Navy Successfully Simulates Effect that May Improve Low-Speed Maneuverability

By Joseph Slomski and Tom Marino
Research Engineers
Naval Surface Warfare Center, Carderock Division
Bethesda, Maryland

The Naval Surface Warfare Center (NSWC) has successfully simulated the Coanda effect using computational fluid dynamics (CFD) - which may help improve low-speed maneuverability of ships and planes. The engineers demonstrated that by blowing air out of a strategically located slot in an airfoil, the rear stagnation point could be moved further aft along the trailing edge of the airfoil, thereby increasing lift. The military has worked on a number of potential applications for this effect, such as making it possible for submarines moving at low speeds to make sharp turns. The key to the advance made by the NWSC was the use of the Reynolds stress model for predicting turbulence in the jet, which the research shows is far more accurate than the more common k-ε models.

The Naval Surface Warfare Center is the principal Navy resource, national focal point, and international leader in surface and undersea vehicle science, ship systems, and related maritime technology. A major technical component of the Naval Sea Systems Command, the Division is a source of innovative technology for other national priorities such as environment, energy and transportation. The Division is responsible for research, development, test and evaluation, fleet support, and in-service engineering for surface and undersea vehicles, including hull, machinery and electrical systems, and propulsors. It conducts logistics research and development, and provides support to the Maritime Administration and the maritime industry. The technical leadership areas of the Carderock Division include materials, structures, ship protection systems, vehicle concepts, hydrodynamics, acoustic and electromagnetic signatures, environmental protection systems, and logistics.

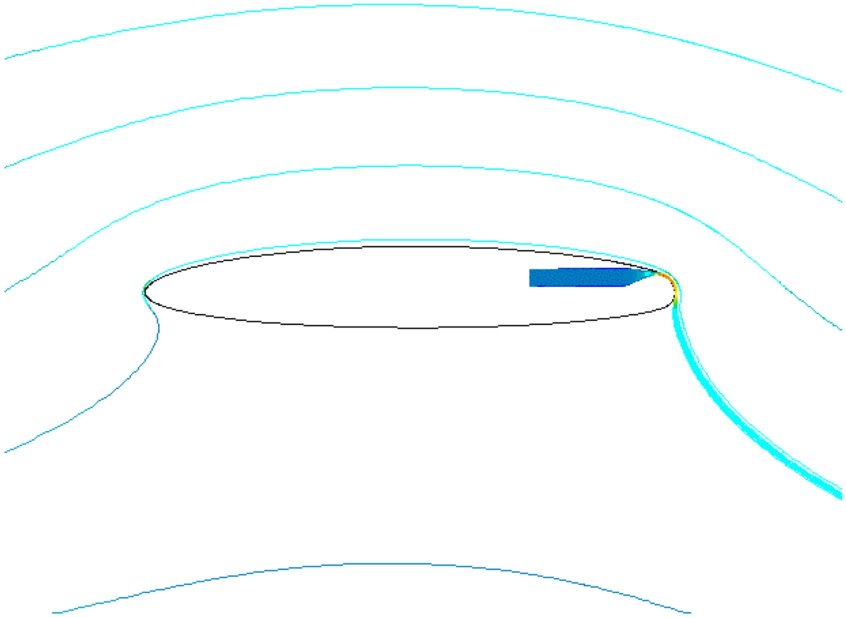

Figure 1: Typical circulation control airfoil showing a Coanda jet at right and surrounding streamlines. The flow is from left to right. The jet is depicted by the thick group of streamlines at the trailing edge of the airfoil. Streamlines are colored by increasing velocity magnitude from blue to red.

The Coanda effect

In 1910, a young Romanian-born engineer named Henri Coanda tested a plane that he had designed and built powered by a gasoline engine-driven centrifugal compressor, a compression chamber, and nozzle - which many call the first jet engine. Coanda put metal plates between the hot jet gases and the plywood fuselage. He expected that the jet would be deflected away from the fuselage but instead it deflected onto the plates, ran along them, and set the fuselage on fire. Amazed by what was going on - which is now known as the Coanda effect - he failed to notice that he was headed straight for a wall. At the last second he noticed the wall, pulled back the stick, just barely cleared it, and was thrown clear as the plane went up in flames. He spent much of the next 20 years studying the phenomenon of ejecting a jet from a narrow slot over a surface.

