

Partially Premixed Combustion - Numerical Results

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](#)

Numerical Results

In FLUENT, go to **File-Write-Case&Data** at the top menu.

Open CFD-POST and click **File-Load Results**. Select the data file that you just wrote as the file to load.

Create contours of:

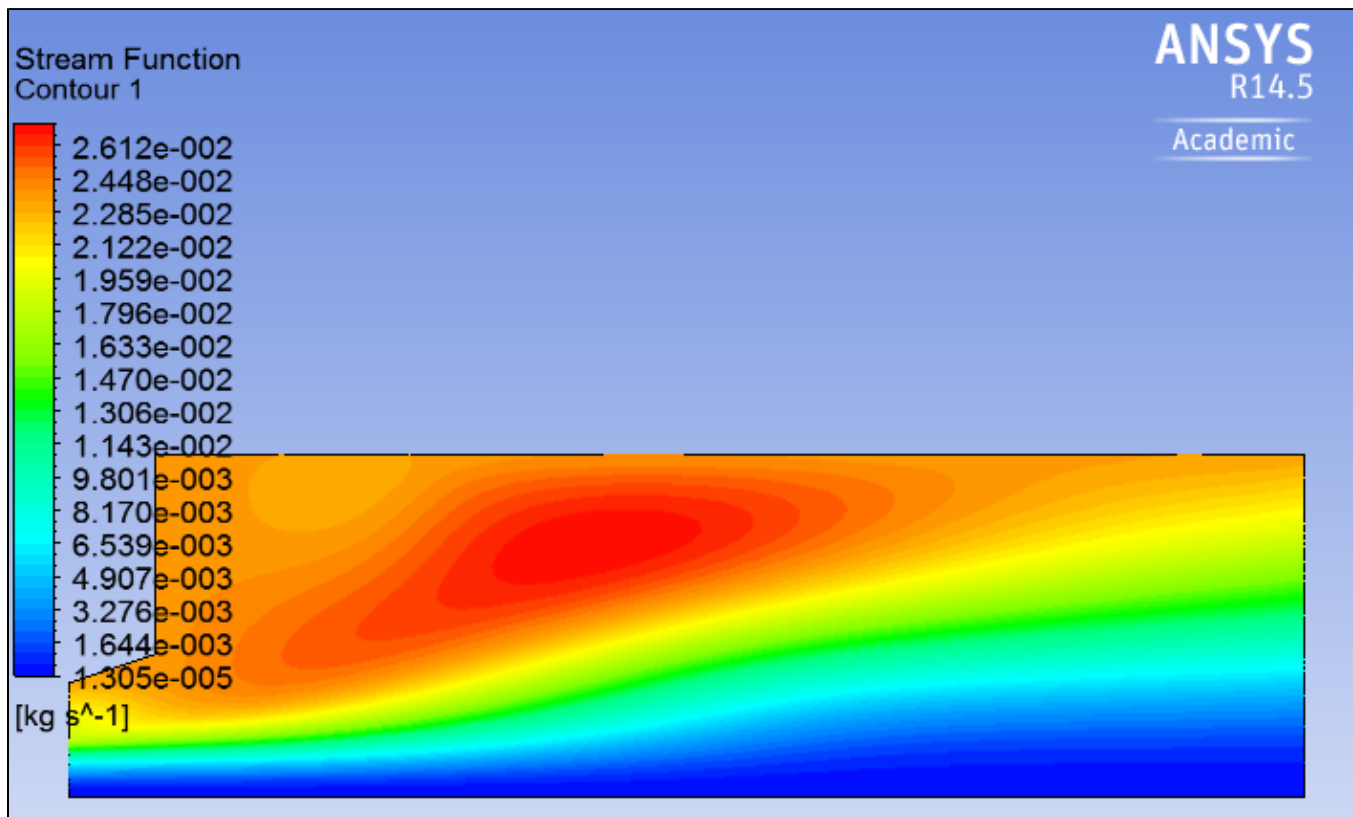
- Stream Function
- Progress Variable (called Reaction Progress)
- Temperature
- Mass Fraction of CH₄
- Turbulent Flame Speed

