ABSTRACT

In the current study, continued efforts to improve a computational in-flight ice accretion prediction tool are introduced together with obtained results. The computational tool follows the usual procedure for computing ice shapes around three-dimensional bodies like wings, intakes, etc., i.e., flow-field calculation, droplet trajectory determination, droplet collection efficiencies calculation, convective heat transfer coefficient distribution computation and finally ice accretion rates determination using the Extended Messinger Method. Finally, integration of ice accretion rates over time yields the ice shapes and the final geometry. The emphasis in this study is on parallel computation of the droplet trajectories using the Langrangian approach. Since almost the entire computational time is used by the calculation of droplet trajectories in the developed computational tool, parallel computation allows fast and accurate analyses to be performed in a fraction of the time required for sequential computing. This allows parametric analyses covering a large number of parameters over wide ranges to be performed during design, development and certification of air vehicles. Large droplet effects, such as non-spherical particles, droplet breakup and droplet splash are accounted for during the computations.

INTRODUCTION

In-flight icing on airframe components is one of the most important problems of aviation. Ice formation on the wings, tail surfaces or sensors like pitot tubes and angle of attack sensors seriously threaten flight safety as the aerodynamic performance and control characteristics become seriously and often unpredictably degraded.

In order to demonstrate that an airplane can fly safely in icing conditions, certification authorities like FAA and EASA have defined meteorological conditions that the airplane must be tested against in flight tests, laboratory tests and computer simulations. An important aspect of computational simulations is accurate representation of droplet physics, taking trajectories, deformation, breakup and splash into account. Unless droplet physics is faithfully represented in the computational tool, accurate estimation of droplet collection efficiency, which constitutes one of the most important parameters for ice accretion rate calculations will not be reliable. However, computation of droplet trajectories constitutes the most time-consuming step in computational simulations, particularly if the Langrangian approach is used. Therefore, slicing computational time is important for studies involving high number of parameters to be swept over wide ranges.

In the current study, the wing is divided into a finite number of spanwise segments, like 2, 4, 8, 16, and 24 for which trajectory calculations and subsequent droplet collection efficiencies are determined. The emphasis is on the acceleration of the analysis through the use of parallel computing. Additionally, the dependence of the results on the number of spanwise segments is also addressed to.
METHODS

In this Section, the method developed for ice accretion calculations applicable to three-dimensional geometries is summarized. A brief flowchart of the calculation procedure and the developed program is presented in Figure 1, modified for parallel approach from [1].

METHOD FOR COMPUTING THE FLOW-FIELD

In order to compute the air flow velocity components so that the droplet trajectories can be calculated, a panel method is utilized [2]. In this method, the wing is divided into quadrilateral panels each associated with a singularity element. The strengths of the singularities are taken to be constants and are unknowns of the problem. The developed computer program uses $N$ quadrilateral panels to solve for $N$ singularity strengths using the flow tangency boundary condition at the collocation points of the panels. The collocation points are the centroids of each panel. Once the singularity strengths are calculated, one can construct a velocity potential and hence calculate the air flow velocity components at any location in the flow field. The velocity components at a given point are the $x$, $y$, $z$-derivatives of the velocity potential constructed at that point. The results of the panel method also serve the boundary-layer calculations, for which the inviscid velocity distribution around the geometry is required. The boundary layer calculations are performed in order to calculate the convective heat transfer distribution around the geometry.

METHOD FOR COMPUTING THE DROPLET TRAJECTORIES

For droplet trajectory computations, it is assumed that the presence of the droplets do not affect the flow field. Gravity and aerodynamic drag are the only forces acting on the droplets. Non-spherical particles are accounted for, which becomes important for large droplets. The droplet trajectories are computed using the following equations:

$$m\ddot{x} = -D \cos \gamma_1,$$

$$m\ddot{y} = -D \cos \gamma_2,$$

$$m\ddot{z} = -(D \cos \gamma_3 + mg),$$

with

$$\gamma_1 = \tan^{-1}\frac{\dot{x}}{V_{rel}}, \quad \gamma_2 = \tan^{-1}\frac{\dot{y}}{V_{rel}}, \quad \gamma_3 = \tan^{-1}\frac{\dot{z}}{V_{rel}},$$
In the above equations, $V_x$, $V_y$ and $V_z$ are the components of the flow velocity at the droplet location, while $\dot{x}_P$, $\dot{y}_P$, $\dot{z}_P$, $\ddot{x}_P$, $\ddot{y}_P$ and $\ddot{z}_P$ are the components of the droplet velocity and acceleration. The symbols $\rho$ and $A_P$ denote the atmospheric density and cross-sectional area of the droplet. $C_D$ denotes the droplet drag coefficient. For the calculation of the drag coefficient, the model by Clift, Grace and Weber, which takes non-spherical shapes of large droplets into account, is used [3, 4]:

$$C_D = eC_{D,\text{disk}} + (1-e)C_{D,\text{sphere}}.$$  \(\text{(7)}\)

According to this model, the effective droplet drag coefficient is obtained by interpolation between a disk and a sphere, where the eccentricity, $e$, depends on the Weber number:

$$e = 1 - \frac{1}{(1 + 0.007\sqrt{We})^6},$$  \(\text{(8)}\)

where $We = \rho V_{rel}^2 d_p / \sigma_w$ is the Weber number.

