

Partially Premixed Combustion - Physics Setup

Author: Lara Backer, Cornell University - taken from ANSYS Inc. tutorial

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

Physics Setup

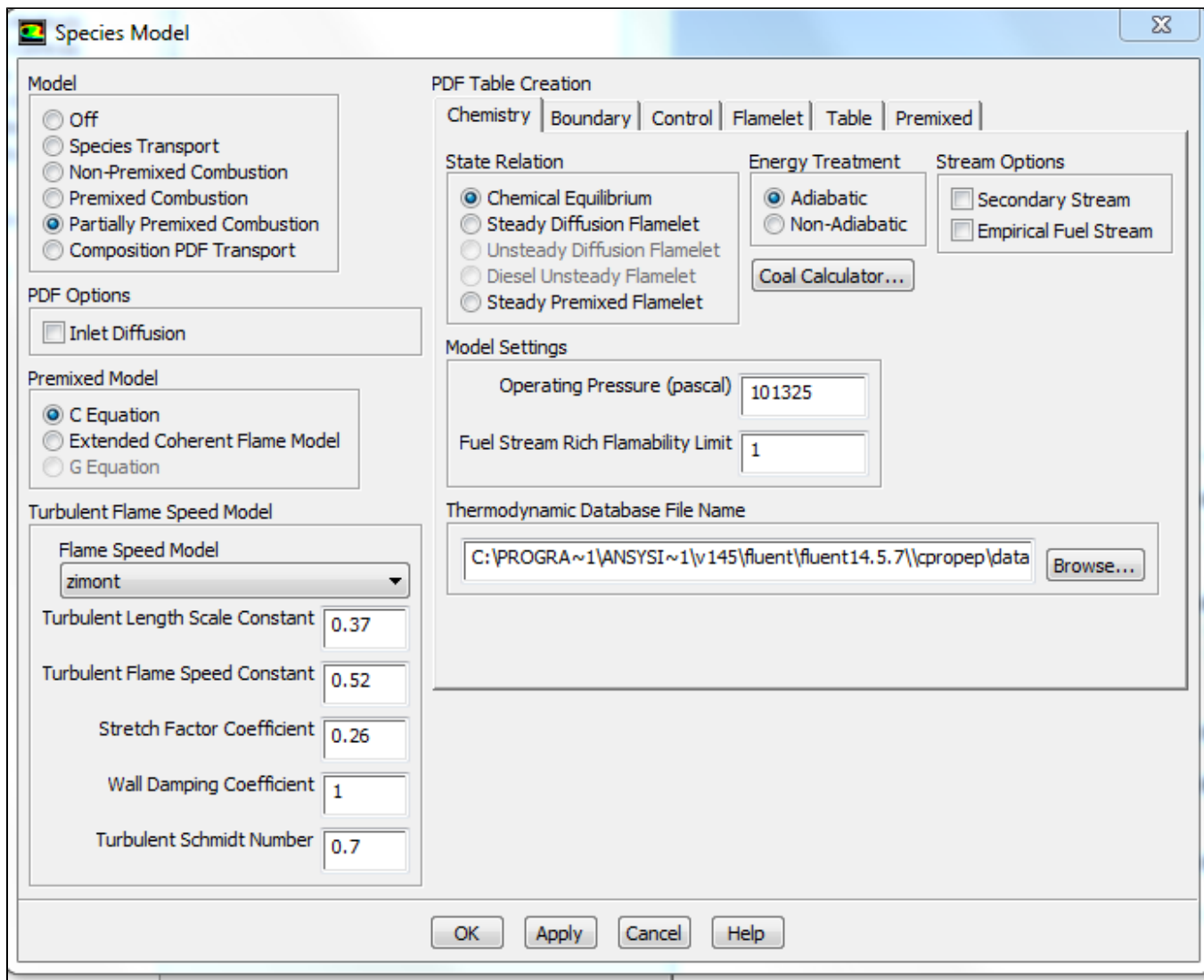
General

In the General tab, select "Axisymmetric Swirl" in the 2D space type. Keep the solver type as Pressure based to allow the premixed combustion model to be used.

Models

In the Models tab, double click on the Viscous option to change it from Viscous-Laminar to "k- (2 eqn)". Do not change any other parameters in the viscous dialogue box. Click "Ok" and close the dialogue box.

