A NUMERICAL INVESTIGATION OF THE FLOWS IN AND AROUND ANNULAR-PLUG NOZZLE CONFIGURATIONS

D.A.Perigo, Prof. N.Qin
Cranfield University

Keywords: *Plug Nozzle, External Expansion Nozzle, Numerical Simulation, Turbulence Modelling*

Abstract

Recently there has been a resurgence of interest in the use of self adapting nozzles, specifically the External Expansion Nozzle (EEN), for space launch applications. This can be considered an effect of the renewed interest in single stage to orbit launch vehicle designs, notably the X33. This poster details the work to date at Cranfield, in conjunction with the European Space Agency, into the investigation the complex flowfields produced by such nozzles and assesses the applicability of modern computational techniques, in particular advanced two equation turbulence models, in the prediction of such flows.

Results are presented for the simulation of the turbulent flow around an axisymmetric EEN geometry with external flow. Comparisons are made with experimental pressure measurements including pressures along the plug surface and pressures in the base region. Comparisons of velocity profiles against LDV measurements are also made. A grid sensitivity study is then performed for the Wilcox $k-\omega$ *model before moving on to compare against more advanced* $k-\omega$ *formulations including cubic and quadratic nonlinear models. Finally, utilising our chosen turbulence models, the phenomenon of wake closure is investigated using two seperate methods and an asesment made of the suitability of the respective methods for the prediction of EEN flows.*

1 Poster Summary

Current research activities at Cranfield, in conjunction with the European Space Agency (ESA), centre on numerical simulation of both annular plug nozzle flow-fields and the more complex Clustered Module Plug Nozzle (CMPN) flow-fields. In its simplest form the annular EEN provides propulsive thrust by transmitting the pressure force acting on the outside of a conical surface to the vehicle. This contrasts with a conventional nozzle which, in the simplest case, transmits the pressure force acting on the inside of a conical surface to the vehicle. This external expansion means that the free boundary of the flow can move, changing the effective geometry, and hence expansion ratio of the nozzle and allowing it to adapt to the changing pressure experienced while climbing through the atmosphere.

The resulting flow-field, created by an EEN, presents numerous challenging features in terms of the prediction of shock-shock and shock-turbulent boundary layer interaction and also in the prediction of base flows. It is also understood that the accurate prediction of these flow-fields is essential if the economic design of such propulsion systems is to be realised. The utilisation of CFD as a design tool is now widespread around the world, however, for flows such as those currently under investigation , it may prove precarious to undertake design work using CFD without an understanding of the physics of the problem, or the validation of the afore-mentioned software against experimental data. This paper attempts to provide both of these things.

This research concerns itself, in the main, with the numerical simulation of EEN flowfields. Solutions in this paper have been calculated using flow solver algorithms developed in house at Cranfield. The flow solver utilises a high resolution discretisation based on Osher's approximate Riemann solver for the calculation of the inviscid flux vector. This is coupled with a MUSCL scheme to provide third order accuracy in space. Central differencing is used for the calculation of the viscous flux vector and turbulence modelling is achieved using a strongly coupled approach and a variety of linear and nonlinear, two equation, eddy viscosity models. The multi-block methodology implemented within the code allows the EEN geometry to be dealt with easily using high quality grids.

Results are presented for the simulation of the turbulent flow in and around an axisymmetric EEN geometry, Figure(1). Comparisons are made with experimental data including pressure measurements across the EEN surface and in the base region. Numerical velocity profiles and flow in the base region are compared with LDV measurements. The work includes a study of grid sensitivity where it is demonstrated that there is a marked sensitivity to coarsness of grid. The calculations made with respect to each experimental case are repeated for a variety of high order turbulence models and the differences in the resulting flowfields are discussed, Figure(2).

The phenomenon of wake closure is investigated along with a comparison of thrust estimates. The accurate prediction of wake closure is deemed to be essential in the prediction of overall EEN thrust performance. In this work two different methods of approaching the prediction of wake closure are examined and the differences in the results that they produce are detailed.

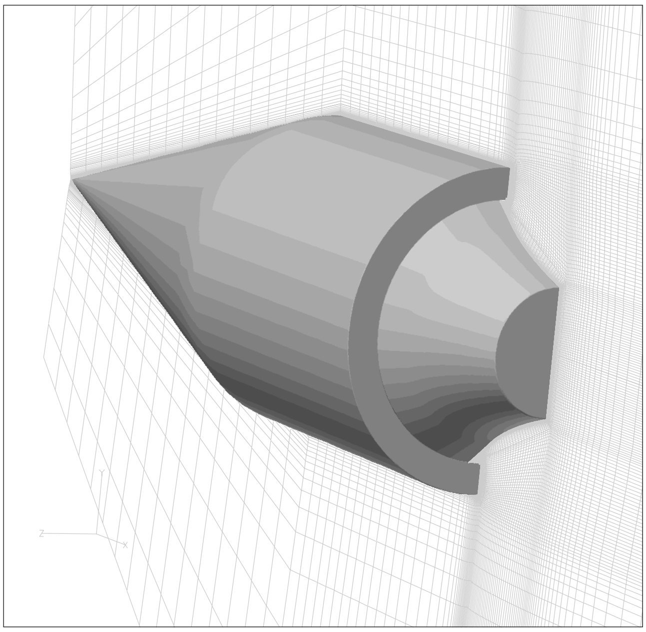

Fig. 1 *EEN surface model - full configuration half-plane*

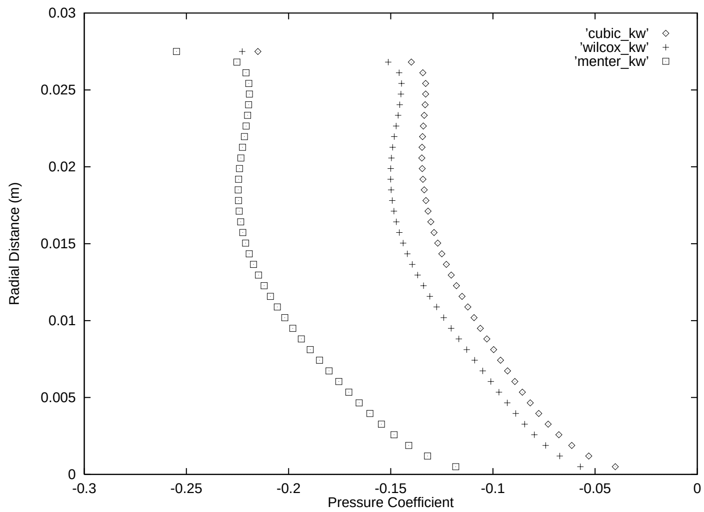

Fig. 2 *Pressure coefficients in base region for selected turbulence models.*