td>z</td>
<td>0.0 … (\pi)h</td>
<td>195</td>
<td>(\Delta z^+ = 4.3)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Re_{\tau})</td>
<td>400</td>
</tr>
<tr>
<td>Number of fields</td>
<td>2</td>
</tr>
<tr>
<td>Number of bubbles</td>
<td>60</td>
</tr>
<tr>
<td>Void fraction</td>
<td>1%</td>
</tr>
<tr>
<td>Two-phase Reynolds number</td>
<td>29,530</td>
</tr>
<tr>
<td>Bubble diameter, (D_b/\delta)</td>
<td>0.203</td>
</tr>
<tr>
<td>Eotvos number</td>
<td>0.110</td>
</tr>
<tr>
<td>Morton number</td>
<td>(1.33 \times 10^{-11})</td>
</tr>
<tr>
<td>Weber number</td>
<td>24.63</td>
</tr>
<tr>
<td>Number of elements</td>
<td>21,405 M</td>
</tr>
<tr>
<td>Mesh resolution in wall units:</td>
<td>4.3; 4.3; 4.3</td>
</tr>
<tr>
<td>Statistical sample size, non-dimensional time units:</td>
<td>(5 \times 15)</td>
</tr>
<tr>
<td>Number of computing cores used:</td>
<td>9,600</td>
</tr>
<tr>
<td>Elements per core:</td>
<td>2,230</td>
</tr>
</tbody>
</table>

\(\delta = 2.0\)
Multi-Phase Flow: Direct Numerical Simulation

Results: bubbly flow ITM (2)
Multi-Phase Flow: Direct Numerical Simulation

Results: bubbly flow ITM (3)
Multi-Phase Flow: Direct Numerical Simulation

Results: Statistics (2)

![Graphs showing single-phase and two-phase flow statistics](image-url)
## Multi-Phase Flow: Direct Numerical Simulation

<table>
<thead>
<tr>
<th>Case:</th>
<th>$U_{\text{mean}}$</th>
<th>Re</th>
<th>$Re_{\tau}$</th>
<th>Bubble Size</th>
<th>Bubbles at 1%</th>
<th>Mesh Size (M)</th>
<th>Cores</th>
<th>Elements per core</th>
</tr>
</thead>
<tbody>
<tr>
<td>Local Test Run</td>
<td>0.1</td>
<td>10,236</td>
<td>159</td>
<td>4.09</td>
<td>3</td>
<td>0.8</td>
<td>64</td>
<td>12,500</td>
</tr>
<tr>
<td>$Re_{\tau} = 400$</td>
<td>0.288</td>
<td>29,481</td>
<td>400</td>
<td>1.60</td>
<td>53</td>
<td>6.0</td>
<td>2,000</td>
<td>3,015</td>
</tr>
<tr>
<td>Tested mesh size</td>
<td>1</td>
<td>102,000</td>
<td>1,211</td>
<td>0.54</td>
<td>1,480</td>
<td>212.0</td>
<td>70,666</td>
<td>3,000</td>
</tr>
<tr>
<td>PWR conditions</td>
<td>5</td>
<td>512,000</td>
<td>5,178</td>
<td>0.13</td>
<td>11,500</td>
<td>8,300.0</td>
<td>300,000</td>
<td>27,667</td>
</tr>
</tbody>
</table>

### Time: 0

![Velocity Magnitude](image)

**Retau=158**

**Local Test Run (2 bubbles)**

*Anand Mishra*
Multi-Phase Flow: Direct Numerical Simulation

Local Bo = 30: Coalescence occurs
Multi-Phase Flow: Direct Numerical Simulation

*Steve Palzewicz
Lift/Drag force measurement

Force estimate is over timestep range of: 1000 - 3000
Drag control force: 0.14969 N/kg
Lift control force: 0.14337 N/kg
Drag coefficient: 0.1754
Lift coefficient: 0.7733

Shear rate: 2.0 s\(^{-1}\)
Relative velocity: 0.06875 m/s
Density: 996.5/1.161 (kg/m\(^3\))
Viscosity: 8.5439E-04 (liquid); 1.858E-05 (gas)
Bubble diameter: 5 mm
Multi-Phase Flow: Direct Numerical Simulation

Lift/Drag force measurement

Force estimate is over timestep range of: 4000 - 10000
Drag control force: 0.06853 N/kg
Lift control force: -0.03147 N/kg
Drag coefficient: 0.0803
Lift coefficient: -0.1697

Shear rate: 2.0 s⁻¹
Relative velocity: 0.06875 m/s
Density: 996.5/1.161 (kg/m³)
Viscosity: 8.5439E-04 (liquid); 1.858E-05 (gas)
Bubble diameter: 5 mm
Modeling of multiphase flows is still a very immature area. Interpret the results with care!

