

CFD Analysis of the Tigerfish Retractable Float System on a DHC-Twin Otter

Artur Medon, Matthew Tetlow and Bassam Dally
The University of Adelaide

Copyright © 2008 SAE International

ABSTRACT

A Retractable Float System is being developed by Tigerfish Aviation as a retro fit to existing aircraft. The concept of the Retractable Float System is to fold the floats into a streamlined pannier under the main body of the aircraft in order to reduce the aerodynamic drag by an estimated 20% compared to current fixed float systems. The reduction in drag improves the performance of the aircraft and reduces its operating costs by reducing fuel consumption and increasing range. In this paper, a computational fluid dynamics analysis of a Retractable Float System is performed on the DHC Twin Otter Seaplane. Fluent is used to perform a 3D analysis to calculate the various forces on the aircraft. The clean aircraft and the aircraft with the float pack are modeled. A DHC Twin Otter equipped with a the Tigerfish Aviation Retractable Float Pack was found to have only 17% more drag than the clean aircraft, as compared to 45% more drag for a fixed float equipped DHC Twin Otter.

INTRODUCTION

Seaplane is a generic term for an aircraft that can operate from water. By definition a pure seaplane can only takeoff and land on water, however, amphibious seaplanes are equipped with wheels so that they are capable of both land and water operation (Chicken, 1996). There are two types of seaplane: the flying boat and the floatplane. The flying boat is a purpose built aircraft, in that the entire aircraft is designed to operate from water, and uses a specially designed fuselage as its floatation device. The floatplane derives its floatation from external floats and in the vast majority of cases, the floatplane is created by the addition of floats to an existing suitable landplane (Chicken, 1996).

Seaplanes, once significant in air transport, went into gradual decline with the dominance of land based aircraft after WW2. However, they have survived and their activity has doubled in the last 20 years. According to the USA Seaplane Pilots Association the current world seaplane fleet is conservatively estimated at 12,000 aircraft and if this total is to grow only as it did in the previous 20 years there will be 12,000 new planes added.

Seaplanes are a niche transport system offering unique capabilities. World-wide there are signs of a welcome revival in seaplanes. From a defence point of view seaplanes provide a faster force projection than surface ship squadrons. Disasters caused by earthquakes, terrorism, civil unrest as well as climate change require rapid response only possible by air transport and usually beyond the range of helicopters. All too often the airfields are destroyed, captured or non existent, and the areas needing aid are on or close to water that would allow seaplanes access far faster than surface vessels.

Day to day transport can also be improved in regional areas in both developed and undeveloped countries by increased use of seaplanes, particularly if they are amphibious. They can be based on land and use existing land facilities but access new areas enroute by water landing with far less infrastructure cost and environmental impact than land landing.

As flying boats are purpose built aircraft, their development is very costly. Floatplanes are much cheaper to develop as they require only a modification to an existing operational aircraft. Tigerfish Aviation is therefore committed to develop a modern floatplane.

Current floatplanes are based on technologies that have not changed appreciably for the last 80 years. This old technology leads to poor performance, high fuel consumption, clumsy workplace operations and an

antiquated public face. The current issue with conventional amphibious floatplanes is that the design configuration is not aerodynamically efficient, since their large boat shape floats incur a large amount of drag during flight. Tigerfish Aviation is developing a retractable float system in an effort to produce low drag amphibious floatplanes. The concept of the retractable float system is to fold the floats up into a streamline pannier below the fuselage during flight. Figure 1 shows the deployed and retracted floats on a scale model of the Cessna Caravan Seaplane.



Figure 1 Cessna Caravan Retractable Float System (Tetlow, 2007)

The drag reduction occurs due to the reduction of surface area exposed to the airstream and the concealing, in flight, of many of the drag creating features of the floats such as mounting struts, planning steps, chines and spray strakes. Reducing the drag of the aircraft results in many benefits, such as better fuel consumption, increased range, a higher cruise speed and lower operating costs.

