Preliminary CFD Investigation of De/anti-icing Fluid Behavior on a Flat Plate

Pekka Koivisto, Mikko Auvinen

Trafi Research Reports
Trafin tutkimuksia
Trafis undersökningsrapporter
5/2015

Preliminary CFD Investigation of De/anti-icing Fluid Behaviour on a Flat Plate

Pekka Koivisto, Mikko Auvinen

Aalto University Department of Applied Mechanics, Arteform Oy

Finnish Transport Safety Agency
Liikenteen turvallisuusvirasto Trafi
Trafiksäkerhetsverket Trafi
Helsinki Helsingfors 2015

ISBN 978-952-311-071-7
ISSN 2342-0294

FOREWORD

This research report is focused on possibilities to simulate De/anti-icing fluid behavior on a flat plate in an airstream over it using methods of computational fluid dynamics (CFD. It forms part of the third year studies in the Icewing project initiated by the Finnish Transport Safety Agency, Trafi.

The research was performed by the team of Arteform Oy, headed by MSc Juha Kivekäs.

Helsinki, September 15th 2015

Erkki Soinne

Chief Adviser, Aeronautics

Finnish Transport Safety Agency, Trafi

Index

ABSTRACT

ABSTRACT

A preliminary investigation on the possibility to utilize CFD to simulate the behavior of a de/anti-icing fluid in airflow passing a flat plate using CFD-calculations was carried out as a part of Ice-wing 3 project. The flow situation was considered as a two dimensional problem. The applied software was OpenFOAM with two different two-phase solvers: multiphaseEulerFoam solver with the most complete physics modeling available and multiphaseInterFoam solver which includes a more limited range of physical mechanisms. The latter solver turned out to be a more preferable option due to its stability and CPU-efficiency properties. Both Newtonian (deicing) and non-Newtonian (Anti-icing) fluids were studied. In case of Newtonian fluids the preliminary results were encouraging giving new perspectives in mechanisms of thin fluid film flow in an airstream. The non-Newtonian fluids turned out to be more challenging due to the lack of knowledge on time dependent rheology of these fluids.

Nomenclature

K	proportionality factor
L	length of flat plate
n	power law constant
u	fluid velocity in x-direction
U	air velocity in x-direction
w	wave speed
y^+	$u_\tau y/\nu$ where u_τ is friction velocity, y is coordinate perpendicular to surface and ν is the kinematic viscosity
μ_a	apparent viscosity

Background and Objectives

The objective of this preliminary numerical investigation is to simulate the airflow driven transport of Newtonian and non-Newtonian de/anti-icing fluid layers on a flat plate. This is a stratified two-phase flow problem where the two fluids are immiscible (unmixable). The lighter fluid (air) is in a gas phase and the heavier fluid (de/anti-icing fluid) is in a liquid phase. Ultimately the goal is to consider lifting surfaces, such as wings in take-off configuration but at this preliminary stage the CFD investigation is constrained to a simple flat plate problem.

Considering the nature of the flow problem, the starting point is challenging: In reality, when the anti-icing fluid is spread over a surface, it forms a 1-2 mm thick layer which naturally becomes thinner at the edges. Therefore, the fluid layer is never subjected to a pure shear-flow, which would offer the possibility to study Kelvin-Helmholtz instability of the fluid interface. Instead, the front of the fluid layer features some frontal area where pressure force originates the first wave formation. Once this initial wave has formed at the front of the plate, the air flow separates at the wave causing the flow downstream of the wave to become unsteady, dominated by vortex shedding. Thus, the relevance of considering a fully developed air flow over a liquid interface is practically lost. All efforts that attempt to find consistency with experiments need to take into account the actual, unsteady flow conditions over the liquid interface. The experiments have confirmed that the liquid layer forms mostly two-dimensional waves which traverse across the surface, either merging with smaller and slower waves or ending up being absorbed by larger ones. Naturally there is some three-dimensionality as the waves seldom reach over the whole span of the model. It would be highly encouraging if the 2D simulations were able to capture this dynamic interaction between waves in a reasonable accordance with experiments. For this reason the calculation domain has been selected to simulate one of the wind tunnel models tested during the Icewing 3 – project.

Objectives may be summarized as follows:

- Establish the numerical framework which is capable of simulating the two-phase flow problem adequately. This includes setting up the inlet gas phase flow condition as a time-dependent ramp to mimic accelerated flow conditions in the wind tunnel experiments.

- Obtain estimates for the characteristic wave speeds, wave lengths, wave heights and overall convection rate of the liquid phase (volume flow). Other interesting measures include the onset airspeed, at which the first wave forms, and the change of liquid layer thickness with time.

- Build templates for rapid case setup, post-processing and analysis.

As stated above the CFD results will be validated by wind tunnel experiments with a flat plate model.

1 Software and Computational Approaches

The simulation problem considered was solved using an open source CFD-code known as OpenFOAM. This code may be regarded as state of the art software for the kind of problems considered in the present study.

OpenFOAM offers two potential techniques to simulate flows of multiple immiscible (unmixable) fluids. Both are based on Euler-Euler approach where all fluid phases are modeled as interpenetrating continuum, each according to Navier-Stokes equations with additional terms that account for the momentum exchange between the phases and interfacial forces.

