PARTNERSHIP FOR ADVANCED COMPUTING IN EUROPE

Express Introductory Training in ANSYS Fluent

Workshop 07 Tank Flushing

Dimitrios Sofialidis Technical Manager, SimTec Ltd.

Mechanical Engineer, PhD

PRACE Autumn School 2013 - Industry Oriented HPC Simulations, September 21-27, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, Slovenia

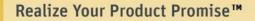








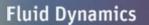




14.5 Release

NNSYS°

Workshop 07a Tank Flushing



Structural Mechanics

Electromagnetics

Systems and Multiphysics

Introduction to ANSYS Fluent

ANSYS Introduction [1]

Workshop Description:

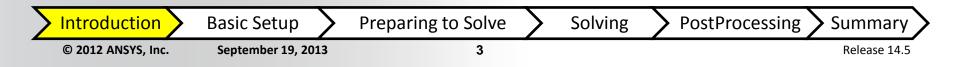
In this workshop, you will model the **filling and emptying of a water tank**. The simulation will be **multiphase** (volume of fluid) and transient (time dependent).

Learning Aims:

- This workshop aims to teach skills in running multiphase simulation in Fluent. The entire simulation approach is covered, including:
 - Setting up a 2-phase simulation.
 - Using "patch" tools to control initialization.
 - Preparing a transient animation.
 - Using Solution Controls to modify the problem definition (turn off the valve).

Learning Objectives:

This workshop teaches skills in the use of multiphase modelling, transient flow modelling, generating images on–the–fly and preparing animations.



ANSYS Mesh Import [1]

Start a new Fluent session.

 3D, Double Precision and Display Mesh After Reading (Display Options).

Read or import the mesh file. tankflush.msh.gz

Use the parallel processing operation if it is available on the training computers.

Click General in the outline tree.

- Scale the mesh to units of cm.
- Set "View Length Unit In" to cm to have Fluent display lengths in centimeters.

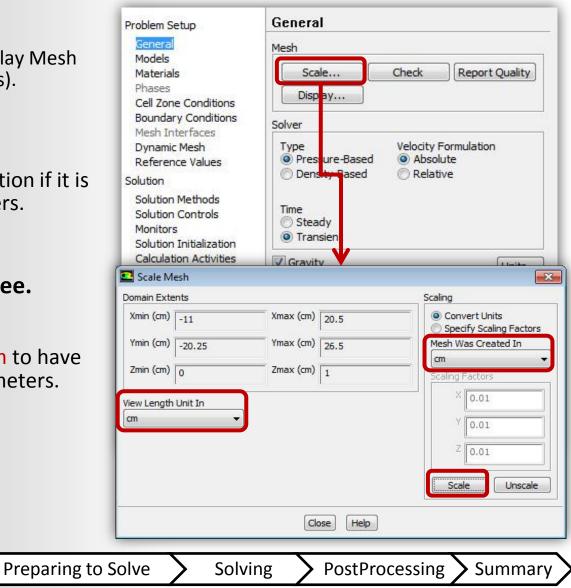
Basic Setup

September 19, 2013

- Verify the domain extents: -11 < x < 20.5 cm -20.25 < y < 26.5 cm 0 < z < 1 cm
- Check the mesh.

Introduction

© 2012 ANSYS. Inc.



Mesh Import [2] **ANSYS**[®]

Orientate the view.

Ambient

Introduction

© 2012 ANSYS, Inc.

- Select "Graphics and Animations • the outline tree.
- Click "Views" button in the cent ulletpane.
- In the panel that opens select • "front" under Views and click "Apply", click "Auto Scale" and t "Close".

Outlet

Basic Setup

September 19, 2013

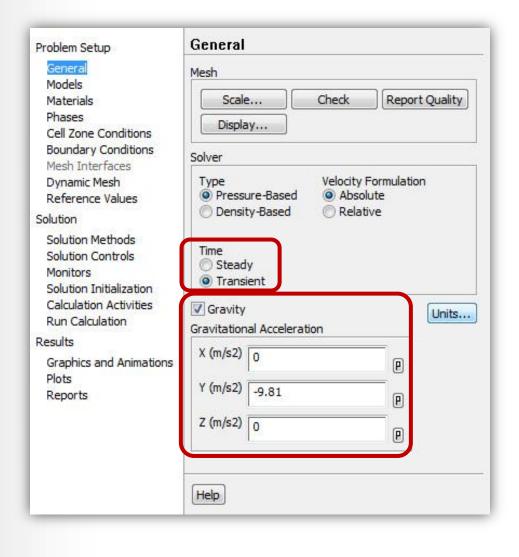
Inlet

	Problem Setup	Views	Views					
	General	Views back	Actions	Mirror Planes 🔳 🗐				
ns" in	Models Materials	bottom front	Default	sym2 sym1				
tre	Phases Cell Zone Condition Boundary Condition	lisometric	Auto Scale Previous					
	Mesh Interfaces Dynamic Mesh	top	Save	Define Plane				
	Reference Values		Read	Periodic Repeats				
then	Solution Methods Solution Controls Monitors	Save Name front	Write	Define				
	Solution Initializatio Calculation Activitie Run Calculation	Apply	mera Close	Help				
t	Results Graphics and Anima	tions						
	Plots Reports	Set Up						
		Options	Scene View	vs				
		Help						
		1						

ANSYS Define Simulation Type

In the General Panel.

