# Tutorial 10. Simulation of Wave Generation in a Tank

#### Introduction

The purpose of this tutorial is to illustrate the setup and solution of the 2D laminar fluid flow in a tank with oscillating motion of a wall.

The oscillating motion of a wall can generate waves in a tank partially filled with a liquid and open to atmosphere. Smooth waves can be generated by setting appropriate frequency and amplitude. One of the tank walls is moved to and fro by specifying a sinusoidal motion.

In this tutorial you will learn how to:

- Read an existing mesh file in FLUENT.
- Check the grid for dimensions and quality.
- Add new fluid in the materials list.
- Set up a multiphase flow problem.
- Use the dynamic mesh model.
- Set up an animation using Execute Commands panel.

#### **Prerequisites**

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

#### **Problem Description**

In this tutorial, we consider a rectangular tank with a length (L) of 15 m and width (W) of 0.8 m (Figure 10.1). The left wall is assigned a motion with sinusoidal time variation. The top wall is open to atmosphere and thus maintained at atmospheric pressure. The flow is assumed to be laminar.



Figure 10.1: Problem Schematic

# Preparation

- 1. Copy the mesh file, wave.msh and libudf folder to your working directory.
- 2. Start the 2D double precision solver of FLUENT.

#### Setup and Solution

#### Step 1: Grid

1. Read the grid file, wave.msh.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Case...}$ 

FLUENT will read the mesh file and report the progress in the console window.

2. Check the grid.

 $\mathsf{Grid} \longrightarrow \mathsf{Check}$ 

This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

 $Grid \longrightarrow Scale...$ 

Scale Grid	2	<
Scale Factors	Unit Conversion	
× 1	Grid Was Created In 🔳	
Y 1	Change Length Units	
Domain Extents		
Xmin (m) 🔋	Xmax (m) 15	
Ymin (m) 🔋	Ymax (m) 0.8	
Scale	Jnscale Close Help	

Check the domain extents to see if they correspond to the actual physical dimensions. If not, the grid has to be scaled with proper units.

4. Display the grid (Figure 10.2).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 

Grid Display		
Options ☐ Nodes ☑ Edges ☐ Faces ☐ Partitions	Edge Type	Surfaces E = bottom-wall default-interior moving-wall outlet side-wall
Shrink Factor	eature Angle 20	
Surface Name	Pattern Match	Surface Types = = axis clip-surf exhaust-fan fan Outline Interior
Display	Colors C	lose Help

(a) Click Colors....

The Grid Colors panel opens.

Grid Colors		X
Options Color by Type Color by ID Sample	Types far-field inlet interior outlet periodic	Colors blue cyan magenta yellow orange
	symmetry axis wall free-surface internal	light red light green light blue light gray
Reset Colors	Close	Help

- i. Under  $\mathsf{Options},$  enable  $\mathsf{Color}$  by ID.
- ii. Click Close.
- (b) In the  $\mathsf{Grid}$  Display panel, click  $\mathsf{Display}$
- (c) Zoom in near the moving-wall (Figure 10.3).

	outlet
moving-wall	
Grid (Time=0.0000e+00)	FLUENT 6.2 (2d, dp, segregated, dynamesh, vof, lam, unsteady)

Figure 10.2: Grid Display



Figure 10.3: Grid Display (Close-up of moving-wall)

# Step 2: Models

1. Specify the solver settings.

Define	$ \longrightarrow$	Models	→Solver…
	1		

Solver	$\mathbf{X}$
Solver Segregated Coupled	Formulation  function  Explicit  Formulation  Formulation
Space	Time
<ul> <li>⊙ 2D</li> <li>○ Axisymmetric</li> </ul>	<ul> <li>C Steady</li> <li>✓ Unsteady</li> </ul>
C 3D	Transient Controls
	<ul> <li>✓ Non-Iterative Time Advancement</li> <li>✓ Frozen Flux Formulation</li> </ul>
Velocity Formulation	Unsteady Formulation
<ul> <li>Absolute</li> <li>C Relative</li> </ul>	C Explicit C 1st-Order Implicit C 2nd-Order Implicit
Gradient Option	Porous Formulation
<ul> <li>Cell-Based</li> <li>Node-Based</li> </ul>	<ul> <li>Superficial Velocity</li> <li>Physical Velocity</li> </ul>
ОК	Cancel Help

- (a) Under Time, enable Unsteady
- (b) Under Transient Controls, enable Non-Iterative Time Advancement.
- (c) Click OK.
- 2. Enable VOF multiphase model.

