

COMPUTATIONAL FLUID DYNAMICS IN A MARINE ENVIRONMENT

Arthur D. Carlson
Naval Underwater Systems Center

INTRODUCTION

The introduction of the supercomputer and recent advances in both Reynolds averaged, and large eddy simulation fluid flow approximation techniques to the Navier/Stokes equations, have created a robust environment for the exploration of problems of interest to the Navy in general, and the Naval Underwater Systems Center in particular. The following discussion will address the nature of problems that are of interest, and the type of resources needed for their solution. The goal is to achieve a good engineering solution to the fluid-structure interaction problem. It is appropriate to indicate that a paper by D. Chapman¹ played a major role in developing our interest in the approach discussed. Further there are several researchers who have pivotal roles in the development of and use of these computational fluid dynamic (CFD) tools. Two of these researchers are Dr. A. J. Kalinowski, who has a leadership role in the large eddy simulation code we are developing, and C. A. Wagner, who has played a similar role in the further development of the Reynolds averaged code INS3D.

NATURE OF PROBLEMS

It is convenient to divide our interests into two classes: the first is where the scale permits a more macroscopic approximation of the flow, i.e., it is not dominated by the boundary layer alone but includes a significant part of the flow outside the boundary layer, the second is dominated by the boundary layer and requires as detailed (as practical today) a description of the flow as possible. Figure 1 illustrates this first class, where we show two bodies in close proximity to one another, perhaps with a relative velocity. The flow is three-dimensional,

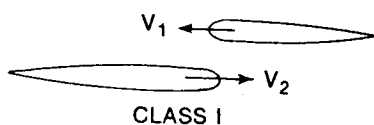
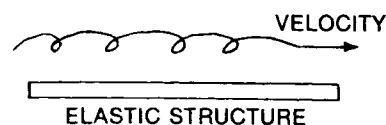


Figure 1.

unsteady, laminar/or turbulent and incompressible.

A second class of problem is shown in Figure 2.



CLASS II

Figure 2.

Here we have an elastic structure that can respond to the boundary layer generated forces, and in turn cause the flow to be influenced by the structural motion. Further the structure may absorb energy and/or be a source of energy. We are interested in as many details of this interaction as are possible to compute.

It appears that a computer code developed at Ames Research Center called INS3D for Incompressible Navier/Stokes in 3 Dimensions will be quite appropriate for class I problems. Reference 2 provides a thorough description of the theoretical and numerical solution techniques employed in INS3D. We are working with this code on a series of laminar problems with viscosity, and for turbulent flow situations. The turbulent solution is being tested for one example against a right circular cylinder for which good experimental data is available. The problems being addressed are quite large and require a machine the size of a CRAY 2 if we are to do practical work. Of course there is no difficulty in thinking of problems that would require N number of CRAY 2's, but for this point in time good insights can be achieved with a CRAY 2. Because the pressure field is important to the Navy, INS3D is very useful since it calculates pressure explicitly. The equation for this approximation is;

$$\frac{\partial P}{\partial t} + \beta \nabla \cdot \vec{V} = 0$$

where P is pressure, t is time, β is constant and V is velocity.

One effect not considered is any coupling of INS3D with the elasticity of a structure.

To examine class II problems NUSC is developing its own code called TURBO (Turbulent Unsteady-flow over Resilient Boundaries). TURBO is being written for a CRAY 2 environment, at first for single CPU, later as a multiple CPU code. An important feature in TURBO is the

coupling of the three-dimensional Navier/Stokes equations to a viscoelastic structure. Following is the order of development of the code and some of its details.

. Version I - The complete Navier/Stokes equations with no modelling are coupled to an orthotropic viscoelastic plate. An analytical solution is used for the plate.

. Version II - The complete Navier/Stokes equations with no modelling are coupled to a three-dimensional viscoelastic structure. The structure is represented by finite elements.

. Version III - A turbulence model in conjunction with the large eddy simulation approximation is used to model the fluid. The structure is viscoelastic and represented by finite elements.

SOLUTION METHODOLOGY

. The fluid equations are transformed by a double Fourier transformation in the streamwise and spanwise direction into a wave number space. This is so we can sample the computational space with fewer computational points, and secondly automatically satisfy periodic boundary conditions. In the vertical direction a closed form solution to the differential equation is used. In principle this should allow us to resolve the flow character near the wall to a small scale.

. Provisions have been made for compressibility effects in the fluid. This is achieved by a formula of the form:

$$\text{Change in Pressure} = \text{Constant (Change in Density)}$$

. Periodic boundary conditions are assumed in the streamwise and spanwise direction. Currently in the vertical direction there is a rigid boundary at the top surface. Research is underway to lift this restriction and provide for a free field.

