Flow Modelling in a Porous-Fluid Domain using ANSYS CFX

First Edition

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.

Flow Modelling in a Porous-Fluid Domain using ANSYS CFX

Introduction

The tutorial was written in a rush so it has spelling mistakes never got 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. Porous fluid problems are encountered in lots of engineering applications, in soils , in food , in filters,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 FluidSolid.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, for our studied case we have two domains one for the porous domain and the other is for the gas.



Step 6:

Postion the cursour on the icon (2 Parts, 2 Bodies) and then press the left button on it a subtree of three directore would occur. Position the cursour on the last two solids (doing each solid seperatly) and then click the left mouse button and select one of the solids. Then go to the Details of boday and select Solid, as you can see the slelected body is colored yellow.



Step 7:

Select the second solid which turns into a yellow color go to Details of body and select Fluid. You have now finished with design modeller our next step is to generate the mesh.



Step 8:

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



Step 9:

Click on the geometry icon, you will get to solids in the sub tree, click on the solid and right click and rename the solid to Fluid, to check that you have selected the right domain look at the highlighted box.



Step 10:

Click on the geometry icon, select the other solid and rename the solid to Porous (representing the porous domain), to check that you have selected the right domain look at the highlighted box.



Step 11:

Click on the connections icon, then select the contact region, the contact region is in red, you can see under the details of 'Contact Region' that the contact Bodies for Porous is highlighted in pinkish red and the target Bodies is highlighted in purple .



Step 12:

Under geometry click on the fluid icon and select suppress body from the drop down list.



Step 13:

Click on the mesh icon, then select insert, then select method.



Step 14:

Select the box which should turn into a green colour, then click on the apply button.



Step 15:

To check that the geometry has been correctly selected the domain should turn into a purple colour.



Step 16:

Select Tetrahedrons as a meshing method, then press update.



Step 17:

Click on the mesh icon and you can see the generated mesh. We are done now in generating the porous domain mesh.



Step 18:

We now want to generate the fluid mesh domain. Select the fluid icon under geometry and right click the mouse button and select Unsuppress Body.



Step 19:

Select Mesh then right click mouse button then select insert then Method.



Step 20:

Select the Fluid box which should turn into a green colour, then click on the apply button.



Step 21:

To check that the geometry has been correctly selected the domain should turn into a purple colour.



Step 22:

Select Tetrahedrons as a meshing method, then press update.



Step 23:

After pressing Update button you will see a window indicating the meshing process.



Step 24:

The genrated mesh for both domains should look somthing similair to what is shown below.



Step 25:

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

icon.

A Userved Desiret - Weyldsmeth			
N Unsaved Project - Workbench			
File View Tools Units Help			
🎦 New 对 Open 🛃 Save 🔣 Save As	👔 Import	∉φ Reconnect	Refresh Project
Toolbox 🝷 🕂 🗙	Project Schemat	ic	
Analysis Systems			
S Fluid Flow (CFX)			
G Fluid Flow (FLUENT)	-	A	
Component Systems	1 🕃	Fluid Flow (CFX)
CFX CFX	2 🕅	Geometry	× .
Engineering Data	3 🍘	Mesh	× .
🔆 External Connection	4 🚵	Setup	2
Finite Element Modeler	- 445	C-Luker	<u> </u>
FLUENT	> 👊	Solution	<u> </u>
🥪 Geometry	6 💓	Results	P 🖌
🍘 Mesh		Fluid Flow (CFX)
Microsoft Office Excel			
Results			
Design Exploration			
Goal Driven Optimization			
Parameters Correlation			
Response Surface			
Six Sigma Analysis			
	1		

Step 26:

Go to principal 3D Region and select the porous domain region (The mesh highlighted in green) right click mouse button and select rename, assign it a name of Porous .



Step 27:

Go to principal 3D Region and select the Fluid domain region (The mesh highlighted in green) right click mouse button and select rename, assign it a name of Fluid .



Step 28:

We need now to add the time stepping option.



Step 29:

Enter 10 seconds for the total time, then enter 1 second for the time step and finally enter 0 second as initial time. Press the apply button and press ok.

