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

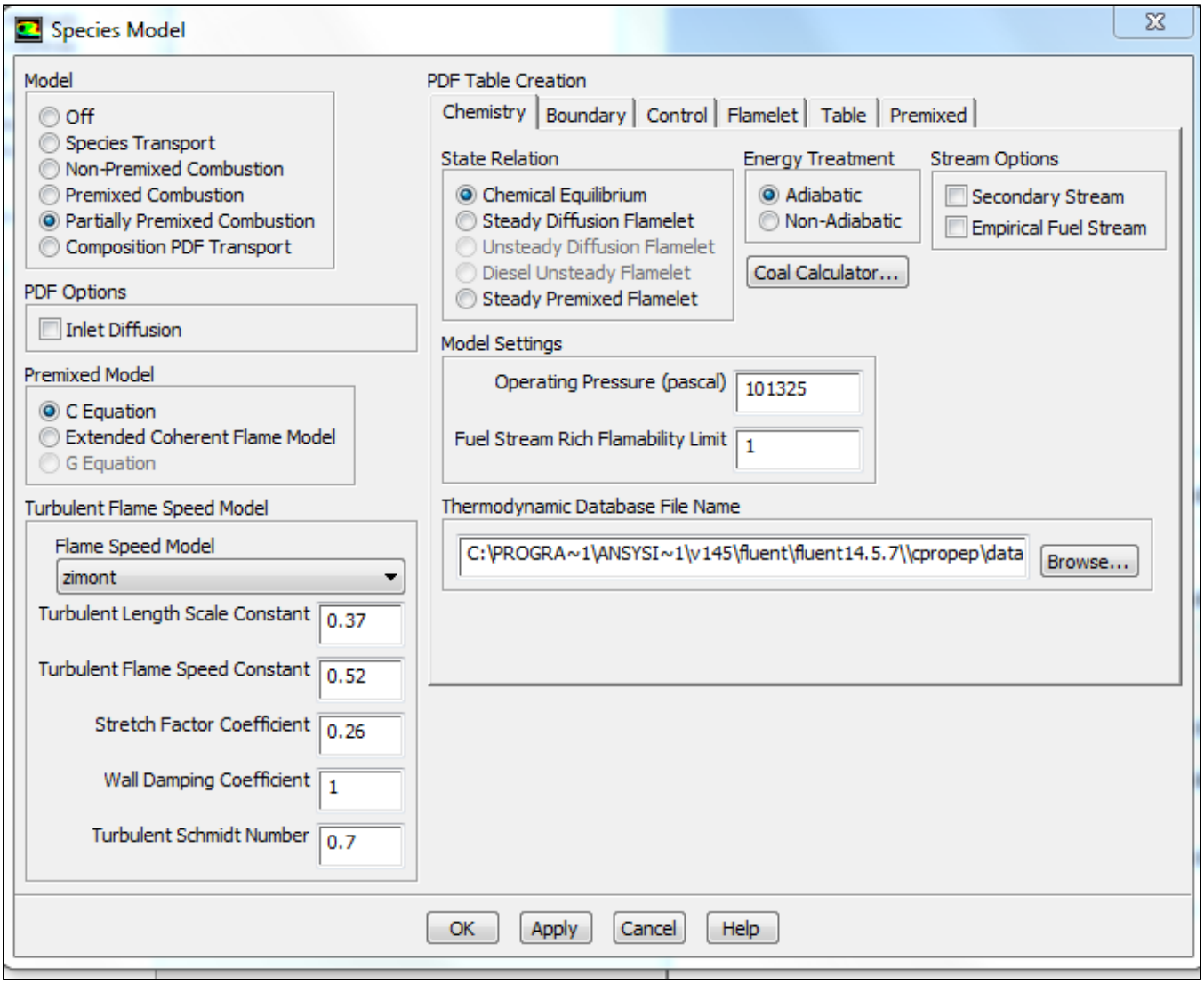
General

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.

