FGM implementation in Ansys Fluent 15.0

Application to Sandia flame D

Roberto Stella

14/12/2014

Case description

This tutorial explains how to correctly implement the Fluent 15.0 built in Flamelet Generated Manifolds method and be able to simulate premixed flames. In particular this tutorial focuses on the simulation of the Sandia flame D.

The jet fluid is a mixture of three parts air and one part CH_4 by volume. The mixing rates are high enough that these flames burn as diffusion flames, with a single reaction zone near the stoichiometric mixture fraction and no indication of significant premixed reaction in the fuel-rich CH_4/air mixtures. Flame D operates at a Reynolds number of 22400. The pilot is a lean mixture of C_2H_2, H_2, air, CO_2, and N_2 with the equivalence ratio of 0.77.

Step-by-step guide to the simulation

Files needed

For the simulation you will need the mesh file "sandia-d.msh", the thermal properties database "thermo.db" and the chemical mechanism file "gri-3.0.che".

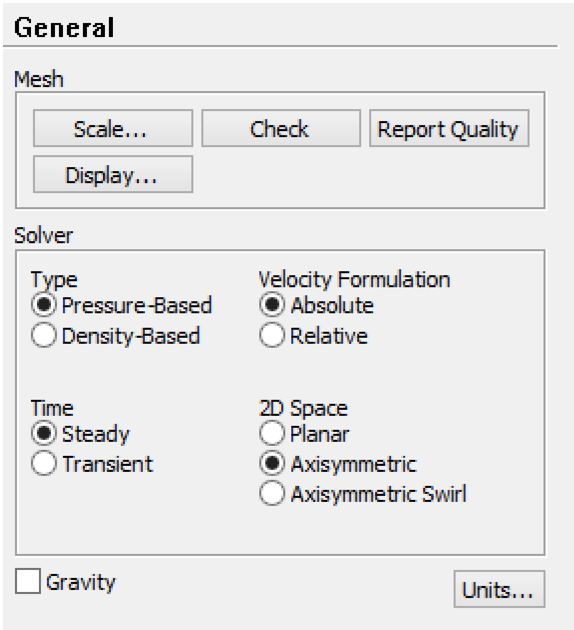
Starting of Fluent 15.0

In the Fluent launcher choose a "2D" dimension and leave the "Double Precision" checkbox unchecked. In "Working directory" browse the working path that you are using for the simulation.

Mesh import

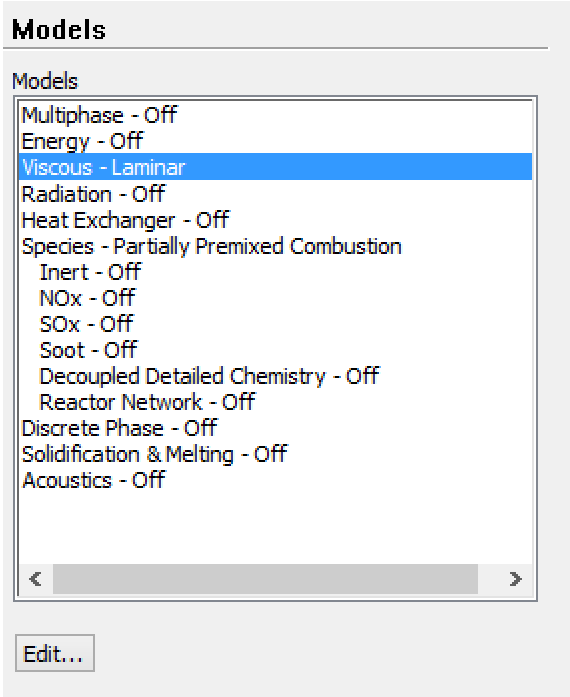
When Fluent is opened import the mesh by clicking on File→Read→Mesh…

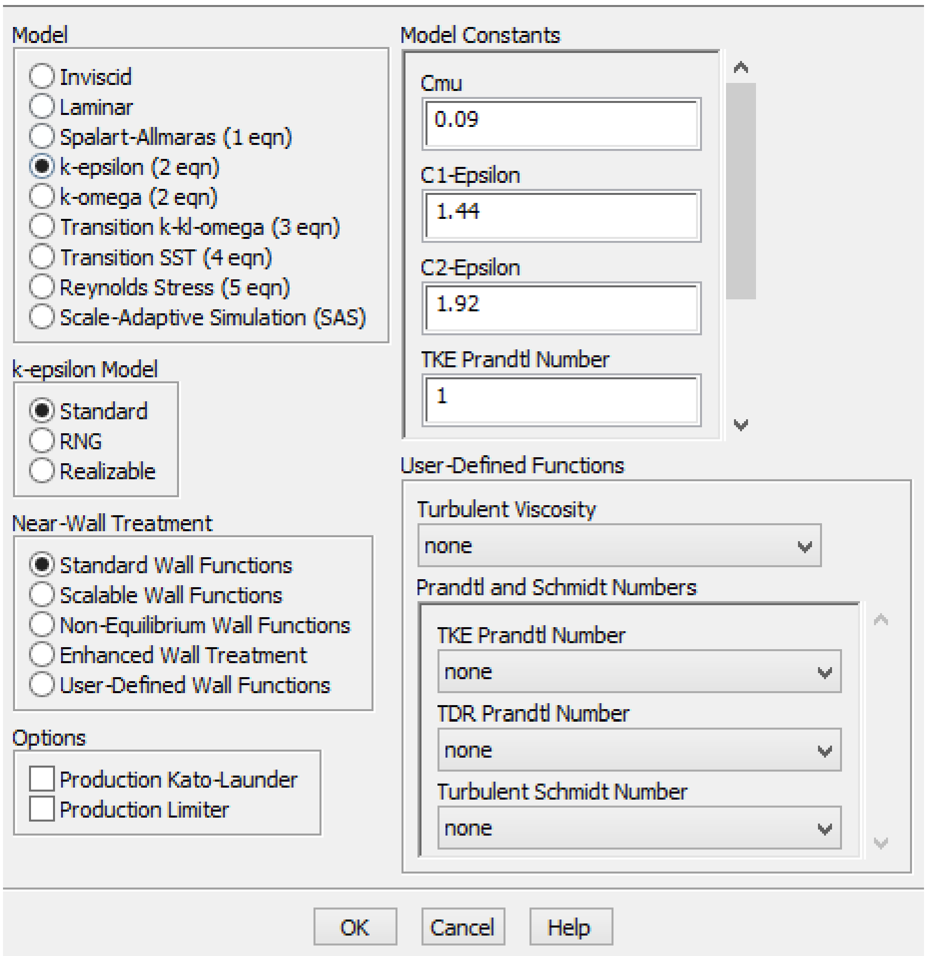
Under the "Solution Setup" tree, in the "General" branch choose "Axisymmetric" under 2D space.