In OpenFOAM 2.3.x the first solver option is called multiphaseEulerFoam. The solver algorithm is based on explicit time-stepping and treats the phases as compressible fluids. It offers the most complete physics model including, for instance, heat transfer. But, in its current implementation, it does not support non-Newtonian viscosity models. The alternative approach is offered by multiphaseInterFoam which is an incompressible solver for multiphase systems. multiphaseInterFoam captures the behavior of the interface and includes surface-tension and contact-angle effects for each phase. Dispersed phases (in this case droplets of fluid below the size of calculation grid size) are better simulated with multiphaseEulerFoam, however multiphaseInterFoam also supports non-Newtonian viscosity models. The latter solver has also more favorable numerical characteristics (implicit formulation for momentum and thus better stability) and allows the utilization of either LES or RANS turbulence models. If this solver ends up yielding meaningful results, it would be the preferable option due to better stability and computation efficiency.

Ultimately, the comparison between the experimental and computational results will dictate which solver should be utilized in future studies.

As stated above the gas phase incorporates unsteady vortex shedding. For this reason a LES-turbulence model was selected to catch the airstream induced pressure and shear forces that transfer the liquid as accurately as possible.

2 Calculation Domain, Grid, Time Steps and Boundary Conditions

The calculation domain, grid structure and boundary conditions are illustrated in Figures 1 and 2. As the results are intended to be validated by wind tunnel tests at Aalto University (see Chapter 1. Background and Objectives) one of the three wind tunnel models were to be selected as a model for the CFD domain flat plate. To achieve reasonable computing times the flat plate model with length of 0.6 m were selected. The grid was refined in vertical direction progressively in two steps: first within 165 mm and then within 1.6 mm from the flat plate surface (Fig. 2). To simulate the eddies near the air-liquid interface as accurately as possible the grid was refined also in horizontal direction close to the interface and wall (see Fig. 2). As there has to be a refinement on this area in both vertical and horizontal direction rectangular cells could not be used. GridPro-software was used for grid generation.

For the gas phase no y^+ - limit was set as the grid size was determined according to the following facts:

- At the leading edge area of the flat plate the flow is very progressive and the boundary layer of the gas phase never develops to relevant dimensions for y^+-analysis

- At the leading edge area a small error may be easily tolerated near the surface as it does not affect the solution of the gas phase as it meets the surface of the liquid phase which after the flow will be rearranged
- The most important feature of the grid size in the vicinity of the plate surface is to be as isotropic as possible and fine enough to catch the interaction between phases and the evolution of the liquid surface
- The fluid (liquid) velocity scale near the flat plate wall is so small that the condition $y^+ < 1$ will easily be fulfilled

The gas flow undergoes perpetual fluctuations, forming vortices at the front of the liquid layer. These vortices are convected downstream over the liquid interface. On a flat plate with a length of 0.6 m and airspeed of 10m/s, the lifetime of a fluid particle in the domain is roughly T_{gas} = 0.06s. We can consider this as a global time scale for the gas phase vortices. In a stark contrast, the highly viscous liquids have barely formed a single wave at the front part of the liquid layer after 0.5s after the gaseous fluid particle entered the domain. This indicates that the global time scale of the waves can reasonably be expected to be $T_{liq} > 100\ T_{gas}$. This is problematic as the numerical requirements of the CFD simulation are dictated by the vortex shedding dominated gas phase physics as well as the interface behavior. These demand an extremely high grid resolution (particularly at the interface) and very small time steps to capture the flow

