2011-38-0023

Proposed Wall Function Models for Heat Transfer around a Cylinder with Rough Surface in Cross Flow

Guilherme Araujo Lima da Silva, Marcos Noboru Arima
Aero-Thermal Solutions for Industry

Natashe Nicoli Branco, Marcos de Mattos Pimenta
University of São Paulo

ABSTRACT

This paper proposes wall function models to simulate the heat transfer around a cylinder in cross flow with an isothermal and rough surface. The selected case has similitudes with aircraft wing icing: the ice roughness shape, height and distribution. Moreover, the flow is somewhat similar to that found on iced airfoil; and the surface is isothermal like when icing. The Reynolds-Averaged Navier-Stokes, turbulence, energy and mass conservation 7-equation system is solved by two Computational Fluid Dynamics (CFD) codes. To represent accurately the effects of roughness on the heat transfer, the present authors had to modify both codes and to propose new thermal wall functions for them. In addition, it was implemented a momentum wall function that is not so common in CFD codes but it is a standard in aircraft icing simulation. Basically, the present paper is focused on presenting the simulation results improvement obtained by the implementation of thermal wall functions that also considered the effect thermal resistance of viscous sub-layer on convective heat transfer. The numerical results were compared to experimental data of heat transfer around cylinders with an isothermally heated surface and equivalent sand grain roughness height of $K_s/D = 900 \cdot 10^{-5}$. For the selected case, the flow regime was trans-critical at Reynolds numbers $Re = u_e \cdot D/\nu = 2.2 \cdot 10^5$. Due to significant blockage effects present in the experimental data, the tunnel walls were also meshed and simulated. In sum, the implementation into CFD codes was considered adequate because the results were close to experimental data around the whole cylinder surface, not only along the leading edge or the separated region. The results accuracy were improved when compared with CFD factory original models. However, the results indicate that further works on wall functions and their validation are required before using CFD in wing aircraft icing.

INTRODUCTION

During glaze ice accretion, the convective heat transfer is the main mechanism to remove the solidification enthalpy causing water to freeze. The mass of accreted ice depends mainly on mean heat transfer but the position where the runback water liquid freezes depends on local heat transfer conditions. The convective heat transfer coefficient sharply increases when laminar-turbulent transition occurs and causes a turbulent flow over a fully rough surface, causing in somecases the water freezing to develop. Other effects related to water film hydrodynamics, surface tension, beads formation and rivulets flow have influence as cause change the surface wetness and heat exchange area between the water film surface and the environment. As the onset of water freezing bears relation to the ice shape, it is important to evaluate the convective process as accurate as possible [1]. Due to icing tunnel operational limitations, the convective heat transfer may differ significantly from that in-flight operation, which makes it necessary to have an icing simulation tool to evaluate the tunnel-flight similitude. The turbulent skin friction and heat transfer are both enhanced. At same time, the roughness also affects the laminar-turbulent transition region onset position and its extension. The rough surfaces effects on heat transfer and transition were studied in icing literature[2, 3, 4, 5] and its influence on glaze ice shape were evaluated recently [6, 7].

In his classic work, Makkonen [2] used Achenbach [8] experimental data over a cylinder with isothermal and rough surface in order to validate his boundary layer integral evaluation procedure. The ultimate objective of the author was to model the laminar, transitional and turbulent heat transfer around icing cylinders in order to predict the mass of ice accreted on electrical power transmission lines. The laminar-turbulent transition was assumed to be abrupt, i.e., the flow passes from laminar to turbulent regime instantaneously downstream the onset position. Some of the classic two-dimensional airfoil icing prediction codes use a heat transfer calculation procedure based on the integral evaluation of the boundary layer, like LEWICE [9], Onera2D [10], TRAJICE [11] and others [2, 5]. These icing codes consider a smooth surface in laminar regime, a fully rough surface in turbulent regime and that the laminar-turbulent transition occurs abruptly downstream its onset position. Those works have similar characteristics to the work of Makkonen[2], with exception of few details. Basically the models use the integral analysis and the ice roughness height is used to predict the transition onset position as well as the momentum and heat transfer enhancement effects. All the authors use the the Nikurases's concept of equivalent sand grain roughness height K_s. This approach was criticized by Pimenta[12] because one parameter is used to represent three independent roughness characteristics: height, distribution and shape. Recently, Stefanini et al.[13] proved that it is necessary to use two parameters in order to represent Achenbach's closed-packed pyramidal roughness[8] by Owen's St_k correlation [14]. Therefore, there is a clear advantage to use a laboratory controled experiment with a precise definition of K_s [8] in the validation of the heat transfer prediction of integral or differential icing simulation codes.

Differential analyses application to aircraft icing problems momentum are not so common as the integral form. The results of California State University Long Beach group have been validated in predicting momentum and heat transfer around icing airfoils [15, 16, 17]. These works also used the Nikurases's concept of equivalent sand grain roughness height K_s in the eddy viscosity formulation. Despite being differential RANS, they are parabolic boundary-layer type of codes. The present authors have not found works on Computational Fluid Dynamics (CFD) or on elliptical codes applied to icing simulation that are specifically validated against convective heat transfer experimental data of cylinders or airfoils.

OBJECTIVE

The objective of the present paper is to propose momentum and thermal wall functions to estimate the local friction and heat transfer distribution around a rough cylinder in cross flow for $Re = 2.2 \cdot 10^5$ with pyramidal roughness of equivalent height of $K_s = 0.00135$ m by means of CFD numerical tools. The CFD results generated herein are compared to the heat transfer experimental data of Achenbach[8] and the integral analysis results of Stefanini et al[13]. As required, the paper discusses the improvements obtained with proposed wall functions, the numerical convergence aspects and the experimental data limitations.

CFD SOLVERS

The present authors selected the commercial-differential-RANS-finite volume solver CFD++[18], which is well known in the aeronautical market and has been validated for aircraft friction drag[19] as well as for engine lip anti-ice analyses[20] in recent years; and also the open source-differential-RANS-finite volume solver OpenFOAM[21, 22] - OF - that has been recently applied to helicopter icing problems[23].

The present paper used the CFD++ version 10.5 provided by Metacomp Technologies in beta testing version, which has the Stefanini et al.[13] heat transfer option available. On the other hand, the OF version was the 1.6-ext, which is the version extended and maintained by INTERNET community of the official version 1.7.1 released by OpenCFD. The OF1.6-ext code used the momentum and thermal wall functions modified by the present authors. The OF code is actually a CFD toolbox in object-oriented language C++, therefore, it allows user customization directly in the source code.

