

# Development of Computational Fluid Dynamics Model for Industrial Gas Turbine Combustor

**Jimmy Robertson**

University of Oklahoma

UTSR Fellowship Participant at Pratt & Whitney

May 30 – August 18, 2006

## **Executive Summary**

---

For my fellowship experience, I had the fortune of being assigned to Pratt & Whitney in East Hartford, CT. My mentor, Albert Veninger, heads the combustor development team for Pratt & Whitney's Power Systems division, which designs and markets industrial gas turbines. During my twelve weeks under Mr. Veninger's guidance, my project consisted of developing and validating a computational fluid dynamics (CFD) model for an existing industrial gas turbine combustor in order to predict results for future designs, thereby establishing the model as a cost-effective design tool. Having a small amount of experience with CFD and combustion, I was excited for the opportunity to learn more and apply my knowledge to an actual gas turbine project.

My time at Pratt & Whitney was spent in a very diverse manner, which gave me the chance to learn about many different aspects of the company, the industry, the business environment, and the full scope of CFD practices. My first few weeks were spent acquainting myself with my coworkers and familiarizing myself with the software tools available to me. Once the solid model for the project was completed by draftsmen and made available to me, I set out to apply my new skills and process the model until it was complete and prepared for simulation. During the final weeks, I closely watched and adjusted the simulation while constantly tracking the results and comparing them to the existing data.

Throughout each stage of my project, I was able to access and consult with numerous experts in the fields of CFD and combustor design, each of whom was a tremendous help to me. I learned much about CFD, combustion, and gas dynamics with their assistance. When I wasn't consumed with the project, I was able to visit and tour various support operations ranging from manufacturing to assembly and repair. These visits were great experiences that allowed me to gain a better understanding of different aspects of engineering in the industry.

In summary, I am tremendously thankful for receiving the opportunity to work with Pratt & Whitney through this fellowship. My days were full and interesting, and the project was continually challenging and rewarding. The people I met at P&W were fantastic, always willing to take some time and help me better understand something or make me feel welcome and valued. I learned much through this fellowship, and not just concerning the subjects related to my project: I have learned that I truly

have a passion for the gas turbine industry and the challenges it provides. As a result of this fellowship, I intend to focus my graduate studies on subjects that apply to gas turbine design with the end goal of working as a development engineer in the industry. I could not have had a better experience, and am truly thankful for the opportunity to participate in this fellowship.

## **Introduction**

---

### ***Project Description***

The research project was intended to augment the design and testing program for an industrial gas turbine combustor by attempting to model the combustor testing rig in a virtual environment. Specifically, the primary goal was to develop a valid virtual rig model using existing test data, and use said model as a design tool to tailor the exit temperature profile of a newly-proposed combustor design to meet or exceed turbine blade durability requirements.

The research was conducted by simulating the flow through a virtual model of the testing rig with computational fluid dynamics (CFD) software, a finite-element analysis method applied to fluid flow simulation. The software package chosen for this project was Fluent, which is considered the industry standard for CFD simulation. The simulation conditions were applied in an identical manner to prior rig tests in order to obtain meaningful results, which were in turn compared to the existing test results to determine the validity of the virtual model. This process included many complicated steps, including the creation of a digital CAD model of the testing rig, preparing the model for finite-element analysis in the CFD software, consultation with experts on the selection of numerical modeling approaches, simulation, post-processing of results, and comparison with actual rig data. After verifying the model against existing test data, the model would be used to predict rig test results for the proposed new design, adjust the design as necessary.

