2011 Gas Turbine Industrial Fellowship

Parker Hannifin Gas Turbine Fuel Systems Division

Christopher J. Paul

Michigan State University
As a fellow of the 2011 University Turbine Systems Research (UTSR) Gas Turbine Industrial Fellowship Program, I spent twelve weeks working at the Gas Turbine Fuel Systems Division of the Parker Hannifin Corporation. The Gas Turbine Fuel Systems Division (GTFSD) of Parker Hannifin is one of the leaders in developing and manufacturing fluid metering, delivery and atomization devices, liquid cooling systems, and aircraft and engine thermal management systems. GTFSD has facilities in Germany, Malaysia, and the states of Arizona, New York, South Carolina, and Washington, with the division headquarters located in Mentor, Ohio.

Upon arrival at the division headquarters, I was assigned to be a part of research and development team, with my mentor being Dr. Erlendur Steinthorsson. Research and development has two primary functions: to develop new technologies and to provide engineering support analyses to product development and manufacturing. While there I was involved in four projects in aircraft fuel delivery systems. These were an analysis of leakage from a valve assembly, failure analysis of a prototype nozzle, a parametric study of airblast atomizers to predict discharge coefficients, and flow field analyses of various spray cup nozzles.

The first project was to analyze a valve cartridge which was exhibiting leakage flows that were impacting the valve assembly during production. Since the current valve flow model was not sufficiently accurate to predict the leakage flow rate, a more sophisticated model was created using CFD analysis. Since the actual geometries, operating conditions, and results are proprietary, they have not been released.

The second project was to investigate and perform a failure analysis of a prototype nozzle. My role was simply to aide Dr. Steinthorsson in evaluating and
postprocessing the CFD results. Since the information is proprietary, it will not be discussed further.

The third project with which I was involved was to perform a parametric study of airblast atomizers to create a tool to predict discharge coefficients. The data for the tool was created by varying geometric parameters and determining the discharge coefficients using 3D CFD analyses. The results of the study are to be validated with empirical data that has yet to be collected. The fourth project was to characterize the flow field of a macro-laminated (ML) spray cup. The geometry representing the fluid domain of the spray cup is shown in Figure 1.

Figure 1
ML Spray Cup

Figure 2 shows the fluid domain geometry including the downstream pressure exit. The shape and size of the downstream geometry were chosen to match the experimental apparatus.
A CFD analysis of this particular ML spray cup was completed in 2003 along with the flow field being measured. See J. Cai et al. The mesh created for the original CFD analysis was based on measurements of the actual parts and had approximately two million hexahedral elements, with edge lengths ranging from 0.000004 m to 0.005 m. This mesh is shown in Figure 3.
The incompressible, isothermal, and isotropic CFD analysis done in Fluent used a Reynolds-averaged Navier-Stokes (RANS) k-ω Shear-Stress Transport (SST) model. Finally, the flow field was measured using Laser Doppler Velocimetry (LDV) and compared to the CFD analysis. Figure 4 shows the mean axial velocity fields in the xz plane, while Figure 5 shows the mean axial velocity fields at $z = 7$ mm downstream of the cup exit.
Mean Axial Velocity at $z = 7\text{mm}$ Downstream

To take advantage of increased computing resources, a new refined mesh was to be used. Using Pro-Engineer models made to match the dimensions of the original part, a new mesh of 49 million tetrahedral elements was created using the ANSYS meshing tool, with edge lengths from 0.000075 m to 0.001 m. Figure 6 shows the new mesh in the spray cup region.

![New Tetrahedral Mesh](image)

Figure 6
New Tetrahedral Mesh

Instead of the RANS $k-\omega$ SST model originally used, a Large Eddy Simulation (LES) turbulence model with a Smagorinsky sub-scale model was to be used. The LES analysis was to be done first using the original hexahedral mesh and then with the new tetrahedral one. Both LES analyses were to be done in OpenFoam. The flow fields in the $xz$ plane and at $z = 7\text{mm}$ downstream are shown in Figures 7 & 8 for the LES analysis using the original hexahedral mesh.
The shape of the flow fields for the LES analysis compares favorably with the LDV measurements from Figures 3 and 4, but the peak velocities differ. This may be due to a difference of how the mean velocities were defined or the inability of the measurement apparatus to pick up the velocities in between probe locations.
Due to time constraints, all of the analyses to be done had not yet been completed. While the LES analysis using the original hexahedral mesh was completed, the LES analysis using the new tetrahedral mesh is still in progress. Additional prototype geometries were to be analyzed as well, with the inclusion of fuel droplets.

I would like to thank the Southwest Research Institute and Parker Hannifin for giving me this excellent opportunity to gain valuable practical experience analyzing fluid flows using CFD. I would specifically like to express my gratitude to Dr. Erlendur Steinthorsson, my mentor, and Dr. Adel Mansour, Research and Development Team Leader, for their help and guidance.
Reference