TECHNICAL BRIEF:
Selected Benchmarks from Commercial CFD Codes

For a large portion of the engineering community, the primary source of CFD capabilities is through the purchase of a commercial CFD code. Typically, the selection of a commercial CFD code is based on discussions with a vendor's sales or marketing staff and on demonstration of sample simulations (often idealized to highlight features of the code). This is usually done in isolation in which one does not have the opportunity to see several different commercial codes at once, solving the same problem. Therefore, the Coordinating Group for Computational Fluid Dynamics, of the Fluids Engineering Division of ASME, sponsored and organized two unique Forums, allowing for such an evaluation to be made. These forums were; The CFD Triathlon: Three Laminar Flow Simulations by Commercial CFD Codes, held at The Fluids Engineering Conference, Washington DC, June 20-24, 1993; and, The CFD Biathlon: Two Turbulent Flow Simulations by Commercial CFD Codes, held at The Fluids Engineering Conference, Lake Tahoe, Nevada, June 19-24, 1994. This Tech Note then, summarizes the results of both of these Forums and attempts to present conclusions in an unbiased form. A complete description of the results is given in the article forming the basis of this summary: *"Perspective: Selected Benchmarks From Commercial CFD Codes,"* Journal of Fluids Engineering, Vol. 117, June, 1995 by C. J. Freitas.

Objectives

In these Forums, only commercial CFD codes were allowed to participate. The definition of a commercial CFD code used here is that the code is the product, not the consultation resulting from the used of a code, and that there is a marketing and user support staff for the code. Also, both laminar and turbulent flows were chosen here to evaluate the codes.

The objective of the Forums was to provide a common point of reference to the engineering community for the quantitative evaluation of commercial CFD codes. In addition, it was also the intent of the Forums to provide a qualitative evaluation of the insights of and the capabilities that each of the vendors bring to the solution process. This was felt to be important in that a user would more than likely have to rely on the vendor for aid and support sometime during the license period of the code in order to solve highly complex problems. Therefore, the participants were asked to describe in some detail their method of problem definition, problem solution, and problem analysis.

Methodology

The forums were organized through letters of invitation, inviting commercial CFD code vendors to participate. Once a vendor accepted the invitation to participate, a problem definition statement for each problem was sent to them. In the first Forum, the benchmark problems consisted of three different laminar flows; i.e., a steady, two-dimensional laminar flow (low Reynolds number flow over a backward-facing step); an unsteady, two-dimensional laminar flow (low Reynolds number flow around a unit cylinder); and an unsteady, three-dimensional laminar flow (shear-driven cavity flow). In the second Forum, two different turbulent flow problems were defined; i.e., an unsteady, two-dimensional turbulent flow (flow around a square cylinder), and a steady, three-dimensional turbulent flow (spatially-developing flow in a 180-degree bend). Each vendor was then given approximately five months to perform the simulations and write a summary paper. The names of the vendors and codes, which participated in one or both of the Forums, are listed in Table 1. Although twenty vendors were invited to participate in these Forums, only five vendors completed each exercise, with only two vendors completing both exercises (with a total of eight vendors participating in these Forums).

Participants

Vendor & Code Names Who Participated in the Triathlon & Biathlon Forums

(CFD Triathlon = T, CFD Biathlon = B).
Company or vendor name--Code name

- CFD Research Corp CFD-ACE-B
- Computational Dynamics LTD STAR-CD-B
- Computational Fluids Dynamics Services CFDS-FLOW3D-T
- Engineering Mechanics Research Corp NISA/3D-FLUID-T
- Flow Science FLOW-3D-T
- Fluent Inc. FLUENT-T,B
- Scientific Services, Inc. (SIMULOG) N3S-B
- Swanson Analysis Systems FLOTRAN-T, B

The CFD Codes

A brief review of the capabilities of each of the codes used to complete either the CFD Triathlon simulation series or the CFD Biathlon simulation series are now given. These codes are presented in no specific order, and the reader is to infer nothing by the order of presentation. In the presentation of each code, the header line gives the name of the code, the company name, and the people involved in performing the simulations.