The aircraft used in the performance analysis of the Tigerfish Aviation retractable float system was the DHC Twin Otter amphibious floatplane, shown in Figure 2. The volume of the float pack was designed based on the Federal Aviation Regulations FAR 23.751, which requires that the two floats of a twin float floatplane have a buoyancy of 180 percent of the maximum weight of the seaplane in fresh water.



Figure 2 DHC Twin Otter Series 300 (Atoll, 2004)

AIM

The aim of research was to perform a Computational Fluid Dynamics (CFD) analysis to investigate the benefits of a Retractable Float System on the DHC Twin Otter floatplane. The overall objective is to show an aerodynamic drag improvement of the DHC Twin Otter with the retractable float system over the conventional fixed float system. Specifically, we determine the difference in drag between the clean aircraft and the aircraft with retracted float pack.

NUMERICAL METHODOLOGY

CFD uses numerical methods and algorithms to solve and analyze problems involving fluid flows. The advantage of numerical methods, such as CFD, is that insight can be gained into the flow field and the performance of the aircraft can be predicted cost effectively without the need of conducting full-scale experiments with prototypes. The CFD package used for this analysis was Fluent.

FLOW CHARACTERIZATION AND ASSUMPTIONS

We first define the flow field characteristics and the assumptions of the aircraft model, since these characterizations influence the numerical methodology that will be discussed. The flow is modeled as steady in time and the flow is assumed to be fully turbulent. In addition, the heat transfer between the flow and the aircraft is negligible for aerodynamic purposes and hence the model assumes no heat transfer. The operating velocity of the aircraft was 82 m/s and standard atmosphere at sea level was used. The Mach Number of the aircraft was 0.24 at the operating velocity and atmospheric conditions. Since compressible effects can be ignored at Mach numbers below 0.3 (Anderson, 2001), the flow regime was set to be incompressible. Finally, the operating pressure was set to standard atmospheric pressure, which was 101,325 Pascal (Anderson, 2001).

FLUENT SOLVER

Fluent allows two numerical methods, the segregated solver and the coupled solver. In the segregated solver, the governing equations are solved sequentially, segregated from one another, whereas, in the coupled solver the governing equations of continuity, momentum and energy are solved simultaneously. The segregated solver has traditionally been used for incompressible and mildly compressible flows, whereas the coupled solver was originally designed for high speed compressible flows (Fluent, 2005). Since the flow around the aircraft is incompressible, the segregated solver was used.

DISCRETIZATION

In Fluent, the diffusion terms of the governing equations are discretized using the central differencing scheme and are second order accurate. However, the convective

terms of the governing equations cannot be discretized with the central differencing scheme due to transportiveness issues (Christo, 2007). Instead the second order upwind scheme is used for this analysis.

LINEARIZATION

In the segregated and coupled numerical methods, the discretized non-linear governing equations are linearised to produce a system of equations for each computational cell (Versteeg, 2007). Linearization may be implicit or explicit, however using the segregated solver in Fluent, only implicit linearization is available. Hence in our analysis implicit linearization was employed. In particular, the implicit Point Gauss-Seidel numerical algorithm was applied (Fluent, 2005).

UNDER RELAXATION FACTORS

Since the non-linear governing equations are linearised, under relaxation factors are necessary to control the change of a given variable. Under relaxation factors reduce the change between successive iterations by multiplying the change of the variable, $\Delta\Phi$, by a constant factor, α , which is less than 1. Equation 1 below mathematically illustrates this process.

$$\Phi_{new} = \Phi_{old} + \alpha\Delta\Phi$$

Equation 1 Variable change with under relaxation factors (Fluent, 2005)

The default under-relaxation parameters in Fluent are set to values that are near optimal for the largest possible number of cases, and for most flows do not require modification. However, if residuals display divergent behavior after several iterations, then the under relaxation factors need to be further reduced.

PRESSURE-VELOCITY COUPLING

A pressure velocity coupling method is required for the segregated solver to link the pressure field to the velocity field, since the momentum and continuity equations are solved separately in the segregated solver. The SIMPLE (Semi-Implicit Method for Pressure Linked Equations) pressure-velocity coupling method for steady flows was used for this analysis.

