ABSTRACT

This article proposes the use of the Box-Behnken design of experiment (DoE) methodology to study an aircraft anti-icing system. The anti-icing system consisted of a piccolo tube with round apertures for producing air jets inside a wing. Mass flow, jet to wall distance, and jet impact angle were varied, starting from an initial design, in order to maximize heat transfer effectiveness. A conjugate heat transfer procedure from commercial CFD software was used to solve for cold air external flow, compressible internal flow, and thermal conduction in the airfoil skin. The DoE methodology was validated using a single impinging jet. A quadratic model of the heat transfer effectiveness of the anti-icing system was then built using the methodology and the maximal value was sought.

Keywords: anti-icing; conjugate heat transfer; design of experiment.
1. INTRODUCTION

In order to reduce the risk of accidents [1], Canadian civil aviation safety authorities allow flight under icing conditions only if critical portions of the aircraft are protected by either anti-icing or de-icing systems, as set forth in regulation 602.11 [2]. Anti-icing systems are commonly used for the continuous protection of wing leading edges, with heat provided usually by hot air circulating inside the wing. This is the type of protective system studied in this paper.

The efficiency of thermal anti-icing systems depends on several parameters, in particular on the amount of water impinging on the wing (external flow) and on heat loss by convective flux. It also depends on heat generation by the anti-icing system. Design of experiment (DoE) could be used advantageously to conduct parametric studies of efficiency. Based on the few published detailed parametric studies of hot-air-based anti-icing systems, the most influential parameters can be identified.

Fregeau et al. [3] established a correlation for the Nusselt number based on jet nozzle-to-surface spacing, jet mass flow rate and jet nozzle-to-nozzle distribution. The average Nusselt number correlation was found to depend strongly on jet nozzle-to-surface spacing. The maximal Nusselt number occurred at the jet stagnation point and depended mainly on mass flow rate, while the jet nozzle-to-nozzle spacing effect was negligible.

Saeed et al. [4] used a commercial computational fluid dynamics (CFD) flow solver to model the different hot-jet arrangements of an anti-icing piccolo system with a curved impingement surface. The study revealed that a single jet array and a jet array staggered at 20 degrees yielded better surface heat transfer than the same jet array staggered at 10 degrees.

Brown et al. [5] used 2D experiments to devise a correlation for heat transfer in an aircraft nacelle anti-icing system. The average Nusselt number was correlated with jet nozzle diameter and the impinging jet mass flow rate per unit area.

Papadakis et al. [6,7] studied the effect of piccolo tube design, diffuser geometry, jet temperature, and jet mass flow rate on system performance. They concluded that wing skin temperature depends on the jet temperature and mass flow rate and on anti-icing system geometric parameters such as jet inclination.

Pellissier et al. [8] presented a methodology for optimizing a hot-air anti-icing system using a single objective function based on three geometric parameters, namely piccolo tube position, tilt angle and spacing between jets. The internal flow local convection coefficient was deduced from the correlation of Goldstein et al. [9].

The goal of the present study is to build a parametric model of heat transfer effectiveness as an objective function based on three variables, namely jet inclination, jet nozzle-to-surface spacing, and jet mass flow rate, from a database populated using CFD and using the Box-Behnken design-of-experiment method. To the best of our knowledge, the Box-Behnken DoE method has not been applied previously to the study of anti-ice protective systems for aircraft wings. The resulting model should facilitate the design of hot-air anti-icing systems.

We begin by describing the mathematical model and the numerical method used to solve the conjugate heat transfer (CHT) problems based on DoE methodology. The use of a single round-nozzle impinging jet for methodology validation and the hot-air-based anti-icing system test cases are then presented. We then validate the CFD results against published results. Finally, optimization results obtained using parametric models are discussed.

2. MATHEMATICAL MODEL AND NUMERICAL METHOD

The commercial flow solver ANSYS CFX 12.1 was used for CFD computation. A brief presentation of the specific mathematical model and the numerical method used for the calculation is presented below.
2.1. Mathematical Model

Airflow was deemed compressible and turbulent. Air properties were presumed to follow the ideal gas law:

$$p = \rho RT$$  \hspace{1cm} (1)

The compressible RANS equations for eddy-viscosity-based turbulence models are expressed as follows:

$$\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U U) = -\nabla p + \nabla \cdot \left( \left( \mu + \mu_t \right) \nabla U + (\nabla U)^T \right),$$  \hspace{1cm} (2)

$$\frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot \left( \lambda \nabla T + \frac{\mu_t}{Pr_t} \nabla h \right) + \nabla \cdot \left( U \cdot (\tau - \rho u \times u) \right) + S_E.$$  \hspace{1cm} (4)

The viscous work $\nabla \cdot (U \cdot (\tau - \rho u \times u))$ was taken into account in the energy equation Eq. (4).