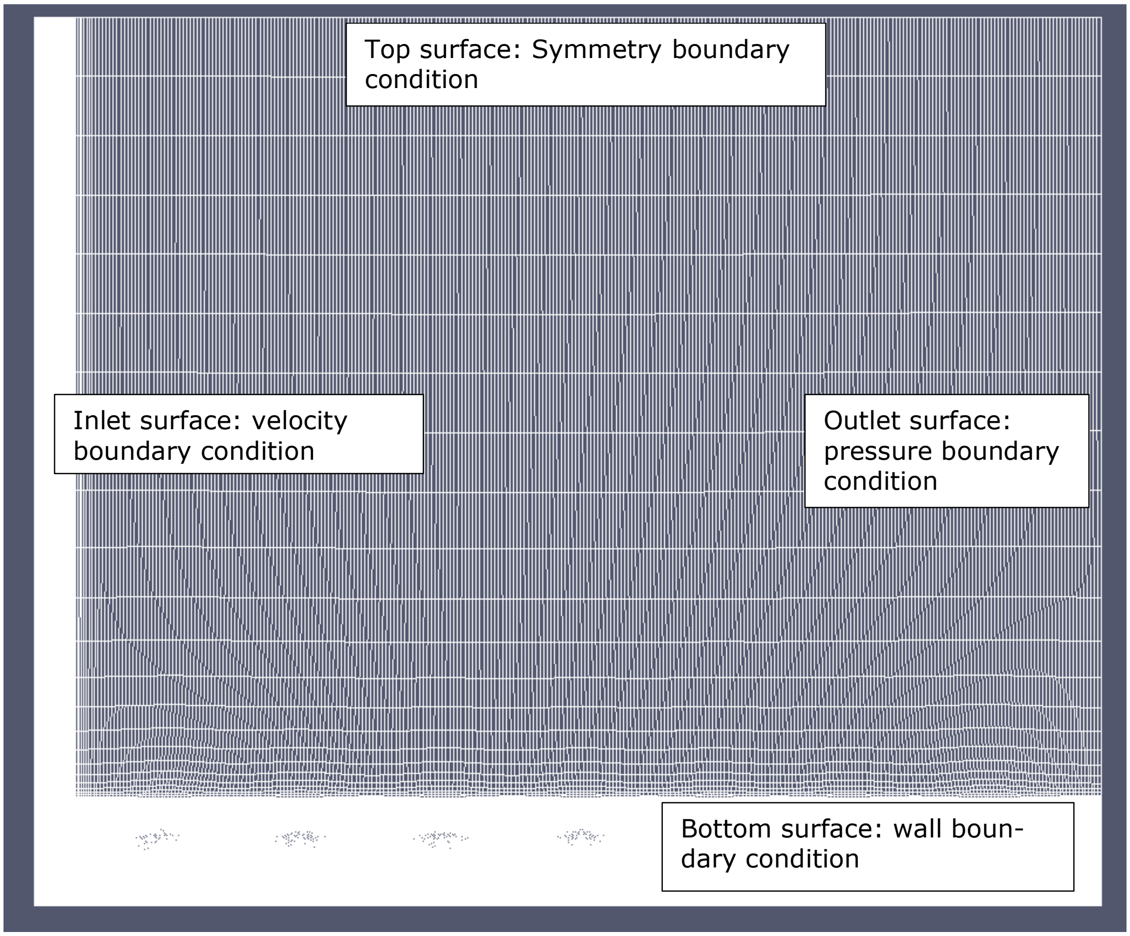

Figure 1. The calculation domain boundary conditions and grid structure (165068 cells). The length of the domain is 0.6 m and the height is 0.5 m.

system's behavior adequately. The grid includes all together 165068 cells. A typical time step in the 2D flat plate simulations is roughly $5*10^{-6}$ s, which unavoidably leads to very long computational times. A 5 second simulation can take up to 2 weeks on a modern computer with 2 CPUs.

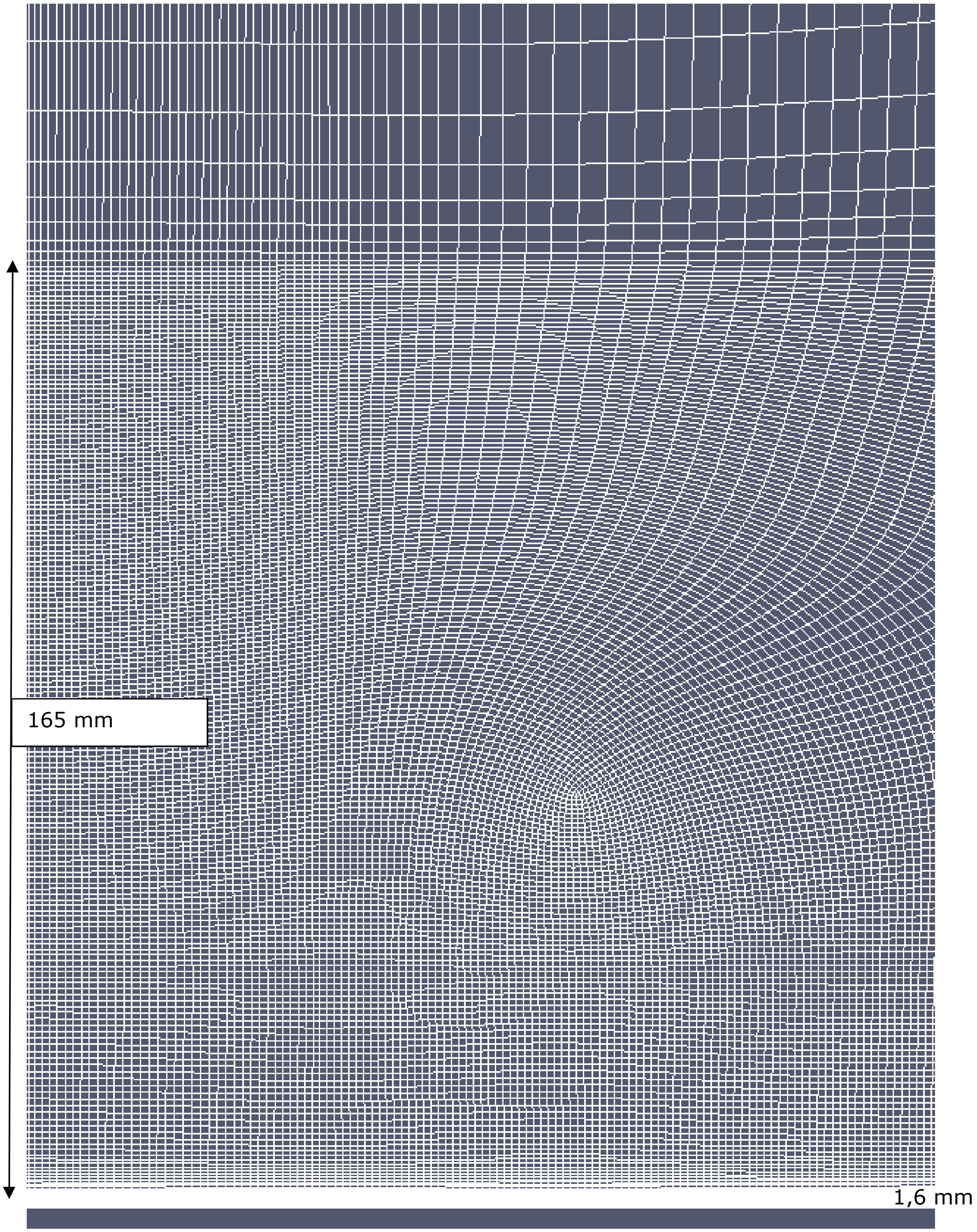

Figure 2. Grid refinement in vertical and horizontal directions.