Multiphase Model	$\overline{\mathbf{X}}$
Model C Off Volume of Fluid Mixture Eulerian Wet Steam VOF Parameters	Number of Phases 2
VOF Scheme C Implicit C Euler Explicit C Donor-Acceptor Geo-Reconstruct Courant Number	
6.25 Solve VOF Every Iteration Open Channel Flow Body Force Formulation Implicit Body Force	]
OK Cancel	Help

(a) Under Model, enable Volume of Fluid.

The panel expands to show the other settings related to VOF model. Retain the other settings as default.

(b) Click OK.

### **Step 3: Materials**

Define  $\longrightarrow$  Materials...

1. Add liquid water to the list of fluid materials by copying it from the materials database.

Materials				X
Name		Material Type		Order Materials By
water-liquid		fluid	•	• Name
Chemical Formula		Fluent Fluid Materials		Chemical Formula
h2o<1>		water-liquid (h2o <l>)</l>	<u> </u>	Fluent Database
		Mixture		User-Defined Database
		none	~	
Properties		ITa [		
Density (kg/m3)	constant	▼ Edit		
	998.2			
Viscosity (kg/m-s)	constant	▼ Edit		
	0.001003			
]		Y		
	Change/Create	Delete Close	Help	

(a) Click Fluent Database....

Fluent Database Materials panel opens.

Fluent Database Materials	
Fluent Fluid Materials     =       vinyl-trichlorosilane (sicl3ch2ch)       vinylidene-chloride (ch2ccl2)       water-iquid (h2ocl>)       water-vapor (h2o)       wood-volatiles (wood_vol)       Copy Materials from Case	Material Type fluid Order Materials By Name C Chemical Formula
Properties	<b>_</b> _
Density (kg/m3)	constant View
	998.2
Cp (j/kg-k)	constant View
	4182
Thermal Conductivity (w/m-k)	constant View
	0.6
Viscosity (kg/m-s)	constant 👻 View
	0.001003
New Edit Save	Copy Close Help

- i. Select water-liquid (h2o<l>) from the Fluent Fluid Materials list. Scroll down to view water-liquid.
- ii. Click Copy and close the panel.
- (b) Click Change/Create and close the panel.

#### Step 4: Phases

Define  $\longrightarrow$  Phases...

1. Set air as primary phase and water as secondary phase.

Phases		
Phase	Туре	
phase-1	primary-phase	
phase-2	secondary-phase	
	ID	
Interaction	2	
Set	Close Help	

(a) Under Phase, select phase-1.

The Type will be shown as primary-phase.

(b) Click Set....

Primary Phase	
Name	
air	
Phase Material air	▼ Edit
OK Cancel	Help

- i. Change  $\mathsf{Name}$  to air.
- ii. Select air in the Phase Material drop-down list.
- iii. Click OK.
- (c) Similarly, change the Name of phase-2 to water and set its Type to water-liquid.
- (d) Close the Phases panel.

#### **Step 5: Operating Conditions**

Define  $\longrightarrow$  Operating Conditions...

1. Set the gravitational acceleration.

Operating Conditions	×
Pressure Operating Pressure (pascal) 101325 Reference Pressure Location X (m) 0 Y (m) 0	Gravity ✓ Gravity Gravitational Acceleration × (m/s2) g Y (m/s2) -9.81 Variable Dessit: Desenters
ок   с	Specified Operating Density     Operating Density (kg/m3)     1.225 Cancel Help

- (a) Enable Gravity.
- (b) Under Gravitational Acceleration, set Y to  $-9.81 \text{ m/s}^2$ .

