

Methane Combustion Modelling Tutorial using ANSYS CFX

First Edition

By Ahmed Al Makkky

@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.

Methane Combustion 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. Combustion is encountered in most engineering applications. Ranging from boilers to gas turbines to internal combustion engines.....etc. This tutorial guides the researcher through a step by step process to get a simple combustion done. From this tutorial the user will be able to get the general required guideline to conduct a combustion simulation. This tutorial might have some inaccurate input values but no need to worry about that at the moment because at this stage it is essential to get an initial simulation run then we can proceed in making more accurate input value selection. The mesh is made with a ball forming an obstacle in the flow. The main intention of having that ball is to form a circulation filed after it and also to form a source of ignition to the mixture. This is done through assigning the ball surface a heat source term.

Step 1:

The first step is to double click the Fluid Flow (CFX) icon, then the next step is to double click on the geometry icon the has beside it the question blue mark.



Step 2:

Under file drop down list select Import External Geometry file. You need to remmber that the file is provided with this tutorial. This tutorial skips the modelling part of the project so there is no need to use DesignModeller. If you do not have a ready mesh then it is essential to model your mesh using DesignModeller.



Step 3:

A window will open up select the file named Ball.x_t. and then press open.



Step 4:

Go back to the main list and check that the green tick sogn is visible beside the geometry icon and then double click the Mesh icon.

Unsaved Project - W	/orkbenc	h							
File View Tools	Units	Help							
New 💕 Open	Save	🔣 Save As	👔 Impo	ort	₽¢ Reconnect	Refresh Project	🕖 Update Project	Project	🕜 Compact Mode
Γoolbox		▼ ₽ Х	Project S	chematio	-				
 Analysis Systems 									
C Fluid Flow (CFX)									
Fluid Flow (FLUENT)	г)		•	•	А				
Component Systems			1	1 🖸 1	Fluid Flow (CF))			
CFX			1	2 🕅 (Geometry	× .			
🥏 Engineering Data				3 🥥 1	Mesh	2			
🔆 External Connection	n			4 🚵	Setup	2			
Finite Element Mod	eler			5 6	Solution				
FLUENT									
Geometry				• 🐠 •	Results	ii ⊿			
🍘 Mesh				F	luid Flow (CF)	0			
	cei								
Ceal Drives Optimi	antino								
Barameters Correla	tion								
Response Surface	10011								
Six Sigma Analysis									
			Message						
				Α					
			1	Туре					
			2	Events	Automotive	Powertrain Fluid-Stri	ucture Interaction (FS	SI)	
			3	Events	Ask the Exp	pert - External Data M	lapping in ANSYS Wo	rkbench &am	p; Mechanical 14.0

Step 5:

Right Click on method then select insert and then from insert select method.



Step 6:

The next step is by clicking on the mesh box which results in it turning into green then pressing the Apply icon in the geometry section.



Step 8:

Click on the update to command the Mehser to start generating the mesh.



Step 9:

By looking where the arrow is pointing you can see the mesh generation process is being done. Just to note that if the meshing process takes more than 5 minutes than most probably no mesh would be generated.

A : Fluid Flow (CFX) - Meshing [ANSYS Academic Teaching CFD]	Access March 10	
File Edit View Units Tools Help File Generate Mesh	। 👪 🛆 🕢 • 💵/Worksheet 🕾 😤 🎝 • 🛱 🖬 🖪 🗃 🚱 • 😂 💠 🍳 🔍 🔍 🔍 🔍 💭 🗮 🗔 •	🔎 Show Vertices 😽 Wirefram
Mesh # Update 1 Mesh - 1 Mesh Control - al Metric	Graph 🍓 Options	
Meta: ∳(updat) @) Mask • @, Mask Control •	Activation Method	
Details of "Automatic Method" - Method		
E Scope		
Scoping Method Geometry Selection		
Geometry 1 Body		
Definition		
Suppressed No		
Method Automatic		
Element Midside Nodes Use Global Setting		
	0.559 0.250 0.460 (m)	
	0.100 0.300	2
NSYS Workbench Update Model Status	ometry / Print Preview / Report Preview /	
Generating mesh for Solid	ages	np
Modeling interior for part		
Stop		

Step 10:

Click the left button on the mouse on the mesh icon to clarfiy that the mesh has been

generated.



Step 11:

Check that the green tick is beside the mesh sign then double click the setup icon.

🔥 Unsaved Project - Workbench	
File View Tools Units Help	
🛅 New 📄 Open 🗒 Save 🔣 Save As	👔 Import 🗧 Reconnect 🥩 Refresh Project 🦻 Update Project 🄇 Proje
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 😨
FLUENT	6 🥪 Results 💡
Geometry	Fluid Flow (CFX)
Mesh Misser & Office Fuel	
Goal Driven Optimization	~
Parameters Correlation	
Response Surface	
J Six Sigma Analysis	
·····	
	Messages
	A
	1 Туре
	2 Events Automotive Powertrain Fluid-Structure Interaction (FSI)
	3 Events Ask the Expert - External Data Mapping in ANSYS Workbench &a
	4 Events Understanding Hardware Selection for Structural Mechanics
	5 Events SPE Annual Technical Conference & amp; Exhibition

Step 12:

Go to the Principle 3D regions and Select the shown boundary and right click the mouse button and select the rename option, then assign the highled boundary in green as: inflow.



Step 13:

After renaming the boundary go to insert then from boundary drop list chose inlet.



Step 14:

Go to boundary details and from the option drop list select Cart. Vel. Components.

🖶 A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
🚽 🕑 📽 🔩 🞯 🤊 🕫 🗳 🍐 🔌 🕢 📾 🖈 🕓 🞯 🖄 👘 🛣 👘 🔛	
Outline Boundary: Tolet	
Details of Inlet in Default Domain in Flow Analysis 1	, <u> </u>
Basic Settings Boundary Details Sources Plot Options	
Flow Regime	
Option Subsonic	
Press Ald Providencial	
Option Normal Speed	
Normal Speed Cart. Vel. Components	
Turbulence Cyl, Vel. Components	
Option Total Pressure (stable)	
Static Pressure	

Step 14:

Assign the U and V velocity slots zero values while the W velocity component assign it a value -10 m/s.