4 : Fluid Flow (CF Edit Session	X)-CFX-Pre Insert Tools Help 通 🤊 🍽 🎬 🍻 🐷 🔌 🗴 🕢 페 🏞 🕓 🗭 🗇 輝 マロッグ 🎶 🐜 🎒	
tline Analysis	ype 🛛	*13
is of Analysis Ty	pe in Flow Analysis 1	View
vternal Solver Co		
ation	None	
ashusia Tuna	TORE	
tion	Transient	
ime Duration		
ption	Total Time	
tal Tima	10 [6]	
	10 [2]	
ine steps		
ption	Imesteps	
mesteps	1[5]	
nitial Time		
ption	Automatic with Value	
ime	0 [5]	
		 Aub Aub
	0	

Step 30:

Right click the mouse button and select edit.



Step 31:

Right click the mouse button and select edit.

tails of Output	itput Con	trol	vsis 1				E
Results Ba	ackup	Trn Result	Trn Stats	Monitor	Export		
Option		Standard					•
File Compressio	on	Default					•
Output E	Equation I	Residuals					Đ
Extra Ou	utput Vari	ables List					Ŧ
Extra Ou	utput Vari	ables List					Œ
Step 32:

Under the Output Frequency enter 1 in the Timestep Interval. Press the Apply button and then press Ok.

A4 : Fluid Flow (CFX) -	CFX-Pre	
ile Edit Session Ins	ert Tools Help	
🚽 🔮 🕰 📷	🔊 🕫 🚰 🎄 💩 👌 🗴 📾 👧 🖉 🗃 🖓 🗤 🖄 👘	P
Outline Output Contro	ol	×
Details of Output Control	in Flow Analysis 1	=
Results Backup	Irn Results Trn Stats Monitor Export	
Transient Results		
Transient Results 1		
Transient Results 1	8	
Option	Standard 👻	
File Compression	Default	
Output Equation	Residuals 🕀	
Extra Output Va	riables List	\sim
Output Frequency		
Option	Timestep Interval 👻	
Timestep Interval	1	
	9	

Step 33:

Select the four highlighted domains. Then press delete. Because we need to add new domains

for the simulation.



Step 34:

Call the new domain Porous and press Ok.



Step 35:

Call the new domain Porous and press Ok.



Step 36:

Press on the button beside the location selection, a window will open up select Fluid and press Ok. Finally press the red x sign to rename the Fluid 1 to Air. Then press the icon that has a document with a yellow star.

A4 : Fluid Flow (CFX)	- CFX-Pre		
File Edit Session I	nsert Tools Help		
📕 🖸 😤 🔩 🗉) 🤊 😋 🗳 🍦 🕹 🔶 🗶 📾 🗛 💿 🗃 🖊 📾 🛱 🥌 🗳 🕐 🔍 🛤	a 🖻	
Outline Domain: Flu	id	X (\\.₽
Details of Fluid in Flow A	analysis 1		View 1 •
Basic Settings Flui	d Models Initialization		
Location and Type		4	Selection Dialog
Location	Fluid 👻 🛄		
Domain Type	Fluid Domain 🔹		Fluid Porquis
Coordinate Frame	Coord 0 🔹		Assembly
Fluid and Particle Defi	nitions		
Fluid 1		←	
Fluid 1			
Option	Material Library		
Material	Air at 25 C 👻 🛄		
Morphology			
Option	Continuous Fluid		
Minimum Volu	me Fraction 🕀		
Domain Models			
Pressure	8		2363856
Reference Pressure	1 [atm]		
Buoyancy Model			
Option	Non Buoyant 👻		
Domain Motion			
Option	Stationary 👻		
Mesh Deformation			
Option	None		
			OK Cancel
			0.02

Step 37:

The next step is to enter in the Name section Air and then press OK, then press apply and then press OK.

A4 : Fluid Flow (CFX)	- CFX-Pre
le Edit Session II	nsert Tools Help
etails of Fluid in Flow A	nalysis 1
Basic Settings Fluid	I Models Initialization
Location and Type	
Location	Fluid 🗸 🛄
Domain Type	Fluid Domain 🔹
Coordinate Frame	Coord 0 🗸
Fluid and Particle Defi	nitions
	X
Domain Models	
Pressure	Θ
Reference Pressure	1 [atm]
Buoyancy Model	Θ
Option	Non Buoyant 🔹
Domain Motion	8
Option	Stationary 👻
Mesh Deformation	
Option	None
	Insert Fluid Defini R
OK ADD	IV Close

Step 38:

Select Flow Analysis and then right click the mouse button go to insert and select Domain.



