Supercavitating bodies can achieve very high speeds under water by virtue of reduced drag: with proper design, a cavitation bubble is generated at the nose and skin friction is drastically reduced. Depending on the type of device under consideration, the drag coefficient can be an order of magnitude less than that of a fully wetted vehicle.

For several years, the United States Navy has supported basic research and development involving supercavitating high-speed bodies. Under this program, the theory of high-Mach-number underwater flows has been investigated and first-principles modeling of cavity development and body dynamics have been addressed. To complement these analytical efforts, a sophisticated experimental program has matured.

This presentation provides an overview of recent research in this topic, including the results of selected experiments with supercavitating projectiles at supersonic speeds in water; experiments involving systems suitable for self-propelled vehicles, and several topics in computational methods and simulation.

INTRODUCTION

Supercavitation is a hydrodynamic process in which an underwater body is almost entirely enveloped in a layer of gas initiated at a cavitator mounted at the forward end. A supercavity can be maintained in one of two ways: (1) by achieving such a high speed that the water vaporizes near the nose of the body; or, (2) by supplying gas to the cavity at nearly ambient pressure. The first technique is known as vaporous cavitation; the second is termed ventilation, or artificial cavitation.

Supercavitating bodies can achieve very high speeds under water by virtue of reduced drag: with proper design, a cavitation bubble is generated at the nose and skin friction drag is drastically reduced. Because the density and viscosity of the gas are dramatically lower than that of seawater, skin friction drag can be reduced significantly. If the body is shaped properly, the attendant pressure drag can be maintained at a very low value, so that the overall body drag is also reduced dramatically: by roughly eighty percent for a self-propelled vehicle capable of executing maneuvers, to an order of magnitude for free-flying gun-launched projectiles. However, because the center of pressure is typically located well forward with respect to the center of gravity, control and maneuvering present special challenges. Also, whereas a fully-wetted vehicle develops substantial lift in a turn due to vortex shedding off the hull, a supercavitating vehicle does not develop significant lift over the gas-enveloped surfaces. This requires a different approach to effecting hydrodynamic control.

For most of the last decade, the United States Navy has sponsored basic research and exploratory development programs addressing the physics and engineering of supercavitating high-speed bodies. This paper highlights some results of the Navy-sponsored activities at the Naval Undersea Warfare Center (NUWC) Newport Division, at the Applied Research Laboratory of The Pennsylvania State University (ARL/PSU), and at the Engineering Technology Center Division of Anteon Corporation (Anteon/ETC). Selected results of experiments and of physics modeling and simulation are briefly presented.

EXPERIMENTS WITH SUPERCAVITATING HIGH-SPEED BODIES

Over the course of the Navy program, many tests with captive models have been performed – both in various water tunnels and in the Langley Tow Tank located in Hampton, Virginia – building on the extensive cavitation research performed in many countries throughout the twentieth century (see, for example, the excellent review in May, 1975). Beginning circa 1993, the Navy began testing free-flying supercavitating projectiles launched directly into water, again building on the results of previous research using launchers located above the water surface. One of the immediate future objectives of the Navy program is the demonstration of safe, self-propelled flight of a supercavitating high-speed body capable of prescribed maneuvers. As with the previous aspects of the program, this activity will build on previous tests, in this case, with small, uncontrolled, underwater rockets.

Captive Model Tests

Photographs of cavities produced by disk and cone cavitators are shown in figures 1a and b. The models used in these experiments at ARL/PSU can incorporate systems for introducing ventilation gas, cavitator force balances, pressure measurement systems, and other instrumentation. Just downstream of the cavitator in figure 1b is a gas deflector that redirects the gas flow to minimize disturbances to the cavity. Care must be taken to ensure that the design for the introduction of ventilation gas does not have a significant effect upon the force measurements. Short exposure times, such as with stroboscopic lighting, should be used when investigating unsteady cavity dynamics. Strobe lights can be synchronized with the scanning rate of video systems to provide exposure times of a few microseconds for each video.
frame. The video record of the cavity can then be compared with transient measurements, such as forces, by use of a master clock. The photograph in figure 1a shows a stable cavity, while cavity instabilities are apparent in figure 1b.