For more information about computing multiphase flow, see:
Appendix:

Summary of governing equations
Integral Form

\[
\frac{\partial}{\partial t} \int \rho \ dv = - \int_S \rho \mathbf{u} \cdot \mathbf{n} \ ds
\]

\[
\frac{\partial}{\partial t} \int \rho \mathbf{u} \ dv = \int \rho \mathbf{f} \ dv + \int \left( nT - \rho \mathbf{u} (\mathbf{u} \cdot \mathbf{n}) \right) ds
\]

\[
\frac{\partial}{\partial t} \int \rho (e + \frac{1}{2} u^2) dv = \int \mathbf{u} \cdot \rho \mathbf{f} dv + \int n \cdot (uT - \rho (e + \frac{1}{2} u^2) - q) ds
\]
Multi-Phase Flow: Direct Numerical Simulation

**Conservative Form**

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0
\]

\[
\frac{\partial \rho u}{\partial t} = \rho f + \nabla \cdot (T - \rho uu)
\]

\[
\frac{\partial}{\partial t} \rho(e + \frac{1}{2}u^2) = \nabla \cdot \left( \rho(e + \frac{1}{2}u^2)u - uT + q \right)
\]

**Convective Form**

\[
\frac{D\rho}{Dt} + \rho \nabla \cdot u = 0
\]

\[
\rho \frac{Du}{Dt} = \rho f + \nabla \cdot T
\]

\[
\rho \frac{De}{Dt} = T \nabla \cdot u - \nabla \cdot q
\]
Pressure and Advection

Advection:

Newton’s law of motion: In the absence of forces, a fluid particle will move in a straight line

The role of pressure:

Needed to accelerate/decelerate a fluid particle: Easy to use if viscous forces are small

Needed to prevent accumulation/depletion of fluid particles: Use if there are strong viscous forces
The pressure opposes local accumulation of fluid. For compressible flow, the pressure increases if the density increases. For incompressible flows, the pressure takes on whatever value necessary to prevent local accumulation:

\[ \nabla \cdot \mathbf{u} < 0 \quad \text{High Pressure} \]

\[ \nabla \cdot \mathbf{u} > 0 \quad \text{Low Pressure} \]
Pressure

Increasing pressure slows the fluid down and decreasing pressure accelerates it
Viscosity

Diffusion of fluid momentum is the result of friction between fluid particles moving at uneven speed. The velocity of fluid particles initially moving with different velocities will gradually become the same. Due to friction, more and more of the fluid next to a solid wall will move with the wall velocity.
Sharply stratified flows

The conservation equations for mass and momentum apply to any flow situation, including flows of multiple immiscible fluids. Each fluid generally has properties that are different from the other constituents and the location of each fluid must therefore be tracked. We usually also have additional physics that must be accounted for at the interface, such as surface tension.

The “regular” conservation equations can be extended to handle these situations by using generalized functions.
Multi-Phase Flow: Direct Numerical Simulation

Governing Equations

Identify each fluid by a marker function $H$

\[
H = \begin{cases} 
1 & \text{in fluid 1} \\
0 & \text{Otherwise} 
\end{cases}
\]

The marker moves with the fluid and is updated by integrating the following advection equation in time

\[
\frac{\partial H}{\partial t} + \mathbf{u} \cdot \nabla H = 0
\]

Updating $H$—in spite of its apparent simplicity—is one of the hard problems in CFD!
Surface Tension

In addition to advecting the marker function accurately, we must often account for physics unique to the interface. The most common example is surface tension.

\[ \mathbf{x}(u,v) = \left( x(u,v), y(u,v), z(u,v) \right) \]

**Definition of a surface**

**Tangent vectors**

\[ \mathbf{x}_u = \frac{\partial \mathbf{x}}{\partial u} ; \quad \mathbf{x}_v = \frac{\partial \mathbf{x}}{\partial v} \]

**Normal to the surface**

\[ \mathbf{n} = \frac{\mathbf{x}_u \times \mathbf{x}_v}{|\mathbf{x}_u \times \mathbf{x}_v|} \]

It can be shown that:

\[ k = -\nabla \cdot \mathbf{n} \quad \text{and} \quad kn = \lim_{\delta A \to 0} \left( \int_{\delta A} \mathbf{m} \, ds \right) \]
The “One-Fluid” approach
Fluid flows where there is an imbedded moving boundary are found in many circumstances and the one fluid formulation has been used for a number of such problems. Here we focus on interfaces and phase boundaries.
Multi-Phase Flow: Direct Numerical Simulation

Governing Equations

\[ H(x, y, t) = \int_{A(t)} \delta(x - x') \delta(y - y') \, da' \]

\[ \nabla H = \int_{A} \nabla' [\delta(x - x') \delta(y - y')] \, da' \]
\[ = -\int_{A} \nabla' [\delta(x - x') \delta(y - y')] \, da' \]
\[ = -\int_{S} \delta(x - x') \delta(y - y') \mathbf{n} \, ds' \]
\[ = -\int_{S} \delta(x - x') \delta(y - y') \mathbf{n} \, ds' \]
\[ = -\int_{S} \delta(s) \delta(n) \, ds' \]
\[ = -\delta(n) \mathbf{n} \]

Using:

\[ \delta(x - x') \delta(y - y') = \delta(s) \delta(n) \]
Multi-Phase Flow: Direct Numerical Simulation