Step 39:

Type in the Name Porous and press OK.



Step 40:

Press on the button beside the location selection, a window will open up select Porous and

press Ok.

A4 : Fluid Flow (CFX)	- CFX-Pre		
File Edit Session Ir	nsert Tools Help	- 12	
	- 今 📲 声 🖉 👌 🗴 🔤 🖬 🗴 🕙 🖉 🍎 🚰 🥱 🔊 🤊		
Outline Domain: Por Details of Porous in Flow	ous E V Analysis 1	X : : : : : : : : : : : : : : : : : : :	
Location and Type			
Location	Porous		
Demain Turne	Fluid Domain	Selection Dialog	
Domain Type		Fluid	
Coordinate Frame		Porous Assembly	
Fluid and Particle Defi	nitions		
Air			
Air			
Option	Material Library		
option M 1 - 1			
Material	Alfat25C		
Morphology			
Uption			
Domain Models			
Pressure			
Reference Pressure	1 [atm]		
Buoyancy Model			
Option	Non Buoyant 👻		
Domain Motion	8		
Option	Stationary 🗸		
Mesh Deformation	8		
Option	None 👻		
		OK Cancel	

Step 41:

Select the porous domain right click the mouse button and select edit, then go to Basic Settings and select Porous Domain.

Is of Porous in Flow	v Analysis 1	Image: Second state Image: Second state	
isic Settings Fluid	d Models Initialization		
cation and Type	Parque		
	Ekid Domain		
inain type	Fluid Domain		
ordinate Frame	Solid Domain		
luid and Particle Defin	Immersed Solid		
Fluid 1	No and an and a design of		
		×	
			~
Fluid 1		8	
Option	Material Library		
Material Science	Aliceh DE C		
Material	All al 25 C		
Morphology	Continues Child		K
Option	Continuous Pluid	*	
	merracion		
omain Models			
Pressure			
leference Pressure	1 [atm]		
Buoyancy Model			
Option	Non Buoyant	•	
Domain Motion		8	
Option	Stationary	•	
Mesh Deformation	8 	8	
Option	None	•	
- Second	Lawrence and the second s		

Step 42:

The next step is to go to Basic Settings and click on the remove selected item, next comes

clicking on the new item icon.

ile Edit Session Insert To Edit Session Insert To Edit Session Insert To Cutine Domain: Porous Details of Porous in Flow Analysi	∞s Hep 월 승 집 ♦ ૹ ፼ 제 A @ 정 영 # # 8 9 11 10 11 10 10 10 10 10 10 10 10 10 10	
Outline Domain: Porous Details of Porous in Flow Analysi		
Outline Domain: Porous Details of Porous in Flow Analysi		
Details of Porous in Flow Analysi	() () () () () () () () () ()	
	is 1	
Basic Settings Fluid Models	Perosity Settings Initialization	View I *
Location and Type	Tendercy occurrige annumacions	
Location Porous	•	
Damain Time	Damin	
contain type		
Coordinate Frame Coord U	•	
Fluid and Particle Definitions		
Fluid 1		
Elizad 1		move selected item
Option Mater	rial literary	
Option	• •	
Material Air at	*	
Morphology		
Option Con	tinuous Fluid	
Minimum Volume Fractio	n 🗄	
Solid Definitions	8	
-		
Domain Models		
Pressure		
Reference Pressure 1 [atm	1	
Buoyancy Model	E	
Option Non B	uoyant 👻	
Domain Motion		
Option Station	nary 🔹	a
Mesh Deformation	E	0 0.00 0.00 (m)
		0.025 0.075
Option None	•	

Step 43:

Enter in the name cell: Air and press Ok. Then press the apply button.