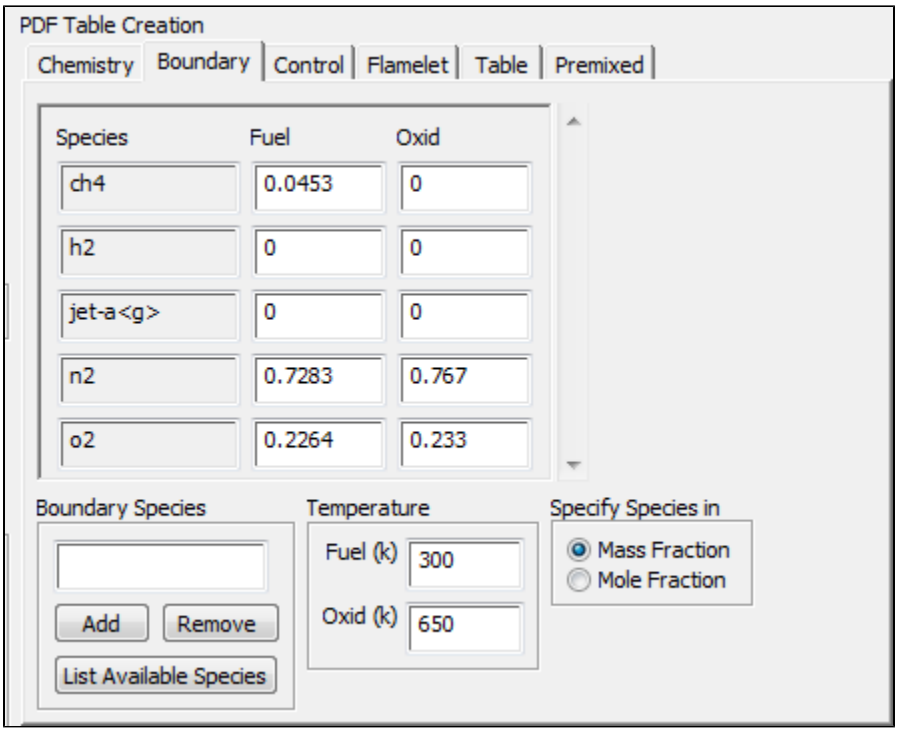
Models

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 = 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.



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.