As the tank bottom is perpendicular to Y axis, gravity points in the negative Y direction.

- 2. Set the operating density.
  - (a) Under Variable-Density Parameters, enable Specified Operating Density.
  - (b) Retain the default density of  $1.225 \text{ kg/m}^3$ .

Set the operating density to the density of the lighter phase. This excludes the build-up of hydrostatic pressure within the lighter phase, improving the round-off accuracy for the momentum balance.

- 3. Set the reference pressure location.
  - (a) Under Reference Pressure Location, retain the default value of zero for both X and Y.

This location corresponds to a region where the fluid will always be 100% of one of the phases (water). If it is not, it is recommended to change the region to a appropriate location where the pressure value does not change much over time. This condition is essential for smooth and rapid convergence.

4. Click OK to accept the settings and close the panel.

# Step 6: Boundary Conditions

FLUENT maintains zero velocity condition on all the walls. Also, the pressure condition for outlet boundary at the top is set by default to zero gauge (or atmospheric). Hence, there is no need to change the boundary conditions. Retain all the boundary conditions as default.

# Step 7: UDF Library

Define → User-Defined	$] \longrightarrow Functions \longrightarrow C$	Compiled
	Compiled UDFs	N 1997
	Source Files = = Add Delete	Header Files = = Add Delete
	Library Name libudf	Build
	Load Ca	ncel Help

1. Click Load to load the UDF library.

The sinusoidal wall motion will be assigned using user defined function. A compiled UDF library named *libudf* is created for this purpose.

### Step 8: Dynamic Mesh Model

1. Set the dynamic mesh parameters.

Define  $|\longrightarrow|$  Dynamic Mesh  $|\longrightarrow|$  Parameters...

Dynamic Mesh Paramet	ers				X
Models Dynamic Mesh In-Cylinder 2.5D Six DOF Solver Mesh Methods Smoothing Layering Remeshing	Smoothing Options Constan Constan Split Fa Collapse Fa	Layering ht Height ht Ratio actor 0.4 actor 0.4	Remeshing	In-Cylinder	Six DOF Solver
OK Cancel Help					

(a) Under Models, enable Dynamic Mesh.

The panel expands.

- (b) Under Mesh Methods, disable Smoothing and enable Layering.
- (c) Under the Layering tab, set Collapse Factor to 0.4.
- (d) Click OK.

Define

2. Set the dynamic mesh zones.

Zone Names		Dynamic Zones
bottom-wall		
C Stationary		
Rigid Body     Deforming		
© User-Defined		
Motion Attributes	s   Geometry Def	inition Meshing Options
Adjacent Zone	fluid	Cell Height (m) 0.008
Adjacent Zone		Cell Height (m) g
		,

- (a) Under Zone Names, select moving-wall.
- (b) Under Type, retain the default selection of Rigid Body.
- (c) Under Meshing Options tab, set Cell Height to 0.008 m.This is the average size of the cell normal to the moving wall.
- (d) Click Create and close the panel.

# Step 9: Solution

1. Retain the default solution controls.

 $\underbrace{\mathsf{Solve}} \longrightarrow \underbrace{\mathsf{Controls}} \longrightarrow \operatorname{\mathsf{Solution}} \ldots$ 

Solution Controls		X
Equations Flow Volume Fraction	Image: Solver Controls         Max.       Correction Residual Relaxation         Corrections       Tolerance         Pressure       10         Image: Solver Controls       0.25         Image: Solver Controls       0.05         Image: Solver Controls       1         Corrections       1         Image: Solver Controls       1         Image: Solver Controls       0.05         Image: Solver Controls       1         Image: Solver Controls       1 </td <td>n A</td>	n A
Pressure-Velocity Coupling PISO Neighbor Correction 1	Discretization Pressure Standard Momentum First Order Upwind v	Y
	OK Default Cancel Help	

2. Initialize the flow.

Solution Initialization
Compute From Reference Frame Reference Frame Relative to Cell Zone C Absolute
Initial Values
Gauge Pressure (pascal)
X Velocity (m/s) 👔
Y Velocity (m/s) 👔
water Volume Fraction g
Init Reset Apply Close Help

(a) Click Init and close the panel.