FLOW-3D, Flow Science Inc., J. Sicilian, J. Ditter, C. Bronisz. FLOW-3D is a finite-difference, transient-solution algorithm solving the conventional, conservation equations of fluid dynamics. It is based on the technology of SOLA-VOF (Hirt and Nichols, 1981). However, it is a completely independent implementation of these techniques. The principal algorithms are all based on a combination of finite difference and finite volume perspectives. SOLA is a technique for the solution of the time -dependent flow equations in primitive variables, solved on a staggered grid system. The grid used is nonuniform and Cartesian (polar/cylindrical coordinate system is also available) in one, two or three dimensions. The VOF algorithm (Volume Of Fluid) tracks the movement of fluid by the calculation of fluid fraction for each mesh cell. FLOW-3D incorporates physical models for porous media flow, conjugate heat transfer, dynamic fluid-interface motion, surface tension, wall adhesion, turbulent flow, incompressible and compressible flow, non-Newtonian fluids, non-inertial reference frames, solidification and melting, thermal buoyancy, and simplified bubble models. FLOW-3D models turbulent flows with either a Prandtl mixing length model, a standard two-equation k-e model, an RNG model (renormalized group theory model, an extension of the standard k-e model), or a Large Eddy Simulation model with a Smagorinsky sub-grid scale model. Finally, FLOW-3D uses either a first-order method or a second-order method of space and time derivatives. In all simulations completed with FLOW-3D in the CFD Triathlon exercise, the monotonicity-preserving, second-order spatial-differencing scheme was used to discretize the convection terms.

FLOTRAN, Swanson Analysis Systems, Inc., T. Chopin, D. Ganjoo, E. Underwood. FLOTRAN is a finite-element based, general-purpose algorithm which solves the Navier-Stokes and energy equations using a segregated or sequential solution method. The velocity-pressure formulation uses an equal-order approximation for velocity and pressure, and solves for each variable field in an iterative manner similar to finite volume methods (Schnipke and Rice, 1986). This results in a method which is bandwidth independent and thereby requires significantly less memory than traditional finite-element methods. FLOTRAN uses a monotone streamline upwind technique to discretize the advection terms and has demonstrated improved accuracy of conventional upwind methods for finite elements. The effects of periodic boundary conditions, porous media flow, distributed resistances, moving walls, conjugate heat transfer, thermal buoyancy, turbulent flow, incompressible and compressible flow, and rotating reference frames may be simulated. FLOTRAN models turbulent flow with the standard two-equation k-e model. In the simulations completed with the FLOTRAN code in both the CFD Triathlon and CFD Biathlon exercises, the monotone, streamline upwind method was used to model the convective terms, which demonstrated greater than second-order accuracy.

STAR-CD, Computational Dynamics LTD, R. Issa, R. Benodekar, R. Sanatian, S. Uslu. STAR-CD is a general purpose, finite-volume algorithm which uses an unstructured grid system to resolve the conservation equations. The unstructured grid allows for a range of optional cell shapes. These cells may exhibit arbitrary deformation, have sliding internal interfaces, and permit cell insertion and deletion. In addition, local mesh refinement may be utilized to locally enhance accuracy of solutions without encumbering the global solution. Extended versions of the SIMPLE (Patankar and Spalding, 1972) and PISO (Issa, 1983) algorithms are used for steady state and transient calculations, respectively, solving all variables on a collocated grid system; i.e., all variables are at the cell center, including the Cartesian velocity components. Spatial differencing is second-order and a fully implicit first-order temporal differencing scheme is used. STAR-CD models turbulent flow using a number of different Reynolds-averaged turbulent models, in particular, the standard two-equation k-e model, an RNG model, and a two-layer variant of the k-e model in which the Norris and Reynolds (1975) one-equation, low Reynolds number model is used in the near-wall region. In general, STAR-CD has extensive flow, heat and mass transfer capabilities, including compressible, multiphase and chemical-reacting flows. In the simulations completed with the STAR-CD code in the CFD Biathlon exercise, the self-filtered, second-order spatial differencing scheme was used.

N3S, SIMULOG/Scientific Services, Inc., J. Canu, C. Fletcher, G. Blankenship. N3S was created to address the solution of problems in fluid dynamics with complex, three-dimensional geometries. N3S is based on the finite element method and uses an unstructured grid topology. It solves the time-dependent Navier-Stokes equations and energy equation using a velocity/pressure formulation. Time discretization is performed by an operator splitting method in which the convection step is calculated using a characteristics method (providing a natural up winding) and the diffusion or Stokes step is calculated by an implicit Euler scheme. Within this technique, first and second-order schemes are implemented. The resulting Stokes problem is discretized in space using triangular elements in two dimensions or tetrahedra in three dimensions and is solved using a preconditioned Uzawa algorithm. N3S incorporates models for multiphase flow, reacting flow, moving boundaries, and an adaptive meshing routine enhancing solution accuracy locally. N3S models turbulent flow with the standard k-e model and a variant of it based on the work of Kato and Launder (Launder et al., 1975). In the simulations completed with the N3S code in the CFD Biathlon exercise, the second-order form of the solver was used.