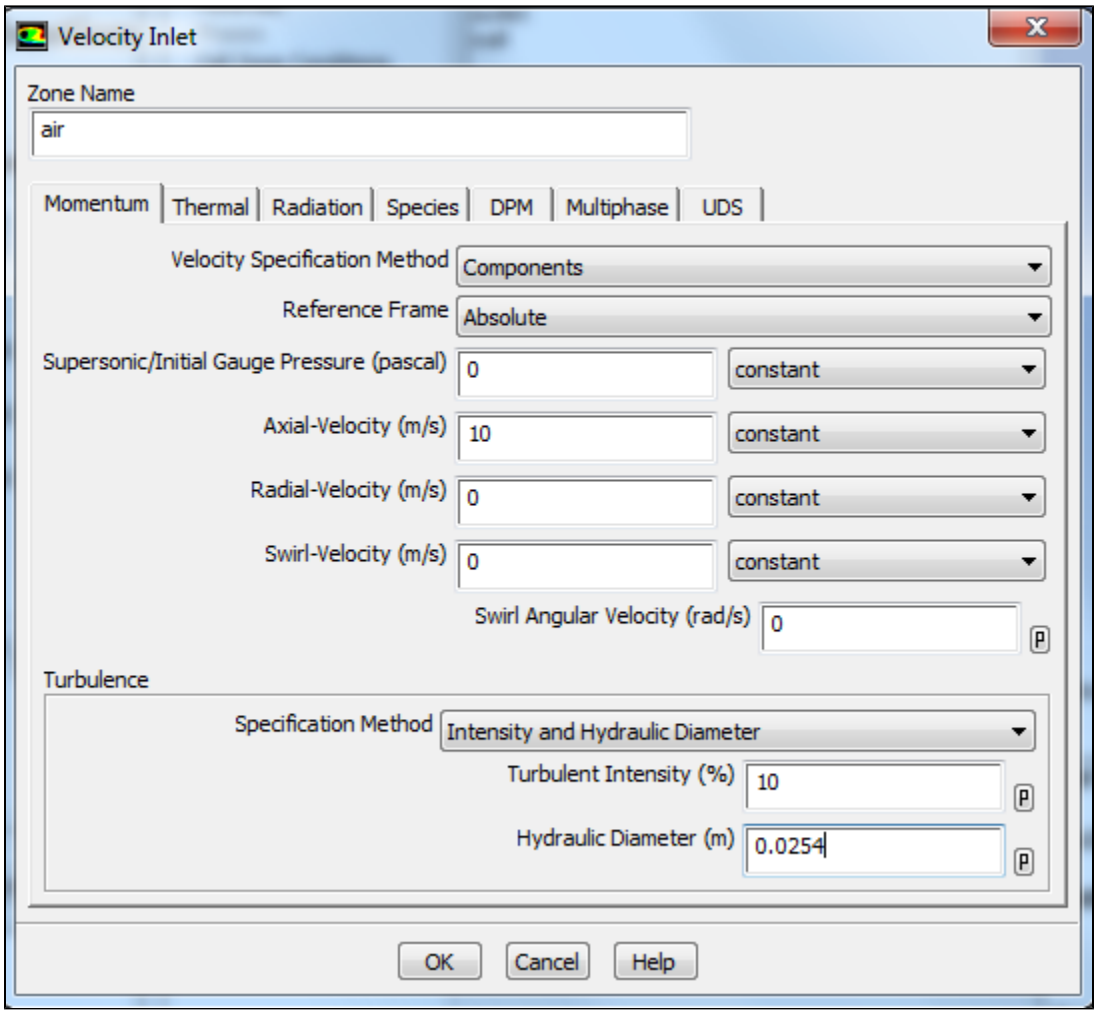
Materials

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".