💼 A4 : Fluid Flow (CFX) - CFX-Pre	2	
File Edit Session Insert To	ols Help	
🔒 🕑 😤 🔩 🚳 🤌 🤇	· 🚰 🏚 💩 👌 🗴 🚾 🖬 🍂 🕓 🞯 🗊 🛤 🕼 📐	a 🖥 👘
Outline Boundary: Inlet		X * * • • • • •
Details of Inlet in Default Domain	i in Flow Analysis 1	View 1 *
Basic Settings Boundary Deta	ils Sources Plot Options	
Flow Regime		
Option Subson	c 🔹	
Mass And Momentum		
Option Cart. Ve	el. Components 👻	
U 0 [m s^	-1]	
V 0 [m s^	-1]	
-10 fm	s^_1]	
Tubulana		
	(Telessity - 50)	
Option	(intensity = 5%)	
	\sim	
	()	

Step 13:

Choose the ball surface and assign it a wall condition and press Ok. At a later stage we will assign this wall a heat source which would perform the role of flame holder.



Step 14:

Select all the 5 boundaries using the control button and the mouse cursor to select the required boundary.



Step 15:

Type in the pressure value of 101325 Pa which represents atmospheric conditions. Then press

Ok.

A4 : Fluid Flow (CFX) ·	CFX-Pre	
Edit Session In	sert Tools Help	
] 🔄 😤 🔩 🔟	🔊 🕫 🚰 🏚 🕹 👌 🕱 🚾 🖬 🔊 🙆 🗿 🛤 🗇 🖓 📾 🖓	
Outline Boundary: op	sening 🛛	*\
ails of opening in Def	ault Domain in Flow Analysis 1	View 1
Basic Settings Boun	dary Details Sources Plot Options	
Flow Regime		
Option	Subsonic	
Mass And Momentum	8	
Option	Opening Pres. and Dirn 🔹	
Relative Pressure	101325 Pa 🗸 🚾	
Flow Direction	T B	
Option	Normal to Boundary Concition	
Loss Coefficient	E	
Turbulence		
Option	Medium (Intensity = 5%)	
	Ъ	
OK Appl	Close	

Step 16:

Check that there is a green tick in the setup section then double click the solution icon.



Step 17:

If you have a single core machine then keep the selection as serial while for more extensive calculations then HP MPI Local Parallel is advisable.

85 : Fluid Flow (CFX) - CFX-Solver Manager	
File Edit Workspace Tools Monitors Help	
Image: Second	
Solver Input File \dp0\CFX\CFX\Fluid Flow CFX.def Image: Control Solution Data (if possible Run Definition Initialization Option Current Solution Data (if possible Initialization Option Initialization Image: Control Solution Data (if possible Initialization Option Full Image: Control Solution Data (if possible Initialization Option Full Image: Control Solution Data (if possible Image: Control Solution Data (if possible Image: Control Solution Data (if possible Image: Control Solution Data (if possible Image: Control Solution Data (if possible Start Run Save Settings Cancel	

Step 18:

The first step is click on the plus icon till the number of partitions gets to four; this would help to conduct a parallel calculation on the lab quad core desktop. A note to the user that he has to check what kind of processor his desktop is.

A5 : Fluid Flow (CFX) -	CFX-Solver Manager			×		
File Edit Workspace	Tools Monitors Hel	p				
i 🐑 😤 🥰 📝 🖽 g			RMS MAX	ject	🕒 Project 🕜 Compact N	10
Workspace 0 Def	ine Run		? ×			
Glob	r Input File (dp0) bal Run Settings un Definition	CFX \CFX \Fluid Flow CFX.def [] rrent Solution Data (if possible + ition II P MPI Local Parallel Partitions 4 Is Is Progress 1 S	A ratus	n (FS S Wor Mech pn	I) kbench & Mechanical 1: ianics	

Step 19:

By running a simulation the user can initially clarify that the domain is applicable to run a combustion simulation.



Step 20:

The visulization part comes here, it starts by double clicking on the results icon.



Step 21:

By using the vector representation to visualize the flow pattern the user can clarify that his input parameters are correct.



Step 20:

Then comes selecting the required domain for the vector field which is in the studied case the default domain and then press Ok.