CFD-ACE, CFD Research Corp., R. Avva, Y. Lai, A. Singhal. CFD-ACE is an advanced, general-purpose CFD code with multi-domain solution capabilities. It is based on a strongly conservative finite-volume formulation using nonorthogonal curvilinear coordinate systems with a structured, collocated grid arrangement. A fully implicit, patched multiblock solution procedure is used which accounts for moving grids, sliding grids, and rotating coordinate systems. The solution algorithms are based on variants of SIMPLEC and PISO, using advanced linear equation solvers including a preconditioned, Conjugate Gradient Squared algorithm and a Symmetric Strongly Implicit procedure. CFD-ACE has models for incompressible and compressible (subsonic to hypersonic) flows, thermal buoyancy, conjugate heat transfer, radiative heat transfer, variable physical properties, mass transfer with multi-component diffusion, solidification/melting, gaseous combustion, and spray dynamics. CFD-ACE models turbulent flows using a variety o f models, in particular, the standard k-e model, RNG model, and a two-layer k-e model (Rodi, 1991). In the simulations completed with the CFD-ACE code in the CFD Biathlon exercise, a second-order central spatial-differencing scheme was used.

FLUENT, Fluent Inc., D. Choudhury, S. Kim, D. Tselepidakis. FLUENT solves the governing conservation equations of fluid dynamics by a finite-volume formulation on a structured, non-orthogonal, curvilinear coordinate grid system using a collocated variable arrangement. Three different spatial discretization schemes may be used; i.e., Power-Law, second-order upwind, and QUICK (Leonard, 1979), a bounded third-order accurate method. Temporal discretization is achieved by a first-order, implicit Euler scheme. Pressure/Velocity coupling is achieved by the SIMPLEC algorithm resulting in a set of algebraic equations which are solved using a line-by-line tridiagonal matrix algorithm, accelerated by an additive-correction type of multigrid method and block-correction. Additional equation solvers are also available to the user. FLUENT models turbulent flows with the standard model, an RNG model, and a second-moment closure or Reynolds-stress model (RSM). In general, FLUENT includes models of single or multiphase flow, with heat transfer and chemical reactions for incompressible and compressible flows. In the simulations completed with the FLUENT code in both the CFD Triathlon and CFD Biathlon Forums, the convective terms were discretized with a bounded, third-order accurate QUICK scheme.

CFDS-FLOW3D, Computational Fluid Dynamics Services, Inc., S. Simcox, H. Pordal, M. Nieburg. CFDS-FLOW3D is also a finite-volume based code using a structured, patched multi-block, nonorthogonal, curvilinear coordinate grid with a collocated variable arrangement. The basic solution algorithm is the SIMPLEC pressure correction scheme which uses a variety of linear equation solvers. Spatial discretization is achieved through the HYBRID scheme, a second-order upwind scheme, and the third-order QUICK scheme. CFDS-FLOW3D has models for multi-phase flow, particle transport, gaseous combustion, chemical species concentration, thermal radiation, compressible and incompressible flows, porous media flow, and conjugate heat transfer. Turbulent flows are modeled with five different closure methods, i.e., the standard k-e model, a low-Reynolds number k-e model, an algebraic stress model, a differential Reynolds stress model, and a differential Reynolds flux model. In the simulations completed with CFDS-FLOW3D in the CFD Triathlon exercise, a combination of HYBRID differencing and second-order upwind differencing for the convective terms was used. The third-order QUICK scheme was used to simulate the flow over a backward-facing step.

NISA/3D-FLUID, Engineering Mechanics Research Corp., K. Bhatia, M. Rahman, B. Agarwal. NISA/3D-FLUID is a finite element based code used to solve the governing conservation equations of fluid dynamics. It uses a Galerkins' approach to discretize the equation system and eliminates pressure through the use of a penalty function. In general, NISA/3D-FLUID has models for heat transfer, radiation heat transfer, non-Newtonian flows, compressible and incompressible flow, turbulent flows, flow through porous media, rotating reference frames, phase change, free surface flows, chemically reacting flows, fluid-solid interaction, and stress analysis. In the simulations completed with the NISA/3D-FLUID code in the CFD Triathlon exercise, linear finite elements were used to resolve the flow fields.

Findings

A series of five benchmark problems were calculated using several commercial CFD codes, all simulations performed by the vendors themselves. The first three problems dealt with laminar flows of varying degrees of complexity. In general, the codes that completed the first two simulations, one, a steady two-dimensional laminar flow, and the second, an unsteady two-dimensional laminar flow, gave acceptable results, when the codes were applied to the problems correctly. However, what the author found most discouraging with the solution of these two "simple" problems was the general lack of insight brought to the solution process by some of the vendors. That is, some vendors were satisfied with solutions generated at a single grid resolution, never attempting to demonstrate a grid convergent solution. With the new policy statement on numerical uncertainty that the Journal of Fluid Engineering has invoked (Freitas, 1993), these results would not have been accepted in an archival publication. Further, with the demonstrated sensitivity of these two solutions to the accuracy of the spatial discretization, it was surprising that some vendors used first-order methods in the solution process. Again, a prediction based on such first-order methods is no longer considered acceptable for inclusion in the Journal of Fluids Engineering.