Turbulence model setup

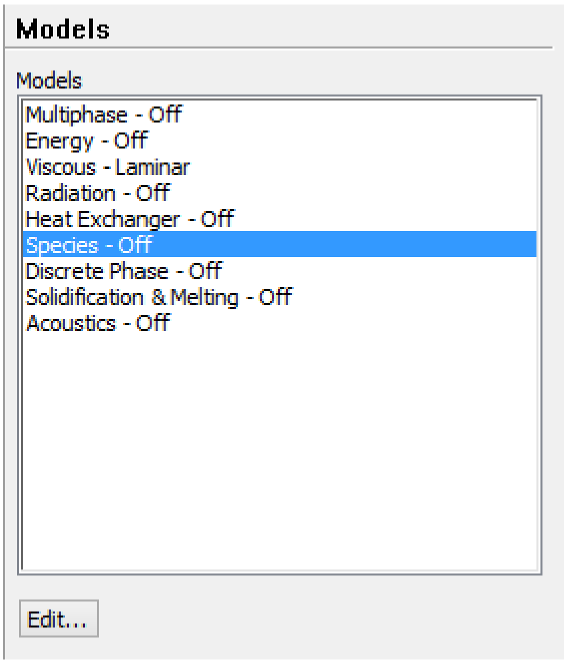
Select "Model" under the "Solution Setup" tree. Click on "Viscous-Laminar" and then click to "Edit…" then check the "k-e (2 equation)" checkbox and leave all the parameters as default.