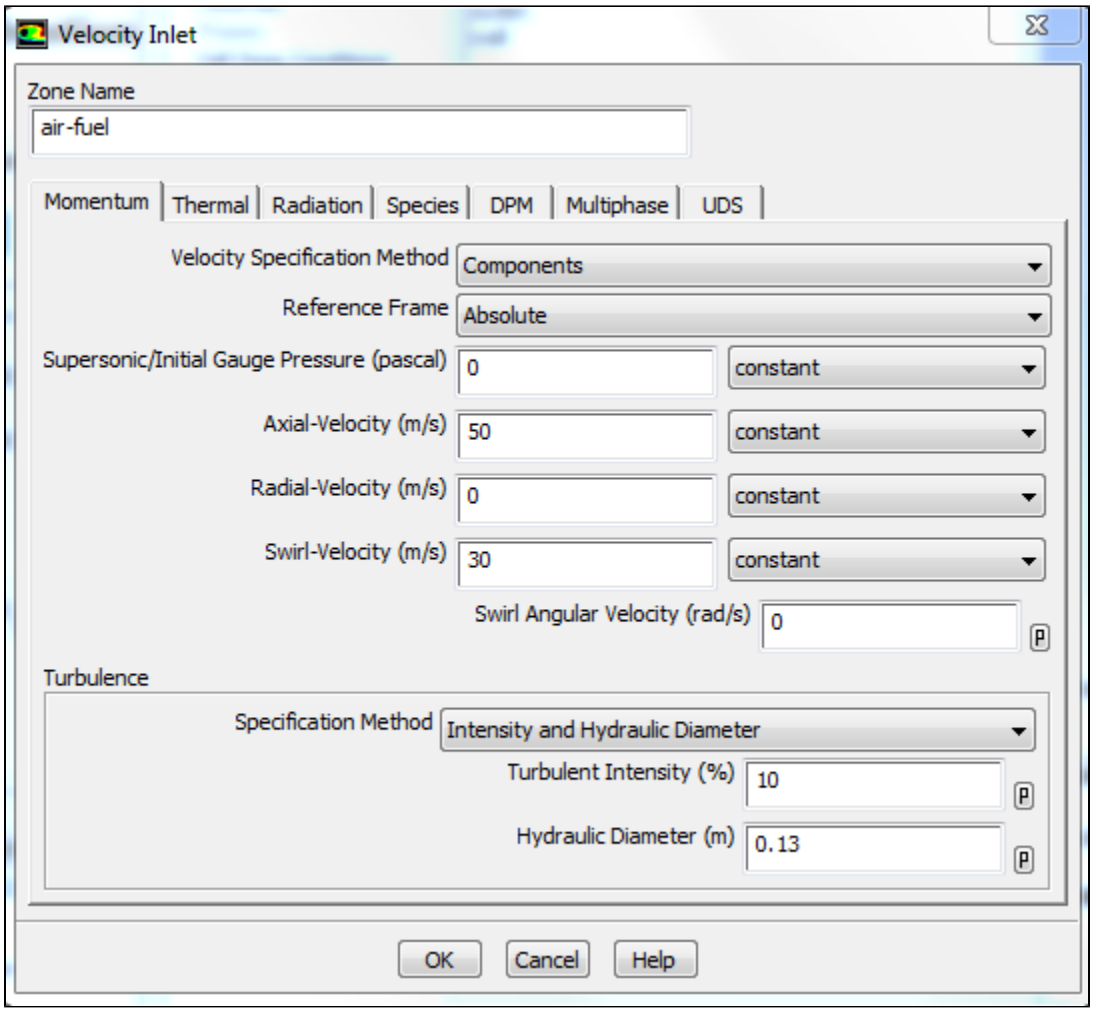
Boundary Conditions

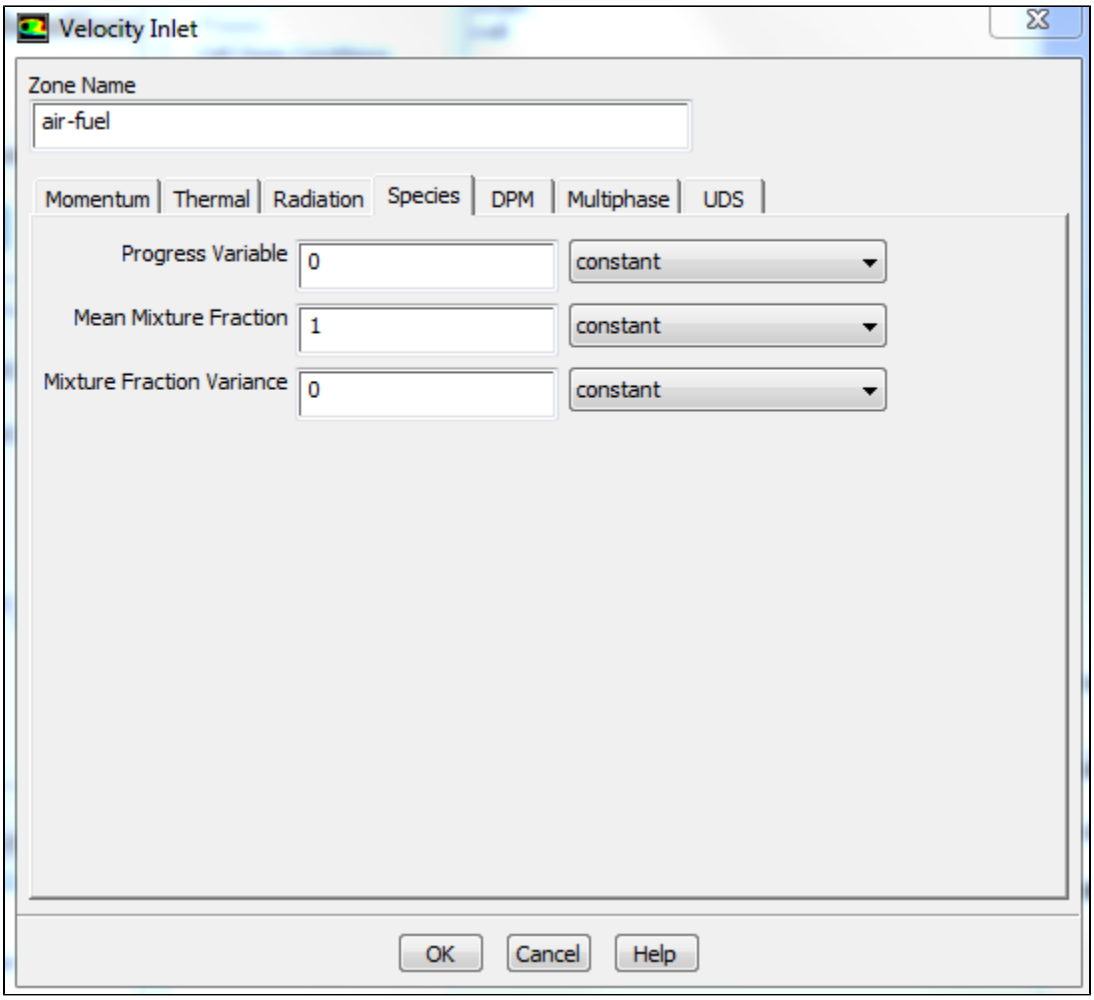
In Solution Setup-Boundary Conditions, because this is a turbulent model, all turbulence parameters must be specified on the boundaries of the combustor.

