Investigation of Flow Turning in a Natural Blockage Thrust Reverser

S. Hall, R.K. Cooper, E. Benard, S. Raghunathan

School of Aeronautical Engineering, Queen's University Belfast, N.Ireland

Summary

Conventional cascade type thrust reversers redirect the fan flow of turbofan engines to produce reverse thrust. They usually consist of a blocker door to block the fan duct and cascade vanes to turn the flow and direct it forwards and outwards through orifices created by translating rearwards the fan cowling. The natural blockage thrust reverser does away with the blocker door. Instead the fan duct is shaped so that when the reverser shroud translates aft to expose the cascade the shroud naturally blocks the fan duct. The conventional natural blockage thrust reverser still relies on a cascade to turn and redirect the fan flow.

The aim of the ongoing project is to first analyse the flow in a specified natural blockage thrust reverser with conventional cascade and then to investigate alternative means of flow turning in the reverser to determine if comparable flow deflection and thrust performance can be obtained. The project will consist of both experimental tests and CFD simulations.

The experiments are being conducted on a 50% 2D scale model in an open circuit low speed subsonic wind tunnel. The thrust reverser model geometry is shown in cross-section in figure 1. The model inlet cross-section is 380mm by 90mm and the maximum free stream velocity is 14m/s. Experimental measurements will include surface static pressures in the duct, inlet weight flow rate, exit total pressure surveys, exit flow direction and axial force on the model.

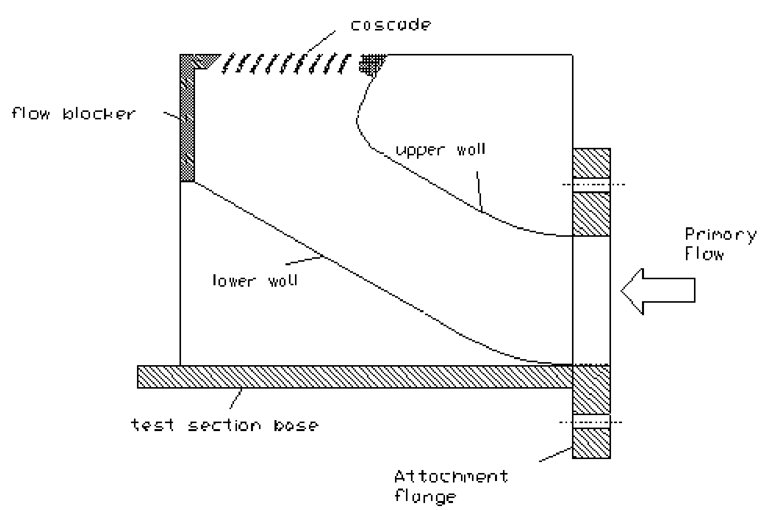

Figure 1. Thrust Reverser Model Geometry

2D CFD simulations will be conducted in FLUENT for the same geometry and operating conditions with the experimental results providing validation of the numerical results. To create experimental models of the fan duct geometry with the reverser both deployed and stowed would be costly and expensive. Therefore after validation the flow through the fan duct geometry with the reverser stowed will be modelled computationally in FLUENT. Results from the stowed duct case will be used to quantify the efficiency of the reverse thrust performance in terms of the forward thrust performance.