WALL FUNCTIONS MODELS DESCRIPTION

STANFORD MODEL The momentum boundary layer thickness, skin friction, heat transfer, turbulent Prandtl number - Pr_t - as well as heat and momentum analogy expressions developed by the Stanford University group from 1958-1983 [12, 24, 25] are still important references for icing works such as those developed by NASA[9], ONERA [10] and Makkonen[2]. They classified the flow regimes over rough surfaces based on equivalent sand grain height:

$$\text{Flow regimes over rough surfaces} \begin{cases} Ks^+ \le 5 & \text{smooth} \\ 5 < Ks^+ < 70 & \text{transitionally rough} \\ Ks^+ \ge 70 & \text{fully rough} \end{cases} \tag{1}$$

where $Ks^+ = u_\tau Ks/\nu$, is the Reynolds number associated with roughness height in wall coordinates and $u_\tau = \sqrt{\tau_w/\rho}$ is the shear velocity. When $Ks^+ < 5$ the surface is hydraulic smooth. After $Ks^+ > 70$ the viscous sublayer disappears and the wall skin friction is independent of Re. In this case, only pressure drag, which is caused by the effect of dynamic pressure in upstream roughness surface, is important. However, the flow subjected to retard by drag continues deteriorating heat transfer and varying with increasing Re[26].

Momentum wall function over rough surfaces The velocity profile in wall coordinates $u^+ = u/u_\tau$ and $y^+ = yu_\tau/\nu$ is given by:

$$u^+ = \frac{1}{\kappa}\ln\left(\frac{32.6\cdot y^+}{Ks^+}+1\right) \approx \frac{1}{\kappa}\ln\left(\frac{32.6\cdot y^+}{Ks^+}\right) \tag{2}$$

where $u^+ = u/u_\tau$, $\kappa = 0.41$ is a constant, u is the mean velocity at point p, $u_\tau = \sqrt{\tau/\rho}$ is the shear velocity, and τ is the shear stress.

Thermal wall function over rough surfaces The momentum and heat transfer analogy function valid for the fully turbulent flow region but not for rough sub-layer is given by:

$$\frac{\mu_t}{\alpha_t} = Pr_t \tag{3}$$

where μ_t is the turbulent viscosity, α_t is the thermal diffusivity and Pr_t is the turbulent Prandtl number. By using with Eq. (2), (3) plus definitions of t^+ and assuming Pr_t constant through the fully turbulent boundary layer,

$$t^+ = \delta t_0^+ + \frac{Pr_t}{\kappa}\ln\left(\frac{32.6\cdot y^+}{Ks^+}\right) \tag{4}$$

The first term of right hand of Eq (4) is related to the thermal resistance of viscous sub-layer influenced by roughness. As this is in series with the fully turbulent layer, the roughness presence causes some deterioration effect on heat transfer that reduces the augmentation effect[26]. Therefore, the shift in t^+ profile can be written as a roughness St:

$$\delta t_0^+ = \frac{1}{St_k} = \frac{\rho c_p u_\tau}{h_k} \tag{5}$$

Thus a thermal convection correlation in sub-layer can be defined in an useful form:

$$St_k = C\cdot (Ks^+)^a \cdot Pr^b \tag{6}$$

where C, a and b are experimental calibration constants and Pr is the molecular Prandtl number. The heat and momentum analogy was developed in terms of Pr_t, Pr, C_f and K_s^+ by Kays and Crawford[24]:

$$St = \frac{C_f/2}{Pr_t + \frac{\sqrt{C_f/2}}{St_k}} \tag{7}$$

where C_f is the rough skin friction, the first term in right hand side denominator of Eq. (7) refers to the heat and momentum analogy of the turbulent thermal boundary layers and the second term of denominator of Eq. (7) takes into account the independent effect of roughness on the viscous sub-layer of the thermal boundary layer. Equations (5) and (7) have one important consequence: the analogy factor Pr_t will be attenuated by the sub-layer thermal resistance that varies with the ratio between $\sqrt{C_f/2}$ and St_k, thus, the lower the St_k and higher the C_f, farthest from the Pr_t the analogy will be.

OPENFOAM MODELS This section describes the current wall functions implemented in OF. The intention is not to detail or discuss all aspects of them herein but present to the reader the equations that have been changed by the present authors, which are the momentum and thermal diffusivities, μ_t and α_t. The complete model is presented by Tapia[27].

OF Momentum wall function for rough surfaces The velocity profile in boundary layer variables:

$$u^+ = \frac{1}{\kappa}\ln(E\cdot y^+) - \Delta B \tag{8}$$

Where the $E = 9.8$ and and $\kappa = 0.41$ are constants. The code follows the Cebeci[28] model:

$$\Delta B = \begin{cases} 0 & Ks^+ \le 2.5 \\ \frac{1}{\kappa}\ln\left[\frac{K_s^+ - 2.25}{87.75} + C_s K_s^+\right]\sin[0.4258\cdot(\ln K_s^+ - 0.811)] & 2.5 < Ks^+ < 90 \\ \frac{1}{\kappa}\ln(1 + C_s K_s^+) & Ks^+ \ge 90 \end{cases} \tag{9}$$

Where C_s depends on shape and distribution of roughness elements and,

$$y^+ = y\frac{\rho C_\mu^{0.25} k^{0.5}}{\mu} \qquad and \qquad K_s^+ = K_s\frac{\rho C_\mu^{0.25} k^{0.5}}{\mu} \tag{10}$$

Due to its equilibrium assumptions, the Eq. (10) is theoretically valid when $y^+ > 20$. Where k is the turbulent kinetic energy and $C_\mu = 0.09$ is a constant. Then, the turbulent viscosity is estimated by the OF's wall function called *mutRoughWallFunction*:

$$\mu_t = \mu\left(\frac{y^+\kappa}{\ln(Ey^+/e^{\kappa\Delta B})} - 1\right) \tag{11}$$

OF Thermal wall function for rough surfaces The original wall function implemented in the OF code - *alphatWallFunction* - assumes that the analogy between heat and momentum transfer is valid throughout the whole layer:

$$\alpha_t = \frac{\mu_t}{Pr_t} \tag{12}$$

Where Pr_t is a constant that can be provided by the OF user. Currently, the OF does not have any model for Pr_t variation across boundary layer. In addition, both OF versions 1.7.x and 1.6-ext do not have a specific thermal wall function for rough surfaces, thus, the Eq. (12) is the only option for users to simulate heat transfer.

PROPOSED MODELS FOR OPENFOAM This paper proposes herein the use of the Stanford model in the momentum wall function because it has been a standard in icing area. In addition, the present authors propose a new model for thermal wall function, which has never been applied in icing problems nor employed in OF code. The intention is to improve the predictions of the momentum and heat transfer over the rough surfaces in OF 1.6-ext because the Eq. (12) does not have the effect of the viscous sub-layer but only the effect of the fully turbulent thermal resistance. Thus, the analogy of Pr_t must be corrected by a factor dependent on fluid and sub-layer roughness properties, K_s, $\sqrt{Cf/2}$, Pr_t and Pr, similar to the form of Eq. (7).