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.

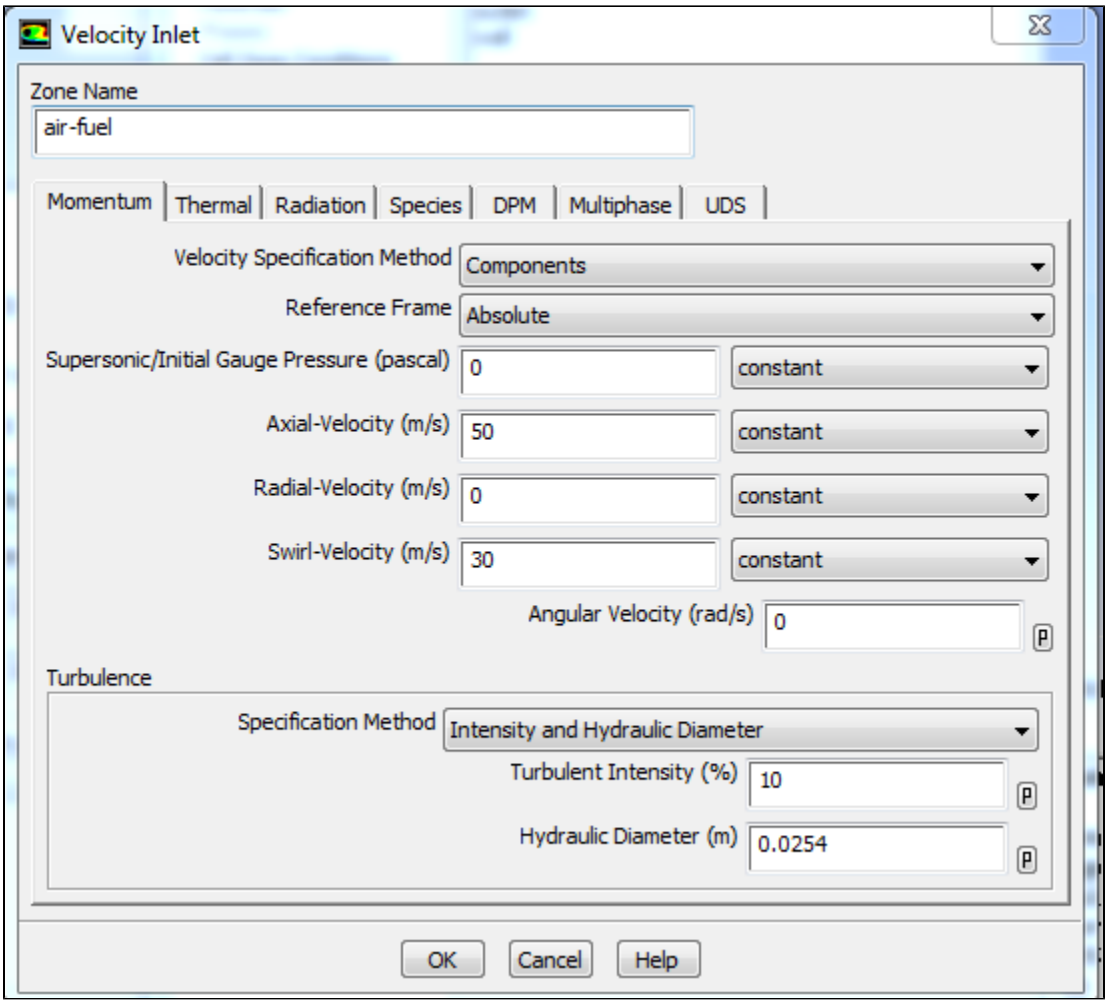


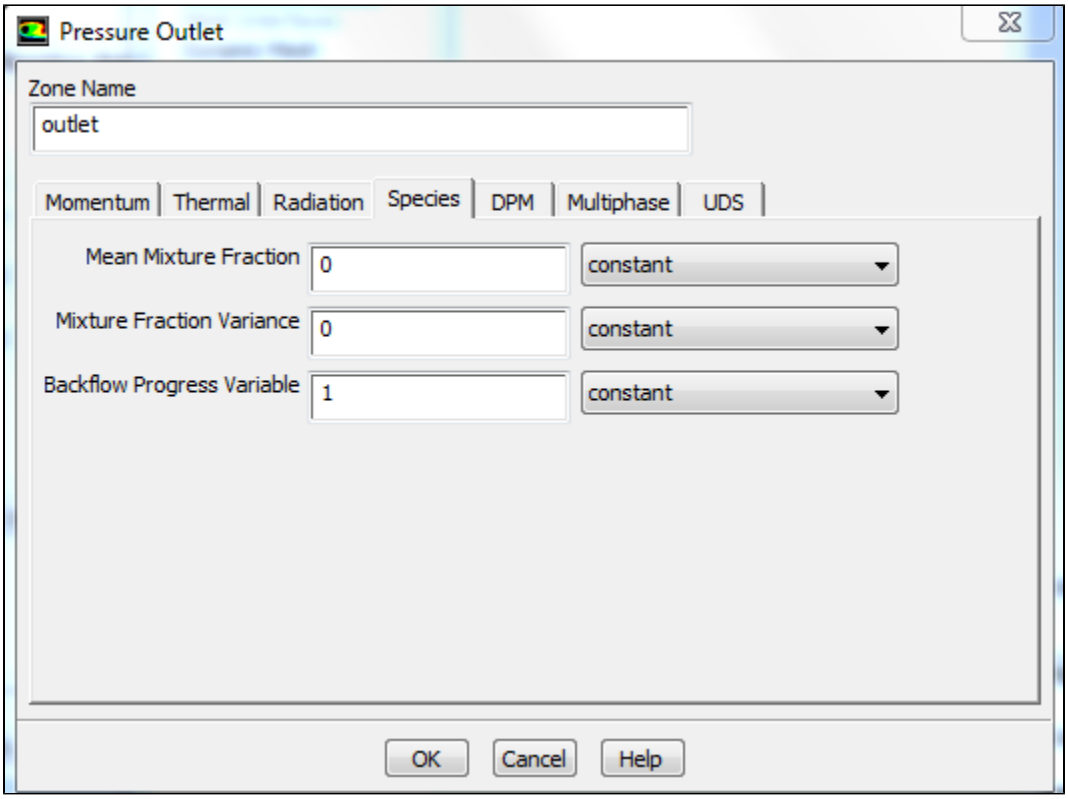
Similarly, edit the air-fuel boundary to have velocity components, with 50m/s axial velocity and 30m/s swirl velocity. The turbulent intensity should be 10%, but the hydraulic diameter should again be 0.0254m. Clicking to the Species tab, change the mean mixture fraction to 1.0. This is because this 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
Turbulence
Specification Method
Intensity and Hydraulic Diameter
Turbulent Intensity (%)
10
Hydraulic Diameter (m)
0.13
OK
Cancel
Help



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
Turbulence
Specification Method
Intensity and Hydraulic Diameter
Turbulent Intensity (%)
10
Hydraulic Diameter (m)
0.0254
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