![Cavity Images](image1.png)

**Figure 1. Photographs of Water Tunnel Cavitation Experiments:** (a) Disk Cavitator at an Angle of Attack; (b) Conical Cavitator with Stroboscopic Lighting

Cavity-piercing supercavitating hydrofoil-type control surfaces such as those shown in figure 2a have been tested in the ARL/PSU 12-in diameter water tunnel. The hydrofoil is attached to a force balance and test fixture that mounts through the water tunnel test section wall and is designed so that there is a small gap between the base of the hydrofoil and the test section wall. The hydrofoil angle of attack can be adjusted by rotating the test fixture. A two-dimensional cavity is created upstream of the hydrofoil by ventilating downstream of a wedge mounted to the tunnel wall. The hydrofoil then pierces the tunnel wall cavity as shown. The measured forces in the water tunnel have been used to validate computational codes, such as the results of boundary-element modeling using an Anteon/ETC program known as LScav (Kirschner, et al, 2001, 2000; Fine and Kinnas, 1993; Fine, 1992) shown in figure 2b.

![Hydrofoil Photos](image2.png)

**Figure 2. Cavity-Piercing Hydrofoils:** (a) Photograph from Water Tunnel Test; (b) Results of Boundary-Element Modeling

Another type of captive model test has been conducted by NUWC at the Langley Tow Tank located in Hampton, Virginia, to investigate the physics of ventilated cavities on large-scale models. The tests were conducted on a freely rotating model that was equipped with an actuated cavitator. The ventilated cavity boundary is a free surface, which may be susceptible to instabilities that could grow and affect the areas of wetted contact on a supercavitating vehicle. This undesirable change in wetted contact could produce large hydrodynamic forces leading to catastrophic loss of control. The stability of the cavity may be affected by changes in the ventilation rate, changes in depth, or unintended coincidence of system natural frequencies with the cavity oscillation frequency. The tow tank tests were performed to measure the amplitude and frequency of the cavity oscillations. Figure 3 shows a photograph of the test model, along with selected cavity dynamics data. High-frequency, solid-state pressure transducers were used to determine the behavior of the cavities. The tests confirmed that the dominant cavity frequency was correlated with cavity length and towing speed.
The data shows the frequency and amplitude of the pressure fluctuations in the ventilated cavity as a function of time. As the ventilation gas supply was increased at 43 s, the cavity grew and the cavitation number decreased. A longer cavity at a constant forward speed corresponds to a lower fundamental frequency of oscillation. This run clearly shows two different frequencies of oscillation at the initial gas flow rate and then a single dominant frequency at the higher gas flow rate, for which the cavity was longer. Similar data was obtained for over 100 test conditions. Such tests will mitigate risk in the design of supercavitating vehicles by allowing prediction of ventilation gas requirements and avoidance of cavity states that are likely to lead to failure.

Free-Running Model Tests

Captive models are valuable for testing components and subsystems. However, many questions related to vehicle control and cavity-body interactions are best answered using free-running models with on-board inertial measurement units and other instrumentation. An example of an ARL/PSU model is shown in figure 4a. Another class of free-running models, known as Adaptable High-Speed Undersea Munitions (AHSUM), is represented in figure 4b. This device was designed by NUWC and personnel now at Anteon/ETC for high speed launch from a fully-submerged gun. The high-speed film frames of figure 4c show AHSUM in supersonic flight under water. The shock wave and cavity are clearly visible.