The complete domain is now initialized with air. The water level required at start (t=0) can be patched.

3. Create a register marking the region of initial water level.

 $\mathsf{Adapt} \longmapsto \mathsf{Region}...$ 

Region Adaption 🛛 🗙		
Options Input Coordinates		
<ul><li>Inside</li><li>Outside</li></ul>	X Min (m) 0	X Max (m) 15
Shapes	Y Min (m)	Y Max (m)
• Quad	0	0.5
C Cylinder	Z Min (m) Ø	Z Max (m)
Manage	0	,
Controls	0	
Select Points with Mouse		
Adapt Mark Close Help		

- (a) Set X Max to be 15 m.
- (b) Set  $Y \ Max$  to be  $0.5 \ \mathrm{m}.$
- (c) Click Mark and close the panel.

FLUENT displays the following message in the console: 8510 cells marked for refinement, 0 cells marked for coarsening.

4. Patch the initial water level.

Solve  $\longrightarrow$  Initialize  $\longrightarrow$  Patch...

Patch		×
Reference Frame  Refative to Cell Zone Absolute  Phase water Variable  Volume Fraction	Value 1 Use Field Function Field Function	Zones to Patch = = fluid Registers to Patch = = hexahedron-r0
	Patch Close Help	

- (a) Under Registers to Patch, select hexahedron-r0.
- (b) Under Phase, select water.
- (c) Under Variable, select Volume Fraction.
- (d) Set Value to 1.
- (e) Click Patch and close the panel.

- 5. Display the zone motion to check the movement of moving-wall.
  - (a) Display the grid (Figure 10.4).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 

Grid Display		×
Options Nodes Edges Faces Partitions	Edge Type	Surfaces = = bottom-wall default-interior moving-wall outlet side-wall
Shrink Factor	eature Angle 20 Pattern	Surface Types = =
	Match	axis clip-surf exhaust-fan fan
		Outline Interior
Display Colors Close Help		

i. Under Surfaces, deselect default-interior.

Zoom-in the graphics window to get the view as shown in Figure 10.4.

ii. Click Display.





(b) Display the zone motion.

 $\mathsf{Display} \longrightarrow \mathsf{Zone} \mathsf{Motion...}$ 

Zone Motion	×
Preview Controls	Motion History Integration
Time Step (s) 0.001	Start Time (s) 👔
Number of Steps 300	Time Step (s) 0.001
300	Number of Steps 300
Apply Preview Integrate	Reset Close Help

- i. Under Motion History Integration, set Time Step to 0.001.
- ii. Set Number of Steps to 300.
- iii. Click Integrate.
- iv. Under Preview Controls, set Time Step to 0.001.
- v. Set Number of Steps to 300.
- vi. Click Preview to observe the wall motion.
- vii. Close the Zone Motion panel.
- 6. View the contours of volume fraction for water (Figure 10.5).

Display  $\longrightarrow$  Contours...

Contours	X
Options          Options         Image: Section of the section	Contours of Phases  Volume fraction Phase Water Via
	Mill     Max       0     1       Surfaces     I       bottom-wall     default-interior       moving-wall     outlet       side-wall
Diantay	Surface Types = = axis clip-surf exhaust-fan fan

- (a) Select Phases... and Volume Fraction in the Contours of drop-down lists.
- (b) Under Phase, select water.
- (c) Under Options, enable Filled.
- (d) Click **Display** and close the panel.



