A Primer on Using CFD to Tackle UAV Aerodynamics Problems

Christoph Hiemcke, PhD, Fluent Inc.

INTRODUCTION

You are an engineering manager or analyst, and your task is to solve several problems involving the aerodynamics of a UAS (unmanned air system). Your experience with CFD (computational fluid dynamics) is limited, and your company's UAV (unmanned aerial vehicle) project is on a tight schedule. The purpose of this white paper is to introduce you to the types of aerodynamic problems that are typically solved using CFD software to help guide your decision process. The paper consists of a series of case studies, each of which includes the motivation for applying fluids engineering to that particular problem.

In the context of this paper, the phrase "CFD" refers to software that solves the full, three-dimensional, compressible, unsteady Navier-Stokes equations together with turbulence equations. This means that you can solve for steady or unsteady flows at subsonic, transonic, supersonic, and low hypersonic speeds. Many CFD software packages can also handle multiple species, and a few offer a reacting flow capability. Some fluids modeling packages allow you to track liquid droplets, solid particles, and the shape of the interface between a liquid and a gas. Finally, some solvers allow for relative motion between different objects in the domain, such as when a control surface is moved or when a store is dropped.

All Navier-Stokes solvers require the user to define a flow volume (the volume occupied by the air), to cover its boundaries with surface mesh, and to fill it with polygonal cells. The solver then writes the governing equations for each cell, assembles them into a large matrix system, and that system is then solved subject to the imposed boundary conditions.
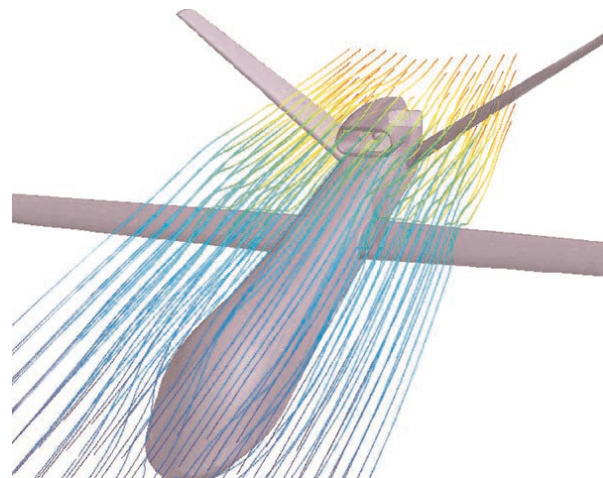
Predator B and resulting CFD analysis
Courtesy of General Atomics Aeronautical Systems, Inc.

Part one of the paper focuses on the aerodynamics of the entire aircraft and includes three case studies. Part two consists of four case studies that involve a subsystem of a UAV.

PART 1: OVERALL VEHICLE AERODYNAMICS

Case Study 1a: Predicting Aerodynamic Forces on a UAV During Extreme Maneuvers

Since UAVs are not subject to the limitations on the number of g's that a human pilot would impose, they can perform very drastic maneuvers. Such maneuvers often involve stalls or near-stalls, with significant regions of separated flow. Flight tests in these conditions will put the vehicle at great risk, and wind tunnel tests are very costly and may suffer from scaling effects. More traditional analysis tools, such as panel methods or vortex lattice methods, cannot tolerate these flow conditions due to the extensive flow separation.

It is worthwhile to note that a good prediction of the aerodynamic forces and moments during subsonic, transonic, and very low supersonic flight (below Mach 1.3 or so) requires the user to create a mesh that resolves the boundary layers with either hexahedral cells or triangular prisms (wedge cells). At speeds above that, the aerodynamic forces will be dominated by the shocks; for example, the drag will be dominated by wave drag. If large areas of separated flow are a concern, then a boundary layer mesh is still mandatory even at high speeds.

Static pressure on a UAV

Case Study 1b: Predicting Aerodynamic Forces on a Morphing UAV in Transonic Flight

A morphing UAV is one which changes its shape during flight. Morphing UAVs, such as Boeing's "Dominator," morph right after they are launched: first the wings are folded out from under the body using a sideward rotation, and then they are extended via a telescoping motion. Other morphing UAVs are controlled through the use of memory alloys in their wings - currents passed through those sections cause the wing to deform, thereby acting as ailerons and flaps. More radical concepts involve a drastic change in the planform shape of the wings.
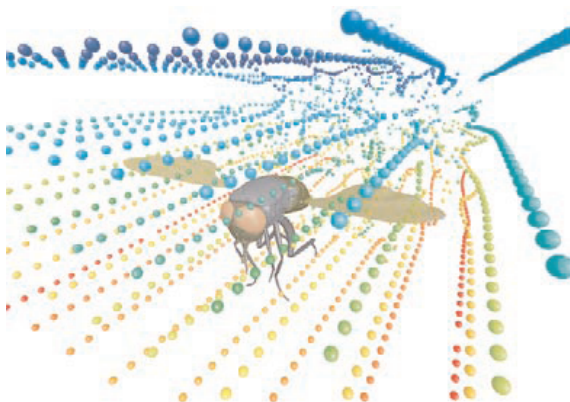
Particle tracks show the air movement caused by the flapping wings on a fly

