Introductory Tutorial on ANSYS

Zhenghong Zhou

zhenghoz@usc.edu

- **DOWNLOAD**

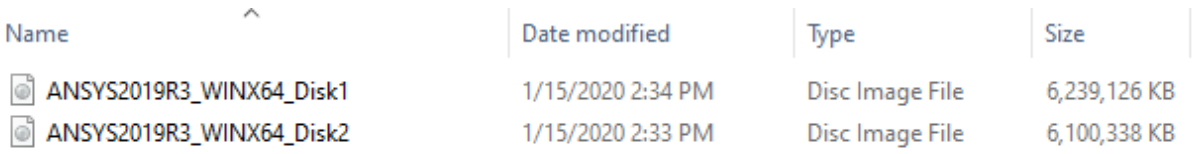
USC AME now has a department license for ANSYS, which includes FLUENT (computational fluid mechanics software, including heat and mass transfer) and Chemkin Pro (chemical kinetics simulation software). The ANSYS package also includes simulation packages for solid mechanics and electromagnetics, but we 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	Type	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".

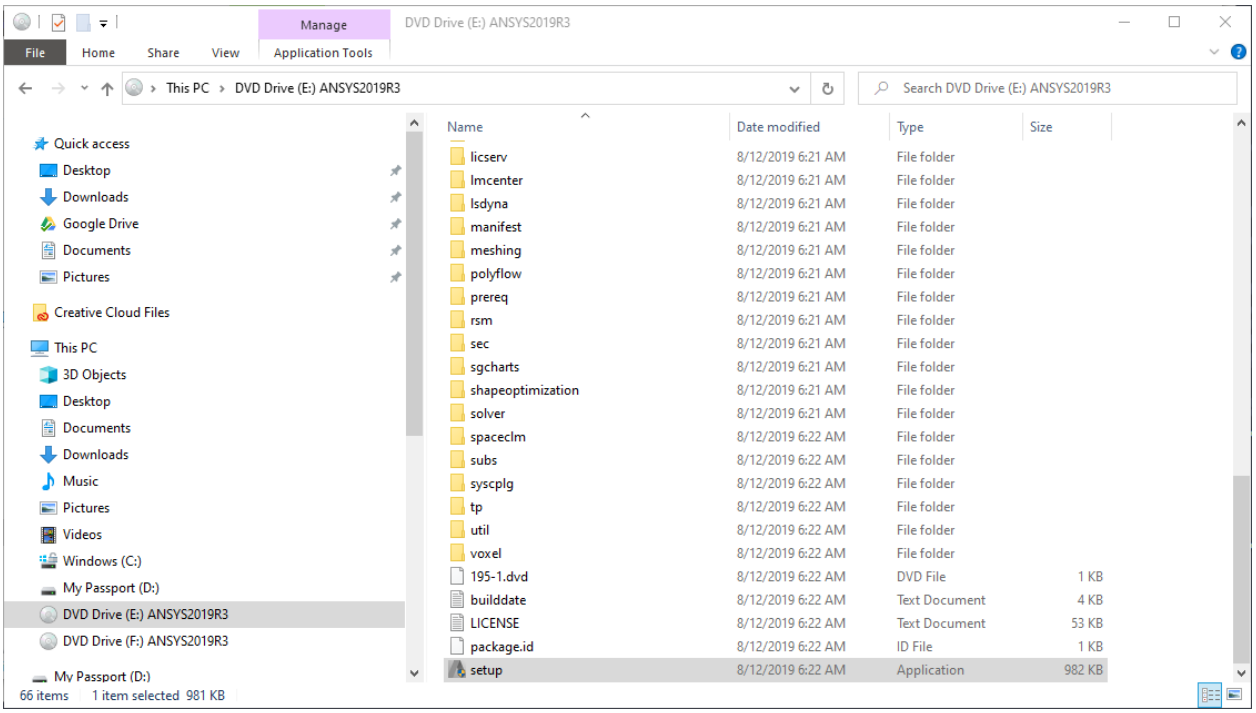


Figure 2. Open "setup"

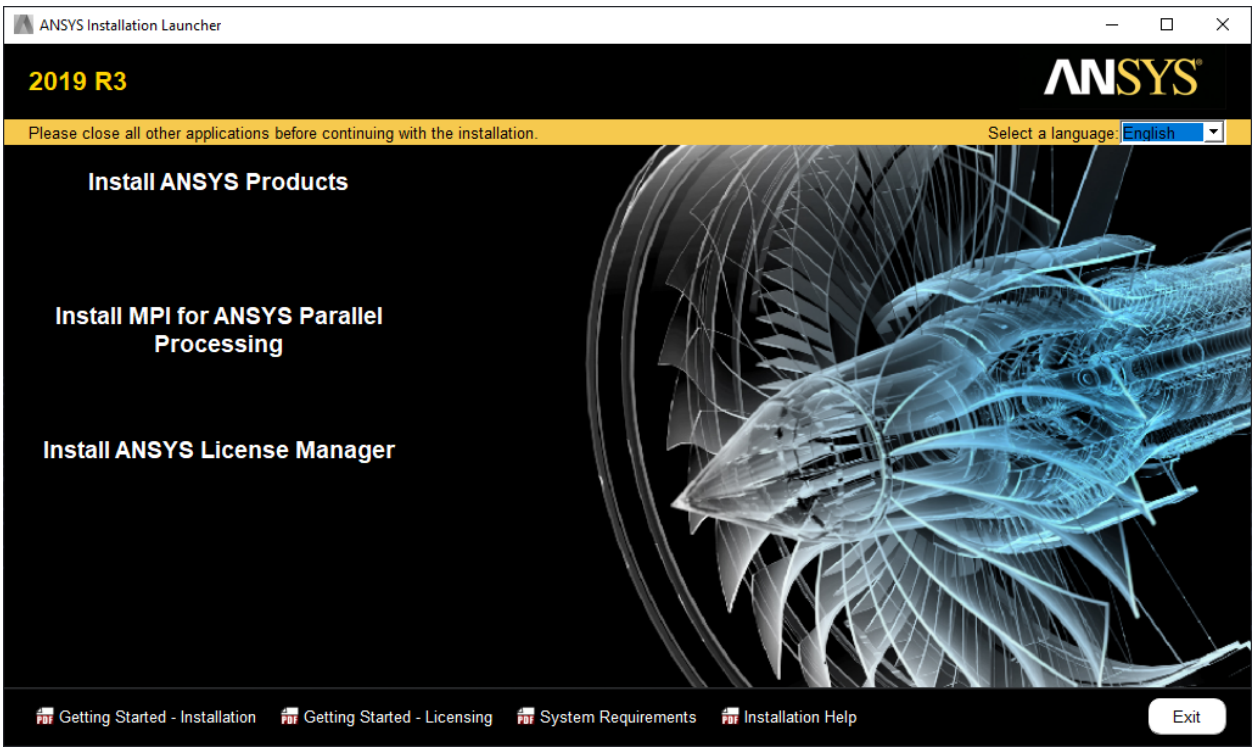


Figure 3. ANSYS Installation Launcher

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

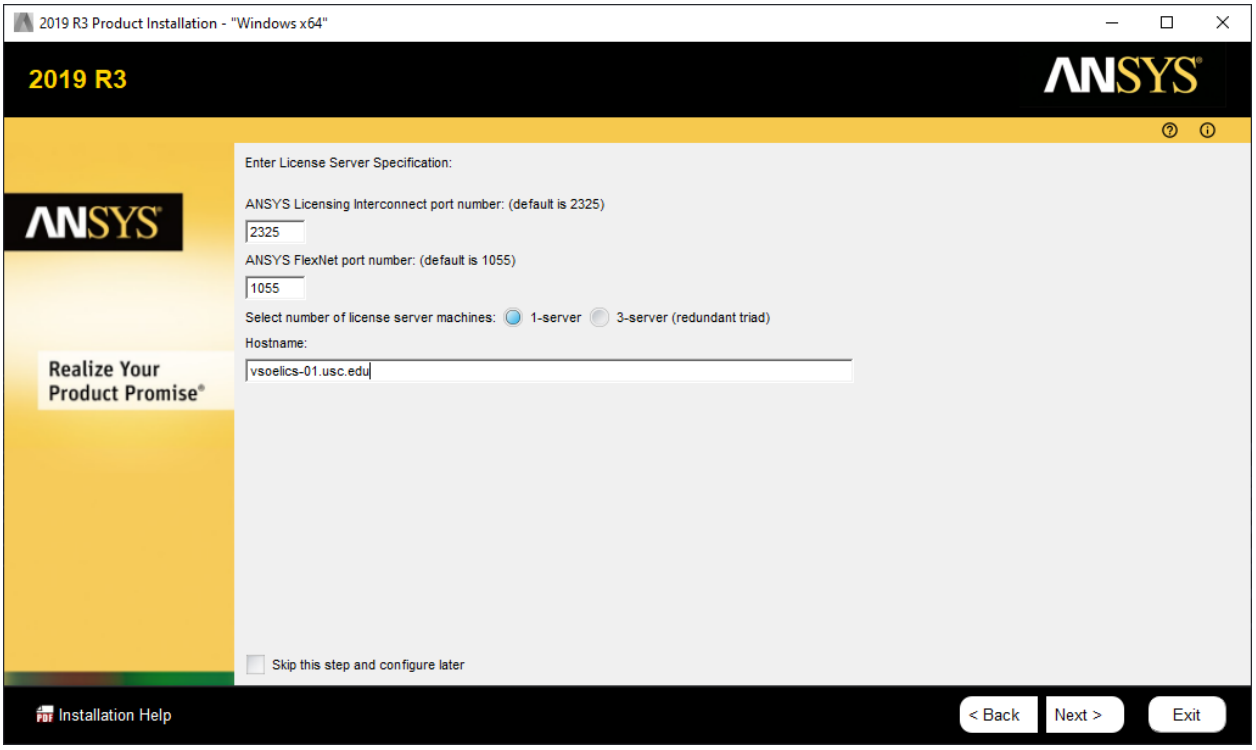


Figure 4. License Server Specification

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.