. At the bottom surface, the very complicated, but very critical, coupling of the flow equations to the flexible structure surface employs an arbitrary Eulerian/Lagrangian coordinate system. This system in effect requires that we observe, i.e., compute the solution along a predetermined, prescribed path. We perform a coordinate transformation that in effect transforms the deformed fluid mesh into a regular mesh in which the fluid equations are solved. Then the new updated forces and fluid displacement are transformed back and applied to the structural surface. The finite element method is then applied to the structural domain and a finite element solution methodology is carried out there.

. Work is currently also underway to use more than a single CPU for the solution of TURBO. The initial version though will use a single CPU.

PRE AND POST PROCESSING OF MATH MODEL

It is imperative that sophisticated graphics be employed if a comprehensive analysis is to be carried out in a production type mode, utilizing rapid turnaround and user friendliness. To this end we have acquired a Silicon Graphics 3030 color workstation along with the GAS (Graphics Animation System) and PLOT3D software developed at NASA Ames. This software enables rapid analysis of large CFD data sets along with automated movie making for visualization and presentation of interesting results. The movies are recorded in 16mm format and then translated to videotape.

OPTIMIZATION

The mathematics of optimization of an N dimensional space, and the computer programming for it, has reached the state where it is reasonable to explore the use of such techniques in conjunction with codes like INS3D and TURBO. Vanderplaats³ gives an excellent outline of the optimization method. To be successful from an engineering point of view will require judicious use of optimization codes and very good estimations of a set of initial designs. Using the structures shown in Figure 3 as an example one can pose the following problem: What shape body will provide a minimum pressure distribution when

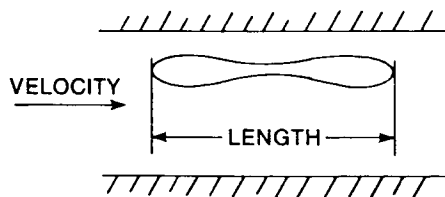


Figure 3.

subject to a set of constraints such as a fixed length and some minimum volume? The problem here is that there is no explicit way to write a relationship between drag and the body dimensions. Hence we start with an initial set of shapes that work "well" and iterate using an approximate search for the minimum objective function. An example of an optimization code that could be combined with INS3D is ADS (Automatic Design Synthesis) and is described in reference 4. This code has software for a number of solution strategies such as sequential linear programming, optimizer options such as the modified method of feasible directions method for constrained minimization, and a number of search options such as a technique to find the minimum of a constrained function using the golden sections method.

There is no guarantee that a good solution can be reached in a realistic time given the computer resources at this particular point in time, but if not today, almost certainly in the near future.

OCEAN DYNAMICS

It appears that INS3D has the necessary ingredients and flexibility for its application to questions involving ocean dynamics. To use it several key issues will have to be resolved: First one must include coriolis forces to account for the earth's rotation effects on the fluid, second the boundary condition at the ocean-air interface must be accounted for, and third the ability to account for variations in density within the fluid, and ultimately with time, will have to be made. The other boundary conditions will depend on the depth of water and the lateral extent over which the model is to be applied on a case by case basis. Our solution strategy is to start with as simple a model of the ocean as possible, and progressively build those characteristics into INS3D, verify these with an experiment(s), and proceed with further complexities. Because we have a good idea of the parameters we are looking for, our efforts will be directed to adding, within an engineering model, just enough to get working insights.

TESTING

The ultimate measure of a computed design is proof by test and then by field use. Several water tunnel test facilities exist today which can be used but these will most likely have to be expanded not only in size, but in the number and accuracy of velocity and pressure sensors that are used. Smaller pressure sensors and three-dimensional velocity measurements at many points simultaneously will be required. Laser velocimetry techniques are useful here.

CONCLUSIONS

Order of magnitude improvements in the ability to separate out cause from effect in complex flow situations is about to occur in important problems. The pacing item is the computer, although such esoteric areas as turbulence modelling will provide significant challenges. As one moves into production use of these new tools, pre and posts processing of the fluid and structure models will also require further improvement.

The immediate future will be devoted to proving out the new CFD tools by both numerical checks against analytical solutions, as well as careful experimental checks. One can think of the product of numerical solutions on supercomputers as providing a numerical microscope with which to view various flow phenomena. What ultimately remains then is the creative use of such tools to solve real problems.

REFERENCES

1. Chapman, Dean R.: Computational Aerodynamics Development and Outlook, AIAA Journal, vol. 17, no. 12, Dec. 1979, pp. 1293-1313.
2. Kwak, Dochan; and Chakravarthy, Sukumar R.: A Three-Dimensional Incompressible Navier-Stokes Flow Solver Using Primitive Variables, AIAA Journal, vol. 24, no. 3, Mar. 1986, pp. 390-396.
3. Vanderplaats, Garret N.: Numerical Optimization Techniques for Engineering Design: With Applications, McGraw-Hill Book Co., Inc., 1984.
4. Vanderplaats, Garret N.: COPES/ADS - A Fortran Control Program for Engineering Synthesis Using the ADS Optimization Program, Engineering Design Optimization, Inc., 1984.