Figure 3. Tow Tank Testing of Captive Models: Sting-Mounted Model (Inset) and Cavity Dynamics Data

Figure 4. Examples of Free-Running Supercavitating Models: (a) Self-Propelled Vehicle; (b) Gun-Launched Supercavitating Munitions Package; (c) Supercavitating Projectile in Supersonic Flight Underwater
The supersonic tests were performed on an indoor range at NUWC that was customized for such scientific experimentation. In order to assess the range performance of such devices, the gun was installed by NUWC for field testing at the Army Research Laboratory’s SuperPond located in Aberdeen, Maryland. A schematic of the test set-up, along with an aerial view of the SuperPond and a view looking down range from the launch area are shown in figure 5. Measured velocity data collected at several stations along the range are also plotted, along with a theoretical prediction based on the Leduc ballistic equation and the best estimate of the projectile drag coefficient.

**MODELING AND SIMULATION OF SUPERCAVITATING HIGH-SPEED BODIES**

An important element of the Navy program involves the development of theoretical and computational models that facilitate the investigation and understanding of the physics of supercavitating high-speed bodies. Such tools are also being incorporated into resources for simulating the behavior of such systems, allowing assessment of their utility. Such simulations are expected to play an important role in planning experiments with free-running, self-propelled vehicles, in assessing the safety of such exercises, and in analyzing and applying test data.

**Boundary-Element Modeling**

Boundary-element modeling has proven successful in cost-effectively predicting many important features of supercavitating flows, including the cavity shape, the cavity drag, and even such details as the re-entrant jet at the cavity closure point (Uhlman, et al, 1998; Savchenko, et al, 1997; Kirschner, et al, 1995). An example of such predictions was discussed above in connection with the lift on supercavitating hydrofoils. Figure 6 shows an application by NUWC of boundary-element methods to predict flows past partially-cavitating axisymmetric bodies of various geometry (Varghese, et al, 2001). The generic geometry of the flow system is depicted in the upper left-hand corner of the figure. The predicted cavity shapes for fixed cavity lengths are shown in the lower left-hand graph. The relationship between the cavitation number and cavity length is presented in the lower right-hand graph for two selected body radii, and for the case of supercavitation. It can be seen that the presence of the body plays an important role. The upper right-hand graph shows a prediction of the drag budget for partially-cavitating flows at high Reynolds number. It can be seen that, although the total drag of partially-cavitating bodies of the selected form is largely dominated by the friction and base drag components, a significant decrease in drag occurs in the regime where the cavity length approximately equals the forebody length. However, it should also be noted that non-feasible solutions are predicted for which the cavity intersects the body profile at the forebody-cylinder junction. A similar effect leading to hysteresis has been observed in experiments (Savchenko, et al, 2000). Such methods are being extended by Anteon/ETC to predict fully-three-dimensional, time-dependent flows with lift (Kring, et al, 2001; Kring, et al, 2000). Note that the forebody geometry is not the same for the various cases presented in figure 6.

**Large-Scale Computational Fluid Dynamics (CFD) Modeling**

UNCLE-M is an implicit, time-accurate, pre-conditioned, multi-phase, Reynolds-averaged Navier-Stokes (RANS) solver, with high-order-accurate discretization, flux limiters, and a three-dimensional, fully generalized multi-block, parallel structure. Mixture volume and constituent volume fraction transport and generation for liquid, condensable vapor, and non-condensable gas fields are solved. Mixture momentum and two-equation turbulence model equations are also solved. Finite rate mass transfer modeling is employed to account for exchange between the liquid and vapor phases. The code can handle buoyancy effects and the presence and interaction of condensable and non-condensable fields. This level of modeling complexity represents the state of the art of CFD cavitation analysis. The code is being adapted and applied at ARL/PSU to model the physics associated with high-speed maneuvers, controls, control surface actuation, body-cavity and fin-cavity interactions, viscous effects such as flow separation, and compressibility effects associated with rocket exhaust, ventilation, bubbly mixtures, and very high speed projectile motion.
Although axisymmetric boundary-element models facilitate an understanding of key aspects of the flow physics and are useful in the design and analysis of supercavitating high-speed bodies, vaporous cavitation over an axisymmetric body is known to be a highly nonlinear and three-dimensional event. This is clearly illustrated in figure 7a, which is a photograph obtained during water tunnel testing of a blunt cavitator at zero angle-of-attack. The cavitation number was $\sigma \approx 0.35$ and the Reynolds number was $Re \approx 150,000$. The cavity behavior compares favorably with the CFD result shown in figure 7b. The modeled conditions were nearly the same as those in the experiment. Approximately 1.2 million computational nodes were used. The cavity boundary in the CFD result is characterized by a void fraction isosurface at $\alpha_l = 0.5$. Selected streamlines are also shown, and the cylinder surface has been colored by volume fraction. Clearly in neither the model result nor the photograph is the flow field in and around the cavity axisymmetric. There is little likelihood of obtaining purely axisymmetric conditions in even the most well-controlled environments. This is compounded by the influence of highly nonlinear turbulent flow dominated by phase transition, et cetera. It is suggested that here, via an enhanced level of modeling, the real flow has been well captured. An in-depth understanding of the detailed causal mechanisms is a goal of ongoing research.