OF Momentum wall function for rough surfaces By making Eq. (2) equal to Eq. (8), one will find the relation between constant C_s and the Stanford model for rough surfaces subjected to turbulent flow:

$$C_s = \frac{E}{32.6} - \frac{1}{K_s^+} \tag{13}$$

Equation (13) is important because it makes possible to use the Stanford model, which is used by classical icing codes, in OF momentum wall function. Therefore, the icing engineer can keep the same K_s values in simulations. When K_s^+ tends to infinity and $E = 0.98$, the C_s of Eq. (13) tends to a constant value of the fully rough surface $C_s \approx 0.3$. Finally, the Eq. (9) is no longer used in the proposed model since Stanford model does not adopt that. Thus, the function ΔB is given by:

$$\Delta B = \frac{1}{\kappa} \ln(1 + C_s K_s^+) \tag{14}$$

Equation (14) is replaced in Eq. (8) and Eq. (11) to provide u^+ and μ_t respectively. The rough flow regime ranges depending on K_s^+ were compared to those values of Stanford model, Eq. (1). Thus, all OF's equations remain the same except the replacement of ΔB of Eq. (9) by the (14) and the introduction of the variable function C_s of Eq. (13) instead of constant C_s given by the user.

It is important to emphasize that the present paper did not attempt to implement major changes in OF's momentum wall functions but only to adapt them to reflect the standard models used in ice accretion area. Paper scope restrictions does not include separated, adverse pressure gradient, compressible or accelerated flow regions. Those limitations are considered herein to be inherent to the type of momentum wall function model of OpenFOAM.

OF Thermal wall function for rough surfaces The present authors recommend the use of momentum and heat transfer analogy factor appropriated to represent the flow over rough surfaces:

$$\eta = \frac{1}{Pr_t + \frac{\sqrt{C_f/2}}{St_k}} \tag{15}$$

Where the St_k is given by Eq. (6) and constants $a = -0.45$, $b = -0.8$ and $C = 1.42$ as defined by Stefanini et al.[13] for closed-packed pyramidal roughness that Makkonen[2] considered to be similar to icing surfaces. In oder to calculate C_f in OpenFOAM solver, the following equations were employed:

$$\frac{u_\tau}{u} = \sqrt{C_f/2} \quad \text{and} \quad u_\tau = \sqrt{\tau/\rho} \quad \text{with} \quad \tau = (\mu + \mu_t) \cdot |\nabla U \cdot \vec{n}|, \tag{16}$$

Equations in (16) do not have the limitations of Eqs. in (10) because it actually calculates the τ at wall. The present authors propose to employ the momentum and heat transfer analogy by coding a new wall function into OF (OpenFOAM) source-code by changing from Eq. (12) to that described below:

$$\alpha_t = \mu_t \cdot \eta \cdot Pr_t \cdot A \tag{17}$$

Where, η is the analogy factor and can be calculated by Eq. (15); A is a non-dimensional constant obtained from dimensional analysis. In the present work, it is suggested to be simply an arbitrary constant, $A = 2$. This assumption may be subjected to change as the model validation is carried out by future works. The present authors coded the Eqs. (15) and (17) into the OF source code in order to modify the wall function called *alphatRoughWallFunction*.

PROPOSED MODEL FOR CFD++ During the paper development, the present authors collaborated with CFD++ scientists in order to develop a new thermal wall function correction to take into account the thermal resistance of rough sub-layer during ice accretion. Regarding the momentum wall function, there was no change or correction factor needed to be implemented in CFD++. Basically, it already has a very complete model that has been extensively validated and covers a wide range of flow effects and regimes. In this case, all the modeling and coding work were performed by CFD++ technical team. For proprietary reasons, the wall functions can not be described here. Despite this restriction, the CFD++ solver was used to to benchmark the OF1.6-ext wall functions development and mainly to firstly validate its own numerical results against experimental data.

CFD++ Thermal wall function for rough surfaces The new proposed analogy factor is based on Stefanini et al.[13] work about heat transfer cylinders with closed packed pyramidal roughness[8, 14] has been used in ice formation by Makkonen[2]. This new heat transfer user option will be available in a next release of CFD++.

EXPERIMENTAL TEST CASE

The Achenbach's[8] tunnel cross section was 900 mm height by 500 mm width. The length as well as the inlet contraction and outlet expansion geometries were not provided. The cylinder was 150 mm diameter and 500 mm long. The present authors selected one a trans-critical condition where the $Re = 2.2 \cdot 10^5$ and $K_s/D = 900 \cdot 10^{-5}$. In the trans-critical condition, the cylinder total drag force is above its minimum point and laminar-turbulent transition is close to the stagnation point. By analyzing the geometry, one can see that the blockage ratio is significant - $6:1$ $(900:150)$. Therefore, it is necessary to simulate whole tunnel in CFD to capture wall effects. The free-stream temperature was 303.15 K. The cylinder isothermal surface temperature varied between test runs but was kept in a constant level above free-stream temperature in each run. The author[8] informed that level was between 9 and 60 K but did not inform the exact value of each test run. Thus, in the present paper, the cylinder wall temperature is assumed as $T_w = 312.15$ K except when noticed. The tunnel pressure for this Re range is close to the atmospheric level. The turbulence level for all experiments was $Tu = 0.45\%$ but there is no information about length scale as well as the size of the grid installed in tunnel or the distance between it and the body. The adopted Re is close to what an actual wing leading edge ice protected region (5% to 15% chord) may encounter in flight or tunnel under icing conditions[13].

For the selected case, $Re = 2.2 \cdot 10^5$ and $K_s/D = 900 \cdot 10^{-5}$, Achenbach's previous results[29] indicated the angular position of separation on $\theta_{sep} = 91.4^\circ$ and of transition on $\theta_{tr} = 15^\circ$ clockwise from stagnation.

CFD SIMULATION SETUP

The mesh used in simulations was the same for both solvers and all cases: 611278 elements, 144768 triangular prisms and 409408 tetrahedral. The X (aligned with flow direction) ranges from -2 to 10 m, Y (vertical direction) from -0.45 to 0.45 m and Z (lateral direction) from 0 to 0.25 m. There was a prism layer around the cylinder with $y1 = 1$ mm in order to guarantee the y^+ within the acceptable range for regular wall functions ($11 < y^+ < 300$) since the y^+ varies with shear velocity u_τ variation.

At inlet, the conditions were set to $U_x = 23.45$ m/s, $T = 303.15$ K, $Tu = 0.45\%$. At outlet, the back pressure was fixed as 101325 Pa. All tunnel walls (side, top and bottom) were considered to be smooth simulated with standard wall functions. At $Z = 0$, a symmetry plane was defined. Around the cylinder, it was considered a rough and isothermal surface at $T_w = 312.15$ K, for which was simulated with wall functions developed herein. The roughness height of $K_s = 1.35$ mm and the Turbulent Prandtl $Pr_t = 0.89$ were used in all simulations.

