Modeling Boundary Layer Ingestion at the Conceptual Level

Andrew Turnbull, Hugo Jouan, Panagiotis Giannakakis, Askin T. Isikveren
Andy.turnbull@safrangroup.com
Dept of Energy and Propulsion
Safran Tech, SAFRAN S.A.
78772 Magny les Hameaux
France

ABSTRACT

Assessing the impact of Boundary Layer Ingestion (BLI) on a conceptual-level design requires a significant reduction in setup time and improved computational throughput in order to make it attractive over the current norm of being dependent on a surrogate model using high-fidelity CFD. This paper describes the implementation of a novel, geometrically-based method for determining the boundary layer characteristics at the engine interface plane for a fraction of the time typically invested. The method, named Propulsion Aerodynamic Integration Link (PAIL), links aircraft sizing with aerodynamic analysis in order to determine the impact of BLI on the engine performance and aircraft sizing. The method uses the concept of using a potential-flow, or non-viscous, flow solution around a parametric geometry model in order to determine which streamlines enter the engine. The inviscid streamlines that trace a 3D path are then used as edge Mach numbers for a 1D boundary layer routine. Averaging the boundary layer characteristic(s) into a distortion map allows for assessment of the total pressure losses, and importantly, the inlet momentum reduction potential of BLI. This process currently takes a few person-days to complete compared to using a viscous, powered CFD modelling technique that can take several person-weeks to arrive at a slightly more accurate flow estimate. In this report, the PAIL tool modelled the MIT D8 configuration, and was found to produce encouraging results as indicated by capturing the inlet energy deficit well.

Keywords: Boundary Layer Ingestion, potential flow, conceptual design, double-bubble, D8

NOMENCLATURE

AOA	Angle-of-Attack
APD™	Aircraft Preliminary Design, product from PACE®
BLI	Boundary Layer Ingestion
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
MIT	Massachusetts Institute of Technology
PAIL	Propulsion Aerodynamic Integration Link
PROOSIS™	Propulsion Object-Oriented Simulation Software
TnW	Tube and Wing Architecture

Symbols

C_f	Local Skin Friction Coefficient
C_p	Coefficient of Pressure
C_{Pk}	Mechanical Flow Power Coefficient, $C_{Pk} = P_K/q_\infty V_\infty S_{ref}$
F	Non-dimensional Mass Entrainment Rate
H	Compressible Shape Factor
$H_{\Delta-\Delta^*}$	Shape Factor Associated with Entrainment Rate
H_i	Equivalent Incompressible Shape Factor
H_{tr}	Transformed Shape Factor

M_e	Mach Number at outer edge of Boundary Layer
n_p	Power Factor ($H_i = (2+n_p)/n_p$)
Re	Reynolds number
S_{ref}	Wing Reference Area
T_e	Static Temperature at outer edge of Boundary Layer
T_0	Total Temperature of External Flow
u	Axial Velocity in the Boundary Layer
u_e	Edge Velocity
U_S	Velocity Parameter at the Wall
U_{tip}	Tangential Velocity of the Tip of the Fan
V_∞	Freestream velocity
X	Transformed Longitudinal Coordinate
y	y-coordinate, perpendicular to surface
δ	Boundary Layer Height
κ_{inl}	Shear Layer Kinetic Energy Defect
θ_i	Transformed or Incompressible Momentum Thickness
ρ	Air Density

1.0 INTRODUCTION

The current trend in transport aircraft design is to either install a pair of engines under and in front of the wings or installed on the upper rear fuselage. As such, except for secondary effects, the engine can for all intents and purposes be designed wholly independently from the airframe and vice versa. It is only during later stages, during preliminary and detailed design phases, where the engine and airframe establish an interface in order to minimize detrimental installation effects. As a result, the disciplines of turbomachinery design and aircraft design don't maintain a link in terms of vocabulary and methods except for late stages of design where the capacity to optimize is limited. This has led to a number of technologies that have not reached their potential due to lack of cross-discipline knowledge and a technical language barrier. One such orphan technological concept is BLI. Simply put, it is the practice of placing the engine inlet in a locale where it can "ingest" the slow-moving "boundary layer" air close to the airframe. This practice has two primary benefits: one of which is to increase propulsive efficiency by reducing the momentum drag of the engine streamtube; and second, the lower momentum inlet air allows for a lower exit velocity (or lower massflow) which reduces mixing and shearing losses in the engine/airframe wake. These and other secondary effects have the estimated potential to reduce fuel burn by 4-10% as shown in a summary constructed from Gladin [1] and more recent studies [2] [3] (see Figure 1). These studies estimate the benefit of BLI as compared to a non-BLI baseline and represent both hybrid-wing-body concepts and TnW concepts. They all have purely mechanical powertrains.