- Choose Transient Solver.
- Enable Gravity.
- Set Gravitational Acceleration to -9.81 (m/s2) in the y direction.



PostProcessing

© 2012 ANSYS, Inc.

Introduction

September 19, 2013

Basic Setup

Preparing to Solve

Solving

Summary Release 14.5

ANSYS Enable Turbulence Model

Preparing to Solve

8

Solving

Activate Models in the Outline Tree.

- Double–click Viscous–Laminar in the central pane under Models.
 - In the Viscous Model panel, select kepsilon (2 eqn).
 - Under k–epsilon model, select Realizable.
 - Retain defaults for all other settings.

Basic Setup

September 19, 2013

Click OK.

Introduction

© 2012 ANSYS, Inc.

Viscous Model	
Model	Model Constants
 Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn) Transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Large Eddy Simulation (LES) 	C2-Epsilon TKE Prandtl Number 1 TDR Prandtl Number 1.2
k-epsilon Model Standard RNG Realizable Near-Wall Treatment	User-Defined Functions Turbulent Viscosity none
Standard Wall Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Options Full Buoyancy Effects	Prandtl Number TKE Prandtl Number TDR Prandtl Number none
ОК	Cancel Help

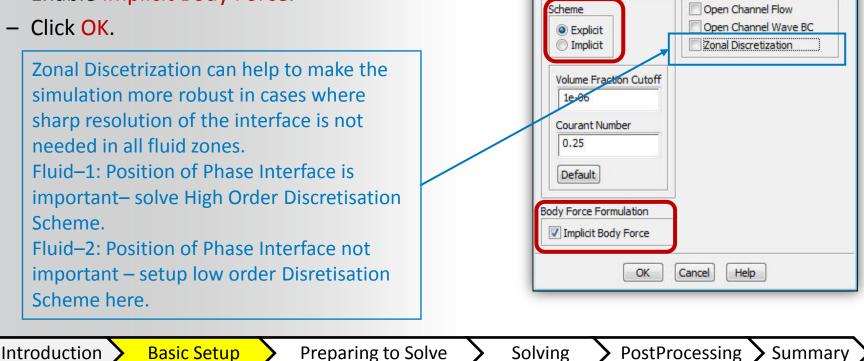
PostProcessing

Summary

Enable VOF Multiphase Model ANSYS®

Enable the VOF multiphase model.

- Double-click on Multiphase.
 - Enable "Volume of Fluid".
 - Set "Number of Fulerian Phases" to 2.
 - Ensure that Scheme is set to Explicit.
 - Enable Implicit Body Force.
 - Click OK.



Volume of Fluid Mixture Eulerian Wet Steam Coupled Level Set + VOF Level Set Volume Fraction Parameters Options

Multiphase Model

Model

O Off

9

X

Number of Eulerian Phases

2

ANSYS Materials and Phases

Add Water to Materials.

- Activate Materials in the Outline Tree.
- Click Create/Edit...
 - In the Materials panel, click Fluent Database...
 - Select "water-liquid" from the Fluent Fluid Materials list, click Copy and then click Close.

Define the phases.

- Activate Phases in the outline tree.
- Double–click phase–1 Primary Phase.
 - Change Name to water.
 - Ensure that water-liquid is selected under Phase Material.
 - Click OK.
- Double–click phase–2 Secondary Phase.

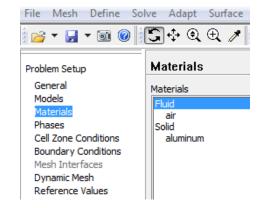
Basic Setup

September 19, 2013

- Change Name to air.
- Select air.
- Click <mark>OK</mark>.

Introduction

© 2012 ANSYS, Inc.



Primary Phase		
lame		
water		
Phase Material wate	r-liquid 🔹	Edit
	OK Cancel Help]

Name	
air	
Phase Material	air 👻 Edit
	OK Cancel Help

PostProcessing

Solving

Preparing to Solve

Summary

Multiphase Model Setup ANSYS[®] Phases Meshina Mesh Generation Phases **Define Phase Interactions.** Solution Setup water - Primary Phase air - Secondary Phase General Click the Interaction Button. Models • Materials Phases In the Phase Interaction Panel that opens, Cell Zone Conditions Boundary Conditions activate the Surface Tension tab. Mesh Interfaces Dynamic Mesh Select constant in the pull-down list and enter Reference Values Solution 0.072 N/m for the Surface Tension Coefficient. Solution Methods Solution Controls Click OK. Monitors Solution Initialization Calculation Activities ID 2 , Ędit Interaction... Dun Colculation Phase Interaction Drag Lift Wall Lubrication Turbulent Dispersion Turbulence Interaction Collisions Slip Heat Mass Reactions Surface Tension atization Interfacial Are Surface Tension Force Modeling Model Adhesion Options Continuum Surface Force Wall Adhesion Continuum Surface Stress Jump Adhesion Surface Tension Coefficients (n/m) air water Edit... constant • 0.072 OK Cancel Help rustriocessing Summary FIEDALING TO SOLVE IIIIIUUUUUUUI Dasic Setup JUIVIIIS 11 © 2012 ANSYS, Inc. September 19, 2013 Release 14.5

ANSYS Set Operating Conditions

Problem Setup>Cell Zone Conditions.

