Spray Modeling Tutorial using ANSYS-CFX

By Ahmed Al Makky

@Ahmed Al Makky 2012

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, mechanical or photo-copying, recording, or otherwise without the prior permission of the publisher.

Spray Modelling using ANSYS-CFX

Introduction

The tutorial was written in a rush so it has spelling mistakes never go the time to correct them, feedback would much appreciated to improve the tutorials. A mesh file is provided with this tutorial in order to focus on the combustion simulation. Sprays are encountered in lots of engineering applications from internal combustion engines to gas turbines to airconditioning systems to etc. This tutorial will show the power of multiphase flow simulation provided in ANSYS CFX.

Step 1:

Under the Analysis tools double click Fluid Flow(CFX), then double click on the geometry icon.



Step 2:

The next step is to select the scale of study the simulation is run at which is in centimetres and then press Ok.

\mathcal{P}	
ANSYS Workbench	
Select desired length unit:	
C Meter C Foot	
Centimeter Cinch	
C Milimeter	
C Micrometer	
Always use project unit Always use selected unit Enable large model support	
ОК	0

Step 3:

The next step is to select the ready modelled mesh that represents a fuel injector used in two stroke engines. The geometry is provided with this tutorial which has the name spray1.x_t and is in parasolid format.



Step 4:

This step is to locate the required mesh to read into the designmodeller. This is done by going to file then from the drop down list selecting Import External Geometry file.



Step 5:

After the file has been selected press generate. This will read the previsouly modled geometry into Design Modeler.



Step 6:

The imported geometery box should look something like this.



Step 6:

Check that there is a green tick beside the Geometry icon. Then double click the mesh icon.

A Unsaved Project - Workbench	
File View Tools Units Help	-
🎦 New 💕 Open 🗟 Save 😹 Save As 👔 Import 🖗 Reconnect 😹 Refresh Pr	()
Toolbox 👻 🖗 🗙 Project Schematic	
Analysis Systems	
S Fluid Flow (CFX)	
💽 Fluid Flow (FLUENT)	
Component Systems I Component Systems I Fluid Flow (CFX)	
🕐 CFX 2 🕼 Geometry 🖌 🗸	
🥏 Engineering Data 3 🍘 Mesh 🖉	
🔅 External Connection 4 🙆 Setup	
Finite Element Modeler	
FLUENT 5 Viii Solution 7	
🤪 Geometry 6 😥 Results 🖓 🖌	
Mesh Fluid Flow (CFX)	
Microsoft Office Excel	

Step 7:

Postion the cursor on the mesh icon and then press the left mouse button and then press the right mouse button first select insert and then select Method.



Step 8:

When you click on the geometry box it will turn green the next step is to press the apply box.



Step 9:

After the apply button had been pressed the box should turn into a blueish purpule colour.



Step 10:

After pressing the update button a window at the bottom will show you the stage at which the Mesher packedge is in the meshgeneration process.



Step 11:

The generated mesh should look something like this. This method is the automatic method.



Step 12:

Check that the a green tick is shown beside the mesh icon. The next step is to double click on the setup icon.

