Aircraft Icing
2-D Ice accretion prediction

Prof. Dr. Serkan ÖZGEN
Dept. Aerospace Engineering, METU
October 2019
Outline

• Flow field solution
• Droplet trajectories and collection efficiencies
  – Governing equations
  – Droplet impact
  – Parameters effecting collection efficiency
• Boundary-layer calculations
• Ice growth model; Extended Messinger Model
• Modified geometry
• Runback water
• Single and multi-step approach
• Validation results
Flow Field Solution

The flow field solution is needed to determine:

• The *velocity and pressure distribution* on the surface of the geometry for subsequent boundary-layer calculations and determination of aerodynamic performance.
• *Off-body velocities* to be used for droplet trajectory calculations.
Flow Field Solution

Alternatives:

• Panel method coupled with a boundary-layer solution: less accurate but fast and cheap.
• Navier-Stokes: more accurate but slow and expensive.

No significant improvement has been observed by using a Navier-Stokes solver instead of a panel code ⇒ use a Hess-Smith panel code.
Flow Field Solution

• In the panel method, the geometry is divided into quadrilateral panels each associated with a source singularity element together with a vortex singularity that is constant for all panels.
• The strengths of the singularities are constants and are unknowns of the problem. The developed computer program uses $N$ quadrilateral panels to solve $N+1$ singularity strengths using the flow tangency boundary condition at the collocation points of the panels and an additional equation is introduced for the Kutta condition. 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 including the boundaries of the geometry. The velocity components at a given point are the $x$-, $y$- derivatives of the velocity potential constructed at that point.
Flow field solution

NACA 23012, $\alpha=-0.27^\circ$
Flow field solution

NACA 23012, $\alpha=-1.05^\circ$
For droplet trajectories, **Lagrangian approach** is used with the following assumptions:

- Droplets are assumed to be **spherical**.
- The flow field is not affected by the presence of the droplets.
- **Gravity and aerodynamic drag** are the only forces acting on the droplets.

These assumptions are valid for droplet sizes with $d_p \leq 500 \, \mu m$. 
The governing equations for droplet trajectories are:

\[ m\ddot{x}_p = -D \cos \gamma, \]
\[ m\ddot{y}_p = -D \sin \gamma + mg, \]
\[ \gamma = \tan^{-1} \frac{\dot{y}_p - V_y}{\dot{x}_p - V_x}, \]
\[ D = \frac{1}{2} \rho V_{rel}^2 C_D A_p, \]
\[ V_{rel} = \sqrt{(\dot{x}_p - V_x)^2 + (\dot{y}_p - V_y)^2} \]
Definitions of parameters:

\( C_D \): droplet drag coefficient,

\( V_x, V_y \): components of the flow field velocity at the droplet location,

\( \dot{x}_p, \dot{y}_p \): components of the droplet velocity,

\( \ddot{x}_p, \ddot{y}_p \): components of the droplet acceleration,

\( \rho \): atmospheric density,

\( A_p = \frac{\pi}{4} d_p^2 \): droplet cross-sectional area.
Droplet trajectories and collection efficiencies

The drag coefficients of the droplets are calculated from:

\[ C_D \frac{Re}{24} = 1 + 0.197 Re^{0.63} + 2.6 \times 10^{-4} Re^{1.38}, \quad Re \leq 3500, \]

\[ C_D \frac{Re}{24} = (1.699 \times 10^{-5}) Re^{1.52}, \quad Re > 3500. \]

\[ Re = \rho V d_p / \mu \]
Droplet trajectories and collection efficiencies

Droplet impact:

\[ x_{i+1}, y_{i+1} \]

\[ x_{dp}, y_{dp} \]

\[ x_d, y_d \]

\[ x_i, y_i \]
Droplet trajectories and collection efficiencies

\[ l_1 = x_d - x_{dp} \]
\[ n_1 = y_d - y_{dp} \] \( \{ \) vector along particle trajectory

\[ l_2 = x_{i+1} - x_i \]
\[ n_2 = y_{i+1} - y_i \] \( \{ \) vector along panel

\[ l_1 t_1 + l_2 t_2 = x_i - x_{dp} \]
\[ n_1 t_1 + n_2 t_2 = y_i - y_{dp} \] \( \{ \) system of two equations with two unknowns

\[
\begin{bmatrix}
  l_1 & l_2 \\
  n_1 & n_2 
\end{bmatrix}
\begin{bmatrix}
  t_1 \\
  t_2 
\end{bmatrix}
= \begin{bmatrix}
  x_i - x_{dp} \\
  y_i - y_{dp} 
\end{bmatrix}
\]
Droplet trajectories and collection efficiencies

\[ t_1 = \frac{n_2(x_i-x_{dp})-l_2(y_i-y_{dp})}{(l_1n_2-l_2n_1)} \]; along the trajectory,

\[ t_2 = \frac{l_1(y_i-y_{dp})-n_1(x_i-x_{dp})}{(l_1n_2-l_2n_1)} \]; along the panel.

