Introductory Tutorial on ANSYS

Zhenghong Zhou

zhenghoz@usc.edu

• DOWNLOAD

USC AME now has a department license for ANSYS, which includes FLUENT (computational fluid mechanics software, including heat and mass transfer) and Chemkin Pro (chemical kinetics simulation software). The ANSYS package also includes simulation packages for solid mechanics and electromagnetics, but we will not use these in this course.

The license allows you to use ANSYS via USC internet connections. If you want to use the licensed software off-campus, you will need to connect to USC via VPN to get the ANSYS license server to work.

Please send an email to Professor Ronney for access to the ANSYS Google drive or email me for a copy of the software.

There are two versions of ANSYS software plus a pdf instruction in the Google drive. Both versions should work, and this tutorial will use ANSYS 2019R3.

Open either one of the folders and download both disk 1 and disk 2 depending on your PC system (Windows or Linux).

Name	Date modified	Туре	Size
ANSYS2019R3_WINX64_Disk1	1/15/2020 2:34 PM	Disc Image File	6,239,126 KB
ANSYS2019R3_WINX64_Disk2	1/15/2020 2:33 PM	Disc Image File	6,100,338 KB

Figure 1. Disc images 1 and 2

• INSTALLATION

Load the two disks, open both of them, and then open "setup" in disk 1.

Wait until the ANSYS Installation Launcher pops out and click on "Install ANSYS Products".

💿 🛃 📕 🖛	Manage	VD Drive (E:) ANSYS2019R3			_	\Box \times
File Home Share View	Application Tools					~ 🕐
\leftarrow \rightarrow \checkmark \uparrow \bigcirc \rightarrow This PC \rightarrow DVE	D Drive (E:) ANSYS2019R3		ٽ ~		E:) ANSYS2019R3	
		Name ^	Date modified	Туре	Size	^
🖈 Quick access		licsery	8/12/2019 6:21 AM	File folder		
E Desktop	*	Incenter	8/12/2019 6:21 AM	File folder		
🕂 Downloads	*	Isdyna	8/12/2019 6:21 AM	File folder		
🛷 Google Drive	*	manifest	8/12/2019 6:21 AM	File folder		
Documents	*	meshing	8/12/2019 6:21 AM	File folder		
Pictures		polyflow	8/12/2019 6:21 AM	File folder		
- Treates	~	prereq	8/12/2019 6:21 AM	File folder		
o Creative Cloud Files		rsm	8/12/2019 6:21 AM	File folder		
This PC		sec	8/12/2019 6:21 AM	File folder		
2D Objects		sgcharts	8/12/2019 6:21 AM	File folder		
		shapeoptimization	8/12/2019 6:21 AM	File folder		
Desktop		solver	8/12/2019 6:21 AM	File folder		
Documents		spaceclm	8/12/2019 6:22 AM	File folder		
👆 Downloads		subs	8/12/2019 6:22 AM	File folder		
b Music		syscplg	8/12/2019 6:22 AM	File folder		
E Pictures		tp	8/12/2019 6:22 AM	File folder		
Videos		📊 util	8/12/2019 6:22 AM	File folder		
Windows (C:)		voxel	8/12/2019 6:22 AM	File folder		
My Passport (D)		195-1.dvd	8/12/2019 6:22 AM	DVD File	1 KB	
		📄 builddate	8/12/2019 6:22 AM	Text Document	4 KB	
DVD Drive (E:) ANSYS2019R3		LICENSE	8/12/2019 6:22 AM	Text Document	53 KB	
DVD Drive (F:) ANSYS2019R3		📄 package.id	8/12/2019 6:22 AM	ID File	1 KB	
My Passport (D:)		🗸 🧥 setup	8/12/2019 6:22 AM	Application	982 KB	~
66 items 1 item selected 981 KB						





Figure 3. ANSYS Installation Launcher

When the ANSYS Installation Launcher asks for license server specification, please follow the instructions in Figure 4.

