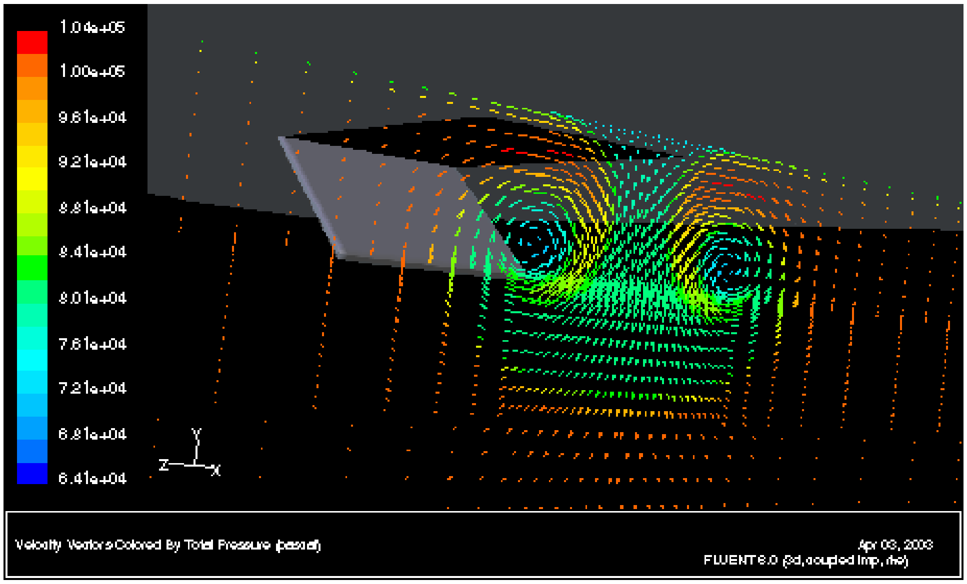
Computational and Experimental Studies of Pressure Relief Doors in Ventilated Nacelle Compartments

Mr P.Pratt, Dr E. Benard, Dr J.Watterson

The Queen's University of Belfast, School of Aeronautical Engineering



Pressure Relief Doors (PRDs) are used on engine nacelles to vent excess air to the free stream in the event of a burst of, or leak from, high-pressure supply lines in the interior of the nacelle. Current designs have been based on experimental data presented in NACA TN4007 (Vick, 1957), regarding the discharge characteristics of flapped, curved duct outlets in transonic flows. More detailed information is now required to aid development and certification of future nacelle system designs.

The flow structures formed around such flapped outlets are a complex combination of vortices, an oblique jet and shear layers. The exact combination and development of these structures is dependent on flap angle, free-stream Mach number and pressure ratio.

A CFD study has been carried out, based on the geometry detailed in NACA TN4007, using FLUENT 6 with data being generated across a range of flap angles, Mach numbers and pressure ratios. The computational data were in agreeance with the experimental results presented by Vick.

Flap angles across a range greater than used in the experiment were considered and it was shown that the discharge flow ratio (DFR) increased with flap angle up to an optimum value, after which increasing flap angle decreases the DFR. Increasing the pressure ratio reduces the value of optimum angle whilst increasing free-stream Mach number increases the value of optimum angle. It was shown that angle for which the pitching moment on the flap was zero fell in the range of 10^o to 15^o for all cases. A freely hinged flap would therefore balance in that range of angles, with the assumption of a negligible flap weight. Increasing Mach number decreases this angle, increasing pressure ratio increases it.

An experiment has been designed and constructed to study a case were the curved duct outlet is replaced with a plenum to more accurately simulate the conditions of a real PRD. An intermittent, draw down, transonic wind tunnel is used. Pressure and discharge data are to be collected in preliminary experiments with further experiments planned to investigate the flow structures formed using flow visualization and optical measurement techniques. Results from this series of experiments will be used to validate a further CFD model to be used in any future design of pressure relief doors for ventilation of engine nacelles.