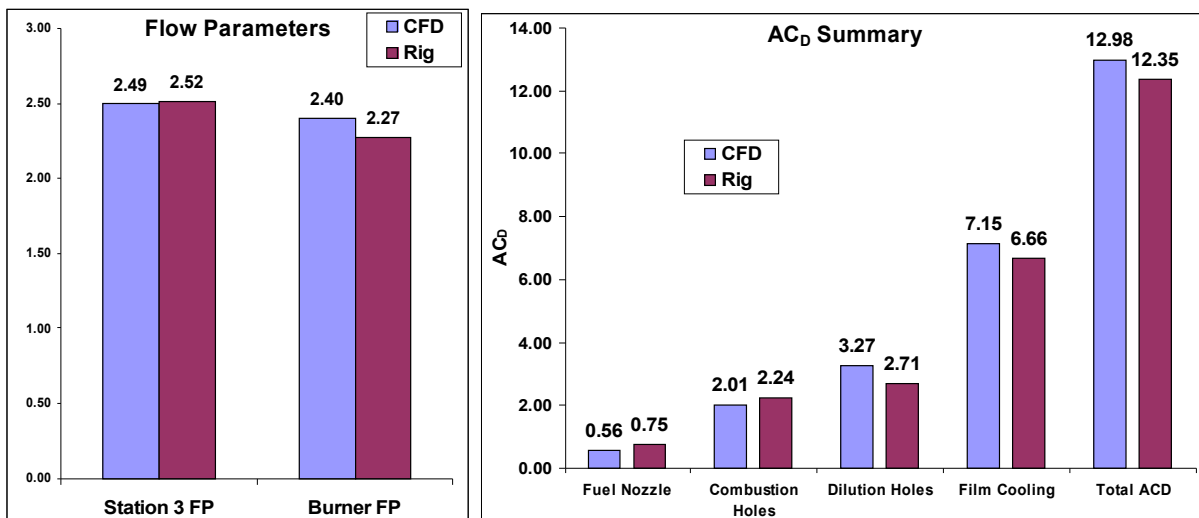
### ***Project Significance & Implications***

Although the initial scope of this research is small, the potential effects of the outcome are far-reaching, and therefore act as motivation to see it succeed. In the short term, the validation of the virtual model could reduce the necessary number of rig test iterations by facilitating the generation of an optimized combustor design that would meet or exceed turbine durability requirements. This would undoubtedly save time, manpower, material, and financial resources. Concurrently, the success of the design program objectives would result in a product with enhanced selling points with regard to durability and emissions. The product's market could then expand, bringing increased sales.

## Results & Discussion

Property	CFD	Rig Data	% error	units
Station 3 Mass Flow	13.506	13.657	-1.1	pps
Combustor Flow	13.047	12.337	5.8	pps
Station 3 Total Pressure	198.5	198.5	0.0	psi
Station 3 Temperature	877	877	0.0	F
Fuel/Air Ratio	0.0195	0.0207	-5.4	

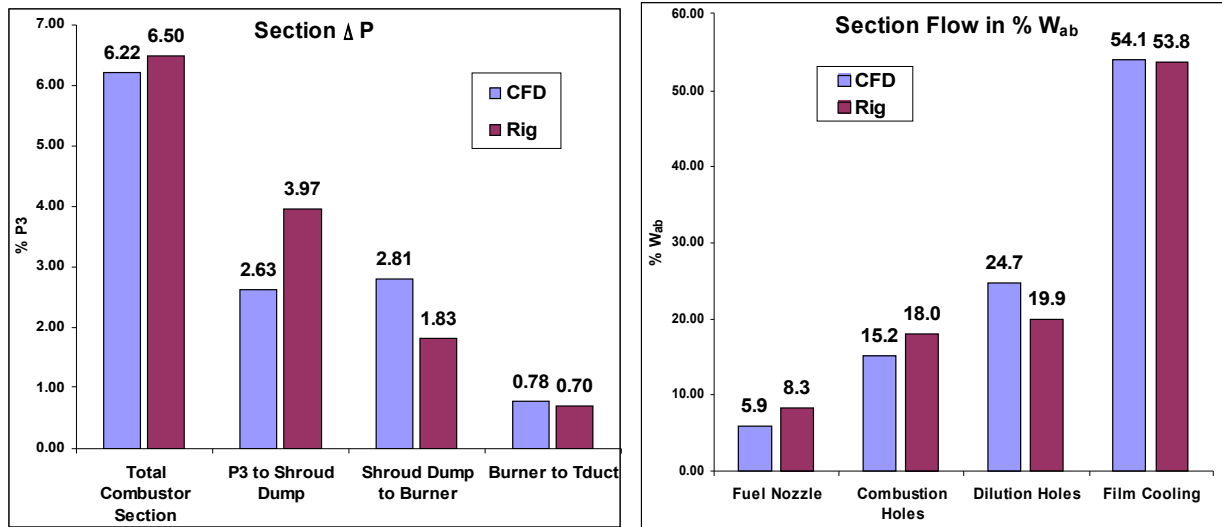
Table 1 contains a summary of the major flow conditions. Station 3 total pressure and temperature were specified in boundary conditions and did not vary; Station 3 mass flow was a targeted boundary condition, but time restraints prevented reaching the required number of iterations to attain the exact rate. However, the difference is only -1.1% from the rig value. Notice that there was nearly a 6% difference in burner air flow, the flow that passes through the burner. It is important to note that, although the CFD measurement is exact, the rig data is not. This is because leakages exist within the rig that cannot be reasonably measured, such as at the interfaces of the nozzle and burner, burner and transition duct, and so on. Since the rig data burner flow is calculated from known orifice  $AC_D$  values and the pressure drops across them, this leaves a discrepancy if leakage flow cannot be measured. These unquantifiable leakages obviously do not exist in the CFD model. It is possible that their exclusion had a significant enough effect on the pressure distribution to prevent the CFD model from developing the correct flow field.



