CFD PROJECT LOGBOOK: Mason Chen

PRE-WORK (Aug-Oct 2025) – Academic Consultation & Research Planning

Objective

- Seek expert guidance on CFD methodology before beginning simulation work.

Tasks Completed

- Researched University of Calgary Mechanical Engineering faculty
- Sent multiple emails explaining project scope and goals
- Followed up after limited initial responses
- Secured a meeting after several outreach attempts
- Discussed proposed CFD approach and modeling strategy

Key Discussion Points

- Importance of isolating baffle geometry as the only manipulated variable
- Recommendation to use Volume of Fluid (VOF) model for free-surface sloshing
- Emphasis on proper boundary condition setup in sealed tanks
- Need to verify numerical stability before comparing force results

Engineering Insight

It required several outreach attempts before finding a professor with available time to meet. This experience reinforced the importance of persistence in professional communication. The discussion helped refine the CFD methodology before any simulations began, strengthening the scientific rigor of the entire project.

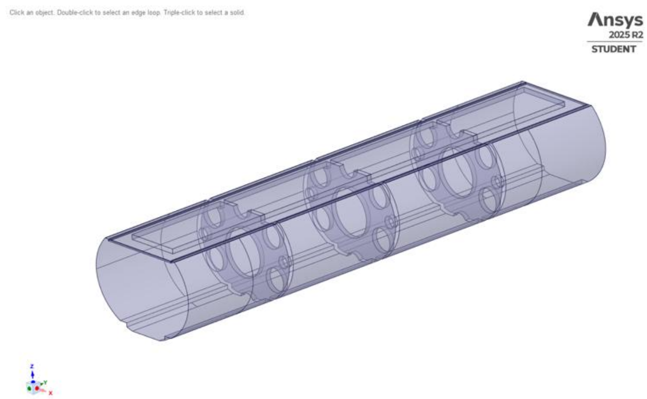
WEEK 1 – (Nov 2-8, 2025) Initial Geometry Preparation (Standard Baffle)

Objective

Prepare simulation-ready geometry for the Standard Baffle tank.

Tasks Completed

- Imported STEP/IGES model into ANSYS SpaceClaim
- Removed decorative text and non-functional features
- Adjusted lid geometry (lowered 5 mm for sealing)
- Combined bodies into a single continuous solid
- Performed Volume Extract to isolate internal fluid region
- Named extracted body volume_volume



Geometry cleanup and volume extraction of Standard Baffle

Engineering Insight

Volume extraction is critical because Fluent solves the flow only inside the fluid region. Any solid bodies mistakenly included would cause solver failure or incorrect force calculations.

WEEK 2 – (Nov 9-15, 2025) Smart Baffle Geometry Preparation

Objective

Prepare Smart Baffle geometry using identical preprocessing steps.

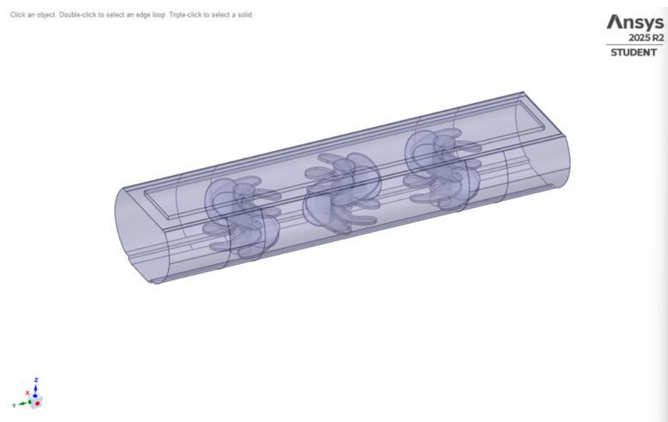
Tasks Completed

- Repeated geometry cleanup

- Verified no open surfaces
- Ensured internal volume extraction was clean
- Confirmed both geometries were consistent in outer dimensions

Key Decision

Geometry consistency between Standard and Smart designs ensures force comparisons are valid.