Figure 10.5: Contours of Volume Fraction for Water

7. Enable the plotting of residuals during the calculation.

$Solve \longrightarrow Monitors$	$\longrightarrow$ Residuals
----------------------------------	-----------------------------

Residual Monit	ors	×
Options	Storage	Plotting
<ul><li>✓ Print</li><li>✓ Plot</li></ul>	Iterations 1000 🛨	Window 1
	Normalization	Iterations 10
	🗆 Normalize 🗹 Scale	Axes Curves
Residual	Check Co Monitor Convergence Cri	nvergence A
continuity		001
x-velocity	V V 0.	001
y-velocity	V N 0.	001
		×
0К	Plot Renorm	Cancel Help

- (a) Under Options, enable Plot.
- (b) Under Plotting, set Iterations to 10.

This will display residuals for only the last 10 iterations.

(c) Click OK.

8. Set hardcopy settings.

File  $\longrightarrow$  Hardcopy...

C EPS C HPGL C IRIS Image C JPEG C PICT C PPM C PostScript C TIFF C Window Dump Command C Window Dump Command C Window Sw	Format	Colorina	File Tyne	Besolution	
· window build 1	C EPS C HPGL C IFIIS Image C JPEG C PICT C PPM C PostScript C TIFF C VRML C Window Dump	<ul> <li>Color</li> <li>Gray Scale</li> <li>Monochrome</li> <li>Options</li> <li>✓ Landscape Or</li> <li>✓ Reverse Fore</li> <li>Window Dump Co</li> <li>import -window</li> </ul>	C Raster C Vector	Width 9 Height 9	

- (a) Under Format, select TIFF.
- (b) Under Coloring, select Color.
- (c) Click Apply.
- (d) Click Preview.

The background of graphics window is changed to white. FLUENT will display a question dialog box asking you whether to reset the window.

- (e) Click Yes in the Question dialog box.
- (f) Close the panel.
- 9. Set the commands to capture the images of contours.

You need to use Text User Interface (TUI) commands to achieve this. For most of the graphical commands, corresponding TUI commands are available.

Solve  $\longrightarrow$  Execute Commands...

	Execu	te Commands		×			
	Defined Commands 3						
	On	Name	Every When Command	^			
		command-1	7 🛨 Time Step 🗸 display set-window 1				
		command-2	7 🛨 Time Step 🗸 display contour water vof 0 1				
		command-3	7 🛨 Time Step 🔻 display hard-copy "vof-%t.tif"				
		command-4	1 Viteration V				
		command-5	1 JIteration	-			
F			OK Define Macro Cancel Help	_			

- (a) Set the number of Defined Commands to 3.
- (b) Enable **On** option for all the commands.
- (c) Under Every, set 7 for all the commands.
- (d) Under When, set Time Step for all the commands.
- (e) For command-1, specify the Command as: display set-window 1

This command will make the window-1 active.

(f) For command-2, specify the Command as: display contour water vof 0 1

This command will display the contours of water volume fraction in the active window.

(g) For command-1, specify the Command as: display hard-copy "vof-%t.tif"

This command will save the image in the TIF format.

The %t option gets replaced with the time step number, when the image file is saved. The TIF files saved can then be used to create a movie. For the information on converting TIF file to an animation file, refer to http://www.bakker.org/cfm/graphics01.htm

- (h) Click OK to accept the settings and close the panel.
- 10. Set the surface monitors.

Solve  $\longrightarrow$  Monitors  $\longrightarrow$  Surface...

Surface Monitors							X	
Surface Monitors 1								
	Name	Plo	t Prin	t Write	Every			
	monitor-1				Time Step	•	Define	
	monitor-2	Γ			Iteration	-	Define	
	monitor-3	Γ			Iteration	-	Define	
	monitor-4				Iteration	7	Define	•
	OK Cancel Help							

- (a) Increase the number of Surface Monitors to 1.
- (b) Enable Plot for monitor-1.
- (c) Under Every, select Time Step.
- (d) Click on Define... next to monitor-1.