**Figures 1 & 2: Flow Parameters and  $AC_D$  Values**

The bulk flow parameters and  $AC_D$  values indicate that the flow scale was closely matched to that observed in the rig and that the major orifices were modeled relatively well. Although the  $AC_D$

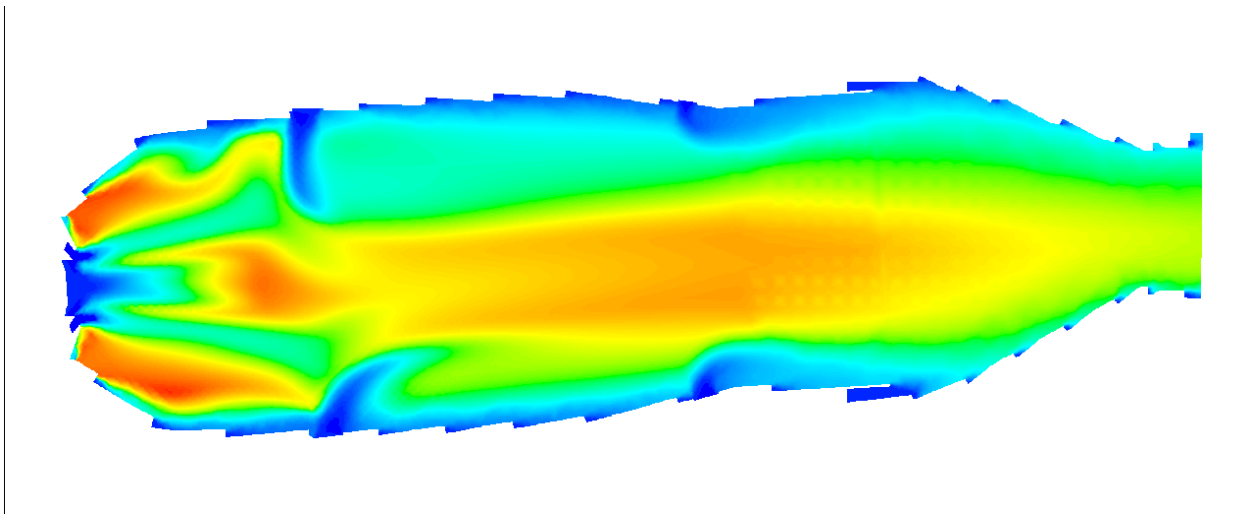
values vary some, they are certainly in the correct range that would be expected. Since  $AC_D$  can sometimes be ambiguously derived (depending on locations of pressure probes and averaging them over large volumes), this amount of variance is not necessarily cause for alarm.



**Figures 3 & 4: Section Pressure Drops and Mass Flows**

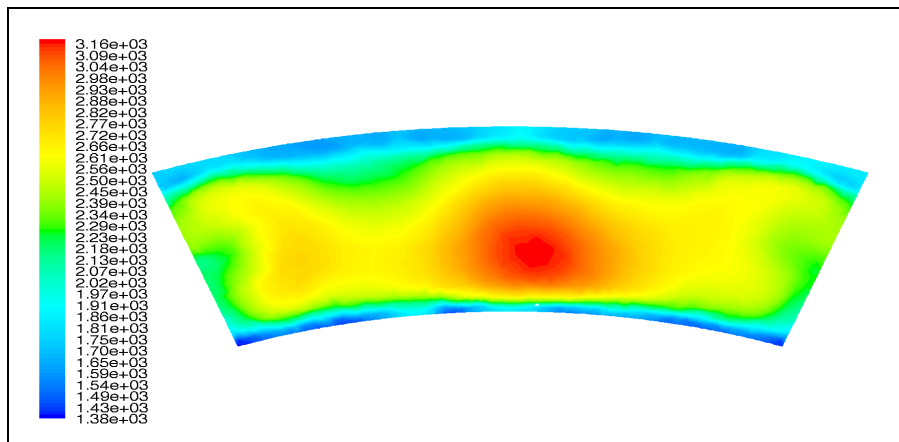
These figures contain section pressure drops and mass flows, scaled to percent of Station 3 total pressure and combustor flow, respectively. Although the overall pressure drop across the combustor section was within five percent of the rig data, the distribution of the total drop did not develop properly. The pressure differential from the combustion to transition zones was in line with expectations, but the same cannot be said for the drop from Station 3 to the diffuser case (shroud dump) region and from that point to inside the combustion zone. This indicates a serious error in the model that has prevented it from replicating the pressure field in the rig data.