ANSYS Licensing interconnect port number: 2325

ANSYS FlexNet port number: 1055

Hostname: vsoelics-01.usc.edu

2019 R3 Product Installation -	'Windows x64"		-	
2019 R3			AN	SYS [®]
				00
	Enter License Server Specification:			
ANSYS	ANSY'S Licensing Interconnect port number: (default is 2325) 2325			
	ANSYS FlexNet port number: (default is 1055)			
	1055			
	Hostname:			
Realize Your Product Promise®	vsoelics-01.usc.edu			
	_			
	Skip this step and configure later			
for Installation Help		< Back	Next >	Exit

Figure 4. License Server Specification

Click "Next", and a window will pop out for you to choose the desired bundles. Please make sure "Chemkin" and "Fluent" are checked. When I was installing, I used all of the default responses.

Hit "Next". It takes about 10 minutes or so to finish the installation, during which time the Installation directory will request the path to next media as in Figure 5. Choose disk 2 ("F:" in my case) and continue.

When the task bar hits 100%, click on "Exit" and you should be ready to go.



Figure 5. Installation directory #2

• CHEMKIN PRO

For Windows users, you could either search "Chemkin" in the start menu or go to the folder "Chemkin" shown in Figure 6.

📙 🕑 📑 🔫 ANSYS 2019 R3							
File Home Share View							~ 🕐
\leftarrow \rightarrow \checkmark \uparrow \bigcirc « ProgramData \rightarrow Microsoft \rightarrow Windo	ws≯	Start Menu > Programs > ANSYS 2019 R3	~	ū		2019 R3	
	^	Name	Date modifie	ed	Туре	Size	^
📌 Quick access		ACP	1/16/2020 11	:38 AM	File folder		
E Desktop	*	Additive	1/16/2020 11	:38 AM	File folder		
🖶 Downloads	*	ANSYS Client Licensing	1/16/2020 11	:35 AM	File folder		
参 Google Drive	*	ANSYS Discovery Suite	1/16/2020 11	:38 AM	File folder		
Documents	*	Aqwa	1/16/2020 11	:38 AM	File folder		
Pictures	*		1/16/2020 11	:38 AM	File folder		
		Fluid Dynamics	1/16/2020 11	:38 AM	File folder		
o Creative Cloud Files			1/16/2020 11	:38 AM	File folder		
💻 This PC			1/16/2020 11	:38 AM	File folder		
3D Objects		- Meshing	1/16/2020 11	:38 AM	File folder		
Desktop		Remote Solve Manager	1/16/2020 11	:35 AM	File folder		
Documents		Utilities	1/16/2020 11	:38 AM	File folder		
Deumlands		💒 ANSYS Icepak 2019 R3	1/16/2020 11	:38 AM	Shortcut	2 KB	
Downloads		ANSYS Viewer 2019 R3	1/16/2020 11	:38 AM	Shortcut	2 KB	
Music		DC Evaluator (DCE) 2019 R3	1/16/2020 11	:35 AM	Shortcut	2 KB	
Pictures		Design Point Service (DPS) 2019 R3	1/16/2020 11	:35 AM	Shortcut	2 KB	
Videos		Mechanical 2019 R3	1/16/2020 11	:38 AM	Shortcut	2 KB	
🏥 Windows (C:)		Mechanical APDL 2019 R3	1/16/2020 11	:38 AM	Shortcut	2 KB	
My Passport (D:)		Mechanical APDL Product Launcher 2019	. 1/16/2020 11	:38 AM	Shortcut	2 KB	
DVD Drive (E:) ANSYS2019R3		SCDM 2019 K3	1/16/2020 11	:35 AM	Shortcut	2 KB	
DVD Drive (F:) ANSYS2019R3		System Coupling 2019 R3	1/10/2020 11	:35 AM	Shortcut	2 KB	
		Wardshamah 2010 P2	1/10/2020 11	:55 AM	Shortcut	2 KB	
My Passport (D:)	~	Workbench 2019 R3	1/10/2020 11	:30 AM	Shortcut	I KB	¥
23 items 1 item selected							

Figure 6. CHEMKIN PRO file location



Figure 7. Create shortcuts for ANSYS

ANSYS Chemkin-Pro 2019 R3 (LAPTOP-463DHQSS) -			
Project Edit View Utility Help Open Projects Open Projects Models Running CHEMINN-PRO Running C	ANSYS Chemkin-Pro 2019 R3 (LAPTOP-4E3DHQ58)	_	×
Open Projects Open Projects Open Projects Running CHEMINN-PRO	Project Edit View Utility Help		
Open Projects Open Projects Models Running CHEMINN-PRO. Maximum jawa memory allowed: 910MB. No projects were open when last session exited. So, none are reopened now. Maximum jawa memory allowed: 910MB.			
Maximum java memory allowed: 910MB. No projects were open when last session exited. So, none are reopened now.	Open Projects Open Projects Models		
	Maximum java memory allowed: 910MB. No projects were open when last session exited. So, none are reopened now.		

