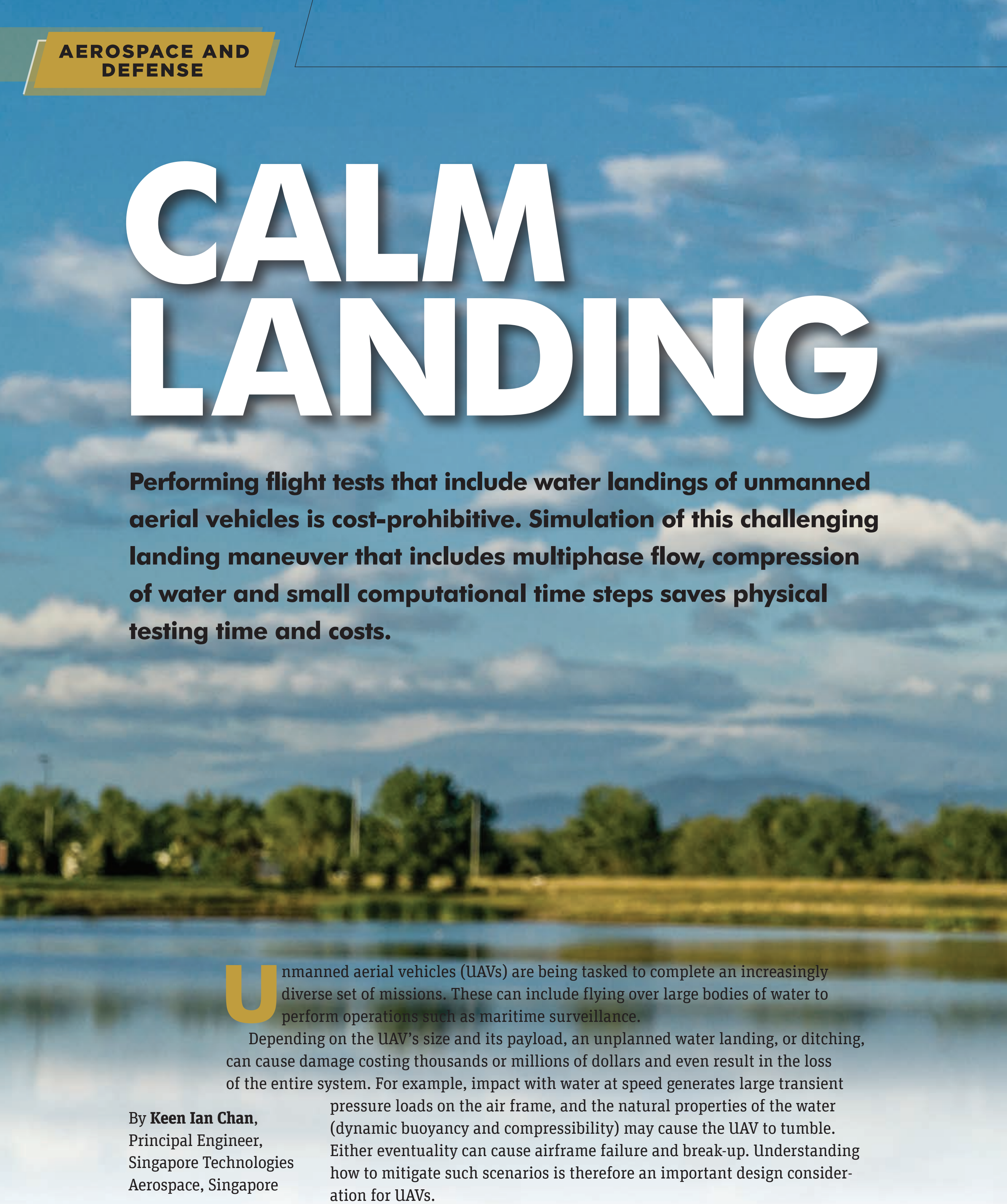


CALM LANDING

Performing flight tests that include water landings of unmanned aerial vehicles is cost-prohibitive. Simulation of this challenging landing maneuver that includes multiphase flow, compression of water and small computational time steps saves physical testing time and costs.

