Fluid dynamics

by **Ugo Piomelli**
Meelan M. Choudhari

The year brought exciting developments in the areas of transition, turbulence, and flow control. The breadth of these research activities and their contribution to major national programs should dispel any ill-founded notion that fluid dynamics is a mature science holding little potential for progress.

Turbulence

Turbulence remains one of the unsolved problems with major impact on engineering analysis and design. The availability of inexpensive computational power has made it possible to apply advanced modeling methodologies to turbulent flows and increase the understanding of flow physics. Direct numerical simulation (DNS) is beginning to be applied in practical configurations. One aeronautical application of DNS was the study of large-eddy breakup devices to decrease cockpit noise. Researchers at Boeing and in St. Petersburg, Russia, computed the pressure field downstream of such devices to determine their effectiveness in reducing cockpit noise.

DNS is also applied in biofluid mechanics, where Reynolds numbers are typically low or moderate. Within this context, immersed-boundary methodologies provide an efficient way to treat moving boundaries such as blood vessel walls, moving parts of heart valves, and deformable insect wings.

At higher Reynolds numbers, large-eddy simulations (LES) and other eddy-resolving techniques are increasingly applied to studies of complex flows. The full-aircraft calculations performed by Boeing, ASU, and Cobalt using the detached eddy simulation method may be the most advanced application of this type.

Among the more complex LES applications are the flows in turbine cascades. Researchers at the Wright-Patterson Air Force Research Laboratory (AFRL) performed experiments and computations for this flow, to explore passive- and active-control strategies. In parallel, computations carried out at the University of Arizona highlighted the effectiveness of forcing in reducing the separation. NASA Langley researchers performed hybrid calculations to understand the noise-generation mechanisms associated with large-scale unsteady structures near the leading-edge slat of a multielement high-lift configuration.

Progress in experimental techniques is making it possible to obtain global flow information that enhances understanding of the turbulent flow physics. Work at the University of Notre Dame on trailing-edge noise is a notable example. A unique combination of hot-wire anemometry, particle image velocimetry (PIV), and acoustic measurements has provided new insights into the trailing-edge sound-generation mechanisms. At Virginia Polytechnic Institute and State University, studies of wall roughness using micro laser-Doppler anemometry have provided similar insights into this ubiquitous problem, which is relevant to both engineering and geophysical applications.

Researchers at Johns Hopkins University used PIV and holographic PIV to study both the physics of interscale energy transfer in homogeneous turbulence and the complex flows in turbine cascades. At Caltech, innovative measurements of the unsteady flow field due to flapping wings are helping to explain the bumblebee paradox. These experiments, together with related calculations by other groups, may significantly accelerate the development of biomimetic flying devices.

Laminar-turbulent transition

Advances in transition research covered a broad range of speed regimes and modeling fidelity. The focus of several investigations (bypass or roughness-induced transition, transition mechanisms on multielement airfoils) either has clear impact on aerospace programs of national interest or affects a variety of disciplines. The technological relevance of these studies underlines the need to maintain long-term research activities in this challenging area.

NASA Langley researchers have analyzed the database acquired during a 3D high-lift experiment using over 500 hot-film sensors on the leading-edge slat, the main element, and a single slotted flap. The boundary layer may be laminar along the slat upper surface, turbulent on the main airfoil, and transitional on the flap; this finding highlights the complexity of high-lift flows and the resulting challenges for computational models. Predictions obtained using unstructured codes demonstrated the ability of turbulence models to match the maximum lift and the stall angle to within 1% and 2 deg, respectively, of the measured values.

Researchers at CFX developed a new transition model based on two transport equations, one for intermittency and one for a transition-onset criterion based on the momentum-thickness Reynolds number. The comparison with measured data for a range of flows including a full helicopter configuration suggests that the model might allow integrated prediction of laminar, transitional, and turbulent flow fields in industrial CFD applications.

