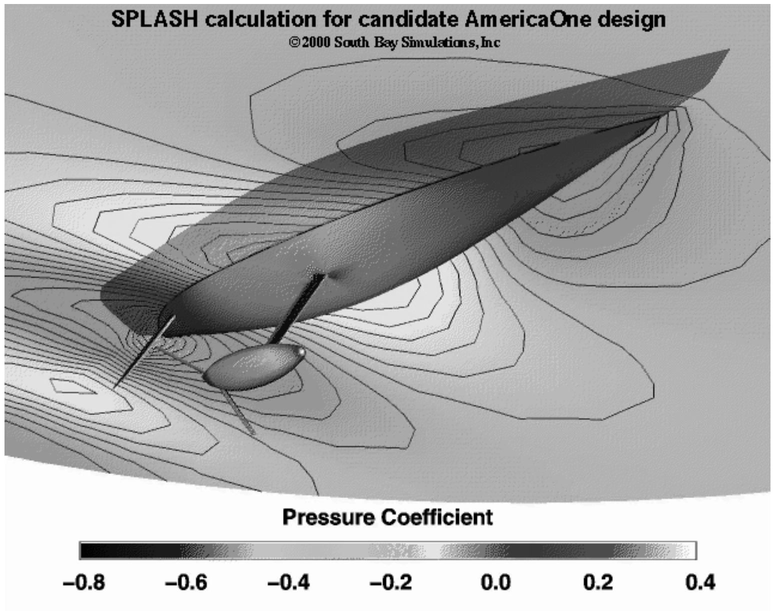


Computational Fluid Dynamics – or How to Make a Good Boat Fast

David Vacanti

The term CFD is showing up more often these days in articles describing the design efforts used to make Volvo 60 round the world racers and America's Cup yachts faster. Computational Fluid Dynamics or CFD actually covers a great many engineering specialties and is not the sole domain of boat and ship design. In this article we will review what types of CFD products exist and hopefully provide some understanding of when and how CFD products are best suited to a project.

Computational Fluid Dynamics is the application of computers to the modeling of fluid characteristics when either the fluid is in motion or when an object disturbs a fluid. A few examples of a fluid in motion are water or chemical flow in pipes, heating and ventilation systems conducting cooling, heating or fresh air supplies to a building. Fluids in motion also include flame and fire effects in combustion or jet engines. Surprised by these fields of interest?

What about examples of an object disturbing a fluid? Examples include stirring paddles submerged in a tank of water and effluent in a waste treatment plant, aircraft of all kinds, cars and trucks at highway or racing speeds and even monohull sailboats, ship, multihull sailboats to name but a few.

Obviously, an open mind is important when considering what constitutes a fluid. Fluids can exist in gaseous and liquid states and science has recently found that even some solids can exhibit fluid like characteristics under right conditions. Scientists have found that some of the most spectacular and deadly landslides or rock falls behave as a fluid while the mass of stone and soil or sand is in motion, only to return to a most decidedly solid form when the motion subsides.

The general field of fluid dynamics differs from the field of boat design in one critical way. Only boat design deals with a vehicle passing through the two fluids of air and water simultaneously.

Our atmosphere is a compressible fluid, though not at yachting or even high-powered boat racing speeds. Air can change in density according to altitude, temperature and humidity. Water is an incompressible fluid that can vary in viscosity according to its salinity and temperature.

For most of us, small effects such as variable salinity and temperature are not of concern, but can make the difference between winning and loosing a major international yacht race.

How do CFD programs Work?

CFD programs are based on the laws of physics, such as the law of conservation of momentum, and special “boundary conditions”. The law of conservation of momentum states that the total momentum of a system remains constant regardless of how the system may change. A boundary condition limits how and where a fluid can travel. A simple example is that motion of the fluid must remain tangent or parallel to the surface of an object passing through it. Another example is that pressure applied by the fluid against the object must be perpendicular to the surface at all points.

