

# Flow Modelling in Heat-Exchangers using ANSYS CFX

# By Ahmed Al Makky

<sup>@</sup>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.

# Flow Modelling in Heat-Exchangers 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 flow modelling side of the problem. Heat exchangers are encountered in lots of engineering applications, in cars , in planes , in home boilers, ....etc.

#### Step 1:

Double click on Fluid Flow (CFX) under the Analysis sytems, then double click on the Fluid Flow (CFX) under A.



#### Step 2:

A window will open up asking you at what length scales do you want to work on, click on centimeter tick box and click Ok.



#### Step 3:

Go to File and select Import External Geometry file.



#### Step 4:

A window will open up click on the file called heatexchanger.x\_t and then press open.



# Step 5:

Press the genrate button and what you will see is the read in geometry into design modeller.

A: Fluid Flow (CFX) - D	esignModeler		the local data of the second data		and the second se
File Create Concept	Tools View Help				
🄊 🗖 🗐 👩 🗍 s	Dilada @Reda Select: * N - R R R R	C- 1 • • • • • • • • • • • • • • • • • •			
	/3* /X*				
XYPlane 💌 ⊁	None 👻 🎽 Generate 👘 Share Lopology	Karling Extrude 🙀 Revolve 🕵 Sweep 🐇 Skin/Loft 💽 Thin/Sur	rface 💊 Blend 👻 💊 Chamfer 🛷 Point 📴 P	arameters	
Tree Outline	+	Graphics			
A Fluid Flow (C VPlane - XPlane - XPlan	FQ lodies			0	
Skatching Markeline					
Modeling					
Details View	÷				
Details of Import1	Townshit				
Import	Imports E\beatexbanger\beatexcbanger v t				
Base Plane	XYPlane		0		
Operation	Add Material				
Process Solid Bodies	Yac				
Process Surface Bodies	Yes				
Process Line Bodies	No				
Simplify Geometry?	No				
Simplify Topology?	No				
Clean Bodies?	Vac				
Refresh	No		0.000		10.000 ()
Neirean			0.000	5.00	10.000 (cm)
			:	2.500 7.50	0
		Model View Print Preview			

#### Step 6:

Postion the cursour on the icon (3 Parts, 3Bodies) and then press the left button on it a subtree of three directore would occur. Position the cursour on the the last two solids ( doing each solid seperatly) and then click the left mouse button and select the suppress body option.

🕅 A: Fluid Flow (CFX) - DesignModeler		Modeler
File Create Concept Tools View Help		s View Help
🔄 🛃 🛃 📫 🕽 Oundo @Redo 🛛 Select: 🍬 🍡	r 🖻 🖻 💽 💽	io @Redo    Select: 🆎 🖏 🖷 🔃 💽 🚺 🔓
. h+ h+ h+ h+ h+		/x-
🛛 XYPlane 🔹 🛧 🛛 None 👻 ಶ 🗍 🧚 Gener	ate 🛯 🗑 Share Topology	ne 🚽 📁 🗍 🧚 Generate 🛛 🕅 Share Topology
Tree Outline	<b>4</b>	4
<ul> <li>→ A: Fluid Flow (CFX)</li> <li>→ XYPlane</li> <li>→ YZPlane</li> <li>→ YZPlane</li> <li>→ Import1</li> <li>→ Import1</li> <li>→ Solid</li> <li>→ Soli</li></ul>		ide Body uppress Body nsuppress All Bodies vert Suppressed Body Set enerate ename

#### Step 7:

The need geometry should look like this. Notice that there is a green tick near the solid meaning the soild is visible while the two suppressed solids have a green x sign beside it.



#### Step 8:

Once you see there is a green tick sign beside geometry, double click on the Mesh icon.



# Step 9:

Position the cursor on the Mesh icon and press on the right button go to insert and select Method.

D	A : Fluid Flow (CF	X) - Meshing [ANSYS Academ	Teaching CFD]
	File Edit View	Units Tools Help 🗍 誟 🤅	nerate Mesh  🌃 \Lambda 🞯 🕶 🇊 Worksheet 🗍 📽 🦎 🖏 🐨 🔃 💽 💽 😵 😵 🏵
N	vlesh 🗦 Update	🍘 Mesh 🔻 🔍 Mesh Cont	I ▼   ,   Metric Graph   \$ Dotions
Dı	utline		Ф
	Project	b) etry inate Systems  Insert  Update  Generate Mesh  Preview Show  Create Pinch Controls  Clear Generated Data b Rename	Mesh 13/12/2012 20:41
De	etails of "Mesh"		
-	Defaults		
	Physics Preference	CFD	
	Solver Preference	CFX	
_	Relevance	0	
+	Sizing		
+	Advanced		
т -	Defeaturing		\         \         \
-	Statistics		
+	Defeaturing Statistics		

#### **Step 10:**

Position the cursor on the geometry then press the left mouse button to select the geometry. As a result the geometry turns green. The next step then to press apply.



