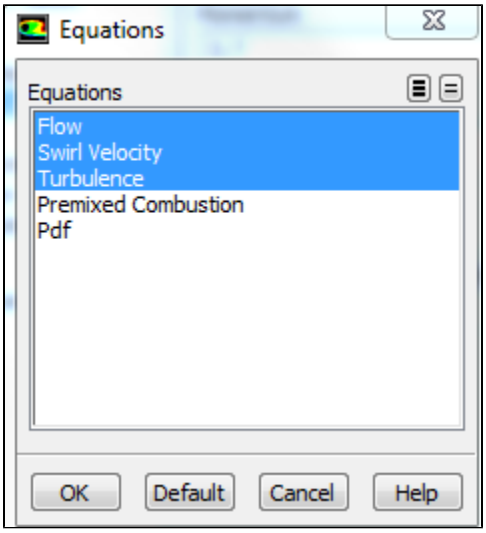
Partially Premixed Combustion - Numerical Solution

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

Numerical Solution

Skipping 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. Press OK.



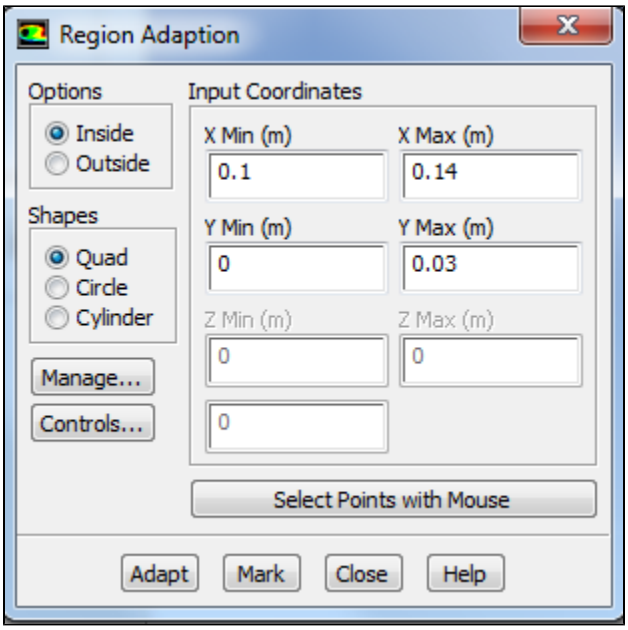
Go to Solution Initialization. Press Initialize; let Fluent initialize the domain as a Hybrid Initialization, taking all boundaries into account.

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). Press OK.

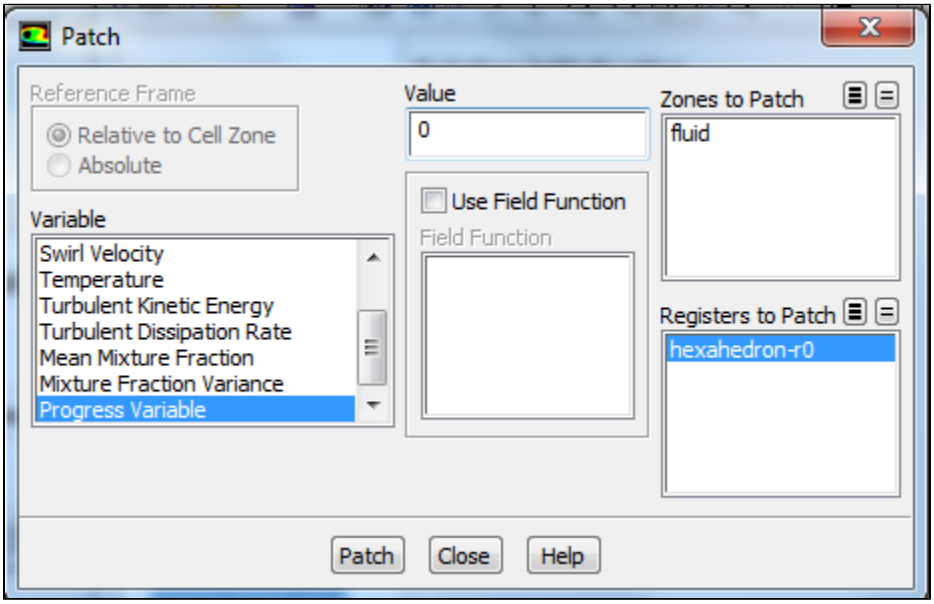
Still on Run Calculation, enter 1000 under the number of iterations and run the simulation until it converges.

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 to 0.14m, Y from 0 to 0.03m. Click "Mark", which marks 207 cells for refinement, and then click "Close".

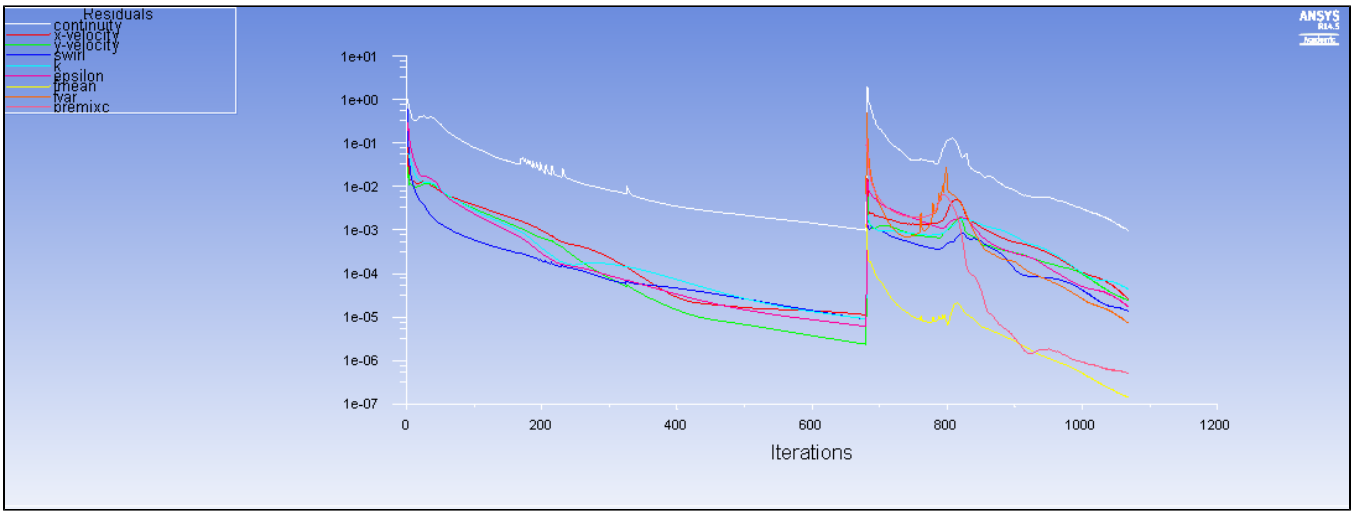


Patch this region by going to Solution Initialization, and click "Patch". Select "Progress Variable" as the variable, and patch the region that you just marked by clicking "Patch". Close the dialogue after patching.



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. Press OK. 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 (convergence criterion being that the residuals are all at least 1E-3 or smaller).



Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules