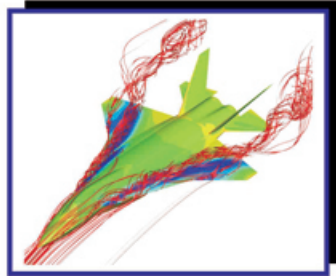




Computational fluid dynamics (CFD) has been used by CSIR to model and optimize the performance of civilian and military aircraft, launch vehicles and missiles.



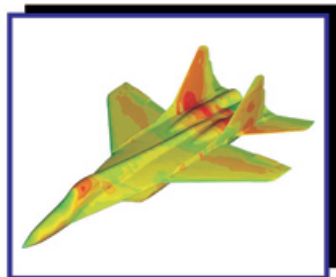
Pressure contours and stream lines

Routine CFD computation are now possible for complex aircraft configurations

A panel method analysis was carried out to assess the installation effects of a pusher propeller.



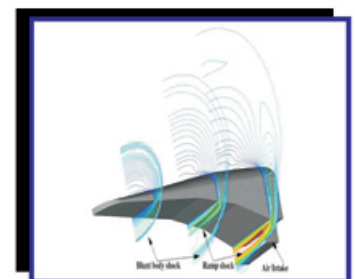
Pressure distribution over the SARAS aircraft with pusher propellers.



Computed pressure distribution over an advanced fighter aircraft.

Euler calculations were carried out for a complete fighter aircraft configuration for evaluation aerodynamic performance.

Reynolds-averaged Navier-Stokes computations (RANS) have been carried out to study the inlet aerodynamics of a hypersonic research vehicle (HRV).



Density contours at different inlet sections of the HRV.