File View Tools Units Help Image: New Open Save Save As Import
New @Open Save & Save As Toolbox Import Analysis Systems Analysis Systems Fluid Flow (CFX) Fluid Flow (CFX) Component Systems CFX Engineering Data External Connection Finite ElementModeler Fluid Flow (CFX) Seometry Secture Fluid Flow (CFX) Secture Mesh Microsoft OfficeExcel Sesults Fluid Flow (CFX)
Toolbox Project Schematic Analysis Systems Fluid Flow (CFX) Fluid Flow (FLUENT) Component Systems CFX Engineering Data External Connection Finite Element Modeler Fluid Flow (CFX) Seconetry Seconetry Seconetry Fluid Flow (CFX) Seconetry Fluid Secup Fluid Secup Fluid Flow (CFX) Secup Fluid Secup Fluid Flow (CFX) Fluid Flow (CFX) Fluid Flow (CFX) Fluid Flow (CFX) Fluid Flow (CFX)
 Analysis Systems Analysis Systems Fluid Flow (CFX) Fluid Flow (FLUENT) Component Systems CFX Engineering Data External Connection Finite Element Modeler FLUENT Geometry Solution Fluid Flow (CFX) Solution Fluid Flow (CFX)
Image: Second systems Image: Second systems <t< td=""></t<>
Image: Second systems Image: Second systems <t< td=""></t<>
 □ Component Systems 1 CFX ○ CFX ○ Engineering Data ○ External Connection ○ Finite ElementModeler ○ FLUENT ○ Geometry ○ Mesh ○ Results
[™] CFX ² Bigineering Data [™] External Connection [™] External Connection [™] Finite ElementModeler [™] Setup [™] Filuid Flow (CFX) [™] Fluid Flow (CFX)
 Engineering Data External Connection Finite ElementModeler FLUENT Geometry Mesh Microsoft OfficeExcel Results
Image: Setup Image: Setup
Image: Finite Element Modeler Image: Flue Flue Image: Flue
Image: Second control of the second control of th
Geometry Results Mesh Fluid Flow (CFX) Nicrosoft OfficeExcel Results
Wesh Fluid Flow (CFX) Microsoft OfficeExcel Image: Constraint of the second se
© Results
Nesalis
E Design Exploration
Parameters Correlation
Response Surface
Jix Sigma Analysis

Step 13:

Position the mouse curser on the default domain icon then press the mouse left handside button then press the mouse righthand side button and select edit.



Step 14:

You would notice that by default under the Fluid and Particle Definitions the name Fluid 1, just select it and press on the red X box, this will remove the default material name.

Edit Session In	sert Too	is He	p								
2 📽 🔩 📾	90	22	ġ.	8	0 :	x 🕜	2 seb	f.	0	3	0
tline Domain: Def	ult Domain										E
als of Default Doma	n in Flow	Analys	is 1								
Basic Settings Fluid	Models	Initiala	ation	1							
Location and Type											
Location	B251									•	
Domain Type	Fluid Dom	ain							_		く
Coordinate Frame	Coord 0									-	
Fluid and Particle Defin	itions										⊟
Fluid 1				4							-
											K
											_
Fluid 1										-	Ξ
Option	Matena	Library								•	
Material	Air at 2	5 C								- [
Morphology	_										Ξ
Option	Contin	uous Fl	uid							1	
										+	
Minimum Volur	e Fraction										
Minimum Volur Domain Models	e Fraction										
Domain Models Pressure	e Fraction										۵
Minimum Volur Domain Models Pressure Reference Pressure	1 [atm]										0
Minimum Volur Domain Models Pressure Reference Pressure Buoyancy Model	e Fraction										•
Minimum Volur Domain Models Pressure Reference Pressure Buoyancy Model Option	1 [atm]	yant								•	8
Minimum Volur Domain Models Pressure Reference Pressure Buoyancy Model Option Domain Motion	1 [atm]	yant								•	8
Minimum Volur Domain Models Pressure Reference Pressure Buoyancy Model Option Domain Motion Option	1 [atm] Non Buc	yant ry								•	8
Minimum Volur Domain Models Pressure Reference Pressure Buoyancy Model Option Domain Motion Option Mesh Deformation	1 [atm] Non Buc	yant ry								•	

Step 15:

Press on the file star icon pointed at by the arrow. This will enable us to add a new material name into the list. A window will open asking you to enter a name of a material, write AIR and press Ok.

File Edit Session	Insert Tools Help	<u>ی</u> ا ه
Outine Domain: Do Details of Default Dom	efault Domain ain in Flow Analysis 1	
Basic Settings Flu	id Models Initialization	
Location and Type		
Location	B251	
Domain Type	Fluid Domain	
Coordinate Frame	Coord 0	•
Fluid and Particle De	finitions	
		×
Domain Models		
Pressure Deference Pressure	1 [atm]	

Step 16:

Then again by pressing on the star file icon add another material called WATER and press Ok. Make sure under Morphology that the continous Fluid is selected this is for the case of AIR.

