University of Nebraska - Lincoln
DigitalCommons@University of Nebraska - Lincoln

US Army Research U.S. Department of Defense

1992

Three-Dimensional Incompressible Flow Calculations with MacCormack's Method

Robert S. Bernard
US Army Engineer waterways Experiment station

Michael L. Schneider
US Army Engineer waterways Experiment station

Follow this and additional works at: http://digitalcommons.unl.edu/usarmyresearch

Part of the Operations Research, Systems Engineering and Industrial Engineering Commons

Bernard, Robert S. and Schneider, Michael L., "Three-Dimensional Incompressible Flow Calculations with MacCormack's Method" (1992). *US Army Research*. 53.
http://digitalcommons.unl.edu/usarmyresearch/53

This Article is brought to you for free and open access by the U.S. Department of Defense at DigitalCommons@University of Nebraska - Lincoln. It has been accepted for inclusion in US Army Research by an authorized administrator of DigitalCommons@University of Nebraska - Lincoln.

Published in Hydraulic Engineering: Saving a Threatened Resource—In Search of Solutions: Proceedings of the Hydraulic Engineering sessions at Water Forum '92.
Baltimore, Maryland, August 2–6, 1992. Published by American Society of Civil Engineers.

Three-Dimensional Incompressible Flow Calculations with MacCormack's Method

Robert S. Bernard and Michael L. Schneider[1]

Abstract

MAC3D is a finite-volume computer code that uses MacCormack's method to calculate three-dimensional incompressible flow on staggered Marker-and-Cell grids. The code accepts nonuniform, nonorthogonal grids for any curvilinear domain that can be mapped onto a single rectangular block. It is applicable for free-surface flow at low Froude number, and for confined flow in general. Computed results from MAC3D are presented for laminar flow in channels with internal obstacles and curved boundaries.

Introduction

Incompressible fluids with constant density lack a time derivative in the equation for conservation of mass, which is given by

$$\nabla \cdot \underline{u} = 0 \tag{1}$$

where $\underline{u}$ is the velocity and ∇ is the gradient operator. Equation 1 is quite different from the momentum equation,

$$\frac{d\underline{u}}{dt} = \nu \nabla^2 \underline{u} - \frac{\nabla p}{\rho} \tag{2}$$

where ν is kinematic viscosity, ρ is density, p is pressure, t is time, and d/dt is the substantive time derivative $\partial/\partial t + \underline{u} \cdot \nabla$.

The absence of d/dt in Equation 1 demands a method

[1]Hydraulics Laboratory, US Army Engineer Waterways Experiment Station, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199.

This article is a U.S. government work, and is not subject to copyright in the United States.

of solution different from that for Equation 2. The only exception is when vertical acceleration is neglected for a flow with a free surface. In that case p is hydrostatic and proportional to the height h of the surface, and Equation 1 then gives way to an equation for dh/dt.

MacCormack's method is an explicit numerical scheme for solving partial differential equations with time derivatives (MacCormack 1969). It has been used quite often for compressible flow, and occasionally for hydrostatic free-surface flow. With minor alterations, it can also be used for confined incompressible flow (Bernard 1986, 1989). In the latter case, staggered Marker-and-Cell (MAC) grids are needed to achieve finite-volume discretization, with pressure and velocity defined at the cell centers and cell faces, respectively.

In the conventional MacCormack scheme, velocity increments are calculated from Equation 2, with the pressure obtained from an equation of state that relates p to ρ or to h. In the altered scheme, however, the velocity increments are first computed without a pressure gradient, and then corrected with a pressure gradient obtained from the solution of a Poisson equation,

$$\nabla^2 p = \rho \nabla \cdot \left(\nu \nabla^2 \underline{u} - \frac{d\underline{u}}{dt} \right) \tag{3}$$

which is discretized in a way that forces the incremented velocity $\underline{u} + d\underline{u}$ to satisfy Equation 1. The altered MacCormack scheme is the algorithm used in the two-dimensional (2-D) STREMR code (Bernard 1989).

The MAC3D code is a three-dimensional (3-D) extension of STREMR. It accepts nonuniform, nonorthogonal grids for curvilinear domains that can be mapped onto a single rectangular block, with internal obstacles mapped onto subblocks (cutouts). The code starts with potential flow and marches forward in time, allowing the flow to develop subject to the boundary conditions and the governing equations. At the discretion of the user, MAC3D imposes any of four different boundary conditions for velocity. For solid boundaries, it offers either a slip condition (zero normal component) or a no-slip condition (zero normal and tangential components). For inflow/outflow boundaries, it offers either a fixed-flux condition (fixed normal component) or a radiation condition (extrapolated normal component). The radiation condition, originally devised by Orlanski (1976), allows internal disturbances to propagate out of the flow field with negligible reflection at the radiation boundaries.