To do this, click the Contour button at the top menu

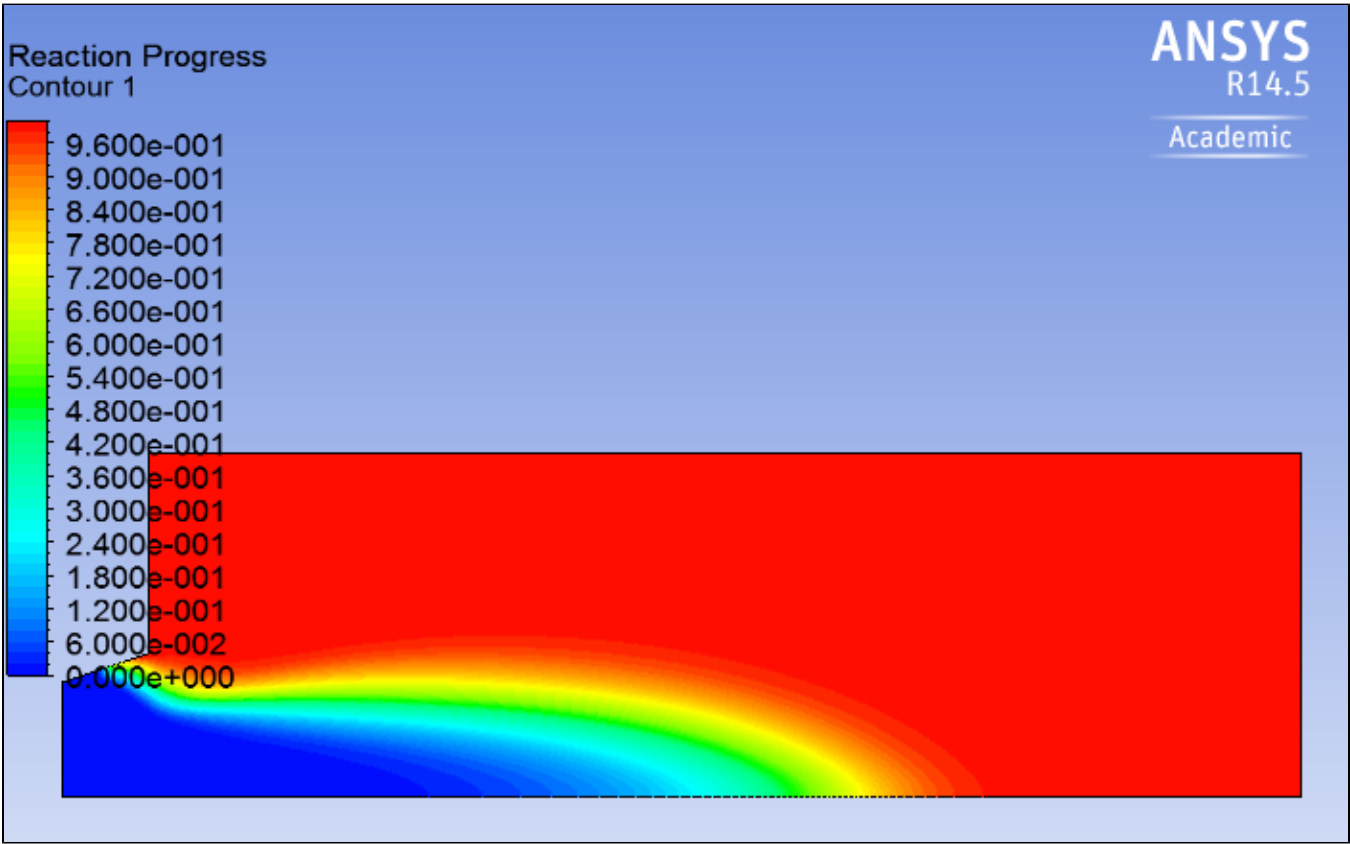


In the pop up dialogue box, name the contour after the variable that it will display and press OK. For all contour plots, select "periodic 1" for the location, and increase the number of contours to 51. Change the drop down box under Variable to the variable that you would like to create the contour of. Press "Apply" to generate the contour, and repeat this process for all contours.

