

**COMPUTATIONAL FLUID DYNAMICS (CFD)
APPLICATIONS IN ROCKET PROPULSION ANALYSIS AND DESIGN**
P. K. McConnaughey, R. Garcia, L. W. Griffin, J. H. Ruf
NASA George C. Marshall Space Flight Center

ABSTRACT

CFD has been used in recent applications to affect subcomponent designs in liquid propulsion rocket engines. This paper elucidates three such applications for turbine stage, pump stage, and combustor chamber geometries. Details of these applications include the development of a high turning airfoil for a gas generator (GG) powered liquid oxygen (LOX) turbopump single-stage turbine using CFD as an integral part of the design process. CFD application to pump stage design has emphasized analysis of inducers, impellers, and diffuser/volute sections. Improvements in pump stage impeller discharge flow uniformity have been seen through CFD optimization on coarse grid models. In the area of combustor design, recent CFD analysis of a film cooled ablating combustion chamber has been used to quantify the interaction between film cooling rate, chamber wall contraction angle, and geometry and their effects of these quantities on local wall temperature. The results are currently guiding combustion chamber design and coolant flow rate for an upcoming subcomponent test. Critical aspects of successful integration of CFD into the design cycle includes a close-coupling of CFD and design organizations, quick turnaround of parametric analyses once a baseline CFD benchmark has been established, and the use of CFD methodology and approaches that address pertinent design issues. In this latter area, some problem details can be simplified while retaining key physical aspects to maintain analytical integrity.

INTRODUCTION

A crucial challenge for CFD has been the application and integration of CFD analysis methods into the rocket propulsion system design process. Several factors encourage intensive analysis in support of the engine design process (e.g., the high cost of testing). Historically, geometric and flow process complexity has precluded efficient and timely analysis to support the design process. This paper briefly highlights three applications that advanced the design process and hardware concept through judicious application of CFD codes. These three applications include turbine stage design, impeller design optimization, and parametric assessment of a low cost ablative combustion chamber design. Discussion of these CFD design applications will be followed by a summary of analytical guidelines that allow for integration of CFD into the design process, and an assessment of future needs to further mature CFD for rocket engine design applications.

TURBINE STAGE DESIGN

Griffin and Huber (ref. 1) summarize advancements in the turbine design process that include the application of a range of CFD codes to both the fuel and LOX turbine stages of a GG engine. Huber, et al, (ref. 2) gives details of

the design process and how integration of several codes, approaches, and organizations contributed to the design of the LOX turbine for the Space Transportation Main Engine (STME). Coordination of these activities was through the CFD Consortium for Applications in Propulsion Technology (CAPT) as discussed by McConnaughey and Schutzenhofer (ref. 3), which allowed for efficient technology transfer from NASA and industry research efforts to a hardware development program.

Results of applying CFD codes to the fuel turbine (denoted G³T) as compared to conventional design methods can be seen in table 1. To achieve this increase in efficiency and decrease in blade count, airfoil camber was increased from 140 to 160 degrees. Analytical refinements to a preliminary design that led to such a highly loaded blade were made through the application of a multi-stage Euler analysis with a surface drag force model, an inverse design code for blade leading edge definition, full Navier-Stokes analysis of both the stator and nozzles (decoupled steady), and two-dimensional (2D) unsteady Navier-Stokes analysis of the first stage. It is with this latter analysis that the axial gap was adjusted to preclude a predicted unsteady shock between the stator and rotor. Optimal axial gaps based on unsteady CFD analysis led to a 24 percent decrease in blade dynamic and loading and a predicted efficiency gain of 1 percent relative to the preliminary design (ref. 1).

The development of the G³T turbine showed how CFD could be used to extend the design envelope of subsonic turbines used in GG engines, and these lessons and hardware design concepts were then applied to the STME LOX turbine. This turbine stage preliminary design was predicted to have an efficiency increase of two points relative to conventional meanline design methods, and further refinement of the design was a result of CFD analysis. This analysis includes multi-stage Euler analysis, steady three-dimensional (3D) Navier-Stokes analysis, and 2D unsteady Navier-Stokes analysis. As in the G³T turbine, unsteady stage analysis was used to adjust axial spacing of the stator and rotor, and both the modified Euler and 3D Navier-Stokes analysis predicted a flow separation downstream of the rotor that was removed by changing blade lean until the analysis predicted elimination of the separation region (refs. 1 and 2). A grid used for inviscid analysis of the design is seen in figure 1.