#### **Step 11:**

The geometry turns to a light purple color.



#### **Step 12:**

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 13:

By pressing the left button onto the Mesh Icon the generated mesh should look something like this.

File Edit View Units Tools Help 🛛 💈 Generate Mesh	† 🖥 \Lambda 🖉 – 🕅 Worksheet 🛛 🖤 👫 🖡 🕞 🕞 🦝 😪 – 🗠
Mech 🛃 Indate   🕅 Mech 👻 🕅 Mech Control 💌 🖟 Meth	
wesh yopuate   topimesh to the mesh control to	
Outline ·	Mesh
Model (A3)	13/12/2012 21:09
E Coordinate Systems	
Addumatic Method	
	APPENDIX SECONDARY SECONDARY S
Details of "Mesh"	
Defaults	
Physics Preference CFD	
Solver Preference CFX	
Relevance 0	
+ Sizing	
Advanced	
+ Defeaturing	
Statistics	

#### Step 14:

Check that there is a green tick sign beside the Mesh icon then double click on the Setup icon.



#### Step 15:

The CFX pre once it is lunched should look something like this.



#### **Step 16:**

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the region numbered F51.50. Then right click and select the Rename option, this action will help us in applying a name to the inflow face.



#### Step 17:

Select the renamed surface called inlet and the right click the mouse button and select Insert, then boundary then Inlet.



#### **Step 18:**

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 coordinate system shown on the bottom right hand side this will clarify that the directions of the selected velocities is correct.

<b>B</b> /	A4 : Fluid Flow (CFX)	- CFX-Pre								
File	e Edit Session I	nsert Tools	Help							
	] 🖸 😤 🔩 🗉	🤊 😋   🖁	🚡 🧄 🖁	ີ 👌 🗴	Vac Sub	f≈	🕓 💆	í 🗇 拜		
C	Dutline Boundary: 1	inlet					×	1 *D		
Det	tails of <b>Inlet</b> in <b>Defau</b>	lt Domain in F	low Analys	is 1				View 1		
	Basic Settings Bou	ndary Details	Sources	Plot Opt	ions					
	Flow Regime									
	Option	Subsonic				•				
	Mass And Momentum						8		$\sim$	•
	Option	Normal Spee	d			-				)
	Normal Speed	Normal Spee	d						$\smile$	
	Tabulaasa	Cart. Vel. Co Cvl. Vel. Cor	mponents ponents				_			
	lurbuience	Mass Flow R	ate			- 1				
	Option	Total Pressu	re (stable)			- 1				
		Static Press	ire			_				

# Step 19:

Enter -10 into the V velocity cell, and enter the value of zero to the U and W cell, then press apply and Ok.

🛞 A4 : Fluid Flow (CFX)	- CFX-Pre		
File Edit Session I	insert Tools Help		
🔄 🛃 🔯	1 🔊 🔍 🚟 🍺 🐷 👌 🗶 🚾 🖬	0 0	í 🗇 🗰 📾 💋 🐜 🖓 ≌ 🔖
Outline Boundary:	Inlet	×	*\; \$
Details of Inlet in Defau	Ilt Domain in Flow Analysis 1		View 1 🔻
Basic Settings Bou	indary Details Sources Plot Options		
Flow Regime			
Option	Subsonic		
Mass And Momentum		Ξ	
Option	Cart. Vel. Components 🗸		
U	0 [m s^-1]		
v	-10 [m s^-1]		
w	0 [m s^-1]		
Turbulence		Ξ	
Option	Medium (Intensity = 5%) 👻		
			U
	_		
	$\frown$		
	$\sim$		
		1	
OK AD	ply Close		

#### **Step 20:**

Go to the CFX.cmdb and from the drop down list for the Principial 2D Regions select the region numbered F60.50. Then right click and select the Rename option, this action will help us in applying a name to the outflow face.



#### **Step 21:**

Select the renamed surface called outlet and the right click the mouse button and select Insert, then boundary then select outlet.



#### **Step 22:**

Under boundary Details, Under the Mass and Momentum go to option and select from the drop down list Average Static Pressure, then enter the value of 111135 value into the Relative Pressure cell, then press apply then Ok.

The value is calculated  $P = \rho gh + P_{atm} = 1000 \times 9.81 \times 1 + 101325 = 111135$  Pa



#### Step 23:

Select the rest of the surfaces then then right click, go to insert ,then go to boundary, then select Wall.



#### **Step 24:**

Enter in the name cell: WALL and press Ok.



