Validation of Fluid Analysis Capabilities in LS-DYNA Based on Experimental Result

Sunao Tokura[1]

[1]Tokura Simulation Research Corporation

1 Introduction

The latest LS-DYNA provides several excellent capabilities for modeling of fluid like materials. These capabilities contains ALE, SPH and CESE for compressible fluid and ICFD for incompressible fluid. Each capability has its own numerical method for computation and characteristics, and is used properly for different target of modeling and purpose of each simulation. For example, ALE and SPH can treat free surface problem automatically without any additional definition of free surface boundary, whereas, explicit definition of free surface boundary is required in ICFD computation. Furthermore, ALE and ICFD are mesh-based approach and mesh definition is required in the region for fluid flow. In contrast, SPH is a kind of meshfree method and then complex mesh generation is not required. In this presentation, fluid is modeled using ALE 2D and 3D, SPH 2D and 3D, and ICFD 2D and 3D. The results obtained from each simulation are compared and validated with the result of an experiment known as the “dam break“ problem.

2 Referenced experiment and material properties used in simulation

The experiment referred in this paper was performed in [1]. A water column is configured using a thin wall in a water tank. As the thin wall is moved very quickly, the water column is collapsed by gravity. The change of the location at the head of the flow is measured in the experiment. The dimensions of the experimental setup are shown in Fig.1 and the photographs taken during the experiment are shown in Fig.2

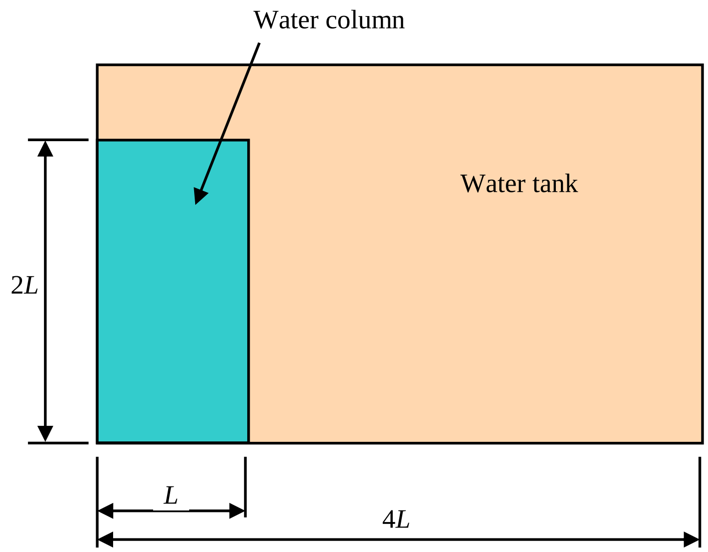


Fig.1: Dimensions of water column and water tank. Actual length of L=0.146 m.

