2012 Gas Turbine Industrial Fellowship

Parker Hannifin Gas Turbine Fuel Systems Division

Sudipa Sarker

University Of Texas at El Paso

As a fellow of the 2012 University Turbine Systems Research (UTSR) Gas Turbine Industrial Fellowship Program, I worked with Parker Hannifin Corporation at the Gas Turbine Fuel Systems Division. I was assigned to be a part of research and development team, with my mentor being Dr. Erlendur Steinthorsson. I was involved in two different projects during my fellowship where I did CFD analysis to help the design improvement of fuel injectors. An example of the work done was a combustion analysis for an injector developed by Parker Hannifin for the NASA UEET project [1]. The Parker UEET injector proved highly successful in high-pressure combustion tests where record low emissions were obtained [2].

Introduction and description of the problem

Three-dimensional numerical simulations of airflow through an experimental Macrolaminated air swirler were performed. The air swirler was designed for target swirl number of 0.8 and was constructed from a stack of metal plates with air swirl passages etched into each plate that fed a 0.43" diameter "spray cup" that was 0.30" deep. A pressure swirl atomizer injected fuel at the bottom of the spray cup exposing the spray directly to the incoming swirling air. The air and fuel from the swirl cup discharged into a 10.0" inch long 3.00" by 3.00" square tube, as shown in Fig 1. The main objective of the analysis was to investigate different approaches for predicting spray flame location and NOx emissions.

In the results shown here, a flamelet model was used as a combustion model where a 31 step reaction mechanisms with 21 species was used with $C_{12}H_{23}$ used as a surrogate for Jet-A fuel. The gas was modeled as incompressible ideal gas. At first, a non-reacting case was run using a fixed pressure drop for the air. Then a reacting case was run using fixed airflow rate with steady flamelet modeling for combustion. Finally, an unsteady flamelet model was used as a post processing step to estimate the emissions of NO.

The CFD Model

The full three-dimensional flow of air through the swirler and the downstream domain was modeled. For non-reacting conditions, a constant pressure drop of 4000 Pa was applied across the swirler, with the exit pressure equal to 1 bar. (i.e., 4% pressure drop). The simulations were done using Fluent. The effects of turbulence on the flow were modeled using realizable k - epsilon turbulence model that is available in fluent. The grid system used in the simulations is a high quality, structured multi-block grid system that was generated using the GridPro grid generator (Fig 2). The grid system contained a little over $2.0 \cdot 10^6$ hexahedral cells. The hexahedral mesh resolves the flow field very efficiently and, in particular, contributes to high accuracy in boundary layers.

A hollow conical spray was used with a cone angle of 80°. A Rosin-Ramler distribution for the initial droplet size was used in this problem using A dynamic drag model was used here to account for the effects of droplet distortion.

Results and Discussion

The results from the simulations are shown in Fig. 3-7. Figures 3 and 4 show the flow field of non-reacting and reacting flow, respectively, showing the strongly swirling flow downstream of the cup and swirl induced vortex breakdown that stabilizes the flame. Fig 5 shows the temperature profile for the reacting flow where the maximum temperature is 2240 K and exit temperature is 1638 K with the burning zone positioned downstream of the cup. Figure 6 and 7 show the mass concentration of OH and NO respectively. The concentration of NO is seen to be highest in the high temperature zone and where long residence time of combustion products is expected. The NOx emissions were predicted to be 3.7 g-NO/kg-fuel.