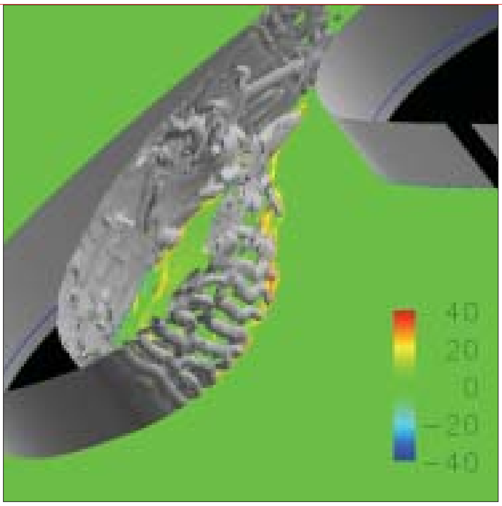
Investigations of boundary-layer transition (and of jet and cavity flows) played a critical role in the space shuttle Return to Flight mission (STS-114), particularly in the development of tools to assess the thermal protection system (TPS) damage and to define repair criteria. Wind tunnel measurements of boundary-layer transition due to discrete surface protuberances and cavities were used to correlate transition onset on the shuttle orbiter with roughness height associated with TPS damage. This correlation played an important role during STS-114 in determining the disposition of protruding tile gap-filler observed on the wing surface after launch.

Return to Flight activities underscore the need for low-disturbance test facilities at hypersonic speeds. Computations at Rutgers University and Mach-6 Ludwieg Tube measurements at Purdue suggest that a separation bubble along the nozzle bleed-lip leads to premature tripping of the nozzle-wall boundary layer. Future computations and experiments appear likely to eliminate the separation and achieve quiet flows at high Reynolds numbers.

A team from NASA and Boeing assessed the risk of a breach in the shuttle wing leading edge by developing models for the internal stagnation heating due to an impinging jet through a damaged wing. Concerns about damage to the shuttle TPS from gouges and missing tiles prompted further study of cavity flows in hypersonic flight conditions. Wind tunnel tests in NASA Langley's 20-in. Mach-6 tunnel were used to develop a correlation between cavity geometry and the change in heating relative to undisturbed levels. CFD calculations revealed the need for additional tests to simulate fully the high-enthalpy, real-gas flow at flight conditions. To enable studies of the effects of real-gas chemistry, boundary-layer transition, and gas-surface interactions on hypersonic flows, the LENS-X facility has been developed at CUBRC (Calspan-UB Research Center).

Theoretical fluid mechanics

Theoretical approaches continue to be useful in various areas. By constructing a causal and globally bounded Green's function for Lilley's equation for jet noise, researchers at NASA Glenn have resolved a longstanding controversy related to the unstable eigensolutions of this equation. Work at Imperial College has provided fundamental insights into the highly directional acoustic radiation from nonlinear, supersonic instability waves in a turbulent jet. Researchers at University College (London) have used a combination of asymptotic and numerical methods to discover the effects of droplet impact on water layers, with the goal of improving the modeling of aircraft icing.



Isosurfaces of spanwise vorticity show the development of large-scale structures within the slat-cove shear layer and their subsequent breakdown into 3D structures. (Calculations carried out at NASA Langley.)

Researchers at Boeing have predicted the critical angle of attack for transonic-buffet onset via a global-instability analysis of the underlying mean flow field, enhancing the prospects for efficient, physics-based buffet predictions. Continuing the progress toward global-instability analyses of complex flows, researchers at Polytechnic University (Madrid) and Caltech have demonstrated the role of this tool in identifying new eigenmodes of the open cavity flow that are inaccessible to the classical linear theory.

Flow control

Flow control is a critical area; engine performance, aircraft maneuverability, and acoustic emission can be improved by passive or active control mechanisms. Thus much research focuses on this topic.

A combination of wind tunnel tests and computational predictions has been used to investigate ultracompact inlet ducts with active flow control. This research, carried out jointly by AFRL, Lockheed Martin, and Honeywell, aims to develop an integrated, advanced propulsion system for a modern fighter.

Wind tunnel experiments at NASA Langley also used active flow control to reduce the engine fan-face distortion in an S-shaped diffuser inlet with large amounts of boundary-layer ingestion. This work has laid the foundation for realizing the aerodynamic performance benefits predicted via systems studies.

Researchers at Rockwell Scientific and the Institute of Theoretical and Applied Mechanics in Russia have developed a plasma actuator method for controlling symmetry-breaking of nose tip forebody vortices. The actuator system provides Joule heating that effectively moves the cross flow separation point and feeding sheets to a stable position to discourage bifurcations leading to the symmetry breaking.