INDIANA
UNIVERSITY
PURDUE
UNIVERSITY
INDIANAPOLIS

Computational Fluid Dynamics LABORATORY

SL 144, 723 West Michigan Street
Indianapolis, IN 46202-5132

For information, contact: Dr. H.U. Akay
email: akay@engr.iupui.edu

A domain decomposition based parallel solver for incompressible Navier-Stokes equations using the finite element method

In this finite element solver, the fractional time-stepping algorithm is used for the solution of the unsteady incompressible Navier-Stokes equations. A previously developped semi-explicit scheme is modifies to a fully implicit algorithm. Isotropic upwinding is added, controlled by a pressure sensor. The program is converted for parallel computation using a database management program GPAR, developped for parallel/domain decomposition. This approach involves subdivision of the flow domain into sub-domains called solution blocks and distribution of the solution blocks to network of computers. The equations for each block are solved in block solvers, while the information exchange between neighboring blocks are solved in interface solvers. The data structure for communication between block and interface solvers are provided by GPAR. Several test cases are analyzed using both the semi explicit and implicit versions, to measure the performance of the parallel algorithms. The accuracy and convergence characteristics fo the different schemes are compared on low and high Reynolds number flows.

Test Cases

The first test case is an entrace flow in a 3-D rectangular duct. This case was used to test the program and compare the results of single and multiple-block cases. The **pressure and velocity fields** are shown here for the one-block case.

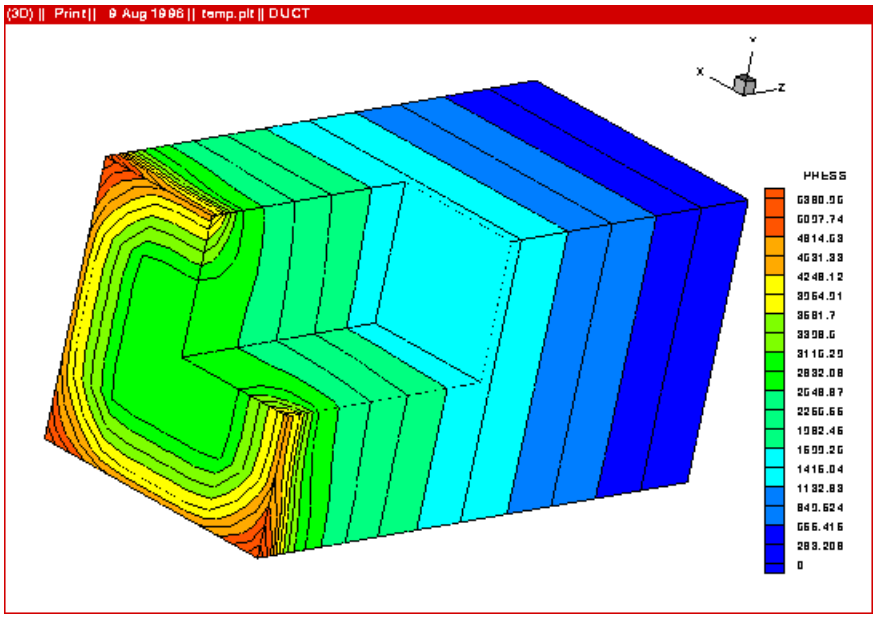
The next case is a 2-D step-duct problem. The **computational grid, pressure and velocity fields** displayed here show the results of the 10-block case.

A 3-D backward facing step-duct problem is also analysed and the results of a **2 and 8-**