These laws and conditions are critical to the development of a CFD program because they allow an aerodynamicist to write equations that describe the system that is being studied. Without the physical laws and boundary conditions there would be no way to write equations that describe fluid motion. The complex equations that result take into account the viscosity, mass and other characteristics of the fluid. The equations are written using integral and differential calculus and require specialized computer techniques to solve them. Typically the programmer writes an algorithm that makes a series of estimates using algorithms that iteratively solve the sets of equations by looking for “balance” in the system of equations. A final answer is obtained when the algorithm converges on a solution with an error that is sufficiently small for the desired accuracy.

Once an algorithm has been developed to implement the laws of momentum and boundary conditions, it cannot be applied to the entire surface of the hull and appendages at once. The surface area of the hull, keel and rudder are broken into thousands of small patches (collectively called a mesh) and the algorithm applied to each patch. Each patch in turn influences the fluid flow on the patch area of its neighbors and therefore the solution must account for the conditions surrounding the patch currently being solved. As a result the program must solve and resolve the equations for all of the patches until the solution obeys the physical laws and boundary conditions.

Sometimes the complexities of the laws of physics are too difficult to implement all at the same time. As a result the aerodynamicists choose to write programs that make certain limiting assumptions that permit the programming to become more practical and still result in reasonable results. A specific example arises in the case of what actually happens to fluids very near the surface of an object. The boundary layer as it is called experiences shear forces in the objects direction of travel that result in viscous drag. These shear forces are described in a special set of equations called the Navier Stokes relationships. The Navier Stokes equations are sufficiently complex themselves that attempting to include them within every aerodynamics or hydrodynamics program would make the solutions nearly impossible. As a result there are Navier Stokes based programs that specifically address viscous drag and Panel method programs that compute lift, wave drag and induced drag. A complete estimate of the drag encountered by a boat requires the data supplied by both programs.

What do CFD programs Calculate?
The most obvious calculation that would be of interest in boat design is the determination of drag forces. But drag comes in several forms that can include, wave, viscous, and induced drag. Therefore, a designer must evaluate the effects of his design in each of these drag areas. The second general area of calculation is lift. The term lift arises from its application to aircraft and becomes a bit confusing when applied to the field of boat design. Lift applies to the forces generated by a keel or centerboard to resist the side force of sails and the driving force of the sails themselves. It also applies to the turning forces of a rudder, and the supportive force acting on “foils” to elevate a hydrofoil sail or powerboat above the water surface.

There is also a distinction between 2 and 3 dimensional fluid dynamics analysis. Specifically, there are programs that predict the performance of foils as if they existed on a wing of infinite length. Here the term “foil” is used to define the shape of a keel or rudder along the chord from the leading to the trailing edge. Foil shapes are best known by the alphanumerical names given to them such as NACA 63A012. So a 2D fluids program would compute the lift, drag, velocity distribution, turbulence onset and the generation of bubbles similar to cavitation for a 2D shape such as a wing or keel foil, and would not include any 3D information such as keel span or thickness distribution or the presence of a bulb. A 3D fluids program would compute wave and induced drag from a hull, keel and rudder, including the effects of a bulbed keel carrying winglets.

CFD codes are critical for more than optimizing the performance of a top-notch America’s Cup class racing yacht. These codes can be of great value to determine loads placed on boat structures of all types and are invaluable when applied to unique marine structures such as oil platforms that are frequently subjected to the world’s worst storms.

Lift and drag effects translate directly into loads that must be sustained by the boat or oil derrick if it is to remain intact in its intended operational conditions. For example, several years ago when the race was known as the Whitbread Round The World Race, many boats developed life threatening hull delamination when subjected to the continuous pounding of high speed downwind surfing and upwind beating. While delamination of a boat at sea is definitely related to structural design errors, those errors were caused by a lack of detailed information about the fluid forces experienced by the boats. Knowledge of these forces would have enabled designers to prevent the hull damage in the first place.

Therefore, the potential application of CFD to your design project should depend on whether or not the design regime that your vessel will operate in has well understood engineering data available to prevent hull damage in addition to overall performance of the vessel. For example, the last few years have seen the development of high-speed hydrofoil sailboats for the consumer market. These top performance boats experience not only significant speeds and loads, but the potential for unstable characteristics could make it highly dangerous to ride in one. However, the judicious use of field-testing and computer analysis has produced a crop of very exciting hydrofoil sport boats that are a joy to fly in.