Additional constraints to prevent fake impact locations:

\[ 0 \leq t_1 \leq 1, \]
\[ 0 \leq t_2 \leq 1. \]

Exact impact location:

\[ x_{imp} = x_i + l_2t_2, \]
\[ y_{imp} = y_i + n_2t_2, \]
Droplet trajectories and collection efficiencies

Collection efficiency:

\[ \beta = \frac{dy_0}{ds} = \frac{\Delta y_0}{\Delta s}, \]
Particle trajectories
Droplet collection efficiency (effect of airframe size)
Droplet collection efficiency
(effect of droplet size)

![Graph showing droplet collection efficiency with different droplet sizes](image)
Droplet collection efficiency (effect of freestream velocity)
Droplet collection efficiency (effect of angle of attack)
Factors increasing collection efficiency:

- Smaller airframe size; geometry creates a smaller obstacle for the incoming droplets and the deviation of the particles away from the body is not sufficient for them to avoid it.
- Greater droplet size; increases droplet inertia and droplets follow ballistic trajectories $\Rightarrow$ collection efficiencies increase, impingement zone gets wider.
- Higher airspeed; increases droplet inertia.
Boundary-layer calculations

- Integral boundary layer method is used for the calculation of the convective heat transfer coefficients.
- With this method, the details of the laminar and turbulent boundary layers are calculated fairly accurately.
Boundary-layer calculations

- Transition prediction is based on the roughness Reynolds number:

\[ Re_k = \frac{\rho U_k k_s}{\mu} \]

\( k_s \): roughness height,

\( U_k \): local flow velocity at the roughness height:

\[ \frac{U_k}{U_e} = 2 \frac{k_s}{\delta} - 2 \left( \frac{k_s}{\delta} \right)^3 + \left( \frac{k_s}{\delta} \right)^4 + \frac{1}{6} \frac{\delta^2}{\nu} \frac{dU_e}{ds} \frac{k_s}{\delta} \left( 1 - \frac{k_s}{\delta} \right)^3 \]

\( U_e \): flow velocity outside the boundary-layer at the roughness location.

\( s \): streamwise distance along the airfoil surface starting at the stagnation point.
Boundary-layer calculations

- Roughness height (NASA model):

\[
\frac{k_S}{\delta} = 0.00117 k_{V_\infty} k_{LWC} k_{T_\infty} k_{d_p}
\]

\[
k_{V_\infty} = 0.4286 + 0.0044139 V_\infty,
\]

\[
k_{LWC} = 0.5714 + 0.2457 \times (1000 LWC) + 1.2571 \times (1000 LWC)^2
\]

\[
k_{T_\infty} = 46.8384 \frac{T_\infty}{1000} - 11.2037
\]

\[
k_{d_p} = 1 \quad \text{if} \quad d_p \leq 20 \, \mu m,
\]

\[
k_{d_p} = 0.033 \quad \text{if} \quad 20 \mu \leq d_p \leq 50 \mu m,
\]

\[
k_{d_p} = \frac{5}{3} - \frac{1}{30} d_p \times 10^6 \quad \text{if} \quad d_p > 50 \mu m.
\]
Boundary-layer calculations

• Laminar boundary-layer thickness:
  \[ \delta = \frac{315}{37} \theta_l \]

• Laminar momentum thickness (Thwaites formulation):
  \[ \frac{\theta_l^2}{\nu} = \frac{0.45}{U_e^6(s)} \int_0^s U_e^5(s) ds \]

• For laminar flow, \( Re_k \leq 600 \), method of Smith & Spalding for the calculation of the convective heat transfer coefficient:
  \[ h_c = \frac{0.296k U_e^{1.435}}{\sqrt{\nu \int_0^s U_e^{1.87}(s) ds}} \]

\( k \): thermal conductivity of air.
Boundary-layer calculations

• For turbulent flow, $Re_k \geq 600$, method of Kays & Crawford for the calculation of the convective heat transfer coefficient:
  
  $h_c = St \rho U_e C_p$

• Stanton number:

  $$St = \frac{C_f/2}{Pr_t + \sqrt{(C_f/2)/St_k}}$$

  $Pr_t = 0.9$: turbulent Prandtl number.