Geometry cleanup and volume extraction of Smart Baffle

WEEK 3 – (Nov 16-22, 2025) Mesh Development and Refinement (Standard vs Smart Baffle)

Objective

Generate stable and comparable meshes for both Standard and Smart baffle geometries suitable for transient multiphase (VOF) simulation.

Tasks Completed

- Generated initial mesh for the Standard Baffle geometry
- Applied structured surface and body meshing methods
- Evaluated mesh quality metrics (skewness, orthogonal quality)
- Attempted refinement while maintaining numerical stability
- Developed a refined mesh for the Smart Baffle geometry
- Ensured consistent meshing strategy between both designs

Challenges

During refinement of the Standard Baffle mesh, excessive refinement attempts led to:

- Mesher instability
- Failed mesh generation
- Poor element quality

This required balancing mesh resolution with solver stability.

The Smart Baffle geometry required significantly finer mesh resolution due to its more complex internal structure. This increased computational cost but improved free-surface interface capture and internal flow resolution.

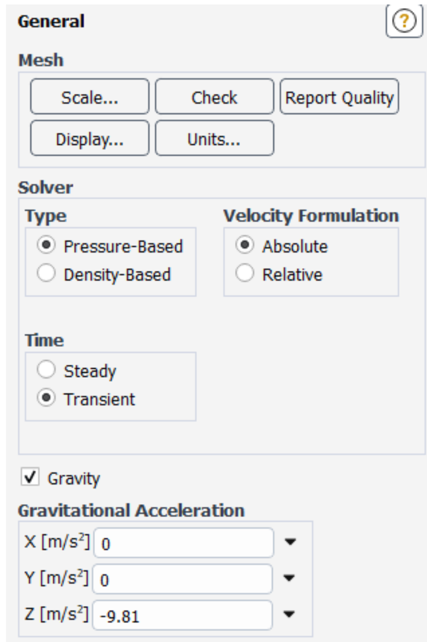
Engineering Insight

Mesh quality directly impacts transient multiphase simulation stability and accuracy. While the Standard geometry-imposed refinement limitations, the Smart Baffle required higher resolution to accurately capture internal circulation effects. Maintaining a consistent meshing strategy between both cases ensured that force comparisons remained scientifically valid.