Now double click on the Species option to edit it, and 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 CH₄ (methane) and air (nitrogen and oxygen) injected at a lean (less than 1.0) equivalence ratio of $\phi = 0.8$, representing the premixed fuel at T=300K. 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 650K. To add other species to the boundary, you would select the species you want and click "Add" under the Boundary Species options in this tab.

PDF Table Creation

Chemistry | **Boundary** | Control | Flamelet | Table | Premixed

Species	Fuel	Oxid
ch4	0.0453	0
h2	0	0
jet-a <g>	0	0
n2	0.7283	0.767
o2	0.2264	0.233

Boundary Species

Temperature

Fuel (k) 300

Oxid (k) 650

Specify Species in

☒ Mass Fraction

☐ Mole Fraction

Add Remove

List Available Species

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".

Back in the Premixed tab, click to edit the Laminar Flame Speed options, and note that various laminar flame speeds were calculated corresponding to various non-premixed mixture fractions. Click Cancel on these editing options to close the box, and click "Apply" and "OK" to close the Species dialogue box.

To write and save the PDF file that you just generated, go to **File-Write-PDF** at the menu at the top of the screen.

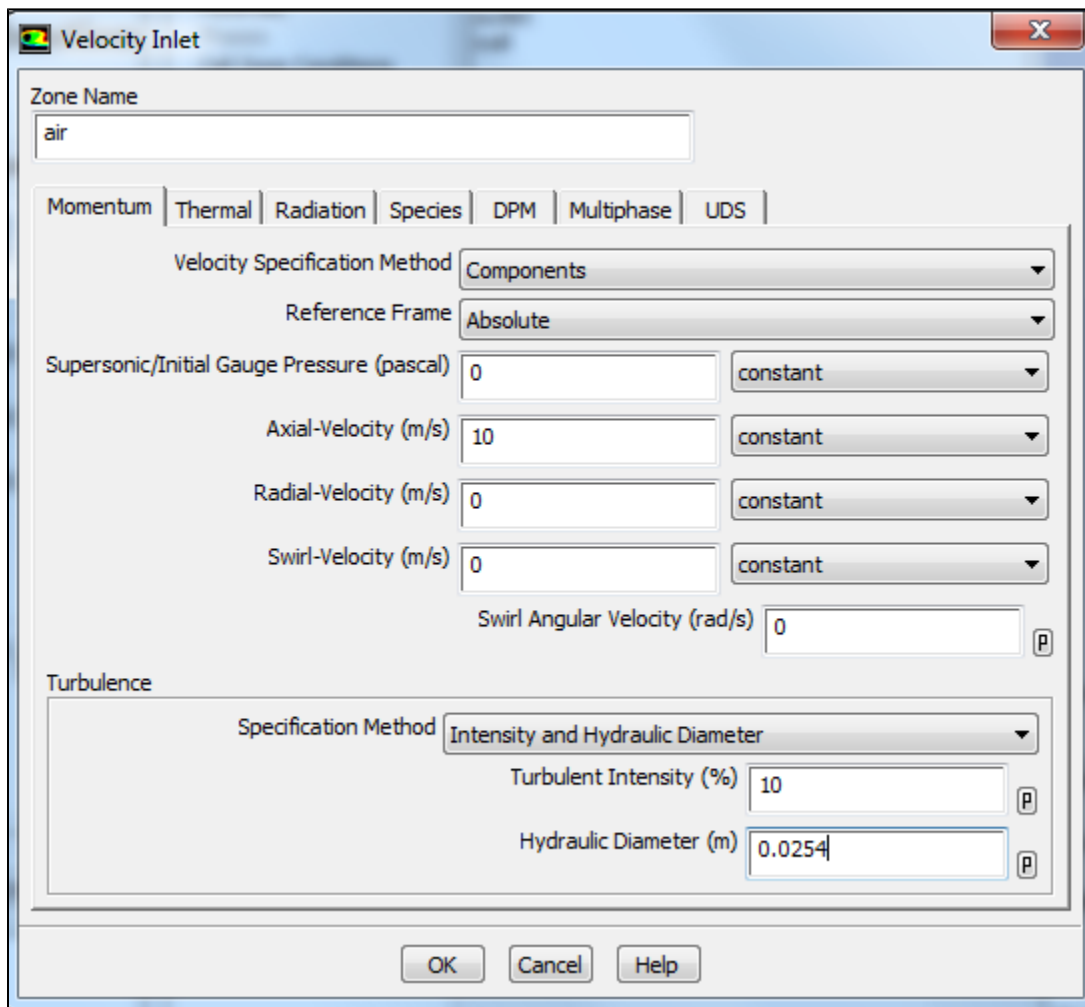
Materials

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. Click "Close".