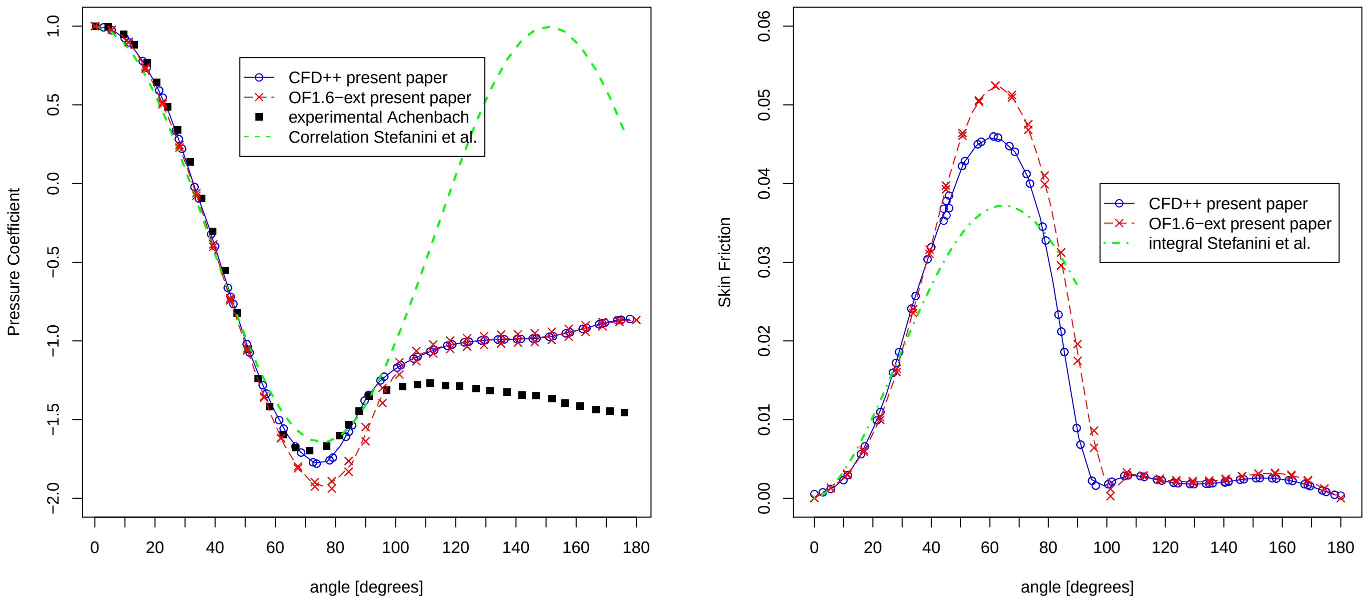
In both solvers, the *realizable-k-ε* turbulence model was used to solve the full turbulent flow without transition. Despite they have different implementations, the present paper will not describe or analyze them since they are out of present scope and objectives. Therefore, it is assumed that the turbulence model is an inherent characteristic of each solver and there will be no attempt to change constants or terms to make them compatible or similar.

RESULTS

Figure 1 presents the flow solution around the rough circular cylinder. The pressure coefficient calculated by CFD++ is compared to OF1.6-ext and experimental data in Fig. 1(a). For all figures shown in present paper the markers do not represent actual points of numerical solutions since the mesh is much finer than the marker spacing. They are a visual cue to improve the legend assignments. The lines are result of a spline interpolation of the upper and lower cylinder surface averaged values, which are indicated by markers. The OF1.6-ext curves were extracted by interpolation from Paraview[30] software but the CFD++ used its own interpolation tool to get points in the intersection of the symmetry plane ($z = 0$) and the cylinder surface. An empirical correlation is used to represent C_p:

$$C_p = 1 - a \cdot \sin(b \cdot \theta) \tag{18}$$

Where $C_p = 2 \cdot (p - p_\infty)/(\rho \cdot U^2)$; a and b are constants; θ is the angle in radians counter clockwise from point $x = -0.075$ m; ρ is the density and U is the velocity at free-stream conditions. Since Eq. (18) is similar to theoretical potential flow solution around a circular smooth cylinder, it was used to fit experimental data in the range of $Re = 2.2 \cdot 10^5$ to $4 \cdot 10^6$ and distance -0.075 m $\leq x \leq 0$ m, i.e., angle in range $0 \leq \theta \leq 90^\circ$. The present paper adopted $a = 2.646$ and $b = 1.194$ as recommended by Stefanini et al.[13]. As one can see in Fig. 1(a), the solution of CFD++ was closer to experimental data than that of OF1.6-ext upstream the separation point but both deviate from experimental data downstream of it. One possible reason for this deviation for both codes is that the tunnel geometry or boundary conditions simulated do not represent the actual installation and operation. The results are considered acceptable because the present paper is not focused to solve only the C_p but mainly heat transfer around cylinder.



(a) Pressure Coefficient - C_p

(b) Skin Friction - C_f

Figure 1: Flow Solution for $Re = 2.2 \cdot 10^5$ and $K_s/D = 900 \cdot 10^{-5}$

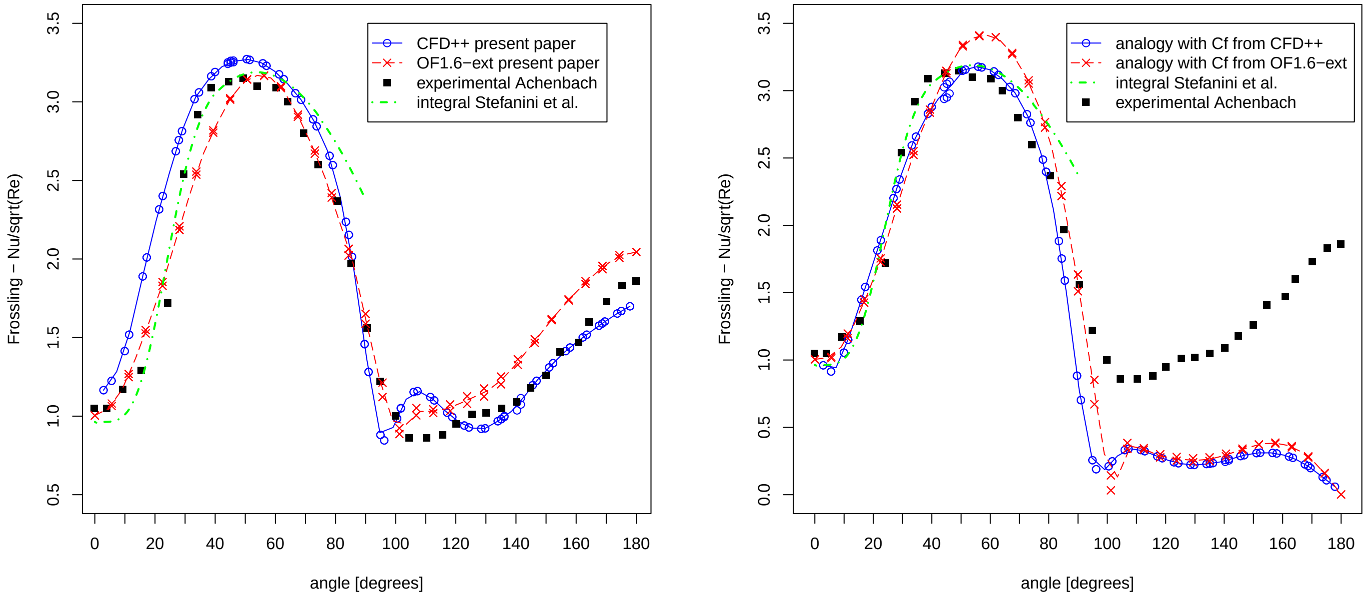
The skin friction, defined herein as $C_f = 2 \cdot \tau/(\rho \cdot U^2)$, where τ is the shear stress, ρ is the density and U is the velocity at free-stream conditions. The C_f numerical results of the differential codes CFD++ and OF1.6-ext are compared with Stefanini et al.[13] integral boundary layer analysis results in Fig. 1(b). The OF1.6-ext predicted a separation point downstream of CFD++, which consequently resulted in a difference in maximum value of C_f. The integral analysis deviated from both differential solutions at same position and could not reach their maximum value of C_f. As the OF1.6-ext does not have a laminar-turbulent transition model implemented yet in its code, it was ran as full turbulent in whole present paper. In the case of CFD++ , it does have a transition model to predict onset and its development but it was not used herein. However, the integral analysis employed a laminar-transition model as defined per Stefanini et al.[13]. The OF1.6-ext used momentum wall function proposed by present authors in Eqs. (11) and (14) with $C_s = 0.3$. On the other hand, CFD++ use its own factory momentum wall function with no change at all. The difference in C_f values calculated by three codes indicates that a dedicated validation with C_f experimental data around cylinders is required for each code.