#### Step 25:

The selected surfaces are highligted in green press Apply then press Ok.



#### **Step 26:**

Position the cursor over the Default Doman then select it using the left mouse button. Right click and then select the Edit option.



# Step 27:

Go to materials select from the drop down list Water, then press apply, then press Ok.

A4 : Fluid Flow (CFX)	- CFX-Pre		
File Edit Session Ir	nsert Tools Help		
- 0 2 2 1	o 🛯 🚰 🍐 👌 🖉 🚾 🖬 🕫 🕑 🖻	<b>] ] # # # # [</b> ] [] [] [] [] [] [] [] [] [] [] [] [] []	f "i 6
Outline Domain: Def	ault Domain	*	2
Details of Default Doma	in in Flow Analysis 1	yiew 1 x	
Basic Settings Fluid	Models Initialization	VICU I	1 A
Location and Type			- i f i f i f i f
Location	B50 👻		
Domain Type	Fluid Domain 👻		
Coordinate Frame	Coord 0		
Fluid and Particle Defin	nitions		
Fluid 1		6	
		S	
			WHICH AND A REAL AND A
Fluid 1		$\sim$	
Option	Material Library 👻		
Material	Air at 25 C 🗾 💽		
Morphology	Air Ideal Gas		
Option	Water		
Minimum Volur	ne Fraction		
Domain Models			
Pressure			
Reference Pressure	1 [atm]		
Buoyancy Model			
Option	Non Buoyant 👻		
Domain Motion			
Option	Stationary 👻		
Mesh Deformation			
Option	None 🔻		
	$\frown$		
	$\gamma$		
	$\frown$		
OK Ann	Close		
		L	

# Step 28:

Check there is a green tick sign beside Setup, then double clip on the Solution icon.

File View Tools Units Help
🎦 New 🚰 Open 层 Save 📓 Save As 👔 Import 🚳 Reconnect 义 Refresh Project 🚽
Toolbox 👻 👎 🗙 Project Schematic
Analysis Systems
C Fluid Flow (CFX)
C Fluid Flow (FLUENT)
Component Systems  I Gruid Flow (CFX)
🕘 CFX 2 🛍 Geometry 🗸
🥏 Engineering Data 3 🍘 Mesh 🗸 🖌
🔆 External Connection 4 🎲 Setup
Finite Element Modeler
Geometry
Wiesh Fluid Flow (CFX)
Barameters Correlation
In Six Sigma Analysis

### Step 29:

Select the HP MPI Local Parallel, this will enable the user to conduct a parallel core calculation on the desktop he is using, then press on Start Run.

🔞 Define Run		? ×	
Solver Input File	0\CFX\CFX\Fluid Flow CFX.def	1	
Global Run Settings			
Run Definition			
Initialization Option	Current Solution Data (if possit ication	ble 👻	
Type of Run	Full	<b>-</b>	
Double Precision Parallel Environment			~
Run Mode	Serial	•	-0
	HP MPI Local Parallel	-	
F211-02	HP MPI Distributed Parallel MPICH2 Local Parallel		
	MPICH2 Distributed Parallel		
Show Advanced Cont	trols		
Start Run Save Settin	lgs	Cancel	

#### **Step 30:**

Once the calculation Kicks off, the user can follow up the progress of his calculation through the CFX-Solver Manager window. The calculation is setup by default to stp after 100 iterations or if reaches its cut-off criteria.



# Step 31:

Check there is a green tick beside the Solution icon, then double click on the Results icon.

🐧 Unsaved Project - Workbench	
File View Tools Units Help	
🎦 New 💕 Open 层 Save  Save As	Import 🛛 🖓 Reconnect 🛛 😹 Refresh Project
Toolbox 🝷 🕈 🗙	Project Schematic
Analysis Systems	
S Fluid Flow (CFX)	
🔄 Fluid Flow (FLUENT)	▼ A
Component Systems	1 🖸 Fluid Flow (CFX)
CFX CFX	2 🕅 Geometry 🗸 🖌
🦪 Engineering Data	3 🎯 Mesh 🗸 🖌
External Connection	4 🎡 Setup 🗸
Finite Element Modeler	5 📓 Solution
Geometry	V V Results
Microsoft Office Excel	Fluid Flow (CFX)
Results	
Design Exploration	
Goal Driven Optimization	
Parameters Correlation	
Response Surface	
Six Sigma Analysis	

#### Step 32:

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



#### Step 33:

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



#### Step 34:

Left click the cursor on the User Locations and Plots icon, then go to insert then locations and select contour,



#### Step 35:

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 36:

The next step is to select the Velocity Variable from the drop down list, then go to Range and select from the drop down list Local. Finally press apply and what would be visible the velocity profile of water. The user has required knowledge now to continue on his own.