The trajectory calculations start from an upstream location far away from the wing leading edge so that air flow velocity components are sufficiently close to their freestream values. The initial droplet velocity is taken to be the terminal velocity [4]:

$$V_{term}^2 = \frac{4}{3} \frac{(\rho_w - \rho) g d_p}{\rho C_D}.$$  \(\text{(11)}\)

The droplet trajectories are obtained by integrating equations (1, 2, 3) over time until the droplet impacts the geometry. The droplet impact pattern on the section determines the amount of water that impinges on the surface and the region subject to icing. The local collection efficiency is defined as the ratio, of the area of impingement to the area through which water passes at some distance upstream of the section, $\beta = A_o / A$.

If the droplet size and velocity is high enough, the droplet can breakup into smaller droplets due to shear forces acting on its surface. Droplet breaks up when a threshold value of the Weber number is exceeded. Although the choice of the threshold value varies widely in the literature, a value of 12 has been chosen in this study. After breakup, the trajectory calculation is resumed with the secondary droplet until impact. The secondary particle diameter is obtained from [5]:

$$d_s = 6.2 \left(\frac{\rho_w}{\rho}\right)^{0.25} \left(\frac{\sigma_w}{\rho V_{rel} d_p / \mu_w}\right)^{0.25} \text{Re}_{w}^{-0.5} d_p \text{ with } \text{Re}_{w} = \rho_w V_{rel} d_p / \mu_w.$$  \(\text{(12)}\)
After a droplet impacts the surface, part of its mass will impinge on the surface, while the remaining mass will bounce and reimpinge on the surface downstream of the initial impact point [6]. The model adopted in this study is to treat the bouncing and reimpinging water as runback water. Cossali et al [7] present their results in terms of the non-dimensional group:

\[ K = \left( \frac{\rho_d d_p V_n}{\sigma_w \mu_w} \right)^{0.25} = Oh^{-2/5} We_n. \]  

(13)

In the above, the Weber number, \( We_n \) is based on the normal component of droplet velocity, \( V_n \) at impact. Ohnesorge number is defined as, \( Oh = \mu_w \sqrt{\rho_d d_p \sigma_w} \). For splash threshold and properties of the splashed droplet a modified version of this parameter is employed: \( K_m = \sqrt{K} f^{-3/8} \), where \( f \) is a dimensionless droplet frequency defined as follows: \( f = (3/2)(\rho_d / \rho_w)^{1/3} \).

According to Wright [5], droplet splash occurs when \( K_m > 17 \) and splashed mass is given by:

\[ \frac{m_s}{m_o} = 0.2 \left[ 1 - e^{-0.85(K_m - 17)} \right]. \]  

(14)

where \( m_o \) is the incident mass, \( m_s \) is the splashed mass. Secondary droplet size is given by [5]:

\[ \frac{d_s}{d_p} = 8.72 e^{-0.0281K}. \]  

(15)

Figure 2 shows the calculated trajectories and Figure 3 depicts the corresponding droplet collection efficiencies. The conditions correspond to an untapered wing with 10.6 m span, 0.53 m chord with NACA 0012 airfoil at \( \alpha=4^\circ \). The freestream velocity is 58 m/s, the droplet size is 20 microns and the ambient temperature is \(-3.9^\circ \)C. In this case, the wing has been divided into 8 spanwise segments.

**METHOD FOR COMPUTING CONVECTIVE HEAT TRANSFER COEFFICIENTS**

When the collection efficiency distribution around the geometry is determined, the convective heat transfer coefficients need to be determined for the thermodynamical analysis. The geometry is divided into strips and boundary-layer calculations are performed for each of these strips by solving the two-dimensional Integral Boundary Layer equation. The boundary-layer calculations start at the stagnation point at the leading edge and proceed downstream using the marching technique for the upper and lower surfaces of the wing or duct. This method enables calculation of the details of the laminar and turbulent boundary layers fairly accurately. Since a two-dimensional approach is used, the same method used by Özgen and Cambek [1] can be adopted for this problem.