![Image of axisymmetric boundary-element model](image1.png)

![Image of cavity behavior comparison](image2.png)

**Figure 6. Boundary-Element Modeling of Partially-Cavitating Axisymmetric Flows (Body Length is 80 Cavitator Diameters)**

![Image of cavity length and radius](image3.png)

![Image of cavitation number and axial coordinate](image4.png)

**Figure 7. Three-Dimensional Vaporous Cavitation over a Blunt Circular Cylinder: (a) Photograph at $\sigma \approx 0.35$; (b) CFD Prediction at $\sigma \approx 0.4$**

A sample of the results obtained by three-dimensional time-domain modeling of vaporous cavitation over a blunt cylinder at zero angle of attack is presented in figure 8. These results appear to agree with both significant qualitative and quantitative experimental observations. As in the experiment, the modeled reentrant flow has been observed to follow a helical pattern. This helical flow revolves around the circumference of the cylinder, driven by a complex reentrant flow.
At the same time, other aspects of the flow tend to cause a cavity cycle that is largely axial. This axial cycle fits the typical observations of reentrant flow (Stinebring, et al, 1979, 1983; May 1975). This axial motion is observable in the snapshots and is also well captured by the drag coefficient history given in figure 8a of the figure. Here the drag history has been given over a cycle as defined by the three-dimensional flow. Clearly the drag coefficient is insufficient, by itself, to provide the true model cycle. However by examination of figure 8b in conjunction with figure 8a, it is possible to deduce the model cycle. In figure 8c, the cycling frequency data based on a three-dimensional analysis is compared with experimental results and those of an axisymmetric analysis. Due to the observed helical nature of the reentrant region, it was necessary, experimentally, to use high speed movies to determine the period cavity cycling (Stinebring 1976, 1983, and personal communications). Generally two consecutive observed cycles were required to determine the reported cycle. This would then coincide with the cycle determined by a complete revolution of the reentrant jet. It is apparent that, particularly in comparison to the axisymmetric UNCLE-M results, the three-dimensional UNCLE-M results compare favorably with the experimental data.

Figure 8. Naturally Cavitating Flow over a Blunt Cylinder: (a) Drag History; (b) Sequence showing Surface Pressure and Isosurface of Liquid Volume Fraction; (c) Comparison of Measured and Computed Cavity Cycling Frequencies

Figure 9 shows an underwater supersonic projectile. Both computational results and a corresponding photograph of an actual test are included in the figure. The flow Mach number for the case shown is 1.03 and the liquid-to-gas density ratio is nominally 1000. The experiments and the computations show the presence of a bow shock upstream of the nose. Also, because of the high velocity, the cavitation number is about $10^{-4}$. Consequently, most of the flow immediately adjacent to the body is completely vaporized as is the downstream wake portion. Field contours of mixture density and surface contours of pressure are shown. The single phase water shock system, vaporous cavity, and vaporous wake are observable in the simulation.