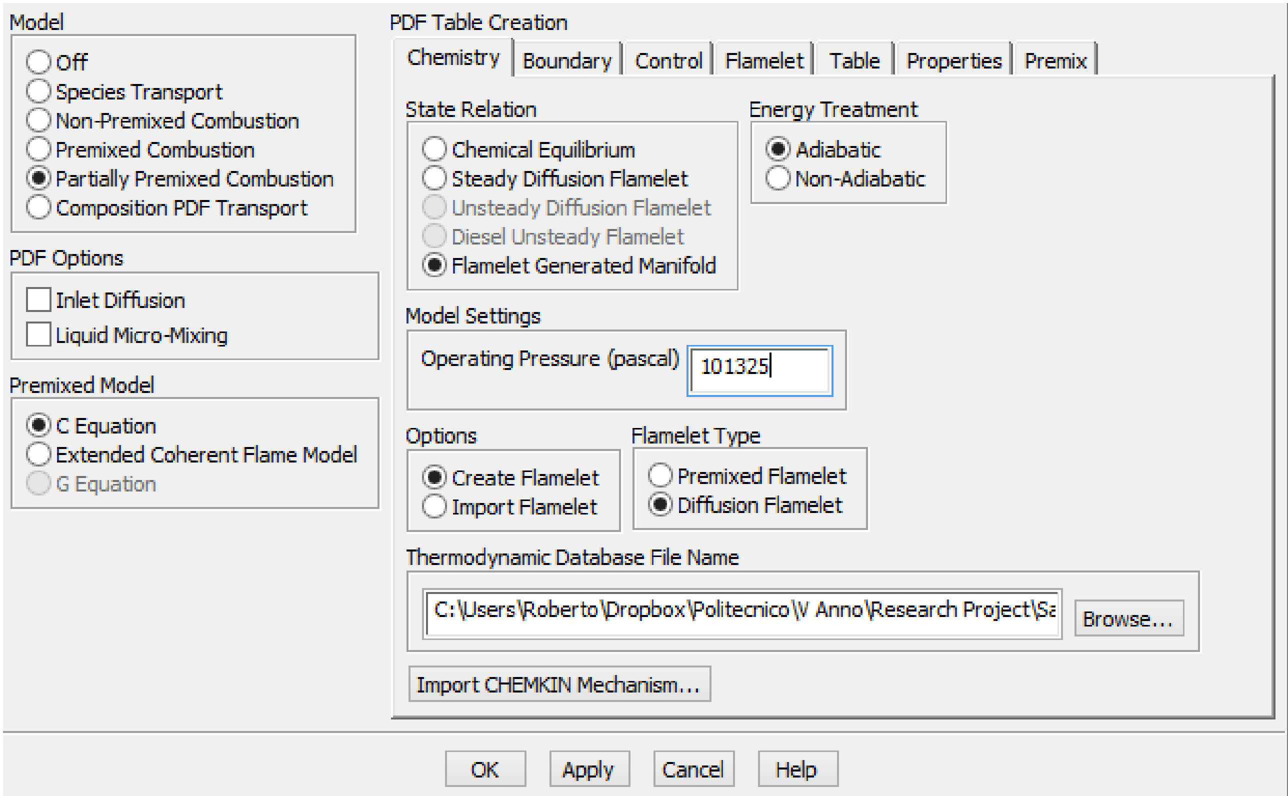


Set up of the Flamelet Generated Manifold model

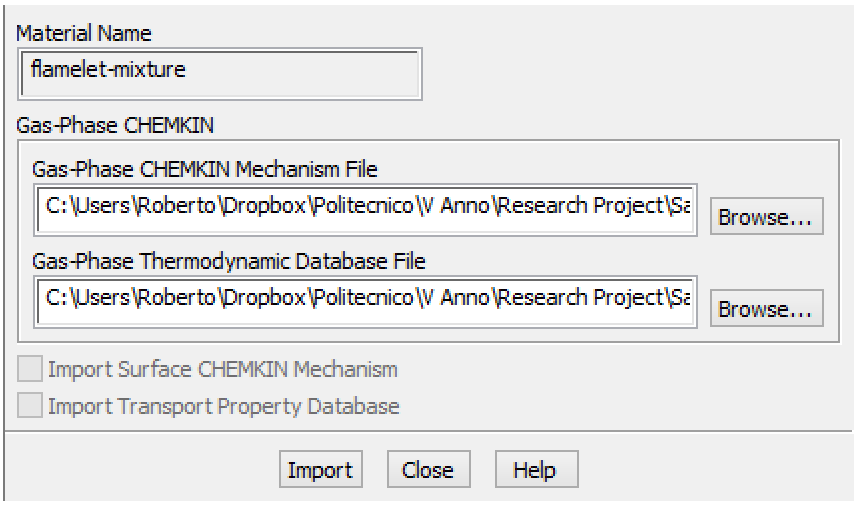
Under the “Model” branch select “Species” and click on “Edit…”



Now check “Partially Premixed Combustion”. Set the operating pressure to 101325 Pa. Under “Options” select “Create Flamelet” and under “Flamelet Type” choose “Diffusion Flamelet”



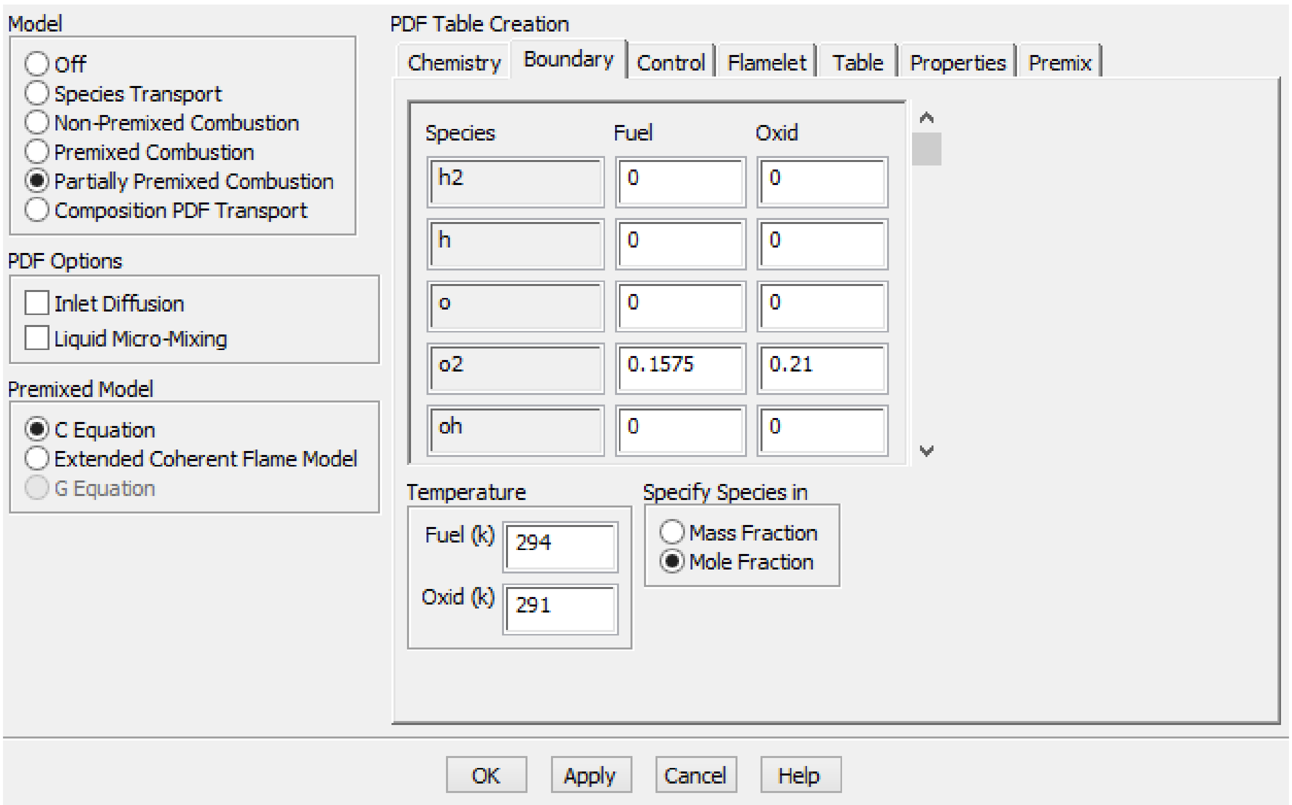
To import the Thermodynamic Database file click on “Browse…” and select the “thermo.db” file that you should keep in the working directory.
To import the chemical mechanism click on “Import CHEMKIN Mechanism…”
Under “Gas-Phase CHEMKIN Mechanism File” browse to the “gri-3.0.che” file that you should keep in the working directory.
Under “Gas-Phase Thermodynamic Database File” browse again to the “thermo.db” file.