le Edt Sesson Inset Tools Heb Celle Cators Turbon + Celle Cators Turbon Cutine Wenddes Expressions Calculators Turbon Cators Calculators Turbon Calculators Turbon Calculators Turbon Calculators Turbon Calculators Calculators Turbon Calculators Turbon Calculators Turbon Calculators Turbon Calculators Calculators Turbon Calculators Turbon Calculators Calculators Calculators Turbon Calculators Calculators Turbon Calculators Calculators Turbon Calculators Calculators Calculators Turbon Calculators Calculators Calculators Turbon Calculators Calculators Calculators Turbon Calculators Calculators Calculators Calculators Turbon Calculators Calculators Ca	A6 : Fluid Flow (CFX) - CFD-Post	
Image: Solution and Soluti	ile Edit Session Insert Tools Help	
Outine Variables Expressions Calculators Turbo * © Cases • © FX • © CX • © CX * © Default Domain • © File • © CX • © CX * © Cases • © CX • © CX • © CX * © Default Domain • © CX • © CX • © CX * © Sering • © CX • © CX • © CX * © User Locations and Plots • © CX • © CX • © CX * © User Locations and Plots • © CX • © CX • © CX * © Default Transform • © CX • © CX • © CX * © Mesh Report • © Coir Symbol Render View • © © File Report • © © File Report * Extails of Vector 1 • © One • © One • © One Cations © © File Report • © © © File Report • © © © File Report * Xatals of Vector 1 • © One • © One • © One Cations © © © File Report • © © © © © © © © © © © © © © © © © © ©	😤 🕰 🛃 🚳 🤊 🥐 🎁 Location 🔻 🛱 🎘	≳ 🗊 🖋 🍻 🔢 🚧 💿 🗶 🚾 🎟 🖄 🏈 🛄 🏄 🧪 🕼
Coses	Outline Variables Expressions Calculators Turbo	*\; 💽 🛟 @, 🕀 @, 🗇 🗖 🗸 ?🖥
CrX Corr CrX Corr Cor <pcorr< p=""> <pcorr< p=""> <pcorr< p=""> <pcorr< p=""></pcorr<></pcorr<></pcorr<></pcorr<>	4 🔞 Cases	▲ View 1 ▼
▲ ● Default Domain ● Events ● ● Mesh Regions ■ 1.220e+001 ● ● Mesh Regions ■ 1.220e+001 ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●	🔺 😰 CFX	1
Velocity Vector 1 Vector 1 Vector 1 Vector 1 Vector 1 1.220e+001 Vector 1 9.150e+000 Vector 1 9.150e+000 Vector 1 9.150e+000 Vector 1 Vector 1	Default Domain	
Vector 1 1.220e+001 Mesh Regions Mesh Regions Mesh Regions Perfult Transform Perfult Tarsform Perfult Denail Mesh Report Mesh	🔲 🕽 🏞 Inlet	Velocity
Image: Field Image: Field Image: Field Image: Field Image: Field	J‡ opening	Vector 1
 ▶ Mesh Regions West Locations and Plots Default Transform ♥ Default Legend View 1 ♥ Wreframe ■ Report ♥ Mesh Report > ♥ Mesh Report > ♥ Mesh Report > P Mesh Report > ♥ Mes	i j)‡ wali	1 220e+001
 Secondary Color Symbol Render View Default Domains Default Domain Fietor 1.0 Octobe + 000 3.050e + 000 Sompling opening wal Reduction Default Domain Fietor I.0 Octobe + 000 Sompling opening Wall Conservative Projection None None 	Mesh Regions	1.22001001
B ⁻ Verault Legend View 1 ♥ B ⁻ Default Legend View 1 ♥ File Report ♥ File	User Locations and Plots	
Vereframe Q = Wireframe Q = Wireframe Q = Mesh Report Vereframe Q = Mesh Report Vereframe Q = Mesh Report Vereframe Q = Mesh Report Off Symbol Render View Comains All Domains Off Symbol Render View Conservative Projection None Projection None Q = Mesh Report Q	Default Transform Default Leased View 1	
Pertentine Pertentin Pertentin Pertentin Pertentine Pertentine	V P Default Legend View 1	
 S. 1500 + 000 S. 1500 + 000	A Depart	9 1500+000
Image of the region of the	Title Page	9.1500+000
File Vector 1 Geometry Color Symbol Render View Domains Al Domains Definition Locations Sampling Reduction Perfault Domain Factor 1.0 Variable Velocity Projection None	File Perpert	
Details of Vector 1 Geometry Color Symbol Render View Domains Al Domains • Definition Locations Default Domain Sampling opening weal Reduction Defout Domain Factor 1.0 Variable Velocity • Boundary Data • Hybrid • Conservative Projection None •	Mesh Report	
Details of Vector 1 Geometry Color Symbol Render View Definition Locations Default Domain Inlet opening wall Reduction Defoult Domain Factor 1.0 Variable Velocity Boundary Data Hybrid Conservative Projection None		
Geometry Color Symbol Render View Domains All Domains Definition Locations Default Domain Reduction Performing Reduction Definite Boundary Data Hybrid Conservative Projection None	Details of Vector 1	6 1000+000
Domains Image: Conservative Projection Al Domains Image: Conservative Projection None Image: Conservative Projection	Geometry Color Symbol Render View	0.10001000
Definition Locations Default Domain Inlet Sampling opening Reduction Default Domain Factor 1.0 Variable Velocity • Boundary Data • Hybrid © Conservative Projection None •	Domains All Domains	
Locations Default Domain Sampling opening Wall Reduction Default Domain Factor 1.0 Variable Velocity Boundary Data Orgonia Conservative Projection None Occuservative		
Locations Default Domain	Definition	
Sampling Unlet opening Wall Reduction Default Domain Factor 1.0 Variable Velocity Boundary Data Hybrid Conservative Projection None	Locations Default Domain	- 3 050e+000
Reduction Defoult Domain Factor 1.0 Variable Velocity Boundary Data Hybrid Conservative Projection None	Sampling	3.000001000
Reduction Default Domain Factor 1.0 Variable Velocity Boundary Data Hybrid Conservative Projection None Image: Conservative Image: Conservative	wall	
Factor 1.0 Variable Velocity Boundary Data Image: Hybrid Projection None	Reduction Default Domain	
Variable Velocity Boundary Data Oconservative Projection None Oconservative	Factor 1.0	
Boundary Data Hybrid Conservative Projection None	Variable Velocity	0.069e+000
Projection None	Boundary Data Hybrid Conservativ	[m s^-1]
	Projection	

Step 22:

By looking closely at the vector tip direction the user can clarfy that the flow field is. Mimicing the case he wants.



Step 23:

The next step is to go back to the fluid flow (CFX) and double click on the setup icon. This is where we start working on the combustion side of things.

File View Tools Units Help					
🛅 New 对 Open 🛃 Save 📓 Save As	👔 Import	≩φ Reconnect	ᄙ Refresh Project	🕖 Update Project	Project 🕥
oolbox 🔻 🕂 🗙	Project Schemat	c			
Analysis Systems					
😋 Fluid Flow (CFX)					
Fluid Flow (FLUENT)	-	A			
Component Systems	1	Huid Flow (CFX)			
2) CFX	2 🔟	Geometry	× 🔺		
Engineering Data	3 🥡 1	Mesh	× .		
External Connection	4 🎡	Setup	< <		
Finite Element Modeler	5 😭 :	Solution			
E FLUENT	6 📦	Results	1		
Geometry		tite (co)	· 4		
📦 Mesh	'	Huid How (CFX)			
Microsoft OfficeExcel					
Results					
Design Exploration					
🧿 Goal Driven Optimization					
Parameters Correlation					
Response Surface					
📙 Six Sigma Analysis					

Note : Regarding important values for the combustion simulation the researcher can extract lots of the needed data from the following reference:

http://www.amazon.co.uk/An-Introduction-Combustion-Concepts-Applications/dp/0071260722/ref=cm_cr-mr-title

This handbook is also rich with important data to:

http://www.amazon.co.uk/Combustion-Engineering-Computational-Mechanics-Analysis/dp/0849320712/ref=cm_cr-mr-title

If you want to learn more about flamelets and computational combustion simulation:

http://www.amazon.co.uk/Theoretical-Numerical-Combustion-2-E/dp/1930217102/ref=cm_cr-mr-title

Step 24:

By right clicking on the Default Domain icon what comes next is selecting the edit option.



Step 25:

Under the basic setting you will see that by defual Fluid 1 is assigned by the software which represents Air at a temperature of 25. We are not concerned with air for our studied case for combustion case of Methane which will be our studied case we will need four species to be taken into account based on the following equation:

utline Domain: D	efault Domain	× 🕄 🖈 🔍	
asic Settings Flu	uid Models Initialization	View 1 🔻	
Location and Type			
ocation	B16	▼	
omain Type	Fluid Domain		
oordinate Frame	Coord 0		
Fluid and Particle De	finitions		
Fluid 1			
		Remove selection and	
Fluid 1			
Option	Material Library		
Material	Air at 25 C	• <u> </u>	
Morphology			
Option	Continuous Fluid		
	ume Pracuon		
Domain Models			
Pressure	4 [44-3]		
Reference Pressure	i lamj		
Buoyancy Model			
Option	Non Buoyant		
Domain Motion			
Uption	Stationary		
Mesn Deformation	NI		
Option	INORE		

$$CH_4(g) + 2O_2(g) \xrightarrow{\text{yields}} CO_2(g) + 2H_2O(l) + 891KJ$$

By clicking on the x red button the Fluid 1 is deleted from the list and we can proceed to add the gas species we want to add.