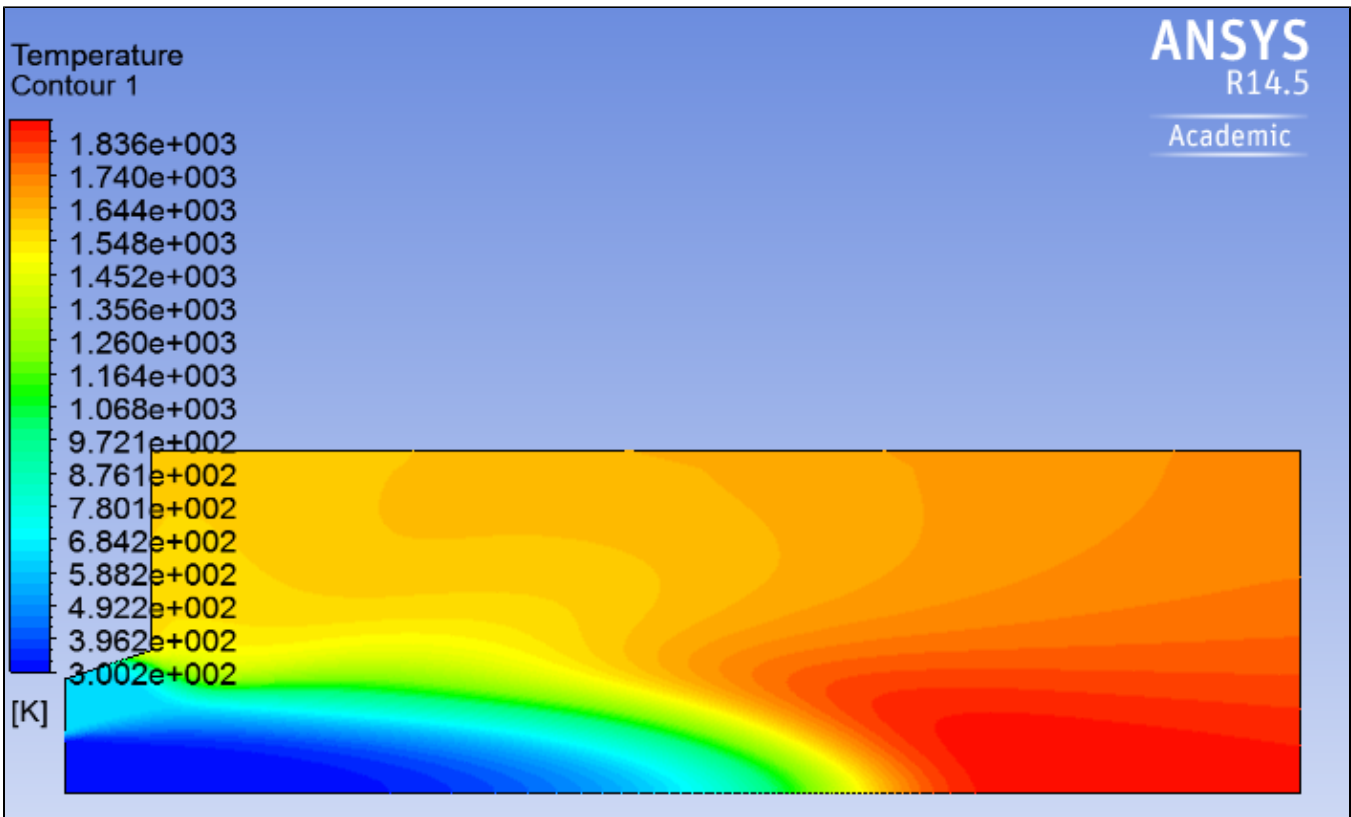
Stream Function:



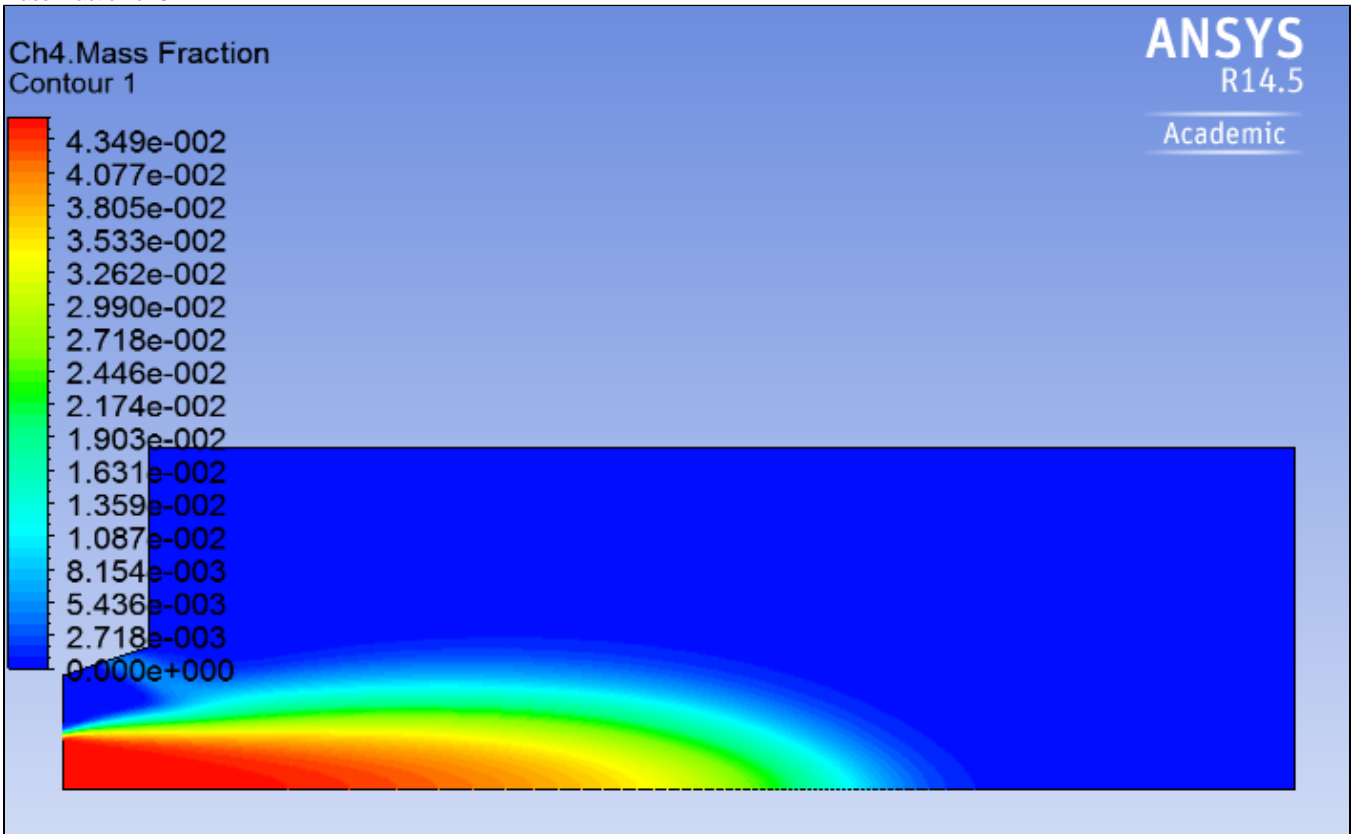
Progress Variable:



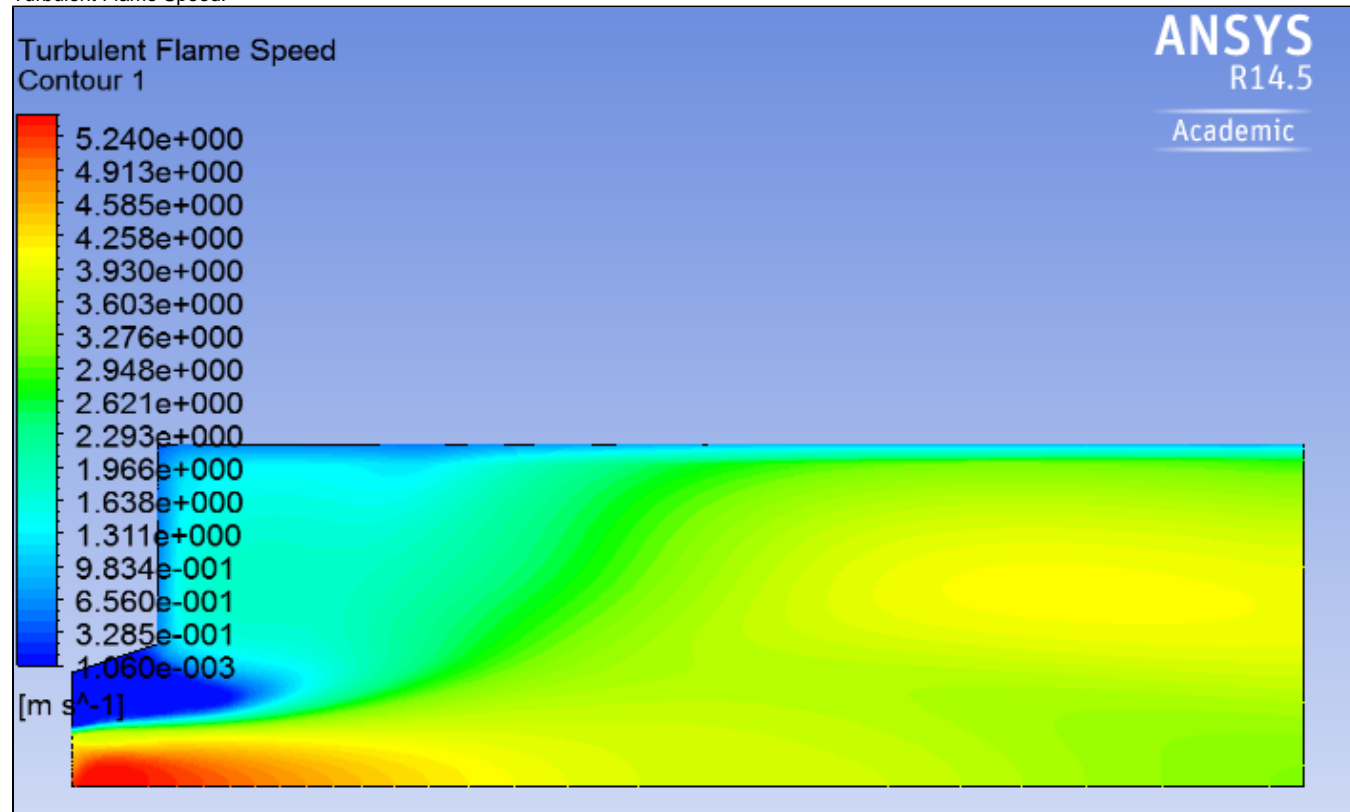
Temperature:



Mass Fraction of CH4:



Turbulent Flame Speed:



[Go to Verification & Validation](#)

[Go to all FLUENT Learning Modules](#)