CFD Analysis and Optimization of Flow Deflector Geometry for a Supersonic Free Jet

Team 58 Project Technical Presentation to the 2017 IREC

Ross B. Alexander
Texas A&M University, College Station, Texas, 77840

INTRODUCTION

At the Texas A&M University Riverside Campus, the Dwight Look College of Engineering and the Department of Aerospace Engineering have established a testing environment for use by both current undergraduate students enrolled in capstone courses in the aerospace engineering curriculum and also current students involved in extracurricular organizations affiliated with the Department of Aerospace Engineering. The Texas A&M University Sounding Rocketry Team (SRT) has utilized this facility extensively to test commercial off-the-shelf (COTS) solid rocket engines and student-researched and -developed (SRAD) hybrid rocket engines. Currently, SRT operates their LN-350 Helios and NP-915 Icarus hybrid rocket engines in the Riverside Test Cell (RTC). The RTC is equipped with cameras and data acquisition (DAQ) devices, including pressure transducers, thermocouples, and load cells, which have recorded data for 8 successful engine hot fires in the past year. After several tests, SRT determined that the deflector plate geometry should be optimized in a computational fluid dynamic (CFD) study focused on the efficient deflection of exhaust gases out of the RTC.

MOTIVATION

The RTC is a comprised of three structures: the cell enclosure, adjacent cell storage, and vertical test stand. The vertical test stand is located inside the cell enclosure and the adjacent cell storage houses DAQ equipment, electrical connections, and several gas cylinders. For a standard engine test, the engine is secured to the vertical test stand through multiple vertical struts, axial loading data is recorded by three load cells at the top of the test stand, and nozzle exhaust gases exit downward until the flow impinges on an angled deflector plate. The existing deflector plate was a 36.0 in x 22.0 in x 0.25 in flat carbon steel plate angled at 47.45° with respect to the vertical – designed to evacuate exhaust gases through a square steel tube passing through the side of the cell enclosure.

During a standard engine test, temperatures inside the cell enclosure reach more than 5,000 °F – this extreme temperature exposure has the potential to cause significant damage to structural components of the vertical test stand and to DAQ components in the test cell enclosure. On May 10th, 2016, the flame of the NP-915 Icarus engine melted through the entire deflector plate less than halfway through the predicted burn time, causing significant damage to the test cell enclosure and DAQ components.

In order to continue testing NP-915 Icarus in the RTC, SRT reached two conclusions: research should be completed to select: 1) an appropriate deflector plate design and 2) an appropriate material for construction. Both of these goals remained focused on efficiently evacuating the enclosure of intense residual temperature in the exhaust gas. Since relatively little research has been performed on deflector plate design, it was determined that a CFD study would be performed in STAR-CCM+ to evaluate the optimal geometry of the deflector plate. Additional research was performed to determine the optimal material for construction but is not discussed in this report.

CFD STUDY: INITIAL FLOWFIELD ANALYSIS

An initial understanding of the geometry of the flow simulation is necessary to identify flowfield concerns. The dominant flow concern in the flowfield is the supersonic free jet exiting the nozzle and impinging on the deflector plate. The secondary flow concern in the flowfield is the entrainment of the quiescent air in the cell enclosure prior to and during flow impingement on the deflector plate, which generates circulation. Since the goal was to improve the evacuation of exhaust gases, mass flow rate through the vent was used as a comparative measure to quantify the effectiveness of a given deflector plate design.

CFD STUDY: PRE-PROCESSING

The selected flowfield simulation domain includes the test enclosure, engine nozzle, deflector plate, and vent. Principally, there were three testing cases considered in the CFD study – a control case with the existing deflector geometry and two test cases for potential implementation in the testing enclosure. The control case was a 36.0 in x 22.0 in x 0.25 in flat plate angled at 47.45° with respect to the vertical. The first of two test cases was a 16.1 in x 22.0 in x 0.25 in flat plate angled at 20° with respect to the vertical leading into a uniformly curved plate with a radius of curvature of 5.85 in leading into an 11.7 x 22.0 in x 0.25 in horizontal flat plate. The second of two test cases was a 14.4 in x 22.0 in x 0.25 in flat plate angled at 40° with respect to the vertical leading into a uniformly curved plate with a radius of curvature of 14.27 in leading into a 13.0 x 22.0 in x 0.25 in flat plate angled at 19.35° with respect to the horizontal. Each of the test cases led to an exhaust manifold located at the end of the deflector plate that focused the flow into the vent. The thickness of all plates was 0.25 in.