3 Results

Several trials were performed with different fluid and solver type combinations. Instructive results and guidelines for further work were achieved with the following cases (initial fluid layer thickness was 1 mm in all cases):

- Water with multiphaseEulerFoam solver and a stepwise increasing airspeed of 10 m/s
- Newtonian de-icing fluid with multiphaseInterFOAM solver and a progressively increasing airspeed (in time)
- Non – Newtonian anti-icing fluid with multiphaseInterFOAM solver and a progressively increasing airspeed

The simulations which were initialized with a constant 10 m/s velocity profile at the inlet resulted in undesirable behavior during the early phase of the simulation. When the sharp flow front became incident with the liquid interface, a sheet of de/anti-icing fluid was rapidly sheared off from the liquid layer and broken into small droplets, which were subsequently convected downstream. This slowed down the computation considerably as the time step had to be dynamically reduced to sustain numerical stability.
To avoid this problem a progressively increasing speed ramp was introduced for de/anti-icing fluids (Fig. 3). This leads to a realistic wave formation in the leading edge area.

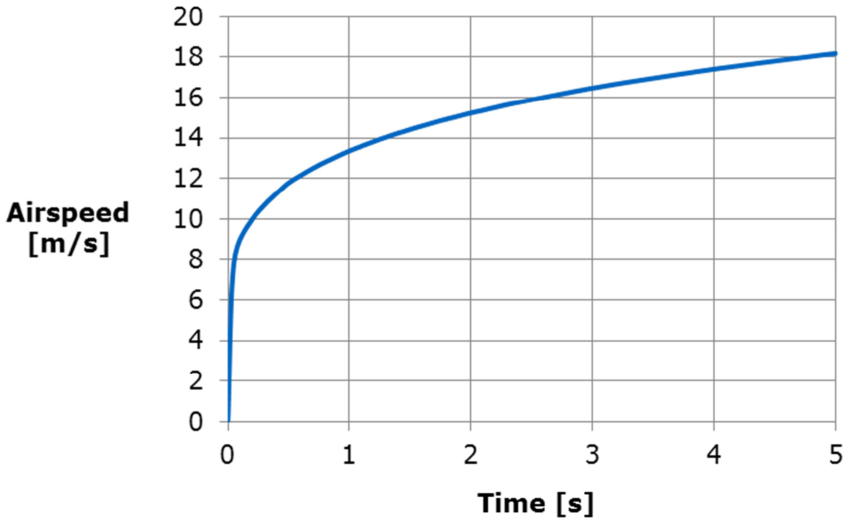

Figure 3. Progressively increasing airspeed applied to calculations for de/anti-icing fluid.

Figure 4 illustrates the water surface contour on the first 0.2 m from the flat plate leading edge at time t = 0.1 s from the start of airflow. The separated drops of water related to the multiphaseEulerFoam model are visible. Also note the small waves (ripples) that actually extended over the whole liquid surface instantly after a few time steps from the beginning of the calculation.

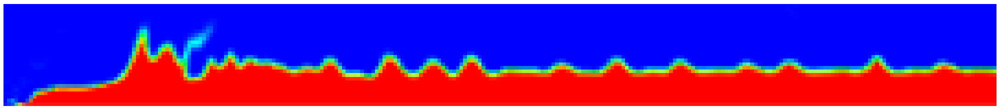

Figure 4. Water film surface on the first 0.1 m from the leading edge in 10 m/s airflow at time t = 0.1 s from the beginning of airflow onset. Note the separated water drops over the surface and ripples on the liquid film surface. Initial film thickness is 1 mm. Note: scale magnified in y-direction with a factor of 7.

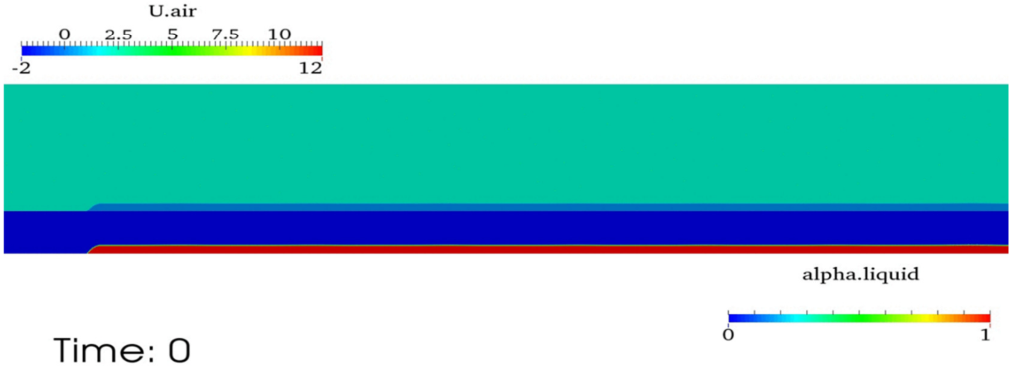

Figure 5. Surface (red) and velocity (blue) of a Type I de-icing fluid at rest (t=0). The full animation is available in Ref. 1.