Transition prediction is based on the roughness Reynolds number, \( Re_k = \rho U_k k / \mu \), where \( k \) is the roughness height and
\( U_k \) is the local flow velocity at the roughness height. Transition from laminar to turbulent flow occurs at the streamwise location where \( Re_k = 600 \), according to Von Doenhoff criterion. The flow velocity at the roughness height is calculated by following Paraschiviu and Saeed [8]:

\[
\frac{U_k}{U_e} = 2 \left( \frac{k_s}{\delta} \right)^{3} - 2 \left( \frac{k_s}{\delta} \right)^{4} + \left( \frac{k_s}{\delta} \right) \left( \frac{dU_e}{ds} \delta \right) \left( \frac{k_s}{\delta} \right)^{3}.
\]

(16)

Above, \( U_e \) is the flow velocity outside the boundary-layer at the roughness location. Roughness height is calculated from [9]:

\[
k_s = \left( \frac{4\sigma_w \mu_w}{\rho_w F \tau} \right)^{1/3},
\]

(17)

where \( \sigma_w, \rho_w \) and \( \mu_w \) denote the surface tension, density and viscosity of water, respectively. Fraction of the wing or duct surface that is wetted by water droplets is expressed by \( F \), while \( \tau \) denotes local surface shear stress. The laminar boundary layer thickness is given by Schlichting [10]:

\[
\delta = \frac{315}{37} \theta_f.
\]

(18)

Laminar momentum thickness is computed using Thwaites' formula [10]:

\[
\theta_f^2 \frac{v}{U_e} = \frac{45}{U_e} \int_0^s U_e^5 ds.
\]

(19)

In the above expression and \( s \) is the streamwise distance along the wing or duct surface measured with respect to the stagnation point. For laminar flow \( Re_k \leq 600 \), the equation of Smith&Spalding is employed to calculate the convective heat transfer coefficient [11]:

\[
h_e = \frac{0.296k U_e^{1.435}}{\sqrt{1 - U_e^{1.87}}} ds.
\]

(20)

where \( k \) is the conductivity of air. The conductivity of air is temperature-dependent and is calculated by using viscosity computed from Sutherland's viscosity law with Prandtl number and specific heat assumed constant.

For turbulent flow \( Re_k > 600 \), the method of Kays & Crawford is employed [11]. The turbulent convective heat transfer coefficient is evaluated by using:

\[
h_c = St \rho U_e C_p,
\]

(21)

where \( C_p \) is the specific heat of air. The Stanton number can be calculated from:

\[
St = \frac{C_f / 2}{Pr_t + \sqrt{(C_f / 2)/St_k}},
\]

(22)

where \( Pr_t = 0.9 \) is the turbulent Prandtl number. The roughness Stanton number is given as:

\[
St_k = 1.92 Re_k^{-0.45} Pr^{-0.8},
\]

(23)

where \( Pr = \mu C_p / k = 0.72 \) is the laminar Prandtl number. The skin friction is calculated by using the Makkonen relation:

\[
\frac{C_f}{2} = \frac{0.1681}{\left[ \ln(864 \theta_f / k_s + 2.56) \right]^2}.
\]

(24)

The turbulent momentum thickness is computed from:

\[
\theta_f = 0.036v_{0.2} \left( \int_0^s U_e^3 ds \right)^{0.8} + \theta_{fr},
\]

(25)

where \( \theta_{fr} \) is the laminar momentum thickness at the transition location.

**METHOD FOR COMPUTING ICE GROWTH RATES, EXTENDED MESSINGER MODEL**

The ice prediction approach employed in this study is essentially the same as the one used by Özgen and Cambek [1], extended to handle ice formation on three-dimensional geometries. The ice shape prediction is based on the standard method of phase change or the Stefan problem. The phase change problem is governed by four equations: energy equations in the ice and water layers, mass conservation equation and a phase change condition at the ice/water interface [12]:

\[
\frac{\partial T}{\partial t} = \frac{k_i}{\rho_i C_{pi}} \frac{\partial^2 T}{\partial y^2},
\]

(26)
where $\theta$ and $T$ are the temperatures, $k_w$ and $k_i$ are the thermal conductivities, $C_{pw}$ and $C_{pi}$ are the specific heats and $h$ and $B$ are the thicknesses of water and ice layers, respectively. In equation (28), $\rho_d b V_{\infty}$, $\dot{m}_{in}$ and $\dot{m}_{e,s}$ are impinging, runback and evaporating (or sublimating) water mass flow rates for a control volume, respectively. In equation (29), $\rho_i$ and $L_F$ denote the density of ice and the latent heat of solidification of water, respectively. Ice density is assumed to be different for rime ice and glaze ice as shown in Table 1. The coordinate $y$ is normal to the surface. The boundary and initial conditions applicable to equations (26, 27, 28, 29) are as follows [12]:

1. Ice is in perfect contact with the wing or duct surface:
   \[ T(0,t) = T_s. \]  
   (30)

The surface temperature is taken to be the recovery temperature [11]:
   \[ T_s = T_a + \frac{V_{\infty}^2 - U_c^2}{2C_p} \left(1 + 0.2rM^2 \right). \]  
   (31)

In the above expression, $M$ is the flow Mach number given as $M = V_{\infty} / a_{\infty}$, while $a_{\infty} = \sqrt{\gamma R T_a}$ is the speed of sound. Additionally, $r$ is the adiabatic recovery factor ($r = Pr^{1/2}$ for laminar flow, $r = Pr^{1/3}$ for turbulent flow).