Vortex formation during the flapping cycle of a fly

All morphing UAVs change their shape as a function of time, so any computational model must also permit the shape to vary with time, unless a series of steady-state analyses is sufficient. This is a case where dynamic mesh or deforming mesh is recommended.

The fact that this UAV is also flying at transonic speed represents another challenge. For one, transonic flows are governed by nonlinear partial differential equations. Furthermore, this flow will be unsteady since the aircraft is morphing, and this means that the shock structures will interact with the boundary layers in an unsteady manner, so the flow may well involve shock-induced separation. From a computational point of view, only Navier-Stokes CFD codes are equipped to handle this type of flow.

It is often necessary to correctly predict both the location and strength of the shocks. Instead of beginning with a very fine mesh from the start (which would be computationally expensive due to the high cell count), some CFD codes allow you to adapt the mesh locally by splitting cells in those areas in which the gradients are the highest. By doing this, you can resolve the shock nicely without using a prohibitively large mesh. When this approach is used for unsteady flows, we speak of dynamic adaption.

Case Study 1c: Predicting Flow Around the Flapping Wings or Rotors of a Micro-UAV

Micro-UAVs (MUAVs) are designed to operate in small spaces, such as inside a building. For that reason, they must be small, quiet, and able to hover. For these reasons, flapping wings or rotors are used to produce lift and thrust.

From the viewpoint of a fluid dynamicist, these flows are especially interesting because they are often transitional and dominated by unsteady vortex dynamics. These facts also make them very challenging for analysis software packages.

In order to predict the flow around a flapping wing, the CFD package must be able to deform the shape of the wings as a function of time. Just as in the case of the morphing UAV, a dynamic/deforming mesh capability is required.

Since the flow will be dominated by the evolution of vortex structures, it will be important to build a fine mesh to avoid artificial dissipation of those vortices. Also, depending on the size and speed of the UAV, the flow may be transitional. If transition from laminar to turbulent flow plays a significant role in the UAV's performance, an LES (large eddy simulation) turbulence model will likely be required.

If we are interested in rotorcraft MUAVs, we may be able to simplify the analysis. For example, if the focus is only the overall interaction between the rotor and the fuselage, we may be able to model the rotor as an actuator disk by implementing a blade element model. The same strategy can be used for tiltrotor designs or for analyzing UAVs with normal propellers.

If a better fidelity of the flow around a tiltrotor or propeller is required (for example, to study the vehicle's acoustics), we can embed the tiltrotor or propeller in a puck-shaped volume and then physically rotate that puck within the overall mesh. This capability is sometimes called a sliding mesh.

A high-fidelity simulation of the flow around a helicopter-like rotor is more complicated because the blade pitch varies as a function of the azimuthal position of the blade. This means that we must use a dynamic/deforming mesh capability to physically translate and rotate each blade during each new time step.
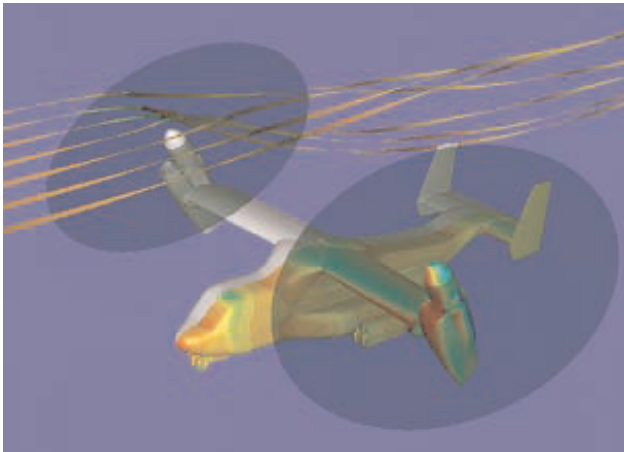
V-22 Osprey during nacelle rotation
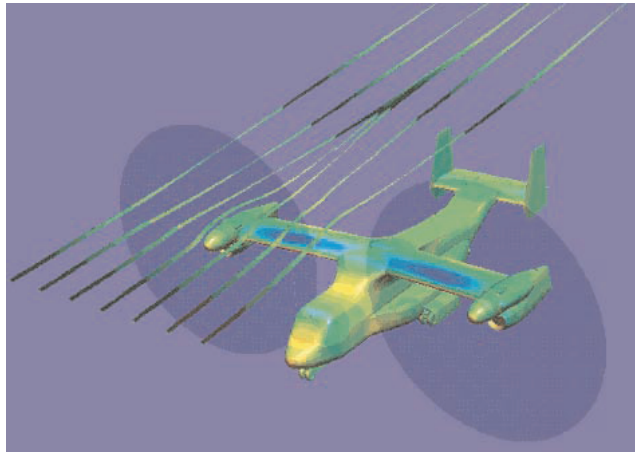
V-22 Osprey in cruise configuration