Figure 8 shows the basic window of Chemkin-Pro.

Figure 8. Chemkin-Pro window

Go to "Help" \rightarrow "Getting started" or "Tutorial", you will be redirected to <u>https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v195/chemkin_gs/section_h2m_vns_kbb.html</u>

where detailed explanations and instructions of the software can be found.

In this tutorial, the project "flame speed freely propagating" is used as an example to show the basics of Chemkin-Pro.

Press "Ctrl+O" or click on "Project" \rightarrow "Open...", go to "samples 2010" folder and then elect "flame_speed__freely_propagating.ckprj". The "flame speed freely propagating" project should appear under Open Project as Figure 9 shows.

Diagram View helps to visualize the simulation process. If everything is set up correctly, all the blocks will be green. If a certain block turns red, it means an error occurred in the corresponding step.



Figure 9. Diagram View

Remember to run Pre-Processing before any further actions. Double click on Pre-Processing, and a window shown in Figure 10 will pop out. This window allows you to select the Chemistry Set (chemical mechanism, e.g. USC Mech II) you want to use. If you are satisfied with ANSYS inciv195 which is the default chemistry set, click on Run Pre-Processor. Notice that Gas-Phase Kinetics File, Thermodynamics Data File and Gas Transport Data File can be modified separately.

ANSYS Chemkin-Pro 2019 R3 (LAF	PTOP-4E3DHQ58)				-		×
Project Edit View Utility Help							
	9 🗙 🍃 🍯 🖻 🗹 💕						
Open Projects	Pre-Processing (flame_speed	dfreely_pro	pagating)			📰 r 🗹	X
Hand_program View Diagram View Pre-Processing Flame_Speed (C1) Run Calculations Analyze Results Monitor Project Pun	Chemistry Set 🛛 🔌 Mechanisn	n Viewer 🍸	Mechanism Parameter Workfood Snip	S			
Monitor Project Run	Working Dir C:\Users\zho	ouz\chemkin\s	amples2010\flame_speed	d\freely_propagating		- 8,2	
	Chemistry Set zhouz\chemi	kin\samples2	010\flame_speed\freely_p	ropagating\flame_speed_	_freely_propagating.ck	s 🔻 🗞	
	New Chemistry Set	Edit Chemistry Set	Run Pre-Processor	View Results	Hide Detail		
	Short Name	flame_speed	freely_propagaling				
	Description						
	Gas-Phase Kinetics File						
	Thermodynamics Data File	C:\Program F	iles\ANSYS Inc\v195\reaction	on\data\therm.dat		370 2	
	Gas Transport Data File	C:\Program F	iles\ANSYS Inc\v195\reacti	on\data\tran.dat		2	
		Process Tra	nsport Properties Fit with	n Normal Output 💌			
		Perform	Data Format Check on Ch	emistry Input Files			
Open Projects Models							
System connectivity validated. Updatin Beginning flame_speedfreely_prop System connectivity validated. Updatin	ng Open Projects tree. agating Network Diagram Validati ng Open Projects tree.	ion					

Figure 10. Pre-Processing window

After running Pre-Processing, double click on "Flame_Speed (C1)" \rightarrow "C1_FlameSpeed" and you'll see what's in Figure 11. You can hover your mouse over each option and a detailed explanation will be displayed. I set the reactor physical properties as shown in Figure 11. You can experiment with the Grid Properties and Species-specific Properties; the simulation accuracy and simulation time will change respectively.