- Click Operating Conditions... in the centre pane below the Cell Zone Conditions box.
 - Verify that Gravity is enabled and the Gravitational Acceleration is set correctly (-9.81 (m/s2) in the y direction).
 - Under Variable Density Parameters, activate Specified Operating Density.
 - Accept the default entry of 1.225 (kg/m3) for the Operating Density.

Pressure		-	Gravity	
	Operating Pressure (pase 101325	cal)	Gravity Gravitational Acceleration	
	101325	P		
	ce Pressure Location	_	X (m/s2)	P
X (cm)	0	P	Y (m/s2) -9.81	P
Y (cm)	0	e	Z (m/s2) 0	(P)
Z (cm)	0	e	Variable-Density Parameters	
L			Specified Operating Densit Operating Density (kg/m3)	y.
			1 225	e

The operating density should be set to the density of the **lightest fluid** in the domain when using the VOF model; otherwise, an erroneous hydrostatic pressure distribution will occur.

Solving

© 2012 ANSYS, Inc.

Introduction

September 19, 2013

Basic Setup

Preparing to Solve

PostProcessing

Summary

ANSYS Define Boundary Conditions [Inlet]

Preparing to

Problem Setup>Boundary Conditions.

- Double click the inlet boundary.
 - Select Normal to Boundary for Direction Specification Method.
 - For the turbulent quantities, select Intensity and Hydraulic Diameter, with TI of 5% and HD of 2.1 cm.
 - Click OK.
- In the Centre Pane, **select water** under Phase and double click on inlet.
 - Set the mass flow rate to 0.2 kg/s.
 - Click <mark>OK</mark>.
- In the Centre Pane, select air under Phase and double–click again on inlet.
 - Set the Mass Flow Rate of air to 0 (kg/s).

Basic Setup

September 19, 2013

Click OK.

Introduction

© 2012 ANSYS, Inc.

one Name		Phase		
inlet		mixtu	re	
Momentum Thermal Radiation Specie	es DPM Mu	ltiphase UDS		
Reference Frame	e Absolute			+
Supersonic/Initial Gauge Pressure (pascal) 0	cor	stant	•
Direction Specification Method	d Normal to Bou	undary		•
Turbulence				
Specification Method	Intensity and H	lydraulic Diameter		•
		lent Intensity (%)	5	
	Hydra	ulic Diameter (cm)	2.1	
0	K Cancel	Help		
Mass-Flow Inlet	K Cancel	Help		
	K Cancel	Help	Phase	
Mass-Flow Inlet	K Cancel	Help	Phase water	
Mass-Flow Inlet Zone Name			water	
Mass-Flow Inlet Zone Name inlet Momentum Thermal Radiation	Species DP		water	
Mass-Flow Inlet Zone Name inlet Momentum Thermal Radiation Mass Flow Specification Method	Species DP	M Multiphase	water	
Mass-Flow Inlet Zone Name inlet Momentum Thermal Radiation Mass Flow Specification Method	Species DP		water	
Mass-Flow Inlet Zone Name inlet Momentum Thermal Radiation Mass Flow Specification Method Mass Flow Rate (kg/s)	Species DP	M Multiphase	water	
Mass-Flow Inlet Zone Name inlet Momentum Thermal Radiation Mass Flow Specification Method Mass Flow Rate (kg/s)	Species DP lass Flow Rate	M Multiphase	water	

ANSYS Define Boundary Conditions [Outlet]

Problem Setup>Boundary Conditions.

- Select mixture under Phase (in the centre pane).
- Select "Outlet" boundary.
 - For the turbulent quantities, select
 Intensity and Hydraulic Diameter, with
 TI of 5% and HD of 12.5 cm.
 - Click OK.
- In the centre pane, select "air" under Phase and click "Edit" again.
 - Switch to Multiphase tab and enter 1 for Backflow Volume Fraction.
 - Click <mark>OK</mark>.

		Phase
outlet		mixture
Momentum Thermal Radi	ation Species DPM Multiphas	e UDS
Gauge Pressu	re (pascal) 0	constant
Backflow Direction Specificat	ion Method Normal to Boundary	
🔲 Radial Equilibrium Pressur		
Turbulence		
ореспсацо	Intensity and Hydraulic I	Diameter 🗸 🗸
	Backflow Turbulent Inter	nsity (%) 5
	Backflow Hydraulic Diame	eter (cm) 12.5

one Name	Phase
outlet	air
Momentum Thermal Radiation Species DPM Mu Backflow Volume Fraction 1	Itiphase UDS
a different the larger from the second	

 Introduction
 Basic Setup
 Preparing to Solve
 Solving
 PostProcessing
 Summary

 © 2012 ANSYS, Inc.
 September 19, 2013
 14
 Release 14.5

ANSYS Define Boundary Conditions [Ambient]

Copy Boundary conditions from outlet to ambient.

- In the centre pane, click "Copy..."
 - Under "From Boundary Zone", select "Outlet".
 - Under "To Boundary Zone", select "Ambient".
 - Select "mixture" under "Phase" and click Copy.
 - Click OK when asked if you want to copy the boundary conditions for mixture.
 - Select "air" under "Phase" and again click Copy.
 - Click OK when asked if you want to copy the boundary conditions for air.
 - Close the Copy Conditions panel.

Basic Setup

September 19, 2013

Introduction

© 2012 ANSYS, Inc.