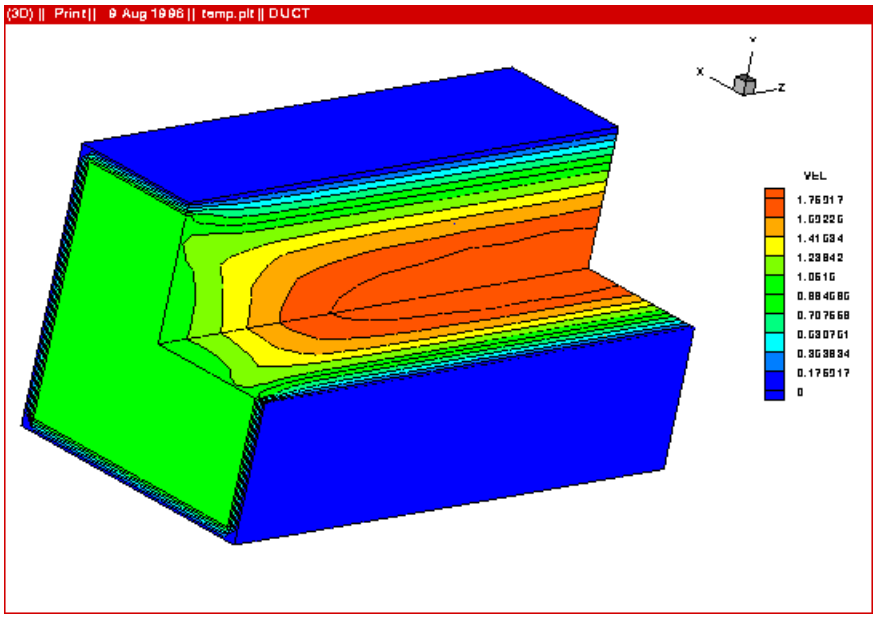
block case compared.

Finally, a **driven cavity problem** in a cube presented with results using 2 and 4-block subdivisions.

Pressure



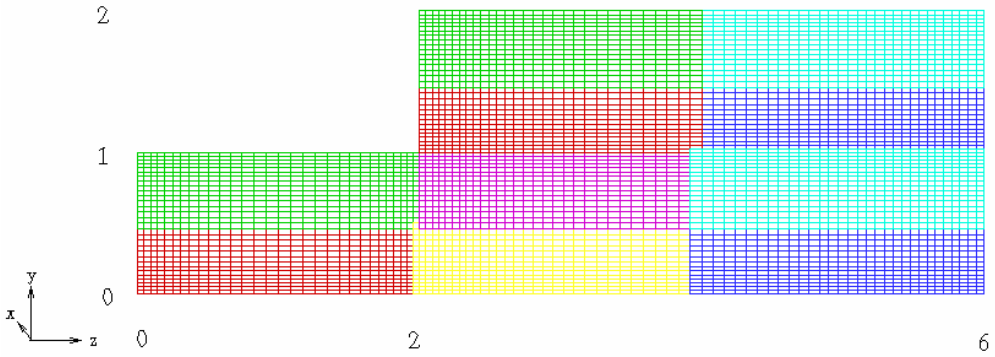
Velocity



STEP DUCT

23,255 - NODE GRID

(5 NODES IN THE X - DIRECTION)