Case 1d: Designing for a Safe Store Separation from a UAV

More and more, UAVs will be used to drop objects while in flight. Such objects include empty fuel tanks, instrumented pods for meteorological or atmospheric research and, in the case of the UCAV (unmanned combat air vehicle), weapons and weapon pods. Often, the object being dropped represents a significant fraction of the total mass and size of the UAV, so its release will have a profound effect on the vehicle. It is important to be able to predict the motion of both the UAV and the store following the release. Wind tunnel testing is only marginally effective here, since the process is very dynamic and involves the motion of both the UAV and the store - tests using a captive trajectory system (CTS) are quasi-steady and only the store is typically moved.

CFD solvers with a dynamic/deforming mesh capability are an excellent choice for this class of problem, since both the UAV and the store can be moved by applying a six-degree-of-freedom (6 DOF) solver to each component. Trajectories can be predicted very accurately using this approach.

PART 2: AERODYNAMICS OF UAV SUBSYSTEMS

Case 2a: Installation Effects and Placement of Subsystems

Picking up on the theme from the preceding case study, consider the question of where a pylon and its store should be placed. Along the same lines, where should sensor pods and antennas be placed to avoid excessive vortex shedding and associated fatigue loading? And perhaps even more importantly, where should the propulsion system (jet engine, propeller, rotor, etc) and its inlets and exhausts be positioned in order to achieve optimum efficiency?

Most of these questions involve the precise prediction of the boundary layer thickness and of the separation point. Therefore, CFD solvers are the best predictive tool in this case. Some of the above-mentioned problems can be simplified by modeling only the part of the UAV near the
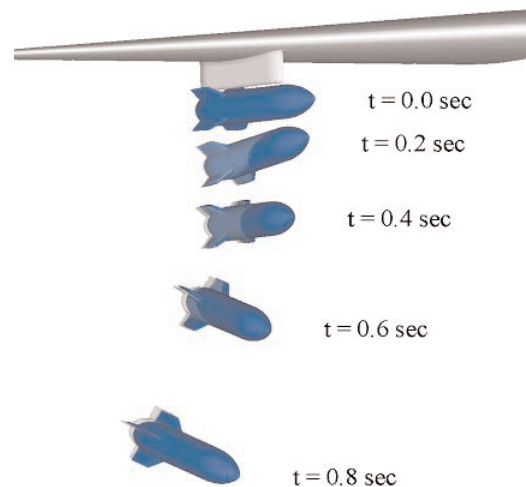
Store separation at Mach 1.2 with CFD results (blue) and experimental results (light blue) Source: AIAA-2003-3919

object of interest. In other cases, you may need to model almost the entire vehicle at low fidelity and then only place a boundary layer mesh upstream of the object of interest (for example, upstream of a ram scoop).

To analyze the full fluid-structure interaction (FSI), the CFD code should allow for coupling to a CSM/FEA (computational structural mechanics, finite element analysis) code, including the interpolation between the CFD and CSM meshes.

Case 2b: Dynamic Forces Due to Tank Sloshing

During the design of the fuel tanks, it is important to minimize the sloshing of the fuel in order to avoid large dynamic forces that affect the stability and control of the UAV. The behavior of the liquid fuel inside the tank is modeled using a free surface multiphase model, such as the VOF (volume of fluid) model. In such a model, the position of the liquid-gas interface is tracked in time. As a result, the center of gravity of the liquid is tracked, as are the forces exerted by the fluid on the tank walls.