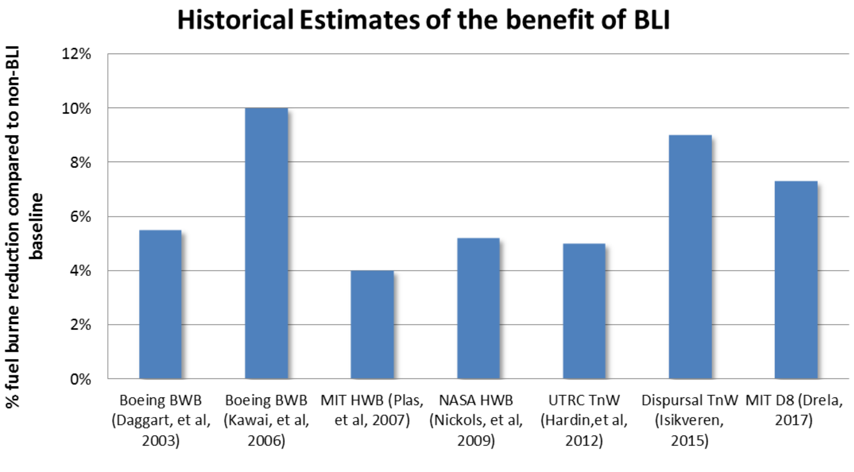

Figure 1 - Previous BLI Studies with estimates of the BLI benefit

1.1 State-of-the-Art Approach

The current industry practice for evaluating concepts that use BLI is to develop viscous, powered CFD models to assess a global benefit. This approach has the advantage of being theoretically consistent and inherently accurate without delving into the individual, often mixed effects of propulsive and aerodynamic sources of power. However, this approach has three drawbacks. The first is the lack of transparency and clear boundaries between airframe and engine control volumes that delineate between propulsion and aerodynamic forces. With only a global benefit quantified, it can be difficult to identify opportunities for optimization, particularly when those opportunities involve interactions between propulsion and aerodynamics. The second drawback is the maturity the design must attain in order to be modeled in a viscous, powered CFD domain. By the time it can be modeled by such a method, a significant number of design decisions have already been made, limiting the potential design envelope without sufficient design space exploration. The details of the inlet design, the forebody shape, the geometric relationship of the engine to the fuselage or wing and even the basic fan parameters (diameter, pressure ratios, massflow, nozzle area ratios) are defined to a very advanced stage in order to properly construct such a CFD simulation. The last drawback is the heavy computational and wall clock cost of a simulation. For instance, the steps to build a CFD simulation suitable for these studies are the aforementioned mature geometry definitions with both a surface and volume grid which often requires a grid sensitivity study, incorporation of a surrogate engine model of sufficient fidelity, and finally the CPU time necessary to run the simulation. While some admirable efforts at automating this process have been and continue to be developed [4] [5], the process is often iterative and manual in order to obtain a clean flow solution. Furthermore, each of those solutions must be repeated over a range of Mach, altitudes, operating lift coefficients, and massflow data points in order to characterize the performance over a good proportion of the aircraft operating envelope. These can combine to require 40 person-weeks to evaluate a single configuration of a larger design study. On top of that, in order to optimize a concept for a given mission, dozens of different configurations need to be evaluated, dependent on the number of variables in the optimization. The effort can easily spiral out of control.