The Copy Conditions panel is a quick way of transferring common settings from one boundary to another. The "To Boundary Zones" automatically displays boundaries of the same type as the "From Boundary Zone" selection.

outlet	ambient
sym1 sym2	
Phase	
air	•
Con	y Close Help
COP	

Solving

Preparing to Solve

ANSYS Define Solution Methods and Controls

Problem Setup>Solution Methods.

- Under Pressure–Velocity Coupling.
 - Set Scheme to **PISO**.
- Under Spatial Discretization.
 - Gradient Least Squares Cell Based.
 - Pressure PRESTO!
 - Momentum Second Order Upwind.
 - Turbulent Kinetic Energy and Turbulent Dissipation Rate – First Order Upwind.
 - Volume Fraction Geo Reconstruct.

Problem Setup>Solution Controls.

- Set the Under–Relaxation Factor for momentum to 0.3.
- Set the Under–Relaxation Factors for "Turbulent Kinetic Energy" and "Turbulent Dissipation Rate" to 0.5.

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
PISO	-
Skewness Correction	
1	
Neighbor Correction	Ľ
1	
Skewness-Neighbor Coupling	
Spatial Discretization	
Gradient	
Least Squares Cell Based	-
Pressure	
PRESTO!	-
Momentum	
Second Order Upwind	-
Volume Fraction	
Geo-Reconstruct	-
Turbulent Kinetic Energy	
First Order Upwind	-
ransient Formulation	_
First Order Implicit	•
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	

Introduction Basic Setup Preparing to Solve Solving PostProcessing Summary
 © 2012 ANSYS, Inc. September 19, 2013
 16
 Release 14.5

ANSYS Initialize the Initial Solution

Initially, the tank is filled to a **level of 6 cm** with water. Here you will first initialize the flow solution, then create an adaption register and use the register to define the initial location of the liquid surface.

- Initialize the flow field.
 - Select Solution Initialization in the outline tree.
 - Select "inlet" from "Compute from" dropdown list.
 - Set air volume fraction to 1.
 - Click Initialize.

This will instruct the solver to fill the tank with air. The next step is to partially fill the tank with water, resulting in the proper initial condition.

Basic Setup

olem Setup	Solution Initialization	
odels aterials nases	nitialization Methods () Hybrid Initialization () Standard Initialization	
ell Zone Conditions	Compute from	
oundary Conditions	inlet	
	Reference Frame	
eference Values	Relative to Cell Zone Absolute	
olution Methods	nitial Values	_
onitors	X Velocity (m/s)	· ·
olution Initialization	0	
alculation Activities	Y Velocity (m/s)	
ults	-0.8004598	-
raphics and Animations	Z Velocity (m/s)	
ots eports	0	7
	Turbulent Kinetic Energy (m2/s2)	- =
	0.00240276	-
	Turbulent Dissipation Rate (m2/s3)	
	0.01255714	
	air Volume Fraction	
	1	
		-
[Initialize Reset Patch	
	Reset DPM Sources [Reset Statistics]	
[Help	
	Initialize Reset Patch Reset DPM Sources Reset Statistics	

Introduction

September 19, 2013

17

Preparing to Solve

ANSYS Patch the Initial Solution – Adaption Register [1]

- Next, define the region of the domain to be filled with liquid.
 - In the top menu, select Adapt>Region.
 - Enter the values shown in the panel to the right.
 - Click Mark. DO NOT CLICK ADAPT!

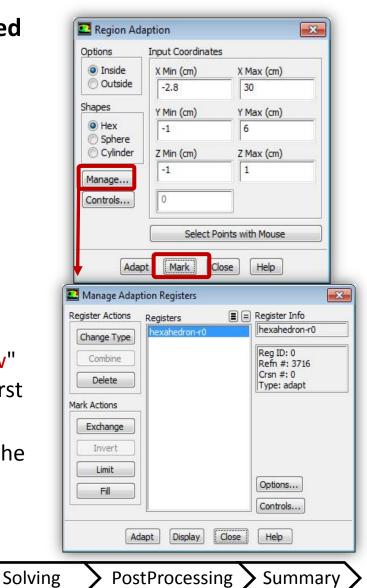
A message appears in the Fluent console informing you that 3716 cells have been marked.

- To view the marked cells, click Manage.
- Verify the register **hexahedron-r0** under Registers is selected and click **Display**.
- You may need to zoom in (use the "Fit to Window" icon) because the mesh was scaled since it was first displayed.
- Close the Manage Adaption Registers panel and the Region Adaption panel.

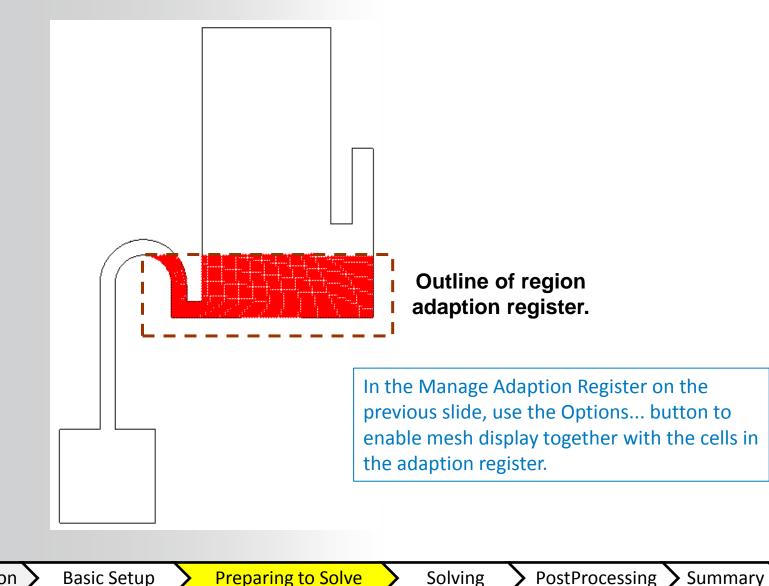
The marked cells will be displayed in the graphics window (see next page).

