The Synergy of CFD and Experiments in Aerodynamics Research at Cranfield University, Shrivenham

A. J. Saddington K. Knowles

Aeromechanical Systems Group,
Department of Aerospace, Power and Sensors,
Cranfield University, RMCS, Shrivenham,
Swindon, Wiltshire, SN6 8LA, UK.

Abstract

For some time now, aerodynamics research carried out at Cranfield University's Shrivenham Campus has made use of both experimental and numerical methods. The combination of these two disciplines has enabled greater understanding of the flow physics being studied than would have been possible through either a purely experimental or purely numerical approach. Using examples of recent research projects, this paper will discuss the synergistic role that CFD and experiments have had in our aerodynamics research programme.

Early combined CFD and experimental research was primarily driven by the need for CFD validation [1]. With computational resources limited, the experimental results in the example given were used to validate CFD models of impinging jet flows. The main objective was to give globally similar results to those measured in the experiments.

More recently, improved computational resources have enabled refinements to the CFD models. Whilst CFD validation is still important, these models have provided additional insight into the physical processes involved, which were not discernable from the experimental measurements [2]. In the study cited, a CFD model was developed of the jet flow-field produced by two axisymmetric convergent castellated nozzles. Experimental measurements had indicated that the castellated nozzles entrained more mass flow into the jet than the baseline design. The CFD models verified that this was the case but crucially also provided a physical explanation of the entrainment mechanism that would have been very difficult to deduce from the available experimental data.

CFD models enable researchers to investigate physical processes that are almost impossible to reproduce experimentally [3]. Here the investigation concentrated on the effect of using external wheel support struts during racecar wind tunnel testing. The struts are used to mechanically decouple the car body from the ground whilst still allowing the wheels to rotate. An experimentally validated CFD model of an isolated racecar wheel and strut was used to quantify the aerodynamic interference effects between the two. The virtual environment of the CFD model enabled the support strut to be easily removed, something that could not have been carried out experimentally!

In our studies on transonic cavity flows, CFD and experiments are being used in synergy to provide design input for flow control techniques. The experimental measurements are being used to provide boundary condition inputs for the CFD models. Once validated, these models are being used used to prototype flow control mechanisms, which will suppress the undesirable acoustic phenomena. The most promising CFD-based control mechanisms will then be tested experimentally.

References

[1] Knowles, K. and Bray, D., "Computation of Normal Impinging Jets in Cross-flow and Comparison with Experiment," *International Journal of Numerical Methods in Fluids*, Vol. 13, No. 10, December 1991, pp. 1225–1233.

[2] Saddington, A. J., Knowles, K., and Wong, R. Y. T., "Numerical Modelling of Mixing in Jets from Castellated Nozzles," *The Aeronautical Journal of the RAeS*, Vol. 106, No. 1066, December 2002, pp. 643–652.

[3] Knowles, R. D., Saddington, A. J., and Knowles, K., "Simulation and Experiments on an Isolated Road Wheel Rotating in Ground Contact," *4th MIRA International Vehicle Aerodynamics Conference*, Warwick, UK, 16-17 October 2002.