Figure 11. Reactor Physical Properties

Go to C1_inlet1, input your guess of the inlet velocity or the mass flow rate. Again, you can use "HELP" to look for detailed explanation on the flame propagation model. Species-specific Properties can also be changed. Notice that the default value gives a stoichiometric $CH_4 + O_2$ mixture with N_2 as the diluent gas. Changing any of the mole fraction will change the mixture and thus the adiabatic flame temperature and the flame speed.

Click on "Run Calculations" and begin simulation. When the simulation is done, the Analyze Results window will appear as Figure 12 shows. You can choose to plot results or analyze reaction paths.



Figure 12. Analyze Results window

If you choose to plot results, after clicking on "Next Step...", you will be able to further choose the data you need. Also, if you choose "use Excel to post-process", the results will be presented as Excel files (see Figure 13).

Tabs 1, 2, 3 show the detailed solution of properties of the mixture before and after combustion. Tab 4 shows the flame speed results. The three flame speeds in Figure 13 is acceptable because they are roughly the same. However, if you go back and refine the grid properties, the three results could be more nearly identical.

	AutoSave 🤇		9 ° C	~ ⇒ B	ook1 - Exce	el 🔎	Search				Hari	ris Zhou HZ	Ē	-//		×
F	ile Hor	ne Inse	ert Pag	je Layout	Formula	is Data	Reviev	v View	Add-in:	s Help	Acrob	oat	ය Shar	e 🖓 🗘	Commen	ts
Pa	aste ✓ ≪	Calibri B I	<u>∪</u> ~ ⊞	 11 ✓ 11 ✓ 4 ✓ 4<td></td><td></td><td>© Ge ₽ ~ \$ ~ €.00</td><td>neral ✓ ✓ % 9</td><td>🔛 Cond 🞲 Form 📝 Cell S</td><td>itional Form at as Table ~ ityles ~</td><td>atting ~</td><td>Elinsert v Delete v</td><td>∑ ~ ↓ ~ ♦ ~</td><td>27 ~ 0 ~</td><td>Ideas</td><td></td>			© Ge ₽ ~ \$ ~ €.00	neral ✓ ✓ % 9	🔛 Cond 🞲 Form 📝 Cell S	itional Form at as Table ~ ityles ~	atting ~	Elinsert v Delete v	∑ ~ ↓ ~ ♦ ~	27 ~ 0 ~	Ideas	
	ippoard is		Font		121	Anghinent		iumber is	1	Styles		Cells	i Eu	ung	lueas	
A:	L	• • • •	×	f _x Sol	ution_No	_()										~
	А	В	С	D	E	F	G	Н	1	J	К	L	М	N	0	
1	Solution_	Number_	Flame_sp	eed_(cm/	sec)											-
3	2.00E+00	2.10E+01	4.30E+01 4.27E+01													
4	3.00E+00	5.50E+01	3.99E+01													
5																
6																
7																-
8																-
10															_	-
11															_	
12																
13																
14																
15																-
10																
18																
19																
20																
21																
22																_
23																
24		2 colo	no 2 3	coln no 2	4 noint	i value va a	umbor	A								
	• • •	. 2.soln_	10_2 3	.soin_no_3	4.point	_value_vs_t	lumber	÷	:	4						P
											=	۵ Ľ	-	1	- + 10	0%

Figure 13. Simulation results in Excel

If you choose to analyze reaction paths, the Reaction Path Analyzer will be called, and you can do further analysis within this window as in Figure 14.



Figure 14. Reaction Path Analyzer

ANSYS Workbench + Fluent Basics (with just heat conduction, no flow or combustion)

Open Workbench 2019 R3 and save the new project with a proper name. Find "Fluid Flow (Fluent)", left click and drag it into the Project Schematic window.



Figure15. Project Schematic window

In this tutorial, a 2D-Heat Transfer in a rectangular region is used as an example.

The first setup involves creating the required geometry for your problem. Besides ANSYS itself, any commonly used CAD software should work. I used Solidworks to create the geometry. When the simulation is 2D, make sure that the sketch plane is the X-Y plane in the CAD software. After building the geometry, save the file as Parasolid (*.x_t) file. Go back to Workbench and right click on "Geometry", and then select "Import Geometry" to import the surface you created.