Step 26:

To add the required species click on the icon that has a document with a yellow star to add a new species. You can start with which ever species you like. I started with carbon dioixde then water vapour then oxygen and finally methane. Remmber that we have just assigned the names of the species, the next step is to asign the thermodynamic properties to the species we have just named.



Step 27:

To acsess the thermodynamic properties of the different specise we have to call the library which is provided with the software.

A4 : Fluid Flow (CFX)	- CFX-Pre		Contraction resulting any ARTS (7). Manual Real	
File Edit Session In	nsert Tools Help			
📄 🔮 💐 🔩 🕥	i 🤊 te 🎬 🏚 🕹 🔶 x 🕢 🖬 🌾 🕙 🗃 📭 🕫 👔	o 🗠 🖓 🔓		
Outline Domain: Def	fault Domain	X	*Ъ Г.+ Q. Q. Q. () □ - 3	
Details of Default Doma	in in Flow Analysis 1		View 1 v	
Basic Settings Fluid	Models Fluid Specific Models Fluid Pair Models Initialization			\smile
Location and Type				
Location	B16	•		\sim
Domain Type	Fluid Domain	•		
Coordinate Frame	Coord 0	•		
Fluid and Particle Defin	nitions	Ξ		
CH4				
CO2				
02				
CH4				_
Option	Material Library	•	Material X	\cap
Material	Air at 25 C	▼	🗸 Air Ideal Gas	
Morphology			👗 Air at 25 C	
Option	Continuous Fluid	•	CHT Solids	
Minimum Volur	me Fraction	Œ	Calorically Perfect Ideal Gases Constant Property Gases	
Domain Models			🕹 Air at 25 C 🗉	
Pressure		8	Constant Property Liquids	
Reference Pressure	1 [atm]		> Soot	
Buoyancy Model		8	> Water Data	
Option	Non Buoyant	•		
Domain Motion			OK Cancel	
Option	Stationary	•		
Mesh Deformation		8		1
Option	None	•	😁 Select Library Data to Import	
			File to Import a/MATERIALS-standard.cd	
			Air Data	
			CHT Solids	
			Calorically Perfect Ideal Gases O.125	
			Constant Property Gases	
			Gas Phase Combustion	
			Interphase Mass Transfer Liquid Phase Combustion	
			Particle Solids	
			Water Data	
			OK Cancel	
ОК Арр	ly Close			

Step 28:

When you open the library a new window is visible. Go to the gas phase combustion section and select the four required species which we named earlier and then press Ok. What you will notice that after pressing Ok the four species have been transferred to the material window.

A4 : Fluid Flow (CFX)	- CFX-Pre	
File Edit Session I	nsert Tools Help	
	· · · · · · · · · · · · · · · · · · ·	
Outline Domain: De	fault Domain	⊠ *\ \$. \$. \$. \$. \$. \$. \$. \$. \$. \$. \$. \$. \$.
Bacic Settings	MILIT FROW ANDIYSIS 1	View 1 👻
Location and Type	a models i filidi Specific Models i filidi Pali Models i filidilizadori	
Location	816	
Domain Type	Eluid Domain	
Coordinate Frame		
Fluid and Particle Defi	nitions	
CH4		t
CO2		$\mathbf{I} = \mathbf{I} + \mathbf{K} \mathbf{N} \mathbf{D}$
02		
CH4		
Option	Material Library 🔹	🗃 Material 🛛 🖉
Material	Air at 25 C 🔹	
Morphology		Air at 25 C
Option	Continuous Fluid 🗸	CHT Solids
I Minimum Volu	me Fraction 🗄	Calorically Perfect Ideal Gases
Domain Models		Air at 25 C =
Pressure	B	Constant Property Liquids
Reference Pressure	1 [atm]	Particle Solids
Buoyancy Model	8	Vater Data
Option	Non Buoyant 👻	
Domain Motion		OK Cancel
Option	Stationary 👻	
Mesh Deformation		2
Option	None 👻	😰 Select Library Data to Import
		File to Import a/MATERIALS-standard.cd
		Д СНЗ
		С СНЗСО
		GH3NO2 0.11
		U CH30
		> 2 CH30H
		CH4
		3 CN 7 CO
OK ADD	lv Close	Cancel

Step 29:

The thermodynamic species have been specfied the next step is to link thermodynamic quantities with our named species.



Step 30:

What comes next is we need to select the Fluid model the importance of this that by selecting the Thermal Energy option from the drop down list we can assign the tempeature heat source term to the sphere surface which would perform the ignition source to the chemical reaction.

B A4 : Fluid Flow (CFX)	- CFX-Pre		
File Edit Session In	isert Tools Help		
🛃 🔮 💐 🔩 🗃	i 🤊 🗞 🚰 🧔 🕹 🔌 🗶 📾 🗛 😡 🖉 🎽 👘 🖓 🖄	1 📲 🖻	
Outline Domain: Def	fault Domain		
Details of Default Doma	in in Flow Analysis 1	;	
Basic Settings Fluid	Models Fluid Specific Models Fluid Pair Models Initialization		
Multiphase			
Homogeneous Mod	tel		
Free Surface Model	8		
Option	None		
Heat Transfer	1	3	
Homogeneous Mod	del del		
Option	Isothermal 🔹		
Fluid Temperature	Isothermal		. + t _∞
	Thermal Energy Total Energy		
lurbulence	Fluid Dependent		
Homogeneous Mod			
Option			
Combustion		╕╴╷╷	
Option	None		
Thermal Radiation		3	
Option	None		
Electromagnetic M	lodel	Ð	
			1 martin
			1
			-
		9) In Analysis 'Flow Analysis 1' - Domain 'Default Domain': It is recommended that a
			The object "/FLOW:Flow Analysis 1/DOMAIN:Default Domain/BOUNDARY:Inlet" of The object "/FLOW:Flow Analysis 1/DOMAIN:Default Domain /FOI INDARY:anapin
			 The object (PLOW:Plow Analysis 1/DOMAIN:Default Domain/BOUNDARY:Openin)
OK Ann			

Step 31:

The next step is click the tick boxes for a Homogeneous Model for the Multiphase part and for the Homogeneous Model for the Heat Transfer part.