Figure 2 shows the results for the heat transfer around the rough circular cylinder for the selected experimental case. The first Fig. 2(a) presents the Frössling number Fr results provided by OF1.6-ext and CFD++ compared to experimental data and integral analysis results. The non-dimensional Fr is defined as:

$$Fr = \frac{Nu}{\sqrt{Re_D}} = \frac{h \cdot D}{k_a ir} \cdot \sqrt{\frac{\nu}{U \cdot D}} = \frac{\dot{q}'' \cdot D}{k_a ir \cdot \Delta T} \cdot \sqrt{\frac{\nu}{U \cdot D}} \tag{19}$$

Where D is the cylinder diameter; h is the local convective heat transfer coefficient; $\dot{q}''$ is the local surface heat flux; $k_a ir$ is the thermal conductivity of the air; ν is the dynamic viscosity; ΔT is the temperature difference between the free-stream and the wall.

Figure 2(a) shows that OF1.6-ext results are closer to experimental data than CFD++ near stagnation and downstream F_r peak value. CFD++ appears to capture better the overall shape of F_r curve than OF1.6-ext and integral analysis[13]. Finally, the integral analysis[13] is representing more accurately the F_r distribution along favorable pressure gradient region than OF1.6-ext and CFD++. The thermal wall function used in OF1.6-ext is given by Eqs. (17) and (15) and same momentum wall function of Fig. 1(b). Regarding CFD++, the new modified wall function called "*Stefanini Heat Transfer*" was used. The deviations between OF1.6-ext results and experimental data in separated region may be due to the code convergence issues, the proposed thermal wall function limitations - Eq. (17) or, likely, the non-adequacy of the analogy factor of Eq. (15) or turbulence model to represent the flow since it comes from boundary-layer attached



(a) Frössling - $F_r = Nu/\sqrt{Re}$

(b) Analogy with Skin Friction C_f from CFD++ and OF1.6-ext

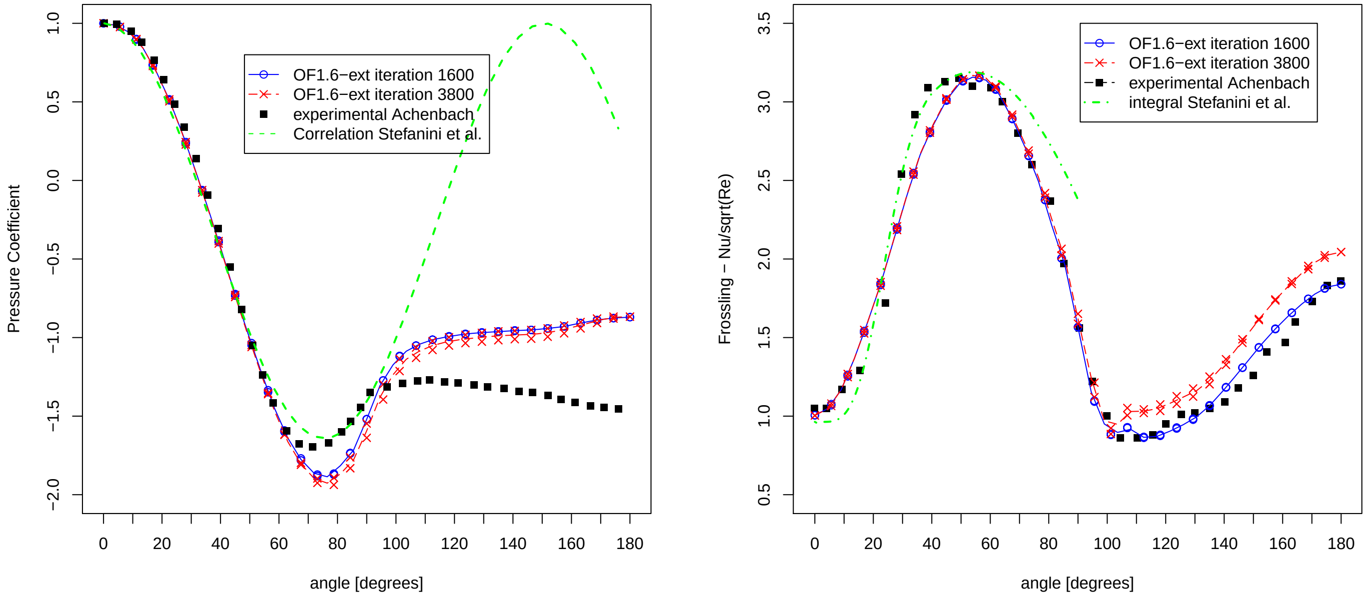
Figure 2: Heat Transfer Solution for $Re = 2.2 \cdot 10^5$ and $K_s/D = 900 \cdot 10^{-5}$

flows. In order to answer those questions a thoroughly validation is required to cover different Re, pressure gradient variation effects and roughness element heights, shape and distribution. As expected, the integral analysis Fr results from other authors[13] shows satisfactory adherence to experimental data where the pressure gradient is almost constant and favorable. Due to its assumptions, the integral analysis failed to predict Fr in adverse pressure gradient and downstream the separation position regions.