Finally, several years ago a multihull sailboat arrived in port after participating in a trans-Atlantic race. When the centerboards were raised in the outer hulls of the trimaran, the skipper was shocked to learn that the boards had been sheered off just below hull depth and he had not had their use for some indeterminate time during the later portion of the race. Clearly, the structural design of the boards had not taken into account the true forces of lift, drag or perhaps cavitation that would be experienced at sea.

CFD programs do not calculate how fast a boat of any type will pass through the water or predict the time to complete a course around the buoys. Predicting speed on a racecourse is the domain of another class of programs called Velocity Prediction Programs or VPP. The VPP makes use of lift and drag numbers calculated in a CFD program to estimate the speed about will sail a course given the sail drive forces and the stability or righting moment of the hull. The VPP is a closed loop simulation continuously varies estimated speed and resulting lift, drag and righting forces until retarding and driving forces are balanced and a stable speed results. A CFD program on the other hand is an open loop simulation that simply states that given an angle of heel and speed for a specified hull and appendage configuration, here are the forces that will result for that instant in time. No consideration is given to how the vessel achieved that speed or sailing condition.

In summary then, CFD programs not only calculate lift and drag forces of a hull with appendages, they can also be used to compute pressure loads due to waves and wave impact at speed. The forces of lift, drag and pressure can be translated into structural requirements and provide the means to optimize a hull working in concert with its appendages to produce lift in the most efficient manner possible while satisfying the needs for stability. Predictions of lift and drag at various speeds can be used to develop a mathematical model needed to accurately close the analysis loop of a velocity prediction program.

When is a CFD Computer Program Required?

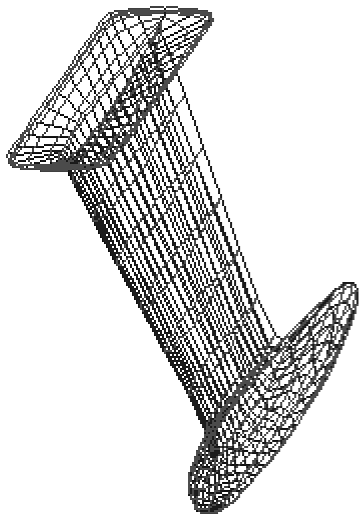
CFD codes are not always required or justified however, when simpler means of estimating the forces involved are available. In the case of a typical sailboat design, the forces generated by the keel and rudder can be easily estimated if the keel lacks a bulb and if the keel and rudder shape are essentially straight leading and trailing edges. It is possible to make use of analytical methods that are easily implemented on personal computers. A simple example is the program I wrote called LOFT that makes use of analytical methods developed by NASA and the US Air Force for initial performance prediction of wing designs.

However, while simple programs like LOFT can adequately address typical bulb-less keels and rudders they cannot analyze the performance of an America’s Cup racing keel with bulb and winglets. Only 3D CFD programs can address that complex task.

Who can operate a CFD program?

While CFD programs can be of tremendous value, getting accurate and meaningful results is not typically within the reach of amateur and many professional boat designers. A degreed Naval Architect or a fluids dynamicist is required to generate the key initial input to a CFD program called a mesh.

The mesh is a mathematical description of the hull and appendages that are to be analyzed. It is not sufficient or even possible to use standard stations, waterlines and buttocks as inputs to a CFD program. The detailed shapes of the hull and appendages must be defined by a mesh of square patches that adjoin one another and whose dimensions are chosen according to the local curvature of the hull or appendage or by the occurrence of the intersection of the hull and a keel, rudder or lifting strut of a hydrofoil. The generation of a mesh is a science unto itself and can require iterations by the analysts running successive trials to be sure that the mesh is sufficiently dense in critical areas. Some meshing can be done automatically and then refined by hand.