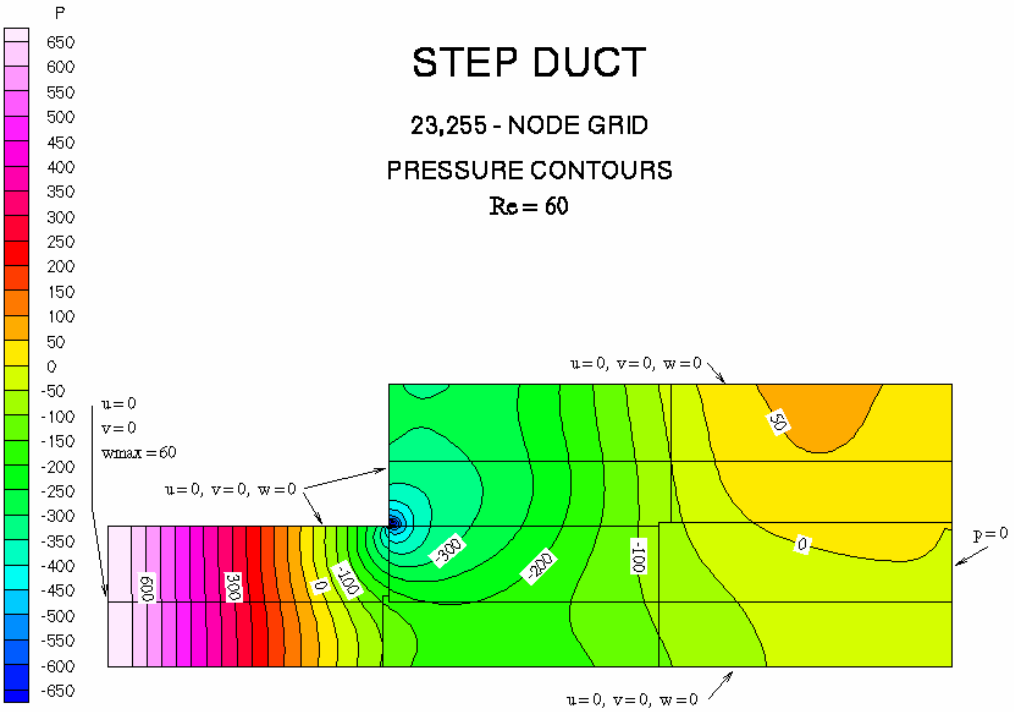


STEP DUCT

23,255 - NODE GRID

PRESSURE CONTOURS

Re = 60

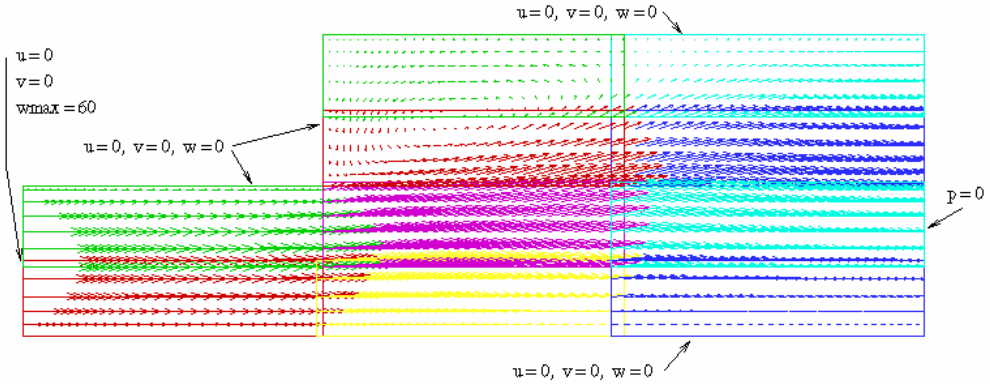