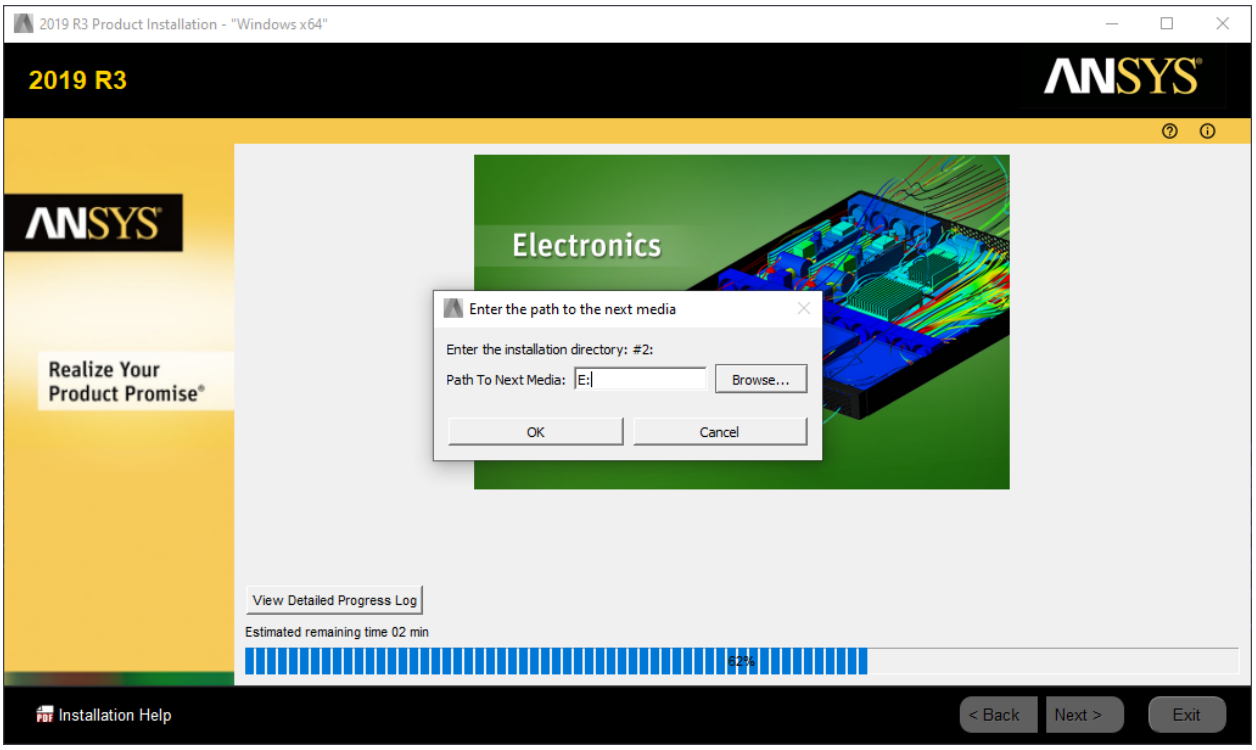


Figure 5. Installation directory #2

- **CHEMKIN PRO**

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

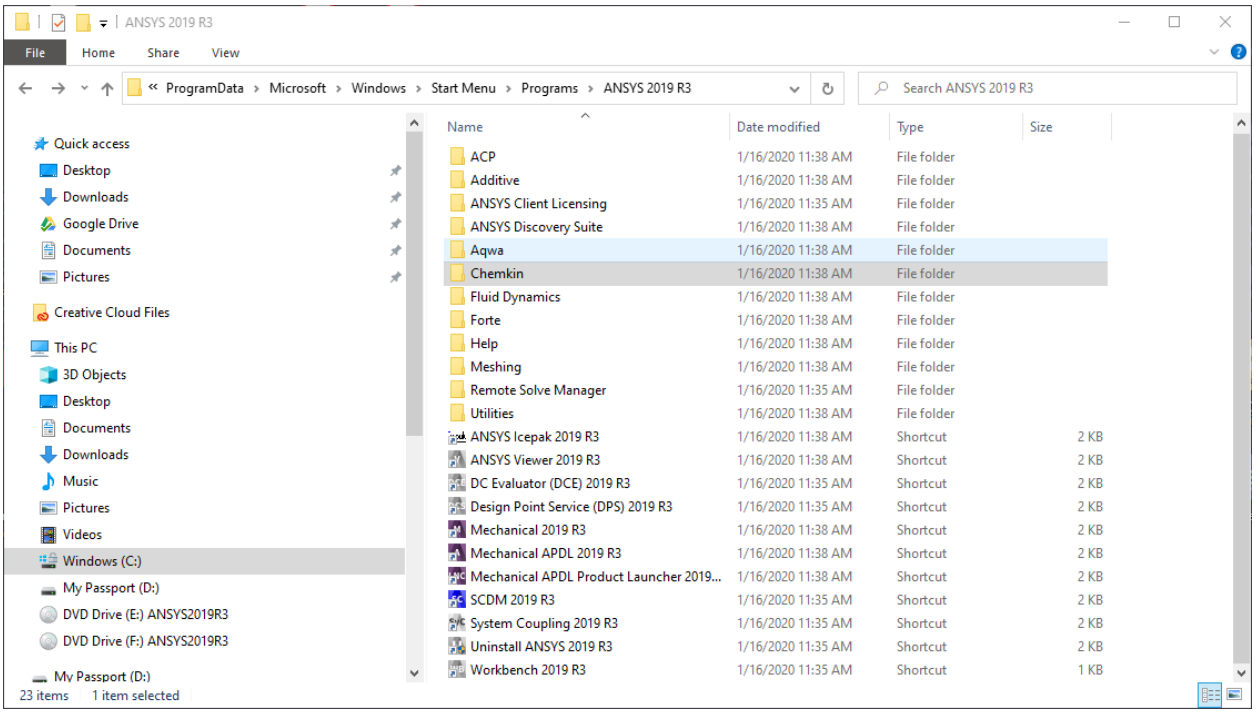


Figure 6. CHEMKIN PRO file location

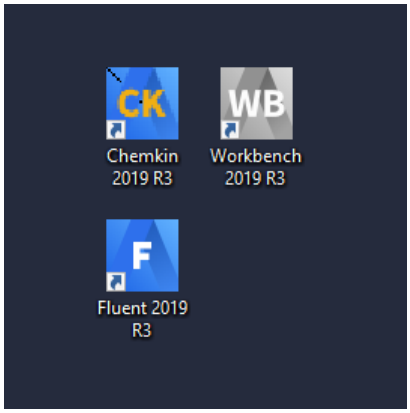


Figure 7. Create shortcuts for ANSYS

Figure 8 shows the basic window of Chemkin-Pro.

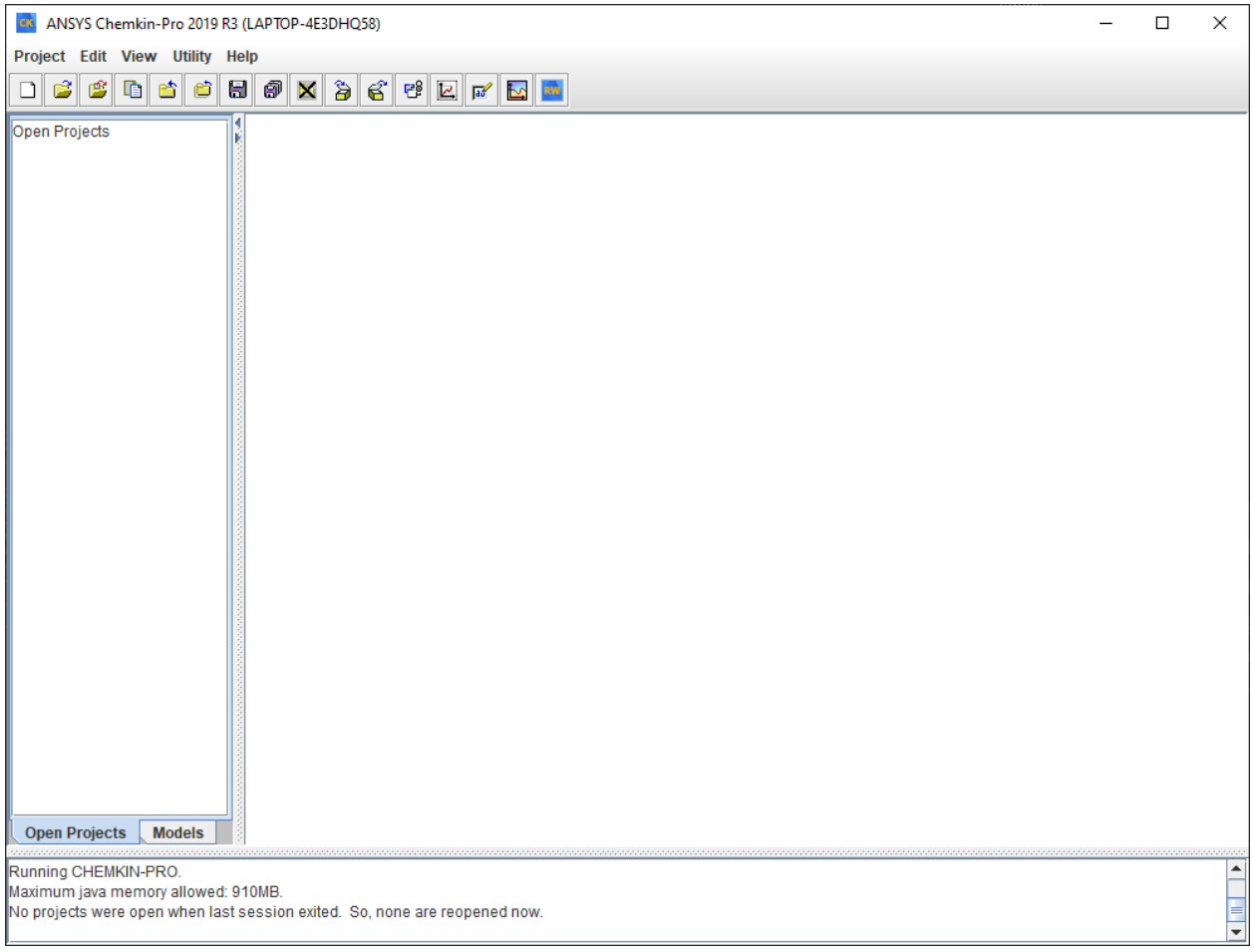


Figure 8. Chemkin-Pro window

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

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" → "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.