Flow Past a Cylinder

The circular cylinder offers a simple test problem for laminar flow calculations. When the Reynolds number u_0D/ν is greater than 40, vortex shedding occurs for cylinders with height H and diameter D in channels with inflow velocity u_0, depth $H >> D$, width $W >> D$, and length $L >> D$. This gives rise to a 2-D vortex street (periodic vortex shedding) normal to the axis of the cylinder.

In two separate calculations with $u_0D/\nu = 100$ and time-step $\Delta t = 0.05\ D/u_0$, MAC3D was run 2000 time-steps for a cylinder centered $2D$ downstream from the entrance of a rectangular channel with $H = W = 4D$ and $L = 13D$. The flow was from left to right in plan view (Figure 1). Uniform inflow was imposed at the entrance, with a radiation condition at the exit and a no-slip condition on the cylinder. The computational grid (not shown) was 60 cells long, 30 cells wide, and 30 cells deep; the cutout for the cylinder was bounded by four surfaces, each of which was 10 cells wide (or long) and 30 cells deep.

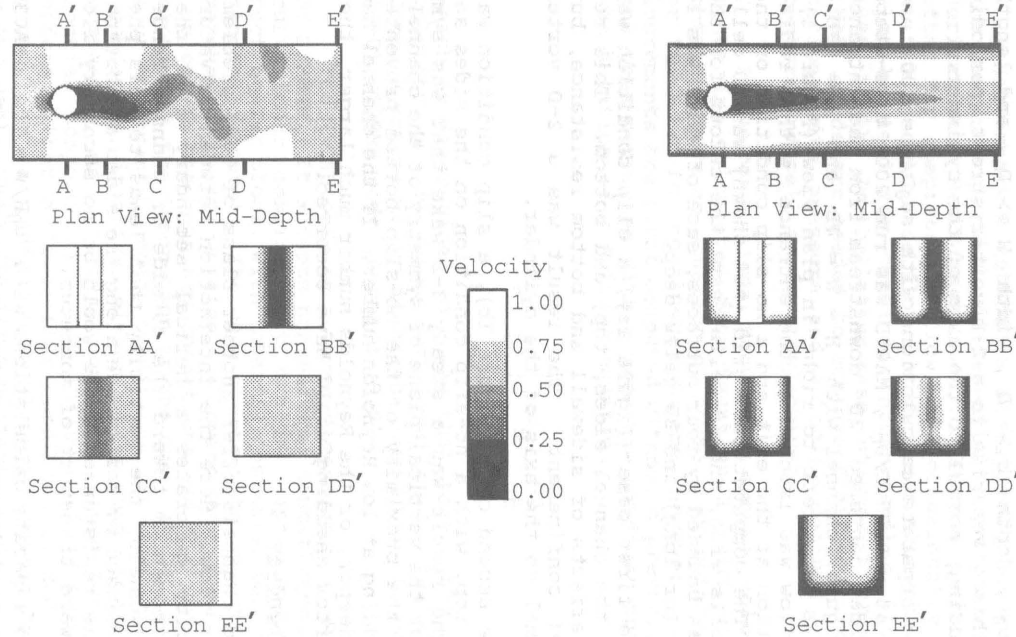
In the first case (Figure 1a), a slip condition was imposed on the channel sides, top, and bottom. This removed the effects of sidewall and bottom resistance, but not channel confinement. The result was a 2-D vortex street normal to the axis of the cylinder.

In the second case (Figure 1b), a slip condition was imposed on top, with a no-slip condition on the sides and bottom. The result was a steady 3-D wake that was symmetric about the vertical plane of symmetry of the channel. Apparently the proximity of the no-slip bottom prevented vortex shedding at low Reynolds number. If the channel had been much deeper, or the Reynolds number much larger, then perhaps vortex shedding would have occurred.

Flow in a Bendway

Curved channels offer another class of test problems for laminar flow. Here the interaction between curvature and vorticity generates a helical secondary flow that transports momentum toward the outside of channel bends. For open channels, the vorticity that leads to this phenomenon is caused by resistance along the sides and bottom. Without this resistance, there would be no secondary flow and no outward transport of momentum.