2. The temperature is continuous at the ice/water boundary and is equal to the freezing temperature:
   \[ T(B,t) = \theta(B,t) = T_f. \]  
   (32)

3. At the air/water (glaze ice) or air/ice (rime ice) interface, heat flux is determined by convection ($Q_c$), radiation ($Q_r$), latent heat release ($Q_i$), cooling by incoming droplets ($Q_d$), heat brought in by runback water ($Q_{in}$), evaporation ($Q_e$) or sublimation ($Q_s$), aerodynamic heating ($Q_a$) and kinetic energy of incoming droplets ($Q_d$):

Glaze ice:
   \[ -k_w \frac{\partial \theta}{\partial y} = (Q_c + Q_e + Q_d + Q_r) - (Q_a + Q_k + Q_{in}) \text{ at } y = B + h. \]  
   (33)

Rime ice:
   \[ -k_i \frac{\partial T}{\partial y} = (Q_c + Q_e + Q_d + Q_r) - (Q_a + Q_k + Q_{in} + Q_s) \text{ at } y = B. \]  
   (34)

4. Wing or duct surface is initially clean:
   \[ B = h = 0, \quad t = 0. \]  
   (35)

In the current approach, each panel constituting the geometry is also a control volume. The above equations are written for each panel and ice is assumed to accumulate perpendicularly to a panel. This is an extension of the one-dimensional model described by Myers [12] to three-dimensional, which is done by taking mass and energy terms due to runback water flow in the conservation equations into account, see equation (28).

### Rime ice growth and temperature profile

Rime ice thickness can be obtained directly from the mass conservation equation (28) as $h=0$:

\[ B(t) = \frac{\rho_d b V_{\infty} + \dot{m}_{in} - \dot{m}_{e,s}}{\rho_r}. \]  
   (36)

It has been shown that, for ice thicknesses less than 2.4 cm (which is the case for most applications), the temperature distribution is governed by $\frac{\partial^2 T}{\partial y^2} = 0$ [12]. Integrating this expression twice and applying the boundary and interface conditions given in equations (30) and (32) results in the temperature distribution in the rime ice layer:

\[ T(y) = T_s + \frac{(Q_a + Q_k + Q_{in} + Q_s) - (Q_c + Q_d + Q_e + Q_r)}{k_i} y. \]  
   (37)

### Glaze ice growth

It has been shown that, if ice and water layer thicknesses are less than 2.4 cm and 3 mm (which is the case for most applications), respectively, the temperature distributions in the ice and water layers are governed by the following equations [12]:

\[ \frac{\partial^2 T}{\partial y^2} = 0, \quad \frac{\partial^2 \theta}{\partial y^2} = 0. \]  
   (38)
After integrating above equation twice and using the conditions (30) and (32), the temperature distribution in the ice layer becomes:

\[ T(y) = \frac{T_f - T_s}{B} y + T_s. \]  

(39)

The temperature distribution in the water layer is obtained by integrating equation (38) twice and employing the interface conditions (32) and (33):

\[ \theta(y) = T_f + \frac{(Q_a + Q_k + Q_m) - (Q_c + Q_d + Q_r)}{k_w} (y - B). \]  

(40)

Integrating mass conservation equation (28) yields the water height, \( h \):

\[ h = \frac{\rho_aB \nu + \dot{m}_{in} - \dot{m}_{out}}{\rho_w} (t - t_g) - \frac{\rho_g}{\rho_w} (B - B_g). \]  

(41)

where \( B_g \) is the rime ice thickness at which glaze ice first appears and \( t_g \) is the corresponding time. When equation (41) is substituted into the phase change condition in equation (29), a first order ordinary differential equation for the ice thickness results:

\[ \rho_g L_F \frac{\partial B}{\partial t} = k \left( T_f - T_s \right) + k_w \frac{(Q_c + Q_d + Q_r) - (Q_a + Q_k + Q_m)}{k_w}. \]  

(42)

In order to calculate the glaze ice thickness as a function of time, equation (42) is integrated numerically, using a Runge-Kutta-Fehlberg method.