There are two likely causes to this problem. First, the model clearly did not force the necessary total pressure drop to produce the correct static pressure in the shroud dump region. With respect to the flow through the combustion, dilution, and cooling holes, this represented a significantly higher upstream pressure, which forced more air through these orifices and resulted in higher jet penetration. By conservation of mass, this reduced the flow through the fuel nozzle, resulting in an inaccurate flow field within the combustion zone. These effects were compounded by the fact that the simulated mass flow rates through the film holes were explicitly defined using the pressure drop across the combustion liner in the rig data, which was significantly lower than that which existed in the model. This resulted in increased upstream static pressure, forcing even more flow through the other orifices. The upsetting effects of the incorrect pressure distribution on the mass flows are also observed in Figure 4.



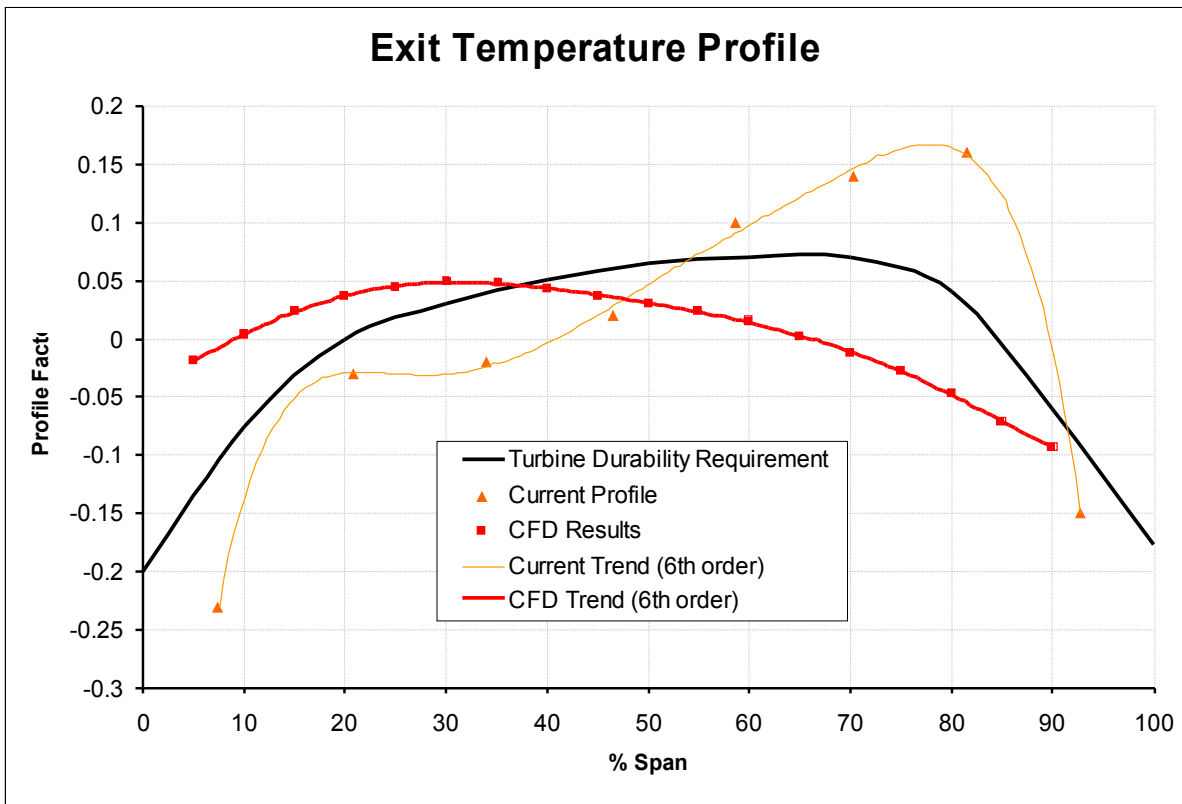
**Figure 5: CFD Temperature Results, Combustion Zone**