In order to compare the predictions of the analogy expression used by classic icing codes [9, 10, 11] against experimental data, which is given by Eq. (7), the C_f calculated by both CFD codes were used to estimate heat transfer over the rough and isothermal surface. In certain manner, this exercise serves also to check the values of C_f calculated since one agrees that Eq. (7) provides accurate results for boundary-layer attached regions. The integral analysis [13] results presented in two sub-figures of Fig. (2) used the same analogy factor to calculate F_r from C_f. All three analyses (OF1.6-ext, CFD++ and integral) provided very close results to each other but the integral was closer to experimental data along the favorable pressure gradient region. However, the CFD++ differential code provided results with smaller deviation from experimental data around whole cylinder, what may indicate that CFD++ is predicting the rough C_f more accurately than the other two codes. The significant deviation between CFD codes predictions and experimental data in adverse pressure gradient and separated regions may be due to two factors: 1) a limitation of the analogy expression Eq. (7) due its assumptions and formulation; 2) an inaccurate estimative of local C_f to calculate St by Eq. (7) due to approximation local velocity by $u_e^2 = (1 - C_p) \cdot \rho \cdot U^2/\rho_e$, which is valid only for boundary-layer attached flows. For icing problems not involving flow separation, the deviations found is not an issue because the icing analysis usually is focused on leading edges of airfoils or cylinders not on the regions with massive separated flow or wake.

NUMERICAL CONVERGENCE IN SEPARATED REGION The OpenFOAM code had some convergence issues in simulating the reference case of the present paper. First of all, the compressible RANS solver *rhoSimpleFoam* that comes with the official code, OF1.7.1, as provided by OpenCFD from their software repository, did not converged to the expected solution. Several configuration attempts were made by varying simulation running strategies, numerical discretization, relaxation and linear solvers parameters, which are given by files *fvSolution* and *fvSchemes*. However, the version OF1.6-ext converged satisfactory with pretty standard configuration by running a potential solver *rhoPotentialFoam* to prepare flowfield and after *rhoSimpleFoam*. This fact may indicate some implementation difference between versions, maybe related to tetrahedral dominant meshes, or a bug correction in the OF1.6-ext version, which is maintained by INTERNET community. As one can see in Fig. 3, the OF1.6-ext solution at iteration 1600 provided closer Fr results to experimental data along separated region than the OF1.6-ext solution at iteration 3800. This divergence is clearly associated with separation point and wake flow. As shown C_p in Fig. 3(a) and Fr in Fig. 3(b), the results are close to each other with small deviation from experimental data in attached flow region.

COMPARISON WITH FACTORY CFD MODELS In order to asses the accuracy improvements generated by the proposed models, it is necessary to compare the heat transfer results before and after the changes proposed by the present authors. Obviously, the baseline

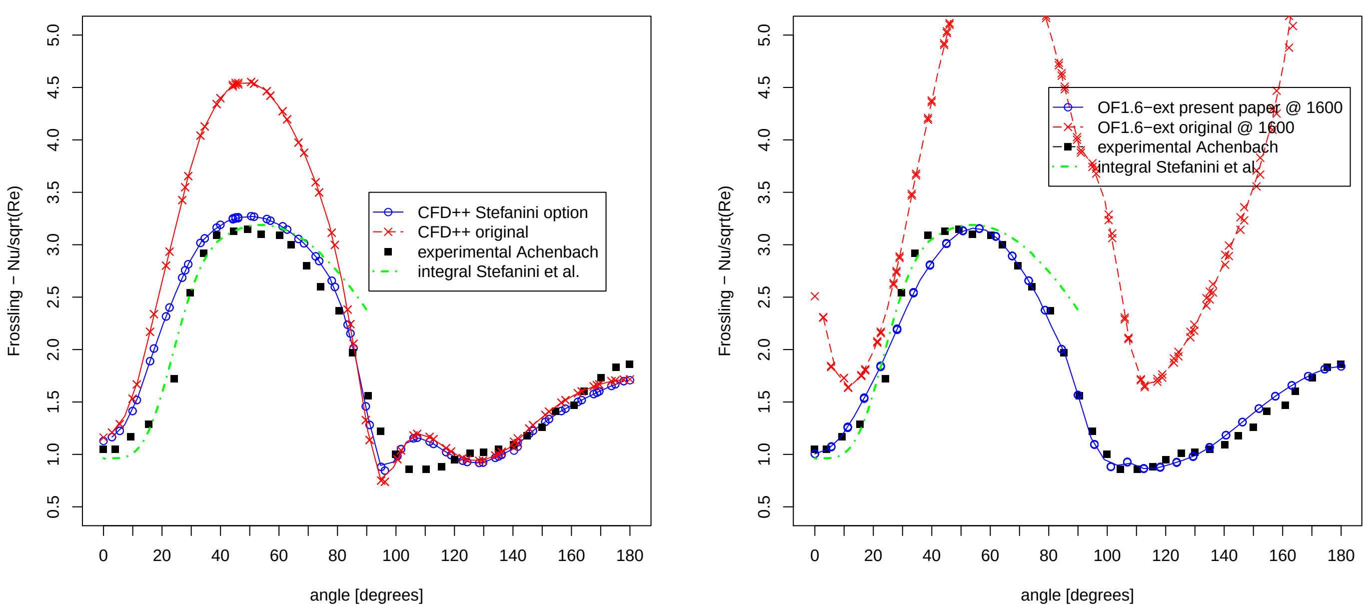


(a) Pressure Coefficient - C_p

(b) Frössling - $F_r = Nu/\sqrt{Re}$

Figure 3: OF1.6-ext results at iteration 1600 and 3800

models are the previous version or the factory implemented wall function models in CFD++ and OF1.6-ext codes respectively, which are called original in Fig. 4. In both cases, only the thermal wall function is varied from factory settings to the proposed models. The CFD++ was run with Stefanini heat transfer option on and off to generated curves in Fig. 4(a). Only the effect of new thermal wall function, Eqs. (17) and (15), is analyzed by comparing OF1.6-ext results in Fig. 4(b) , since the momentum wall function is the proposed one for both runs, Eqs. (11) and (14). It is clear that the modified models contributed to improve accuracy in both codes because it attenuates the heat transfer enhancement caused by roughness due to the thermal resistance of viscous sub-layer. The OF1.6-ext was compared at iteration 1600 because the original model presented worse convergence issues than proposed model.