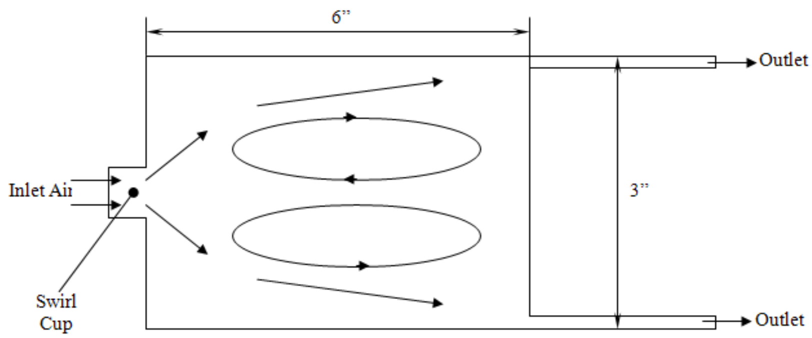

Figure 1: schematic of flow domain used in CFD

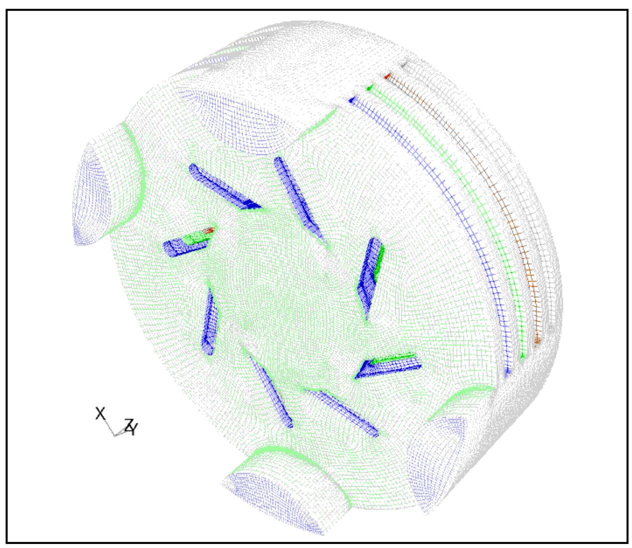

Figure 2: A multi-block grid system for the air passages within the ML air swirler.