The third benchmark problem, the unsteady, three-dimensional flow in a cavity, mandates for a successful solution the use of a spatial discretization scheme that minimizes or eliminates numerical diffusion. With numerical diffusion present, the moderate scale, secondary motions never develop, i.e., the Gortler vortices. This was clearly demonstrated in some of the simulations performed here, when the Gortler vortices were not predicted or sufficiently resolved by the grid. And again, some of the vendors simply did not demonstrate having any insights into the solution of the problem, and no attempts were made to show a grid convergent solution by any of the vendors.

In performing the two turbulent flow benchmark simulations in this exercise, the vendors did demonstrate greater insight to the problem solution and, in some cases did solve the problems on multiple grids with significantly different resolutions. The two turbulent flow benchmarks used here, the two-dimensional flow around a square cylinder, and the three-dimensional flow in a 180-degree bend, taxed both the numerics of the codes and the turbulence models of the codes. With regard to the turbulence models, the current state-of-the-art in commercial CFD codes are the standard k-e model and its variants (primarily RNG forms). Unfortunately, many of the problems for which commercial CFD codes are applied to, push the limits and assumptions upon which these models are based. With regard to the discretization methods, the current state-of-the-art is second-order schemes which may prove to still not be accurate enough for some classes of flow; for example, the flow through a 180-degree bend. Historically, it was believed that in spite of the limitations of the models and methods used that a series of simulations would at least resolve the trends as key parameters were varied and certainly for classes of problems this is true. However, as demonstrated by comparing the same solution using the standard k-e model, the RNG model, and the Reynolds Stress model, details of the flow field may be widely different even while the mean axial velocity profiles are similar. The bottom line is that the user of these codes must remember to balance grid resolution and discretization accuracy, and use the appropriate closure model.

In the course of this exercise, it was suggested by some vendors (who selected not to participate in the forum) that these two turbulent flow problems were not appropriate as benchmarks, since they potentially violated the conditions upon which traditional turbulence models are based. However, these commercial CFD codes are being used today by industry, to study similar flows. The author has been at vendor demonstrations in which vendor staff members have claimed the ability to solve these problems. In addition, with some commercial codes using new types of turbulence models such as the RNG extension to the model and other nonlinear turbulence closures, it is relevant and appropriate to attempt these types of benchmarks. Clearly demonstrated in this exercise, is that further research into more advanced turbulence models for use in commercial CFD codes is required (not a big surprise). And, implementation of higher-order discretization schemes may also be required. These are not unreasonable demands; especially when one considers that both of these turbulent flow problems are essentially, idealizations of typical, industrial flow situations; i.e., one an external flow and the other an internal flow.

References

- Freitas, C. J., 1993, "Policy Statement on the Control of Numerical Accuracy," ASME Journal of Fluids Engineering, Vol. 115, No. 3, pp. 339-340.

- Hirt, C. W., and Nichols, B. D., 1981, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries," Journal of Computational Physics, Vol. 39, pp. 201.

- Issa, R. I., 1983, "Solution of the Implicitly Discretized Fluid Flow Equations by Operator-Splitting," Imperial College of Science & Technology, Report FS/82/15, Sept.

- Launder, B. F., Reece, G. J., and Rodi, W., 1975, "Progress in the Development of a Reynolds-stress Turbulence Closure," Journal of Fluid Mechanics, Vol. 68, pp. 537-566.

- Leonard, B. P., 1979, "A Stable and Accurate Convective Modeling Procedure Based on Quadratic Upstream Interpolation," Computer Methods in Applied Mechanics and Engineering, Vol. 19, pp. 59-98.

- Norris, L. H., and Reynolds, W. C., 1975, "Turbulent Channel Flow with a Moving Wavy Boundary," Stanford University, Report No. FM-10.

- Patankar, S. V., and Spalding, D. B., 1972, "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," International Journal of Heat and Mass Transfer, Vol. 15, pp. 1787.

- Rodi, W., 1991, "Experience with Two-Layer Models Combining the k-e Model with a One-equation Model Near the Wall," AIAA-91-0216.

- Schnipke, R. J., and Rice, J. G., 1986, "A Streamline Upwind Finite Element Method for Laminar and Turbulent Flow," Department of Mechanical and Aerospace Engineering, University of Virginia, Report No. UVA/643092/MAE86/342.

Further information can be obtained from the author :

Dr. C. J. Freitas
Principal Engineer
Computational Mechanics Section
Southwest Research Institute
6220 Culebra Rd.
San Antonio, TX 78228-0510
Email: cfreitas@swri.edu