For the CHT problem, Eq. (5) was solved in the solid domain with the subscript “s” referring to solid:

$$\frac{\partial (\rho_s h_s)}{\partial t} = \nabla \cdot (\lambda \nabla T) + S_E.$$  \hspace{1cm} (5)

The CFD solutions for external flow, internal flow and solid conduction were coupled using the CHT procedure of ANSYS CFX 12.1. Air temperature was presumed equal to solid temperature at air solid interfaces and heat fluxes were presumed conserved.

The $k – \omega$ SST turbulence model equations used for the turbulent viscosity calculation are detailed in Menter [10]. The $k – \omega$ SST model of ANSYS CFX 12.1 allows a smooth transition between logarithmic interpolation functions and modeling the laminar boundary layer if the first mesh node meets the condition $y^+ < 1$.

2.2. Numerical Method

Equations (2–4) and the turbulence model equations were solved using a collocated finite volume method. The ideal gas law (Eq. 1) relates density to pressure and temperature. From Eqs. (2) and (3), a system of linearized equations was built and solved. Using the velocity field, the energy Eq. (4) and the turbulence model were then solved sequentially. The linearized equations were solved using an algebraic multi-grid method. The equation system was solved iteratively and was assumed to have converged sufficiently when the normalized residual of each equation fell below $10^{-6}$.

The ANSYS-CFX 12.1 blend factor advection scheme was used for numerical stability. This scheme is a mix between a first order upwind difference scheme, for a value of 0, and a second order (in form) central difference scheme, for a value of 1. At a value of 0.75, second order is nearly achieved without introducing local artificial oscillation.

2.3. The Design of Experiment Method

The Box-Behnken design reduces the design points required in order to consider quadratic effects in the parametric model [11]. It is often used with experiments to identify correlations between a few variables and to determine optimal values in chemical processes [12]. Application of the Box-Behnken design to the optimization of analytical methods remains rare [13]. In response surface methodology [11], the most influential parameters must be selected first. In the present study, the selection was based on parameters used in published studies. First-order parametric studies are not necessary, assuming that the initial design is not too far-removed from the optimal design.
DoE determines a parametric model that connects an objective function to three design variables. From the dimensionless variables \( v'_1, v'_2 \) and \( v'_3 \), a second-order parametric model was built:

\[
\beta_0 C + \beta_1 v'_1 + \beta_2 v'_2 + \beta_3 v'_3
\]

\[
f_{obj} (v'_1, v'_2, v'_3) = \beta_4 v'_1 v'_2 + \beta_5 v'_1 v'_3 + \beta_6 v'_2 v'_3 + \beta_7 v'_1^2 + \beta_8 v'_2^2 + \beta_9 v'_3^2
\]

The Box-Behnken design requires at least three design parameters [13]. Each dimensionless variable has a high value (+1), a low value (−1), and an intermediate value (0).

The design matrix is summarized in Table 1. It lists the numerical experiments required for each design variable combination. Each line corresponds to one numerical simulation. The first column is the coefficient value \( C \) in front of the zero order parameter \( \beta_0 \). The next columns are the dimensionless values of the three design variables, followed by the products of their combinations, which represent the interactions between the paired design parameters in the resulting parametric model. The final three columns provide a second-order polynomial model in order to consider the curvature of the results. The last three rows are evaluated at a central point. The objective function values are thus the same for numerical simulation.

<table>
<thead>
<tr>
<th>Nº</th>
<th>( C )</th>
<th>( v'_1 )</th>
<th>( v'_2 )</th>
<th>( v'_3 )</th>
<th>( v'_1 v'_2 )</th>
<th>( v'_1 v'_3 )</th>
<th>( v'_2 v'_3 )</th>
<th>( v'_1^2 )</th>
<th>( v'_2^2 )</th>
<th>( v'_3^2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>−1</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>1</td>
<td>−1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>−1</td>
<td>1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>−1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0</td>
<td>−1</td>
<td>−1</td>
<td>0</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>−1</td>
<td>0</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0</td>
<td>−1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>−1</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 1. Box-Behnken design matrix for three parameters.