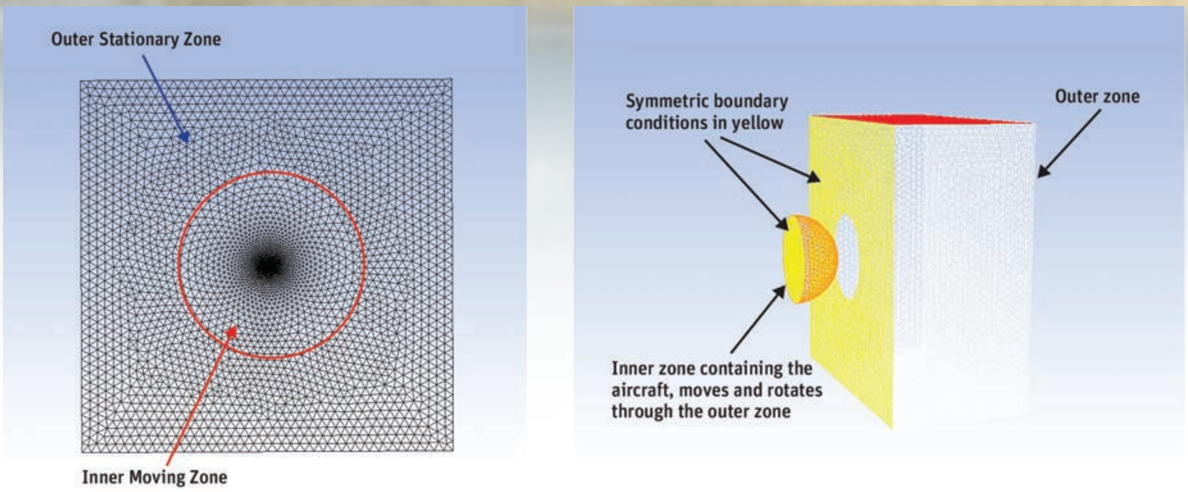
By **Keen Ian Chan**,
Principal Engineer,
Singapore Technologies Aerospace, Singapore

Unmanned aerial vehicles (UAVs) are being tasked to complete an increasingly diverse set of missions. These can include flying over large bodies of water to perform operations such as maritime surveillance.

Depending on the UAV's size and its payload, an unplanned water landing, or ditching, can cause damage costing thousands or millions of dollars and even result in the loss of the entire system. For example, impact with water at speed generates large transient pressure loads on the air frame, and the natural properties of the water (dynamic buoyancy and compressibility) may cause the UAV to tumble. Either eventuality can cause airframe failure and break-up. Understanding how to mitigate such scenarios is therefore an important design consideration for UAVs.

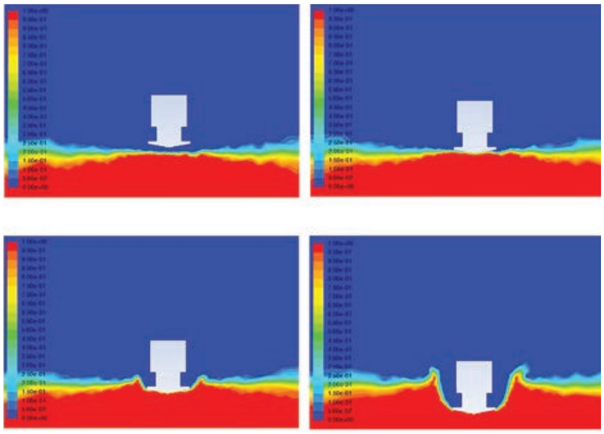
However, performing flight tests of a water-landing maneuver for a new UAV design is not practical because of the time and cost involved to build prototypes, arrange airspace clearance, extensively instrument the test aircraft, and understand and replicate the sea state and environment in which the impact occurred.