A4 : Fluid Flow (CFX) File Edit Session In Cutline Domain: Por Details of Porous in Flow	- CF2-Fre vert Tools Help 1 9 0 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	L A = L S ↔ Q Q Q D - 1 View 1 +
Basic Settings Fluid Location and Type Location Domain Type Coordinate Frame Fluid and Particle Defin	Models Porosity Settings Initialization Porous Porous Domain Coord 0 Ordinations Operation of the set o	B Insert Fluid Defini 2 Name AIR
Solid Definitions Domain Models Pressure Reference Pressure Buoyancy Model Option	1 [atm] Non Buoyant	
Domain Motion Option Mesh Deformation Option	Stationary .	
OK and	N Close	

Step 44:

Ignore for now the caution sign shown in red, go to Solid Definition and press the material assign icon.

A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session	Insert Tools Help	
	1 7 C 📽 🕹 🕹 🖇 🖉 🖬 🖈 🖸 🗃 🖬 - 🖻 - 🗗 - 🗗 - 🗗 - 🗗 - 🗗 - 🗗 - 🗗	
Outline Domain: Pr	VOIR	
Details of Porous in Flo	w Analysis 1	
Basic Settings Flu	id Models Parasity Settings Initialization	view 1 *
Location and Type		
Location	Porous +	
Domain Tune	Poraus Domain	
Conditi Type	Courd 0	
Coordinate Frame	COOR 0	
Fluid and Particle De	finitions	
AIR		
	×	
AIR	8	
Ontion	Material Library	
opton		
Material	Ar at 25 C	
Morphology	Commenter and the second se	
Option		
Minimum voi	ume Fraction La	
Solid Definitions		
Pressure	8	
Reference Pressure	1 [atm]	
Buoyancy Model	E	
Option	Non Buovant	
Domain Motion	R	
Option	Stationary	
March Daformation		0 0.050 0.100 (m)
Option	None	0.025 0.075
opuon	TININ	
		Inc parameter Location in JPLUVI/How Analysis 1/DOMAIN:Default Domain/BOUNDARY:Pud* holds the following disallowed values: T-19.18/F20.18/F21.18/F23.18/F24.18* (Allowed Values) T-19.18/F20.18/F21.18/F24.18* (Allowed Values) T-19.18/F20.18/F21.18/F20.18/F21.18/F20.18* (Allowed Values) T-19.18/F20.18/F21.18/F20.18* (Allowed Values) T-19.18*
OK Ap	Close	

Step 45:

Enter the name wood and press Ok. The material names will be added to the list.

A4 : Fluid Flow (CFX)) - CFX-Pre	
File Edit Session 1		
Outline Domain: Po	rous	≟ № S∻QQQ @ □- 1a
Paris Cattions		View 1 +
Leastien and Time	a Models Porosity Setangs Initialization	
Location	Paraus	
Demain Turne	Perce at Domain	
Condente Erama	Coord 0	
Coordinate Frame		
Hud and Parade Der		
AIR		
	×	
AIR	B	
Option	Material Library 👻	
Material	Air at 25 C	
Morphology	8	
Option	Continuous Fluid 👻	
Minimum Volu	ume Fraction 🗉	
Solid Definitions	8	
	1	B) Insert Solid Defini.
	9	Name Wood
Domain Models		
Pressure		
Reference Pressure	1 [atm]	
Buoyancy Model		
Option	Non Buoyant 👻	\mathbf{i}
Domain Motion		
Option	Stationary 💌	0 0.050 0.100 (m)
Mesh Deformation		0.025 0.075
Option	None	
		S The parameter "Location" in "/FLOW:Flow Analysis 1/DOMAIN:Default Domain/BOUNDARY:Fluid" holds the following disallowed values: "F19.18,F20.18,F21.18,F23.18,F24.18". (Allo
ОК Ар	ply Close	

Step 46:

Press on the material library button, then access the library by pressing on the icon that has a floppy disc sign, a new window will open select the material Building Board Softwood (You can choose a material that mimics your studied case). If the material is not available in the library then the user can manually input the properties of the material.