3. TEST CASES

The first geometry studied was a single round nozzle producing an air jet impinging on a wall. Empirical correlations between average Nusselt number, \( Nu_{avg} \), Reynolds number, \( Re \), and jet nozzle-to-surface spacing for this geometry have been published [14]. The parameter combinations that will maximize the objective function \( Nu_{avg} \) can therefore be determined analytically. The DoE study based on numerical simulations was used to verify the proposed procedure.
The second geometry is an anti-icing system representative of a swept wing used on modern commercial transport aircraft and provides a basis of comparison between our numerical results and those computed by Wright [15].

3.1. Round Impinging Jet

The single-round impinging jet has been studied both experimentally and numerically [14]. Jet confinement does not significantly affect the heat transfer coefficients for confinement wall spacing greater than one jet diameter [16]. For the hot-air anti-icing system considered in Section 3.2, the spacing (in the normal direction) between the piccolo tube and the airfoil wall is from five to seven jet diameters. Jet confinement is thus negligible for the anti-icing system simulation. The unconfined round jet proposed by Vieser et al. [17] and presented in Fig. 1 was used for validation.

For a jet diameter $D$, the computational domain consists of a $13D \times 13D$ square. A fully developed pipe-flow turbulent velocity profile with constant temperature is imposed at the inlet. The air follows a pipe with an adiabatic wall upon entering the computational domain. The jet impinges on a plate at constant temperature. The opening boundary condition allows the air to exit or to enter by entrainment through the top and the right side of the computational domain.

![Fig. 1. Schematic representation of the computational domain for the round impinging jet.](image)

The design parameters used to build the parametric model were the jet Reynolds number $Re$, the ratio $H/D$, and the jet Mach number $M$. 

\[
Re = \frac{\rho U_{avg} D}{\mu} .
\]

The computational domain was made discreet by hexahedral meshes of 165 nodes in the radial direction $\times$ 150 nodes in the normal direction for $H/D = 2$, 165 $\times$ 250 nodes for $H/D = 4$ and 165 $\times$ 350 nodes for $H/D = 6$. A finer mesh of 329 $\times$ 499 nodes was also used for $H/D = 4$. The element sizes were made smaller near walls, such that $y+$ values were below 1.

3.2. Anti-Icing System

3.2.1. Anti-Icing System Geometry

For the numerical simulations, only a portion of the experimental anti-icing system is represented. The anti-icing system comprised three rows of jets. Using proper periodic conditions, each row was reduced to a single jet inlet in the computational domain. The wing geometry was based on a NACA23014 airfoil of
chord length $c = 1.52$ m. Figures 2 and 3 show 2D cuts in the $x$-$y$ and $y$-$z$ planes of the chosen geometry. Figure 3 shows that the entire anti-icing system and wing were swept at an angle of 8° relative to the $z$ axis. At the top and bottom, the two exits were narrow channels 0.001 m in width. The piccolo tube diameter was 0.038 m. This tube acted essentially as a plenum for the high-pressure hot filtered air. The internal flow computational domain excluded the piccolo tube internal area. The jet nozzle diameter was 0.00132 m.

Two computational methods were used. In one case, the internal domain was coupled to the solid domain and the external domain to obtain a CHT solution for the CFD validation. In the other case, only the internal domain was modeled and a constant temperature of 320 K was imposed at the leading edge wall. This second case was used to build the parametric model. For both cases, the piccolo tube wall temperature was 449.817 K and an adiabatic wall condition was maintained on the inner-liner wall. A pressure of $-10$ kPa was maintained at the hot-air outlets.

The design parameters considered in this paper are shown in Fig. 4. The reference point is at the intersection between the leading edge line and the horizontal line that passes through the center of the piccolo tube. $H$ is the horizontal distance in the $x$-$y$ plane between the piccolo tube wall and the reference point. Each centerline of the other two jets intersects the horizontal axis at angle $\alpha$. The mass flow rate, $m$, is considered the same at each jet aperture.