A4 : Fluid Flow (CFX) e Edit Session In	- CFX-Pre		
. 2 % % 0	a 🕸 🖬 🖉 🕗 🗛 📾 🛪 🤞 🕹 👍 🎬 🤊 🕫 🛙	oopt‰ ≿ A % ©	
Outline Domain: Def	fault Domain	*\$ \$.€€€@ @ □•	
tais of Default Doma	uin in Flow Analysis 1	View 1 ×	
Basic Settings Fluid	1 Models Initialization	\sim	
Location and Type			
Location	8251 • · · · ·		
Domain Type	Fluid Domain 👻		
Coordinate Frame	Coord 0		
Fluid and Particle Defin	nitions		
AIR			
	$\overline{\mathbf{x}}$		
		Insert Fluid Defini ?	
AIR			
Option	Material Library	ame WATER	
Material	At at 25 C	OK Cancel	
Material			
Ontion	Continuous Fluid		\frown
- Minimum Volur	me Fraction		()
			$\mathbf{\cup}$
Domain Models Pressure	B		
Reference Pressure	1 [atm]		
Buoyancy Model	E		
Option	Non Buoyant		
Domain Motion			
Ontion	Stationary		
Mesh Deformation			
Ontion	None		
opuon	110 m		
ОК Арр	Ay Close		

Step 17:

In the fluid and partical definitions select AIR then select from the drop down list in the material section Air at 25C. This is important becouse this step links the input name by the user to the thermodyanmics parameters data base in the software library.

A4 : Fluid Flow (CFX) -	- CFX-Pre	
e Edit Session Ins	sert Tools Help	
] 🛃 😤 🔩 📷	📫 🖬 🗟 👌 🗶 🚾 🖬 🔊 🕐	
Dutline Domain: Defa	ault Domain	
tails of Default Domai	in in Flow Analysis 1	
Basic Settings Fluid	Models Fluid Specific Models Fluid Pair Models Initialization	
Location and Type		
Location	B251 -	١
Domain Type	Ekid Domain	J
Condition to Forma		
Coordinate Frame		
Fluid and Particle Defini	itions	
AIR		
WATER	\mathbf{x}	
AIR		
Option	Material Library	
Material	Air at 25 C	
Morphology	Air Ideal Gas	
Option	Water	
I Minimum Volum	ne Fraction	
Denvelo Madela		
Pressure		
Reference Pressure	1 [atm]	
Rueuzagu Medel	- feenil	
buoyancy Model	Ner Devent	
Option	Ivon Buoyant	
Domain Motion		
Option	Stationary 👻	
Mesh Deformation		
Option	None	

Step 18:

The next step is to select in the fluid and particle definition WATER then from the material drop down list select Water again this is important becouse this step links the input name by the user to the thermodyanmics parameters data base in the software library.

Edit Session In	nsert Tools Help	
2 😤 🔩 🎯	j 🔊 🛯 🚰 🏚 🎂 🔶 🗴 🚾 🖬 🎜 🕑 🖉 🗖 📭	ð 🗊 🚺
utline Domain: Def	fault Domain	*> : :
ails of Default Doma i	in in Flow Analysis 1	View 1
asic Settings Fluid	Models Fluid Specific Models Fluid Pair Models Initialization	
Location and Type		
ocation	B251 👻 🛄	\sim
Iomain Type	Fluid Domain	
Coordinate Frame	Coord 0	
Eluid and Particle Defin	nitions	
WATER		
	×	
WATER		
Option	Material Library 👻	
Material	Air Ideal Gas	
Morphology	Air Ideal Gas	
Option	Water	
	me Fraction 🗄	
Domain Models		
Pressure		
Reference Pressure	1 [atm]	
Buoyancy Model	B	
Option	Non Buoyant 👻	
Domain Motion	8	
Option	Stationary	
Mesh Deformation		
Ontine	None	
	THORE V	

Step 19:

Having selected WATER under the Fluid and Partical Definition go to Morphology and select from the drop down list a dispersed fluid.