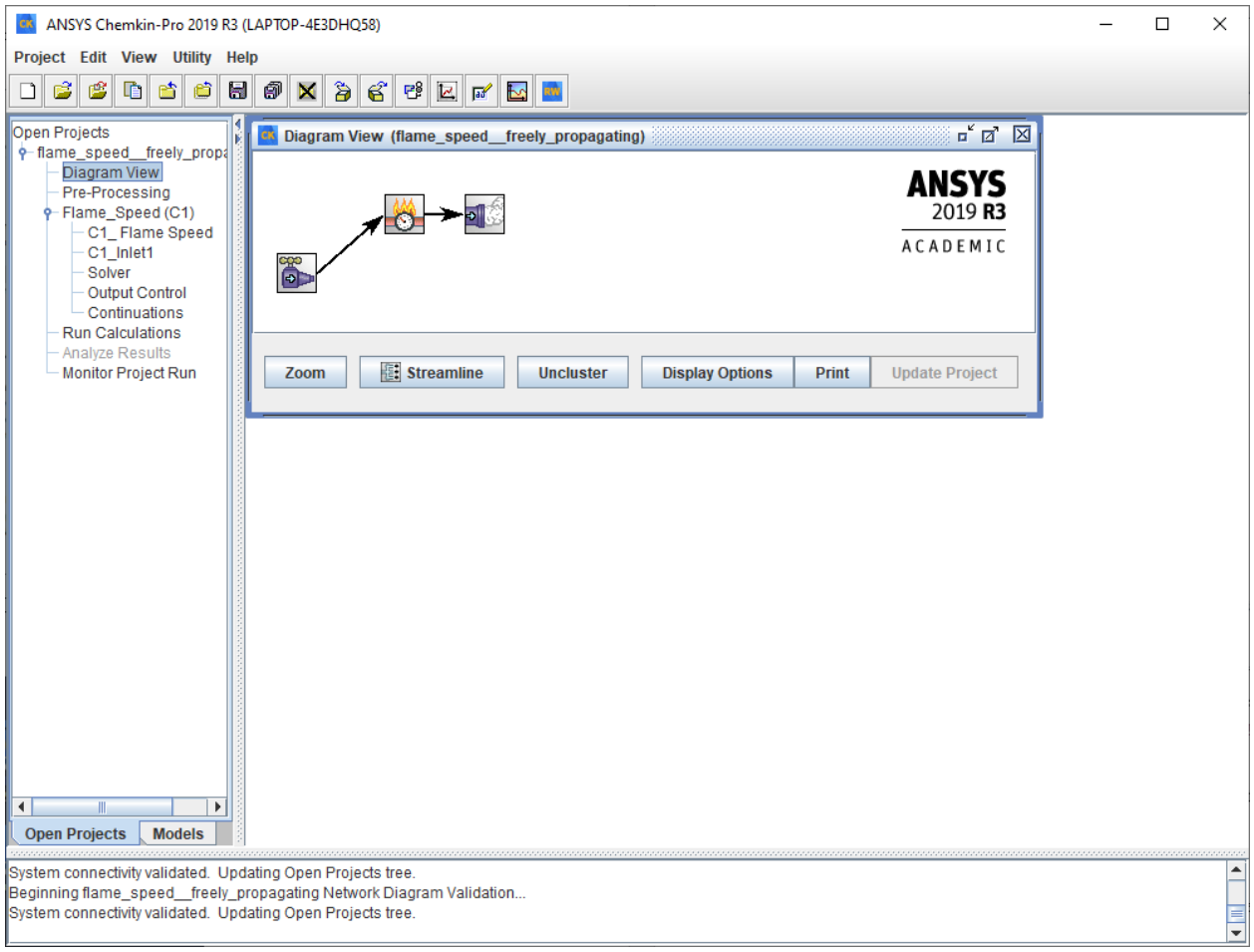


Figure 9. Diagram View

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.

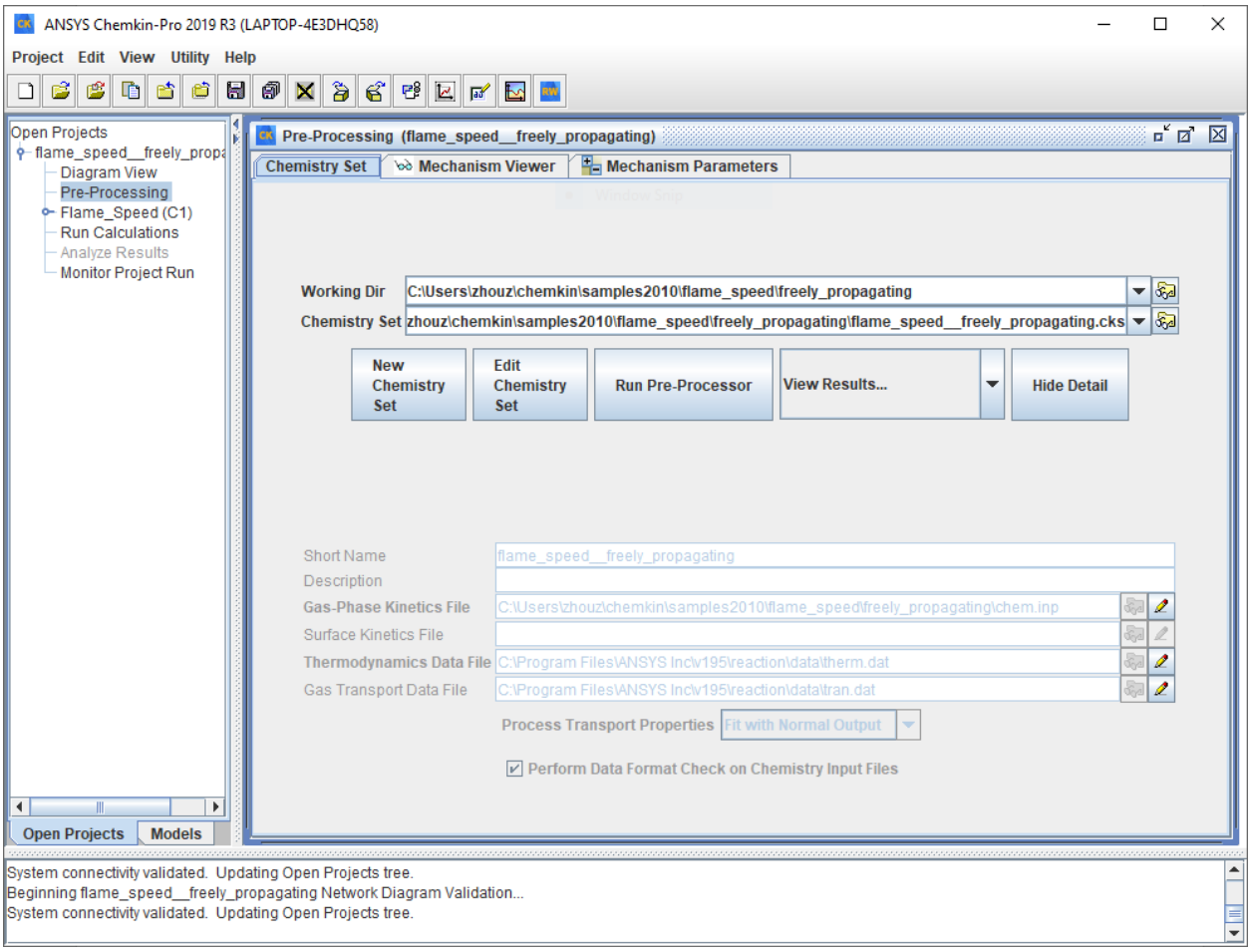


Figure 10. Pre-Processing window

After running Pre-Processing, double click on "Flame_Speed (C1)" → "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.

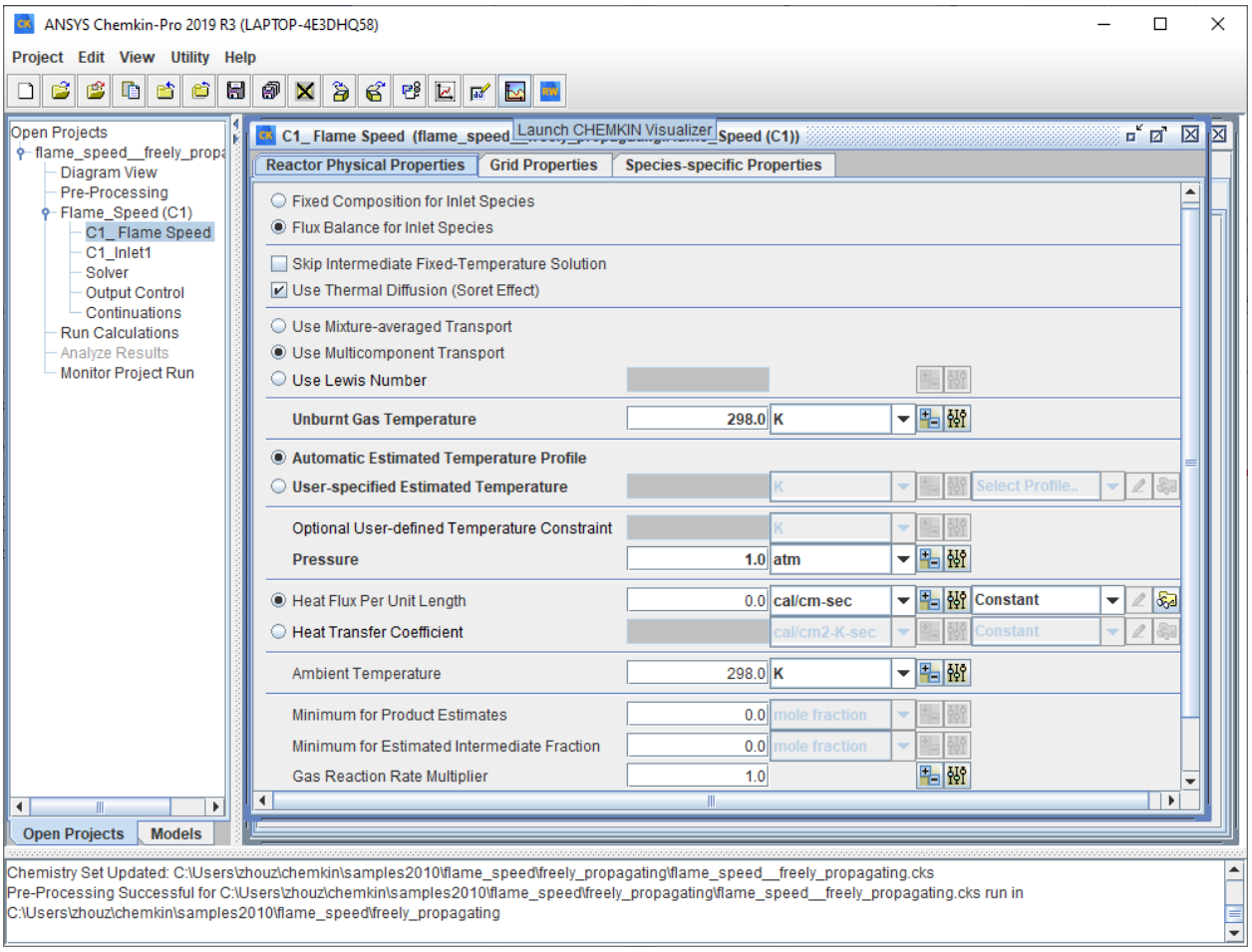


Figure 11. Reactor Physical Properties

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.