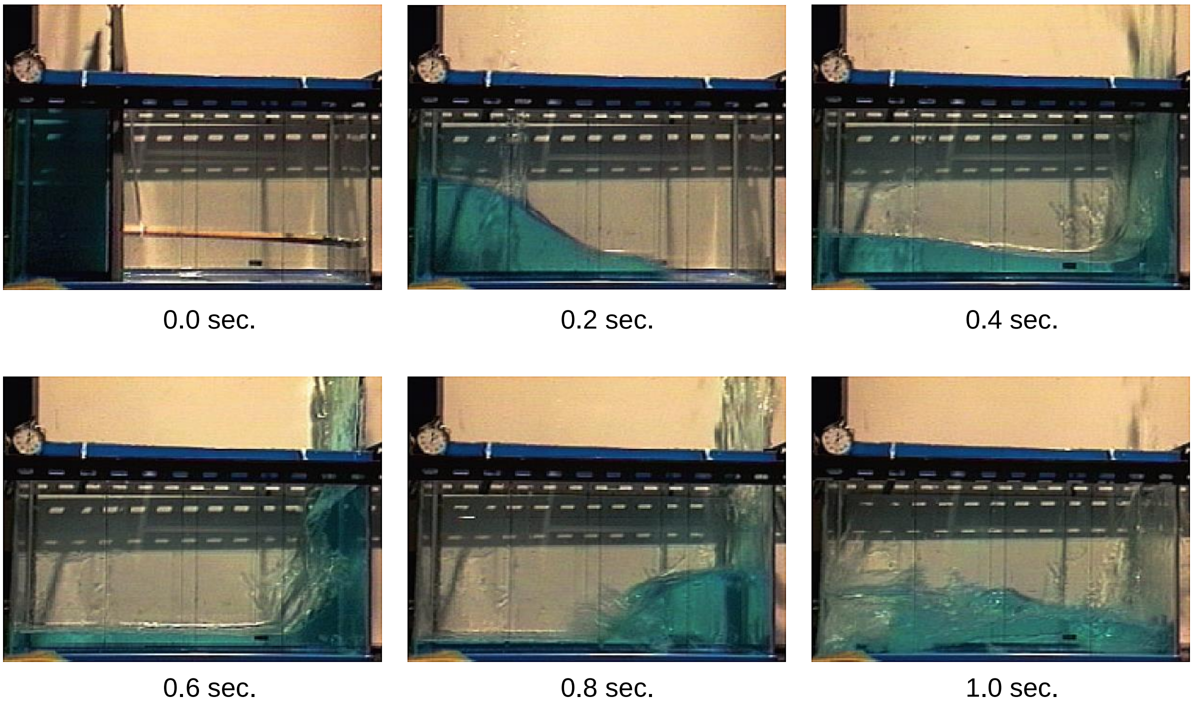


Fig.2: Experimental result of water column collapse [2]

The material properties of water and air (for multi-material definition in ALE) used in the simulation are as follows. Some parameters are used only for compressive flow solver ALE and SPH.

- Water
  - Density = 998.2 kg/m^3
  - Dynamic viscosity = 1.02 x 10^{-3} Pa.s
  - Cut-off pressure = -1.0 x 10^{20} Pa
  - Equation of state = **`*EOS_LINEAR_POLYNOMIAL`**
    - Bulk modulus = 2.22 x 10^9 Pa

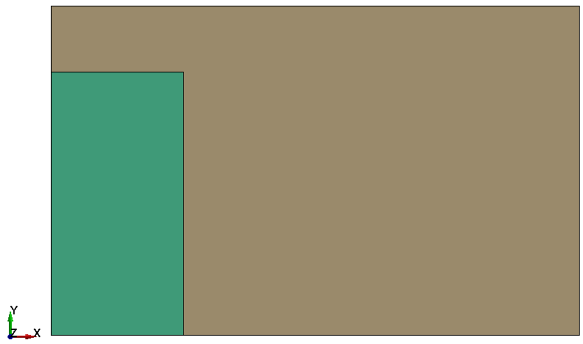
- Air
  - Density = 1.205 kg/m^3
  - Dynamic viscosity = 1.002 x 10^{-3} Pa.s
  - Cut-off pressure = -1.0 x 10^{20} Pa
  - Equation of state = **`*EOS_GRUNEISEN`**
    - Sound speed = 343.6 m/s

Each method is executed using the default values of corresponding parameters. All of the simulation are executed to 0.4 seconds using LS-DYNA R7.1.2 SMP double precision on Windows PC.

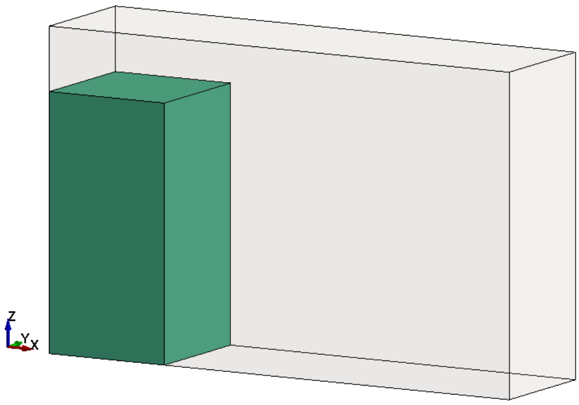
3 Model description

3.1 ALE

Water and air in the water tank are modeled using ALE element. 2D and 3D models are created. The geometry of the models is shown in Fig.3. Depth of 3D model is same as L=0.146 m in Fig.1. Water and air are modeled using muti-material formulation. The boundary is defined as free slip boundary. The model size are summarized in Table 1.