1.2 Aim of this Technical Paper

As such, the goal of this tool development, as described in this paper, is to greatly simplify the required computational cost of estimating the impact of propulsive-aerodynamic interaction on conceptual designs. Dubbed “Propulsion Aerodynamic Integration Link”, or PAIL, it is a method that provides a BLI interface between the conceptual aircraft sizing tool PACE APD™ [6] with the engine cycle modeling code, PROOSIS™ [7]. As stated previously, it is necessary to characterize the viscous flowfield entering an inlet with a varying pressure-ratio engine boundary condition in order to assess the impact of BLI. Other attempts use either scaled boundary layer characteristics from similar configurations [8] or have a dedicated, parallel CFD model development path that can provide an estimate of the total energy deficit ingested by the engine to the conceptual toolset [9]. This paper describes a third option that grounds the analysis in conceptual-level geometry characteristics while avoiding the vast majority of gridding and geometrical immaturity issues associated with performing a viscous, powered CFD assessment. It will demonstrate the potential of the code by comparing it to experimental results from obtained MIT double-bubble or D8 experiments, as conducted in 2011, and subsequently summarized in Reference [10].

2.0 METHOD DESCRIPTION

The starting point of the analysis is a preliminary or surrogate aircraft concept (for example, a non-BLI version of the aircraft) as modeled in PACE APD™, or similar conceptual aircraft design tools [11] [12] [13]). Once the mission and initial geometry are defined by the first round of sizing, the basic geometry parameters are replicated in openVSP [14], namely, Step (1) as depicted in Figure 2. The openVSP tool generates a watertight surface mesh that can be directly linked to downstream aerodynamic analysis, which in this case is Flightstream™ [15], a potential-flow viscosity solver, Step (2). The Flightstream™ [15] flow solution defines streamlines that enter the inlet from the potential flow solution. Those streamlines are used to determine the edge Mach number of a 1D estimate of the boundary layer height and shape using an integral momentum method. The boundary layer profile at the inlet is then averaged over the inlet to make the engine pressure recovery and distortion maps which are passed to PROOSIS™, Steps (3) and (4), to define the inlet conditions at each flight point. The "raison d'etre" of PAIL is to significantly reduce the effort in performing an assessment of the impact of BLI, particularly the inlet conditions, on an arbitrary configuration over its flight envelope from months by using viscous, powered CFD models to about a man-day. The process often takes a few days in its current form, but planned development should reduce computation time to the stated goal.