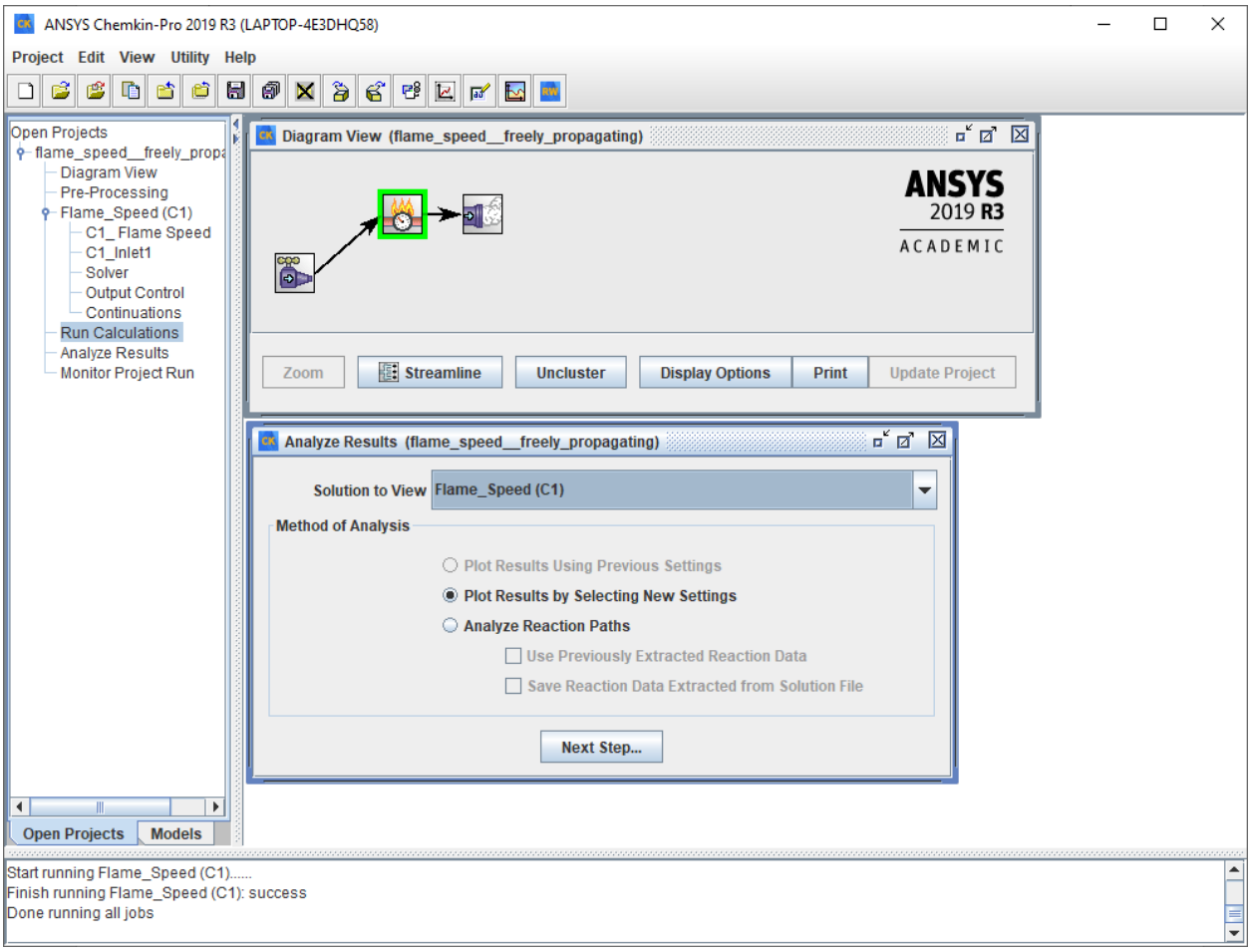


Figure 12. Analyze Results window

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.

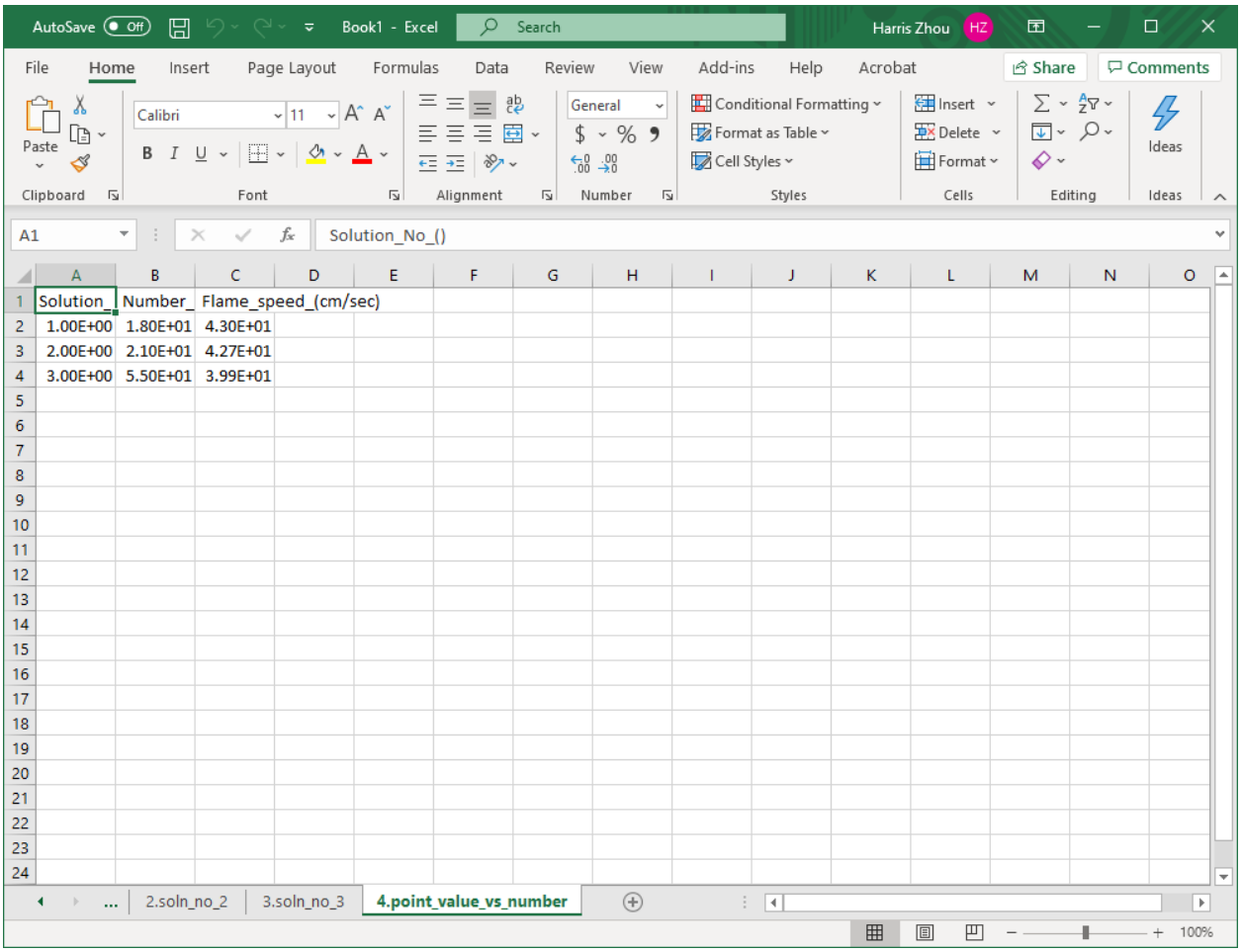


Figure 13. Simulation results in Excel

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.

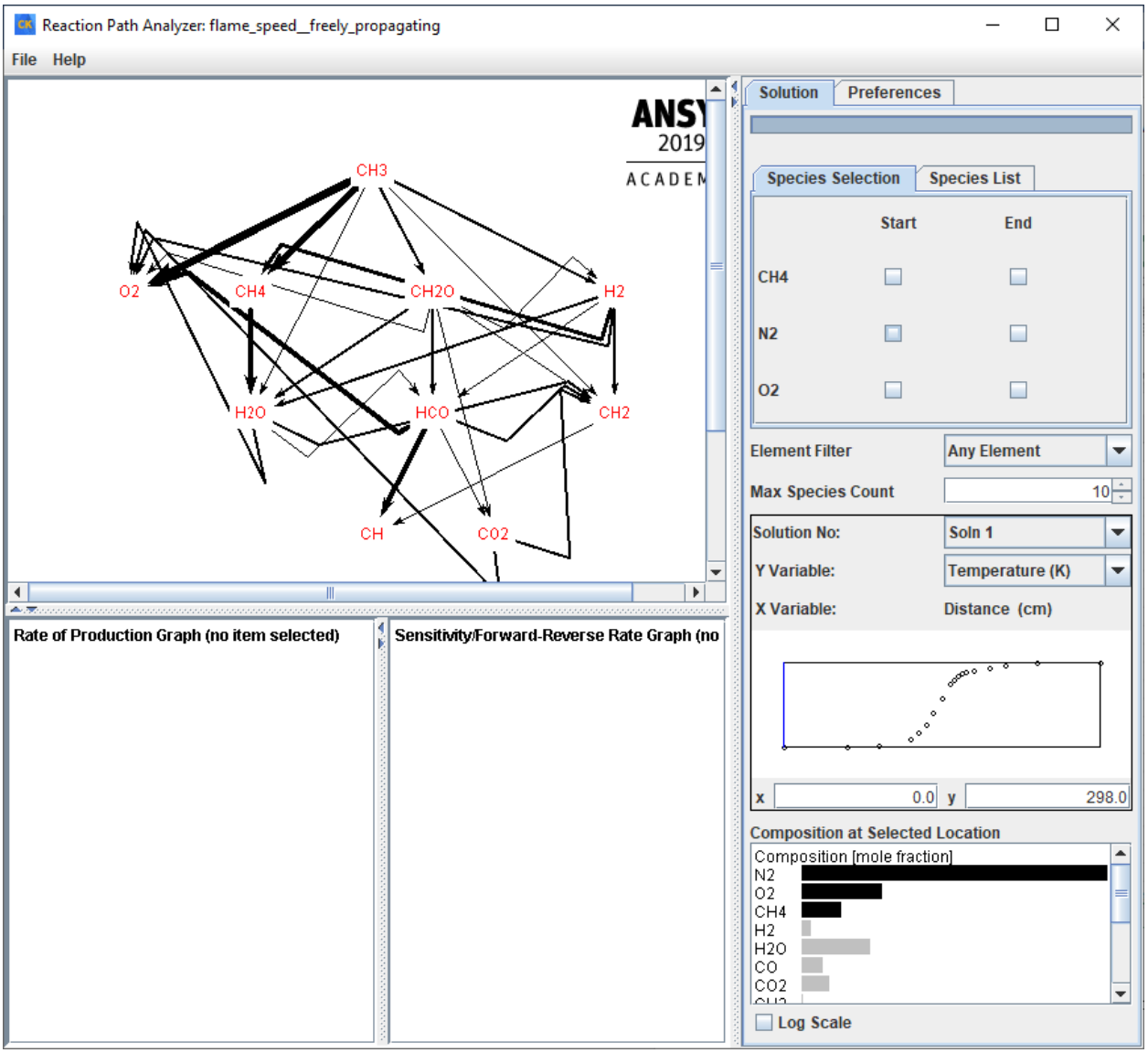


Figure 14. Reaction Path Analyzer

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.