(a) 2D model

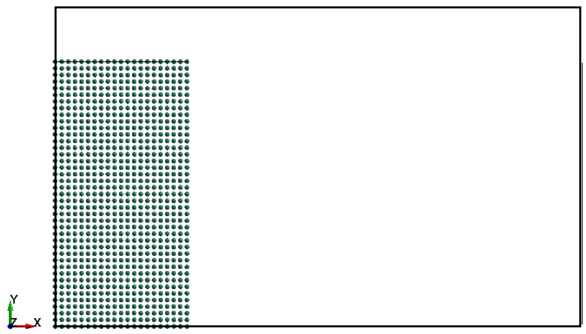


(b) 3D model

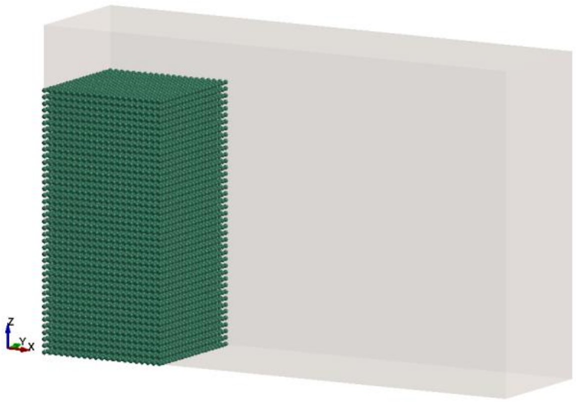
Fig.3: ALE 2D and 3D model

3.2 SPH

Water is modeled using SPH element. 2D and 3D models are created. The geometry of the models is shown in Fig.4. Depth of 3D model is same as L=0.146 m in Fig.1. The walls, floor and ceilling of the tank are modeled using ***RIGIDWALL_PLANAR** for 2D model and shell elements and ***CONTACT_AUTOMATIC_NODES_TO_SURFACE** for 3D model. Friction between SPH elements and the tank is not considered. The model size are summarized in Table 1.



(a) 2D model

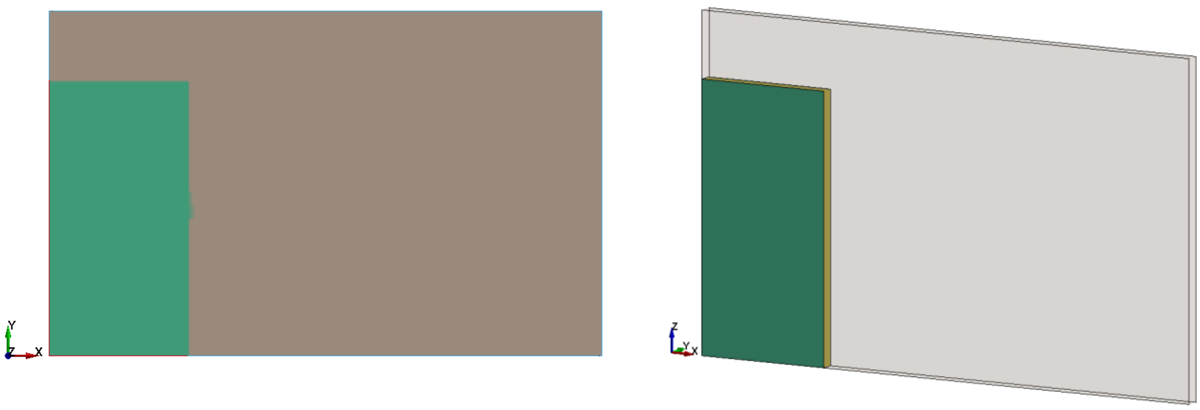


(b) 3D model

Fig.4: SPH 2D and 3D model

3.3 ICFD

Water and air are modeled using ICFD element. Air is defined as void. The geometry of the models is shown in Fig.5. Depth of 3D model is L/10=0.0146 m.All the boundaries of 2D model are defined as nonslip boundary. The front and back walls, floor and ceilling are defined as nonslip and the side walls are defined as free slip boundary in 3D model. In addition nelth=4 is defined on ***MESH_BL** card to define thin boundary layer elements. The model size are summarized in Table 1, however the number of division for x,y and z directions are approximation for ICFD model because of automatic mesh generation.