A4 : Fluid Flow (CFX) -	CFX-Pre	
File Edit Session Ins	ert Tools Help	
🚽 🕸 😤 🔩 🚳	🤊 🕫 🚰 ja 🌡 👌 🗴 🚾 🖬 🖍 🕓 🧭 🗇 輝 📾 🗊 🦺 🗽	A 🔓 👘
Outline Domain: Defa	ult Domain	
Details of Default Domain	n in Flow Analysis 1	View 1 ×
Basic Settings Fluid N	Models Fluid Pair Models Initialization	
Multiphase		
Homogeneous Mode		
Free Surface Model		
Option	None	
Heat Transfer	\frown	8
Homogeneous Mode		
Option	Thermal Energy	
Incl. Viscous Dissipa	tion	
Turbulence		
Option	k-Epsilon 👻	
Wall Function	Scalable	
Turbulent Flux Clo	sure for Heat Transfer	
Advanced Turbulence	Control	• •
Combustion		
Option	None	
Thermal Radiation		8
Option	None 👻	
Electromagnetic Mo	idel	

Step 32:

The next step is to define the occuring reaction in the flow this is done through right clicking on the Reactions folder in the Outline section and then from the insert drop down list select Reaction.



Step 33:

By default the reactants is specfied as Air Ideal Gas, while for our case are the two species in the red color shown in the following equation:

$$CH_4(g) + 2O_2(g) \xrightarrow{\text{yields}} CO_2(g) + 2H_2O(l) + 891KJ$$

🗃 A4 : Fluid Flow (CFX) - CF	FX-Pre
File Edit Session Inser	t Tools Help
📄 🛃 💐 🙀 🚳 🕷	? 🗢 🎬 🏚 🕹 🔌 🚾 🖬 🔊 🕑 🖉 🗇 🍱 📾 🕸 🌾 🖓 🐃 🐼 💊
Outline Reaction: React	ion 1 🛛 🔀
Details of Reaction 1	
Basic Settings Reactan	ts Products Reaction Rates
Reactants	
Option	hild Materials
Materials List A	iir Ideal Gas
Child Materials	
Air Ideal Gas	
Air Ideal Gas	
Option	Stoichiometric
Stoichiometric Coefficie	1.0
Reaction Order	

Step 34:

So we add the two species by pressing on the squred box and from the drop list selecting Methane and Oxygen.

Intert Intert Intert Intert <th>4 : Fluid Flow (CFX) -</th> <th>CFX-Pre</th> <th></th> <th></th> <th></th> <th></th> <th></th> <th></th> <th></th> <th></th> <th></th>	4 : Fluid Flow (CFX) -	CFX-Pre									
Image: Setting: Reaction 1 field: Reaction 1 sex::Setting: Reaction 1 sex::Setting: Reaction 1 sex::Setting: Reaction 1 sex::Setting: Reaction 2 point Colid Materials Ar Ideal Gas Image: Stadhometric Option Stadhometric Stachometric Coeffice: 1.0 Reaction Order Image: Setting: Image: Reaction Order Image: Reaction Order	Edit Session Ins	sert Iools Help	. 🏹 🔥 🤉	د 🗔 🔜 د	£. 0	ह्यान	1t e	1 🗐 i	r 1 1 .	। IN मि	
Internals Vectors reaction 1 sex Settings Reaction 1 sex Settings Child Materials Vectors reaction 1 Image: Setting 1 sex Settings Child Materials Air Ideal Gas Image: Setting 1 Air Ideal Gas Image: Setting 1 Air Ideal Gas Image: Setting 1 Stachiometric Image: Setting 1 Stachiometric Image: Setting 1 Image: Setting 1 Image: Seting 1 <td></td> <td></td> <td>· • • •</td> <td></td> <td>* : •</td> <td></td> <td>y← LL</td> <td></td> <td>•~• ⁻t=0</td> <td>1 E F</td> <td></td>			· • • •		* : •		y ← LL		•~• ⁻t=0	1 E F	
excentres reducts reaction Rates	ils of Reaction 1	action 1									
Rectants pipon Inidi Materials atr Ideal Gas Child Materials Atr Ideal Gas Option Stachiometric Stachiometr	asic Settings React	tants Products	Reaction Rat	es							
pton Child Materials taterials List Ar Ideal Gas Ar Ideal Gas Ar Ideal Gas Cotion Stachometric Stachometric Coeffice 1.0 Reaction Order (Materials List Ar Ideal Gas Ar at 25 C Child Solds Ar Ideal Gas Constant Property Liquids Soci Constant Property Gases Constant Property Gase	Reactants									Ξ	
At Ideal Gas Option Stochtometric Stochtometric Coeffice 1.0 Reacton Order (Constant Property Gases Constant Property Liquids Gas Phase Combustion Constant Property Liquids Gas Phase Combustion Constant Property Liquids Gas Phase Combustion Constant Property Liquids Gas Phase Combustion Constant Property Liquids Constant Property	Option	Child Materials								•	
Child Materials	1aterials List	Air Ideal Gas								•	
Ar Ideal Gas Option Stoichtometric Stoichtometric Coeffice 1.0 Reaction Order	Child Materials										
Ar Ideal Gas Option Stuchtiometric Stochtiometric Coeffice 1.0 Reaction Order	Air Ideal Gas										
Ar Ideal Gas Option Stochiometric Stochiometric Coeffice 1.0 Reaction Order III Materials List III folde Gas Ar Ideal Gas Ar Ideal Gas Constant Property Ideal Gases Ar Ideal Gas Constant Property Ideal Gas Co											
Ar Ideal Gas Opton Stoidhiometric Stoidhiometric Coefficie 1.0 Reaction Order											
Option Stoichiometric Stoichiometric Coefficie 1.0 Reaction Order Image: Constant Property Gases Ar Ideal Gas Ar Ideal Gas Ar Ideal Gas Ar Ideal Gas Water Ideal Gas Water Ideal Gas Constant Property Gases Constant Property Liquids Gas Phase Combustion Image: Constant Property Liquids Gas Phase Combustion Image: Constant Property Liquids Mater Data Constant Property Liquids Interphase Mass Transfer Particle Solids Valer Data Concel Water Data Concel	Air Ideal Gas										
Stochiometric Coefficie 1.0 Reaction Order K Apply Coe	Option	Stoichiometric								-	
Reaction Order	Stoichiometric Coeffic	tie 1.0									
Materials List Air Ideal Gas Air it at 25 C Ott Solids Calorically Perfect Ideal Gases Air Ideal Gas Orstant Property Gases Constant Property Liquids Gas Phase Combustion Air 200 Interphase Mass Transfer Particle Solids Soot Water Data Ott Ott Cancel 	Reaction Order	r								Ŧ	
OK Apply Close			 Air CH Cal Coi Coi Coi Gas Int Pair Soc Wa 	Data Ar Ideal Ar at 25 Folids orically Perf Ar Ideal Water Id stant Prope stant Prope stant Prope s Phase Com CO2 CO2 CO2 CO2 CO2 CO2 CO2 CO2	Gas C fect Idea Gas Jeal Gas erty Gase erty Liqu ibustion	I Gases es ids	Ca	ncel			
\sim	OK Apply	y Close									
						Y					

Step 34:

It is visible from the material list that Methane and oxygen have been added.

$$1\text{CH}_4(g) + 2\text{O}_2(g) \xrightarrow{\text{yields}} \text{CO}_2(g) + 2\text{H}_2\text{O}(l) + 891\text{KJ}$$

First of all we select methane in the Child Materials list. The stoichiometric coefficient is shown in green and is taken having the value of one. The reaction order therfore as a consequance the reaction order is taken as 1.