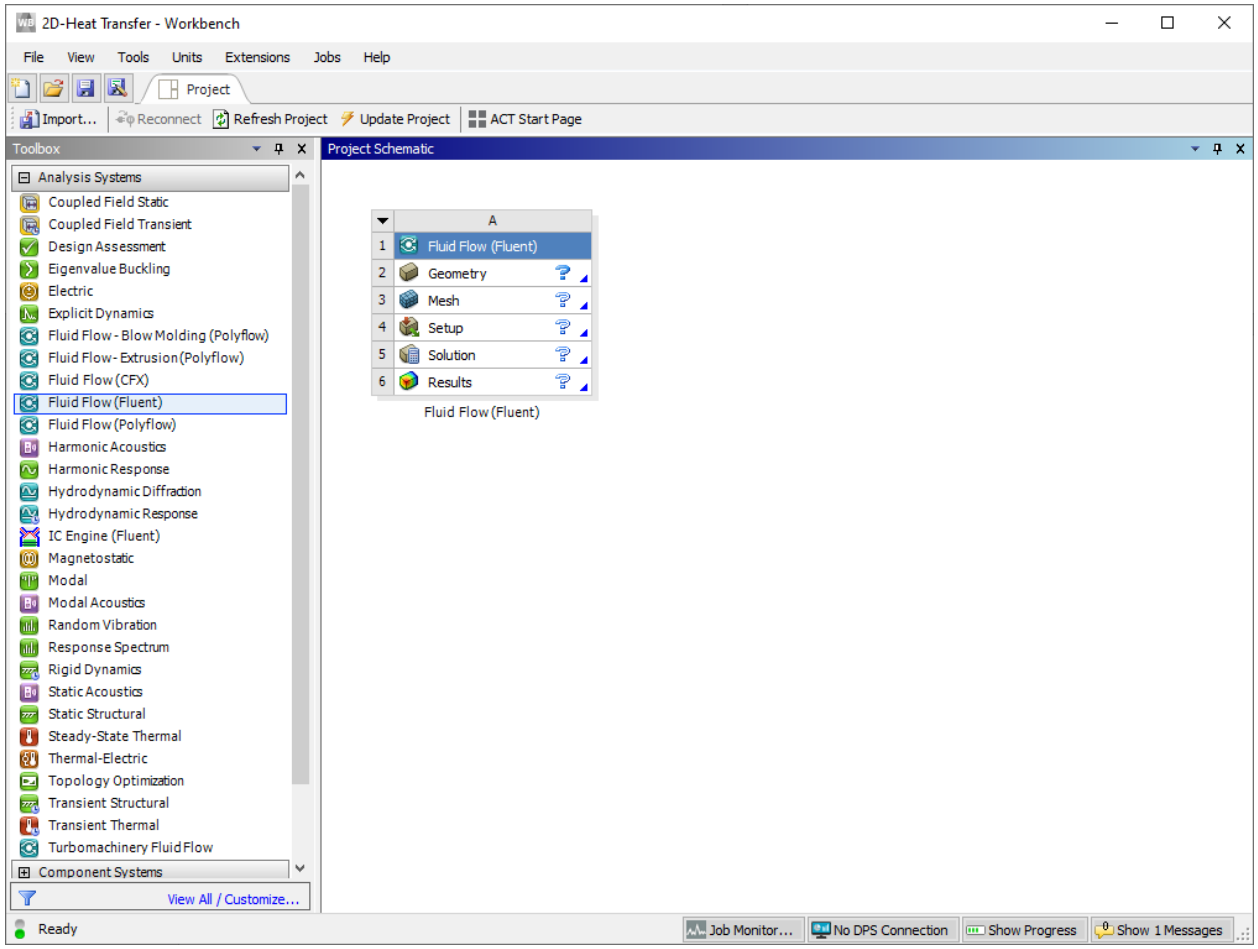


Figure15. Project Schematic window

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.

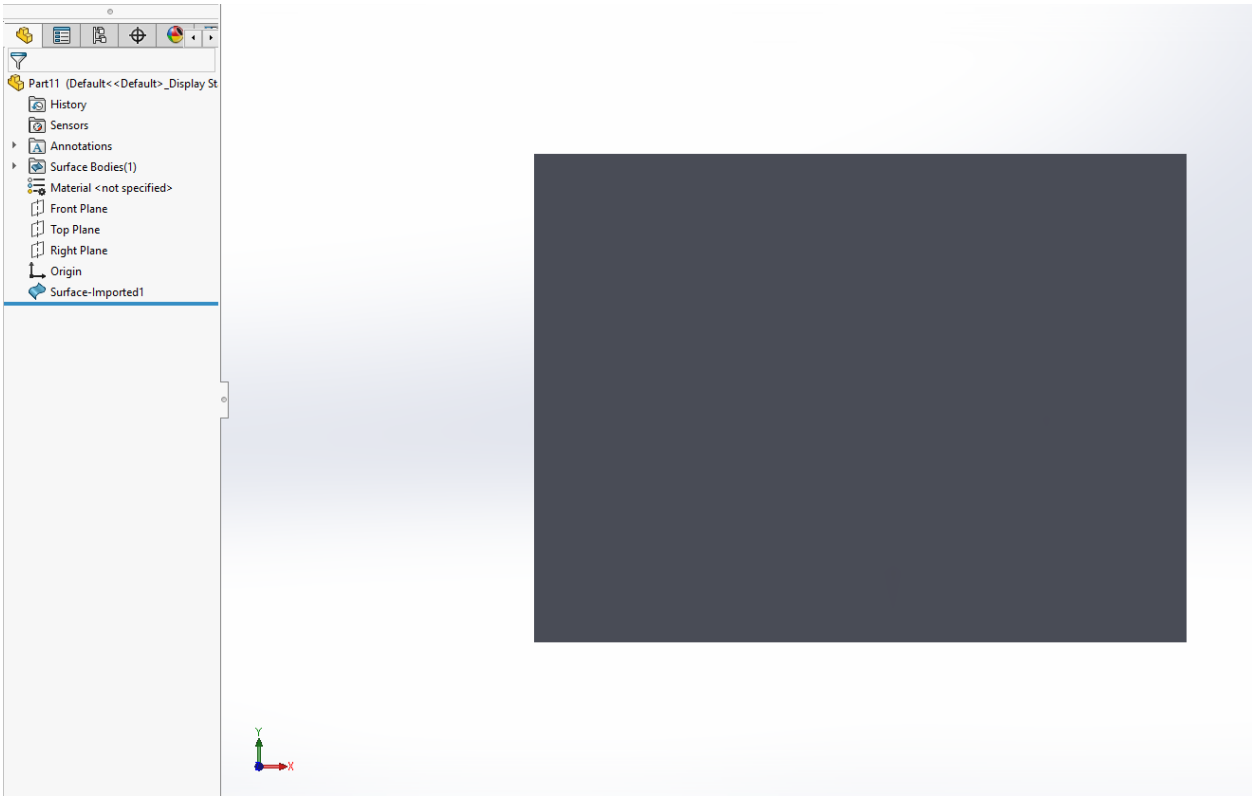


Figure 16. 2D-surface

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" → select the surface → click on "no selection" next to "Geometry" and then apply. "Ctrl+E" → select the four sides → 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.