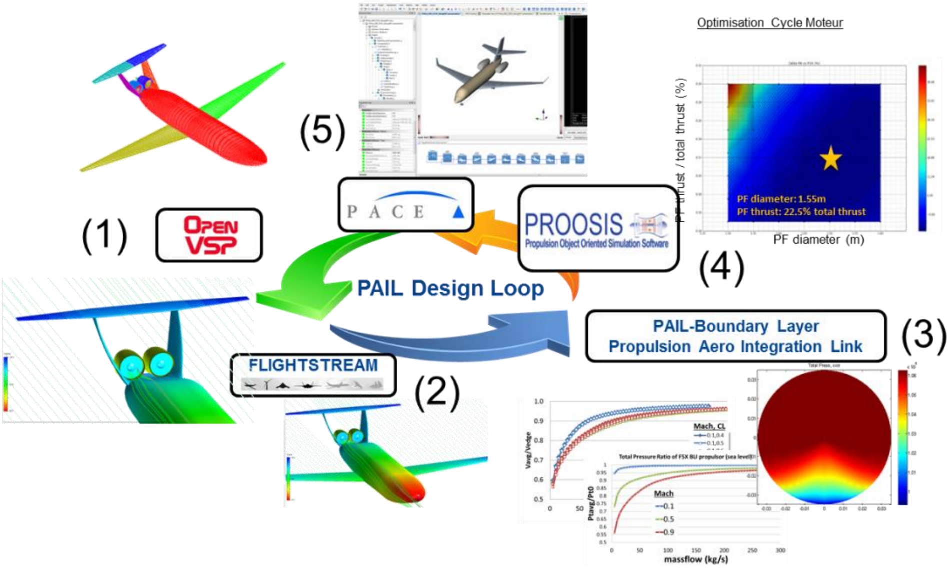

Figure 2 - Propulsion Aerodynamic Integration Link (PAIL) tool scheme

2.1 openVSP

The openVSP [14] software is a free parametric geometry tool, developed by NASA, centered around rapidly and accurately developing aircraft geometries. On top of being an avenue for brainstorming and/or visualizing initial layouts of an aircraft configuration, it also provides a foundation for downstream analysis capability. Geometry using openVSP must be generated manually from parameters provided by PACE APD™, but the downstream tool chain does not require any further manual manipulation. The goal of openVSP in this process is to generate a closed surface mesh, such as a .STL file, including watertight intersections between adjoining components that can be exported directly into the aerodynamic analysis module.

2.2 Flightstream™

Flightstream™ is a potential flow aerodynamic solver developed by Research In Flight [16]. It is a non-viscous, or potential flow solver that requires a surface grid from a geometry source but does not require a volume grid because the flow solution is solved analytically by converging influence coefficients of the vorticity of each panel [16]. As a result of the method, it is a computationally lightweight way to determine the global flow solution around the aircraft even though the viscous effects are not directly determined. It has a direct link with openVSP and runs very quickly on a standard PC with a single flow solution taking between about 1 min to 1 hour, dependent upon the density of the surface grid. An extensive series of validation exercises have been undertaken by Flightstream™ and these are given in [15] [17].

For the purposes of propulsion-aero integration the engine interface plane has a velocity boundary condition that takes into account the impact of inlet or exhaust flow on the airframe. Additionally, it is possible to define a streamline that enters the boundary condition and trace it upstream to define the inviscid velocity distribution of streamlines entering the engine. Therefore, the goal of the Flightstream™ part of the PAIL tool chain is to define the velocity along a streamline that enters the engine. This streamline (or streamlines) is set the streamline Mach number as the edge velocity for boundary layer development. The current method can use 1-3 streamlines from the flow solution to determine the characteristics of the engine inlet flow. Future work will incorporate the boundary layer characteristics directly inside the potential flow solution and save several steps while increasing the fidelity of the method.

It should be noted that this approach has two drawbacks. The first is that the compressibility correction inside the flow solution is a simple Prandtl-Glauert correction [18] and, as such, it cannot predict the onset of a shock. The second drawback is that there currently is no reliable way to predict flow separation. The analytical flowfield does not predict separation by definition, and generally, 1D boundary layer codes are not considered to be reliable in predicting separation along a single streamline. Irrespective of these limitations, in the domain of conceptual aircraft design, one can make the assumption that properly fashioned inlet and fuselage geometries will not cause either of those phenomena to occur just upstream of the inlet. So their absence in the lower-fidelity analysis is a fortunate coincidence that alleviates some of the need for a mature geometrical definition. A significant amount of time can be (and has been) lost by trying to mature an inlet geometry to the point where a higher-fidelity code will not indicate separation or the formation of shocks.

2.3 1D Boundary Layer Method

Streamlines defined by the Flightstream™ flow solution serve to define edge Mach numbers, which is then used in an integral-based 1D boundary layer method to determine the displacement and momentum thicknesses at the inlet boundary conditions. The 1D boundary layer method chosen for this version of the tool is from Standen [19] and includes the ability to model compressible flow under an arbitrary pressure gradient. The method assumes fully turbulent flow and has a C_f-based criteria for determining flow separation. Based upon this approach the transformed version of the integral momentum equation is

$$\frac{d\theta_i}{dx}+\frac{\theta_i}{M_e}\frac{dM_e}{dx}\left(2+H_{tr}\right)=\frac{C_f}{2}\left(\frac{T_e}{T_o}\right)^3 \tag{1}$$

and the transformed shape factor equation is presented as

$$\frac{dH_i}{dx}=-\frac{\left(H_i-0.7\right)^{3.715}}{4.17}\left[\frac{F}{\theta_i}\left(\frac{dX}{dx}\right)-\frac{H_{\Delta-\Delta*}}{M_e}\left(\frac{dM_e}{dx}\right)-\frac{H_{\Delta-\Delta*}}{\theta_i}\left(\frac{d\theta_i}{dx}\right)\right] \tag{2}$$