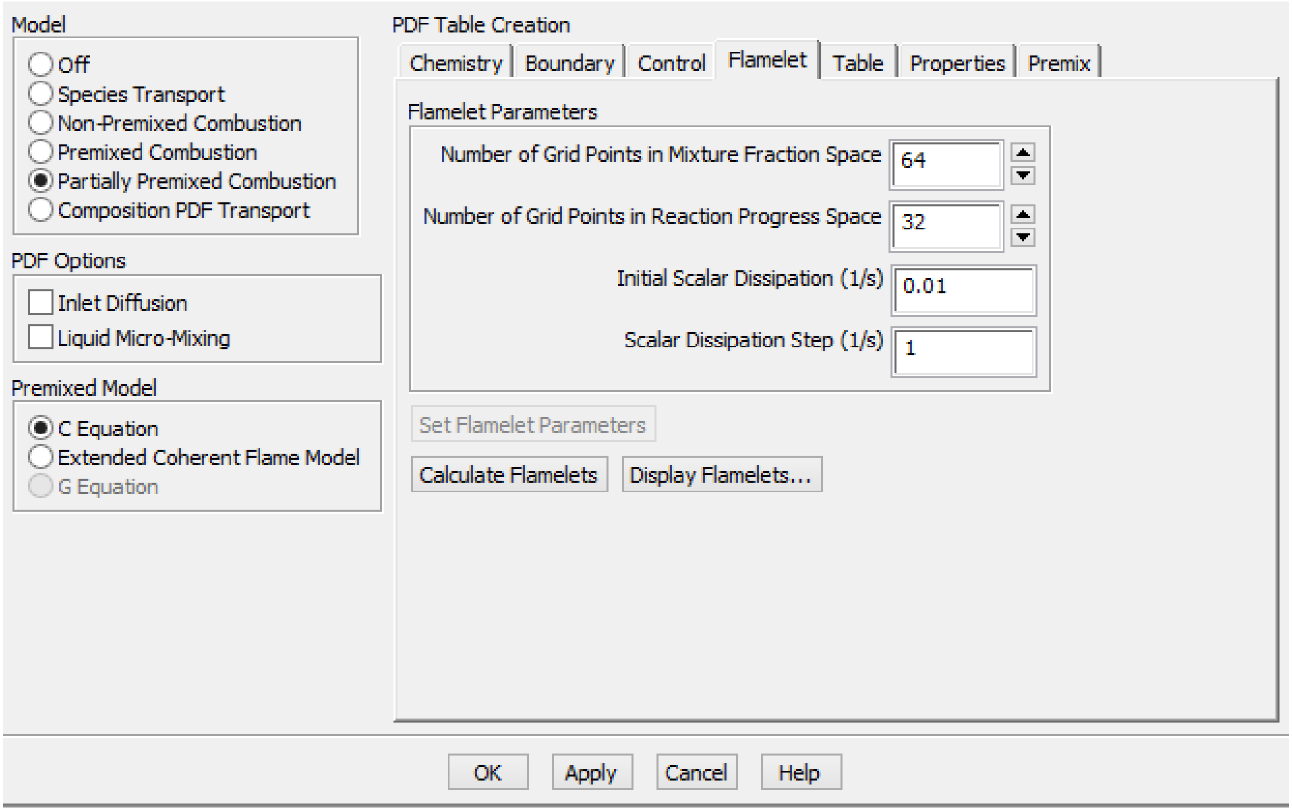
Now move to the "Boundary Tab" and set the value of the mole fractions of the species as seen in the following table. Remember to check "Mole Fraction" under "Specify species in".

Species	FUEL	OXIDIZER
O2	0.1575	0.21
N2	0.5925	0.79
CH4	0.25	0

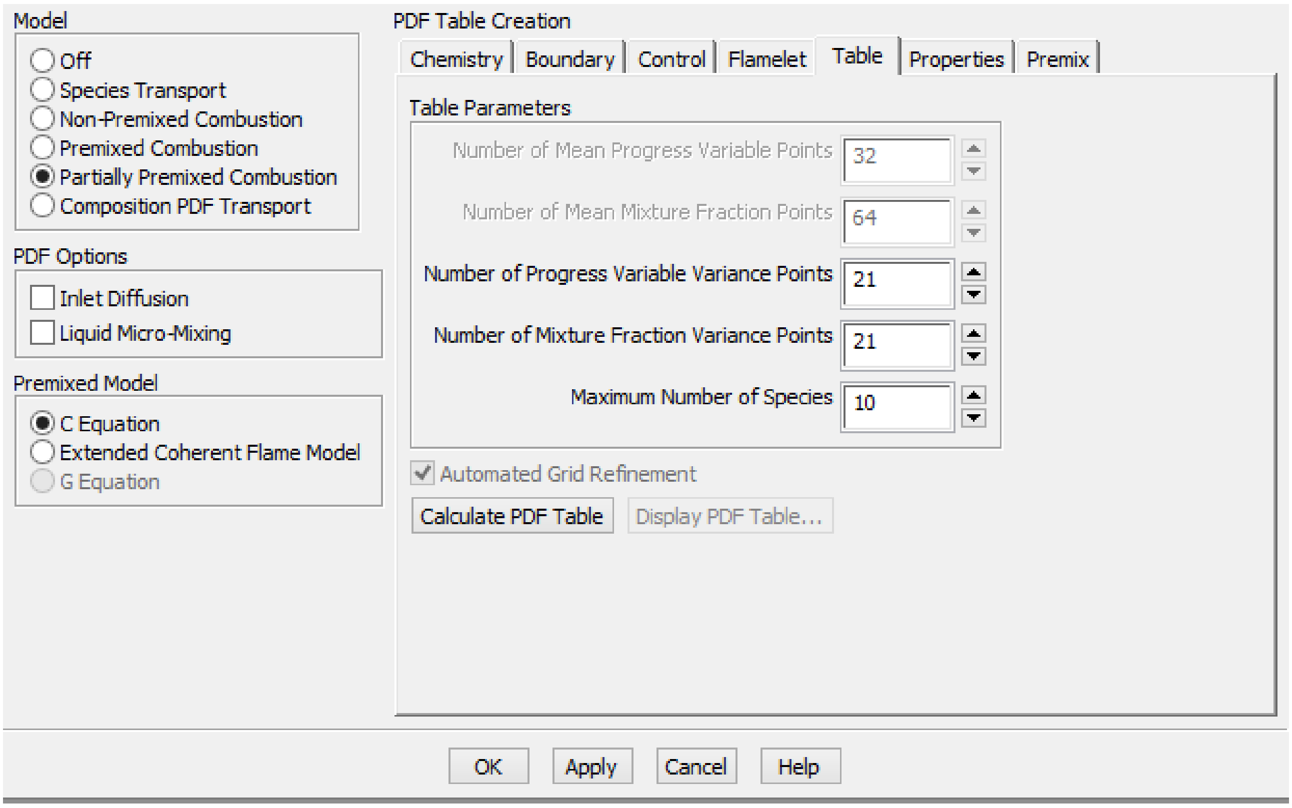
Set the fuel temperature to 294 K and the oxidizer one to 291 K.