WEEK 4 – (Nov 23-29, 2025) Fluent Solver Setup

Solver Settings

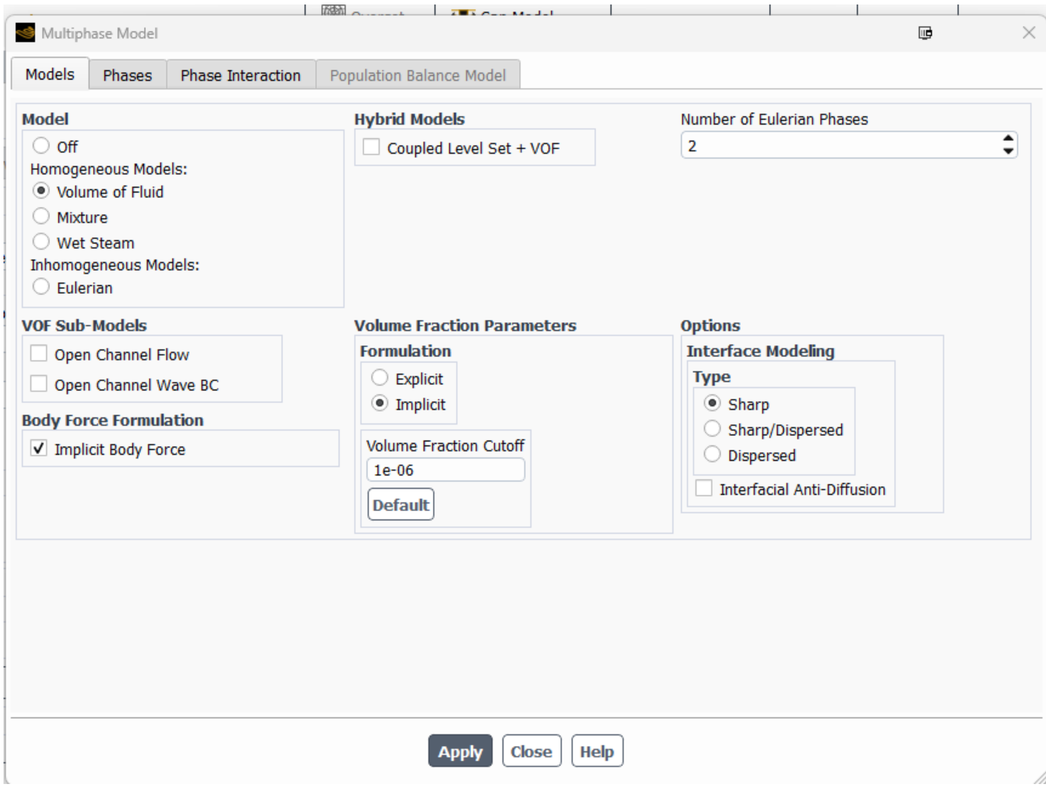
- Pressure-based solver
- Transient
- Gravity: $-9.81\ m/s^2$



Gravity settings

Multiphase Model

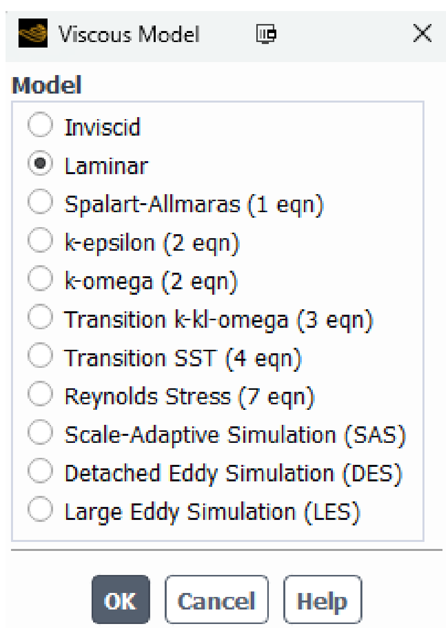
- Volume of Fluid (VOF)
- Air (primary)
- Water (secondary)
- Surface tension enabled



VOF setup panel

Turbulence Model

- Laminar selected



Viscous model panel

WEEK 5 – (Nov 30-Dec 6, 2025) Boundary Condition Testing

Initial Setup

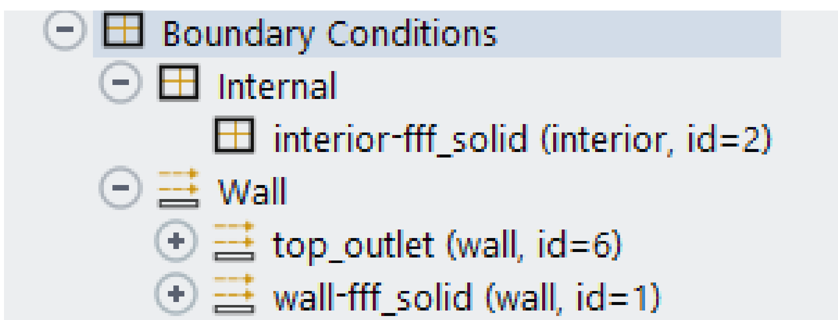
- Top defined as Pressure Outlet

Problem Observed

- Water drained from tank
- Mass not conserved
- Force results unrealistic

Correction

- Lid changed to Wall (sealed tank)



