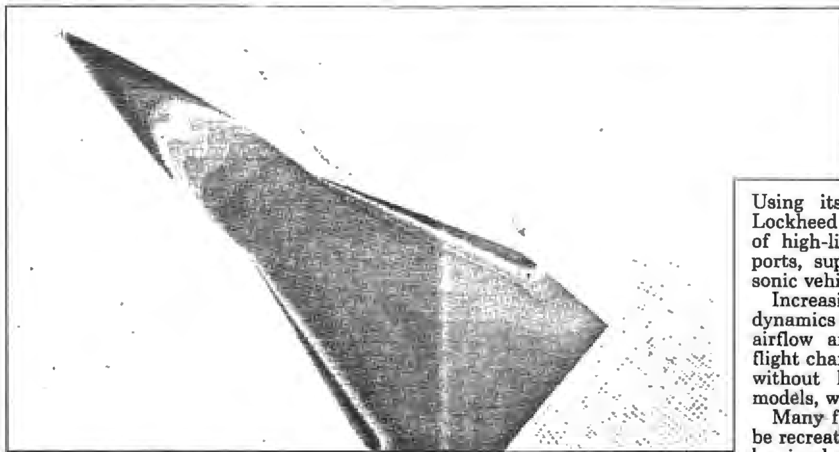
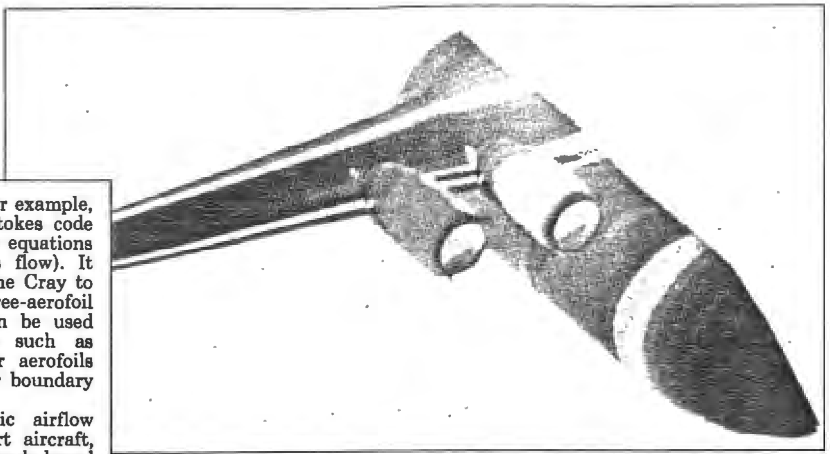
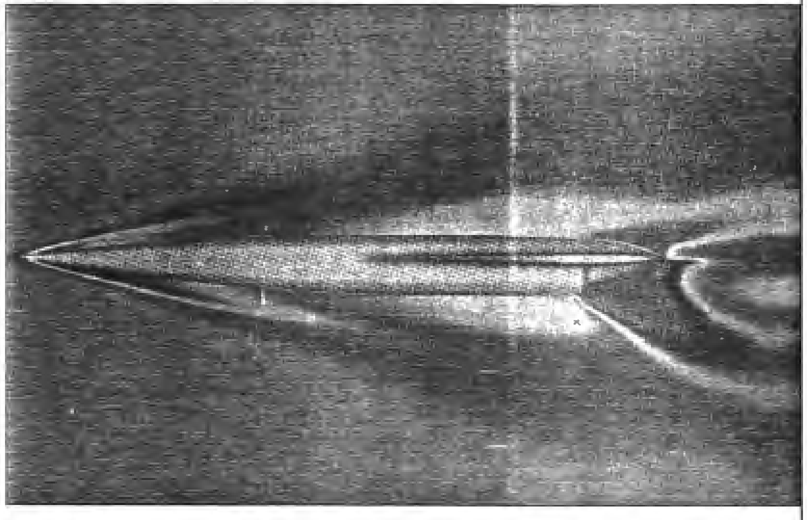
Supercomputer aerodynamics shows Lockheed's way ahead



Computational aerodynamics can be used to describe the airflow around hypersonic vehicles **(left** *and* **below)**, *while the computed pressure fields over advanced fighters* **(bottom** *and* **centre)** *can be displayed graphically*

The interaction of engine exhaust and wing airflow can be modelled for over-the-wing **(right)** *and upper-surface-blowing* **(below)** *propulsive-lift concepts.* **Bottom** *Low-pressure (blue) regions are created by leading-edge vortices*





Using its own Cray supercomputer, Lockheed is studying the aerodynamics of high-lift wings, powered-lift transports, supersonic fighters, and hypersonic vehicles.

Increasingly, computational aerodynamics is being used to simulate the airflow around aircraft so that their flight characteristics can be determined without having to build windtunnel models, writes **Graham Warwick.**

Many flight conditions which cannot be recreated in the windtunnel can now be simulated using computational aerodynamics. This is particularly true of hypersonic flight.

At Lockheed-Georgia's Advanced Flight Sciences Department, some 30 people are generating and applying computational fluid dynamics (CFD) code using a Cray X-MP/24 dual processor supercomputer. To check that the code generated is valid, Lockheed Georgia uses a variety of windtunnels and a number of laser velocimeters to make aerodynamic flow measurements.

The procedure used at Lockheed is to define the geometry of the wing or aircraft to be analysed using a computer-aided design system. A computational grid is then generated in the region surrounding the body. The equations which describe airflow behaviour at each point on this grid are then solved, and a complete flow description is synthesised. This is then used to calculate the pressures and forces acting on the body so that its flight characteristics can be determined.

Lockheed uses different codes for different applications. For multi-element high-lift aerofoils, for example, a two-dimensional Navier-Stokes code is used. (The Navier-Stokes equations of motion apply to viscous flow). It usually requires 30min on the Cray to compute the flow around a three-aerofoil system. The same code can be used to model blown aerofoils such as circulation-control wings, or aerofoils using suction or blowing for boundary layer control.

To analyse the transonic airflow around a complete transport aircraft, Lockheed engineers use methods based on the less-complex Euler equations, which describe non-viscous flow. This code is used to determine the transonic drag penalty and wing flow interference that result from powered-lift configurations such as over-the-wing engines and upper surface blowing.

For the subsonic, transonic, and supersonic analysis of advanced fighter designs, Lockheed uses a three-dimensional version of the Navier-Stokes viscous-flow equations, called ENS3D. Typical Cray computation times for a blended wing/body fighter using a 100,000-point grid are around one hour, says Lockheed.

The low-speed and transonic characteristics of hypersonic vehicles can be analysed using the ENS3D code. Analysis at hypersonic speeds is more demanding, however, because of the low flow densities and high stagnation temperatures, and a modified Navier-Stokes code is used. Another version has been developed to model hydrogen-air combustion flowfields in the afterbody/exhaust region.