Figure 5 presents the initial stage of computed animations of the fluid surface and air and fluid speeds during 5 s from airflow onset for a deicing fluid. Animations for water, Type I deicing fluid and Type IV anti-icing fluid are available as a Powerpoint presentation[1].

In Figure 6 the initial stage (t=0) is presented of an animation about local viscosity changes in Type IV fluid. The non-Newtonian viscosity of Type IV fluid is modelled in OpenFOAM code using a power law:

$$\mu_a = K\,(du/dy)^{n-1} \qquad (1)$$

where μ_a is apparent viscosity, u fluid velocity in x-direction, K proportionality factor **n** is 0,67 for TIV fluid in consideration.

The properties of fluids used in calculations are collected to Table 1. The Type IV fluid was a 25% diluted water – TypeIV – mixture which is less viscous than neat Type IV fluid.

Table 1. Fluid properties as inputs to OpenFOAM code.

	Density	**Dynamic viscosity**	**Surface tension**
	[kg/m^3]	**[cP]**	**[mN/m]**
Water	1000	1.0	72.0
Type I	1040	20.0	35.5
Type IV	1060	168(du/dy)$^{-0.33}$	29.8

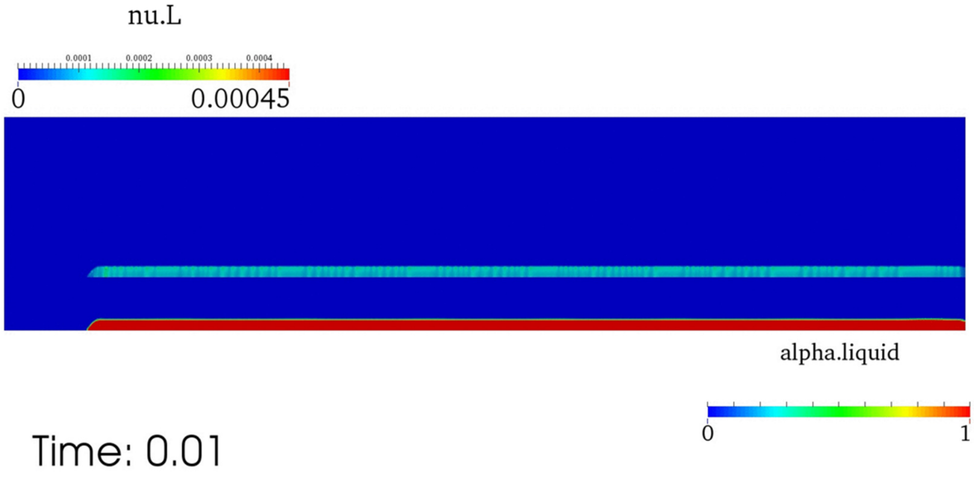

Figure 6. Surface (red) and local viscosity (light blue) of a Type IV de-icing fluid at rest (t=0). The full animation is available in Ref. 1.

The two animations (Figures 5 and 6) referred above[1] demonstrate the following details in the de/anti-icing fluid wave motions on the surface:

- Wave motion starts via solitary waves in the leading edge area
- Within 1.5 s there appears a clearly observable solitary wave at the trailing edge of the plate although the wave front which originated on the leading edge, is still at half way from the trailing edge.
- Wave speeds vary with wave height. The higher wave the faster it moves
- Merging of waves as well as die out of solitary waves occur
- In the non-Newtonian viscosity model the local viscosity value changes instantly without any regard to relaxation time (or viscosity time dependency). This is a source of inaccuracies in the model

The last point needs a further explanation. As the power law does not include any time factor of viscosity restoring properties the viscosity follows immediately - with no time delay - the shear (du/dy) in the fluid. In reality there is a delay in restoring properties of a fluid. The once decreased viscosity may stay at a low level although the local shear level decreases temporarily.

Regarding wave onset airspeed U and wave speeds w the results of OpenFOAM calculations were in line with measurements of onset airspeeds and wave speeds on a wing model of 0.65 m tested with anti-icing fluids in Aalto University Wind Tunnel[2]. The wave speed results of OpenFOAM calculations are illustrated in Figure 7 among some measurements of anti-icing fluid wave speeds on the wing model referred above[2]. The wave onset airspeed was calculated to be around 10 m/s for both fluids.

Most of the earlier published studies[3-19] on airflow over an anti/deicing fluids consider a more or less steady laminar or turbulent boundary layer over the wavy fluid layer which is considered as a rough surface. However this study revealed that there is a clearly unsteady airflow dominated by vortex shedding over the liquid wave surface. This is illustrated in Figures 8 and 9. Air vortices are moving at a speed close to air free stream velocity being two orders of magnitude more than speed of liquid waves. The air vortices produce strong pressure gradients as may be seen in Figures 10 and 11.