Leave the settings on the "Control" tab as default and move to the "Flamelet" tab.
Modify the "Number of Grid Points in Mixture Fraction Space" to 64 and the "Number of Grid Points in Reaction Progress Space" to 32. Modify the "Scalar Dissipation Step" to 1. Now click on "Calculate Flamelets"



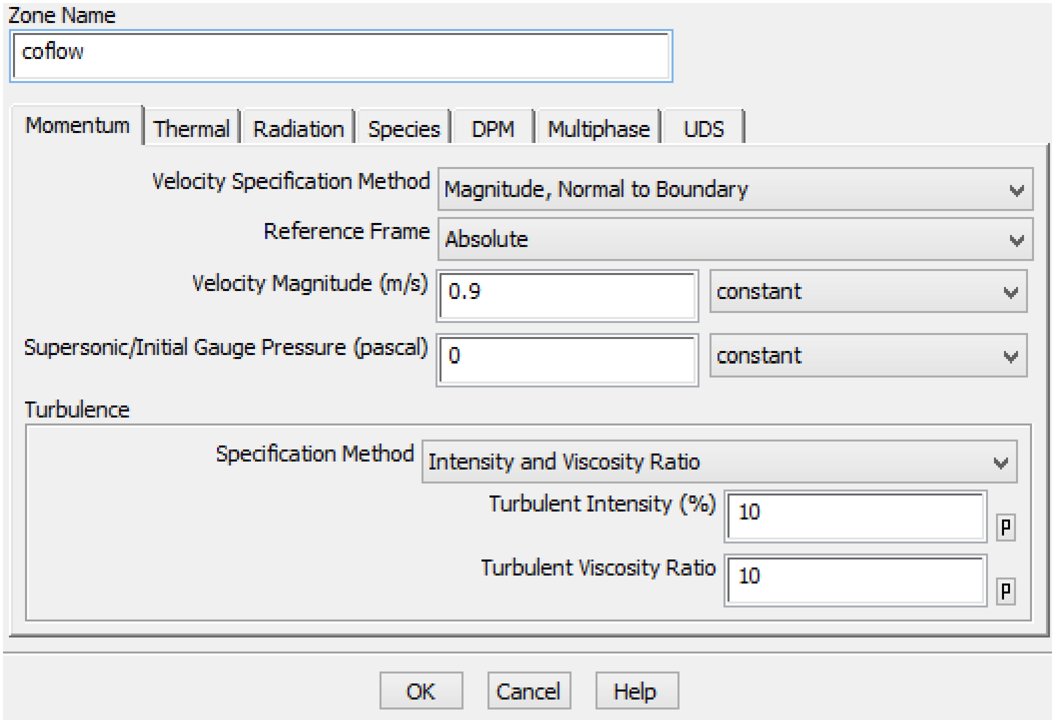
After the calculation is finished move to the "Table" tab. Select 21 for both "Number of Progress Variable Variance Points" and "Number of Mixture Fraction Variance Points". Modify to 10 the "Maximum Number of Species".



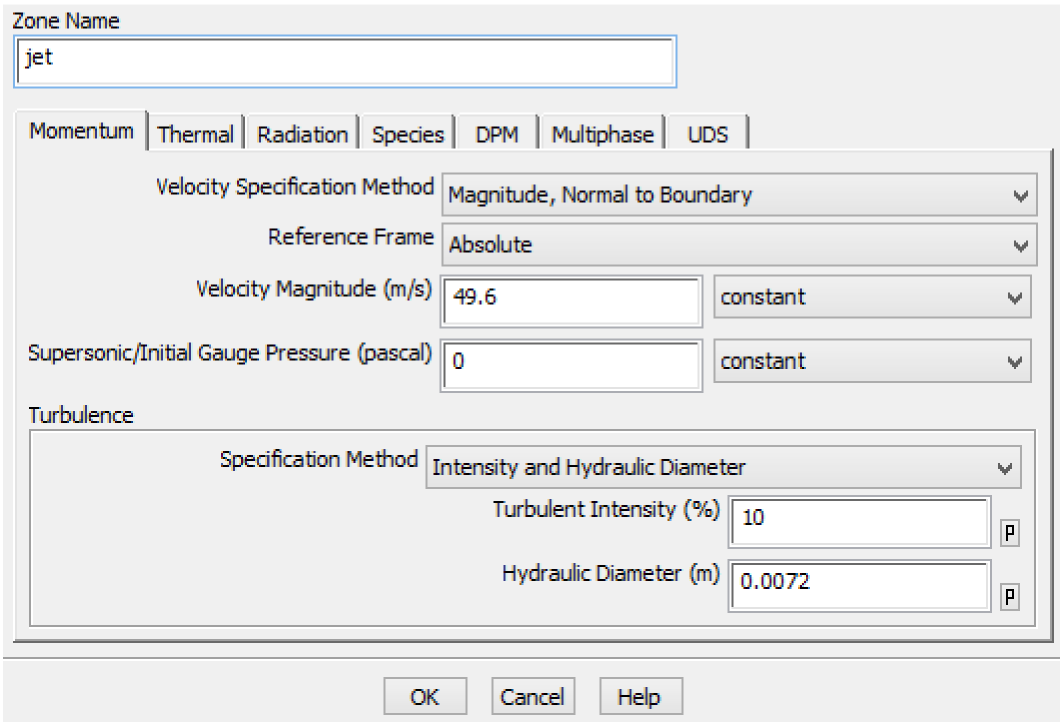
Click on "Calculate PDF Table". When the calculations are finished move to the "Premix" Tab and under "Turbulence-Chemistry Interaction" select "Finite Rate".

