

AUTOMOTIVE FAN COMPONENT CFD MODELLING FOR DESIGN : BLUFF BODY COMPONENTS

Background

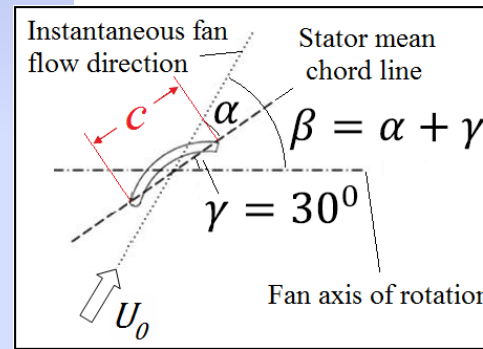
The performance of axial flow fans used in vehicle engine cooling modules relies on the aerodynamics of different components, several of which are blunt or “bluff” in shape (e.g. stators and support arms).



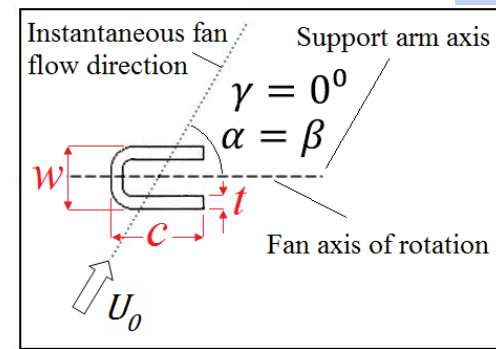
(a)



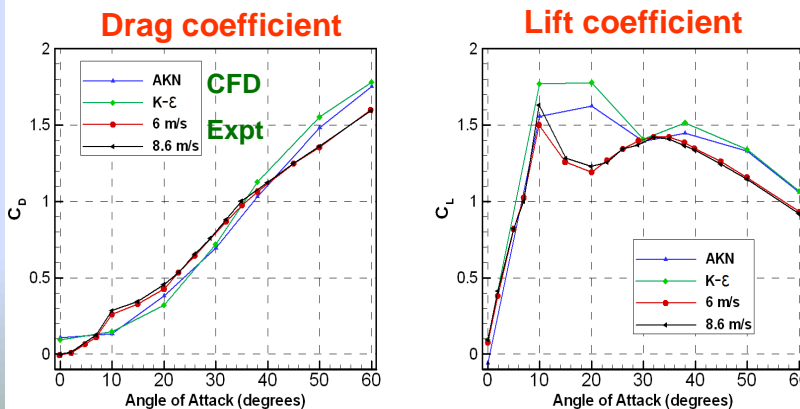
(b)



(a) *Cambered plate airfoil used as a stator and*



(b) *U-shaped fan support arm*



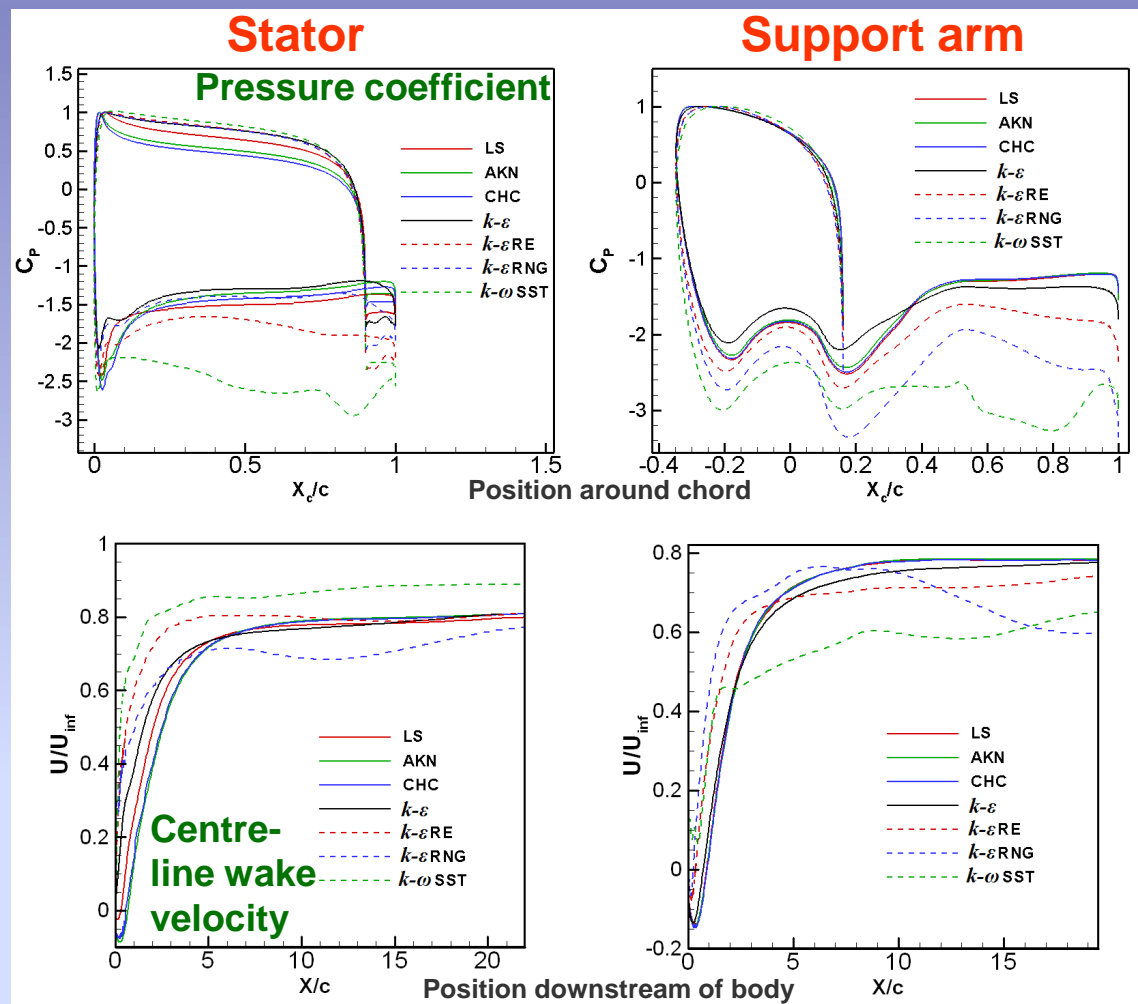
CFD Stator drag and lift + experimental data

Objective

Determine best CFD model to predict these bluff body flows.

Research Carried Out

Numerical modelling (CFD) of the flow and the resulting lift and drag coefficients of a typical support arm cross section and a typical stator, using different turbulence model closures, with comparison to experimental data.



Pressure distributions and wake velocities

Key Findings

Abe-Kondoh-Nagano (AKN) model gives best results and greatest sensitivity to angle of attack. Cascade simulations showed interactions between adjacent support structures.