Boundary condition panel

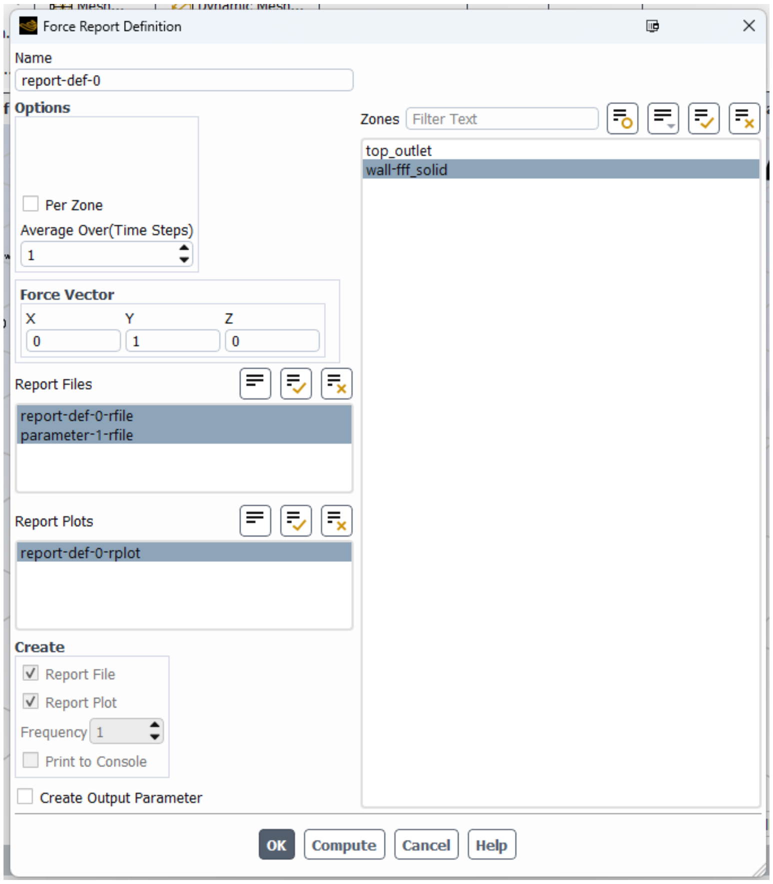
Lesson Learned

Incorrect boundary conditions invalidate multiphase simulations.

WEEK 6 (Dec 7-13, 2025) – Force Report Definition

Setup

- Created force monitor in Y-direction
- Selected wall surfaces
- Enabled time-history plot
- Enabled file export



Force report panel

Goal

Measure impact forces during sloshing events.

WEEK 7 (Dec 14-20, 2026) – Initial Body Acceleration Approach

Method

- Applied −8000 mm/s^2 body force
- Ran 750 timesteps
- Switched acceleration to 0