Figure 10 shows the flow field of an underwater rocket exhaust. The plume is supersonic and is slightly under-expanded. It is surrounded by a co-flowing secondary subsonic gas stream, that in turn is surrounded by a liquid water free-stream flow. The nominal liquid-to-gas density ratio is 1000. Figure 10a shows the density field. Figure 10b shows the shock function field, which exhibits the classic expansion pattern. In particular, the interaction of the compressible gas stream with the incompressible liquid is demonstrated first by the contraction and then by the expansion of the gas stream. In addition, the interface between the liquid and gas phases is comprised of a two-phase mixture that is also fully supersonic due to the low magnitude of the mixture sound speed.

**Flight Simulation of Supercavitating Vehicles**

Large-scale simulation has also been applied in the time domain to begin to address vehicle maneuvering. In figure 11, a set of preliminary results for a notional self-propelled, ventilated, supercavitating vehicle is presented. Figure 11a illustrates a view of the vehicle geometry, which has a relatively blunt cavitator and several annular ventilation ports with aft oriented gas deflectors. A cavity gas ventilation rate was prescribed sufficient to enshroud the entire vehicle during steady flight. A gas propellant flow rate was also specified at the exhaust nozzle. For this analysis, ARL/PSU applied the incompressible UNCLE-M model.
Figures 11b-h show cavity shapes corresponding to different instants during the prescribed pitch cycling maneuver shown in figure 11i. A non-dimensional time step of $\Delta t/t_{\text{ref}} = 0.09473$ was specified, where $t_{\text{ref}} = L_{\text{vehicle}}/U_{\infty}$. A grid consisting of over 1.2 million vertices was used. The simulation was run on 48 processors of a Cray T3E. The snapshots of the evolving cavity during the maneuver are characterized by isosurfaces of liquid volume fraction at a value $\alpha_l=0.5$. Figure 11i shows the predicted lift history for the vehicle during the maneuver, in addition to the prescribed angle-of-attack. A three-field simulation was carried out. Vaporous cavitation occurs upstream of the first gas deflector. The significant perturbation of the cavity geometry for this maneuver is clearly evident. Indeed, the cavity impinges on the body at $t/t_{\text{ref}} = 47.4$. Also, natural cavitation near the leading edge is not sufficient to keep the first injection port unwetted.

Such large-scale simulation and the related simulation using fully three-dimensional unsteady boundary element models allow investigation of the detailed interaction between the body and the cavity during the course of a maneuver. In some cases, however, basic albeit important aspects of the vehicle dynamics can be more easily captured using simpler models. ETC_Supercav and ETC_Simplecav are two such tools being developed by Anteon/ETC. ETC_Supercav is a six-degree-of-freedom underwater flight simulator that incorporates simple hydrodynamics models of the cavitator, the fins, the afterbody, and the cavity itself into a numerical simulation resource that allows the investigation of various physical aspects of the vehicle (such as fin sweep-back under constant-torque actuation), maneuvering strategies (such as a banked turn), and the control system itself (including such aspects as state estimation to account for noisy hydrodynamic excitation, as discussed above, and robustness to uncertainty). Example output from the flight simulator is presented in figure 12, which shows a comparison of vehicle response with the two different fin configurations indicated.
An even simpler tool, ETC_Simplecav, has been found to be very useful in mapping the basic parametric space of such vehicles. This tool is used to simulate the model problem presented in figure 13a. The simplified system consists of a mass and a cavity-like surface that is deformed via motion of the mass. A system of nonlinear springs is used to represent the forces acting on the mass via the fins and afterbody planing on the cavity boundary. The key features of both the full and the simplified systems are the discontinuous forces associated with the rather sharp boundary between the gas and liquid phases of the ambient fluid, and the memory effects associated with advection of disturbances downstream from the cavitator. This highly nonlinear behavior leads to a system map with the fractal character shown in figure 13b, which presents the Lyapunov exponent as a function of the time delay (as set by the cavitator position and the vehicle speed) and of a characteristic system frequency (as set by the mass and spring constants).