Step 47:

The next step comes is through going to the porosity Settings section and assigning a value of 0.45 to the volume porosity input cell. To apply permeability click in the tick box. A note to the user that he can assign the value of porosity according to the material he is studying meaning that different soils have different values of porosity the same applies for fruits, woodsetc. The values of porosity for different material are found in handbooks or specialized books such as :

Principles of Heat Transfer in Porous Media (Mechanical Engineering Series) by Maasoud Kaviany



Step 48:

The next step comes here in applying the permeablity values which are the following

$$K = 1.2 \times 10^{-11} m^2$$

Again the value of permiability varies depending on the type of studied material. Assign a value of 0.001m for the Interfacial Area Den. Finaly assign the Heat TRANSFER. Coefficient a value of 1 again you choose its value depending on your problem.

A4 : Fluid Flow (CFX)	- CFX-Pre	
Edit Session I	insert Tools Help	
1 (2) ² ² ² ² ² ² ²		° ⊑
Dutline Domain: Do	main 1	
Calls of Domain 1 In r	Iow Analysis 1	
Area Porosity		E
Ontion	Isotronic	
Volume Porosity		
Ontion	Value	
Valuera Danasiku	0.45	
volume Porosity		_
Costion	Teatronic Loss	
Loss Velocity Type	Superiidai	
Ontion	Permeability and Loss Coefficient	
Permeability	E	1
Permeability	1.2e-11 [m^2]	
Resistance Lo	ss Coefficient	1
Ehvid Calid Area Dana		
	Interfacial Area Density	-
Tabarén ini Anna Dan		
Interfacial Area Den.	0.001 [m···1]	_
Fluid Solid Heat Trans	Hast Tempfer Coefficient	
Option	v	
Heat Trans. Coeff.	1, Wm^-2K^-1 -	Vα
	_	
	\frown	
OK Ap	ply class	

Step 49:

Select the highlighted region and rename it to inflow.



Step 50:

We need to solve the heat equation therefore we need to specify that. This is done through selecting the Fluid domain and then clicking the right mouse button and selecting edit.



Step 51:

Go to Fluid Models and select Thermal Energy. Then press apply and OK.

Curt Session	Insert Tools Help 面合 医 医 A A A A C 同 品 た 一〇 同 計 1 日 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
tine Domain Ir	terface: F22 18 Domain: Fluid
sic Cottings El	id Models
laat Transfer	
	Thermal Energy
Duon Include Press	none None
Incl. Viscous Dis	Isothermal
urbulence	Total Energy
otion	k-Eosilon V
	Contration (
all Function	Closure for Heat Transfer
Advanced Turbule	nce Control
ombustion	
otion	None
hormal Dadiation	
nermai Radiation	
ption	None
	9

Step 52:

Select the green regions right click on the mouse button and select Insert boundary rename it as Fluid Opening and then press OK.



Step 53:

Select the boundary details, enter in the relative pressure section 101325 Pa, also enter the value of 300 K in the Opening Temperature Section. Then press apply and press OK.

Flow Regime	Subsonic		
Mass And Momentum			
Option	Opening Pres, and Dirn	,	
Relative Pressure	101325 [Pa]		
Elow Direction		F	
Option	Normal to Boundary Condition	,	
Loss Coefficient		Ŧ	
Turbulence		Ξ	
Option	Medium (Intensity = 5%)	•	
Heat Transfer		⊡	
Option	Opening Temperature	•	
Opening Temperature	300 K	• 🗖	

Step 54:

Select the inflow region, click on the mouse right button go to boundary, then select Inlet.



Step 55:

Click on Boundary details, then select Cart Vel. Components.



Step 56:

Enter the value of 10 m/s for the U velocity while apply zero for both the V and W component. Then assign a static temperature value of 290 K. Then press apply and OK.

A4 : Fluid Flow (CFX)	- CFX-Pre		
File Edit Session I	nsert Tools Help		
	(🤊 🖻 🚰 🎍 🖉 👌 🗴 🕢 🖬 🛣 🔍 💆 🖬 🎼 🔊 🔍	·∂·∂·¤ ¼ ½ Å Å'	
Outline Domain Inte	rface: F22 18 Boundary: Inflow	×	
Details of Inflow in Fluid	in How Analysis 1		
Elow Regime		P	
Ontion	Subsonic		
Mass And Momentum			\frown
Option	Cart, Vel, Components		ーイク
	10 [m s^-1]		
v			
v			
W	0 [m s^-1]		
Turbulence	At the Orthographic PR()		
Option	Medium (Intensity = 5%)	· · · · · · · · · · · · · · · · · · ·	
Heat Transfer)
Option	Static Temperature		
Static Temperature	290	K V	
	P		
	ily Close		

Step 57:

Select the 5 walls of the Porous domain (excluding the interface wall) then click the right mouse button, select insert, select Boundary then wall.