Result

- Force plateaued quickly
- No sloshing motion

- Fluid reached static equilibrium

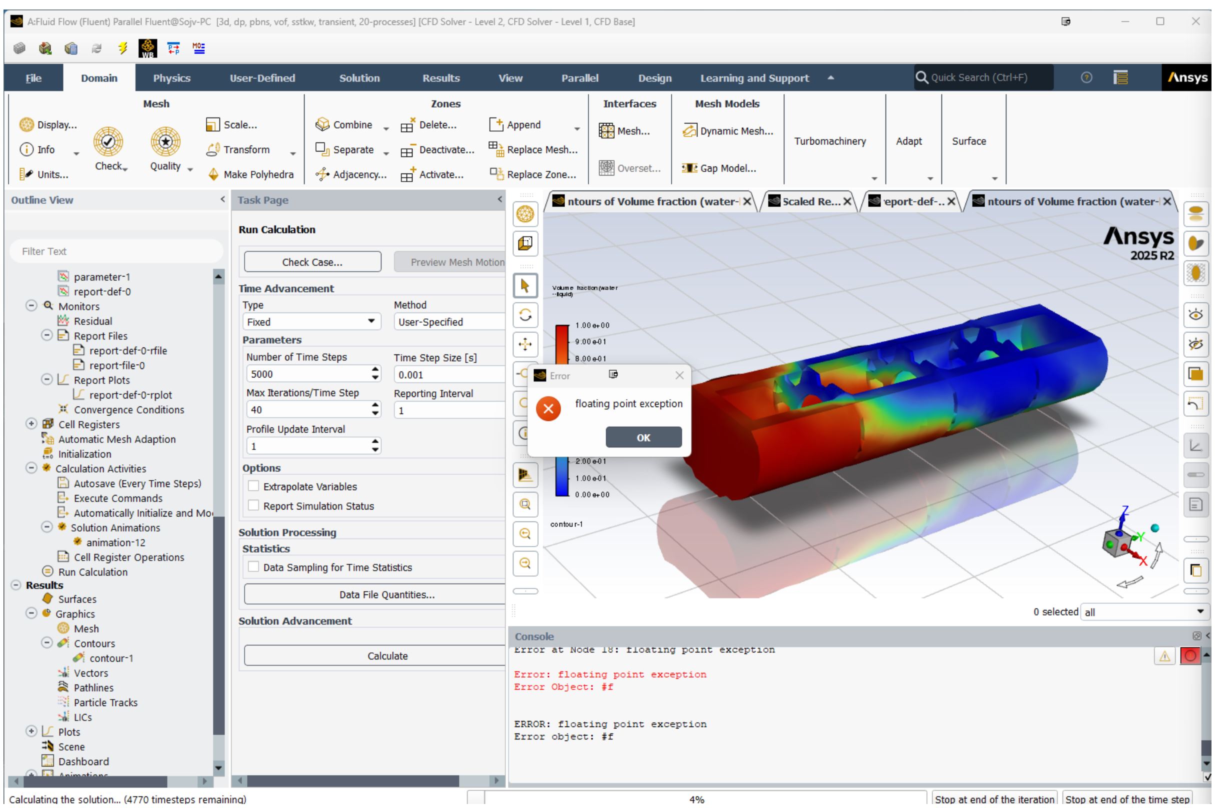
Physics Explanation

Constant acceleration caused pressure equilibrium:
F = m·a
No kinetic energy remained after removal.

WEEK 8 (Dec 28, 2025-Jan 3, 2026) – Debugging Numerical Instability

Issue

- Floating point errors
- Residual divergence
- Solver crashes



Residual explosion / floating point error

Cause

- High acceleration gradients
- Coarse mesh in some regions
- Aggressive timestep

Fix

- Reduced timestep
- Adjusted under-relaxation
- Switched schemes temporarily to first order

WEEK 9 (Jan 4-10, 2026) – Transition to Initial Velocity Method

New Approach

Instead of accelerating tank:

- Removed body force
- Patched initial velocity to fluid
- Simulated sudden braking event

This model's inertia-driven slosh more realistically.

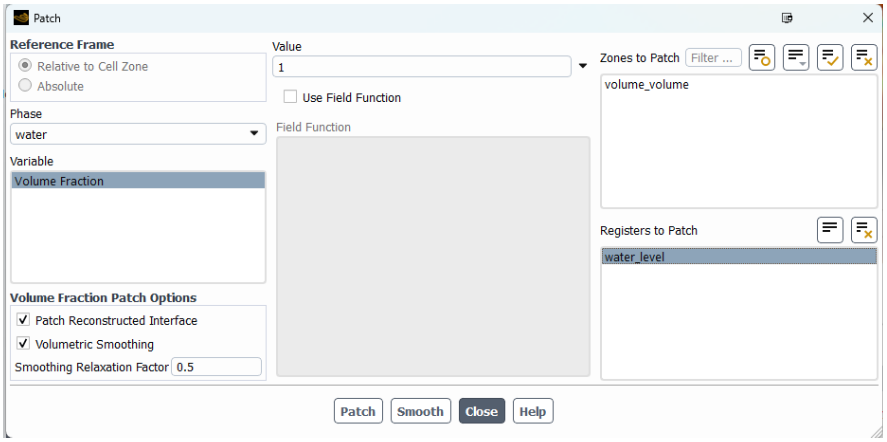
WEEK 10 (Jan 18-24, 2026) – Correct Initialization Procedure

Step 1 – Initialize

Standard Initialization from fluid zone.