🗃 A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
🚽 🔮 🖳 🞯 🦻 🐑 👌 🕹 👌 🎢 🔜 🖍 🔛 🖉 🖓 🖾 👘	"i 🖻
Outline Reaction: Reaction 1	X *
Details of Reaction 1	v
Basic Settings Reactants Products Reaction Rates	
Reactants	
Option Child Materials	
Materials List CH4,02	1
Child Materials	,
CH4	
CH4	
Option Stoichiometric 🗸	
Stoichiometric Coefficie 1.0	
Reaction Order	
Reaction Order 1.0	
	<u> </u>

Step 35:

We select oxygen from the Child Materials list.

$$1CH_4(g) + 2O_2(g) \xrightarrow{\text{yields}} CO_2(g) + 2H_2O(l) + 891KJ$$

The stoichiometric Coefficient is taken 2 for oxygen represented in green in the previous equation. Therefore the reaction order is taken as 2.

A4 : Fluid Flow (CFX) - CFX-Pre	1.00
File Edit Session Insert Tools Help	
Outline Reaction 1	× 🛛 🛛
Details of Reaction 1	View 1
Basic Settings Reactants Products Reaction Rates	
Reactants	
Option Child Materials	
Materials List CH4,02	
Child Materials	
02	
02	
Option Stoichiometric 🗸	
Stoichiometric Coefficie 2	
Reaction Order	
Reaction Order 2	

Step 34:

Next step is to select carbon dioxide and water vapour as products of combustion as seen in the red coloured species in the equation below:

$$1CH_4(g) + 2O_2(g) \xrightarrow{\text{yields}} CO_2(g) + 2H_2O(l) + 891KJ$$

We can see that the default species is chosen as Air ideal gas, by pressing on the materials list box we can access the library of species, under the gas phase combustion list select C

A4 : Fluid Flow (CFX) - CFX-Pre		
File Edit Session Insert Tools Help		
🗐 🖓 👯 📾 🤊 🐑 🎬 🏚 🕹 🔶 🛪 📾 🖈 🙆 🗐 🗊 🗊	● 🕫 🕼 🗠 🛱 🖫 🖸	
Outline Reaction: Reaction 1	🗵 🐁 💽 ↔ @, @, @ 🗐 🗆 ▾ 🎦	
Details of Reaction 1	View 1 -	
Basic Settings Reactants Products Reaction Rates		
Products		
Option Child Materials		
Materials List Air Ideal Gas		
Child Materials	B	
Air Ideal Gas		
Air Ideal Gas		
Option Stoichiometric		
Stoichiometric Coefficie 1.0		
Reaction Order	Materials List	
	Ar Ideal Gas Ar at 25 C Ort T Solids Calorically Perfect Ideal Gases Ar Ideal Gas Water Ideal Gas Orostant Property Gases Constant Property Uquids Gas Phase Combustion Article Solids Sot Water Data Water Data	C

Step 35:

After pressing ok we find that the two species have been added under the Child Materials list. We select carbon dioxide in the Child Materials list.

$$1CH_4(g) + 2O_2(g) \xrightarrow{\text{yields}} 1CO_2(g) + 2H_2O(l) + 891KJ$$

From the previous equation we can see that the stoichiometric Coefficient is 1. Therefore the reaction order is 1.

A4 : Fluid Flow (CFX) -	- CFX-Pre
File Edit Session Ins	sert Tools Help 🤊 💌 📸 🍓 👌 🗶 🚾 📾 🕫 🕐 🗃 🖬 🖬 🖬 🖬 🖓 📲
Outline Reaction: Reaction 1	eaction 1
Basic Settings React	tants Products Reaction Rates
Option	Child Materials
Materials List Child Materials	C02,H20 •
H2O	
CO2 Option	Stoichiometric
Stoichiometric Coeffic	de 1.0
Reaction Order	1.0

Step 36:

The next step in the products section is to select water vapour shown in red in the equation below:

$$1CH_4(g) + 2O_2(g) \xrightarrow{\text{yields}} 1CO_2(g) + 2H_2O(l) + 891KJ$$

By selecting the water vapour species in the Child Materials list, we take the Stoichiometric coefficient as 2 which is seen in green in the equation above.

A4 : Fluid Flow (CFX) -	- CFX-Pre sert Tools Help	
	- · · · · · · · · · · · · · · · · · · ·] ‡ 6
Outline Reaction: Reaction: Reaction 1	action 1	2 .
Basic Settings React	tants Products Reaction Rates	ICW I
Products		
Option	Child Materials	
Materials List	С02,Н20 🗸	
Child Materials		
CO2		
H2Q		
Option	Stoichiometric	
Stoichiometric Coeffic		
-IV Reaction Order		
Reaction Order	2	
Reaction Order	2	

Step 37:

The following step is to work on the Reaction Rates section, the reaction rate equation is shown below:

$$r = k(T)[A]^a[B]^b$$

what is of our intrest is the raction rate coefficient k(T) which is also known as the arrahnuies equatiion:

$$k(T) = \frac{A}{A}e^{\frac{E}{RT}}$$

What will be required by us is to find in litrature the following values starting with the pre exponential factor:

$$A = 2.3 \times 10^{-6}$$

The activation temperature in the exponential part is:

$$T = 1500$$

The activation energy:

$$E = 2000$$

By substituting all the found values in the required cells we can finally press apply and Ok.