Step 58:

Click Boundary details then select the Option Temperature instead of adiabatic then enter a value of 350 K press apply and click OK.

4 : Fluid Flow (CFX) Edit Session II	- CFX-Pre sert Tools Help	-	
	ッ (* 📽 j) ふ (* ※ (e) 副 ★ の (*) (*) (*)	□ • [1 1 • 1	
utline Domain Inte	face: F22 18 Boundary: Porous Walls	X	
Basic Settings Bou	dary Details Sources Plot Options		
Mass And Momentum			
Option	No Slip Wall	_	
Wall Velocity		±	
Option	Smooth Wall		\cap
Heat Transfer			
Option	Temperature	6.	_
Fixed Temperature	350	κ → Γ α	
	L		(
(
ų series se			

Step 59:

In the Fluid domain Principal 2D regions rename the interface surface with FI, in addition to renaming the interface surface in the Porous Principal 2D regions with PI. Select Domain interface and right click the mouse button and select Edit.



Step 60:

In the interface Type select Fluid Fluid, in the interface side 1 select in domain filter Fluid and in the regions List FI. In the interface side 2 select in domain filter Porous and in the regions List PI. The next step is to press apply and OK.

A4 : Fluid Flow (CF)	K) - CFX-Pre	
File Edit Session		
Outline Domain Into	1 7 C 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
Basic Settings Ac	Iditional Interface Models	View 1 •
Interface Type	Fluid Fluid	
Interface Side 1		
Domain (Filter)	Fluid	
Region List	FI	
Interface Side 2		
Domain (Filter)	Porous	
Region List	PI	
Interface Models		
Option	General Connection	
Frame Change/Mix	ing Model	
Option	None	
Pitch Change		
Option	None	
Option	GGI	
OK A	pply Close	Automatic generation of default domains is not currently active. This f Automatic generation of default interfaces is not currently active. This

Step 61:

Edit the Fluid domain.



Step 62:

Click on Initialization, then assign the U component a velocity of 1 m/s, type in a relative pressure of 101325 Pa, and finally type a temperature value of 300 K. Then press apply and OK.

A4 : Fluid Flow (CFX) -	CFX-Pre			
File Edit Session Ins	sert Tools Help			
	🤊 🏹 🚰 🙆 🖓 🚾 🔤			
Outline Domain: Fluid			×	*\$ \$\$. + Q € Q @ □ - ?=
Details of Fluid in Flow An	alysis 1			View 1 👻
Basic Settings Fluid I	Models Initialization			
Domain Initialization	n			
Coord Frame			± 1	
Initial Conditions				\sim
Velocity Type	Cartesian		<u> </u>	
Cartesian Velocity C	omponents		U	
Option	Automatic with Value		-	2
U	1 [m s^-1]			
v	0 [m s^-1]			
w	0 [m s^-1]			
Static Pressure				
Option	Automatic with Value			>
option				
Relative Pressure	101325 [Pa]			
Temperature				
Option	Automatic with Value		-	
Temperature	300 [K]		<	
Turbulence				
Option	Medium (Intensity = 5%)		•	
\cap				
		\frown		
		\sim		
				Automatic generation of default domains is not currently
				Automatic generation of default interfaces is not current
OK Apply				

Step 63:

Edit the Porous domain.



Step 64:

Click on Initialization, then assign the U,V and Wcomponents a velocity of 0.00001 m/s, type in a relative pressure of 101325 Pa, and finally type a temperature value of 300 K. Then press apply and OK.

c Settings Fluid Domain Initializatio	Models Initialization	
Coord Frame		
iitial Conditions	·	
locity Type	Cartesian	
Cartesian Velocity C	omponents	
Option	Automatic with Value	
l.	0.00001 [m s^-1]	
	0.00001 [m s^-1]	
v -	0.00001 [m s^-1]	
Static Processo		
	Automatic with Value	
puon		
elative Pressure	101325 [Pa]	
[emperature		
Option	Automatic with Value	
emperature	300 [K]	
Turbulence		8
Option	Medium (Intensity = 5%)	-
	\sim	

