Partially Premixed Combustion

In this tutorial, you will learn how to set up and solve a turbulent reactive flow—particularly, a partially premixed combustion case, in which there are both premixed and non-premixed conditions. You will:

- Use the probability density function (PDF) method to track the mixture fraction and modeling the chemistry in the system (used for non-premixed, mixing combustion cases).
- Learn the appropriate inputs and solver techniques using the turbulent Zimont Flame Speed model to close the turbulent quantities, typically used for premixed combustion cases.
- Analyze the results of the system.

The non-premixed combustion model solves transport equations for conserved scalars and mixture fractions. The amounts of chemical species present are derived from the predicted mixture fraction distribution, present in the precomputed PDF tables. These tables are generated by knowing the species that can be present, as well as the inflow conditions and properties of the mixture.

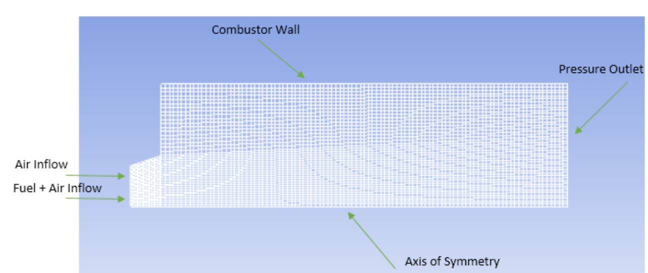
For the premixed combustion component which will be solved at simulation runtime, the Zimont turbulent flame speed model includes the laminar flame speed (which determines the chemistry of the system) and the flame front evolution due to turbulence. The assumption to use this model is that the turbulence lengthscale in the flame is smaller than the flame thickness (Karlovitz number Ka > 1 where $Ka = \frac{(\nu\epsilon)^{1/4}}{(u_{laminar\ flame\ speed})^2}$.

Combining these two models is straightforward. The reaction progress variable *c* is used to track the location of the flame, called the flame front. Before (to the left of) the flame front where $c = 0$, the mixture is unburnt, and the mass fractions and other variables are computed using the precomputed mixture fraction PDFs. Inside the flame, a combination of the two models is used. In the burnt area (to the right of the flame where $c = 1$), the equilibrium mixture fraction is computed.

This method is typically limited to combustion systems that only contain two inflow streams. Using swirl conditions on one of the streams is useful as it promotes mixing of the two streams; reducing problems with flame initialization and extinction. The turbulence model that will be used is the two-equation k–ε model.

Problem Description

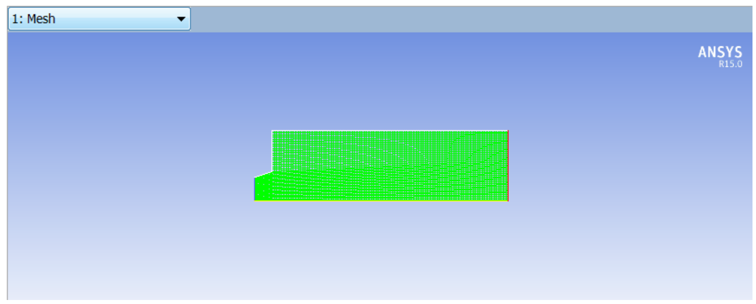
This is a partially premixed combustion case, which has inflows that reflect both premixed (fuel inflow) and no premixed (fuel and air mixing) conditions.



The fuel (CH_4) and air mixture has an equivalence ratio of 0.8, defined in the *Physics Setup*. It is injected at T = 300 K with an axial velocity of 50 m/s and swirl velocity (Θ direction) of 30 m/s. The air inflow is injected at T = 650 K and 10 m/s axially, with no swirl. This case is axisymmetric and so the physical combustion chamber is assumed to be cylindrical, rotated about the axis of symmetry. The outflow is the pressure outlet at atmospheric pressure.

Mesh

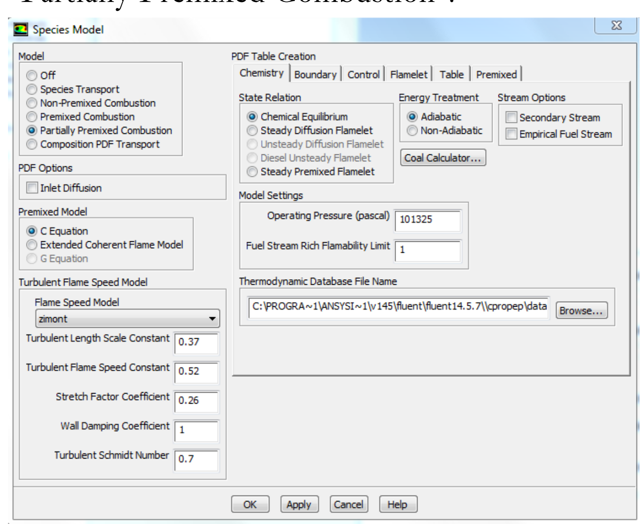
- Download the mesh file `par-premixed.msh` and extract the Fluent mesh file.
- Open Ansys Workbench and drag and drop a *Fluid Flow—Fluent* case into the main workbench area. Save the project.
- Double click on *Setup*, skipping the *Geometry* and *Mesh* options. In the pop-up dialogue, select 2D double precision.
- In *Fluent*, go to *File→Import→Mesh* and select the file that you just downloaded.
- Go to *Solution Setup→General* and click *Display* under mesh options to show the mesh.
- *Mesh→Info→Size* should show 4,700 cells in the domain.
- The mesh was originally created in inches. Click on *Scale* under *Mesh Options*; then select "in" in the drop down box called *Mesh Was Created In*, then click *Scale* and close the dialog box.



