

Computational and Experimental Studies of the Virginia Tech Hypersonic Wind Tunnel*

Rui Chen, Dr. Joseph Schetz
Department of Aerospace and Ocean Engineering, Virginia Tech

My poster presentation will display the results of some Computational fluid dynamics (CFD) predictions and experimental data of the flowfields inside the Virginia Tech Hypersonic Wind Tunnel (VT HST) shown in Figure 1 at nozzle exit Mach numbers of approximately 4.0 and 7.0. Comparisons between the CFD predictions and the experimental data will also be presented in the poster. The purposes of my research are to assess the quality of the flowfield inside the VT HST and to improve the understanding of its working characteristics. Obtaining numerical and experimental results for Mach 7 flowfield is especially important because those results are not previously available. In the CFD solution, the entire VT HST is approximated as axi-symmetric. Steady-state CFD calculations are performed using AeroSoft's GASP 4.2.1. Wilcox's (1998) $k - \Omega$ turbulence model is used. A sample CFD solution is shown in Figure 2. The experimental study is still in progress. I am currently working on taking shadowgraph photographs of the flow past a wedge at the nozzle exit to determine approximately the Mach number in the beginning of the test section. I plan to measure the Mach number along a vertical line in the center of the test section to determine the flow uniformity inside the VT HST at various nozzle exit Mach numbers.

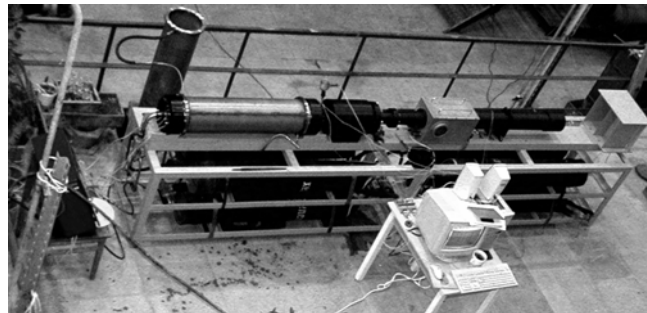


Figure 1: VT HST

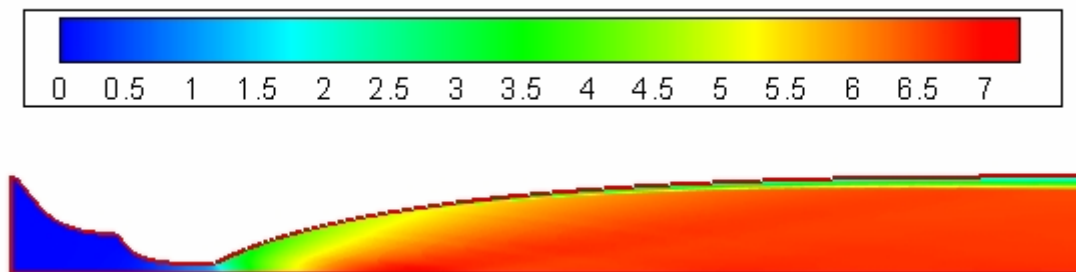


Figure 2: Mach Contour of the VT HST's Mach 7 Nozzle (CFD Solution)

* I would like to thank my good friends and colleagues Aristides Bonanos and Luca Maddalena for their help in my research project.