STEP DUCT

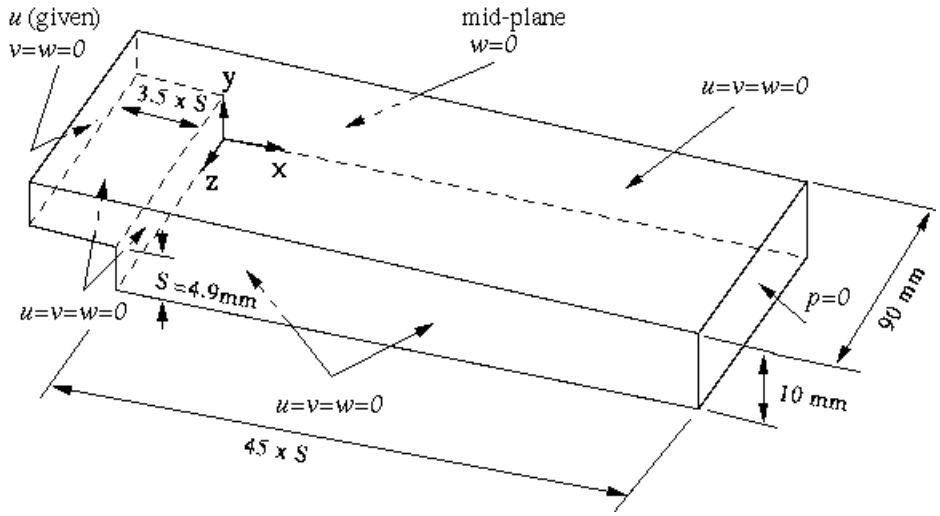
23,255 - NODE GRID

VELOCITY VECTORS

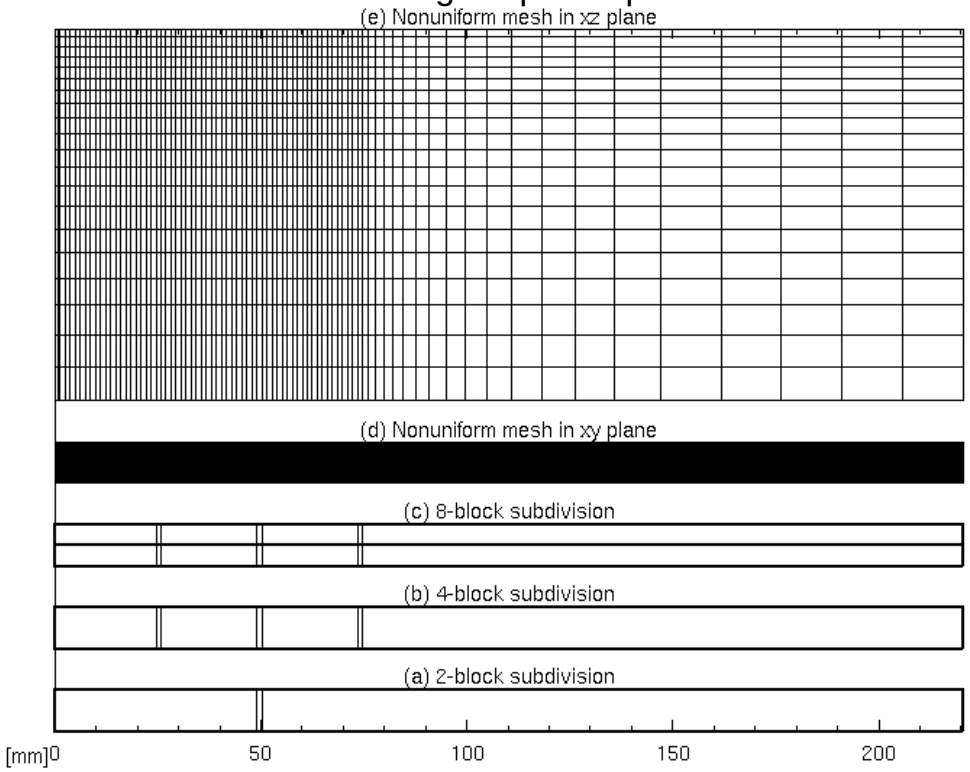
Re = 60