During transition from rime ice to glaze ice, ice growth rate must be continuous:

\[ \left( \frac{\partial B}{\partial t} \right)_{\text{rime}} = \left( \frac{\partial B}{\partial t} \right)_{\text{glaze}} \]  

at \( B = B_g \) or \( t = t_g \).

Using equations (36) and (42) yields:

\[ B_g = \frac{k \left( T_f - T_s \right)}{\left( \rho_aB \nu + \dot{m}_{in} - \dot{m}_{sub} \right) L_F + (Q_a + Q_k + Q_m) - (Q_c + Q_d + Q_r + Q_r)}, \]  

(44)

\[ t_g = \frac{\rho_r}{\rho_aB \nu + \dot{m}_{in} - \dot{m}_{sub}} B_g. \]  

(45)

Notice that equation (44) may yield negative or positive values of \( B_g \), such as:

- \( B_g \geq 0 \): equation (44) yields the ice thickness when glaze ice first appears. Consequently, equation (40) yields the time at which this happens.
- \( B_g < 0 \): indicates that glaze ice will not appear. This may be due to two reasons:
  - \( T_f - T_s < 0 \): indicating that the substrate is too warm for ice to grow.
  - The denominator of equation (44) is less than zero meaning that there is insufficient energy to produce liquid water and pure rime ice is produced.

### Table 1. Parameter values used in the icing calculations.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_a )</td>
<td>Specific heat of air</td>
<td>1006 J/kg.K</td>
</tr>
<tr>
<td>( C_{pf} )</td>
<td>Specific heat of ice</td>
<td>2050 J/kg.K</td>
</tr>
<tr>
<td>( C_{pm} )</td>
<td>Specific heat of water</td>
<td>4218 J/kg.K</td>
</tr>
<tr>
<td>( e_a )</td>
<td>Saturation vapor pressure constant</td>
<td>27.03</td>
</tr>
<tr>
<td>( g )</td>
<td>Gravitational acceleration</td>
<td>9.81 m/s²</td>
</tr>
<tr>
<td>( k_i )</td>
<td>Thermal conductivity of ice</td>
<td>2.18 W/m.K</td>
</tr>
<tr>
<td>( k_w )</td>
<td>Thermal conductivity of water</td>
<td>0.571 W/m.K</td>
</tr>
<tr>
<td>( Le )</td>
<td>Lewis number</td>
<td>1/Pr</td>
</tr>
<tr>
<td>( L_F )</td>
<td>Latent heat of solidification</td>
<td>3.344 x 10⁵ J/kg</td>
</tr>
<tr>
<td>( L_g )</td>
<td>Latent heat of evaporation</td>
<td>2.50 x 10⁵ J/kg</td>
</tr>
<tr>
<td>( L_S )</td>
<td>Latent heat of sublimation</td>
<td>2.8344 x 10⁵ J/kg</td>
</tr>
<tr>
<td>( Pr )</td>
<td>Laminar Prandtl number of air</td>
<td>0.72</td>
</tr>
<tr>
<td>( Pr_t )</td>
<td>Turbulent Prandtl number of air</td>
<td>0.9</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>Radiative surface emissivity of ice</td>
<td>0.5-0.8</td>
</tr>
<tr>
<td>( \mu_w )</td>
<td>Viscosity of water</td>
<td>1.795 x 10⁻³ Pa.s</td>
</tr>
<tr>
<td>( \rho_r )</td>
<td>Density of rime ice</td>
<td>880 kg/m³</td>
</tr>
<tr>
<td>( \rho_g )</td>
<td>Density of glaze ice</td>
<td>917 kg/m³</td>
</tr>
<tr>
<td>( \rho_w )</td>
<td>Density of water</td>
<td>999 kg/m³</td>
</tr>
<tr>
<td>( \sigma_e )</td>
<td>Stefan-Boltzmann constant</td>
<td>5.6704 x 10⁻⁷</td>
</tr>
<tr>
<td>( \sigma_w )</td>
<td>Surface tension of water</td>
<td>0.072 N/m</td>
</tr>
</tbody>
</table>

**Energy terms**

The energy terms appearing in the above equations need to be expressed in terms of the field variables. Although convective heat transfer \( (Q_c) \) and latent heat \( (Q_l) \) are the most prominent terms, all relevant energy terms are considered and used in the developed computer program. In the following, \( T_{sur} \) is the temperature at the ice (rime ice) or water surface (glaze ice).