Step 65:

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

🔥 Unsaved Project - Workbench	
File View Tools Units Help	
🎦 New 对 Open 🛃 Save 🔣 Save As	🚮 Import 🏾 🖗 Reconnect 🛛 🔁 Refresh Project 🍼 Up
Toolbox 🔹 🖣 🗙	Project Schematic
 Analysis Systems 	
S Fluid Flow (CFX)	
S Fluid Flow (FLUENT)	▼ A
Component Systems	1 C Fluid Flow (CFX)
CFX CFX	2 🕅 Geometry 🗸 🖌
🥏 Engineering Data	3 🍘 Mesh 🗸 🖌
External Connection	4 🎡 Setup 🗸
Finite Element Modeler	5 Solution
Geometry Mash	
Microsoft OfficeExcel	Fluid Flow (CFX)
Results	
Design Exploration	
Goal Driven Optimization	
Parameters Correlation	
Response Surface	
Six Sigma Analysis	
_	

Step 66:

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.

olver Input File	\dp0\CFX\CFX\Fluid Flow CF	-X.def 📄 🚺	
Global Run Settings			
Run Definition			
Initialization Option	Current Solution Data (if	possible 👻	
- 🔲 Initial Values S	pecification		
Type of Run	Full	v	
Double Precision	1		
-Parallel Environme	nt	Ξ-	
Dura Marda	Carial		(
Run Mode	Serial	•	
	HP MPI Local Parallel		
5244.02	HP MPI Distributed Para	allel	
F211-02	MPICH2 Local Parallel		
	MPICH2 Distributed Par	allel	
Show Advanced	Controls		
	\frown		

Step 67:

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.