Basic Setup

Introduction © 2012 ANSYS, Inc.



Patch the Initial Solution – Adaption Register [2] ANSYS[®]



Introduction © 2012 ANSYS, Inc.

September 19, 2013

Preparing to Solve

19

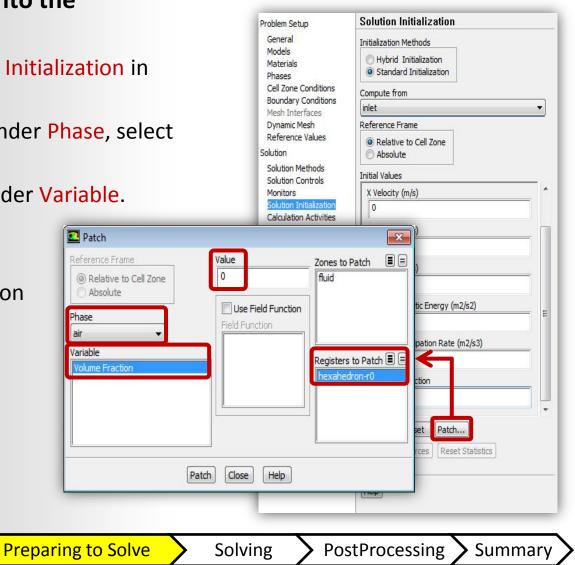
Solving

PostProcessing Summary

Release 14.5

ANSYS Patch the Initial Solution

- Patch the initial solution into the adaption register.
 - Click Patch under Solution Initialization in the outline tree.
 - In the panel that opens, under Phase, select air.
 - Select Volume Fraction under Variable.
 - Set Value to 0.
 - Under Registers
 to Patch, select the adaption register you created.
 - Click Patch.
 - Close the Patch panel.



Introduction

Basic Setup

20

ANSYS Initialize Display Settings [1]

Use the Arrange Windows layout button, and set up 2 graphics windows side-by-side.

By clicking at Window 2 you can activate this Window to display the Initial Solution.

Window 1

- Choose Graphics and Animations in the Outline Tree.
- Choose Contour in Graphics.
 - Switch to Phases.
 - … Volume Fraction air.

 choo Check Options Filled Node Values Global Range Auto Range Clip to Range Draw Profiles Draw Mesh 		•	rface list.		In multiph problems, contours of fraction to correct init condition beginning highly reco	disp of vo con itial befo to it	lume firm the re erate is	Rias	1000000 9,550-01 9,000-01 8,500-01 7,500-01 7,500-01 7,000-01 9,500-000-000000000000000000000000000000		(Time=0.0000e+00) WSYS Fluent 14.5 (3d, c		R14.5
Introduction	<u> </u>	ic Setup	Pre	barir	ng to Solve	$\mathbf{>}$	Solving		PostPr	ocessin	g 🔪 Sum	imary	>
© 2012 ANSYS, Inc	:. Sej	otember 19, 2	013		21						Rele	ease 14.5	;

•

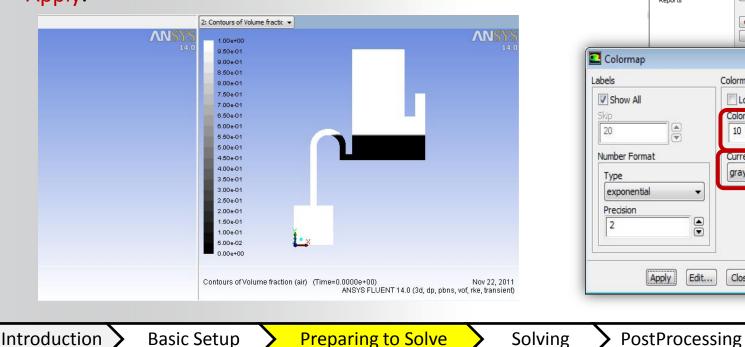


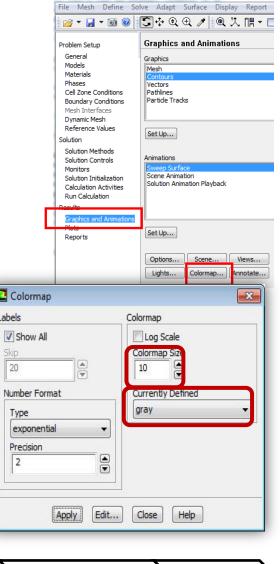
2: Contours of Volume fract -

ANSYS Initialize Display Settings [2]

You can change the colour map used for plotting images. We will change from Blue–Green–Red to a Grayscale scheme.

- Choose Colormap.
 - Set Colormap size to 10.
 - Choose the gray Scheme.
 - Apply.