This color contour plot depicts the temperature field on a vertical cross-sectional slice through the center of the combustion zone (1300-4300 Rankine); the stream flows from left to right. Note that the trajectory of the post-combustion hot zone band appears to be governed by the penetration of the combustion air jets entering on the top half of Louver #4. It has already been stated that the pressure field was malformed, but even with this considered, the static pressure in the outer diameter region of the shroud dump was higher than those in the inner diameter region, which did not occur in the rig. Thus, the OD orifice jets penetrated further, forcing the hot gases toward the bottom of the combustor.



**Figure 6: CFD Temperature Results, Combustor Exit**

This color plot depicts the temperature map at the exit of the combustor section, looking upstream. This is the location where the exit temperature profile is calculated. The plot mirrors the results observed in Figure 5; the hotter gases were forced toward the inner diameter.



**Figure 7: Exit Temperature Profile**

This plot contains the exit temperature profile data from the rig, the CFD simulation, and the requested profile for turbine durability. The x-axis represents the radial location of the data point in percent of span from the ID to OD surfaces. The temperature profile is calculated by taking an average of the temperature at constant radial locations, and normalizing them against the Station 3 and Station 4 average temperatures using the following equation:

$$P.F = \frac{T_{local} - T_{3,avg}}{T_{4,avg} - T_{3,avg}} - 1$$

The validation of the model hinges on this very plot; the CFD results must reasonably match the rig data. The data verifies what was already assumed: the model is not correct yet. As mentioned, the hotter gasses are pushed toward the inner diameter by the combustion and dilution jets. However, there are many possible improvements that could be applied to the model in order to increase accuracy.

## **Recommendations**

---

As previously stated, the success of this research project was hindered by time constraints. Although the preliminary results did not closely match rig data, there are many avenues that could be taken in attempt to correct this, and would have been if time allowed. Consequently, the highest possible recommendation is to encourage and promote the continuation of this research until completion. The potential rewards are significant compared to the resources required. If this project were to continue, the following actions are recommended in order to improve the model and reduce the sources of uncertainty

### ***Primary Recommendations***

1. Obtain and utilize the most accurate rig solid model as possible.
2. Attempt to identify any leakage points in the rig that could lead to unaccounted-for mass flow, and place them in the solid model so that mass flow boundary conditions can be applied to them.
3. Check that all designated cooling simulation surfaces in the simplified model accurately correspond to the rig configuration.
4. Make an effort to obtain or develop a dynamic cooling model that would update the simulated cooling flow as a function of the orifice ACD and pressure drop. This sort of custom model is called a “User-Defined Function” (UDF) in Fluent, and must be scripted in C language to link with and adjust the calculations.
5. Obtain more reliable rig testing data for setting boundary conditions and checking model validation. An ideal data set would possess a full instrumentation map, fully accounted-for mass flows, and accurate temperature profile measurements.
6. Place artificial geometry in the diffuser case region, such as baffles, that would produce the correct diffuser case pressure without affecting the burner can and transition duct.

Additionally, the following actions could be taken to address lesser sources of uncertainty; actions that might improve the accuracy of the model but would likely not result in the more significant differences observed between the rig and CFD model data

## ***Secondary Recommendations***

1. Remove less reasonable simplifications from the solid model; attempt to model the rig more accurately. Alternatively, closely examine the rig for geometry not represented in the un-simplified solid model.
2. Obtain more details on the nature of the water injection and apply them to the corresponding discrete phase model properties in Fluent.
3. Adjust the fuel properties to those for the exact natural gas mixture used in the rig tests. This would require research into corrected specific heat polynomial coefficients for the lesser species in the mix, or temperatures will be over-estimated in the results.
4. Refine the finite element mesh. That is, add more elements to better capture the flow. This could result in a higher computing resources requirement and/or longer calculation time.
5. Fully resolve the film cooling passages with mesh elements. This could easily double or triple the necessary computing resources, but would certainly help the model to develop the proper flow and pressure distribution.
6. Apply an enhanced wall treatment model in Fluent to better resolve boundary layer flow and its effects on the bulk stream.
7. Adjust the turbulence model settings, or use a different turbulence model altogether.
8. Consult experienced CFD users and obtain recommendations to improve the model.