Free surface models have many uses in addition to the sloshing problem. They are used to design the hulls of planing seaplanes and to predict the wave drag on hulls of seaplanes in displacement mode and of USVs (unmanned surface vehicles). They have been used to predict the breakup and spreading of water dumped from firefighting UAVs, as well as the breakup and spreading of fuels near the fuel injectors of IC engines and jet engines.

Case 2c: Cooling Problems and Heat Transfer

During the design of a UAV, there are numerous heat transfer problems: the engine, avionics and other electronics need to be cooled, leading edges may need to be heated to avoid icing, and exhausts need to be designed to minimize infrared signatures. CFD has been successfully used to resolve all of these problems.

With regard to engine cooling, typical challenges involve the exact placement of the engine and the heat exchangers within the cowling or nacelle, and the optimal placement of inlets, scoops, and exhausts. The
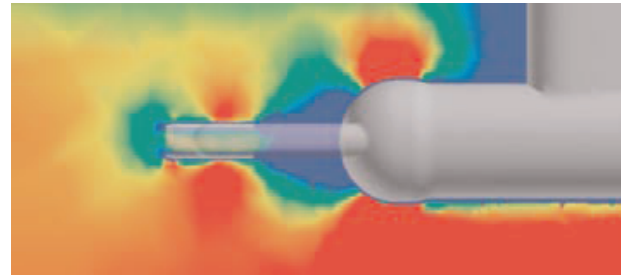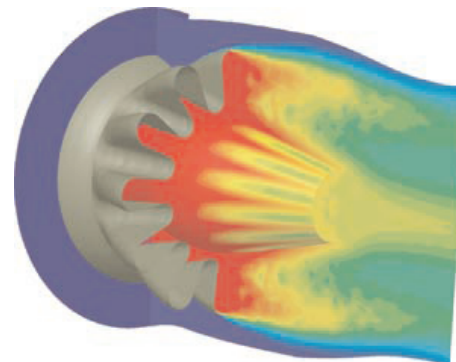

The disturbance of the airflow in the vicinity of a sensor Courtesy of Météo-France
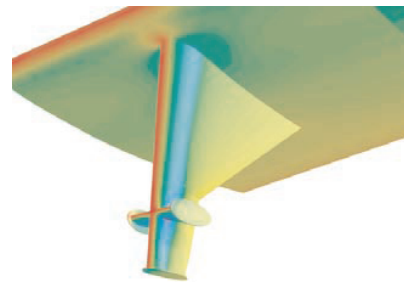
Pressure forces acting on a sensor pylon mounted under a wing
Courtesy of Météo-France
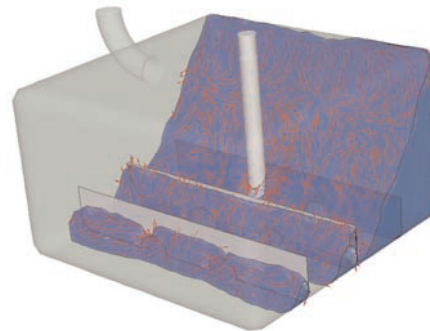
Analysis of how the sloshing in a fuel tank can be reduced through the use of baffles

Exhaust plume from a jet engine

problem is so ubiquitous that it has several names in aerospace jargon, such as "nacelle cooling," "undercowl flow," and "core compartment thermal management." One of the main difficulties here is the complexity of the CAD geometry – it requires the CAD operator and the analyst to work together seamlessly. In the CFD environment, radiation models may be required for this class of problems.

Another problem involving geometric complexity is the cooling of avionics and other on-board electronics. There is an entire "electronics cooling" industry and its best-known problem may be the cooling of computer CPU boxes. In airplanes and UAVs, the geometries are usually more complex because of the curviness of the fuselage. As a result, the meshes will tend to be unstructured instead of consisting of multiple blocks of hexahedral cells.

The same arguments apply to the design of anti-icing systems that are based on bleed air. The hot air is blown into the intricately shaped cavities within the wings in order to heat the wing skin. An additional challenge here is that both the high-speed jets and the flows adjacent to the walls must be resolved properly in order to predict plume spreading rates and wall heat transfer rates. The same stringent requirements apply when you are analyzing the heat transfer from the engine exhaust plume. One needs to determine the plume and the boundary layers adjacent to the walls that are near the plume in order to determine the correct wall temperatures. To find out about the heat transfer due to particles (such as soot particles) in the plume, your CFD code must be able to track particles and account for radiation from those particles.

CONCLUSION

The above case studies demonstrate that CFD is a versatile and essential tool for analyzing many flow problems involving UAVs and their subsystems. Given that inexpensive computational power is now readily available, CFD has become a standard tool for use in the design process.