- Convective heat transfer at the water surface \( (Q_c) \):
\[ Q_c = h_c (T_{\text{surf}} - T_a). \]  

(46)

- Cooling by incoming droplets \((Q_d)\):

\[ Q_d = \rho_0 V_\infty C_{pw} (T_{\text{surf}} - T_a). \]  

(47)

- Evaporative heat loss \((Q_e)\):

\[ Q_e = \chi_e e_o (T_{\text{surf}} - T_a), \]  

(48)

where \(\chi_e\) is the evaporation coefficient and \(e_o = 27.03\). Evaporation coefficient is expressed as \([12]\):

\[ \chi_e = \frac{0.622 h_c L_e}{C_p P_t L_e^{2/3}}, \]  

(49)

where \(P_t\) is the total pressure of the airflow.

- Sublimation heat loss \((Q_s)\):

\[ Q_s = \chi_s e_o (T_{\text{surf}} - T_a), \]  

(50)

Sublimation coefficient \(\chi_s\) is expressed as \([12]\):

\[ \chi_s = \frac{0.622 h_c L_s}{C_p P_t L_e^{2/3}}, \]  

(51)

- Heat loss due to radiation \((Q_r)\) \([13]\):

\[ Q_r = 4\varepsilon \sigma_s T_a^4 (T_{\text{surf}} - T_a), \]  

(52)

where \(\varepsilon\) is the surface emissivity and \(\sigma_s\) is the Stefan-Boltzmann constant.

- Aerodynamic heating term \((Q_a)\):

\[ Q_a = \frac{r h_c V_\infty^2}{2 C_p}, \]  

(53)

- Kinetic energy of incoming droplets \((Q_k)\):

\[ Q_k = \rho_0 V_\infty^2, \]  

(54)

- Energy brought in by runback water \((Q_{in})\):

\[ Q_{in} = m_{\text{in}} C_{pw} (T_f - T_{\text{surf}}), \]  

(55)

where \(m_{\text{in}}\) is the mass flow rate of the incoming runback water.

- Latent heat of solidification \((Q_l)\):

\[ Q_l = \rho_1 L_F \frac{\partial B}{\partial t}. \]  

(56)

With these definitions, it is possible to express equations \((37), (40), (42)\) and \((44)\) in terms of the surface temperature of the geometry \((T_a)\) and ambient temperature \((T_a)\) only.

**Rime ice temperature distribution**

Equation \((37)\) can be written as:

\[ T(y) = \frac{Q_{0r} - Q_{1r} T_s y + T_s}{k_i + Q_{1r} B}, \]  

(57)

where

\[ Q_{0r} = \rho_r L_F \frac{\partial B}{\partial t} + \rho_0 V_\infty \frac{V_\infty^2}{2} + r h_c \frac{V_\infty^2}{2 C_p} + \rho_0 V_\infty C_{pw} T_a + h_c T_a + 4\varepsilon \sigma_s T_a^4 + \chi_s e_o T_a + m_{\text{in}} C_{pw} T_f, \]  

(58)

\[ Q_{1r} = \rho_0 V_\infty C_{pw} + h_c + 4\varepsilon \sigma_s T_a^4 + \chi_s e_o + m_{\text{in}} C_{pw}. \]  

(59)

**Glaze ice temperature distribution and ice growth rate**

Equation \((40)\) can be written as:

\[ \theta(y) = \frac{Q_0 - Q_1 T_f h + T_f}{k_w + Q_1 h}, \]  

(60)

where

\[ Q_0 = \rho_r L_F \frac{V_\infty^2}{2} + r h_c \frac{V_\infty^2}{2 C_p} + \rho_0 V_\infty C_{pw} T_a + h_c T_a + 4\varepsilon \sigma_s T_a^4 + \chi_s e_o T_a + m_{\text{in}} C_{pw} T_f, \]  

(61)

\[ Q_1 = \rho_0 V_\infty C_{pw} + h_c + 4\varepsilon \sigma_s T_a^4 + \chi_s e_o + m_{\text{in}} C_{pw}. \]  

(62)

Equation \((42)\) can be written as:


\[ \rho_s L_F \frac{\partial B}{\partial t} = k_i \frac{T_f - T_s}{B} - k_w - \frac{Q_0 - Q_T f}{k_w + Q_i h}. \]  \hspace{1cm} (63)

Equation (44) can be written as:

\[ B_g = \frac{k_i (T_f - T_s)}{\rho_s L_F \left( \rho_d \beta V_{\infty} + \dot{m}_in - \dot{m}_{sub} \right) / \rho_r + (Q_0 - Q_T f)} \]  \hspace{1cm} (64)

**Freezing fractions and runback water**

Freezing fraction for a given control volume (or a panel in this case) is the ratio of the amount of water that solidifies to the amount of water that impinges on the control volume plus the water that enters the panel as runback water.