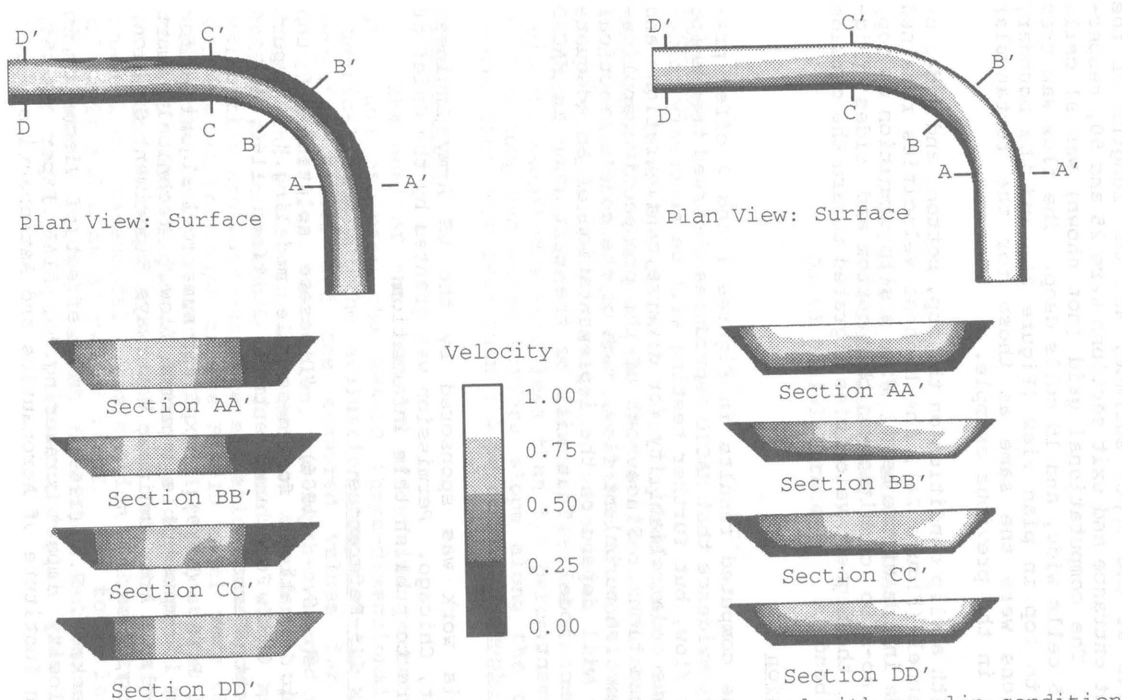
In two separate calculations with $u_0H/\nu = 272$, MAC3D was run to steady state for a 90-degree bendway (Figure 2) with a uniform trapezoidal cross section and a centerline depth H. In multiples of H, the width of the channel



a. Computed with slip condition on bottom and sides of channel

b. Computed with no-slip condition on bottom and sides of channel

Figure 1. Velocity magnitudes for cylinder in rectangular channel



a. Computed with slip condition on bottom and sides of channel

b. Computed with no-slip condition on bottom and sides of channel

Figure 2. Velocity magnitudes for trapezoidal bendway

was 12.46 at the upper surface and 8.46 at the bottom. Likewise in multiples of H , the inner radius of the bend was 30.43 at the upper surface, and the lengths of the straight entrance and exit sections were 25 and 50, respectively. The computational grid (not shown) was 91 cells long, 25 cells wide, and 15 cells deep. The flow was from bottom to top in plan view (Figure 2), and the boundary conditions were the same as those for the rectangular channel in the previous example.

With a slip condition on the top, bottom, and sides of the channel (Figure 2a), the highest velocities remained near the inside of the bend. With a slip condition on top, and a no-slip condition on the bottom and sides (Figure 2b), the highest velocities migrated toward the outside of the bend.

Conclusion

The computed results in Figures 1 and 2 offer preliminary evidence that MAC3D reproduces observed trends for laminar flow, but further testing will be necessary to demonstrate code reliability for diverse configurations and flow conditions. Since most of the prospective applications involve turbulent flow, much of the code's practical utility will depend on the implementation of an adequate turbulence model. That will be the next step in MAC3D development.

Acknowledgement

This work was sponsored by the US Army Engineer District, Chicago. Permission was granted by the Chief of Engineers to publish this information.

Appendix I.--References

Bernard, R. S. (1986). "Discrete solution of the anelastic equations for mesoscale modeling." Report 86/E/51, GKSS Forschungszentrum, Postfach 1160, W-2054 Geesthacht, Germany.

Bernard, R. S. (1989). "Explicit numerical algorithm for modeling incompressible approach flow." Technical Report REMR-HY-5, US Army Engineer Waterways Experiment Station, Vicksburg, Miss.

MacCormack, R. W. (1969). "The effect of viscosity in hypervelocity impact cratering." AIAA Paper 69-354, American Institute of Aeronautics and Astronautics.

Orlanski, I. (1976). "A simple boundary condition for unbounded hyperbolic flows." J. Comp. Phys., 21, 251-269.