MATERIAL PROPERTIES OF THE FLOW

In this analysis, density is modeled as constant, since the flow regime is incompressible and the problem does not involve heat transfer. In addition, dynamic viscosity is modeled as constant, as there is no heat transfer. The constant density and dynamic viscosity of air was 1.225 kg/m^3 and $1.7894\text{E-}5 \text{ kg/m-s}$ respectively for standard atmosphere at sea level (Munson, 2002).

TURBULENCE MODELS

A RANS (Reynolds Averaged Navier Stokes) turbulence model, in which all scales of turbulence are modeled, was employed in this analysis. In particular, the two-equation $k-\epsilon$ Realizable model was used.

NEAR WALL TREATMENT

The near wall region can be subdivided into three layers, the viscous sublayer, which is the layer adjacent to the wall where the flow is almost laminar, the buffer layer and then the fully turbulent layer, also known as the log law region. These various regions of the boundary layer are represented in Figure 3 below, which displays a graph of u^+ against y^+ . The variable u^+ is velocity, u , divided by the friction velocity, u_τ . The variable y^+ is a non-dimensional parameter that indicates a fluid particles position in the boundary layer. Hence, in terms of y^+ , the viscous sublayer begins at 0 and ends at approximately 5. Then, the buffer layer begins after the viscous sublayer and spans to a y^+ of 30. The fully turbulent layer, i.e. the log law region, then starts from 30 and continues to a y^+ of around 400 (Kelso, 2005). The significance of y^+ is that it is a guide to the meshing requirements for the near wall region discussed later in the near wall models.

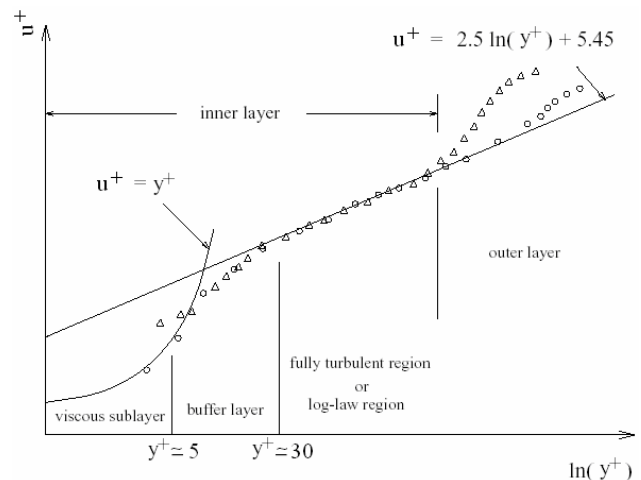


Figure 3 Decomposition of the Near Wall Region (Fluent, 2005)

y^+ is calculated using Equation 2, where ρ is the fluid density, y is the distance from the surface of interest, μ is the dynamic viscosity and μ_τ is the friction velocity, which is a property of the flow and given by Equation 3. Hence, from equation 3, as the distance from the surface, y , increases, then y^+ also increases.

$$y^+ = \frac{\rho y \mu_\tau}{\mu}$$

Equation 2 y^+

$$\mu_{\tau} = \sqrt{\frac{\tau_w}{\rho}}$$

Equation 3 Friction velocity

There are two general methods to model the near wall region, the near wall model approach and the wall function approach, which are both illustrated in Figure 4.

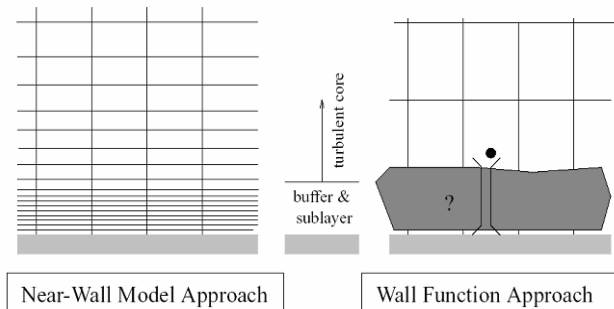


Figure 4 Near-Wall Model and Wall Function Approach (Fluent, 2005)

The near wall model involves resolving the viscous sublayer and buffer layer all the way down to the wall, whereas in the wall function approach, the viscous sublayer and buffer layer are not resolved and are instead approximated by empirical wall functions.