Define Surface Monitor	×
Name monitor-1	Report of Grid
Report Type Area-Weighted Average	X-Coordinate
X Axis Time Step 👻	
Plot Window 2	bottom-wall default-interior
,	moving-wall outlet side-wall
File Name	·
monitor-1.out	
OK Curves	Axes Cancel Help

- (e) Select Area Weighted Average in the Report Type drop-down list.
- (f) Select Grid and X-Coordinate in the Report of drop-down list.
- (g) Under Surfaces, select moving-wall.
- (h) Click OK to close both the panels.
- 11. Save the case and data files (wave-init.cas.gz and wave-init.dat.gz).

 $\boxed{\mathsf{File}} \longrightarrow \mathsf{Write}} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$ 

Retain the default Write Binary Files option so that you can write a binary file. The .gz extension will save zipped files on both, Windows and UNIX platforms.

Select File						? 🗙
Look in:	🚞 wave		•	- + 6	🗅 💣 🎟 •	
My Recent Documents Desktop						
My Documents						
My Network Places	Case/Data File Files of type:	wave-init.cas Case/Data File: les	8		•	OK Cancel

12. Start the calculation.

Solve  $\longrightarrow$  Iterate...

Iterate 🛛 🗙
Time
Time Step Size (s) 0.001
Number of Time Steps 4000
Time Stepping Method
• Fixed
C Variable
Options
Data Sampling for Time Statistics
Iteration
Reporting Interval 1
UDF Profile Update Interval 1
Iterate Apply Close Help

- (a) Set the Time Step Size as 0.001 s
- (b) Set Number of Time Steps to 4000.
- (c) Click Iterate.



Figure 10.6: Scaled Residuals

13. Save the case and data files (wave-4000.cas.gz and wave-4000.dat.gz).



Figure 10.7: Monitor Plot of Area Weighted Average on moving-wall

# Step 10: Postprocessing

1. Display filled contours of static pressure (Figure 10.8).

 $\mathsf{Display} \longrightarrow \mathsf{Contours...}$ 

Contours	X			
Options ✓ Filled ✓ Node Values ✓ Global Range ✓ Auto Range Clip to Range □ Draw Profiles □ Draw Grid	Contours of Pressure   Static Pressure  Phase  mixture  Min (pascal)  Max (pascal)			
Levels Setup     28     1     Surfaces     =       Surface Name Pattern     bottom-wall default-interior moving-wall outlet side-wall     Surface Types     =       Match     Surface Types     =       axis clip-surf exhaust-fan fan				

- (a) Select Pressure... and Static Pressure in the Contours of drop-down lists.
- (b) Click Display.

The pressure at the bottom of the tank is maximum and goes on decreasing towards the top. This shows the variation of hydrostatic pressure due to the height of the liquid.



Figure 10.8: Contours of Static Pressure

#### Summary

The dynamic mesh model is used to apply periodic sinusoidal motion to the wall. This generates a wave in the fluid. The VOF model is used to track the air-water interface and consequently the wave motion. Non-iterative time advancement (NITA) was used to reduce the run time of transient simulation. Images displaying contours of water phase were captured to visualize the transient effects.

#### References

1. Flow Around the Itsukushima Gate, an example from Fluent Inc. Marketing Catalog, 2003.

#### **Exercises/Discussions**

- 1. Run the simulation for longer flow time to check the wave pattern.
- 2. Try running the simulation without non-iterative time advancement (NITA) option.
  - (a) Are the flow patterns different?
  - (b) Compare the wall clock time taken to reach the same flow time.
- 3. Run the simulation using variable time step option.
- Try different motions to the wall and observe wave patterns.
   This will need specific C compiler to create UDF library from the source code.
- 5. What other situation can be simulated using the same mesh file?

# Links for Further Reading

- http://www.prads2004.de/pdf/027.pdf
- http://www.prads2004.de/pdf/138.pdf
- http://www.math.rug.nl/~veldman/preprints/OMAE2004-51084.pdf