🔒 A4 : Fluid Flow (CFX) - C	FX-Pre		
File Edit Session Inser	rt Tools Help		
📄 🔮 🞇 🔩 🔟 🕨	🤊 💌 🛱 📾 🗗 🗶 🚾 🖬 🖈 🖉 🐼 🖗	f •	
Outline Reaction: React	tion 1	×	*13
Details of Reaction 1			Vie
Basic Settings Reactan	nts Products Reaction Rates		
- Forward Reaction Rat	te	-8-	
Option	Arrhenius		
Pre Exp. Factor 2	2.3E-6		
Temperature Exp. 1	1500		
Reaction Activation			
Option	Activation Energy		
Energy	2000 [] mol^-1]		
Backward Reaction Ra	ate	Đ	
Third Body Term		ŧ	

Step 38:

In this section we start applying the volume fractions at the boundary conditions. We start with the inlet section where we select it under the Default Domain in the Outline Section. Then Edit is selected.



Step 39:

Under the fluid values section we will consider that methane has a volume fraction of 0.5.

😰 A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
i 🕞 🗗 🚟 🖬 🞯 🦻 🗮 🧔 🕹 👌 🖉 📾 🖈 🖉 😡 🖈	3
Outline Boundary: Inlet	*12
Details of Inlet in Default Domain in Flow Analysis 1	Vie
Basic Settings Boundary Details Fluit Values Sources Plot Options	
Boundary Conditions	
CH4	
C02	
H2O	
02	
CH4	
Volume Fraction	
Ontion Value	
opuir -	
Volume Fraction 0.5	

Step 40:

Due to that we consider no combustion occuring previously we can take carbon dixode volume fraction as 0.

Edit Session I	nsert Tools	Help							_
🚯 🚟 📲 💿	000	🖥 🧔	👌 🕱 🖟	sub f 🖉 🕻) 🖸 🗇 🛛]‡ 🖻 🖬	1	🏹 🖞	" å (
Itline Boundary: 1	Inlet								×
ils of Inlet in Defau	It Domain in F	low Analysis 1							
asic Settings Bou	ndary Details	Fluid Values	Sources	Plot Options					
Boundary Conditions								E	
CH4					$\mathbf{\vee}$				
CO2			K						
H2O									
02									
CO2									
Volume Fraction			\frown					Ξ	
Option	Value		$\mathbf{\lambda}$						
option	Tenere								
Volume Fraction	ା 🚄							Vac	

Step 41:

Due to that we consider dry air is flowing into the combustor we can take the water vapour volume

fraction as 0.

🗃 A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
i 🕞 🔁 😫 📾 🤊 🥐 🎬 🏚 🕹 🔶 X 🕼 🖬 🏂 🗐 🖬 👘 🗊	Lo 🖎 🖓 📽 🔽
Outline Boundary: Inlet	×13
Details of Inlet in Default Domain in Flow Analysis 1	View
Basic Settings Boundary Details Fluid Values Sources Plot Options	
Boundary Conditions	
CH4	
C02	
H20	
02	
H20	
Volume Fraction	
Option Value	•
Volume Fraction 0	

Step 42:

Finaly a combustion process needs an oxidzer therfore we take the volume fraction of oxygen as 0.5.

😰 A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
🗄 🕞 😰 🖼 🚳 🔊 🍋 😤 🧑 🎂 👌 🛠 😡 🖬 🖈 🕹 🖉 🖓	, 🖪
Outline Boundary: Inlet	
Details of Inlet in Default Domain in Flow Analysis 1	٦ŕ
Basic Settings Boundary Details Fluid Values Sources Plot Options	
Boundary Conditions	
CH4	
Volume Fraction	
Option Value	
Volume Fraction 0.5	

Step 43:

The next step under boundary details we select an input velocity of -10 m/s. The reason for a minus sign is due to the direction of the flow in the mesh is in the opposite direction of default coordinate system in Fluid Flow(CFX). While the rest of the velocities are takenas zero. Finaly the ambient temperature is taken as 300K.

File Edit Session Insert Tools Help Image: Contract of the session Image: Contract of the session Outline Boundary: Inlet Details of Inlet in Default Domain in Flow Analysis 1 Basic Settings Boundary Details Flow Regime Option Subsonic V 0 [m s^-1] V 0 [m s^-1] W -10 [m s^-1] Turbulence Option Medium (Intensity = 5%)	
Image: Contract of the contract	
Outline Boundary: Inlet Details of Inlet in Default Domain in Flow Analysis 1 Basic Settings Boundary Details Flow Regime Flow Regime Option Subsonic Mass And Momentum Image: Cart. Vel. Components Option Cart. Vel. Components U 0 [m s^-1] V 0 [m s^-1] Turbulence E Option Medium (Intensity = 5%)	1
Details of Inlet in Default Domain in Flow Analysis 1 Basic Settings Boundary Details Fluid Values Sources Plot Options Flow Regime Option Subsonic Mass And Momentum Option Cart. Vel. Components V 0 [m s^-1] V 0 [m s^-1] Turbulence Option Medium (Intensity = 5%) 	🔀 i *b
Basic Settings Boundary Details Fluid Values Sources Plot Options Flow Regime Option Subsonic Image: Component set of the set o	
Flow Regime E Option Subsonic Mass And Momentum Image: Cart. Vel. Components Option Cart. Vel. Components U 0 [m s^-1] V 0 [m s^-1] W -10 [m s^-1] Turbulence E Option Medium (Intensity = 5%)	
Option Subsonic Image: Constraint of the second se	
Mass And Momentum E Option Cart. Vel. Components Image: Cart. Vel. Components I	
Option Cart. Vel. Components Image: Cart. Vel. Components	
U 0 [m s^-1] V 0 [m s^-1] W -10 [m s^-1] Turbulence E Option Medium (Intensity = 5%)	
0 0 [m s^-1] V 0 [m s^-1] W -10 [m s^-1] Turbulence E Option Medium (Intensity = 5%)	
V 0 [m s^-1] W -10 [m s^-1] Turbulence E Option Medium (Intensity = 5%)	
W -10 [m s^-1] E Turbulence E Option Medium (Intensity = 5%)	
Turbulence E Option Medium (Intensity = 5%)	
Option Medium (Intensity = 5%)	5
Heat Transfer E	
Option Static Temperature	
Static Temporature 300 [K]	

Step 44:

The next step is to assign the opening section the species volume fractions. By selecting the openning boundary and then selecting edit.



Step 45:

By going to the fluid values section and selecting the required species volume fractions. The opening volume fraction for methane is taken as 0.1.