Simulation of water-landing scenarios is a practical alternative to extensive flight testing, but it can be challenging because engineers need to consider multiphase flows (air and water), the compressibility of water, and the very small computational time steps

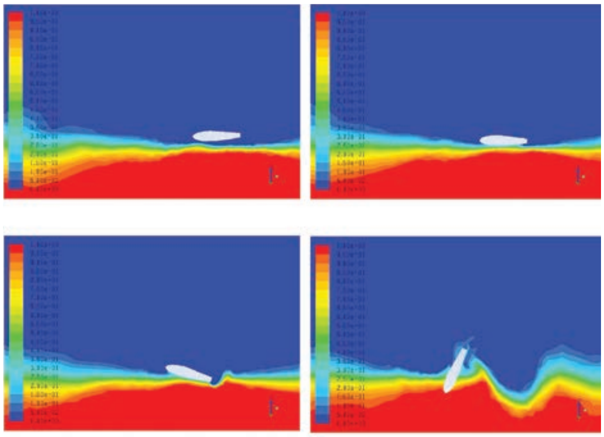


^ Engineers were able to reduce time step size by dividing fluid domain into two zones.

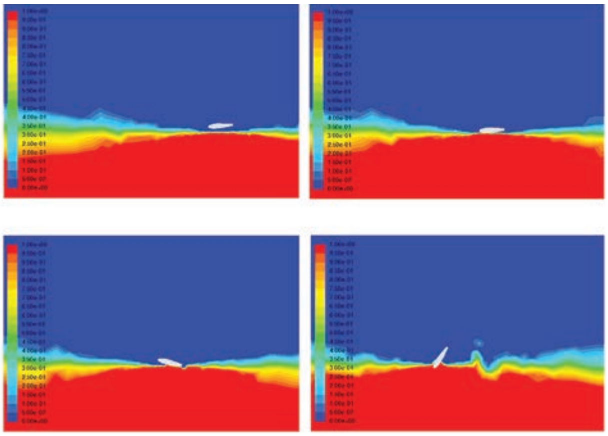
© 2016 ANSYS, INC.



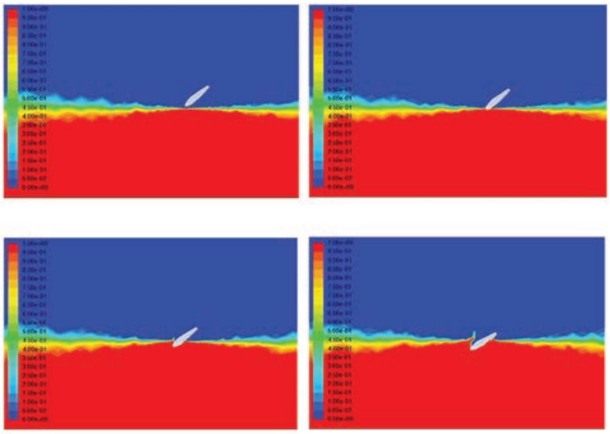
^ Validation of the simulation method



^ Steep descent landing shows undesirable tumbling behavior.



^ Belly landing maneuver simulation reveals undesirable tumbling behavior.



^ Nosedive landing maneuver simulation with desirable results

required to capture impulse loading. Singapore Technologies Aerospace (ST Aerospace) engineers used ANSYS CFD software to overcome these challenges and accurately simulate a wide range of water-landing scenarios. This saved a large amount of time and money.

MULTIPHASE FLOW

ST Aerospace is an integrated service provider that offers a wide spectrum of maintenance and engineering services to a customer base that includes the world's leading airlines, airfreight and military operators. To capture the multiphase properties of the flow fields in water impact simulations, ST Aerospace engineers used the volume of fluid (VOF) model in ANSYS Fluent. In this model, the volume fraction of each phase, which is defined as a fraction of volume occupied by that phase in a computational cell, is tracked throughout the domain, and the interface between phases is captured simultaneously. The geometric reconstruction interface-capturing scheme used in this study computes the evolution of the water surface by representing it using a piecewise–linear approach. This scheme is most accurate and compatible with unstructured, moving and deforming meshes (MDMs).