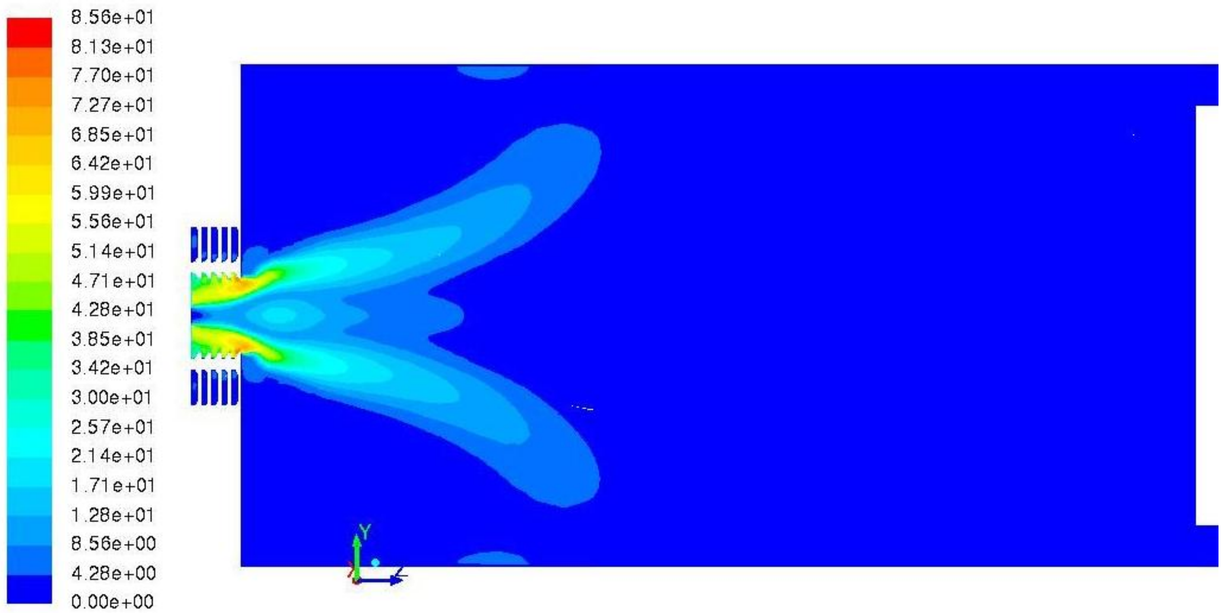

Figure 3: Velocity field of non-reacting flow

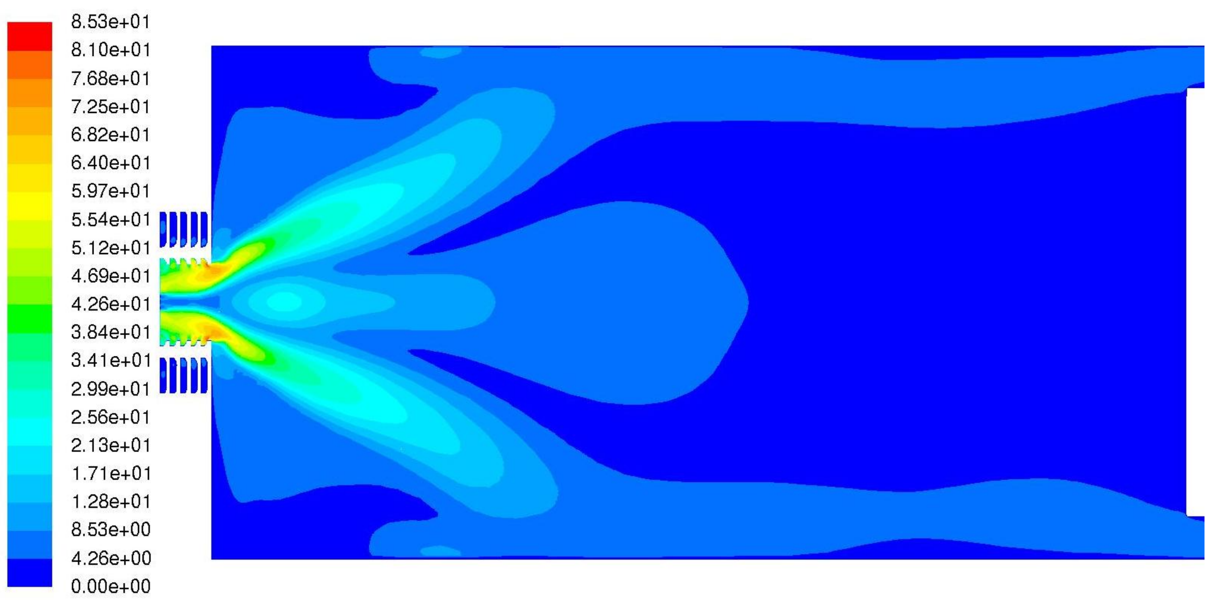

Figure 4: Velocity field of reacting flow

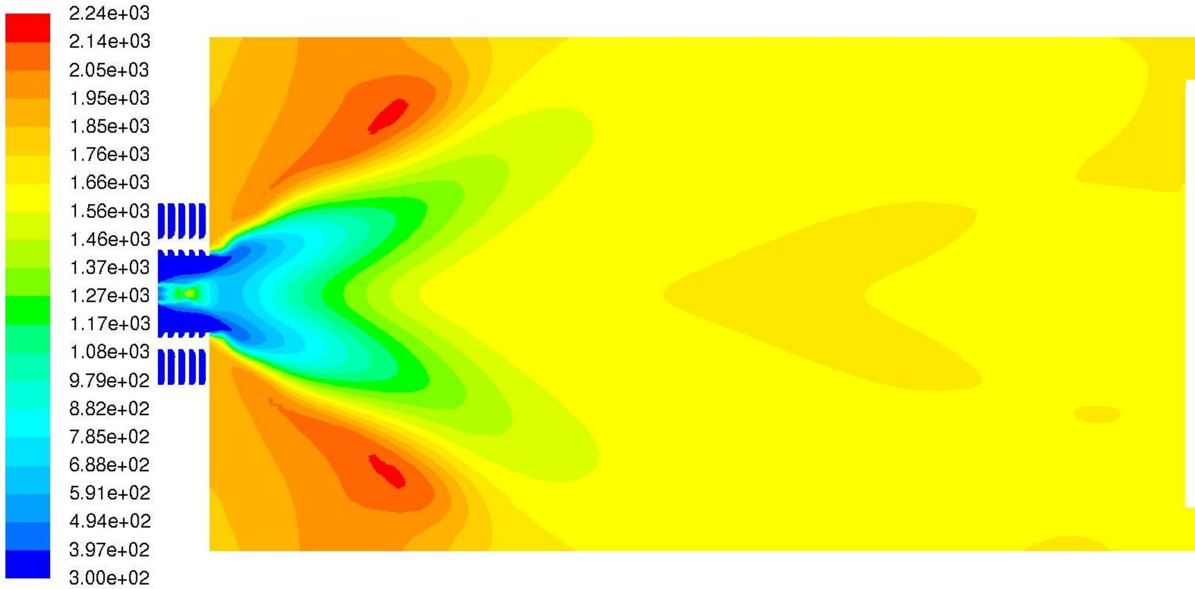

Figure 5: Temperature

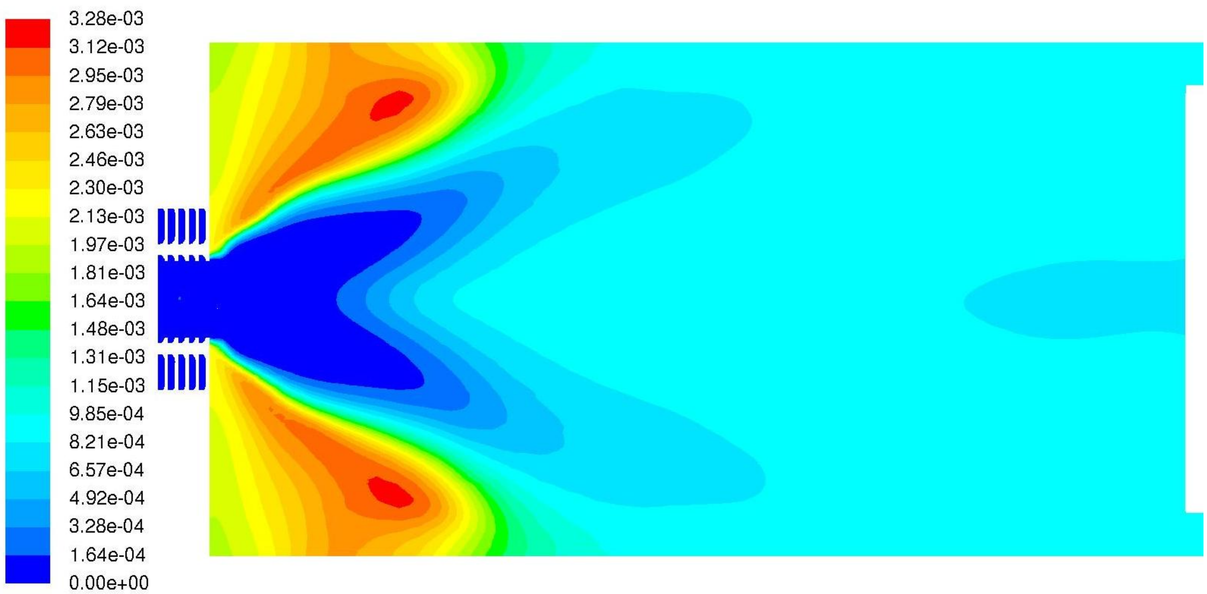

Figure 6: Mass fraction of OH

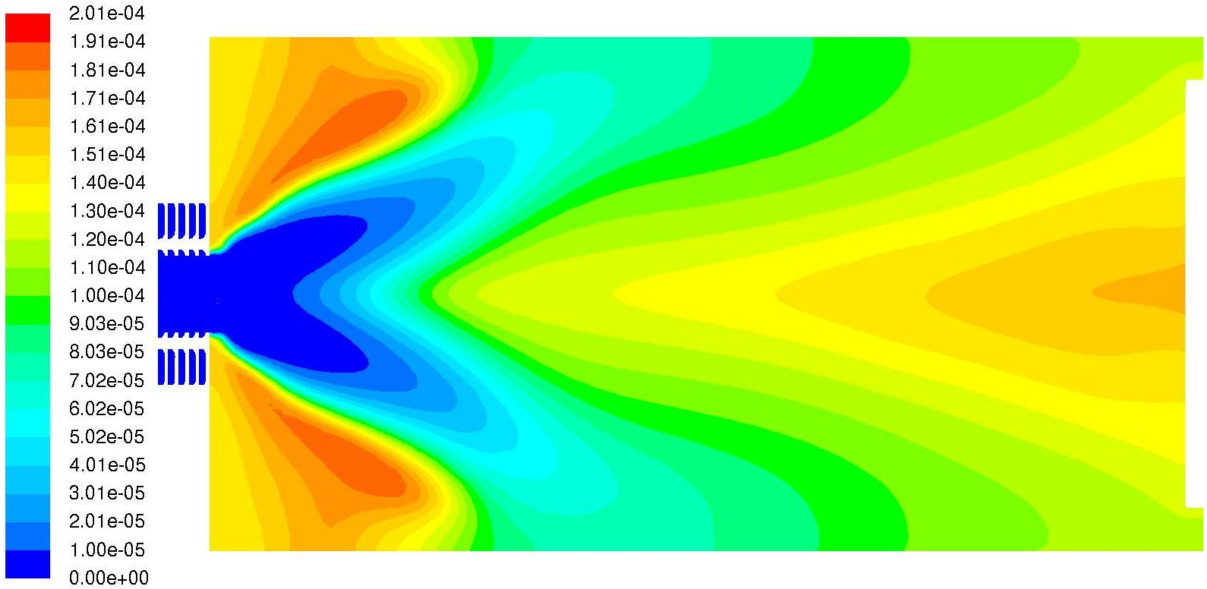

Figure 7: Mass fraction of NO

Acknowledgements

I would like to thank the Southwest Research Institute and Parker Hannifin for giving me this excellent opportunity. I earned a valuable experience regarding CFD analysis. I would specially like to express my gratitude to Dr. Erlendur Steinthorsson for his help and guidance.

References

1. Cai, J., Jeng, S.-M., Steinthorsson,E., "Experimental and Numerical Investigation of a Macro-Laminated Radial Swirler", AIAA 2003-0826.
2. Tacina,R.,Wey,C.,Laing,P.,Mansour,A., "Sector test of a low NOx, lean-direct-injection, multipoint integrated module combustor concept,"ASME Turbo Expo2002.