Coanda jets are created by blowing a moderate to high pressure gas or liquid through a narrow slot over a surface. The velocity of the jet evacuates the layer of fluid between it and the surface. The low-pressure region between the jet and the surface cannot be relieved by ambient inflow, however, because the ambient fluid is on the other side of the jet. This explains why the jet deflects towards the surface and adheres to it. Coanda jets will run tangentially over a surface and can even circulate around 180 degree bends. The jets are often very turbulent because they are composed of many smaller localized eddies. The speed of the fluid within these turbulent eddies varies with time, location, and direction.

Aerospace and naval applications

Coanda jets are being investigated for both aerospace and naval applications because they can significantly increase the amount of lift generated by an airfoil. The lift of an airfoil is influenced by the location of the stagnation points at its leading and trailing edges. For a conventional airfoil producing lift, the front stagnation point is just under the leading edge where the oncoming flow splits into upper and lower surface streamlines, and the aft one is at the trailing edge where the flow re-joins. When air is blown from a slot just above the rounded trailing edge of an airfoil, it increases the circulation around the airfoil. In the case of a circulation control airfoil, which has a rounded trailing edge to take advantage of the Coanda effect, this increased circulation moves the rear stagnation point further aft. The front stagnation point is also shifted to a location further below the leading edge of the airfoil. As a result, the lift can increase by a factor as high as four and can be varied with the amount of air blown through the slot. In the 1980s the Navy proposed to use the concept - which has become known as circulation control - for an aircraft that could take off like a helicopter and, once up to speed, stop the rotor and use it as a wing. This concept became known as the "X-wing." As an airplane, it was projected to be capable of high subsonic speeds. Analytical, wind tunnel and whirl tower studies conducted by Boeing, Lockheed and Sikorsky all verified the feasibility of the concept while pointing out the challenging development tasks. But ensuring safety of flight would have required many endurance tests, redundant systems, and an elaborate crew ejection strategy. The program was terminated in 1987 because of technical difficulties and cost overruns.

Recently, low speed maneuverability has become an important design requirement for aircraft, ships, and submarines. At low speed, the control authority (that is, the normal, or lifting force) associated with conventional hinged control surfaces is often insufficient to perform certain maneuvers. As a result, designers have begun to investigate the use of circulation control airfoils to achieve the required control authority at low speeds. In an effort to avoid the difficulty and high cost involved in experimentally investigating many different circulation control configurations, researchers and designers have begun to focus on the use of computer simulation to analyze these devices.

The modeling challenge

Although the majority of the computational problem of the circulation control airfoil is straightforward, complications arise in the area of the Coanda jet itself. The extent to which the jet remains attached controls the circulation, and, hence, the lift generated by the airfoil. Thus, any computational technique, in order to be successfully applied to the circulation control problem, must be able to accurately predict the spreading rate of the jet and the location at which the jet finally separates from the curved trailing edge of the airfoil. To accomplish this, the CFD solver must be able to correctly predict the