These two equations must be simultaneously solved as ordinary differential equations to determine the displacement and momentum thicknesses. The method is primarily dependent upon the edge Mach along the streamline and runs for less than 1 sec. when using Matlab™ on a personal computer. This combination of a 3D flow solution and a 1D boundary layer method is sufficient to capture the 3D characteristics of the flow entering the inlet and for establishing the essential character of the boundary layer at the inlet while significantly increasing throughput compared to not only the pure computational cost of a full-CFD model, but also the large majority of the gridding necessary for such CFD models. This approach was inspired by a similar approach proposed and implemented by AVID LLC for a NASA NRA solicitation for improved propulsion-aerodynamic interaction on a hybrid wing-body [20].

Once the Standen method calculates the momentum and displacement thicknesses, PAIL then requires the assumption of a velocity profile in order to establish the full defect within the boundary layer. Currently, the code offers a choice between a power law profile

$$\frac{u}{u_e} = \left(\frac{y}{\delta}\right)^{1/n_p} \tag{3}$$

and a Coles profile [21]

$$\frac{u}{u_e} = U_S + \frac{(1-U_S)}{2}\left[1-\cos\left(\pi\frac{y}{\delta}\right)\right] \tag{4}$$

The tool has the capability to model either axisymmetric configurations or those with semi-buried engines. The boundary layer flow can be oriented along any radius around the inlet and can even include a 3D correction [22, p. 88] to take into account, at least in a qualitative sense, the impact of the nearby higher pressure flow that tends to intrude on the borders of the stratified boundary layers. For this validation study, due to the relatively low total pressure of the inlet flow, as well as the asymmetric nature of the D8 configuration, this effect is small and the flow was left as a stratified boundary layer.

The goal of the boundary layer portion of the PAIL tool chain is to return the total pressure losses and the momentum deficit of the inlet flow while ingesting a boundary layer. These estimates of total pressure and momentum losses are a function of inlet massflow, the freestream conditions (Mach, altitude), and the AOA of the upstream body. Normally, the AOA can be directly tied to the aircraft operating lift coefficient, which is usually used as a demarcation in the engine model and aircraft mission modeler (for example, using PACE APD™). The dependency on aircraft operating lift coefficient is due to the different path the inlet streamlines take as the lift coefficient changes. The benefit of BLI is derived from the reduced momentum of the incoming flow, but the cost comes as a result of lower total pressure.

2.4 PROOSIS™

The data-tables generated by the boundary layer solver of PAIL are subsequently fed into the engine cycle modelling tool called PROOSIS™ [7]. PROOSIS™ is a commercial software developed by Empresarios Agrupados Internacional S.A and a consortium of European universities, research institutes and companies. It is a flexible, object-oriented tool that is capable of modelling any propulsive system configuration. For studying architectures featuring integrated BLI devices, a new engine intake model needs to be created. This intake model encompasses the data-tables generated by PAIL and calculates the equivalent ram drag reduction and inlet pressure losses included in the ingested stream-tube, for any given flight conditions, aircraft operating lift coefficient and engine mass flow. The new intake model then provides a revised total pressure and inlet velocity to the engine cycle and the engine can then be sized in order to satisfy the requirements of multiple operating points and its performance can be predicted at any point of the flight envelope.

2.5 PACE APD™

The engine model from PROOSIS™ is then linked to the aircraft configuration sizing and mission modeler, PACE APD™ [6]. PACE APD™ performs aircraft sizing using an engine model that incorporates the impact of BLI derived from the PAIL tool chain. While "flying" the mission profile, PACE APD™ requests the fuel flow of the engine at a given flight point from the engine model. Since the fuel flow is also dependent upon the current operating lift coefficient, PACE has been modified to provide the lift coefficient to PROOSIS along with the altitude, Mach, and required thrust it nominally provided. While the overall process involves significant links between tools and creation of new ones, the link with PACE APD™ is relatively straightforward and requires no major modifications.