Edit Session Ir	nsert Tools Help	
1 🖓 😤 🛃 🔟	🤊 🕫 🚟 🎰 🕹 👌 🗴 🚾 🖬 🏞 🖸 🗃 🗖 🗱 🦣 🗄	•
utline Domain: Def	ault Domain 🛛 👘 🚓 🤄	2
ails of Default Doma	in in Flow Analysis 1	í
asic Settings Fluid	Models Fluid Specific Models Fluid Pair Models Initialization	
Location and Type		
ocation	B251	
iomain Type		
oordinate Frame	Coord 0	-
Fluid and Particle Defir	nitions	
AIR		
WATER		
WATER		
Ontion	Material Library	1
opaon	The sector of y	1
Material	Water	
Morphology		
Option	Continuous Fluid	
- Minimum Volur	me Fr Continuous Fluid	
	Dispersed Fluid	
Domain Models	Particle Transport Fluid	
Pressure	Particle Transport Solid	
Reference Pressure	1 Polydispersed Fluid	
Buoyancy Model	Droplets (Phase Change)	
Option	Non Buoyant	
Domain Motion		
Domain Mouon		
Option	Stationary	
Mesh Deformation		
Option	None	

Step 20:

Once dispersed fluid is selected the user is asked input a fluid partical mean diameter. Apply the mean droplet diameter of 0.5 micron. Then press Ok.

1 🖗 😤 🔩 🔟	🤊 🗞 🔤 🌣 🎂 🔶 🖉 👘	5 🗇 J‡	a o 🞜	
utline Domain: Def	ault Domain	×	*& S	
ails of Default Doma	in in Flow Analysis 1		View 1 💌	
Basic Settings Fluid	Models Fluid Specific Models Fluid Pair Models Initia	lization		
Location and Type	2254			
ocation	8251			
Jomain Type	Fluid Domain	-		
Coordinate Frame	Coord 0	•		
Fluid and Particle Defin	itions			
AIR				
WATER				(
Option	Material Library 🗸]		
Material	Water -			
Morphology		8		
Option	Dispersed Fluid 🗸			
Mean Diameter	0.5 [micron]			
	ne Fraction	ŧ		
I Maximum Pad	ing -	±.		
Domain Models				
Pressure				
Reference Pressure	1 [atm]			
Buoyancy Model		Ξ		
Option	Non Buoyant 🗸			
Domain Motion				
Option	Stationary -			
Mesh Deformation				
Option	None			
			110	

Step 21:

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the region numbered F211.251. Then right click and select the Rename option, this action will help us in applying a name to the inflow face.



Step 22:

After renaming the surface right click on the renamed region, go to Insert, then go to boundary and then under boundary select inlet.



Step 23:

Once the inlet option is selected go to boundary details and under Mass and Momentum select the Cart. Vel. Components. In the main view look at the coordiante system shown on the bottom right hand side this will clarfiy that the directions of the selected velocities is correct.

E	A4 : Fluid Flow (CF	X) - CFX-Pre		
F	ile Edit Session	Insert Tools Help		
****	📑 🕑 i 😤 🔩 i (🛯 🤊 🍋 🚰 💩 🧄 🏷 🚾 💀 🔊	🗇 J‡	
	Outline Boundary	: Inlet	×	
Ē	etails of Inlet in Defa	ult Domain in Flow Analysis 1		
	Basic Settings Bo	oundary Details Fluid Values Sources Plot Options		
	Flow Regime			
	Ontion	Subsonic		\cap
				\mathcal{A}
	Mass And Momentu	m		
	Option	Normal Speed 🗸		
	Normal Speed	Normal Speed		
	Normal Speed	Cart. Vel. Components		
	Turbulence	Cyl. Vel. Components		
		Bulk Mass Flow Rate		
	Option	Total Pressure (stable)		
		Static Pressure		
		Fluid Dependent		

Step 24:

Type the value of 30 into the U velocity input box, while for the V and W cell type in zero.