For the CHT solution, the internal flow was solved for $H = 0.0091$ m, $\alpha = 45^\circ$ and $m = 0.327394$ g/s. The internal flow was subsonic and compressible with $Re = 10,300$ ($M = 0.6$) at the inlets, the ratio of viscosities $\mu_{Turb}/\mu_{Lam} = 10^{-5}$ and the turbulence intensity $I = 1\%$.
3.2.2. Wing Geometry

For CHT calculations, the external flow represented a dry airflow cooling a wing. Figure 5 shows the computational domain, which includes three to five times the chord length from the airfoil. An air velocity of $59.2 \text{ ms}^{-1}$ (115 knots) at an angle of attack of $3^\circ$ was established uniformly in the far field with a uniform temperature of 268.2 K. Periodic boundary conditions were established on both sides of the external domain, spaced 0.132 m apart. The ANSYS CFX 12.1 opening boundary condition was applied to the top and bottom surfaces.

3.2.3. Computational Domain Discretization

A mixed tetrahedral-prism mesh was used to obtain a discreet internal domain. The surface mesh used is shown in Fig. 6. Prismatic element layers were used near all walls and tetrahedral elements were used elsewhere. The prism layer thickness was set to $4 \times 10^{-4}$ m with 20 prismatic layers and a growth ratio of 1.1 to ensure $y^+ \leq 2$. The distance from the first node to the wall was $7 \times 10^{-6}$ m. Density cone mesh refinement regions included the jet core from the aperture to the wall in the leading edge area, as shown in detail in Fig. 7.
Figure 8 shows a 3D view of the hexahedral elements used to mesh the external domain, along with a close-up on the internal space containing the piccolo tube only. The hexahedral layer closest to the airfoil wall was located $4 \times 10^{-6}$ m from the surface. A growth ratio of 1.1 was applied.
4. VALIDATION

This section is devoted to the validation of the CFD results obtained using the ANSYS CFX 12.1 code. The impinging jet results were validated against previously published numerical and experimental results obtained for round nozzles. The anti-icing system results were validated by comparison with previously published numerical results [15].

4.1. Impinging Jet

4.1.1. Grid Refinement Study

The effect of mesh density on the wall Nusselt number was studied for \( H/D = 4, \ Re = 23,000, \) and \( M = 0.01, \) with a medium mesh of 165 \( \times 250 \) nodes and a finer mesh of 329 \( \times 499 \) nodes. The local Nusselt number, \( Nu, \) was calculated as follows:

\[
Nu = \frac{q''_W D}{(T_W - T_{0, jet}) k_f}.
\]

where \( T_W \) and \( q''_W \) are respectively wall temperature and wall heat flux, \( T_{0, jet} \) is the total temperature at the jet inlet, \( k_f \) is the air conductivity, and \( D \) is the jet inlet diameter. As shown in Fig. 9(a), \( Nu \) was slightly higher in the impingement area for the medium mesh, \( 0 < r/D < 1 \), and the results agree quite well overall. For \( 0 < r/D < 6 \), the value for \( Nu_{avg} \) was computed using the following function:

\[
Nu_{avg} = \frac{1}{R} \int_0^R \frac{q''_W (r) D}{(T_W - T_{jet}) k_f}.
\]

The medium mesh value was 74.70 and the fine mesh value was 73.64, the difference being less than 1.5 %. The medium mesh was considered sufficient for computation of the average values needed for the objective function. A comparison of the \( y^+ \) values is shown in Fig. 9(b). As expected, reducing the element size by a factor of two reduced the \( y^+ \) values by a factor of two.

4.1.2. Comparison of the Results

A comparison of the \( Nu \) distribution obtained at \( Re = 23,000 \) and \( H/D = 2 \) using the ANSYS CFX 12.1 \( k - \omega \) SST model as well as various other numerical and experimental methods is shown in Fig. 10.
The results from Vieser et al. [17] were obtained using the $k-\omega$ SST turbulence model provided in CFX 5. Average $y^+$ values around 2, in addition to possible differences between versions may explain their higher Nusselt numbers. The data of Colucci and Viskanta [18], Baughn et al. [19], Lee and Lee [20], and Yann [21] were obtained from experiments. These results show a significant scatter of about 25% in the impingement area. The numerical results are within the margin of uncertainty of the experimental data.