3.0 MIT D8 COMPARISON

In 2008, NASA proposed a study [23] to industry with the goal of determining if significant fuel burn (-70% relative to year 2000 baseline) and emissions reductions were possible by year 2030-2035. The respondents proposed advanced airframe and propulsion concepts including an account of which corresponding enabling technologies that should be declared as funding priorities. One such concept, proposed by the MIT, centered a good proportion of fuel savings based upon the application of BLI, thus resulting in the so-called D8 concept [24]. The most distinguishing feature of D8 is the morphological evolution of a short-to-middle range TnW airliner accommodating BLI. The entire configuration was oriented towards extracting the maximum advantages of BLI with conventional turbofans. Using fully viscous, powered CFD models, MIT determined that the conventional turbofans ingest approximately 17% of the fuselage energy deficit from viscous sources [2] and uses them to reduce wake losses and to increase overall propulsion efficiency.

3.1 MIT D8 Wind Tunnel Experiment and Pressure Rake Map

As a follow-on to the Phase 1 N+3 report [25], which showed a significant potential of the D8 to reduce fuel burn for a short-to-medium range aircraft, NASA contracted with them to perform a series of wind-tunnel experiments using the NASA Langley 14 x 22 wind-tunnel in order to better quantify the propulsive-aerodynamic benefits of BLI. While validating the totality of the global benefits of the D8 configuration was not part of the original funding, it provided a wealth of opportunities to refine and/or develop methodologies, as well as provide validation data associated with BLI as modelled by conceptual aircraft design methods, fully viscous, powered CFD models, and finally wind-tunnel experiments. It is the most complete publically disseminated treatment of BLI to date. In one of the many theses spawned by the difficult problem of properly assessing BLI in a wind-tunnel, Lieu [10] details the D8 wind-tunnel model geometry together with the pressure rake geometry used to assess the characteristics of the inlet flow. Figure 3 shows the test setup with a total pressure rake that is larger in diameter than the engine inlet and approximately one diameter upstream from the engine inlet. It is this test and results against which PAIL is compared in the following section.

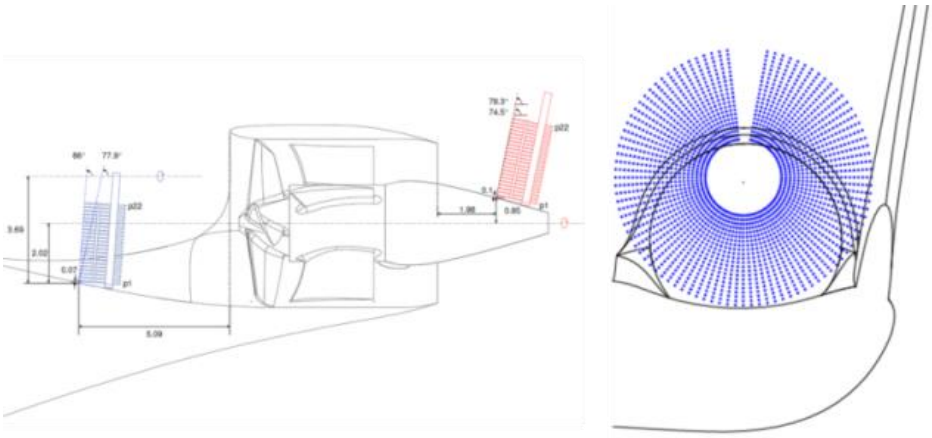

Figure 3 - Total pressure rake geometry for evaluating the integrated engine inlet of the D8 configuration (Lieu [10])

3.2 PAIL Results for MIT D8

The first step of the PAIL process is to replicate the 3D geometry of the wind-tunnel model that MIT and NASA used in the 14 ft x 22 ft wind-tunnel at NASA Langley. The geometry of the model [10] is faithfully replicated in an openVSP model shown in Figure 4.