A4 : Fluid Flow (CFX) - CFX-Pre			
File Edit Session Insert Tools Help			
i i i i i i i i i i i i i i i i i i i	¢.	1	1
Outline Boundary: opening			*D3
Details of opening in Default Domain in Flow Analysis 1			Vie
Basic Settings Boundary Details Fluid Values Sources Plot Options		_	
Boundary Conditions	Ξ		
CH4 CO2 H2O O2 CH4 Volume Fraction Option Value Volume Fraction 0.1		~	

Step 46:

The opening volume fraction for Carbon dixode is taken as 0.1

😰 A4 : Fluid Flow (CFX) - CFX-Pre		
File Edit Session Insert Tools Help		
🗄 🛃 😰 🖼 ன 🤊 🤨 🞬 🏚 🕹 👌 🕱 🚾 🖬 🖍 😓 🖉	: 🗗 🖥	
Outline Boundary: opening	×	*₽3
Details of opening in Default Domain in Flow Analysis 1		Vie
Basic Settings Boundary Details Fluid Values Sources Plot Options		
Boundary Conditions		
CH4		
	1	
02		
Volume Fraction		
Option Value 🗸		
Volume Fraction 0.4		

Step 47:

The opening volume fraction for water vapour is taken as 0.4.

😰 A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
🔚 🗗 🛤 📾 🕫 📬 🖬 🕲 🔌 🕹 🚸 🖉 🖬 🔊 🕲 👘	R
Outline Boundary: opening	*13
Details of opening in Default Domain in Flow Analysis 1	Vie
Basic Settings Boundary Details Fluid Values Sources Plot Options	
Boundary Conditions	
CH4	
C02	
H20	
02	
H20	
Volume Fraction	
Option Value	
Volume Fraction 0.4	

Step 48:

The opening volume fraction for oxygen is taken as 0.1.

A4 : Fluid Flow (CFX) - CFX-Pre	
File Edit Session Insert Tools Help	
📄 🗿 🕰 🔩 🚳 🤊 🤊 🎬 🏚 🕹 🔶 🗴 📾 🖬 🖈 🖁 🚳 🛣 👘	f = 1
Outline Boundary: opening	🛛 🗄 🏷 🔤
Details of opening in Default Domain in Flow Analysis 1	View 1
Basic Settings Boundary Details Fluid Values Sources Plot Options	
Boundary Conditions	
CH4 CO2 H20 O2 Volume Fraction Option Value Volume Fraction 0.1	

Step 49:

The next step is to go to the boundary details and choose the Relative Pressure as 101325 Pa.

ł	A4 : Fluid Flow (CFX) -	CFX-Pre		
Fi	e Edit Session Ins	ert Tools Help		
	- 🖸 😤 🔩 🗃	🕐 🖻 🚰 🍐 👌 🌠 🚮 🛧 🛛 🕐 🖉 🗇 📭 🗊 🗱 🎠 🗽	₽	
	Outline Boundary: op	ening	X	: *n
De	etails of opening in Defa	ault Domain in Flow Amalysis 1		- 43 Vie
16	Basic Settings Bound	Jary Details Fluid Values Sources Plot Options		Vie
ľ	Flow Regime		-8-	
	Option	Subsonic		
	Mass And Momentum	\square		
	Option	Opening Press and Dirp		
	option			
	Relative Pressure	101325 [Pa]		
	Flow Direction			
	Option	Normal to Boundary Condition 👻		
	Loss Coefficient		Ŧ	
	Turbulence			
	Option	Medium (Intensity = 5%)		
	Heat Transfer			
	Option	Opening Temperature		
	Opening Temperature	300 K V		

Step 50:

In this step we edit the wall boundary condition for the located sphere inside the meshed domain, so by right clicking on the wall icon and then selecting the edit option.

e Edit Session Insert	Tools Help	
] 🗿 🚟 🗣 🔟 🤊	e 👫 🏚 🕹 👌 🛪 🚾 🖬 🏞 🕓 🗃 📭 🗃	' 💋 🖁 🗠 🖄 🖾 🔰 '
Dutline		
🙆 Mesh		
▲ 👰 CFX.cmdb		
Principal 3D Region	ns	
4 📝 🌍 B16		
4 Principal 2	D Regions	
v 🎾	F17.16	
V 🎾	F18.16	
V 🎾	F20.16	
v 🔰	F21.16	
v 🔰	F22.16	
V 1	F23.16	
V 🔰	Inet	
Ginulation		
4 Row Analysis 1		
Analysis Type		
✓ Analysis type ✓ Ø Default Dor	ain	
☑ 1 ‡ Inlet		
V 1 opening		
V]‡ wall		
🚺 Interface	Edit	
4 😥 Solver 📻	Mash Statistica	
🙎 kš Solutic 🕮	Mesh Statistics	
Solver	Render +	
Outpu	Show	
Coordinate F	Hide	
Materials 1	Thue Contract of the Contract	
A Reactions	Edit in Command Editor	
Reaction 1 —		
4 R Evenessions	Сору	
▲ 🔀 Expressions,	Duplicate	
Expressions, Additiona Expressia	bopicate	
Expressions, Additiona Expression Expression	Delete	
Kyressions, Additiona Expressic Suser Fun Wyser Rou	Delete	
Expressions, Expressions, Additiona Additiona Expressic Expressic User Fun Weer Rou Weer Rou Weer Rou	Delete Rename	
Expressions, Additions Expressic Expressic User Fun User Rou B Gonfigurations	Delete Rename	

Step 51:

Then in this step we go to the Boundary Details and apply the Fixed Temperature value of 2000K and then press apply and Ok.

B A4 : Fluid Flow (CF)	X) - CFX-Pre		
File Edit Session	Insert Tools Help		
📑 🕑 🚟 🔩 🛛	o 🕫 🗢 🖬 🏚 🕹 👌 🗴 🜠 💽 🖌 🙆 🖉 👘 🎼 🖄	A 4	
Outline Boundary	: wall	×	*\; 🖸 💠 @, €
Details of wall in Defau	ult Domain in Flow Analyzis 1		View 1 👻
Mass And Momentu	m	8	
Option	No Slip Wall 👻]	
Wall Velocity		±.	
Wall Roughness			
Option	Smooth Wall		
Heat Transfer			
Option	Temperature)	
Fixed Temperature	2000 K 🗸		

Step 52:

Congratulations the setting up process is finished go back to the project schematic view and double click on solution.



Step 53:

You can use the serial mode to run your calculation but I would prefer to make use of the quad core machine I am working on.

A5 : Fluid Flow	(CFX) - CFX-Solver Manager	
File Edit Work	space Tools Monitors Help	
1 😤 😤 👔		
Workspace	Co Define Run	
	Solver Input File (dp0\CFX\CFX\Fluid Flow CFX.def)	
	Run Mode HP MPI Local Parallel	
	Host Name Partitions	
	Show Advanced Controls	
	Start Run Save Settings Canc	

Step 54:

You should get and output that looks like something like this .