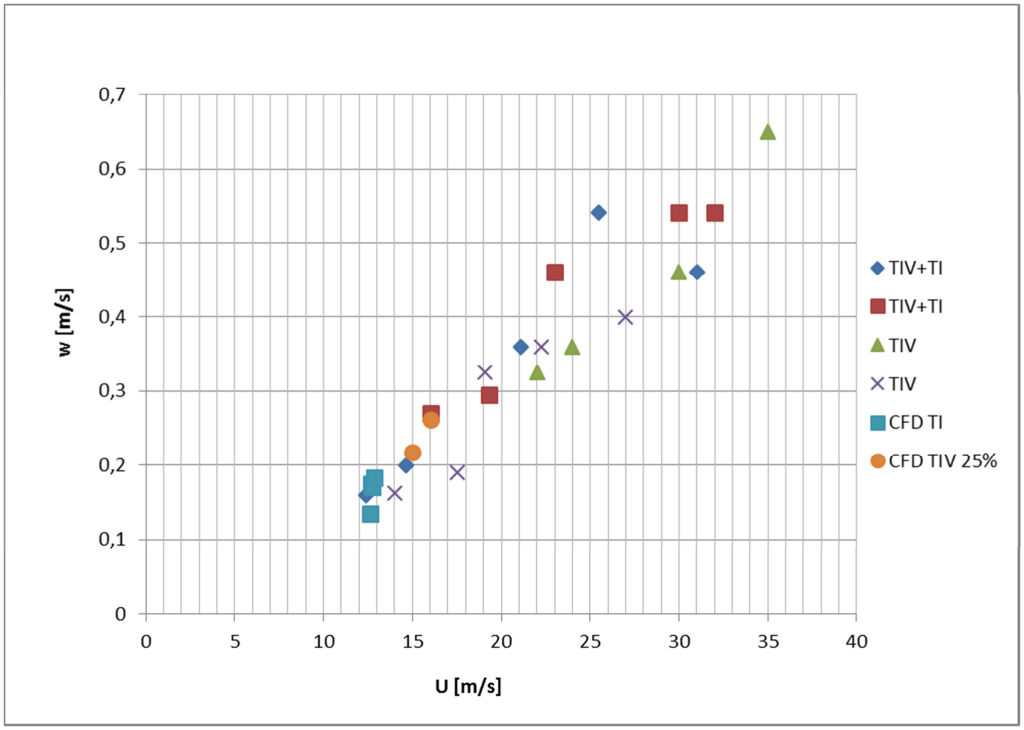

Figure 7. Comparison of wave speed variations with airspeeds between OpenFOAM calculations (blue squares and yellow circles – CFD TI and TIV) and measurements[2] (all the other results).

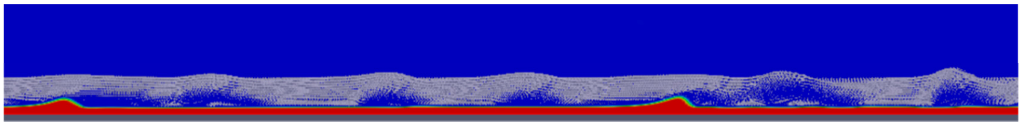

Figure 8. Vortex shedding of airflow (white arrows are airspeed vectors) over wavy liquid surface (red surface).

Perhaps the most interesting results of the present study were related to the fluid flow off process. Many of the previous studies[4, 15] on this topic assume the fluid is transported by shear stresses. Though the detailed calculations are still to be accomplished, a preliminary analysis indicates that the pressure forces are responsible for fluid convection instead of shear stresses. Figure 10 illustrates how pressure gradient is formed around a solitary wave just after its formation. In Figure 11 a fully developed vortex shedding has made the pressure gradient field apparent.

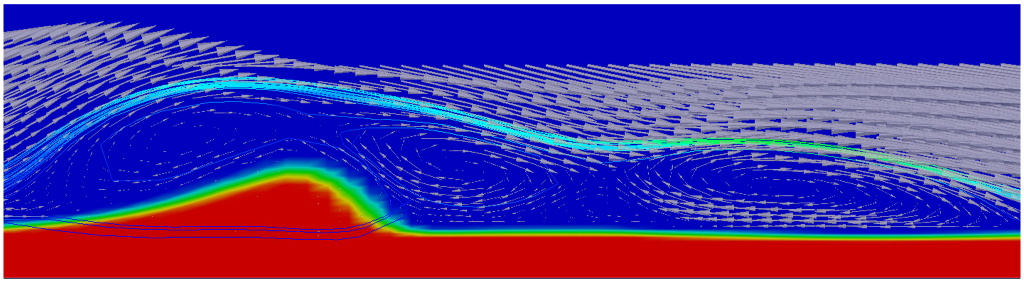

Figure 9. A magnified detail of airflow vortices over the wavy liquid surface.