The advantage of the near wall model approach is that it is the most physically sound model. However, the limiting factor of the near wall models applicability is that it imposes a very large computational requirement, as the near wall region must be resolved with a fine mesh to resolve the viscous sublayer. The mesh requirements for this model is that at least ten grid points lie within the viscous sub layer and the first grid point must have a y^+ at least less than 1 (Fluent, 2005). This model is not appropriate for this analysis, since the number of elements required for an entire aircraft would be too demanding. Instead, the wall function approach is utilized for this analysis. The advantage of the wall function approach is that the model substantially saves computational resources and still gives reasonably accurate predictions for high Reynolds number flows (Christo, 2007). However, the disadvantages of the wall function approach are that the model is inadequate for low Reynolds number effects and not valid for flow separation (Fluent, 2005). The mesh recommendation for this approach is that the first grid point must lie just outside the buffer layer, hence at a y^+ of 30. Excessively fine meshes near the walls, i.e. a y^+ below 30, should be avoided since the wall functions cease to be valid in the viscous sublayer. In addition, Fluent suggests that y^+ for the first grid point can extend up to 400 for accurate results (Fluent, 2005).

BOUNDARY CONDITIONS

Boundary conditions are specified for the flow inlet, flow outlet, aircraft surfaces, flow volume walls and the symmetry plane of the aircraft. The flow inlet is defined as a velocity inlet. The velocity at the inlet is 82 m/s (160 knots), which is the maximum cruise speed of the DHC Twin Otter seaplane (Trans Maldivian Airways, 2006). The flow outlet is defined as a pressure outlet. The outlet gauge pressure is set to zero. Gauge pressure is the pressure relative to the local atmospheric pressure. Hence, the pressure at the flow outlet is equal to the operating pressure, which is 101,325 Pascal.

The aircraft surfaces are all defined as a wall boundary. The wall boundary is specified as a no slip wall, meaning that the flow velocity at the surface is zero. Hence, the fluid sticks to the wall and moves with the same velocity as the wall as expected in the actual flow. In addition, the wall roughness of the wall is modeled as smooth, so the roughness height, K_s , is set to zero. The flow volume walls are also defined as wall boundaries. It is desired that the outer walls of the flow volume have no effect on the flow and hence are modeled as a slip wall with zero roughness. A slip wall means that the wall has no viscous effects on the flow and is modeled by specifying a wall shear stress of zero.

The symmetry plane of the aircraft is defined as a symmetry boundary. Symmetry boundary conditions can only be used when both the physical geometry and the expected flow pattern have mirror symmetry about that plane, which are both satisfied for the symmetry plane of the aircraft. Note a symmetry boundary does not take into account the flow on the other side. A symmetry plane is merely represented by having zero normal gradients of all variable at the symmetry plane. Hence, the force output does not take into account the other half of the aircraft, so therefore must be doubled to yield the force on the entire aircraft.

TURBULENCE QUANTITIES AT INLET

Turbulent kinetic energy, k , and turbulent dissipation rate, ϵ , must be specified at the flow inlet in order to estimate the incoming turbulence for the k - ϵ turbulence model. Firstly, the turbulent kinetic energy, which is the mean kinetic energy per unit mass associated with eddies in turbulent flow, is determined by Equation 4, where U is the initial velocity magnitude and I is the initial turbulence magnitude. Initial turbulence magnitude, I , is approximated using Equation 5, where Re is the Reynolds number.

$$k = 1.5U^2 I^2$$

Equation 4 Turbulent kinetic energy (Fluent, 2005)

$$I = 0.16 Re^{-\frac{1}{8}}$$

Equation 5 Initial turbulence magnitude (Fluent, 2005)