Next, right click on "Mesh" and select "Edit…" and the meshing window will appear. Use "Ctrl+E" or click on the Edge selection as Figure 17 shows and select the left side of the rectangle. Then edge would turn green if selected. Right click on the selected side, select "create Named Section", name it "hot boundary", then hit "OK". Hold "Ctrl" and left click on the rest three sides, and name them "cold boundaries".

Figure 17. Graphics Toolbar

Click on "Mesh" and you could change the mesh properties in the left bottom Details of "Mesh" window. I changed the Mesh size to 0.001m. In some situations, you may need to have finer mesh at the boundaries. One way of doing this is to right click on "Mesh" and insert inflation. "Ctrl+B" \rightarrow select the surface \rightarrow click on "no selection" next to "Geometry" and then apply. "Ctrl+E" \rightarrow select the four sides \rightarrow click on "no selection" next to "Boundary" and then apply. You could adjust the maximum layers and growth rate as you like. Also, if you want to create different inflations for different boundaries, just create another inflation and repeat the procedure.

Generate mesh, save the mesh file, go back to Workbench and update the project until you see a green "check" symbol next to "Mesh". Then, open "Setup". When the Fluent launcher pops out, you can choose if you want double precision and parallel processing.



Figure 18. Generate Mesh



Figure 19. Inflation



Figure 20. Mesh with inflation

Expand models and right click on "Energy" to switch on the energy equation. You can choose the materials to be solid or liquid in "Cell Zone Conditions".

The most important step is to set up the boundary conditions. Double click on "cold_boundaries", go to "thermal", and set the temperature to 300K. By the same procedure, set the "hot_boundary" temperature to 800K.

Right click on initialization and hit initialize. You could also change the initialization method. Double left clicks on "Run Calculation", put 20 in the "Number of Iterations" and hit calculate. Wait until the "Calculation complete" window appears, and the results are ready to be checked.

For example, if we double click on Contours, select "temperature" in the float window, and hit "Save/Display", a temperature contour of the rectangular region will be generated as Figure 22 shows. You could play with Graphics, Plots and Animations to see what happens.

If we close the Fluent Calculation window and go back to open "Results" in Workbench, we can further customize more detailed contours, plots, reports, etc.

If you're looking for more complicated examples or detailed tutorials, there are tons online either in the form of videos or interactive webpages (ANSYS learning hub).



Figure 21. Outline View



Figure 22. Temperature Contour

2D premixed combustion problem

First, create a 50cm×500cm rectangular surface and then import it into a new project (I call it 2D-Combustion). For the Named Sections, name the left side edge as "inlet", right side edge as "outlet", top and bottom sides as "walls". Generate 0.01 m uniform mesh for the surface as Figure 23 shows.

Outline	↓ □ ×	Q Q 📦 📦 🎲 😘	🔾 - 🛟 🍳 🕘 🍭 😒 sel	ect 🐂 Moder 😰 🗊 🕤	🕞 📴 📾 📾 🐨 🌱 💎	Clipboard -	[Empty] 😜 Extend -	9 Select By *	Convert -	
Name 💌 Se	arch Outline 🔽 🖕									
Project										
B- B Model (A3)										
💮 🗸 🐨 Geometry										
Coordinate	Systems									
Mesn Named Sala	stione									
En Eg Nalled See	Cooris									
Thoutet										
Datails of "Mash"	- 1 - 2									
Dirplay										
Display Style	Lice Geometry Setting									
Defaults	ose ocometry setting									
Physics Preference	CED									
Solver Preference	Fluent									
Element Order	Linear									
Element Size	1.e-002 m									
Export Format	Standard									
Export Preview Surface	Mesh No									
+ Sizing										
Quality										
Inflation										
Batch Connections										
Assembly Meshing										
Advanced					0.000		1.000		2.000 (m)	
Statistics									Lines (in)	
						0.500		1 500		
						0.300		1.500		

Figure 23. Mesh for 2D-Combustion

After generating the mesh, save the project and go back to the Workbench. Update the mesh and wait until the yellow lightening icon becomes a green check.

Open "setup". Turn on the energy equation in "Models".