(a) 2D model (b) 3D model

Fig.5: ICFD 2D and 3D model

Table 1: Model size summary

Method	Dimension	Element size/interval (m)	Number of division/elements			Number of elements in fluid region
			X direction	Y direction	Z direction	
ALE	2D	0.0073	80	50	-	4000
	3D	0.0073	80	20	50	80000
SPH	2D	0.0073	21	41	-	861
	3D	0.0073	21	21	41	18081
ICFD	2D	0.00292	200	125	-	56242
	3D	0.00292	200	5	125	1891341

4 Results and comparison with experiment

The results of the simulation are shown in Fig.6. The time histories of the location of the head of the flow are compared with the experimental result in Fig.7. The results of ALE 2D and 3D cases and the results of ICFD 2D and 3D cases are very similar respectively. Hence only single line represents both 2D and 3D cases for these methods in Fig.7 for clarity of the graph.

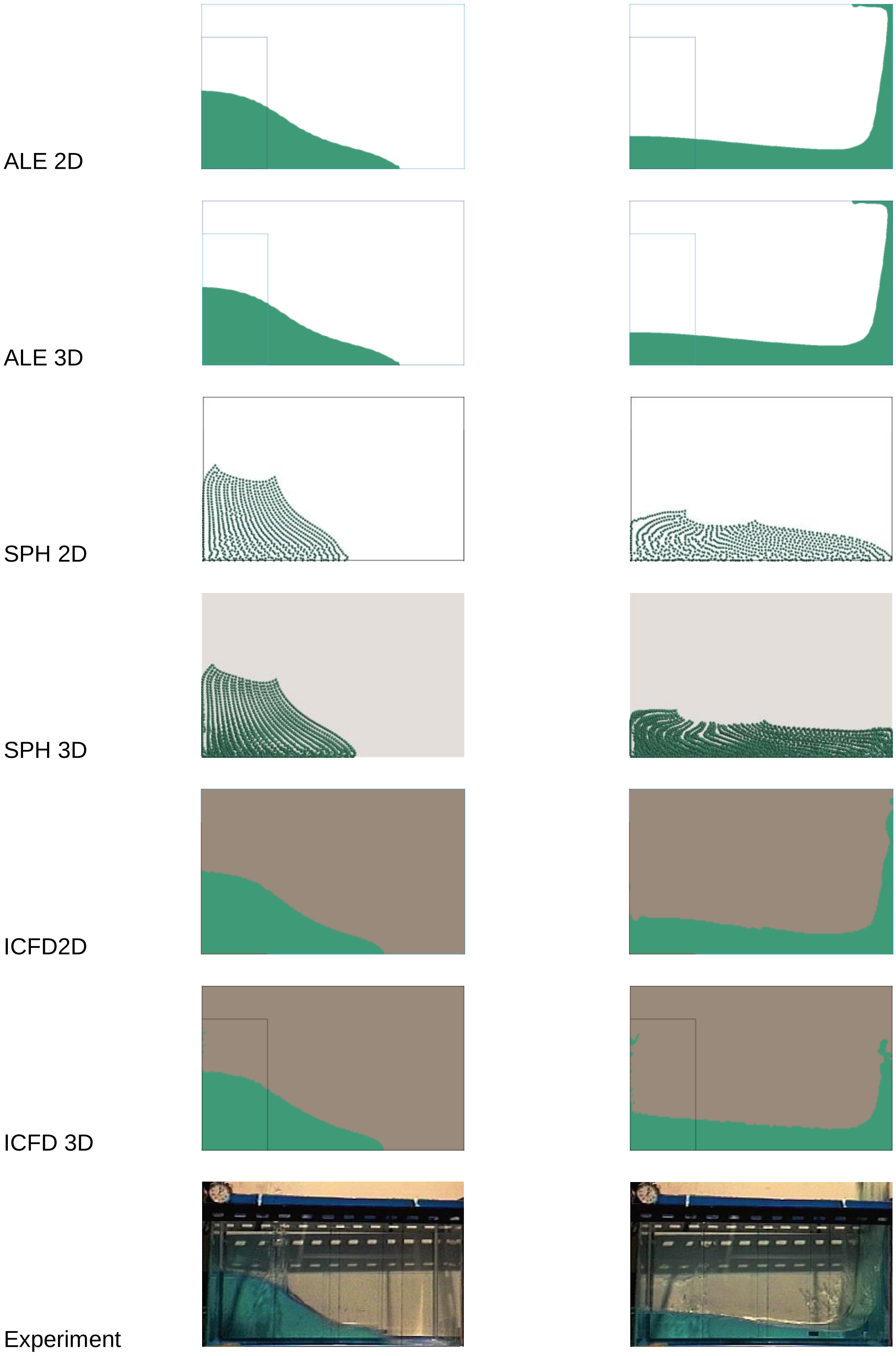


Fig.6: Simulation and experimental results at 0.2 seconds (left) and 0.4 seconds (right)