Typically the developers run complex 3D CFD programs or those trained in their use and as result are not really meant for use by the rest of us. However, 2D fluids programs designed for the analysis and development of 2D air or hydrofoil shapes (recall the 63A010 example) are sufficiently easy to use for a designer with basic mathematics skills and general knowledge of airfoil characteristics. Analytical programs such as Vacanti Yacht Design's FOIL program can aero / hydrofoil lift, drag, turbulence onset and bubble formation characteristics for anyone with basic computer skills and a working knowledge of basic foil design.

What CFD Programs Exist?
Panel Method and Navier Stokes programs are two general classes of CFD programs that apply to the issues of boat design. The most commonly used and most available are Panel Method programs. Panel methods allow the prediction of wave drag, free surface effects and induced drag due to lift generated by a keel or rudder but they do not account for viscous drag. Programs using panel methods assume that there are only forces normal to the surface of the hull within the fluid. However, due to viscosity, the fluid is subject to forces in shear – more or less parallel to the hull surface that causes turbulence. Therefore the panel programs are referred to as "inviscid" analysis methods. As a result they compute wave and induced drag but not the effects of viscous drag. Viscous drag computations are computed by specialized codes known as Navier Stokes programs. These programs are difficult to use and apply and are best left to a professional skilled in their use.

When a designer has a task that justifies the use of CFD programs, he should be using design tools that that can export true 3D surface shapes in the form of common Computer Aided Design (CAD) file formats. Designing in a typical CAD program such as AutoCAD using lines and polylines, even though in 3D are not sufficient for use with CFD programs. True surface definitions such as Non-Uniform Rational B-spline (NURB) surfaces are required. Most professional versions of the commonly known yacht design programs (AeroHydro, AutoShip, Maxsurf, New Wave, PROLINES) all provide this kind of file exchange.
Licensing costs or consulting time is available from the companies or sources listed below.

Company Name	Program(s)	Web Address
Aerologic	Cmarc, Postmarc	**http://www.aerologic.com/dwt.html**
Analytical Methods Inc	VSAERO, FSWAVE	**http://www.am-in.com**
Fluent	FLUENT	**http://www.fluent.com/solutions/marine/index.htm**
South Bay Simulations	SPLASH	**http://www.panix.com/~brosen**
Vacanti Yacht Design	FOIL 97	**http://www.vacantisw.com**
Virginia Technical University	Several Free simple programs – Code Compiler may be required	**http://www.aoe.vt.edu/aoe/faculty/marchman/software**
CFD Online	Very extensive links to many suppliers of CFD programs of every possible type	**http://www.cfd-online.com**

Specialized consulting companies include:

Bruce Rosen
South Bay Simulations
44 Sumpwams Ave
Babylon, NY 11702 631 587 3770, brosen@panix.com

Joe Laisoa
Fluid Motion Analysis Consulting, Inc.
3062 Queensberry Dr.
Huntingtown, MD 20639, 410 535 0307 X3351, laiosa@panix.com

Conclusion

CFD programs are best applied when there are either significant engineering unknown effects or load levels or where design optimization for a specific application in specific conditions are essential to the goal. For instance, there are many books of scantlings or building standards for typical sailboat or powerboat designs intended for inland cruising. But an attempt at the world record speed sailing at the “ditch” in France at speeds approaching 50 knots clearly calls for specialized analysis to prevent catastrophic failure that could risk lives or incur that last bit of drag that could prevent success in inching the speed record that much higher.

Some CFD codes are only usable in the hands of a skilled practioner and others are designed and intended for use by those with reasonable technical skills and willingness to do a bit of reading or research to help them understand the results and limitations of their modeling efforts. CFD and analytical programs are very important to the development of high performance vessels from the perspective of optimization for speed and safety. High speed sailing craft and those destined for offshore use can benefit the most from computer analysis methods. One final key point here is that we have only discussed vessels in displacement mode and have not referred to high performance planning powerboats. The prediction of planning vessel performance is an art unto itself and is the domain of yet another class of programs. I refer those of you who wish to know more about that subject area to research the Society of Naval Architects and Marine Engineers (http://www.sname.org) web site.