**Rime ice:**

\[ FF = \frac{\rho_r B}{(\rho_d \beta V_{\infty} + \dot{m}_in)}. \]  \hspace{1cm} (65)

**Glaze ice:**

\[ FF = \frac{\rho_r B + B - B_g}{(\rho_d \beta V_{\infty} + \dot{m}_in)}. \]  \hspace{1cm} (66)

Runback water mass flow rate:

\[ \dot{m}_{out} = (1 - FF) \left( \rho_d \beta V_{\infty} + \dot{m}_in \right) - \dot{m}_b. \]  \hspace{1cm} (67)

This becomes \( \dot{m}_in \) for the neighboring downstream panel. It is assumed that, all unfrozen water passes to the next downstream panel for the upper surface of the geometry. For the lower surface, it is assumed that all the unfrozen water is shed [14].

**Evaporating or sublimating mass**

Evaporating mass is given as:

\[ \dot{m}_e = \frac{Q_E}{L_E}. \]  \hspace{1cm} (68)

Likewise, sublimating mass is expressed as:

\[ \dot{m}_s = \frac{Q_s}{L_S}. \]  \hspace{1cm} (69)

**METHOD FOR PARALLEL PROCESSING**

Parallel processing technique is implemented using mpi (Message Passing Interface) with mpich libraries. The computer environment is composed of distributed PC's with Linux type of operating systems, forming a parallel PC cluster located at the Turkish Aerospace Industries. Each PC has equipped with Intel(R) Core(TM) 2 Duo processors and 8 Gigabyte RAM size memory. The cluster has 64 nodes with dual cores in its environment which is dedicated to aerodynamic CFD calculations only. Each node is interconnected with Gigabit switches in the parallel computing environment.

**RESULTS AND DISCUSSION**

The developed tool is tested against experimental and numerical ice shapes reported in the literature. Three test cases are selected such that the effects of geometry, ambient and icing conditions on ice formation and the degree at which the code is able to predict these are brought out. Geometric and ambient conditions, as well as icing data for these cases are presented in Table 2. The airfoil is NACA 0012 for all cases. Since experimental and numerical ice data for three-dimensional geometries are extremely scarce in the literature, airfoil data is used, which are widely available. Two-dimensional test cases are selected from the literature such that the resulting ice shapes are complex, i.e. displaying single or double horns and the effects of compressibility are prominent [9]. In the computations, the span of the wing is chosen to be ten times its chord in order to simulate the flow over an airfoil. Also, the multi-layer approach is used, where the total exposure time is divided into 4 layers in all subsequent computations.

**Table 2. Geometric characteristics and flow conditions for code validation.**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Case 31°</th>
<th>Case 34°</th>
<th>Case 39°</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha ), angle of attack (°)</td>
<td>4.</td>
<td>4.</td>
<td>0.</td>
</tr>
<tr>
<td>( c ), wing chord or duct length (m)</td>
<td>0.53</td>
<td>0.53</td>
<td>0.465</td>
</tr>
<tr>
<td>( V_{\infty} ), freestream velocity (m/s)</td>
<td>58.1</td>
<td>93.9</td>
<td>131.5</td>
</tr>
<tr>
<td>( p_{\infty} ), ambient pressure (Pa)</td>
<td>95610.</td>
<td>92060.</td>
<td>85000.</td>
</tr>
<tr>
<td>( T_{\infty} ), ambient temperature (°C)</td>
<td>-3.9</td>
<td>-16.6</td>
<td>-3.9</td>
</tr>
<tr>
<td>( \rho_{\infty} ), liquid water content (g/m³)</td>
<td>1.3</td>
<td>1.05</td>
<td>0.6</td>
</tr>
<tr>
<td>( t_{\text{exp}} ), exposure time (s)</td>
<td>480.</td>
<td>372.</td>
<td>180.</td>
</tr>
<tr>
<td>( d_p ), droplet diameter (μm)</td>
<td>20.</td>
<td>20.</td>
<td>20.</td>
</tr>
</tbody>
</table>

**RESULTS FOR DIFFERENT NUMBER OF PANEL WISE STRIPS**

Figure 4 shows the effect of using different spanwise panels on the ice shapes. Case 31 defined in Table 2 is selected for
these comparisons. In each computation, the number of chordwise panels is 149. The ice shapes are taken from $y = 2.65$ m from the wing centerline, i.e. at quarter span. As can be seen, all computations yield similar results with minor differences, the most significant outliers being 2 and 4 strips. Ice shapes for 8, 16 and 24 spanwise strips yield almost identical results suggesting that the optimum number of spanwise strips for such computations and geometries is probably around 8. These results are supported by the total ice mass data, which is presented in Figure 5. This data is obtained by integrating the ice mass over the span of the wing. It can be seen that the total mass of accumulated ice is predicted to be nearly constant for strip numbers higher than 8. Therefore all the results in the subsequent sections will be shown for 8 spanwise strips. It is possible however that more strips may be needed for more complex shapes like wings with sweep, twist, etc.