As mentioned earlier, a $N_u_{avg}$ of 73.64 was obtained with $r/D =$ in the range of 0 to 6 is 73.64. Several published correlations may be used for validation. The correlation proposed by Martin [22] is valid for $Re$ from 2000 to 400,000, $H/D$ from 2 to 12, and $r/D$ from 2.5 to 7.5:

$$N_u_{avg} = Pr^{0.42} \frac{D}{r}^{1-\frac{1.1}{H} 2Re^{1/2} (1 + 0.005Re^{0.55})^{1/2}}.$$  \hfill (10)

Lee and Lee [23] proposed a correlation for $N_u_{avg}$ over the $r/D$ range of 0 to 4, valid for $Re$ from 5000 to 30,000 and $H/D$ from 2 to 10:

$$N_u_{avg} = 0.083Re^{0.708} (\frac{H}{D})^{-0.144}.$$  \hfill (11)

Finally, Tawfek [14] also suggest a correlation valid for $Re$ from 3400 to 41,000, $H/D$ ranging from 6 to 58, and $r/D$ from 2 to 30:

$$N_u_{avg} = 0.453Pr^{1/3}Re^{0.691} (\frac{H}{D})^{-0.22} (\frac{r}{D})^{-0.38}.$$  \hfill (12)

<table>
<thead>
<tr>
<th></th>
<th>CFX</th>
<th>Eq. (10)</th>
<th>Eq. (11)</th>
<th>Eq. (12)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$H/D = 2$</td>
<td>92.4</td>
<td>79.4</td>
<td>92.0</td>
<td>-</td>
</tr>
<tr>
<td>$H/D = 6$</td>
<td>116</td>
<td>71.5</td>
<td>78.5</td>
<td>166</td>
</tr>
</tbody>
</table>

Table 2. Average Nusselt numbers obtained using ANSYS CFX 12.1 and correlations.

Table 2 compares the Nusselt numbers calculated using these correlations to the numerical simulation results for $H/D = 2$ and 6, with $r/D$ ranging from 0 and 4 and $Re = 23,000$. For $H/D = 2$, the CFX value is higher than the value obtained using Eq. (10), but almost identical to the value obtained using Eq. (11).
At higher $H/D$, two correlations predict a decrease in average value, while CFX predicts an increase and Eq. (12) a much greater increase. The large variation between the correlations appears to be caused by differences in nozzle type [14]. The predictions of $Nu_{avg}$ by CFX are within the ranges of uncertainty associated with the correlations.

4.2. Anti-Icing System

The $Nu$ values are plotted along the curvilinear axis defined in Fig. 11. The curve corresponds to the intersection of the wing leading edge and the plane passing through the centerline of inclined jets 2 and 3 and perpendicular to the piccolo tube surface. The black arrow extends from the origin of the curvilinear distance $S$ on the upper side of the airfoil.

Fig. 11. Curve location on the inner wall of the leading edge for 2D analysis.

4.2.1. Grid Convergence Index GCI

A grid refinement study with three meshes was conducted in order to evaluate the GCI [24]. The coarser mesh had $3 \times 10^5$ nodes, the medium mesh had $1.5 \times 10^6$ nodes, and the finer mesh had $8 \times 10^6$ nodes. Prism layer thickness was $1 \times 10^{-4}$ m for the finer mesh and $4 \times 10^{-4}$ m for the medium and coarse meshes. For the fine and medium meshes, the local Nusselt numerical profiles are almost superimposable as shown in Fig. 12. A curve average GCI of 9.24% was calculated for the fine and the medium meshes.

4.2.2. Comparison of Results

The span-wise $Nu$ values are plotted against the dimensionless distance $s/c$ in Fig. 13. The heat transfer computed by solving internal flow only (CFX) was compared to the heat transfer computed using the CFX CHT procedure and to results obtained by Wright [15].

Computed results were consistently lower than those of Wright, except near the stagnation points. However, the Goldstein correlation [9], used in [15] instead of the detailed internal flow solution, does not consider the recirculation areas observed in computed results for $s/c$ ranging from $-0.1$ to $-0.05$ or $0.05$ to $0.1$. Moreover, the jet $Re$ was below the valid range of application of the correlation. For these two reasons, the anti-icing simulation results may be considered satisfactory.
Solving the CHT model or the internal domain alone gave similar $\text{Nu}$, except at the jet stagnation points. The difference between the two solutions for the $\text{Nu}$ averaged values in the leading area was 2.4 %, which is less than the GCI. For the parametric model study, only the internal flow was solved on the medium mesh.