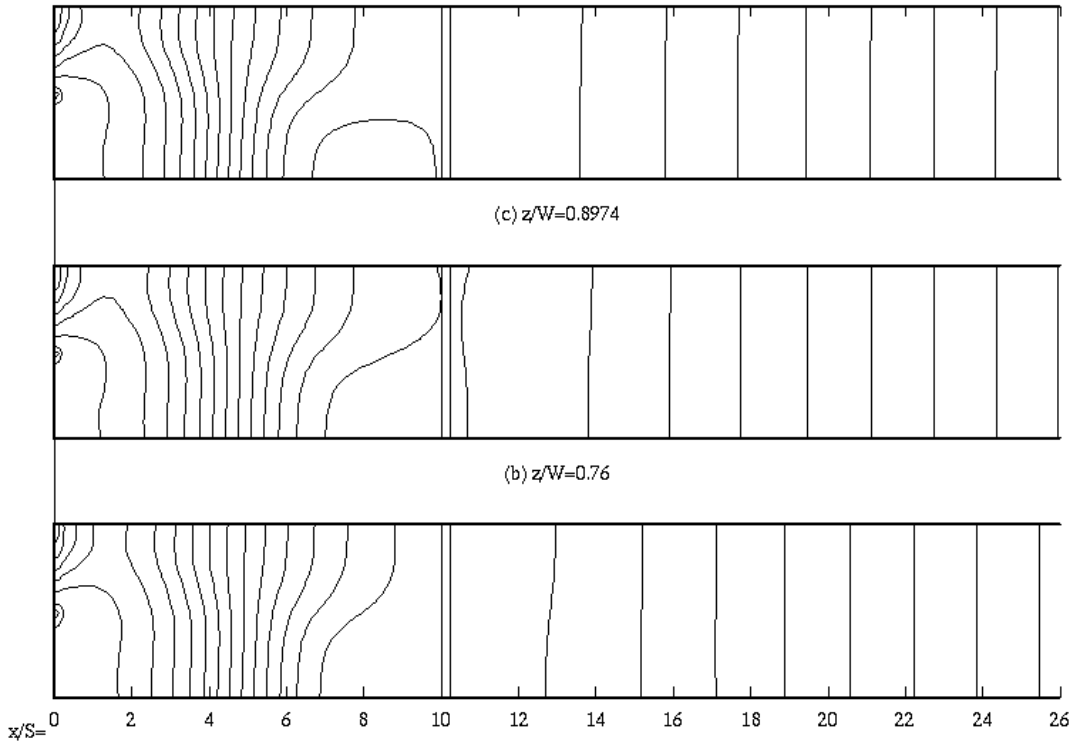
3-D Backward Facing Step



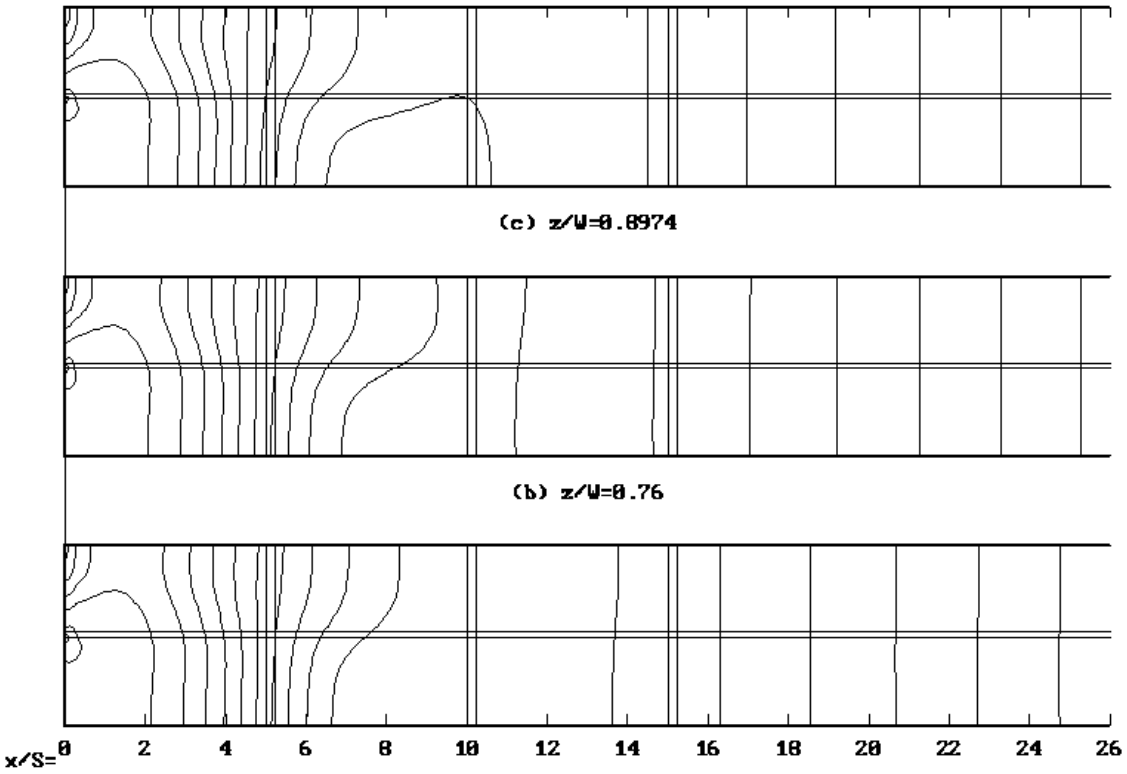
3-D Backward Facing Step Computational Grid