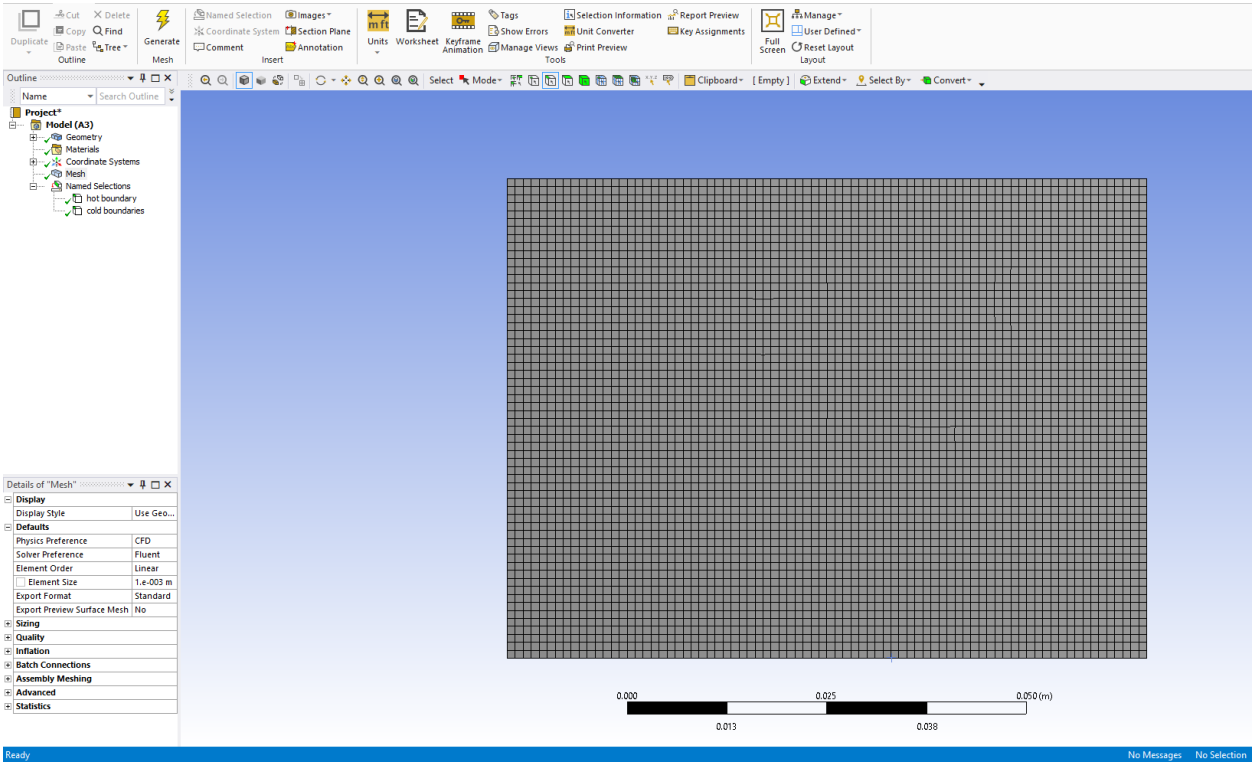


Figure 18. Generate Mesh

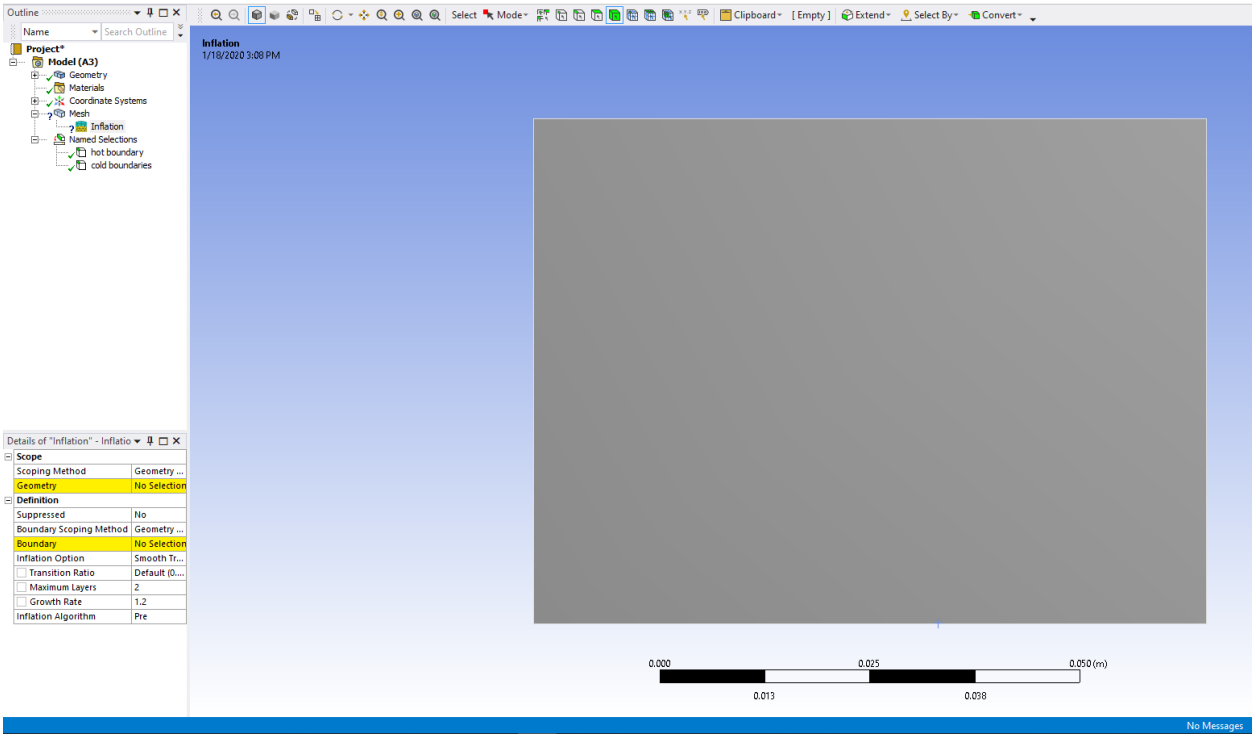


Figure 19. Inflation

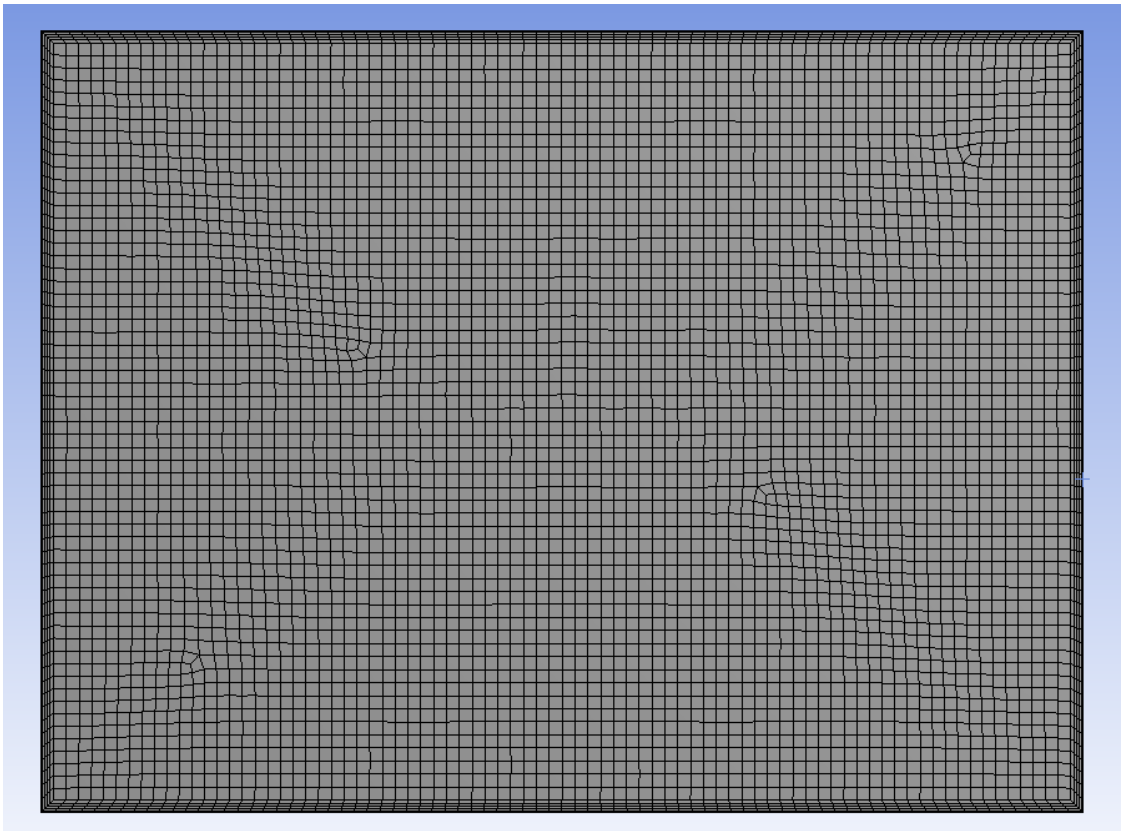


Figure 20. Mesh with inflation

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).

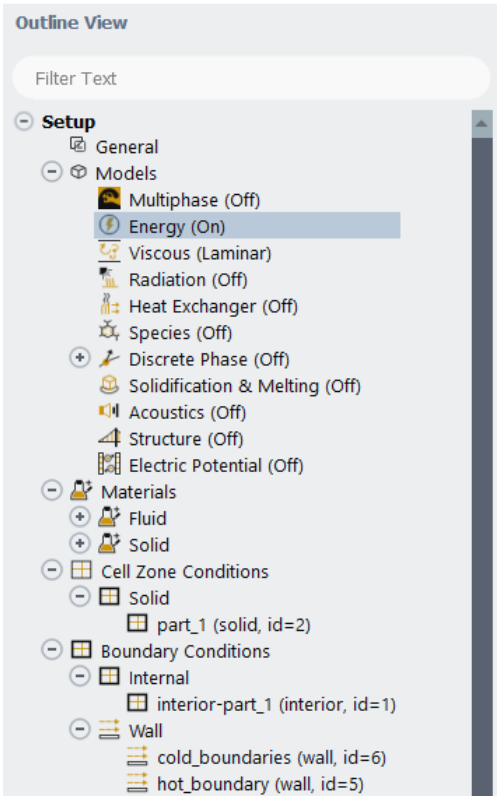


Figure 21. Outline View

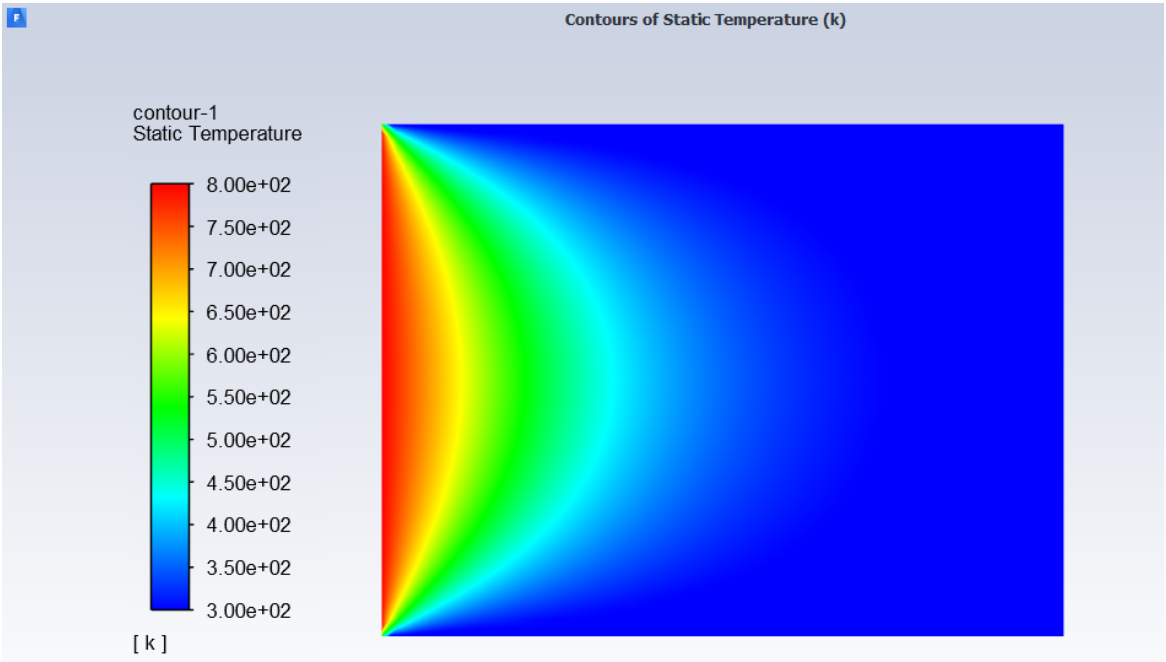


Figure 22. Temperature Contour

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.