September 19, 2013

Summarv

ANSYS Define Calculation Activities [1]

In this step you will define activities that Fluent will perform during the calculation. These activities are as follows:

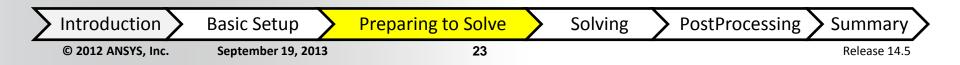
- To autosave case and data files.
- To turn off the supply of water after t = 1 second. (Mass flow rate boundary condition will be changed to zero).

Set autosave options.

Select Calculation Activities in the outline tree.

- Click Edit... next to Autosave.
 - Set Autosave Every (Time Steps) to 25.
 - [If running Fluent standalone, rather than under workbench]
 In the panel that opens, enter the file name tank–flush.gz.
 - [If running under workbench] No action needed.
 - Retain the defaults for all other settings and click OK.

Autosave X				
Save Data File Every (Time Steps) 25				
Data File Quantities				
Save Associated Case Files				
 Only if Modified Each Time 				
File Storage Options				
Retain Only the Most Recent Files				
Maximum Number of Data Files 0 V Only Associated Case Files are Retained				
File Name				
tank-flush.gz Browse				
Append File Name with time-step				
OK Cancel Help				



ANSYS Define Calculation Activities [2]

Define a command to modify the boundary condition after 1 second:

In the Centre Pane, under Execute Commands.

- Click Create/Edit.
 - In the Panel that opens, set Defined Commands to 1.
 - Check Active next to the command line.
 - Enter the following command to be executed. Please make sure the spelling is exactly as written as below, take special care with the hyphens "-": define boundary-conditions mass-flow-inlet inlet water yes no 0

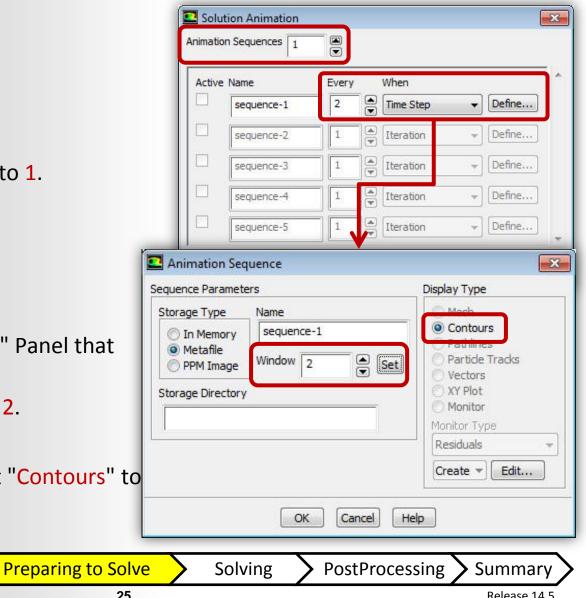
-	Execute Commands	
 Set "Every" to 100. Set "When" to "Time Step". Click OK. 	Defined Commands 1 Active Name Every When Command command-1 100 Time Step define boundary-conditions mass-flow-inlet inlet water yes no 0 command-2 1 Iteration define boundary-conditions mass-flow-inlet inlet water yes command-5 1 Iteration OK Define Macro Cancel Help	s no 0
Introduction > Basic Setup > Pre	eparing to Solve Solving Solving Summary	$\overline{\mathbf{y}}$
© 2012 ANSYS, Inc. September 19, 2013	24 Release 14.	.5

Define Animation Solution [1] **ANSYS**[®]

Set the Animation Sequence.

Calculation Activities>Solution Animations>Create/Edit.

- In the Panel that opens.
 - Set "Animation Sequences" to 1.
 - Set "Every" to 2.
 - Set "When" to "Time Step".
 - Click "Define".
 - In the "Animation Sequence" Panel that opens.
 - Set "Window" number to 2.
 - Click "Set". •
 - Under Display Type select "Contours" to • open the contours panel.



Introduction

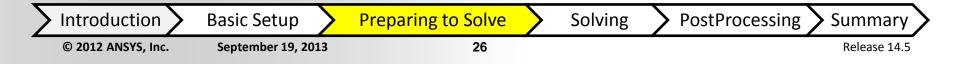
Basic Setup

ANSYS Define Animation Solution [2]

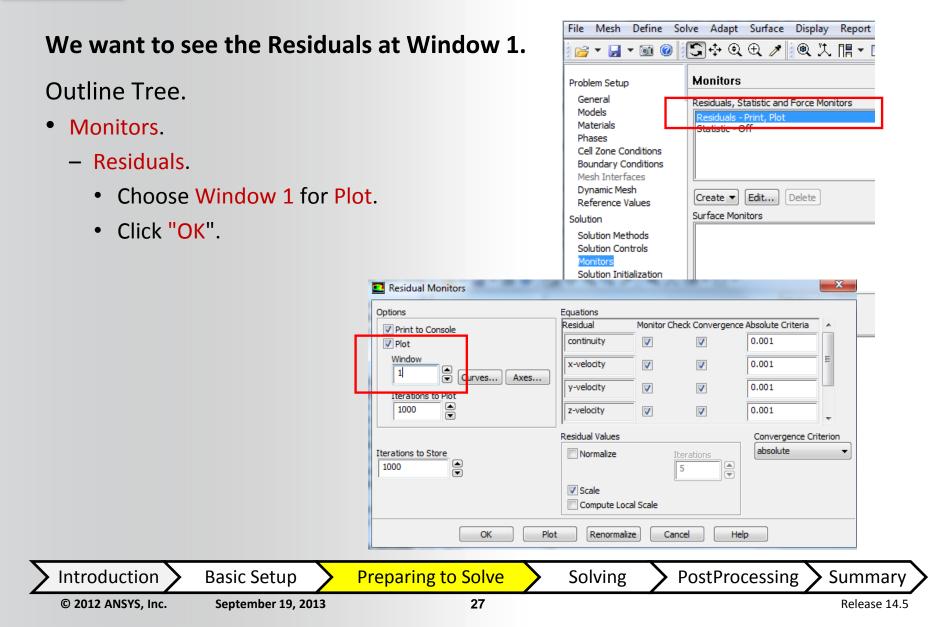
Set the animation sequence cont ...