Step 2 – Patch Water Volume Fraction

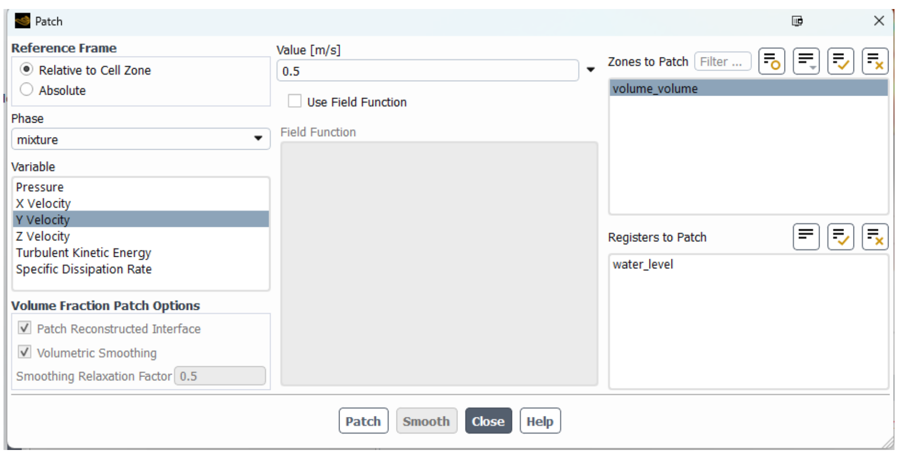
Volume fraction = 1 in lower 60%.



Volume fraction patch

Step 3 – Patch Initial Velocity

Y-velocity = 1 m/s to entire fluid domain.



Velocity patch panel

WEEK 11 (Jan 25-31, 2026) – Numerical Damping Investigation

Observation

Motion stopped around 0.01 s.

Physical Check

Tank dimensions:

- Length: 25 cm

- Height: 4.5 cm
- Fill: 60%

Estimated slosh period ≈ 1 second.

Conclusion:
Damping was numerical, not physical.

WEEK 12 (Feb 8-14, 2026) – Solver Refinement

Final Stable Configuration

- Pressure: PRESTO
- Momentum: Second Order Upwind
- Volume Fraction: Compressive
- Viscous: Laminar
- $\Delta t = 0.0005$ s
- 2000+ timesteps

This reduced artificial diffusion and preserved slosh motion.

WEEK 13 (Feb 15-21, 2026) – Final Comparison: Standard vs Smart

Observations

Standard Baffle

- Higher peak force
- Less flow disruption
- Coarser mesh

Smart Baffle

- Lower peak force
- More distributed energy dissipation
- Better interface behavior

Force curves showed improved damping characteristics in Smart design.

WEEK 14 (Feb 22-28, 2026) – Academic License Upgrade

Objective

Remove software limitations affecting mesh and solver capability.

Tasks Completed

- Identified element and feature limits in Student Version
- Contacted ANSYS support in mid-February
- Requested academic CFD license
- Received approval late February
- Re-ran refined simulations

Engineering Insight

Expanded license improved mesh resolution and solver stability, increasing confidence in final results.

WEEK 15 (Mar 1-4, 2026) – Final Validation & Report Preparation

Verification Steps

- Checked mass conservation
- Verified water remained in tank
- Ensured identical solver settings between cases
- Compared peak force and oscillation duration