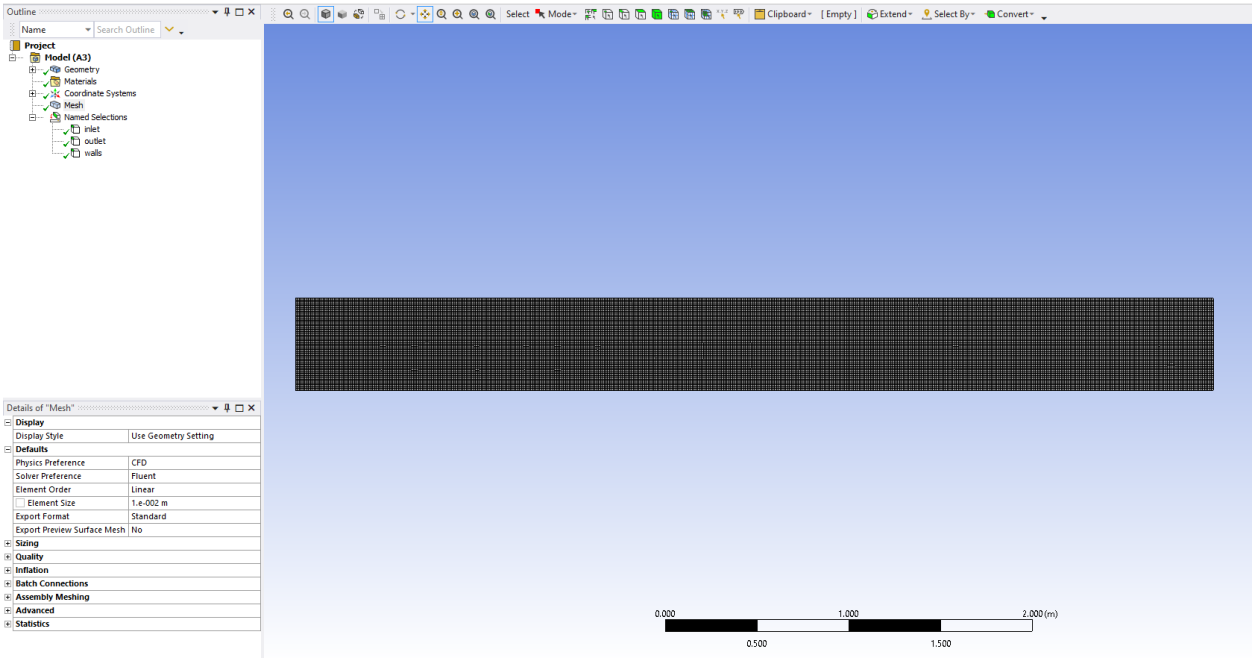


Figure 23. Mesh for 2D-Combustion

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.

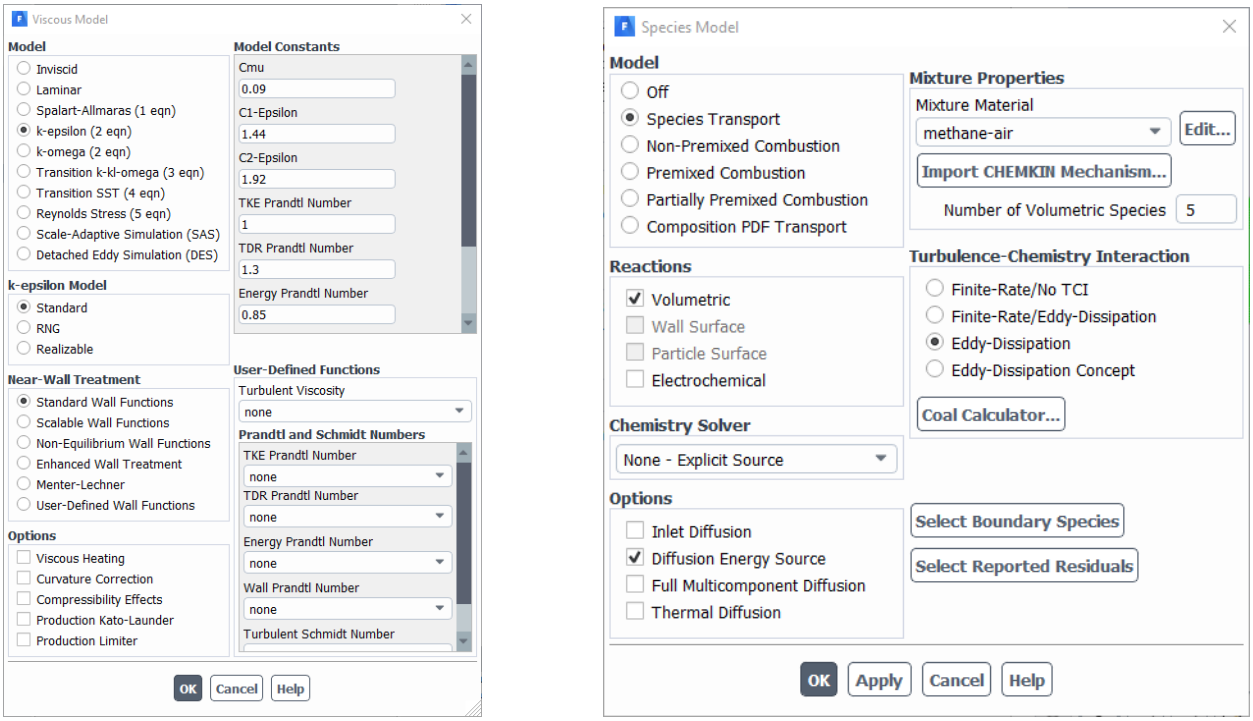


Figure 24. Setups in "Models"

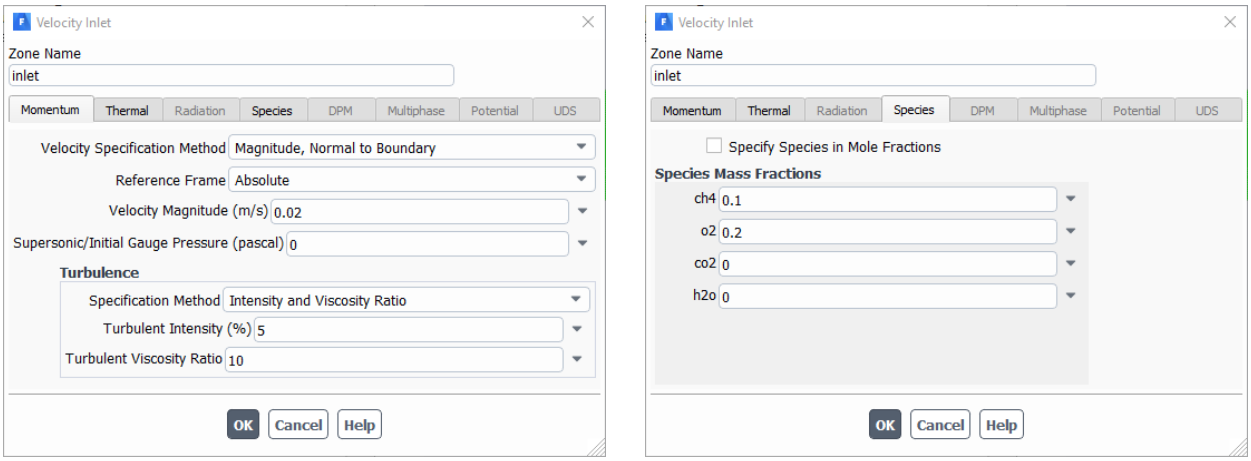


Figure 25. BCs at the inlet

Next, go to "Solutions" → "Monitors" → "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.

Set the number of iterations to 250 and start calculation.