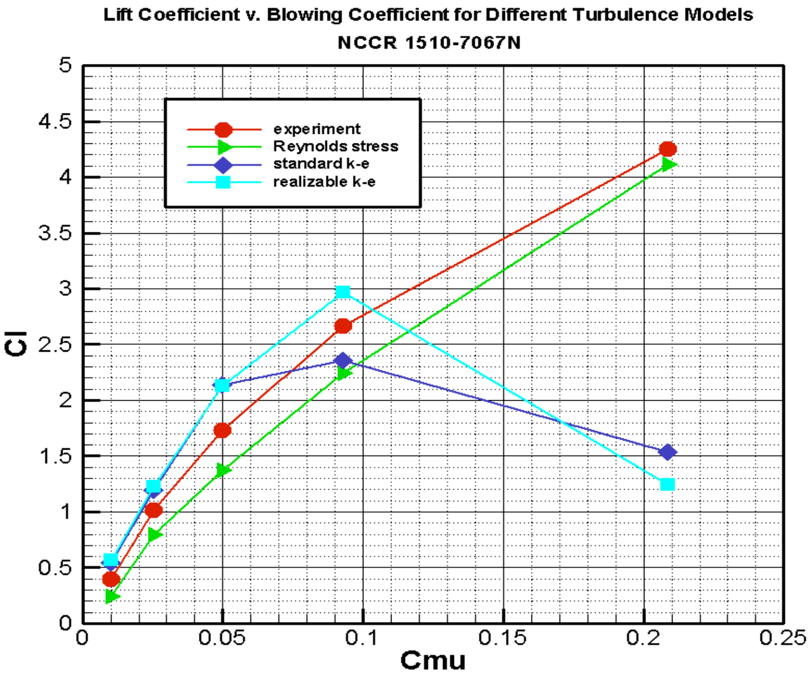

Figure 2: Comparison of experimental data with numerically calculated sectional lift coefficients for three different turbulence models. Cl is the sectional lift coefficient and Cmu is the blowing coefficient.

exchange of momentum between the Coanda jet and the surrounding fluid. The type of turbulence model chosen for the problem is crucial to successfully capturing this interaction and the subsequent prediction of the lift force generated by the circulation control airfoil.

In an effort to demonstrate that Coanda jets could accurately be modeled with CFD, NSWC researchers modeled an NCCR 1510-7067N airfoil with a Coanda jet emerging from a slot located just above the trailing edge. The Reynolds number for the problem was 5.45 x 106, corresponding to a free stream Mach number of 0.12. The flow was assumed to be governed by the two-dimensional, compressible, Navier-Stokes equations that describe the behavior of fluids under both laminar and turbulent flow conditions. Strictly speaking, to analyze a turbulent flow (which is inherently unsteady because each of the flow field variables has a fluctuating component), a transient, or time-accurate calculation would be necessary, using a time step small enough to capture all turbulent fluctuations on even the smallest time scale. For many turbulent flows, however, time-averaged (or Reynolds-averaged) Navier-Stokes (RANS) equations can be used instead. The RANS equations contain extra terms called the Reynolds stresses that account for the time-averaged effects of turbulence on the flow field. In particular, these terms give rise to increased diffusion of the flow field variables.

Turbulence model helps solve problem

The introduction of the Reynolds stresses after time-averaging the fluctuating variables requires some form of closure so that all variables can be uniquely determined. There have been a wide range of methods used to do this, beginning with the simplest zero-equation models and extending to the much more complex Reynolds stress transport model (RSM). Most CFD simulations make use of the popular k-ε turbulence model or one of its derivatives. The k-ε models use two extra differential equations to provide closure. The Reynolds stress model, on the other hand, requires one equation for each of the four (2D simulations) or six (3D) Reynolds stresses. While the k-ε models assume that the effects of turbulence are isotropic, or the same in all coordinate directions, the RSM does not. It is therefore a better choice for flows involving high swirl or strong curvature. NSWC researchers selected FLUENT CFD software from Fluent Incorporated, Lebanon, New Hampshire, for this application primarily because it is one of the few commercial codes that offers several of the simpler k-ε models as well as the more rigorous Reynolds stress model.

NSWC researchers modeled the Coanda jet using three different turbulence models: the standard k-ε model, the realizable k-ε model (favored for predicting the behavior of round jets), and the Reynolds stress model. The numerical solutions were compared to experimental data. Figure 2 illustrates this comparison. In Figure 2, Cl is the sectional lift coefficient, and Cmu is a measure of the Coanda jet blowing rate. The solutions show that for the lowest rates of blowing, the k-ε and realizable k-ε turbulence models can reasonably predict the lift generated by the circulation control airfoil. At higher blowing rates, however, only the Reynolds stress turbulence model continues to capture the physics of the circulation control problem, allowing reasonable prediction of lift. Figure 3 shows the predicted