For "Viscous" and "Species", please follow what's shown in Figure 24. Here we selected the k- ε viscous model which you'll learn about later in AME 513b; for now just accept that the k- ε model is a simplified way of modeling the mass, momentum and energy transport caused by turbulence without having to solve the 3D unsteady Navier-Stokes equations directly; it assumes that the turbulence is *isotropic*, that is, the same in all coordinate directions. You could also try the "Reynolds Stress" (an extension of k- ε that allows for anisotropic turbulence) or the "Laminar" (which suppresses all turbulence effects) viscous model to see what happens.

In "Species Model", select "Species Transport" and make sure that "Volumetric" is checked. Also, check "Eddy-Dissipation" in the "Turbulence-Chemistry Interaction". (The eddy-dissipation model essentially assumes that the rate of reaction is determined by the rate of turbulent mixing of products with reactants and that reactants react infinitely rapidly once mixed with products; we'll discuss this later in AME 513b). I've tried multiple cases and believe that even if the flow speed is low, the "Eddy-Dissipation" still works best because of the existence of boundary layers and thus some turbulence. You can manually change the turbulence intensity, but if it is set to zero, the computation might not converge.

Figure 25 shows how the inlet boundary conditions are set. And for the wall boundary conditions, make sure "Heat Flux" is checked in the "thermal conditions" and anything else is zero.

🚺 Viscous Model	×	Species Model	×
Model	Model Constants		
Inviscid Laminar Spalart-Allmaras (1 eqn) is k-epsilon (2 eqn) transition k-kl-omega (3 eqn) Transition SST (4 eqn) Reynolds Stress (5 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) k-epsilon Model is Standard RNG Realizable Near-Wall Treatment is Standard Wall Functions	Cmu 0.09 C1-Epsilon 1.44 C2-Epsilon 1.92 TKE Prandtl Number 1 TDR Prandtl Number 1.3 Energy Prandtl Number 0.85 User-Defined Functions Turbulent Viscosity none	Model Off Species Transport Non-Premixed Combustion Premixed Combustion Partially Premixed Combustion Composition PDF Transport Reactions Volumetric Wall Surface Particle Surface Electrochemical	Mixture Properties Mixture Material methane-air Edit Import CHEMKIN Mechanism Number of Volumetric Species 5 Turbulence-Chemistry Interaction Finite-Rate/No TCI Finite-Rate/Eddy-Dissipation Eddy-Dissipation Eddy-Dissipation Concept Coal Calculator
Scalable Wail Functions Non-Equilibrium Wall Functions Enhanced Wall Treatment User-Defined Wall Functions Options Uscous Heating Curvature Correction Compressibility Effects Production Limiter OK Ca	Prandtl and Schmidt Numbers TKE Prandtl Number none TDR Prandtl Number none Energy Prandtl Number none Wall Prandtl Number none Turbulent Schmidt Number ancel Help	Chemistry Solver None - Explicit Source ▼ Options □ Inlet Diffusion □ Diffusion Energy Source □ Full Multicomponent Diffusion □ Thermal Diffusion OK Appl	Select Boundary Species Select Reported Residuals

Figure 24. Setups in "Models"

🖪 Veloo	ity Inlet							×	🖪 Velocity I	nlet						×
Zone Nar	ne								Zone Name							
inlet									inlet							
Moment	um Th	nermal	Radiation	Species	DPM	Multiphase	Potential	UDS	Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS
Velo	ocity Spe	ecificatio	on Method	Magnitude,	Normal to	Boundary		•		Specify Spe	cies in Mole	Fractions				
		Referer	nce Frame	Absolute				•	Species Ma	ss Fractio	ns					
	v	/elocitv	Magnitude	(m/s) 0 02				_	ch4 0.	.1				•		
Superco	nic/Initia	al Cauro	o Proceuro	(naccal) 0				- I	o2 0.	2				•		
Superso		ar Gaugi	e riessure	(pascal) 0				•	co2 0					-		
	Sner	nce cificatio	n Method	Intensity an	1 Viscosity	Ratio		-	h2o 0							
	Tu	irhulent	t Intensity ((%) 5	, ,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,			_								
	Turbuler	nt Visco	osity Ratio	10				-								
				OK Can	el) (Help	,					l	OK Can	cel Hel	р		

Figure 25. BCs at the inlet

Next, go to "Solutions" \rightarrow "Monitors" \rightarrow "Residuals" and set it as what Figure 26 shows. Initialize the project by clicking on the "Initialize", and then click on "Patch…" as Figure 27 shows. Make sure that the surface in "zones to Patch" is also selected, and then click on "Patch". This gives the computer an initial input for the computation, which would make the convergence easier.