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 at 10m/s. Change the Specification method to Intensity and Hydraulic diameter, and adjust the turbulence parameters as in the dialogue. Press OK once all the changes have been made.



The image shows a 'Velocity Inlet' dialog box with a title bar containing a small icon and a close button. The 'Zone Name' field is set to 'air'. Below this are tabs for 'Momentum', 'Thermal', 'Radiation', 'Species', 'DPM', 'Multiphase', and 'UDS'. The 'Momentum' tab is active, showing a 'Velocity Specification Method' dropdown set to 'Components' and a 'Reference Frame' dropdown set to 'Absolute'. There are five rows of input fields for velocity components, each with a text box and a dropdown menu set to 'constant': 'Supersonic/Initial Gauge Pressure (pascal)' with value 0, 'Axial-Velocity (m/s)' with value 10, 'Radial-Velocity (m/s)' with value 0, 'Swirl-Velocity (m/s)' with value 0, and 'Swirl Angular Velocity (rad/s)' with value 0. Below these is a 'Turbulence' section with a 'Specification Method' dropdown set to 'Intensity and Hydraulic Diameter'. It contains two input fields: 'Turbulent Intensity (%)' with value 10 and 'Hydraulic Diameter (m)' with value 0.0254. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Zone Name
air

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Velocity Specification Method: Components

Reference Frame: Absolute

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Axial-Velocity (m/s): 10 constant

Radial-Velocity (m/s): 0 constant

Swirl-Velocity (m/s): 0 constant

Swirl Angular Velocity (rad/s): 0 P

Turbulence

Specification Method: Intensity and Hydraulic Diameter

Turbulent Intensity (%): 10 P

Hydraulic Diameter (m): 0.0254 P

OK Cancel Help

Similarly, edit the air-fuel boundary to have velocity components, with 50m/s axial velocity and 30m/s swirl velocity. The turbulent intensity should be 10%, but the hydraulic diameter should again be 0.0254m. Clicking to 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. Click "Ok".

 Velocity Inlet

Zone Name

air-fuel

Momentum

Thermal

Radiation

Species

DPM

Multiphase

UDS

Velocity Specification Method

Components

Reference Frame

Absolute

Supersonic/Initial Gauge Pressure (pascal)

0

constant

Axial-Velocity (m/s)

50

constant

Radial-Velocity (m/s)

0

constant

Swirl-Velocity (m/s)

30

constant

Swirl Angular Velocity (rad/s)

0

P

Turbulence

Specification Method

Intensity and Hydraulic Diameter

Turbulent Intensity (%)

10

P

Hydraulic Diameter (m)