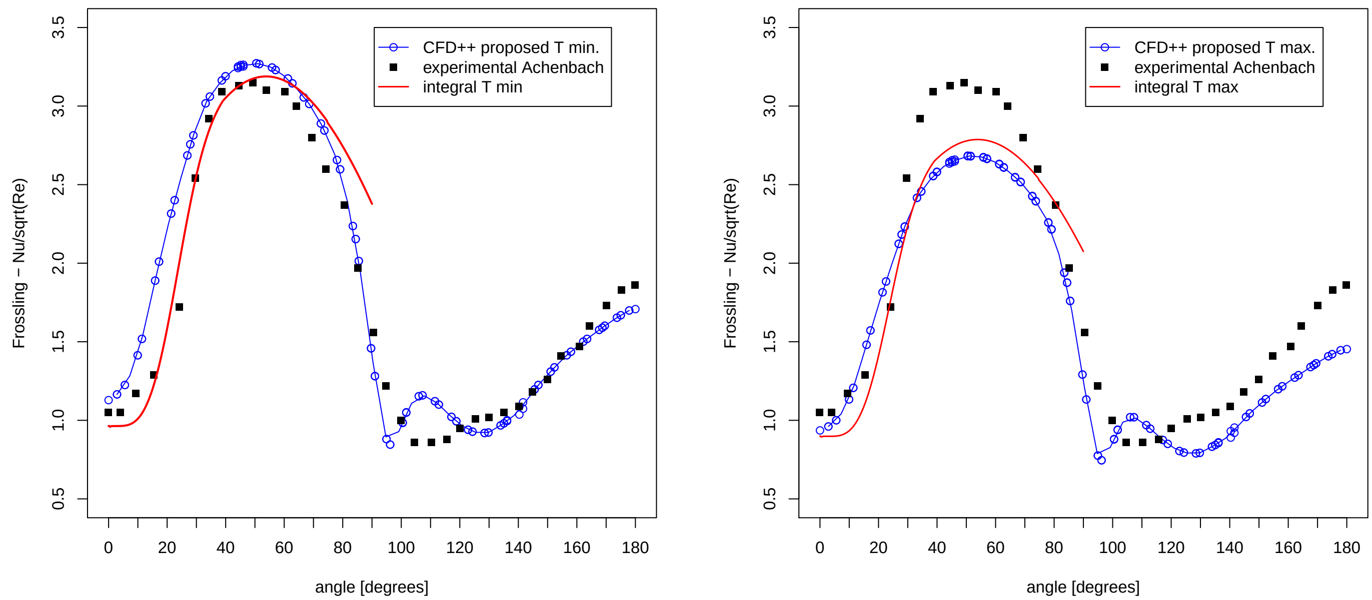


(a) CFD++ results

(b) OF1.6-ext results

Figure 4: Improvement obtained with proposed models in Fr prediction

SURFACE TEMPERATURE LEVEL EFFECT Figure 5 presents the effect of surface temperature level on heat transfer coefficient estimation. As noticed in previous section, Achenbach[8] did not provide the surface temperature for each test run but only the temperature range that it varied between test runs, from 312.15 to 363.15 K, i.e., 9K $< \Delta T <$ 60 K. Therefore, its necessary to assess the impact of the temperature level on the solution obtained by the CFD code. If the peak value of Fr is considered, the results provided by CFD++ were below the integral results[13] for the maximum temperature level as show in Fig 5(b) but above the integral[13] for the minimum as shown in Fig. 5(a). In both conditions, the integral analysis provided results closer to experimental data than differential analysis along favorable pressure gradient region. The over or undershooting characteristic as well as the deviations in forward part of cylinder may be related to the full turbulent flow solution because CFD++ did not solve the transitional flow as the integral[13] did. Another conclusion from Fig. 5 is that Achenbach[8] experimental data set can be used for heat transfer and wall function models development but it can not provide sufficiently accurate basis for models fine calibration, i.e., constants definition, since the surface temperature level has not been published.



(a) Surface Temperature at 312.15 K

(b) Surface Temperature at 363.15 K

Figure 5: Cylinder Isothermal Surface Temperature Level Effect

SUMMARY/CONCLUSIONS

A new thermal wall function model was developed and implemented by present authors in OF1.6-ext source code. The CFD++ software was changed to incorporate collaborations of present authors by its providers. The OF1.6-ext results are good in forward part of cylinder mainly and the CFD++ results are considered good in all regions with a curve shape close to experimental data despite slightly overestimated in some points. In addition, a momentum wall function, which is standard in icing area, was implemented into OF1.6-ext code by present authors. Both codes numerical results were compared to selected experimental data for the same testing conditions. The deviations, limitations and inaccuracies were analyzed in order to provide future subsidies for a fully three-dimensional icing or de-icing base on computational fluid dynamics (CFD) technology. The new developments proposed herein improved the prediction of the heat transfer coefficient around the circular cylinder with isothermal and rough surface in cross flow by CFD numerical codes when compared with CFD factory original wall functions. In addition, as the cylinder temperature level of experiment was not known, the present authors were not able to calibrate model constants with accuracy. Another conclusion is that open source codes like OF1.6-ext may have different versions available in public software repositories so the user must perform a thoroughly research to find the version and setup that will bring most accurate results for his application. Therefore, the validation of the heat transfer from rough surfaces - such those found in airfoil icing - and related wall function modeling, is a requirement for any CFD numerical tool that is intended to be applied in in-flight aircraft icing or ice protection simulations by the industry.

REFERENCES

[1] Stefanini, L. M., Silvares, O. M., Silva, G. A. L., and Zerbini, E. J. G. J., 2008. "Boundary-layers integral analysis - airfoil icing". In AIAA Paper 2008-0474, Aerospace Sciences Meeting and Exhibit, 46th, 2008, Reno, American Institute of Aeronautics and Astronautics.

[2] Makkonen, L., 1985. "Heat transfer and icing of a rough cylinder". *Cold Regions Science and Technology,* **10**, pp. 105–116.

[3] Poinsatte, P. E., 1989. "Heat transfer measurements from a NACA 0012 airfoil in flight and in the nasa lewis icing research tunnel". Master's thesis, The University of Toledo, Toledo, Ohio, EUA, Agosto.

[4] Bragg, M. B., Cummings, S. L., and Henze, C. M., 1996. "Boundary-layer and heat transfer measurements on an airfoil with simulated ice roughness". In AIAA Paper 1996-0866, Aerospace Sciences and Meeting, 34., 1996, Reno, America Institute of Aeronautics and Astronautics, pp. 1–16.

[5] Fortin, G., 2001. "Simulation de l'accretion de glace sur un obstacle bidimensionnel par la méthode des bissectrices et par la modelisation des ruisselets et des gouttes de surface". PhD thesis, Universite du Quebec a Chicoutimi, Quebec, Canada.

[6] Stefanini, L. M., Silvares, O. M., Silva, G. A. L., and Zerbini, E. J. G. J., 2007. "Convective heat transfer effects in airfoil icing". In Procedings of COBEM 2007, 19th International Congress of Mechanical Engineering, Brazilian Society of Mechanical Sciences and Engineering.

[7] Stefanini, L. M., 2009. "Efeitos da camada limite térmica na formação de gelo em aerofólios de uso aeronáutico". Master's thesis, Escola Politécnica da Universidade de São Paulo, São Paulo, Brasil, Maio.

[8] Achenbach, E., 1977. "The effect of surface roughness on the heat transfer from a circular cylinder to the cross flow of air". *Int. J. Heat and Mass Transfer,* **20**, pp. 359–69.

[9] Wright, W. B., 1995. *User Manual for the Improved NASA Lewis Ice Accretion Code LEWICE 1.6*. National Aeronautics and Space Administration, Cleveland, May. 95 p. (Contractor Report, 198355).

