THE NPARC FLOW SIMULATOR

NASA LeRC • USAF AEDC
NPARC
Government • Industry • Academia

GENERAL DESCRIPTION

The NPARC Navier-Stokes code is a general purpose computational fluid dynamics (CFD) tool which is applicable to a wide variety of aerospace design and analysis problems involving fluid flow. It is actively supported by the NPARC Alliance, a partnership between the NASA Glenn Research Center and the Arnold Engineering Development Center. This Alliance seeks to enhance the military and commercial competitiveness of the United States through the establishment of the NPARC code as a national resource.

The NPARC flow simulation program is used to calculate the properties of a fluid flow, based on specified boundary surfaces and flow conditions. These boundary surfaces can be quite complex and the fluid can be treated generally. Inviscid and viscous flows can be calculated. Viscous flows can be laminar or turbulent and can be treated as fully viscous or as shear layer flows. This computer program may be used to simulate steady-state and transient flows. The NPARC code, which is written in FORTRAN 77, is easily ported to most computer architectures (e.g. SGI and IBM workstations, Vax minicomputers, Cray and Convex supercomputers). It is robust and fairly easy to use. Anyone who is familiar with computers and has a basic understanding of the physics of fluid flows should have little trouble learning how to apply the NPARC code.

NPARC code flow simulations have had a demonstrable impact on aerospace propulsive applications as diverse as supersonic and hypersonic inlet design, rocket nozzle failure analysis, and turbine engine exhaust mixer design. This computer program has also proven capable of treating many other aerodynamic problems, such as, missile nose cone analysis, instrumentation probe design, and ducted flow analysis.

TECHNICAL FEATURES

Generality

The basis of the algorithms used in the NPARC code is the complete Navier-Stokes equations in conservation law form. Various specializations are provided. For example, the viscous terms can be selectively calculated so that a thin-layer simulation can be performed or an inviscid (Euler) flow-field calculated. Similarly, for viscous simulations the fluid flow can be treated as laminar or turbulent as desired.

Complex Geometry

The principal distinguishing feature of the NPARC code is its generalized treatment of boundary conditions. Physical and computational boundaries may be located on any grid surface without restriction. Any portion of any grid surface may be a boundary surface. Common boundary conditions (e.g., slip and no-slip wall, symmetry plane, free-stream) can easily be selected for each boundary. These features allow complex geometries to be readily treated using a single grid. Complex geometries may also be broken into a number of grid blocks. Grid blocking circumvents computer memory limitations, simplifies grid generation about complex geometries and permits grid embedding techniques. Communication between grid blocks is accomplished through the overlapping of grids. Adjoining grid blocks do not need to have an exact match of grid points.

Robustness

The NPARC code is also very robust. A semiautomatic time-step control feature allows flow simulations to proceed from nearly arbitrary initial conditions with little danger of divergence. This is accomplished by monitoring and limiting the maximum change in the flow field between iterations. In steady-state calculations, the time-step size is determined by stability considerations. An optimal time-step size is used for each grid point, reducing the total number of iterations to reach steady-state.

Turbulence Models

Another distinguishing feature of the NPARC code is its general purpose algebraic turbulence model. Although optimized for free shear layers at moderate Mach numbers, it provides reasonable viscous effects for almost any combination of shear layers. A standard Baldwin and Lomax algebraic turbulence model is also provided as an option when maximum accuracy is required in skin friction and heat transfer. For maximum flexibility, a K-epsilon two-equation turbulence model may also be used in turbulent flow simulations.

Algorithms

Almost any flow idealization can be simulated: 2-D, axisymmetric, or 3-D; inviscid, laminar, or turbulent; steady state, or transient. These flow simulations are calculated using either the pentadiagonalized form of the Beam and Warming approximate factorization algorithm or the Jameson multilevel scheme. The Beam and Warming algorithm is an implicit, computationally robust scheme for solving the Navier-Stokes equations. The most attractive feature of the algorithm, as modified by Pulliam, is that it forms an ADI type of scheme in which each sweep involves the inversion of a set of scalar pentadiagonal matrices. The Jameson multilevel algorithm is second-order accurate in time. It may be used in the form of a three-, four- or five-stage scheme. The Beam and Warming algorithm has the best convergence properties, while the Jameson scheme has the best time-accuracy.

BRIEFLY

The NPARC code embodies the most recent, proven CFD technology available. It is a general-purpose fluid flow simulator ideally suited to applications which involve complex geometries. In the hands of a fluid dynamicist with a basic familiarity with CFD technology, the NPARC code is a powerful tool for the analysis of many complex flows of immediate interest.

POINT OF CONTACT

If you have any questions or comments concerning the NPARC code or the NPARC Alliance, call 931-454-7455 or send E-mail to cooper@hap.arnold.af.mil.