5. PARAMETRIC MODELS

5.1. Impinging Jet

The heat transfer from a single impinging jet of a given fluid from a round nozzle depends mostly on $Re$ and $H/D$. The third parameter needed for the proposed Box-Behnken methodology is the Mach number. The maximum $M$ value was set at 0.3 in order to avoid any large compressible effects on the flow. Table 3 shows the minimal, intermediate, and maximal values for the three design variables selected. The objective function is $\text{Nu}_{\text{avg}}$ for $r/D$ from 0 to 6.25.

Based on Table 1, 13 simulations were run, since the last three rows are identical. Simulation results were compared to results obtained using Eq. (10), the only correlation valid for all the values in Table 3. The error
Table 3. DoE matrix used for the impinging jet.

was defined as follows:

\[ \varepsilon = 100 \times \frac{|Nu_{avg} - Nu_{Martin}|}{Nu_{avg}}. \]  \hspace{1cm} (13)

As Table 4 shows, the computed values were 20–30% higher than those predicted by the correlation.

Table 4. Numerical results DoE.

<table>
<thead>
<tr>
<th>Run number</th>
<th>(Nu_{avg})</th>
<th>(Nu_{Martin}) (Eq. 10)</th>
<th>(\varepsilon)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40</td>
<td>32.7</td>
<td>18.25</td>
</tr>
<tr>
<td>2</td>
<td>111.7</td>
<td>80.2</td>
<td>28.2</td>
</tr>
<tr>
<td>3</td>
<td>44.78</td>
<td>30.6</td>
<td>31.7</td>
</tr>
<tr>
<td>4</td>
<td>117.6</td>
<td>75.0</td>
<td>36.2</td>
</tr>
<tr>
<td>5</td>
<td>42.18</td>
<td>31.6</td>
<td>25.1</td>
</tr>
<tr>
<td>6</td>
<td>112.8</td>
<td>77.5</td>
<td>31.3</td>
</tr>
<tr>
<td>7</td>
<td>40.4</td>
<td>31.6</td>
<td>21.8</td>
</tr>
<tr>
<td>8</td>
<td>107.2</td>
<td>77.5</td>
<td>27.7</td>
</tr>
<tr>
<td>9</td>
<td>80</td>
<td>58.6</td>
<td>26.8</td>
</tr>
<tr>
<td>10</td>
<td>82.6</td>
<td>54.9</td>
<td>33.5</td>
</tr>
<tr>
<td>11</td>
<td>80.6</td>
<td>58.6</td>
<td>27.3</td>
</tr>
<tr>
<td>12</td>
<td>83.4</td>
<td>54.9</td>
<td>34.2</td>
</tr>
<tr>
<td>13</td>
<td>84.1</td>
<td>56.7</td>
<td>32.6</td>
</tr>
<tr>
<td>14</td>
<td>84.1</td>
<td>56.7</td>
<td>32.6</td>
</tr>
<tr>
<td>15</td>
<td>84.1</td>
<td>56.7</td>
<td>32.6</td>
</tr>
</tbody>
</table>

5.1.1. Parametric Model for the Impinging Jet

The system of equations \(A\beta = B\) was solved, with the \(15 \times 10\) rectangular matrix \(A\) corresponding to Table 1 and column vector \(B\) corresponding to the computed \(Nu_{avg}\). Since the system is overdetermined, the \(\beta\) coefficients of the polynomial are the solution in the least squares sense of the system of equations \(A\beta = B\). The polynomial is expressed as follows:

\[ f_{obj}(Re', (H/D)', M') = 84.1 + 35.242Re' + 2.01(H/D)' - 0.747M' + 0.28Re'(H/D)' - 0.955Re'z' + 0.05(H/D)' M' - 5.7925Re'^2 + 0.2125(H/D)^2 - 2.663M'^2, \]  \hspace{1cm} (14)

where \(Re', (H/D)', \) and \(M'\) replace \(\nu_4', \nu_2', \) and \(\nu_3'\) for clarity. For \(M = 0.2\), the resulting surface is plotted on Fig. 14 with numerical predictions. In the worst case, at point \((-1,-1)\), the difference between
the parametric surface calculation and the numerical prediction was about 10%. The maximum $Nu_{avg}$ value occurred at point $(1, 1, -0.31)$, which corresponds to $Re = 40,000$, $H/D = 6$, and $M = 0.167$. Equation (14) does not take $M$ into account and does predict that the maximum occurs at $Re = 40,000$, but for $H/D = 2$ for the parameter range considered. Figure 15 compares the $Nu_{avg}$ obtained from (10) to the parametric model at $M = 0.2$. It is clear that $Re$ has a greater effect than does $H/D$ on $Nu_{avg}$. In addition to 30% higher $Nu_{avg}$ values, the parametric model also predicts a faster increase as a function of $Re$ than does Eq. (10). Also, Eq. (10) predicts a small increase of $Nu_{avg}$ with a decrease of $H/D$ values, contrary to the parametric model.

