The Institute of Aeronautics and Applied Mechanics
Department of Aerodynamics

subject: Advanced Computational Fluid Dynamics
Komputerowa Analiza Przepływów

Exercise 6

Break-down of small dam – volume of fluid method

Purpose

Get familiar with volume of fluid method for simulation of the two phase flow.

Duration: 2h

Mesher

The computational mesh is shown in Fig.1. One has to define the two zones (surface names), one for water and the other for air. The object 'wall_4' can be of any shape (square, cylinder etc.). The object has to be placed sufficiently close to the initial volume of water. The mesh can be unstructured in the zone 'air'

Fluent (settings for fluent ver 12)

1. **Define/General.** Single precision solver, transient formulation. Activate Gravity, Y=-9.81 m^2/s.
2. The computational domain in Fluent is defined in meters.
3. **Define/Models/Multiphase**. Choose the VOF method. Activate the implicit body force.
4. **Define/Materials**. Select the second fluid: water (liquid) from the Fluent database. Use **copy** to confirm.
5. **Define/Phases**. Define as a first phase: air and a second phase: water. Specify your own names to each phase.
6. **Cell Zone Conditions/Operating Conditions**. Verify the gravitational acceleration (Y=-9.81m/s^2). Set the coordinates for the reference pressure to x=5.5 and y=4.5 (it should be far away from water). Activate the Specified Operating Density (1.225 kg/m^3).
7. Initialize the solution with 0 values for velocity and pressure. In **Patch** specify the volume fraction for water (second phase) to 1. Verify the volume fraction of the phase 1 and 2 in **Display/Graphics and Animations/Phases.**
8. Set-up the animation sequences for density of mixture. Control the pressure force acting on the object 'wall_4' by monitoring e.g. the integral of the static pressure in **Solve/Monitors/Surface Monitors** (note that for 2D cases the calculations are performed

for 1 m section of the computational domain in the spanwise direction – perpendicular to the screen).

9. Save the *.cas and *.dat files in your working directory before starting the simulation.
10. Run the simulation with the time step set initially to dt=0.001s and then to dt=0.01s. Set the maximum number of the inner iteration steps in each time step to 40. Verify the convergence of the solution. At each time step the residuals should drop below 1e-2.
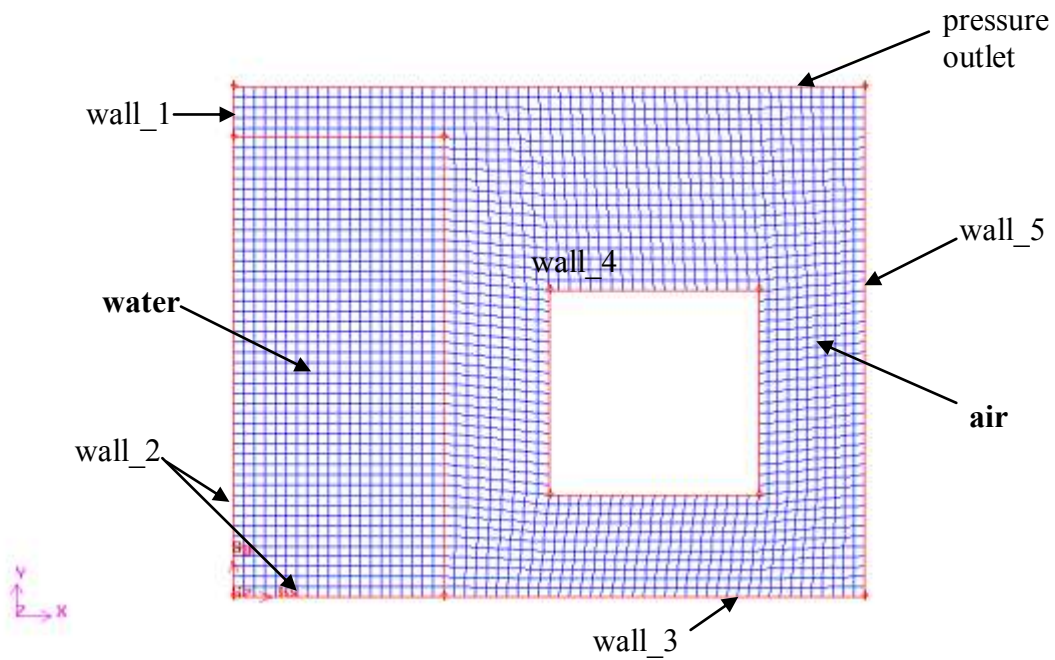11. All animations and the monitoring data should be presented.

Fig.1. Computational mesh. The size of the whole computational domain is 6x5m. Define the zones 'water and 'air' in Gambit (**Operation/Zones/Specify Continuum Type**).