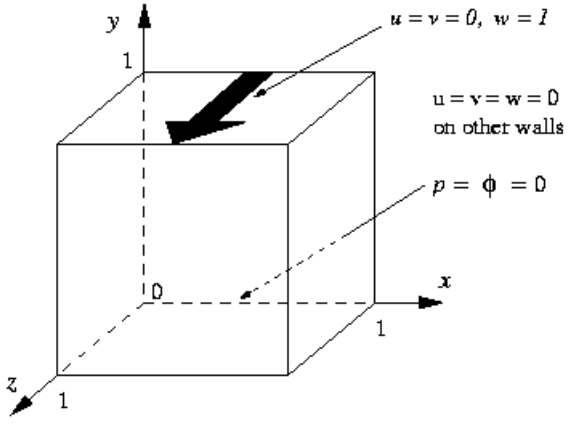
3-D Backward Facing Step Pressure Contours, 2-block case



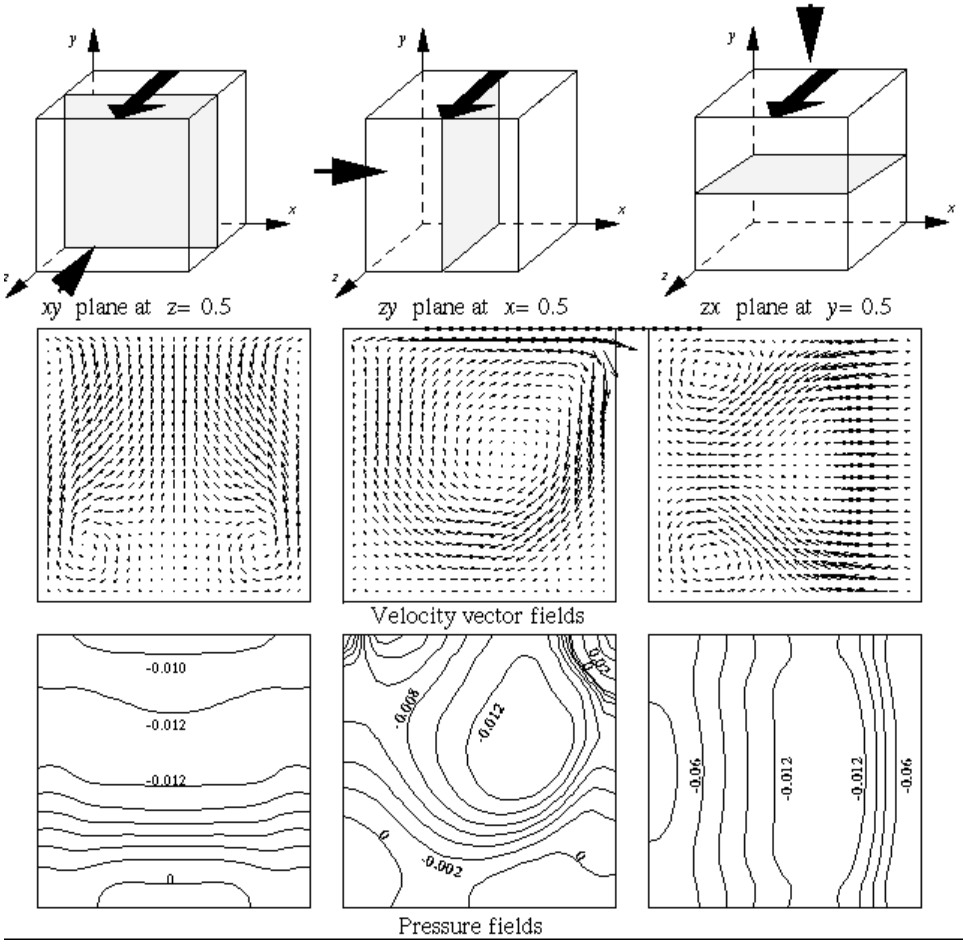
3-D Backward Facing Step Pressure Contours, 8-block case



Driven Cavity



Driven Cavity, 1-block case



Driven Cavity, 4-block case