Boundary Conditions

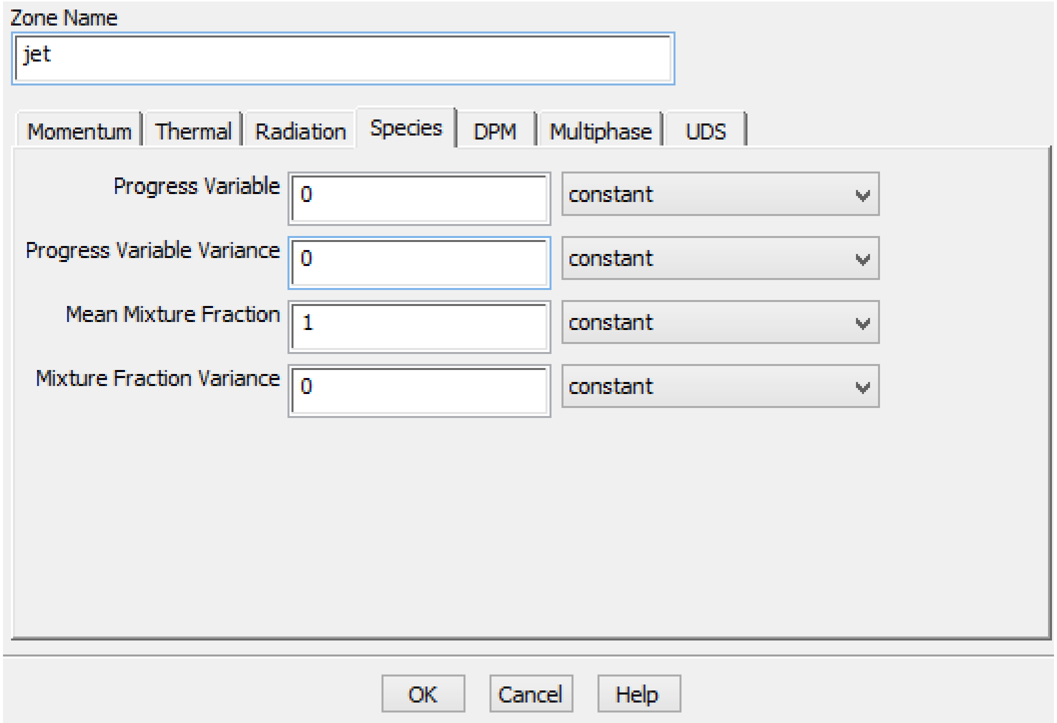
Under the "Solution Setup" tree select the "Boundary Conditions" branch. Select "Coflow", click "Edit…" and apply the momentum boundary conditions as shown in the figure below. Since "Coflow" is an oxidizer only velocity inlet the default species boundary conditions don't need to be modified.



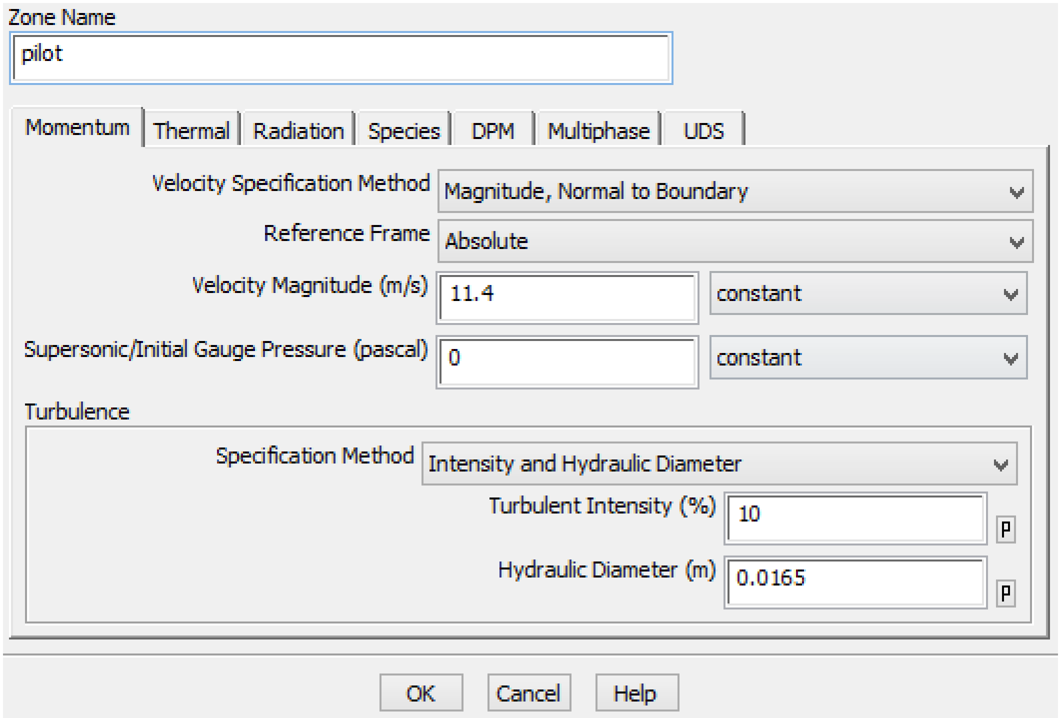
Select "Jet", click "Edit…" and apply the momentum boundary conditions as shown in the figure below.



