OPEN ACCESS Freely available online

Journal of Hotel and Business Management

Short Communication

Aerodynamic Design and Computational Analysis of Yaw Sphere for Subsonic Wind Tunnel Calibration

Akhila Rupesh

Lovely Professional University, India

Abstract

Flow analysis is considered to be the most crucial procedure in aerodynamics. Analysis of flow and its parameters over any object has to be done with considering aerodynamic loading acting over it. In the field of aerodynamics, wind tunnel test setup is used for flow analysis. The wind tunnel test section should always afford a laminar and uniform flow to provide exact results during flow parameter determination. But attaining cent percent laminar flow inside a wind tunnel test section is practically not possible. Hence there is an immense requirement of performing calibration before starting any research experiments in a wind tunnel. It is to be noted that wind tunnel calibration is done with ultimate care to avoid any error in the analysis. Generally a pitot-static probe is used to calibrate subsonic wind tunnel. But the pitot –static tube has many limitations like single point data sensing. Evaluation of drag coefficient often requires wind tunnel experiments and can be prohibitively expensive if not impossible for large objects or systems. Computational Fluid Dynamics (CFD) aerodynamic analysis offers an alternative approach and can be used as a very effective design tool in many industries: automotive, aerospace, marine, etc. The main objective of this research is to investigate feasibility of using non-contact digitizers for developing finite element models of large objects for subsequent CFD analysis. The developed methodology is applied to investigation of time-trial bicycle rider efficiency. Companies competing in this class of racing spend millions trying to optimize bicycle and rider geometry in order to reduce aerodynamic drag. This project investigates an alternative way to optimize the aerodynamic efficiency of the rider, considering the rider contributes the majority of the drag force of the rider-bicycle system. If small riding position adjustments could be made to the rider's body during a race, drag may be significantly reduced. This idea, and the fact that the direction of wind impacting the rider can vary, influenced the concept of this project. It was hypothesized that adjusting the time-trial handlebars on the bicycle to stagger the fore-aft position of the rider's hands would influence the upper body to rotate iii slightly. This could then reduce the frontal area of the rider in the wind direction, therefore reducing the aerodynamic drag. To simulate this situation, the Konica Minolta VIVID 910 non-contact 3-D digitizer was used to scan two separate riders, each aboard a different bicycle, in several positions, as described above. The 3-D scans were then imported into the CFD software package Star-CCM+ and several simulations were run using each of the two rider-bicycle models. The initial simulations seemed to support the theory as the asymmetrical riding position experienced decreased drag at significant wind yaw angles while the normal riding position did not. A second study, using a different rider and bicycle, yielded less conclusive results. The two studies represent the groundwork for similar large system CFD analysis and provide useful recommendations for continued research into bicycle rider aerodynamics. With the fast-paced growth of computational horsepower and its affordability, computational fluid dynamics (CFD) has been rapidly evolving as a popular and effective tool for aerodynamic design and analysis in the automotive industry. In the real world, a road vehicle is subject to varying wind and operating conditions that affect its aerodynamic characteristics, and are difficult to reproduce in a traditional wind tunnel. CFD has the potential of becoming a cost-effective way of achieving this, through the application of different boundary conditions. Additionally, one can view wind tunnel testing, be it a fixed-floor or rolling road tunnel, as a physical simulation of actual on-road driving. The use of on-road track testing, and static-floor, and rolling-road wind tunnel measurements are common practices in industry. Subsequently, we investigated the influences of these test conditions and the related boundary conditions on the predictions of the aerodynamic characteristics of the flow field around a vehicle using CFD. A detailed full-scale model of Hyundai Veloster with two vehicle configurations, one with the original and the other with an improved spoiler, were tested using a commercial CFD code STAR-CCM+ from Siemens. Both vehicle configurations were simulated using four different test conditions, providing overall eight different sets of simulation settings. The CFD methodology was validated with experimental data from the Hyundai Aero-acoustic Wind Tunnel (HAWT), by accurately reproducing the test section with static floor boundary conditions. Wind tunnel is a device that artificially produces airflow relative to a stationary body and measures aerodynamic force and pressure distribution to simulate with actual conditions. Wind Tunnels offer a rapid, economical, and accurate means for aerodynamic research. An important aspect of wind tunnels is their ability to accurately simulate the full complexity of fluid flow. Small-to-medium sized wind tunnels are used in research laboratories for experimental and educational purposes. Though these are relatively smaller in size as compared to commercial wind tunnels, meeting their accurate and precise design and fabrication specifications is quite a tough task. This paper focuses on the design aspects of various wind tunnel components like the test section, contraction cone, diffuser, drive system and the settling chamber. It also discusses various manufacturing approaches considered in previous researches. This dissertation was written as a part of the MSc in Strategic Product Design at the International Hellenic University. During the past few decades, CFD has entered the world of product development. Companies follow

certain processes for improving their design using CFD tools. This thesis contains a demonstration of such a process. Altair University designed a 3-wheeler motorbike geometry which was tested in a wind tunnel in the University of Mosbach. The results from this test were used as validating data for a CFD simulation that was built and run using the CAE software suite HyperWorks (HyperMesh, Acusolve, Hyperview) provided by Altair. A necessary literature review was conducted for the correct setup of the CFD simulations. The wind tunnel test was simulated and the results showed a satisfactory correlation with the physical test after a mesh independency study. The geometry was then simplified for an external aerodynamics CFD simulation. The settings of the CFD study for the external aerodynamics case, were based on the wind tunnel test simulation setup. The drag coefficient of the 3-wheeler motorbike was estimated. A comparison was made between the resultant drag coefficient of the wind tunnel test and the external aerodynamics simulations. Subsequently, the original model was re-designed. The goal was to improve the original model by reducing the drag coefficient of the vehicle. The new design led to a more streamlined body and on the new geometry, CFD simulations were conducted. These simulations resulted in a significantly lower drag coefficient than the one of the original design. The drag coefficient of the original model was approximately 0.35 and after the re-design process, it dropped to 0.15, a 57% improvement. The reason behind this significant difference is the absence of extensive recirculation areas past the rear of the redesigned model, in contrast with the original design

Note: This work is partly presented at 2nd International Conference on Aerospace, Defense and Mechanical Engineering; Webinar- August 17-18, 2020.