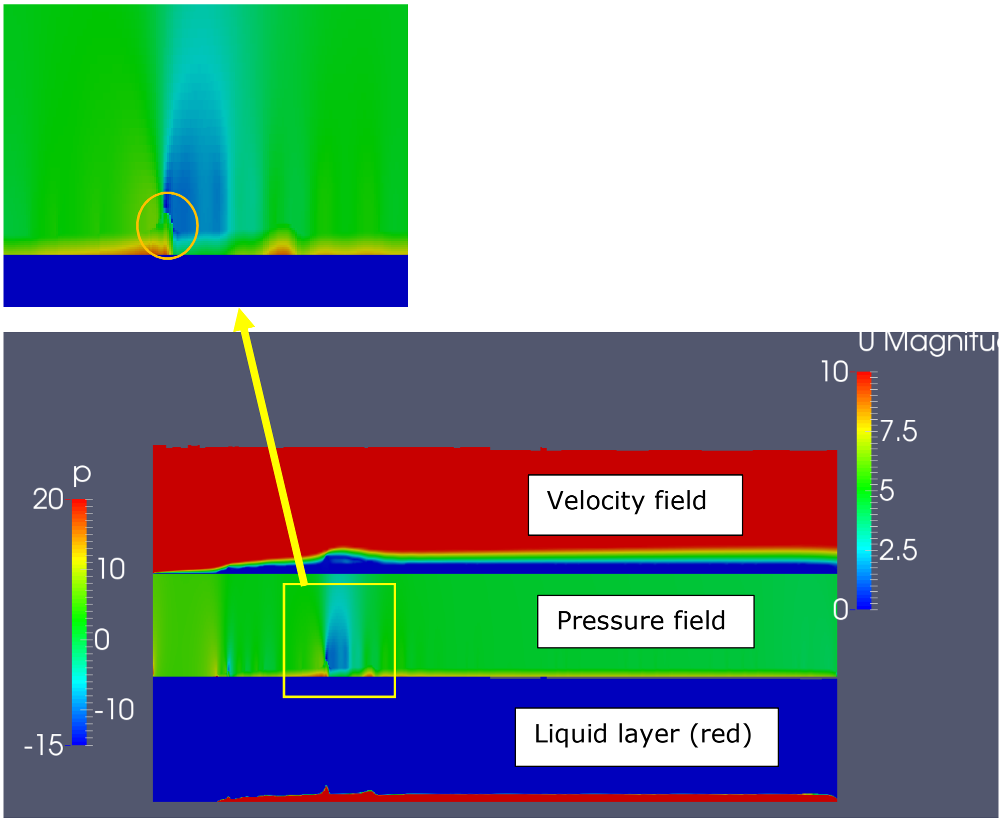

Figure 10. Pressure field around a developing wave. The lowest red layer is the liquid layer, pressure value color code is presented in left corner of the lower figure. The contour of the wave in consideration is visible inside the circle of the upper figure.

The present study also revealed how the fluid motion is quite different from the one assumed in earlier publications[3-19]. Figure 12 illustrates this phenomenon. It is evident that practically all of the fluid convection is concentrated in the wave motion. As is seen by the velocity vectors (white) and the fluid velocity component color code (ULX) in x-direction (along the plate), the majority of movement of fluid is concentrated in the wave. Outside the wave the fluid is practically stagnant.

As the maximum time interval in the calculations were 5 s there is not enough data to estimate the overall volume flow rate of the fluid in general.

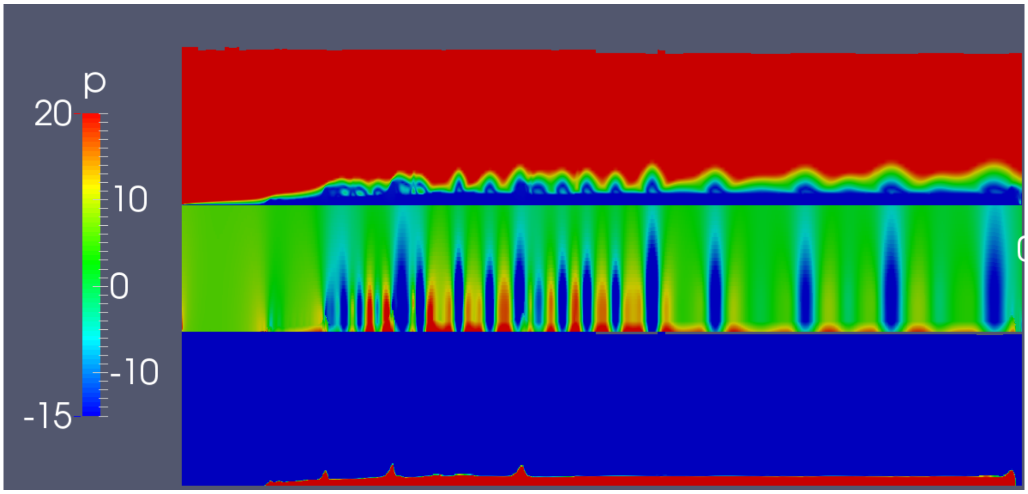

Figure 11. The centermost part of the figure illustrates the pressure field of fully developed vortex shedding. The upper figure describes the fluid velocity field and the lower figure the liquid layer contour.

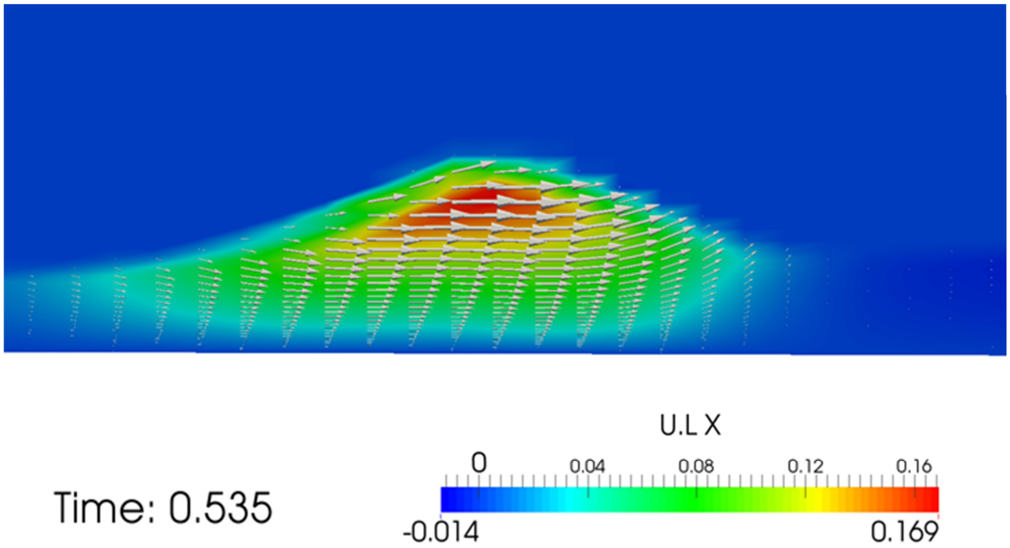

Figure 12. Liquid fluid velocity field in and around a fluid wave.

