

ABSTRACT

The performance evaluation and detailed flow field characterization of an aerodynamic vehicle are typically achieved using combined experimental and computational approaches. Experimental data are utilized as a foundation for aerodynamic databases and as a validation for numerical results. Meanwhile, numerical simulations can provide important supplementary data that is otherwise difficult or expensive to obtain experimentally. Therefore, it is important to understand and utilize both approaches to develop a next-generation aircraft or projectile. For air vehicle development, slender body aerodynamics play an enormous role, as most aerospace systems are fundamentally designed around slender body geometry, including launch vehicles, missile systems, and jet aircraft. However, these vehicles in real-world applications also include a variety of aerodynamic surfaces that cause additional features in the flow field and lead to complex flow interactions. In addition to this, these flow features and interactions vary significantly across a large Mach regime, leading to varying aerodynamic performance.

In the present investigation, a series of test campaigns were undergone to evaluate the aerodynamic performance of a generic projectile configuration and characterize its flow field at supersonic speeds. Various test diagnostics were utilized, including force and moment measurements and optical techniques. These experimental results were combined with high-fidelity computational fluid dynamics (CFD) to facilitate a better understanding of the flow field. The performance was evaluated over an extended Mach regime (0.4 to 4.0) in which the effects of compressibility on the aerodynamic coefficients were observed. A significant variation in force and moment characteristics was observed at supersonic speeds, indicating a reduction in vortex influence on the aerodynamics. CFD results agreed well with experimental measurements and, therefore, were utilized to analyze the flow field in detail. The results showed a decrease in vortex-induced low-pressure distributions on the various projectile components. The variations in surface pressure distribution were shown to be largely a function of relative vortex strength decrease and less dependent on vortex size and location, which remained relatively consistent across the Mach regime. The decrease in relative strength with increase in Mach number was shown to be primarily driven by the expansion of previously compressed flow as it rolls around the fin edges and into a vortex. Misalignment in density and pressure gradients also contributed by slightly impeding the vorticity generation at high supersonic speeds as well.

For further analysis of the supersonic flow field, advanced experimental techniques, including pressure-sensitive paint (PSP) and stereo particle image velocimetry (SPIV), were utilized to experimentally observe the variation of the vortex flow field and quantify the effects of vortex interactions at supersonic speeds. The PSP results were investigated in conjunction with other optical techniques, including oil flow and shadowgraph visualization, to validate its ability to resolve the effects of the various flow features that are known to occur. The measurements were able to observe effects of the leeward fin shock wave in relation to the primary body separation. The effects of both primary and secondary vortex structures on the surface pressure distributions were able to be observed in further detail as well. The increased strength of both the windward and leeward fin vortices were shown to impose pressure variation on both the fins and the cylinder body at 12° . At a lower angle of attack, 6° , the same vortices only had notable effects on the fins themselves.

To further investigate the variation in vortex influence, off-surface flow measurements were made on the leeward side of the model using SPIV. The results showed that the vortices on the top of the model were significantly smaller in size at 6° than at 12° , creating a notably different flow field. While distinctive vortex footprints were made on the fins at each angle of attack, the larger vortices at 12° entrained fluid and imposed velocity variation across the entire top surface of the model, compared to only the localized effect of the vortices at 6° . The location of the vortices in relation to the fin edges was largely unaffected by angle of attack, but the distance from the fin increased as the vortices were lifted further away from the fin surfaces at 12° . Despite this, the vortices imposed lower pressure values at the higher angle of attack as the relative strength increased significantly.

PSP and SPIV measurements were lastly combined and utilized to conduct a detailed evaluation of the prediction of the flow field at 12° . The comparison was made with the numerical tools designed for the computational database generation of this configuration from previous works. Surface pressure results showed quality agreement between experimental and computation results from regions of low pressure gradient, but notable discrepancies were shown to exist at the fin shock wave and for the locations of separation and vortex formation. Regarding the fin vortices, it was shown that the CFD underpredicted vortex influence on the model, including on both the fins themselves and on the cylinder surfaces. PIV results corroborated the PSP results in regard to the leeward vortices on top of the model, as the peak normalized vorticity was lower as predicted by the CFD compared to the experimental results. The experimentally measured vortices also had a larger overall size within the flow field, which led to an increased influence on the rear end of the model cylinder. Lastly, the vertical location of the vortices was well predicted, but variation existed in the location

in relation to distance from the model centerline. This work demonstrated that, while the computational tools could do an acceptable job in predicting overall integrated forces and moments, some noteworthy shortcomings exist in the prediction of certain flow features. A better understanding of how the trade-offs made in the prediction tool selection process play a role in the prediction of flow physics, will have a significant impact on future aerospace development. The experimental work presented here also demonstrated advancements in the application of various techniques and has already contributed to the successful testing of other geometries using lessons learned.