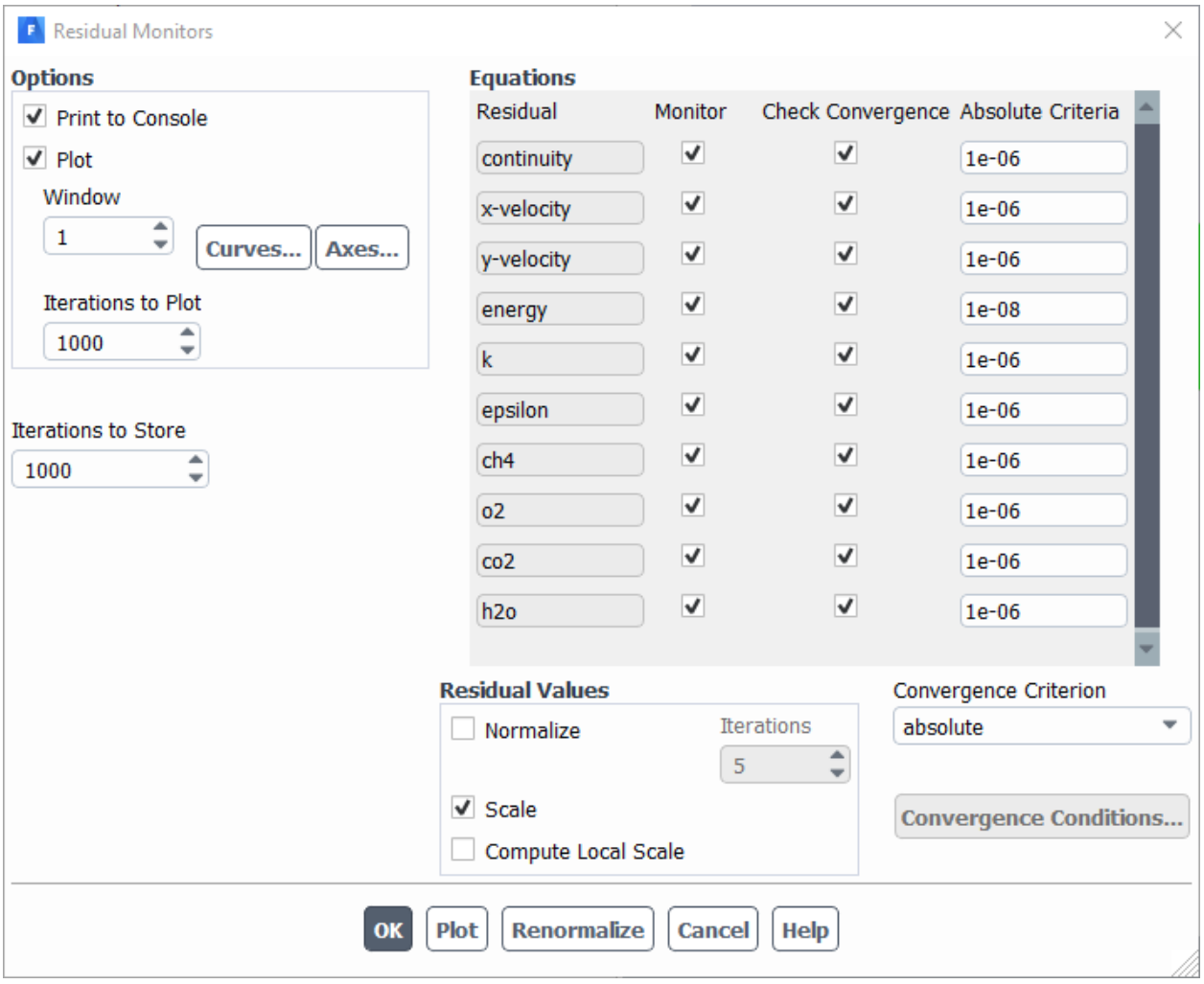


Figure 26. Residual Monitors

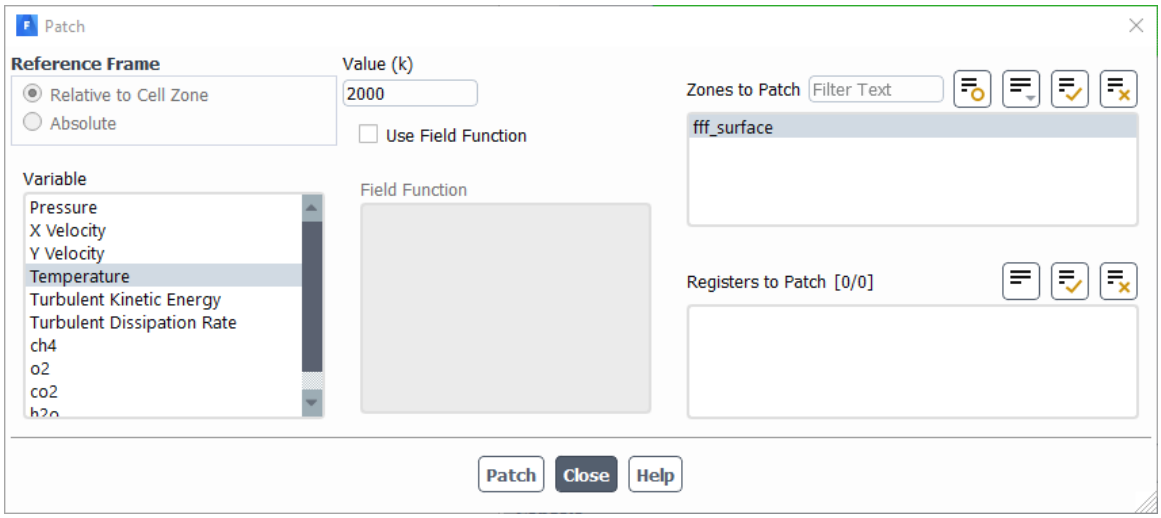


Figure 27. Patch

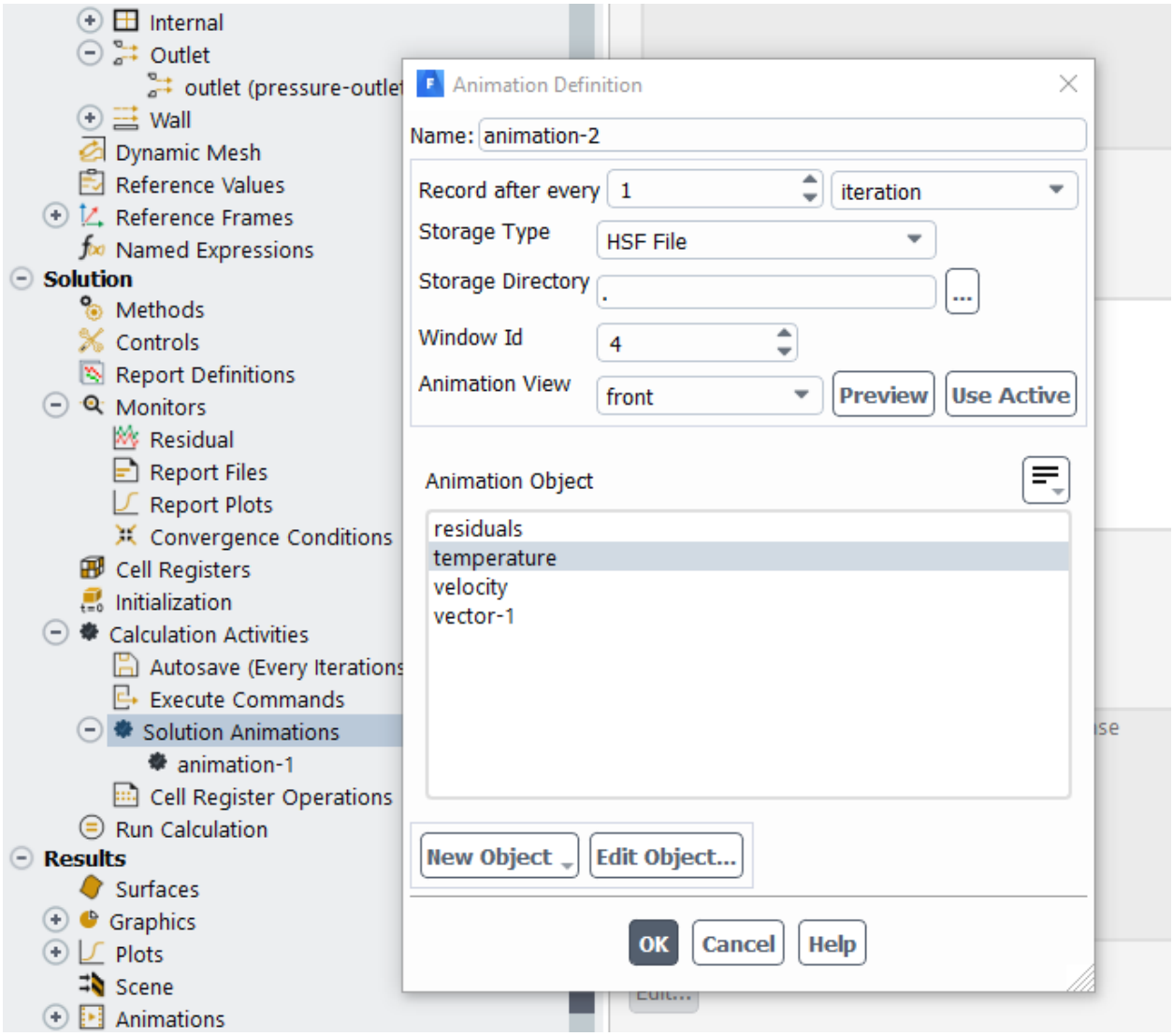


Figure 28. Solution Animations

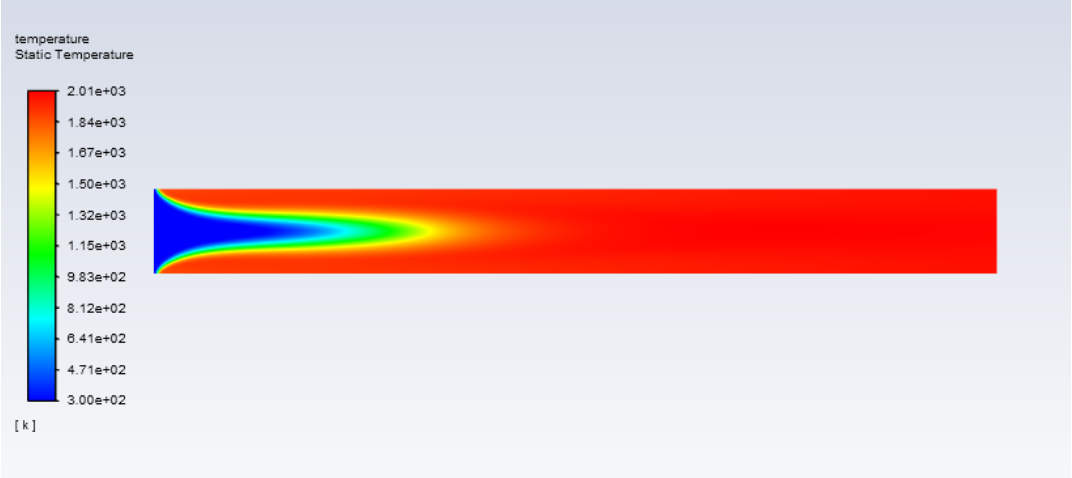


Figure 29. The temperature contour for 2D-Combustion

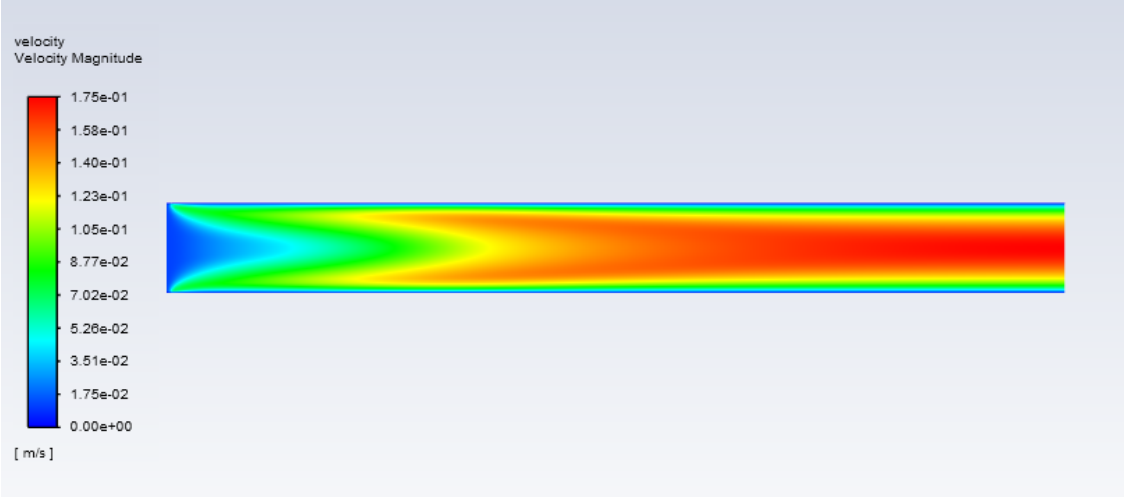


Figure 30. The velocity contour for 2D-Combustion

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.