To summarize the impact of CFD on turbine stage design for rocket propulsion systems, the incorporation of Euler and both steady and unsteady Navier-Stokes analyses have led to advanced designs that extended the design envelope with increased performance and turning angle such that turbine part count and stage number are decreased.

PUMP STAGE DESIGN

CFD analysis of the STME fuel pump impeller has led to comparable advances in both hardware concepts and design methodology comparable to the turbine stage design. Prior to this design effort, a significant effort in benchmarking CFD codes for inducer and impeller flows was completed, and continues to be in progress. As seen in Garcia, et al, (refs. 4 and 5), a preliminary design of the STME fuel impeller was completed using current design methodology,

and then optimized using 3D Navier-Stokes analysis. The impeller was designed to minimize exit flow distortion, and hence, unsteady diffuser loads while maintaining or increasing impeller performance. Variation in impeller exit flow velocities for a range of CFD code predictions can be seen in figure 2. This optimized impeller design was then used to evaluate non-standard impeller design changes that would lead to further reduction in flow distortion and enhanced stage performance. Design changes evaluated included tandem (or offset) bladed impellers, length and location of partial blades, the use of pressure side/suction bleed holes at the shroud, and significant blade lean that limits the flow overturning that contributes to the exit distortion level. The latter design incorporating a change of blade lean was seen to be most effective in reducing impeller exit flow distortion.

The net result of this CFD-based pump stage design activity (also through the CFD CAPT) has been the development of advanced impeller hardware concepts and the integration on CFD into the pump stage design cycle. Even though computational grids may be coarse by some standards, both inviscid and viscous CFD analyses are routinely used to support inducer and impeller design for rocket engines at the major manufacturers and design organizations. It is in the use of benchmarked codes for parametric design assessment that CFD has and will continue to play a key role in advanced pump stage design.

COMBUSTION CHAMBER DESIGN

A recent example of CFD used to support combustion chamber design is assessment of both injection coolant schemes and chamber geometry for an ablative insert chamber design. This combustion chamber design is associated with the Advanced Technology Low Cost Engine (ATLCE) where parametric assessment of coolant injection mass, injection velocity, and chamber geometry and their effect on wall temperature was completed using a Navier-Stokes code. In this advanced engine, simplicity is maximized and cost is minimized through the use of radiatively cooled composites and ablative liner materials. New materials (such as silica phenolics) have lower ablation rates, but material behavior and properties at higher temperatures are not well characterized. Thus, the objectives of this analysis were to support a preliminary design for a hot-fire chamber characterization test while minimizing risk through lowering wall temperatures to below a known acceptable value (3800 degrees R). The code used in this study was previously benchmarked for coolant jet injection heat transfer (ref. 6), and a similar approach was used for this design application with the exception of some turbulence model modifications. The chamber geometries considered are seen in figure 3, with the range of parameters for injection and predicted maximum wall temperatures seen in table 2. A Taguchi approach was used in the parametric assessment, and the 'best' design was predicted to be the constant contraction chamber geometry with a 'low' coolant injection velocity and a coolant mass flow rate of 3.6 lbm/sec. Parametric analysis indicates this flow rate could be reduced to 2.6 lbm/sec while providing some margin relative to the chosen material limit. These results were provided to the designer to then complete the combustion chamber design from a system perspective. It is expected that the chamber will be fabricated and tested at NASA/Marshall Space Flight Center in the near future.

SUMMARY AND FUTURE NEEDS

Key aspects in these analyses that allowed for timely completion of design parametrics include the use of benchmarked Navier-Stokes codes for the chosen range of applications, close coupling between the design group and CFD analyst, and an understanding of key physical aspects of the flow problem. The latter point allows for the CFD analyst to assess important design issues and not overcomplicate the CFD analysis. Continued development and maturation of this philosophy should lead to further integration of CFD methods into subcomponent design organizations.

