2003 ME Graduate Student Conference
May 3, 2003

AN EFFICIENT PARALLEL METHOD FOR PRESSURE VELOCITY COUPLING OF THE UNSTEADY INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

Frank Muldoon
Ph.D. Candidate

Faculty Advisor: Dr. Sumanta Acharya

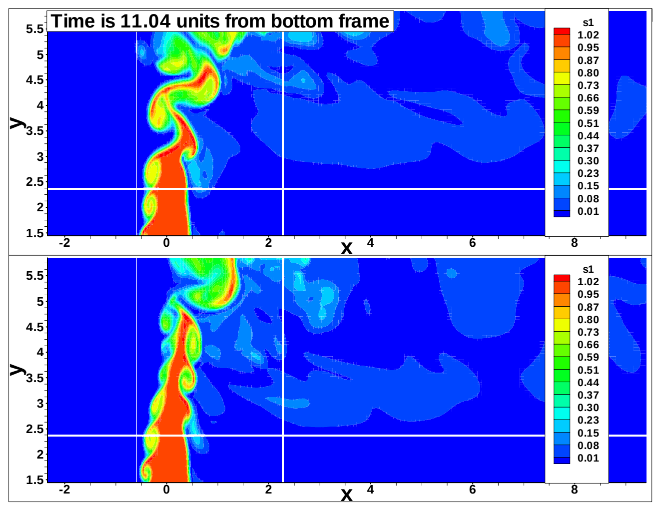
ABSTRACT

A method for the efficient parallel computation of the unsteady incompressible Navier-Stokes equations is presented. A computer program has been written and used to perform direct numerical simulation of a number of flows with, in some cases, scalar transport. These flows include channel flow, mixing jets, and flow over a backward facing step. Statistical data resulting from the direct numerical simulations have been collected so as to aid in the improvement of turbulence models.

The non conservative form of the unsteady incompressible three-dimensional Navier-Stokes equations are solved. In addition an equation describing the transport of a passive scalar is solved.

Turbulent flows are characterized by a range of spatial and temporal scales. The range of these scales is determined by the Reynolds number (Re). A turbulent flow is characterized by the presence of many different sized structures or eddies. For instance a flowing stream has large eddies that contain smaller eddies that contain still smaller eddies etc. This process continues until we reach the smallest eddies. At this level the eddies do not break down into smaller eddies but instead are destroyed by the viscous dissipation which turns their kinetic energy into heat. Direct Numerical Simulation (DNS) is the numerical solution of the three-dimensional Navier-Stokes equations with a resolution sufficient to capture all the important scales. This means that the numerical solution of the flow field must have a resolution able to capture the viscous dissipation of the small eddies. The spatial resolution requirements are given as proportional to Re^(9/4) in (Pope 2000) who also gives an estimate of the number of floating point operations to be proportional to Re^3. These large exponents restrict DNS to a small range of flows.

FIGURES AND TABLES



Mixing jets in crossflow

ACKNOWLEDGMENTS

This work was supported by the DOE, the Navy, NASA Stennis and Louisiana State University. The simulations were run on an IBM SP2 (LSU), IBM SP3 (ARSC), IBM SP3 (ARSC), IBM SP4 (NAVO), Linux PIII cluster (Maui Supercomputing Center), Linux Xeon cluster (LSU) and a dual processor Compaq Dec Alpha at the Computational Fluid Dynamics Lab at Louisiana State University.

REFERENCES

1. Stephen B. Pope "Turbulent Flows" Cambridge University Press, The Edinburgh Building, Cambridge, CB2 2RU, UK, (2000)