Coanda jet streamlines at the highest blowing rate investigated, Cmu= 0.209. For the isotropic (k-ε) turbulence models, only the realizable k-ε results are shown (Figure 3a), since the standard k-ε results are similar. These results suggest that the jet remains attached to the airfoil surface for nearly 1 ½ revolutions around the airfoil, which was not seen in the experiments. The Reynolds stress model (Figure 3b), on the other hand, predicts separation cleanly at the trailing edge, which is physically correct. The RSM results, which demonstrate that CFD simulation can accurately model Coanda jets, will serve to expedite the development of circulation control airfoils to improve low speed maneuverability on water and air craft in the future.

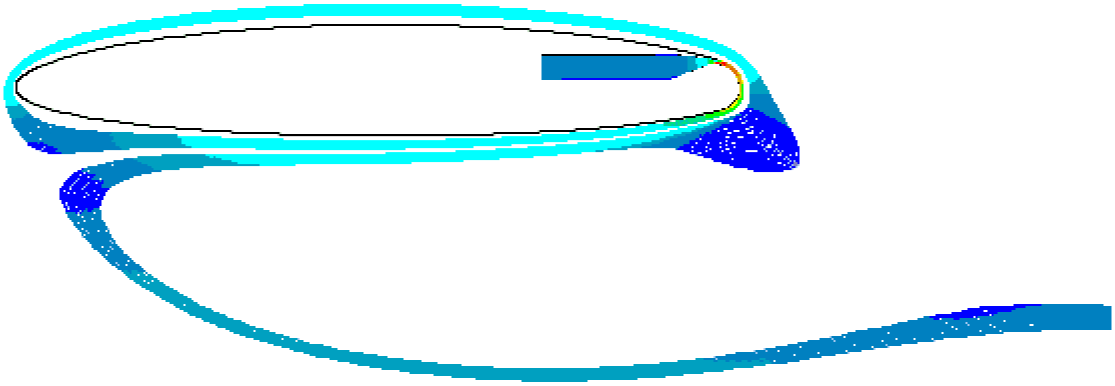

Figure 3a: Coanda jet path predicted by the realizable k-e turbulence model for Cm = 0.209. Streamlines are colored by increasing velocity magnitude from blue to red.

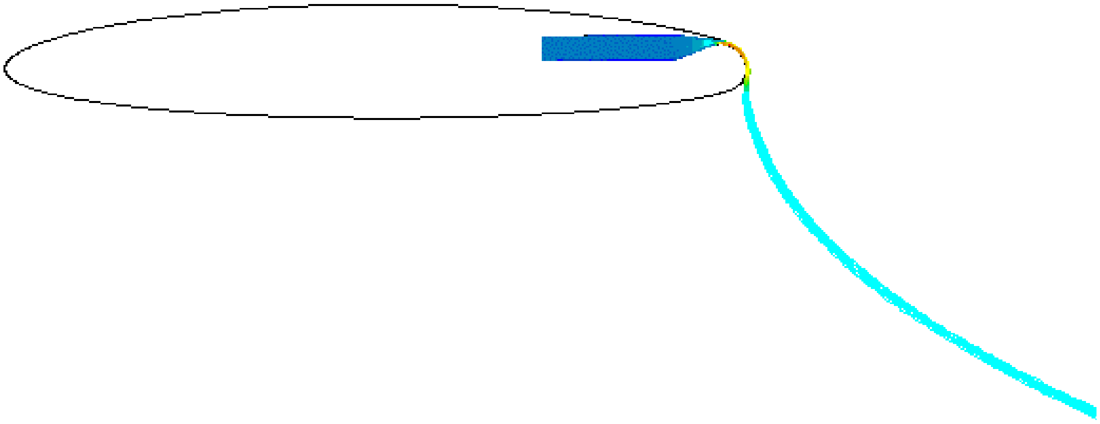

Figure 3b: Coanda jet path predicted by the Reynolds stress turbulence model for Cm = 0.209. Streamlines are colored by increasing velocity magnitude from blue to red.