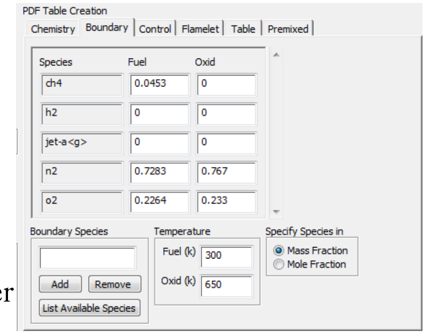
- To speed up future computations, select *Mesh→Reorder→Domain* in the menu at the top of the screen. (This increases memory access efficiency for neighboring elements.)

Physics Setup

- In the *General* tab, select "Axisymmetric Swirl" as the 2D space type. Keep the solver type as *Pressure* based to allow the premixed combustion model to be used.
- In the *Models* tab, double click on the *Viscous* option to change it from "Viscous—Laminar" to "k–ε (2 eqn.)".
- For *Species*, change the model from "Off" to "Partially Premixed Combustion".
- In the *Chemistry* tab, change the *Fuel Stream Rich Flammability Limit* to 1. Changing this value means that FLUENT will perform equilibrium calculations at all mixture fractions (1.0 being the largest possible fraction). If this was not the case, the composition would be computed based on mixing instead, for fractions above the specified limit.



- Click on the *Boundary* tab. Enter the values as shown in the following image.
  These values correspond to methane and air (nitrogen and oxygen) injected at a lean (< 1.0) equivalence ratio of $\Phi = 0.8$, representing the premixed fuel at $T = 300$ K. This ratio is defined as the ratio of the fuel to oxidizer ratio, to the fuel to oxidizer ratio at stoichiometric conditions. In addition, oxidizer (air) will be injected as a co-flow outside of the fuel stream, at 650 K. To add other species to the inflow boundary, you would select the species you want and click *Add* under the *Boundary Species* options in this tab.



- Ignore the *Control* and *Flamelet* tabs, and select the *Table* tab.
  - Click *Calculate PDF Table* to generate the pre-computed PDF table denoting mixture fractions that will handle the non-premixed combustion analysis.
  - Investigate PDF table parameters and dependencies by clicking on *Display PDF Table*, selecting desired options, and clicking *Display*.
  - Click Apply and OK to close the *Species* dialogue box.
- To write and save the PDF file that you just generated, go to *File→Export→PDF* at the menu at the top of the screen.

Materials

- In the *Materials* section, select the material *Mixture*, and click *Create/Edit*.
- Note how Fluent has pre-selected the material as a mixture, and computes the density based on the PDF. Leave these pre-selected options as they are.

Boundary Conditions

- In *Solution Setup→Boundary Conditions*, because this is a turbulent model, all turbulence parameters must be specified on the boundaries of the combustor.
- Click to edit the zone "air" (a velocity inlet). Change the *Velocity Specification Method* to "Components", and set the *Axial Velocity* to 10 m/s.
- Change the *Specification Method* to "Intensity and Hydraulic diameter", and adjust the turbulence parameters as in the dialogue.
- Similarly, edit the air–fuel boundary to have velocity components, with 50 m/s axial velocity and 30 m/s swirl velocity.
- The turbulent intensity should be 10%, but the hydraulic diameter should be 0.0254 m.
- On the *Species* tab, change the mean mixture fraction to 1.0. This is because this stream will be taken as the fuel stream, whereas the previous air boundary did not need to be set beyond the default value of 0 for the oxidizer stream.
- Edit the outlet boundary condition, which is a pressure outlet.
- Use 10% for *Backflow Turbulent Intensity*, and 0.13 m for the *Backflow Hydraulic Diameter*.

- Clicking to the *Species* tab, change the *Backflow Progress Variable* to 1. This means that the boundary condition where the progress variable is 1.0 will only be applied in the case that there is backflow from this outlet.

  Numerical Solution
- Skip to *Solution Controls*. Click *Equations* and deselect the *Premixed Combustion* and *PDF* options so that the only equations solved for are flow, swirl velocity, and turbulence as a preliminary solution.
- Go to *Solution Initialization*. Press *Initialize*; let Fluent initialize the domain as a *Hybrid Initialization*, taking all boundaries into account.
- Go to *Run Calculation*. Click on *Data File Quantities* and select all of the mass fractions of species, the turbulent flame speed, and the stream function (or click the button at the top right of the selection screen to select all if you would like to monitor other variables).
  - Enter 1000 under the *Number of Iterations* and run the simulation until it converges.
- Now adapt a small region near the inlet with a progress variable of zero (this region will be entirely recalculated with combustion included):
  - Go to Adapt-Region at the menus at the top of the page.
  - Select a small region near the inlet: x from 0.1–0,14 m, y from 0.00–0.03 m$.
  - Click *Mark*, which should mark 207 cells for refinement, and then click Close.
- Patch this region by going to *Solution Initialization*, and click *Patch*.
  - Select *Progress Variable* as the variable, and patch the region that you just marked by clicking *Patch*.
- Now go back to *Solution Controls* and click *Equations*.
  - Reselect the *PDF* and *Premixed Combustion* options so that all options are highlighted and will be solved for.
- In *Solution Methods*, make sure that all solvers (Momentum, Swirl Velocity, and Turbulent Kinetic Energy) are set to a second-order solver scheme.
- Go to *Run Calculation*. Press *Calculate*.
- Allow the solution to converge (residuals all $\leq 1\times10^{-3}$).

  Post-processing
- Save the project.
- Create contours of the following. For all plots, use 51 contours.
  - Stream Function
  - Progress Variable (called Reaction Progress)
  - Temperature
  - Mass Fraction of CH4
  - Turbulent Flame Speed