The pressures generated during water impact are large enough to compress seawater, so the compressibility of water must be included in the simulation. During the simulation, a user-defined function (UDF) calculates the compressibility of water by determining its density based on its bulk modulus, which is defined in terms of pressure and density change.

DIVIDING FLUID DOMAIN TO LENGTHEN TIME STEPS

To simulate the aircraft moving relative to adjacent cells, the time step needs to be small based on the fine adjacent grid resolution. In this case, engineers were able to increase

Aerospace Industry Demands Accurate, Fast and Reliable Simulation Technology
ansys.com/aerospace

> “Performing ***flight tests*** of a water-landing maneuver for a ***new UAV design*** is not practical.”

the time step size by dividing the fluid domain into two zones. An inner hemispheric zone contains the aircraft and remains fixed relative to the aircraft, so that as the aircraft moves and rotates in response to forces generated by water impact, the inner zone also moves and rotates. The outer zone is stationary and fixed in space. This is accomplished in ANSYS Fluent using the MDM modeling approach. MDM efficiently re-meshes the volume cells at the interface of the two zones as the inner zone moves through the outer zone as the computation progresses. The time step size is based on the larger volume cells at the interface of the two zones, rather than the much smaller cells directly adjacent to the aircraft, enabling larger time steps to be used and greatly reducing the number of time steps required to complete the simulation.

Engineers used symmetry boundary conditions in the CFD model so that only half of the aircraft was modeled. This halved the number of volume cells and reduced the computational time by 50 percent. A limitation of this approach is that pitching motion can be captured but rolling and yawing motions cannot.

The water impact simulation starts with the aircraft a short distance above the water and proceeds in small time steps. At each time step, CFD simulations are performed to resolve the flow field at that instant. The flow field yields the forces and moments acting on the aircraft. The forces and moments are input to Fluent’s built-in six degree of freedom (6DOF) solver to compute an incremental translation and rotation for that time step. The UAV is moved to the new position and orientation, carrying the inner fluid zone with it. The movement of the aircraft and body-fixed inner zone distorts the volume cells at the boundary with the outer fluid zone. Regions of distorted cells are re-meshed by MDM to maintain good quality. The cycle is repeated for each successive time step.

VALIDATING THE METHOD

ST Aerospace engineers validated their computational approach by simulating a published experimental test case [1]. The case involves dropping a 160-degree cone into the water at different masses and impact velocities. The impulse forces upon impact were measured. Simulations were performed for the case of a 0.324 kg mass impacting the water at 5.04 m/s. The experimental measurements showed a peak force of 317.844 N while the simulation showed a peak force of 310.977 N, a difference of only 2.2 percent.

EVALUATING DIFFERENT WATER-LANDING APPROACHES

With the simulation method validated, ST Aerospace engineers ran 20 different water-landing simulation cases for the new UAV. The team simulated steep-descent landings, belly landings and nosedive landings. They also modeled a belly landing in which the UAV’s belly was replaced with a NACA 84 flying boat hull.

The steep descent, belly landing and flying boat hull landings all showed tumbling behavior, which is an undesirable result because it increases the forces on the UAV. The nosedive landing, on the other hand, was free of tumbling behavior and provided the lowest forces. Images of the water landing are as seen from the symmetry plane of the UAV, extracted from animations of the simulations.

The CFD simulation of the UAV landing on water yielded valuable results and insights that were used in the airframe’s structural design to enable it to withstand impact with the water. The results will also be valuable for UAV operators to determine the best procedure to execute a water-landing maneuver. These solutions were achieved without having to embark upon a costly and high-risk flight test campaign, thus substantially reducing the time and cost required to design the UAV. Λ

Singapore Technologies Aerospace is supported by ANSYS channel partner CAD-IT Consultants (Asia) Pte Ltd.

References

[1] Watanabe, S. Resistance of Impact on Water Surface, Part I – Cone, *Inst Phys Chem Res Tokyo*, **1930**, Vol. 12, No. 226, pp. 251–267.

© 2016 ANSYS, INC.