• Roughness Stanton number:

  $$St_k = 1.92 Re_k^{-0.45} Pr^{-0.8}$$

  $Pr = 0.72$: laminar Prandtl number.
Boundary-layer calculations

- Skin friction coefficient is calculated from Makkonen relation:

\[
C_f = \frac{0.1681}{2} \left[ \ln \left( \frac{864 \theta_t}{k_S} + 2.568 \right) \right]^2
\]

- Turbulent momentum thickness:

\[
\theta_t = \frac{0.036 \nu^{0.2}}{U_e^{3.29}} \left( \int_0^s U_e^{3.86}(s) ds \right)^{0.8} + \theta_{tr}
\]
Boundary-layer calculations

- Boundary-layer calculations start at the leading edge and proceed downstream using the **marching technique** for the upper and lower surfaces of the airfoil.
- **Transition** is fixed at the streamwise location, where $Re_k = 600$ according to Von Doenhoff criterion.
Boundary-layer calculations
Boundary-layer calculations
Extended Messinger Model

See lecture notes.
Modified geometry

• After the ice growth rates given by equation (10) for rime ice and equation (52) for glaze ice are solved, an ice thickness results for a given control volume (panel).

• In order to obtain the modified geometry with ice formation, it is assumed that the ice grows perpendicularly to the surface.
\[ \Psi_i = \text{panel inclination angle} \]

\[ B_i : \text{ice thickness calculated at node } i, \]

\[ \bar{x}_i = x_i + B_i \sin \left( \frac{\Psi_{i-1} + \Psi_i}{2} \right) \]

\[ \bar{y}_i = y_i + B_i \cos \left( \frac{\Psi_{i-1} + \Psi_i}{2} \right) \]
Runback water

• Under glaze ice conditions, freezing fraction is less than one ($FF < 1$), meaning that only a fraction of the impinging water freezes.

• Remaining water may either flow downstream as runback water or may be shed due to high shear and gravity.

• According to Fortin et al:
  – Upper surface: All the unfrozen water passes to the neighboring downstream panel as runback water,
  – Lower surface: All unfrozen water is shed (due to gravity).

• Including the runback water introduces some modifications to the Messinger Model formulation (see lecture notes).
Single and multi-step approach

• Computing the ice shapes in a single cycle or computational step for the entire duration of the exposure may yield **unrealistic** and **incorrect** ice shapes.

• This is particularly true for:
  – Glaze ice conditions where **runback water** effect is prominent,
  – Icing exposures where the **icing conditions are varying**, i.e. climbing flight, descending flight.
  – Long exposures.
In the current solution method, an unsteady problem is solved by a quasi-steady approach. Therefore a continuous flow of water, impingement, ice and water accumulation has to be analyzed in as small as possible intervals to represent these faithfully, but this increases computational time. With a multi-step approach, the effect of ice formation on the flow field, droplet trajectories, ice formation and runback water are taken into account to some degree. At each computational step, flow field solution, droplet trajectories, collection efficiencies, energy terms and the ice formation routines are repeated. Experience shows that $\Delta t = 60 - 120$ seconds is a good compromise between computational time and accuracy.
## Validation results

<table>
<thead>
<tr>
<th>Variables</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of attack</td>
<td>4°</td>
</tr>
<tr>
<td>Chord length</td>
<td>0.53 m</td>
</tr>
<tr>
<td>Freestream velocity</td>
<td>58.1 m/s</td>
</tr>
<tr>
<td>Ambient pressure</td>
<td>95610 Pa</td>
</tr>
<tr>
<td>Liquid water content</td>
<td>1.3 g/m³</td>
</tr>
<tr>
<td>Exposure time</td>
<td>480 s</td>
</tr>
<tr>
<td>Droplet median diameter</td>
<td>20 μm</td>
</tr>
</tbody>
</table>
Validation results \( (T_a=-27.8^\circ C) \)
Validation results ($T_a = -19.8^\circ C$)
Validation results ($T_a=-13.9^\circ C$)
Validation results \((T_a=-6.7^\circ C)\)
Multi-step approach ($T_a=-27.8^\circ C$)
Multi-step approach ($T_a=-19.8^\circ \text{C}$)
Multi-step approach ($T_a = -13.9^\circ C$)
Multi-step approach \((T_a = -6.7^\circ C)\)
Airfoil with flap

<table>
<thead>
<tr>
<th>Variables</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of attack</td>
<td>4°</td>
</tr>
<tr>
<td>Chord length</td>
<td>1. element: 0.53 m</td>
</tr>
<tr>
<td></td>
<td>2. Element: 0.1 m</td>
</tr>
<tr>
<td>Freestream velocity</td>
<td>60 m/s</td>
</tr>
<tr>
<td>Ambient pressure</td>
<td>101300 Pa</td>
</tr>
<tr>
<td>Liquid water content</td>
<td>1.0 g/m³</td>
</tr>
<tr>
<td>Exposure time</td>
<td>360 s</td>
</tr>
<tr>
<td>Droplet median diameter</td>
<td>20 μm</td>
</tr>
</tbody>
</table>
Particle trajectories (airfoil with flap)
Ice shapes in two element airfoil

(δ_f=10°)
Ice shapes in two element airfoil
(main airfoil, $\delta_f=10^\circ$)
Ice shapes in two element airfoil
(flap, $\delta_f=10^\circ$)