-	Edit	Work	space	Тс	ools	Mon	itors	Help	2											
9	F	3	° 4	11	E	00			G	STOP			×	2	RMS	MAX	×			
/ork	space	Rur	Fluid	Flow	CFX	003					•									
Mor	mentum a	and M	ass	He	at Tr	ansfe	r ·	Turbul	ence	(KE)										(
1	1.0e+00	ъE					-				-				_					
	1.0- 01	=																		
	1.02-01	3					1													
alue	1.0e-02	1	-								-									
able \	1.0e-03	1	-	-	~	-	-				-				_					_
Variè	1.0e-04	1													_					
	1.0e-05	=																		
	100 00	E																		
	1.0e-06	ЦL. Т	- 1	-1					- 1	T		1	-				- 1			-
	1.0e-06	-Ц Г 0		-1	-1	Т	1 5	1 1	Acc	, mula	10 ted Ti	ime St		1 1	1	5 5	ı		г - т	20
	1.0e-06	 ⊓ □	- 1	1	T	1	1 5	, ,	Acc	umula	1 10 ted Ti	ime St	ep	1 1	1	5	•	-	i i i	20
	1.0e-06	⊣ر∟ ⊓ □	-		RMS	S P-Ma	I 5 ISS		Acc RMS I	umula J-Mom	10 ted Ti	ime St	ep (MS)	/-Mom	1	5	RMS	W-W	1om	20
0t	1.0e-06	- ס 	-	1	RMS	S P-Ma	5 ss		Acc RMS I	umula J-Mom	10 ted Ti	ime St	ep MS 1	/-Mom	1	5	RMS	W-M	1om	20
Dut	1.0e-06	۲۱ ۱ ۱	-	-	RMS	S P-Ma	T 5		Acc RMS I	umula J-Mom	10 ted Ti	ime St	ep RMS V	/-Mom	1	5	RMS	W-M	1om	20
Out	1.0e-06	۲ <u>ا</u> لد ۲ ۱			RMS	S P-Ma	1 5 ss •	+	Acc RMS I	umula J-Mom	10 ted Ti	ime St	ep MS 1	/-Mom	1	5	RMS	W-M	1om	20
Out +	1.0e-06				RMS	S P-Ma	1 5 ss '	+	Acc RMS I	umula J-Mom	10 ted Ti	ime St	ep MS \ +	/-Mom	1	5	RMS	+	1om	20
Out + 1	1.0e-06	 g tr :	ansi Tra	 ent	RMS	S P-Ma	ful.	+	Acc RMS I	umula J-Mom	10 ted Ti	me St	ep RMS 1 +	/-Mom	1	5	RMS	W-M	1om	20
Out + 1 	1.0e-06	 0 	ansi Tra Sta	ent nsi(RMS fi:	SP-Ma	I 5 ISS ·	+ 1.tri 1	Acc RMS I	umula J-Mom	10 ted Ti	me St	ep RMS \ +	/-Mom	1	5	RMS	+ + 	1om	20
Out + 1 	1.0e-06		ansi Tra Sta Tim	ent nsi nda:	RMS fi: ent rd ep :	SP-Ma	full full ilts	+	Acc RMS I	umula J-Mom	10 ted Ti	ime St	ep RMS 1 +	/-Mom	1		RMS	+ + 	10m	2(
Out + 1 +	:File Writin Name Type Opti	 0 g tz : : :	ansi Tra Sta Tim	ent nda est	RMS fi: ent rd ep :	5 P-Ma -+ le 5_ Resu Intes	full fulls	+ 1.trn 1	Acc RMS I	umula J-Mom	1 10 ted Ti	me St	+		1		RMS	W-M	1om	20
Dut + 1 +	:File Writin Name Type Opti	 0 g tr : : : :	ansi Tra Sta Tim	ent nsi est	RMS fi: ent rd ep :	-+ le 5_ Inter	fullts	+ 1.trr 1	Accc RMS I	umula J-Mom	1 10 ted Ti		+		1		RMS	+ + 1 1 1	10m	20
Out + 1 +	File Writin Name Type Opti	 g tr ::	ansi Tra Sta Tim	ent nsi est	RMS fi: ent rd ep :	SP-Ma	fullts	+ 1.trr 1	Acc RMS I			R R	+		1		RMS	+ + + +	lom	20
Out + 1 1 1 1 +	1.0e-06		ansi Tra Sta Tim	ent nsi est	RMS fi: ent rd ep :	SP-Ma	ful ilts cval	+ 1.trr 1	Acc RMS I	umula J-Mon +	10 ted Ti	R	ep MS 1 +		1		RMS	W-M	1om	20
Out + 1 +	:File Writin Name Type Opti	 0 g tr : : : : : : : : : : : : : : : : : : :	ansi Tra Sta Tim 	ent nsi est	RMS fi: ent rd ep	5 P-Ma -+ le 5_ Resu Inter Time	_ful sss ' _ful ults cval	+ 1.trn 1 pping	Accc RMS I	Umula J-Mom	10 ted Ti a	ime St R R 	ep RMS 1 +	/-Mom	1	5 	RMS	+ + + 	10m	20
Out + 1 1 1 1 + 1 + 1	:File Writin Name Type Opti	 0 () () () () () () () () () () () () ()	ansi Tra Sta Tim step 	ent nda est 	RMS fi: ent rd ep	5 P-Ma	full ilts estep	+ 1 ppping Coura 99:	Accc RMS I	Umula J-Mom		n 1 M 1	ep (MS) +	/-Mom	1	 Num 9	RMS	W-M + + + + + + +	1 om	20
Step 68:

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 📙 Save 🔣 Save As	🚰 Import 🛛 🍣 Reconnect 🛛 🥏 Refresh Project
Toolbox 🝷 🕂 🗙	Project Schematic
 Analysis Systems 	
Fluid Flow (CFX)	
G 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
	5 Solddon V
Geometry	• Fresults
Missooft Office Even	Fluid Flow (CFX)
Goal Driven Optimization	
Parameters Correlation	
Response Surface	
Six Sigma Analysis	

Step 69:

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 70:

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



Step 71:

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



Step 72:

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. In the variable section select Temperature. Then go to (# of Contours) and enter into the input cell 100.



Step 73:

The next step is double click the case comparison, then click on the tick box of case comparative Active. Then double click on the time step numbered 6. You will notice that the comparative case gets deactivated.



Step 73:

Select time step 6. Then press Load now. Finally press apply.



Step 74:

The results shown represent time step number 6, time step number 10, and finally the subtraction between the two time steps which help in calculating the heat loss (gain).