Future requirements for CFD methodology and applications development to support design integration include the following: (1) The use of unstructured grids and/or expanded multiblock interface capabilities (e.g., chimera or patched) to address more complicated domains, (2) increasingly robust and efficient solution algorithms, (3) solution algorithms that are efficient and port to either moderate or massively parallel computer architectures, (4) design optimization techniques, (5) an assessment of the level of physical modeling necessary for a given design application, and (6) coupling of various discipline codes to support integrated engineering analysis. It is in this latter area that the unstructured approach allows for flexibility, albeit with some cost in code efficiency.

REFERENCES

1. Griffin, L. W., and F. W. Huber, "Advancement of Turbine Aerodynamics Design Techniques," ASME Paper No. 93-GT-370.
2. Huber, F. W., P. D. Johnson, X. A. Montesdeoca, R. J. Rowey, and L. W. Griffin, "Design of Advanced Turbopump Drive Turbines for National Launch System Application," AIAA Paper No. 92-322.
3. McConnaughey, P. K., and L. A. Schutzenhofer, "An Overview of the NASA/Marshall Space Flight Center Consortium for Applications in Propulsion Technology," AIAA Paper No. 92-3219.
4. Garcia, R., E. D. Jackson, and L. A. Schutzenhofer, "A Summary of the Activities of the NASA/Marshall Space Flight Center Pump Stage Technology Team," Proceedings of the 4th International Symposium on Transport Phenomena and Dynamics in Rotating Machinery.
5. Garcia, R., P. McConnaughey, and A. Eastland, "Activities of the Marshall Space Flight Center Pump Stage Technology Team," AIAA Paper No. 92-3222.
6. Tucker, P. K. and M. Croteau-Gillespie, "Combustion Devices Technology Team: An Overview and Status of STME-Related Activities," AIAA Paper No. 92-3224.

Table 1 - Comparison of Conventional Design and G³T (as predicted by meanline analysis)

	Conventional	G ³ T
No. of Stages	2	2
Work Split	70/30	50/50
Blade Turning	2.36 rad	2.79 rad
Max. Blade Mach No.	1.32	0.87
Efficiency	base	+9.8%
Airfoil Count	base	-55%

Table 2 - Maximum Temperature (degrees R) of ATLCE Combustion Chamber Wall from Parametric Study

	Geometry 1		Geometry 2		Geometry 3	
	low	high	low	high	low	high
Boundary Layer Injection Vel.						
Mass Injection Rate						
1.0 lbm/sec	5400	5500	5200	-	-	-
2.6 lbm/sec	4000	-	3540	-	-	-
3.6 lbm/sec.	3400	3900	2950	-	3550	-

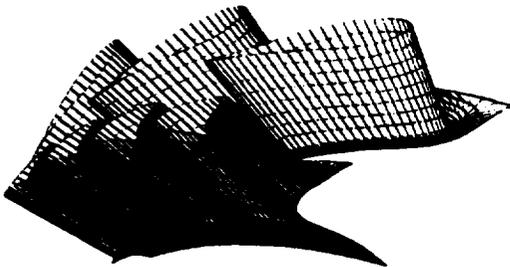


Figure 1 - STME LOX Turbine Geometry and Grid (Ref. 1)

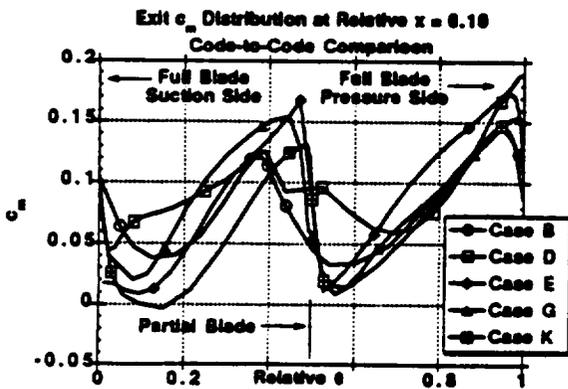


Figure 3 - STME Impeller Blade-to-Blade Exit C_m Distribution Near Shroud ($x = 0.10$) (Ref. 5)



Geometry 1



Geometry 2



Geometry 3

Figure 2 - Geometries Analyzed for Ablative Combustion Chamber Design