Under the Turbulnce option select High (Intensity=10%).

🔒 A4 : Fluid Flow (CFX) - (CFX-Pre
File Edit Session Inse	ert Tools Help
📑 👌 😤 🔩 🚳	🤊 🐑 🚰 🍈 🕹 🔶 🕱 🚾 🖬 🏂 🕙 🗃 📭
Outline Boundary: Inle	et 🛛
Details of Inlet in Default I	Domain in Flow Analysis 1
Basic Settings Bound	ary Details Fluid Values Sources Plot Options
Flow Regime	
Option	Subsonic
(
Mass And Momentum	
Option	Cart. Vel. Components
U	30 [m s^-1]
V	0 [m s^-1]
W	0 [m s^-1]
Turbulence	
Option	Medium (Intensity = 5%)
	Medium (Intensity = 5%)
	High (Intensity = 10%)
	Intensity and Length Scale
	Intensity and Eddy Viscosity Ratio
	k and Epsilon
	k and Eddy Viscosity Ratio
	k and Length Scale
	Default Intensity and Autocompute Length Scale
	Intensity and Auto Compute Length
	Zero Gradient
	Fluid Dependent 👻

Step 25:

In the boundary conditions section go to Fluid Values and select AIR then in the volume fraction section input zero as.

A4 : Fluid Flow (CFX) - (CFX-Pre		
ile Edit Session Inse	rt Tools Help		
📑 🗿 😤 🔩 🔟	🤊 🐑 🚰 🍰 🕹 🔶	X 🚾 홰 🏂 🕓 🛃	🗇 j:
Outline Boundary: Inle	t		×
etails of Inlet in Default I	Domain in Flow Analysis 1		
Basic Settings Bounda	ary Details Fluid Values	Sources Plot Options	
Boundary Conditions			
AIR			
WATER			
AIR Volume Erection			
Volume Praction			
Option	Value	•	
Volume Fraction	0	K	

Step 26:

The next step is to go again to Fluid Values and select WATER under boundary conditions. Then apply 1 as the volume fraction for WATER.

Edit Session In	sert Tools H	elp			
👌 🚔 🛃 🔟	9 🛯 😤	ja 🕹 🌔) 🗴 🚾	500 fx 🕓	🛃 🗇 j
tline Boundary: In	let				E
ils of Inlet in Defaul	t Domain in Flov	v Analysis 1			
asic Settings Bour	dary Details	Fluid Values	Sources	Plot Options	
Boundary Conditions					
AIR					
WATER				K	
WATER					\frown
Volume Fraction				/	
Option	Value				•
Volume Fraction	1		K		

Step 27:

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the five surfaces having the names of F265.251, F267.251, F268.251, F269.251 and 270.251, the go to Insert, then go to boundary and then under boundary select Opening.



Step 28:

Type into the relative pressure cell a pressure value of 101325, this would apply atmosphric conditions to the spray domain.

44 : Fluid Flow (CFX) - CFX-Pre					
Edit Session	🔟 🤊 🥐 😤 🍰 👌 🛠 🚾 🖬 🖍 🕓 🗃 🖬					
utline Bound	ary: opening					
als of opening i Basic Settings	n Default Domain in Flow Analysis 1 Boundary Details Fluid Values Sources Plot Options					
Flow Regime						
Option	Subsonic 👻					
Mass And Mome	ntum 📃					
Option	Opening Pres. and Dirn					
Relative Pressure	e 101325 [Pa]					
Flow Direction						
Option	Normal to Boundary Condition 👻					
Loss Coeffic	ient 🗉					
Turbulence						
Option	Medium (Intensity = 5%)					

Step 29:

In the boundary conditions section go to Fluid Values and select AIR then in the volume fraction section input a value of 0.9.