[10] Guffond, D., and Brunet, L., 1988. *Validation du programme bidimensionnel de capitation*. Office National D'Études et de Recherches Aérospatiales, Chátillon Cedex, France. (Rapport Technique , RP 20/5146 SY).

[11] Gent, R. W., Dart, N. P., and Cansdale, J., 2000. "Aircraft icing". *Phil. Trans. Royal Society London A*(358), pp. 2873–2911.

[12] Pimenta, M. M., 1975. "The turbulent boundary layer: an experimental study of the transport of momentum and heat with the effects of roughness". PhD thesis, Stanford University, Stanford, May.

[13] Stefanini, L. M., Silvares, O. M., Silva, G. A. L., and Zerbini, E. J. G. J., 2010. "Heat transfer on iced cylinders". In AIAA Paper AIAA-2010-7672, AIAA Atmospheric and Space Environments Conference, 1., 2010, Toronto, America Institute of Aeronautics and Astronautics.

[14] Owen, P. R., and Thomson, W. R., 1963. "Heat transfer accross rough surfaces". *J. Fluid Mech.,* **15**(3), pp. 321–334.

[15] Cebeci, T., 1989. "Calculation of flow over iced airfoils". *Journal of Aircraft,* **27**(7), Junho, pp. 853–61.

[16] Shin, J., Chen, H. H., and Cebeci, T., 1992. *A Turbulence Model for Iced Airfoils and Its Validation*. National Aeronautics and Space Administration, Washington, January. 18 p. (Technical Memorandum, 105373).

[17] Cebeci, T., Chen, H., Kaups, K., Schimke, S., and Shin, J., 1992. *Analysis of Iced Wings*. National Aeronautics and Space Administration, Washington, January. 16 p. (Technical Memorandum, 105773).

[18] Chakravarthy, S., Peroomian, O., Goldberg, U., and Palaniswamy, S., 1998. "The cfd++ computational fluid dynamics software suite". In SAE Paper 985564, World Aviation Congress & Exposition, Anaheim, 1998, Society of Automotive Engineers.

[19] Vassberg, J. C., Mani, E. N. T. M., Rider, B., Zickuhr, T., Levy, D. W., Brodersen, O. P., Eisfeld, B., Crippa, S., Wahls, R. A., Morrison, J. H., Mavriplis, D. J., and Murayama, M., 2010. "Summary of the fourth aiaa cfd drag prediction workshop". In AIAA Paper AIAA Paper 2010-4547, 28th AIAA Applied Aerodynamics Conference, Chicago, 2010, America Institute of Aeronautics and Astronautics.

[20] Domingos, R. H., Pustelnik, M., Trapp, L. G., Silva, G. A. L., Campo, W., and Santos, L. C. C., 2007. "Development of an engine anti-ice protection system using experimental and numerical approaches". In SAE paper 2007-01-3355, SAE Aircraft and Engine Icing International Conference, 2007, Seville, Society of Automotive Engineers.

[21] Weller, H., Tabor, G., Jasak, H., and Fureby, C., 1998. "A tensorial approach to computational continuum mechanics using object orientated techniques". *Computers in Physics,* **12**(6), pp. 620 – 631.

[22] Jasak, H., 1996. "Error analysis and estimation in the finite volume method with applications to fluid flows". PhD thesis, Imperial College, University of London.

[23] Kinzel, M. P., Noack, R. W., Sarofeen, C. M., Boger, D. A., and Miller, S. T., 2010. "An openfoam implementation of ice accretion for rotorcraft". 5th OpenFOAM Workshop, Chalmers University of Technology. http://www.openfoamworkshop.org.

[24] Kays, W. M., and Crawford, M. E., 1993. *Convective heat and mass transfer*, 3rd ed. McGraw-Hill, New York.

[25] Moffat, R., and Kays, W., 1984. “A review of turbulent-boundary-layer heat transfer research at Stanford, 1958-1983”. Vol. 16 of *Advances in Heat Transfer*. Elsevier, pp. 241 – 365.

[26] Yaglom, A. M., and Kader, B. A., 1974. “Heat and mass transfer between a rough wall and turbulent fluid flow at high Reynolds and Peclet numbers”. *Journal of Fluid Mechanics,* **62**(3), pp. 601–623.

[27] Tapia, X. P., 2009. “Modelling of wind flow over complex terrain using OpenFoam”. Master’s thesis, University of Gavle.

[28] Cebeci, T., 2004. *Analysis of turbulent flows*, 2 ed. Elsevier, Amsterdam.

[29] Achenbach, E., 1971. “Influence of surface roughness on the cross-flow around a circular cylinder”. *J. Fluid Mechanics,* **46**(2), pp. 321–335.

[30] Henderson, A., 2007. *ParaView Guide, A Parallel Visualization Application.* Kitware Inc.

CONTACT INFORMATION

Dr. Guilherme Araujo Lima da Silva, consultant, ATS4i - Aero-Thermal Solutions for Industry, Rua Cotoxo, 611, cj. 63, São Paulo, SP, Brazil, 05021-000, email gasilva@ats4i.com.br, site www.ats4i.com.br.

Prof. Dr. Marcos de Mattos Pimenta, Mechanical Engineering Department, Escola Politécnica, University of São Paulo, Av. Mello Moraes, 2231, São Paulo, SP, Brazil, 05508-970, email mpimenta@usp.br

ACKNOWLEDGMENTS

The first author and second authors would like to thanks to Francisco Domingues Alves de Sousa from Aero-Thermal Solutions for Industry by his support and advices.

The authors would like to acknowledge Prof. Dr. Otávio de Mattos Silvares, Prof. Dr. Euryale J. G. de Jesus Zerbini and Luciano Martinez Stefanini, from Icing Research Group of University of Sao Paulo, by all support and data exchanged regarding previous work on integral analysis of heat transfer around rough cylinders.

In addition, the authors would like to acknowledge Dr. Uriel Goldberg, Dr. Sukumar Chakravarthy and Dr. Nili Bachchan from Metacomp Technologies, California, USA, for all the efficient support and inspiration.

Last but not least, the first author acknowledge the open forums www.cfd-online.com and groups.google.com/group/samba-openfoam by the patience with new users and the high quality information freely available on-line about CFD.

APPENDIX A: WALL FUNCTIONS SOURCE CODE AVAILABILITY

As it may be of public interest, the wall functions source code developed for OpenFOAM (compatible with versions OF1.6-ext and OF1.7.1) as well as a test case related to this work will be freely available on the INTERNET at repository - code.google.com/p/ats4iopentools/ or at Brazilian fluid engineers discussion group - groups.google.com/group/samba-devel.

APPENDIX B: NOTICES

CFD++ is a trademark of Metacomp Technologies, CA, USA. OpenFoam is a trademark of OpenCFD, UK. OF1.6-ext is an independent community-driven release of OpenFOAM by Extend Project, www.extend-project.de. This is a research paper made by engineers who use CFD tools funded by own resources of ATS_4i - Aerothermal Solutions for Industry. The present authors development are not endorsed or approved by any CFD supplying companies.