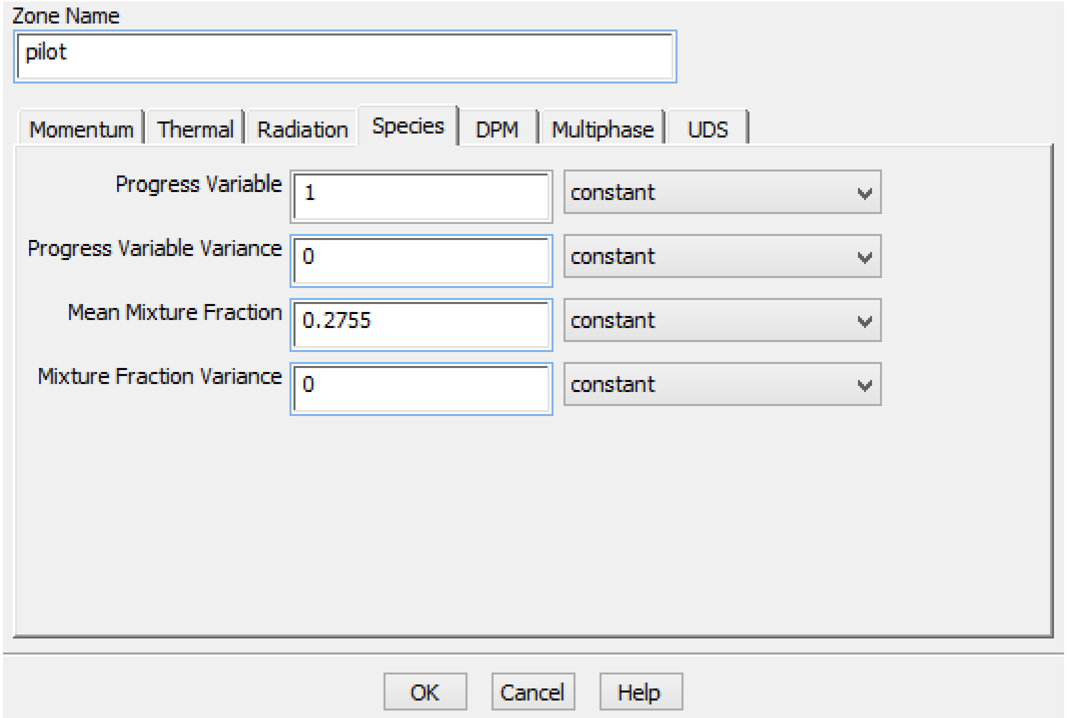
Since "Jet" is the fuel inlet it is necessary to modify the species boundary condition changing the mixture fraction value to 1 (corresponding to 100% fuel and 0% oxidizer).



Select "Pilot", click "Edit…" and apply the momentum boundary conditions as shown in the figure below.



Since "Pilot" is the pilot flame it is necessary to modify the species boundary condition changing the mixture fraction value to 0.2755 (corresponding to 27,55% fuel, 72.45% air) and the progress variable value to 1 (meaning that the mixture is already burnt).



Solution methods
Now select "Solution Methods" branch under the "Solution" tree. Under "Spatial discretization" modify "Gradient" to "Green-Gauss Cell Based" and select a first order approximation for "Turbulent Kinetic Energy" and "Turbulent Dissipation Rate" while a second order approximation for all the other variables.

Solution controls
Select "Solution Controls" branch under the "Solution" tree. Modify the Pressure Under-Relaxation Factor to 0.7 and the Momentum one to 0.3. Leave all the other values to default.

Solution initialization
Select "Solution Initialization" branch under the "Solution" tree. Check "Standard Initialization". Initialize the values as follows:
Axial Velocity: 0.9 m/s
Turbulent Kinetic Energy: 0.01215 m2/s2
Turbulent Dissipation Rate: 0.09 m2/s3
Progress Variable: 1
Leave all the other values to 0 then click on "Initialize".

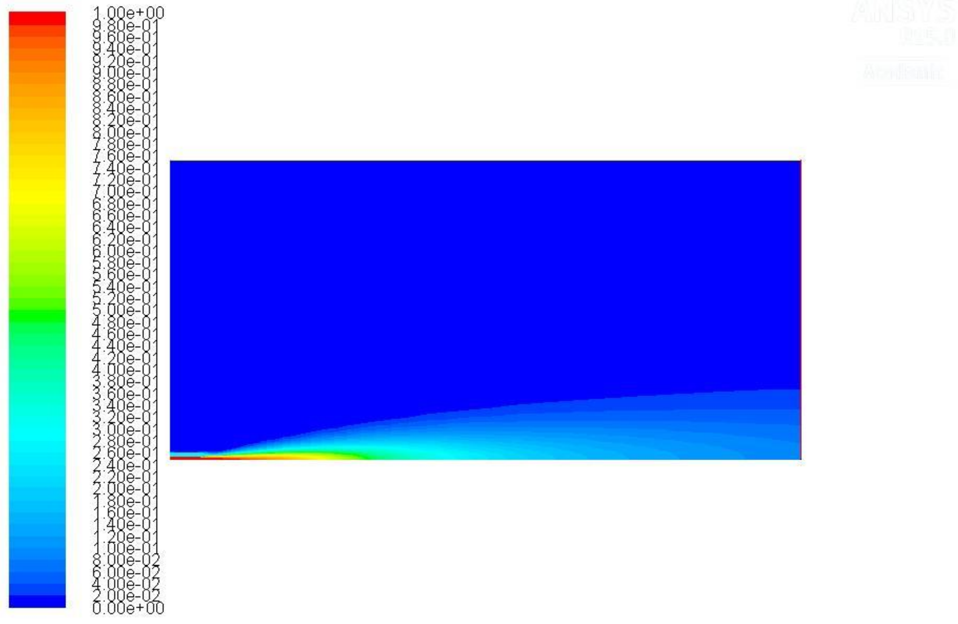
Monitors

Select "Monitors" under the "Solution" tree. Select "Residuals" and click "Edit". Then set "Convergence Criterion" to "none".