4 Conclusions

A preliminary CFD study was carried out to examine the possibilities to study de/anti-icing fluid flows on a flat plate when subjected to an airflow. The software utilized was OpenFOAM code which is regarded as a state of the art code for the kind of problems considered in this study. As the objects of the study were quite limited a 2 dimensional model was selected to save CPU time. The results were encouraging. The wave formation and wave velocities appeared to be realistic. Some new views to the behavior and reasons of and for the fluid motion under an airflow were revealed. Most of the fluid transfer is concentrated on the waves and the wave carrying forces are most probably pressure forces not shear stresses as is assumed widely in the earlier studies.

It is well motivated to continue the CFD calculations further on. The following topics should then be studied:

- Grid convergence study
- Effect of fluid layer thickness on the fluid waves and flow
- Effect of fluid viscosity (Type I), specific gravity and surface tension on the fluid motion in general
- Continue the calculations to at least 15 - 30 s to establish an estimate for fluid flow off rates
- Effect of flat plate (and fluid layer) length to discover possible scale effects
- Effect of horizontal acceleration of the frame to simulate the airliner take-off acceleration

The results of calculations with CFD shall be validated with the already ongoing wind tunnel tests.

References

1. Koivisto, P. PowerPoint presentation given at Trafi 30.10.2014. http://www.trafi.fi/filebank/a/1415257432/816813f558787f44486266fe3678f00c/16151-Icewing_esitys_Trafi_5_11_2014.pdf
2. Koivisto, P. "Effects of Anti-icing Treatment on Lift Degradation during Simulated Take-off" Trafi Publications 25/2013.
3. Yih, C.S. "Wave formation on a liquid layer for deicing airplane wings." Journal of Fluid Mechanics, 212, 1990
4. Nelson, J.J., Alving, A.E., Joseph, D.D., "Boundary layer flow of air over water on a flat plate", Journal of Fluid Mechanics, 284, 1995
5. Perron, E., Louchez P.R. and Laforte J-L., "Study of the Shearing of Ground Deicing and Anti-icing Fluids", AIAA 95-0658, 33rd Aerospace Meeting and Exhibit, Jan 9-12, 1995, Reno, Nevada
6. Perron, E., Louchez P.R.and Laforte, J-L., "Introductory Analysis of Boundary layer Development on De/Anti-icing Fluid", Journal of Aircraft, Vol 33, No 1., Jan-Feb, 1996
7. Boelens, O.,J., Moeleker,P.J.I, deJong, H: and Hoejmakers, H:,W.,M., "Numerical Methods For Simulating the Flow Over an Airfoil, Covered with a thin Layer of Liquid, ICAS-96-1.4.3
8. Tsao, J-C., Rothmayer, A.P., and Ruban, A.I., "Stability of airflow past thin liquid films on airfoils", AIAA 96-2155, Jun 1996.
9. Tsao, J-C., Rothmayer, A.P., and Ruban, A.I., "Stability of airflow past thin liquid films on airfoils", Computers and Fluids Vol. 26, No 5, pp. 427-452, 1997.
10. Myers, T.G. and Thompson, C. P., "Modeling the Flow of Water on Aircraft in Icing Conditions.", AIAA journal, Vol 36, No 6, Jun 1998.
11. Boelens, O.J. and Sijp, J.C., "Investigation into the Formation of Waves on Thin Layers of De/Anti-icing Fluids on Wings." ICAS – A98-31718.
12. Özgen, S., Degrez,G. and Sarma, G. S. R., ''Two-fluid boundary layer stability,'' Phys. Fluids 10, 2746 ~1998.
13. Miesen, R. and Boersma , B. J. ''Hydrodynamic stability of a sheared liquid film,'' J. Fluid Mech. 301, 175 ~1995
14. Boelens, O.J. and Hoeijmakers, H.W.M, "Investigation into the Formation of Waves on Thin Viscous Fluid Layers", AIAA 99-0621, 37th Aerospace Sciences Meeting and Ex-hibit, Jan 11-14, 1999, Reno, Nevada
15. Dart N.P., "Numerical Analysis of Deicing Fluid Flow Off from Aircraft Wings", AI-AA 2010 – 7838, Atmospheric and Space Environments Conference 2-5 Aug. 2010, Toronto, Ontario, Canada
16. Timoshin, S.N. ''Instabilities in a high-Reynolds-number boundary layer on a film coated surface,'' J. Fluid Mech. 353, 163 ~1997.
17. Craik, A.D.D:''Wind generated waves in thin liquid films,'' J. Fluid Mech. 26, 369 -1966
18. Rothmayer A.P., Matheis B.B. and Timoshin S.N., "Thin liquid films flowing over external aerodynamic surfaces", Journal of Engineering Mathematics, 42, 341-357, 2002.
19. Vlachomitrou M. and Pelekasis N., "Short to long wave resonance and soliton formation in boundary layer interaction with a liquid film", J.Fluid Mech., vol 660, pp 162-196, 2010.