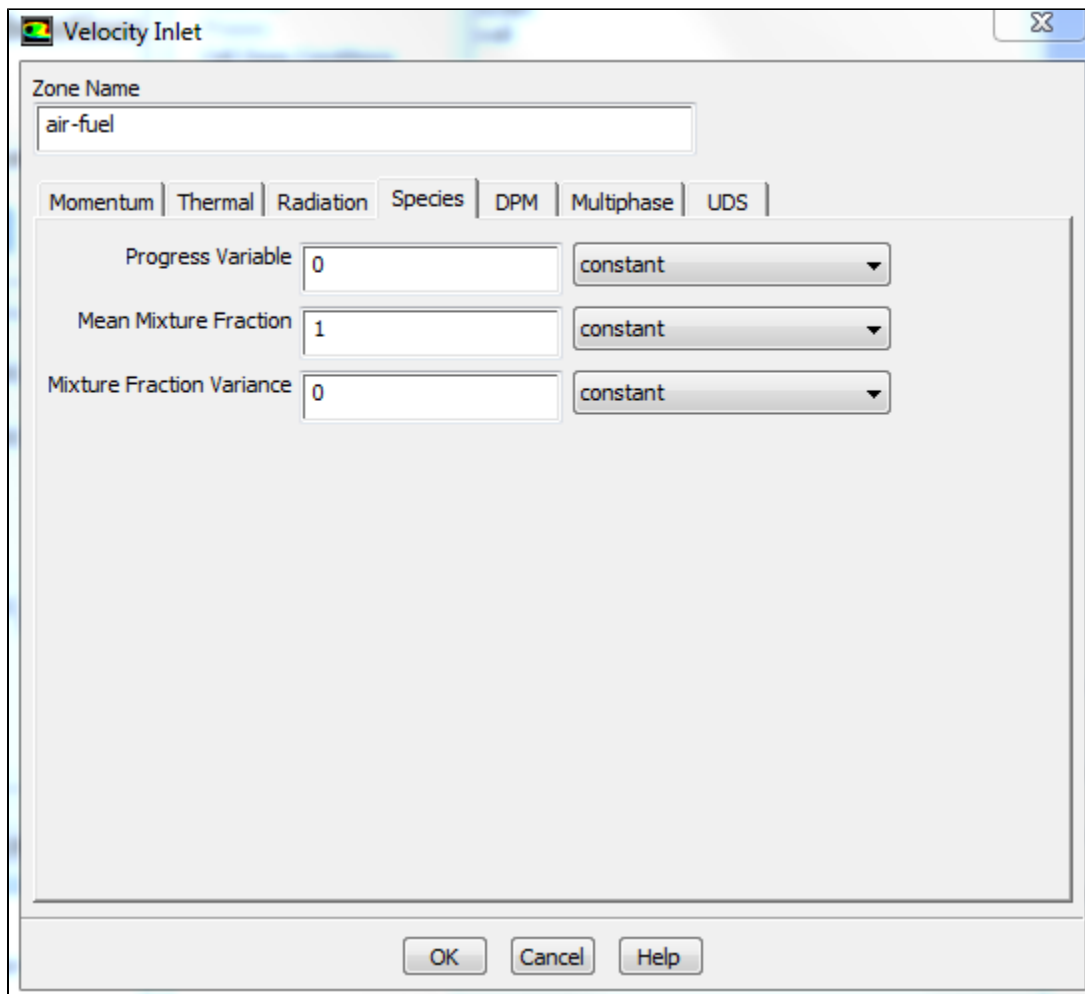
0.13

P

OK

Cancel

Help



The image shows a 'Velocity Inlet' dialog box with a close button in the top right corner. The 'Zone Name' field contains 'air-fuel'. Below this are tabs for 'Momentum', 'Thermal', 'Radiation', 'Species', 'DPM', 'Multiphase', and 'UDS'. The 'Species' tab is selected, showing three rows of settings: 'Progress Variable' with a value of 0 and a 'constant' dropdown; 'Mean Mixture Fraction' with a value of 1 and a 'constant' dropdown; and 'Mixture Fraction Variance' with a value of 0 and a 'constant' dropdown. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Property	Value	Dropdown
Progress Variable	0	constant
Mean Mixture Fraction	1	constant
Mixture Fraction Variance	0	constant

Edit the outlet boundary condition, which is a pressure outlet. Use 10% for Backflow Turbulent Intensity, and 0.13m 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.

 Velocity Inlet

Zone Name

air-fuel

Momentum

Thermal

Radiation

Species

DPM

Multiphase

UDS

Velocity Specification Method

Components

Reference Frame

Absolute

Supersonic/Initial Gauge Pressure (pascal)

0

constant

Axial-Velocity (m/s)

50

constant

Radial-Velocity (m/s)

0

constant

Swirl-Velocity (m/s)

30

constant

Angular Velocity (rad/s)

0

P

Turbulence

Specification Method

Intensity and Hydraulic Diameter

Turbulent Intensity (%)

10

P

Hydraulic Diameter (m)

0.0254

P

OK

Cancel

Help

 Pressure Outlet

Zone Name

outlet

Momentum

Thermal

Radiation

Species

DPM

Multiphase

UDS

Mean Mixture Fraction

0

constant

Mixture Fraction Variance

0

constant

Backflow Progress Variable

1

constant

OK

Cancel

Help

[Go to Step 5: Numerical Solution](#)

[Go to all FLUENT Learning Modules](#)