- In the Contours panel select "Filled" under "Options".
- Under "Contours of" select "Phases..." and choose "air" for the "Phase" to be displayed.
- Under "Surfaces" select "sym1" zone.
- Click "Display" and close the panel.
- Close remaining panels by clicking "OK".

Contours		_
Options Filled Options Global Range Auto Range Clip to Range Draw Profiles Draw Mesh	Contours of Phases Volume fraction Phase air IMIN Max 0 1	•
Levels Setup 10 1 Surface Name Pattern Match	outlet sym1 sym2	
	New Surface ▼ Surface Types axis dip-surf exhaust-fan fan	
Display	Compute Close Help]



ANSYS Define Residual Monitor



ANSYS Run the Calculation [1]

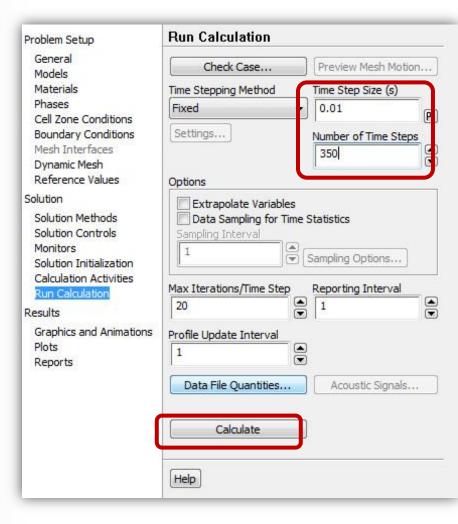
Before run the calculation, you should save the case and data files.

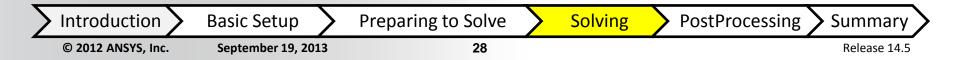
- Use the Save toolbar button to write case and data files as "tank–flush–init.cas.gz".
- If running Fluent within ANSYS Workbench, Select Save Project.

Select Run Calculation from the Outline Tree.

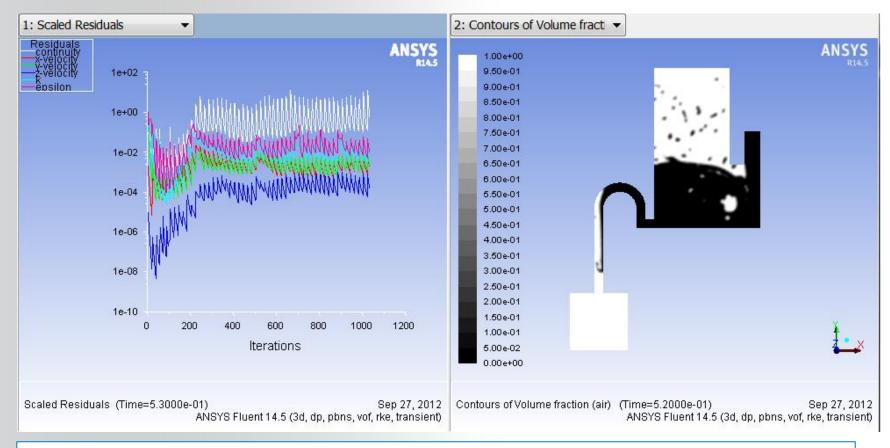
- Enter 0.01 s for Time Step Size.
- Enter 350 under Number of Time Steps.
- Click Calculate.

The solution will require approximately half an hour to compute. You can choose to run all of the calculations or stop the iterations, read final data file or check the provided animation.

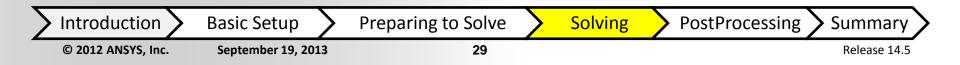




ANSYS Run the Calculation [2]