Governing Equations

Conservation of Momentum

\[ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \nabla \cdot \mathbf{uu} = -\nabla p + \mathbf{f} + \nabla \cdot \mu \left( \nabla \mathbf{u} + \nabla^T \mathbf{u} \right) + \int_{\Gamma} \sigma_{\kappa} \mathbf{n} \delta(x - x_f) \, da \]

Conservation of Mass

\[ \nabla \cdot \mathbf{u} = 0 \quad \text{Incompressible flow} \]

Equation of State:

\[ \frac{D \rho}{Dt} = 0; \quad \frac{D \mu}{Dt} = 0 \quad \text{Constant properties} \]

The conservation equations are solved on a regular fixed grid and the front is tracked by connected marker points. The “one fluid” formulation of the conservation equations is the starting point for several numerical methods, including MAC, VOF, level sets, and front tracking.
Multi-Phase Flow: Direct Numerical Simulation

Governing Equations

The “one-fluid” formulation implicitly contains the proper interface jump conditions. Integrating each term across a small control volume centered at the interface:

\[
\int_{\delta V} \left[ \rho \frac{Du}{Dt} \right] dv = -\int_{\delta V} \nabla p dv + \int_{\delta V} f dv + \int_{\delta V} \nabla \cdot \mu (\nabla u + \nabla^T u) dv + \int_{\delta V} k\sigma n \delta(n) dv
\]

Jump Condition:

\[
\left[ -p + \mu (\nabla u + \nabla^T u) \right] n = -k\sigma n
\]
Multi-Phase Flow: Direct Numerical Simulation

Governing Equations

We can also show that the “one-fluid” formulation contains the equations written separately for each fluid and the jump conditions:

Write:

\[ u = H_1 u_1 + H_2 u_2 \]
\[ P = H_1 p_1 + H_2 p_2 \]
\[ \rho = H_1 \rho_1 + H_2 \rho_2 \]

Substitute into the momentum equation

\[ H_1 \left( \begin{array}{c}
\text{Momentum equation in phase 1} \\
= 0
\end{array} \right) + H_2 \left( \begin{array}{c}
\text{Momentum equation in phase 2} \\
= 0
\end{array} \right) + \delta(x_f) \left( \begin{array}{c}
\text{Interface conditions} \\
= 0
\end{array} \right) = 0 \]
The one-fluid formulation allows us to treat multiphase flows in more or less the same way as single phase flows. The main differences are:

The density and viscosity change discontinuously across the interface and have to be updated as the interface moves.

Surface tension needs to be evaluated and added to the Navier-Stokes equations.

Because the density is no longer constant, the pressure equations is not a separable elliptic equation anymore.
The Navier Stokes equations can be written in many slightly different forms. Here we will generally work with:

\[
\rho \frac{\partial u}{\partial t} + \rho \nabla uu = -\nabla p + \rho f_b + \nabla \cdot \mu \left( \nabla u + \nabla^T u \right)
\]

When the viscosity is the same everywhere, the viscous terms can be simplified slightly

\[
\nabla \cdot \mu_0 (\nabla u + \nabla^T u) = \mu_0 \nabla^2 u.
\]

For two-dimensional flows:

\[
\frac{\partial}{\partial x} 2\mu_0 \frac{\partial u}{\partial x} + \frac{\partial}{\partial y} \mu_0 \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) = \mu_0 \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\]

\[
\frac{\partial}{\partial x} \mu_0 \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} 2\mu_0 \frac{\partial v}{\partial y} = \mu_0 \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\]
Another Appendix:

Nondimensional Numbers In Multiphase Flows
Multi-Phase Flow: Direct Numerical Simulation

$L$ Size
$U$ Velocity
$\rho$ Density
$g$ Gravity
$\mu$ Viscosity
$\sigma$ Surface Tension

$m$ length
$s$ time
$kg$ Mass

\[
\begin{array}{ccccccc}
L & U & \rho & g & \mu & \sigma \\
\text{m} & \text{s} & \text{kg} & \text{m}^3 & \text{s}^2 & \text{kg} & \text{kg} \\
\text{m} & \text{s} & \text{m}^3 & \text{s}^2 & \text{ms} & \text{s}^2 \\
\end{array}
\]
Multi-Phase Flow: Direct Numerical Simulation

Reynolds Number

\[ \text{Re} = \frac{\rho L U}{\mu} \]

Galileo Number

\[ N = \frac{g \rho \Delta \rho L^3}{\mu^2} \]

Weber Number

\[ \text{We} = \frac{\rho L U^2}{\sigma} \]

Capillary Number

\[ \text{Ca} = \frac{\mu U}{\sigma} \]

Ohnesorge Number

\[ \text{Oh} = \frac{\mu}{\sqrt{\rho \sigma L}} \]

Eotvos (Bond) Number

\[ \text{Eo} = \frac{\Delta \rho g L^2}{\sigma} \]

Morton Number

\[ \text{Mo} = \frac{\Delta \rho g \mu^4}{\rho^2 \sigma^3} \]

Froude Number

\[ \text{Fr} = \frac{U^2}{g L} \]