The physics continua selected were a standard, steady, three-dimensional flow, assuming ideal air, coupled energy, and coupled flow. Reference conditions for the region were standard atmospheric conditions: a pressure of 101.325 kPa, a temperature of 300 K, and a density of 1.0 kg/m^3. The walls of the domain were modeled with an adiabatic, no-slip condition and each wall type was prescribed a specified surface roughness. For the test enclosure, vent, and deflector plate, surface roughness was 100 μm; for the tile floor, surface roughness was 10 μm; for the graphite nozzle, surface roughness was 1 μm. For the converging-diverging nozzle, a reduction was made to stagnation conditions of 335.0 psia and 5223.0 °F. The flow solver was assumed to generate an accurate flow profile at the nozzle exit. The vent was set as an environmental-extrapolated pressure outlet at reference conditions.

The selected meshing method was an unstructured, polyhedral mesh with prism layers and an automated surface remesher. The base mesh size was 0.5 in with a surface growth rate of 1.3. 10 prism layers with a prism layer stretching factor of 1.5 were generated spanning 0.5 in. Final mesh cell counts were on the order of 1.3 million cells.

CFD STUDY: PROCESSING

The selected solver equations were the Reynolds-Averaged Navier-Stokes (RANS) equations, with closure provided by the two-equation realizable k-epsilon (k-ε) turbulence model with an underrelaxation factor of 0.8. Additionally, the coupled inviscid flux function selected was the Advective Upstream Splitting Method (AUSM) with Flux Volume Splitting (FVS) with second-order discretization scheme. The study was configured with the coupled implicit solver and a Courant-Friedrichs-Lewy (CFL) condition of 2.0 for 5,000 iterations and a CFL condition of 5.0 for 5,000 additional iterations. For all cases, the solver setup was identical and 10,000 iterations were performed.

The computational power was provided by the Texas A&M University Division of Research High Performance Research Computing (HPRC). Approximately 2,000 hours of wall-clock time was used on the Ada and Terra clusters.

RESULTS AND DISCUSSION

Simulations were determined to be converged after 10,000 iterations. Pressure and velocity scalar field profiles appeared to be accurate and all simulations demonstrated strong residual convergence. In all cases, mass flow rate was used as a comparative metric as a measure for the effectiveness of the deflector plate design. The mass flow rates for the control case, first test case, and second test case were 24.5 lb/s, 31.0 lb/s, and 26.2 lb/s, respectively. For a majority of the 10,000 iterations, the first test case mass flow rate was larger than the other two cases. The results of the simulation control case indeed confirmed the existence of the design problem and both test cases showed improvement over the control case. The first test configuration was determined to be the most effective design for flow deflection.

CONCLUSION AND FURTHER WORK

In mid-April 2017, the first test case design was implemented into the RTC using a combination of 0.5 in thick steel plates and refractory cement which provides improved thermal resistance[1] at the point of flow impingement. The design was successfully verified on April 23rd, 2017 with a standard engine test of NP-915 Icarus in the RTC. The deflector plate withstood mechanical and thermal loads from the flow impingement and a marked improvement was observed for the flow exiting the cell enclosure. In large part, the implementation of the modified deflector plate design enabled safe testing and validation of the hybrid engine SRT will use in the Spaceport America Cup in June 2017. Despite this success, there is significant work to be done in deflector plate design documentation for vertical engine test stands – in particular, high-enthalpy flow-mixing configurations. While NASA and others have documented their methodologies, there are few studies focused on supersonic free jet impingement on various deflector plate geometries and the effect on the flowfield. Correlation studies should be performed with experimental data and CFD data[2] to verify the accuracy of the results.

REFERENCES

[1]Calle, L. M. et al., "Refractory Materials for Flame Deflector Protection System Corrosion Control: Similar Industries and/or Launch Facilities Survey," NASA/TM-2013-217910, 2009, pp. 27.

[2]Garcia, R. G., "CFD Simulation of Flow Fields Associated with High Speed Jet Impingement on Deflectors," 2007.