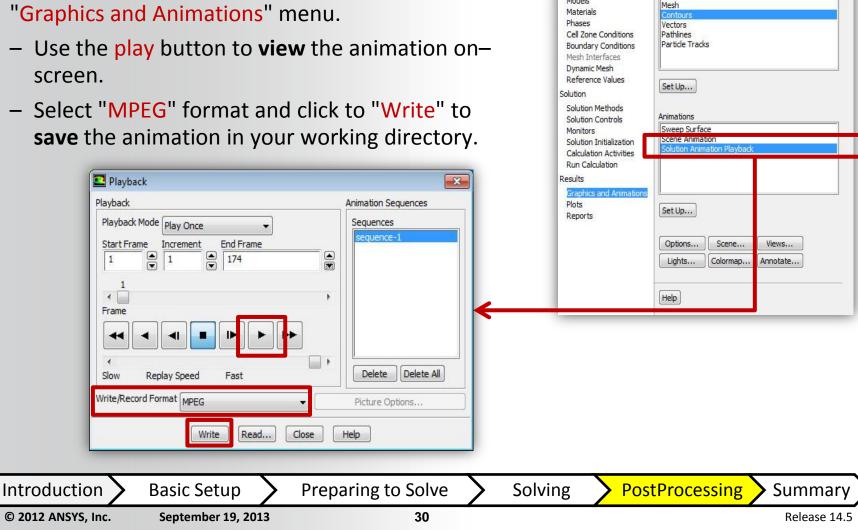
This is a snapshot of the graphics windows after the completion of the first 53 time steps.



PostProcess Results [1] **ANSYS**®

Generate Animation

- Select "Solution Animation Playback" from the "Graphics and Animations" menu.
 - Use the play button to view the animation onscreen.
 - Select "MPFG" format and click to "Write" to **save** the animation in your working directory.



Graphics and Animations

Graphics

Problem Setup

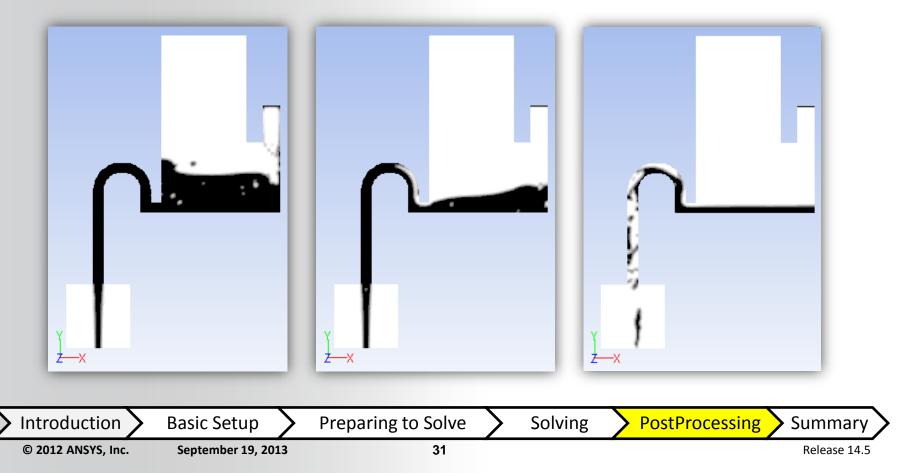
General

Models

ANSYS PostProcess Results [2]

The animation can be played using most of the standard multimedia Players like **Windows Media Player**.

The "Animation Playback" tool can also be used to generate a sequence of picture frames.



ANSYS Further Work

There are many ways the simulation in this tutorial could be extended for instance reloading the saved initial case and data files and then try:

- Switch to Different Discretization Schemes for Volume Fraction.
 - Compressive or Modifed HRIC.
- Modify the Time Step size.

© 20

- Reduce the Time Step Size by Factor 2 or 5.
- Use Variable Time Stepping to ensure that the time step size corresponds to a predetermined value for the Courant Number in the region of the phase interface.

File Mesh Define		Variable Time Step Settings	
i 📂 🕶 🔛 👻 🎯 🥝	[\$\$\$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$	Global Courant Number 2	Courant Number of 2
Problem Setup General	Run Calculation	Ending Time (s) 4	means the Phase
Models Materials	Check Case Preview Mesh Motion Time Stepping Method Time Step Size (s)	Minimum Time Step Size (s) 1e-05	Interphase is passing only two Cells per
Phases Cell Zone Conditions Boundary Conditions	Fixed O.01 Settings	Maximum Time Step Size (s) 1	Time Step.
Mesh Interfaces Dynamic Mesh	Settings Number of Time Steps	Minimum Step Change Factor 0,5	Time Step.
Reference Values	Options	Maximum Step Change Factor 1.5	
Solution Solution Methods	Extrapolate Variables Data Sampling for Time Statistics	Number of Fixed Time Steps 1	
Solution Controls Monitors Solution Initialization	Sampling Interval	User-Defined Time Step none	
Calculation Activities Run Calculation	Time Sampled (s)	OK Cancel Help	
oduction	Basic Setup > Preparing to	Solve 🔪 Solving 🔪 I	PostProcessing Summary
12 ANSYS, Inc.	September 19, 2013	32	Release 14.5

ANSYS[®] Wrap–Up

This workshop has shown the basic steps that are applied in VOF simulations:

- Setup Phase and Interaction.
- Setting boundary conditions per Phase and Solver Settings.
- Running a transient simulation whilst write data and animation data.
- Postprocessing the results.

One of the important things to remember in your own work is, before even starting the ANSYS software, is to think WHY you are performing the simulation:

- What information are you looking for.
- What do you know about the inlet conditions.

In this case we were interested in the how long it would take to completely empty the tank.

Knowing your aims from the start will help you make sensible decisions of how much of the part to simulate, the level of mesh refinement needed, and which numerical schemes should be selected.