Note that "Residuals" is a source of great confusion - everybody who uses FLUENT reports it but almost nobody knows what it means. It's a measure of how nearly a conservation equation (e.g., x-velocity, which should read x-momentum since velocity is not a conserved quantity but momentum is, or more precisely, the rate of change of momentum is equal to the sum of the forces) is satisfied.

If you want to monitor an animation of the computation. You can go to "Calculation Activities and follow what Figure 28 shows.

Residual Monitors					\times
Options	Equations				
✓ Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	A
✓ Plot	continuity	✓	✓	1e-06	
Window	x-velocity	✓	✓	1e-06	
1 Curves Axes	y-velocity	✓	✓	1e-06	
Iterations to Plot	energy	✓	✓	1e-08	
1000	k	✓	\checkmark	1e-06	
Iterations to Store	epsilon	✓	\checkmark	1e-06	
1000	ch4	✓	\checkmark	1e-06	
	02	✓	\checkmark	1e-06	
	co2	✓	✓	1e-06	
	h2o	✓	✓	1e-06	
					-
R	esidual Values		Conve	ergence Criterion	
	Normalize	Ite	abso abso	lute	•
		5	÷		
E	✓ Scale		Con	vergence Conditi	ons
	Compute Local S	cale			
ОК РІ	ot	Cance	l) (Help)		

Set the number of iterations to 250 and start calculation.

Figure 26. Residual Monitors

Reference Frame	Value (k)	
		Zones to Patch Filter Text 📃 🚍 🚍
	2000	
Absolute	Use Field Function	Π_surface
Variable		
Pressure	Field Function	
X Velocity		
Y Velocity		
Turbulent Kinetic Energy		Registers to Patch [0/0]
Turbulent Dissipation Rate		
ch4		
02		
h2o	•	
	Patch	Help
	Figure 27.	Patch
Internal Outlet		
Structure (married (married)	a autor D Animation Definit	tion
outlet (pressu	re-outler Animation Definit	
	Name: animation-2	
🖉 Dynamic Mesh	anna anna anna anna anna anna anna ann	
🔁 Reference Values	Record after every	1 Iteration
🛛 🔼 Reference Frames	Charmen Tama	
Named Expressions	Storage Type	HSF File 🔻
olution	Storage Directory	
% Methods	ye bir cabiy .	
% Controls	Window Id	4
Poport Definitions		· · · ·
Report Definitions	Animation View	front Preview Use Active
Monitors		
🖄 Residual		
🖻 Report Files	Animation Object	
Report Plots		
X Convergence Cor	ditions residuals	
Cell Registers	temperature	
	velocity	
Coloulation Antiviti	vector-1	
 Calculation Activities 		
🔁 Autosave (Every I	terations	
🕒 Execute Comman	ds	
😑 🏶 Solution Animation	15	2
animation-1		
Cell Register One	rations	
Run Calculation		
	Now Object	dit Object
esuits	new object -	uic object
Surfaces		
) 🤎 Graphics		OK Canaal IIIala
) 💆 Plots		
Scene Scene		





Figure 29. The temperature contour for 2D-Combustion



Figure 30. The velocity contour for 2D-Combustion

From the temperature contour we could tell that the premixed flame sheet is a very curved layer despite that our initial velocity of the flow is only 2cm/s. Compare the temperature results with the velocity contour, we may say that the curved profile is due to the thermal expansion effect. That is, the burnt gas density is drastically dropped, and the flow is accelerated after combustion.

You could try:

- 1. Changing the inlet velocity and the premixed mixture mole fractions.
- 2. Turning on radiation or giving the wall boundaries a fixed temperature.
- 3. Changing mesh size.
- 4. Having two inlets instead of one, which makes the combustion mostly non-premixed.