A4 : Fluid Flow (CFX)	- CFX-Pre	100 million (1990)					
File Edit Session Ir	isert Tools Help						
🚽 🔮 😤 🔩 🚳	🔊 陀 🚰 🍰	👌 % 🚾 🐽 丸 🕓	gi 🗇 🕽 🗱				
Outline Boundary: o	pening		-				
Details of opening in Def	ault Domain in Flow Analy	sis 1					
Basic Settings Bour	dary Details Fluid Values	Sources Plot Options					
Boundary Conditions							
AIR							
WATER							
AIR							
volume Fraction	G						
Option	Option Value						
Volume Fraction	Volume Fraction 0.9						

Step 30:

The next step is to go again to Fluid Values and select WATER under boundary conditions. Then apply 0. 1 as the volume fraction for WATER.

🖽 A	4 : Flui	id Flow	(CFX) - C	FX-Pre														
File	Edit	Sessio	on Inse	t Too	s He	р												
	1	💦 🖳	01	9 0		ja 👌	5 👌	X	Vac sub] <i>f</i> ≈	0 💈	(1]‡ 🖻	1 🗊	t:	0 🏡	₿ ∎	
0	utline	Bound	dary: F26	5 251 1]				_								×	*12
Deta	ails of F	265 25	1 1 in De	ault Do	maini	n Flow A	nalys	sis 1	\frown									Viev
E	asic Se	ttings	Bounda	y Detail	s Fl	uid Value	s i	source	es P	lot Opti	ons							
	Bounda	ary Cond	itions														-8-	
	AIR			K														
	WAT	TER		-														
	WATE	R															—(
	- Volun	ne Fracti	ion														F	
	Optio	n		Value											/	-		
	Volum	ne Fractio	on	0.1									K					
1 C																	=	

Step 31:

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the rest of the regions which havent been selected before, then go to Insert, then go to boundary and then under boundary select wall.



Step 32:

The next step is to check that there is a green tick beside the setup icon then double click on solution.



Step 33:

The final step is press on Start Run and the calculation should start.



Step 34:

The solver should show you something like this. You can press stop after a number of time steps then check if the required results fall into the wanted ranges. In this case I will press stop you can go back and continue the simulation from the stoped point.



Step 35:

The user can see a pause and a green tick mark beside the solution icon by double clicking on the results icon the researcher can proceed to the data analysis part.

\Lambda spray2 - Workbench	
File View Tools Units Help	
🎦 New 对 Open 📙 Save 🔣 Save As	👔 Import 🛛 🍣 Reconnect 🛛 🥰 Refresh Project 🗲 U
Toolbox 🔻 🕂 🗙	Project Schematic
Analysis Systems	
S Fluid Flow (CFX)	
🔇 Fluid Flow (FLUENT)	▼ A
Component Systems	1 S Fluid Flow (CFX)
CFX	2 🕅 Geometry 🗸
🥏 Engineering Data	3 🍘 Mesh 🗸
🔆 External Connection	4 🏟 Setup
Finite Element Modeler	5 Solution "
FLUENT	
Geometry	o 😰 Results 🗸 🛧
Misson Charles Turnel	Fluid Flow (CFX)
Goal Driven Optimization	
Response Surface	
Six Sigma Analysis	

Step 37:

After pressing the apply button you should see a cross sectional plane (grey in color) running through the domain in the xy plane.



Step 36:

Left click the cursor on the User Locatins and Plots icon, then go to insert then locations and select plane, a window will open with a plane default name of plane 1 higlighted in blue changing its name is upto the user.



Step 37:

Left click the cursor on the User Locatins and Plots icon, then go to insert then from the drop down list select Contour, a window will open with a plane default name of contour 1 higlighted in blue changing its name is upto the user.



Step 38:

A new side window will open which has the details of the created plane, go to locations and chose Plane1 from the drop down list. Then go to (# of Contours) and enter into the input cell 100.



Step 39:

The next step is to select the Water.Velocity Variable from the drop down list, then go to the Range and select from the drop down list User Specfied.When user specfied is selected two new options are added the minimum and the maximum enter a minimum velocity value of zero and a maximum velocity value 0.9. Finally press apply and what would be visibile the velocity profile of water. The user has required knowlagde now to continue on his own.