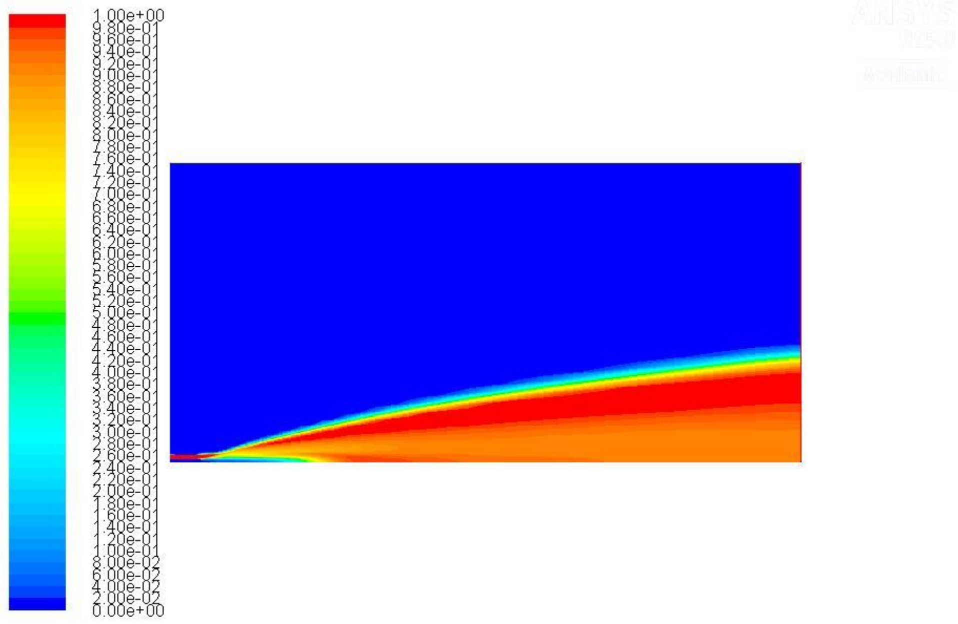
Calculation
Select "Run Calculation" branch under the "Solution" tree. Modify the "Number of Iterations" to 2500. Click on "Calculate".

Solution

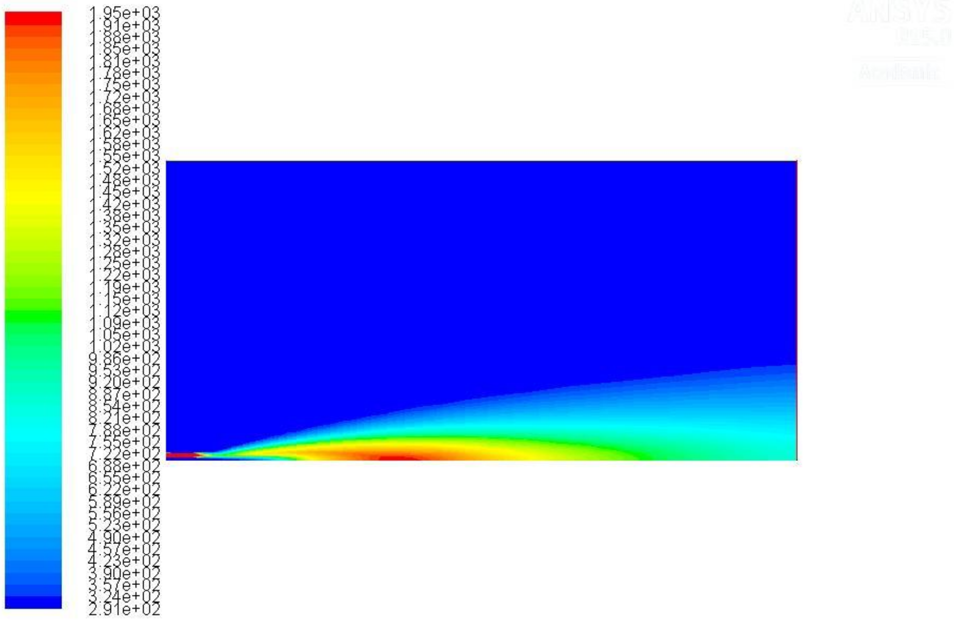
After the calculation is converged you should obtain a solution that is similar to the following:



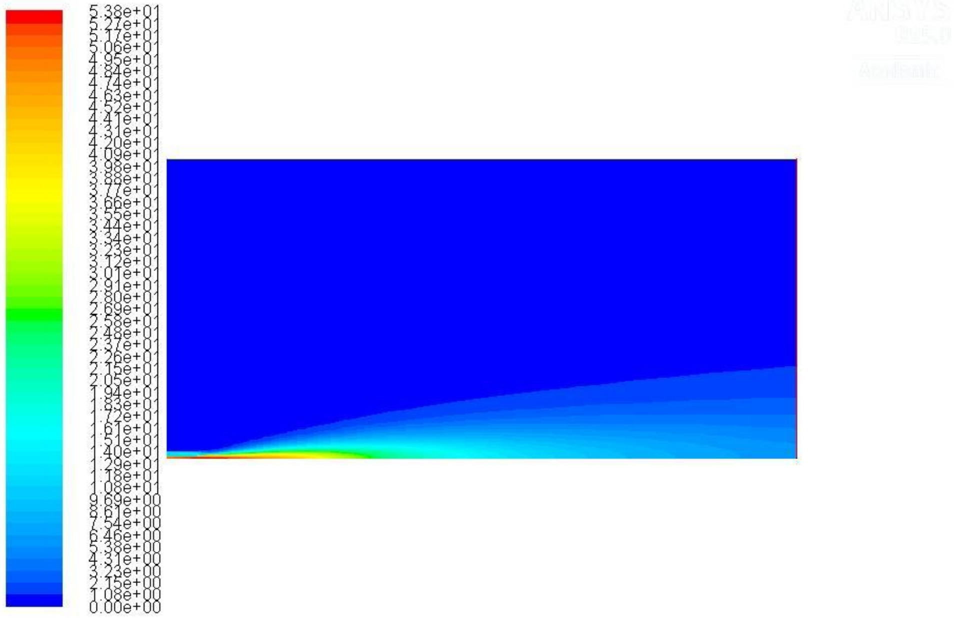
Contours of Mean Mixture Fraction — Feb 19, 2015 — ANSYS Fluent 15.0 (axi, dp, pbns, pdf10, ske)



Contours of Progress Variable — Feb 19, 2015 — ANSYS Fluent 15.0 (axi, dp, pbns, pdf10, ske)



Contours of Static Temperature (k) Feb 19, 2015
ANSYS Fluent 15.0 (axi, dp, pbns, pdf10, ske)



Contours of Velocity Magnitude (m/s) Feb 19, 2015
ANSYS Fluent 15.0 (axi, dp, pbns, pdf10, ske)