Because of the presence of a time delay and non-linear coupling, it appears that even this simplified system satisfies the necessary conditions such that chaotic behavior might be encountered (Baker and Gollub 1991). This may be related to the fractal nature of the plot shown in figure 13b.
Simulation capabilities are also being developed at ARL/PSU to support the development and testing of supercavitating high-speed bodies. Analytical and computational models have been developed describing the full six-degree-of-freedom dynamics of a supercavitating body. In addition to the dynamics of the body, forces on the cavitator, fins, and planing contact point are modeled. These forces are modeled using a combination of empirical, analytical, and experimental data in a highly modular simulation environment that has been specifically developed over the last 10 years to support the integration and testing of prototype vehicles. A novel scheme for tracking the cavity has been implemented that allows the full simulation model to run in real-time. This simulation capability is being used in the hydrodynamic design, optimization, and analysis of supercavitating vehicles; in the development and bench shakedown of practical control systems for in-water testing; and in the analysis of in-water data.

The vehicle simulation model has been coupled with the immersive visualization capabilities of the ARL/PSU SEALab. This immersive environment is a 4-wall cave automated virtual environment (CAVE) that provides a user with a stereoscopic view of a scene that may include just a vehicle or imagery of a specific geographical location. Figure 14 shows a snapshot of a visualization of the simulation described above. Various vehicle simulations run in a distributed fashion on a network and are tied to the visualization environment via a distributed simulation/high-level architecture (DIS/HLA)-compliant system.

![Dynamics Visualization Tool in SEALab](image)

The user can interact with a simulation as it progresses to change a viewpoint, issue a command to a player, change the initial location of players and their characteristics, and control player responsiveness to environmental stimuli. This capability is being used to evaluate different vehicle concepts in operational environments and to educate the technical community in the capabilities of supercavitating vehicle technology. The hydrodynamic performance of supercavitating vehicles in maneuvers is of particular interest. Relevant quantities include the transient forces and moments, as well as transient cavity behavior, each of which is important in the design of vehicle control systems and gas ventilation schemes.

**SUMMARY AND CONCLUSIONS**

This article has provided an overview of recent basic research and exploratory development results in the topic of supercavitating high-speed bodies. Both experimental and modeling aspects of the Navy program have been discussed. The focused effort by a nationwide team of scientists and engineers over the last decade has resulted in an improved understanding of the physics of these devices, as well as an enhanced capability to predict their performance and assess their utility. The confidence gained in overcoming the many technical challenges associated with vehicle and projectile operation at very high speeds underwater has positioned the Navy to set more challenging objectives for the next few years, including field testing of free-flying vehicles in maneuvers. Continued development of the resources established to date will help ensure that such activities can be carried out safely and cost-effectively.

**ACKNOWLEDGEMENTS**

This research was sponsored, in part, by the Office of Naval Research, High-Speed Super Cavitation Undersea Weaponry program, Dr. Kam Ng, Program Officer, who also sponsored the preparation, publication, and presentation of this article. Continuing exploratory development of the AHSUM concept and adaptation of the supercavitation flight simulator for analysis of supercavitating projectiles is sponsored by the Office of Naval Research Counterweapons and Countermeasures program, Dr. Teresa McMullen, Program Officer, who also sponsored the AHSUM field tests. Certain results presented herein were achieved under projects jointly sponsored by the Defense Advanced Research Projects Agency, Dr. Theo Kooij, Program Officer, and by the Naval Undersea Warfare Center In-House Laboratory Independent Research and Bid and Proposal Programs (Messrs. Richard Phillips and Bernard Myers, Program Managers, respectively). The gun launchers used in the supercavitating projectile tests were supplied by General Dynamics Armament Systems. The author’s would like to thank Dr. Ann W. Stokes, Mr. Jeff O’Brien and Ms. Jamison L. Miller for their contributions to the vehicle dynamics and control systems simulation discussed in this article.

**REFERENCES**