Figure 6 shows the effect of using parallel computing on the computational time. Again, case 31 defined in Table 2 is chosen for the comparisons with 149 chordwise and 8 spanwise panels. Results are shown for 1 (sequential computing), 2, 4 and 8 processors for 8 spanwise strips. As can be seen, the computational time decreases exponentially with increasing number of processors, which is an indication that almost the entire computational time is consumed by trajectory calculations.

**COMPARISON OF THE RESULTS WITH EXPERIMENTAL AND NUMERICAL DATA**

In the following computations, the span of the wing is chosen to be ten times its chord in order to simulate the flow over an airfoil, and the ice data is taken from a spanwise section close to the symmetry plane of the wing in order to simulate the airfoil flow conditions.
Figure 7 illustrates the results of the analysis for a high temperature case, where the ice formation is mostly glaze with runback water effects. As can be seen, the extent of the ice both on the upper and lower surface is well predicted in the current study compared with the experimental ice shape. Also the prominent bump and its position on the upper surface are reproduced by the computations, although underestimated in size. Moreover, the second bump just downstream of the prominent bump has also been well captured. The size and distribution of the ice formation on the lower surface is reproduced with accuracy in the current study. The maximum ice thickness is 1.1 cm. Meanwhile, the overall ice volume is slightly underestimated by the current computations. On the other hand, the numerical results obtained from fully two-dimensional computations by other investigators overpredict the extent of the iced region on the upper surface and the bumps on the upper surface are not fully reproduced. This suggests that runback water is overpredicted by other studies and the current study is slightly superior in that respect. However, it has to be kept in mind that all the numerical results that are used here for comparison date back to 1997 and the computational tools that are used for obtaining them almost certainly must have evolved over time and the current results would probably be in much better agreement with the experimental results now.

In Fig. 8, another glaze ice case is shown, this time with a lower temperature. The experimental ice shape displays two horns, with almost equal sizes. As can be seen, all the numerical computations including the current one slightly underestimate the iced region on the upper surface, while there is the current study provides a better agreement on the lower surface. The horns are predicted by the current study, also by the results of DRA and ONERA. Both the locations and the sizes of the horns are almost perfectly predicted in the current study. As a result, the total ice volume seems to be also well predicted. For this case the maximum ice thickness is roughly 3 cm occurring at the upper horn.

Figure 9 illustrates a case where the freestream velocity is high enough to render aerodynamic heating effects important. For this case, there are no experimental results available; however the agreement of the results of the current study with those of other computations, especially with those of NASA is fair. For this case, although the extent of the airfoil wetted by the incoming droplets is much larger than the region that is iced, aerodynamic heating raises the surface temperature of a large portion of the leading edge above the freezing temperature, preventing ice formation there. It is also noteworthy that the very thin layer of ice on the lower surface is not observed in any other computational result that is shown except the current study. The maximum ice thickness occurs at the bump at the leading edge and is about 1 cm.

Figure 7. Comparison of experimental and numerical results with the present study (Case 31).
Figure 8. Comparison of experimental and numerical results with the present study (Case 34).

Figure 9. Comparison of experimental and numerical results with the present study (Case 39).
RESULTS WITH DROPLET BREAKUP AND SPLASH EFFECTS

In Fig. 10, the results of an analysis with droplet breakup and splash effects is illustrated. For this case the droplet size is 160 microns, the liquid water content is 1.5 g/m$^3$, the freestream velocity is 52 m/s, the temperature is −19.5°C and the exposure time is 300 seconds. The airfoil is again NACA 0012. It can be seen that inclusion of droplet splash and breakup features in the developed tool does not significantly improve the agreement of the obtained results with the experimental ones for this parameter combination. The mass of ice predicted by the computations with droplet breakup and splash effects is slightly less than the ice mass obtained by not including them. Non-spherical particles are also considered according to equations (7, 8, 9, 10).

Another improvement would be to treat the water flow on the surface as an additional problem with a higher order approach like the one studied by Myers, et al [13]. This would allow runback water effects to be simulated more faithfully, which are important for glaze ice conditions yielding irregular ice shapes. Nevertheless, both this and the previously suggested improvements would increase the computational immensely, rendering the current approach a good compromise between accuracy and speed.

The developed tool could be easily modified and used for determining the capacities of a heated anti/de-icing leading edge device, and also for certification and design purposes.

REFERENCES


CONTACT INFORMATION

Prof. Dr. Serkan ÖZGEN
Dept. Aerospace Engineering
Middle East Technical University
06800 Ankara, Turkey
Phone: +90-312-210 4273
sozgen@ae.metu.edu.tr