### 5.2. Anti-Icing System

The three design variables chosen were $\alpha$, $H$, $\dot{m}$. Table 5 shows their minimal, intermediate, and maximal values.
<table>
<thead>
<tr>
<th>Dimensionless Variable</th>
<th>(\alpha) (deg)</th>
<th>(H) (m)</th>
<th>(\dot{m}\left(\frac{g}{s}\right))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>22.5</td>
<td>0.03</td>
<td>0.32</td>
</tr>
<tr>
<td>0</td>
<td>45</td>
<td>0.06</td>
<td>0.38</td>
</tr>
<tr>
<td>1</td>
<td>67.5</td>
<td>0.09</td>
<td>0.44</td>
</tr>
</tbody>
</table>

Table 5. DoE matrix used for the anti-icing system.

The objective function is heat transfer effectiveness, defined by Eq. (15) as the ratio of the area average wall heat flux between jet airflow and wall, \(\bar{q}_{W,\text{area}}\), divided by the summation of the average heat flow at each jet inlet, \(\sum q_{\text{in},\text{area}}\).

\[
f_{\text{obj}}(v'_1, v'_2, v'_3) = \frac{\bar{q}_{W,\text{area}}'}{\sum q_{\text{in},\text{area}}}.
\]

(15)

5.2.1. Parametric Model for the Anti-Icing System

The system of equations \(A\beta = B\) was solved as in Section 5.1.1, but with the column vector \(B\) corresponding this time to the computed heat transfer effectiveness. The polynomial is expressed as follows:

\[
f_{\text{obj}}(\alpha', H', \dot{m}') = 0.634 + 0.551\alpha' + 0.558H' - 0.09\dot{m}' - 0.1\alpha'H' + 0.05\alpha'\dot{m}' - 0.089H'\dot{m}' + 0.225\alpha'^2 + 0.284H'^2 + 0\dot{m}'^2.
\]

(16)

Fig. 16. Heat transfer effectiveness as a function of \(H'\) and \(\alpha'\), for \(\dot{m}' = 0\).

The zero coefficient of the term \(\dot{m}'^2\) indicates that no quadratic effect is associated with mass flow rate. The 3D response surface model with the design variable \(\dot{m}'\) kept at the mid-range value 0 is shown on Fig. 16. Heat transfer effectiveness exceeded 1 since heat was also added by convection between the hot piccolo tube wall and the internal flow. The maximum \(f_{\text{obj}} = 1.6\) was observed at \((H', \alpha') = (-1, 1)\) or \((H, \alpha) = (0.03\ m, 67.5^{\circ})\).
6. CONCLUSION

Parametric models based on three design variables were obtained using the Box-Behnken DoE approach. The database was populated using ANSYS CFX 12.1 CFD code to solve the RANS equations using the $k-\omega$ SST turbulence model. The $Nu$ values predicted by CFD were validated against empirical, numerical, and experimental results for an unconfined impinging jet from a round nozzle. The CFD results were within the range of uncertainty. For an anti-icing system, the internal $Nu$ values predicted by CFD using CHT or a constant wall temperature were compared with other numerical results. In both cases, results agreed with values published for a wing exposed to dry airflow. A second order parametric model was then built for $Nu_{avg}$. Maxima coincided with maximal $Re$ and $H/D$ spacing, in agreement with empirical correlations. For the anti-icing system, a second-order quadratic model of heat transfer efficiency is proposed. The optimal anti-icing system design deduced from RSM plots features intermediate mass flux, maximal jet angle, and minimal piccolo tube to inner wall distance.

The Box-Behnken DoE approach is useful as long as initial solutions are nearly optimal. One of its weaknesses is the difficulty of determining new objective function values at additional points. However, its simplicity makes it possible to build parametric models subsequently to predict heat flux at several locations along the wing leading edge.

ACKNOWLEDGEMENTS

The authors acknowledge the contribution of the National Science and Research Council of Canada to the funding of portions of this research.

REFERENCES

2. Canadian Aviation Regulations, Minister of Justice, 602.11, Canada, 2012.