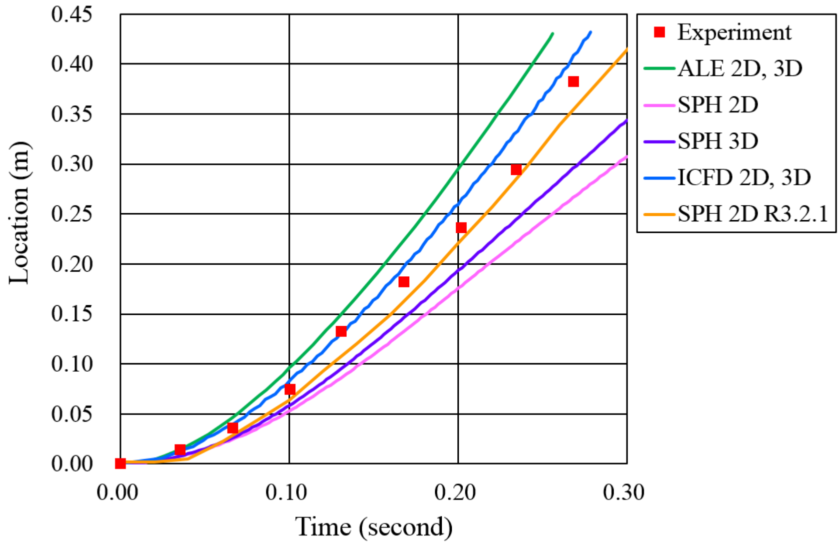


Fig.7: Time history of the location of the head of flow

We can find some facts from the results. ALE is faster than the experiment. The reason can be supposed as the free slip condition (no constraint boundary condition for tangential direction of ALE region) may not be adequate in this case. ICFD shows slightly faster flow velocity. However there is a well-known fact that wet floor of the tank causes slight resistance for fluid flow in a real experiment. So flow velocity obtained from an experiment is slightly delayed. If this fact is considered, it can be regarded that ICFD shows good agreement with the experimental result. SPH 2D and 3D cases show highly viscid behavior. As mentioned above, these simulation are performed using LS-DYNA R7.1.2. The author tried same simulation in 2009 using LS-DYNA V971 R3.2.1 [3]. At that moment SPH showed closer result to the experiment. This former result is shown in Fig.7 as “SPH 2D R3.2.1”. It is supposed that the default formulation of SPH in LS-DYNA may have been changed. CPU times for the simulation methods are summarized in Table 2.

Table 2: CPU time summary (SMP)

model	Number of CPUs	Time (hh:mm:ss)	Averaged time step (microsecond)	Problem cycle	CPU time per zone cycle (nanosecond)
ALE 2D	1	00:03:08	4.149	96407	474
ALE 3D	2	01:50:30	4.149	96407	8594
SPH 2D	4	00:00:48	7.963	50232	989
SPH 3D	4	00:46:59	4.415	90606	992
ICFD 2D	1	00:04:37	1000	400	-
ICFD 3D	1	03:32:06	1000	400	-

5 Conclusions

The fluid analysis capabilities in LS-DYNA were examined using the “dam break” model and compared with the experimental result. ALE and ICFD show relatively good results. In contrast SPH in the latest version of LS-DYNA shows viscid behavior. Users should take much care to define proper input parameters and boundary conditions depending on problems to get accurate simulation results using these excellent fluid analysis capabilities.

Reference

[1] Koshizuka, S., Oka, Y., “Moving-Particle Semi-Implicit Method for Fragmentation of Incompressible Fluid”, Nuclear Science and Engineering: 123, 1996, pp.421-434
[2] Koshizuka, S., “Numerical Analysis of Flow using Particle Method”, Nagare 21, 2002, pp.230-239 (in Japanese)
[3] Tokura, S., “Key points to use SPH Capability Implemented in Multi Purpose Structural Analysis Software“, Comp. Mech. Div. NewsLetter No.42, Japan Society of Mechanical Engineering, May 2009, pp.16-17 (in Japanese)