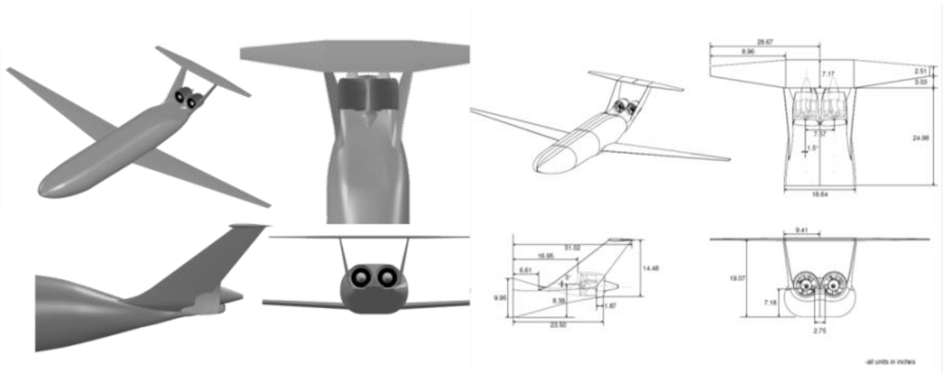

Figure 4 - D8 wind-tunnel model representation using openVSP (left) and a schematic of the wind-tunnel model given by Lieu [10] (right)

The geometry was exported as a .STL mesh and directly into Flightstream™. The resulting Flightstream™ model was run at wind-tunnel conditions (31.3 m/s tunnel speed, Re = 5.5×10^5, AOA = 2.0° and a 28.0 m/s inlet velocity boundary condition). The streamlines that were chosen along which the boundary layer characteristics were derived are shown in Figure 5. The latter velocity boundary condition of 28.0 m/s was estimated from the ΔC_p plot shown in Figure 3-3(b) of reference [10] using an average of $\Delta C_p = 0.2$. By varying the inlet velocity boundary condition by -20% to +20%, it was determined that the impact on the boundary layer height/shape at the inlet is minimal. Even if the velocity inlet condition is not quite correct, the impact on the edge Mach number, and therefore, the impact on the boundary layer height estimate only covers a short distance. In fact, comparing figures from Appendix C.2.3 in Lieu [10], the average reduction in δ (with little apparent change in shape) between a $U_{tip}/V_\infty = 3.20$ and $U_{tip}/V_\infty = 2.64$ is only 4%.

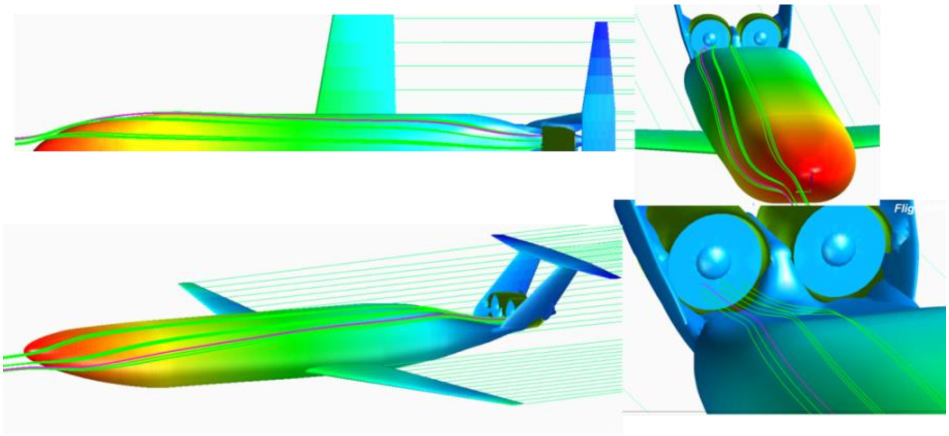

Figure 5 - Streamlines used to generate the distortion map from Flightstream model

The resulting boundary layer estimate, as taken from the middle outside streamline, is very close to the value of the boundary layer published by MIT. From MIT's CFD, due to the upstream swirl from the fan and the shape of the upstream fuselage, there is a difference between the left side of the fan face and right side in terms of δ so close inspection of Fig. 3-3(a) in Lieu [10] yielded a different δ, for the left side and right side. In comparison to the PAIL output, as shown in Figure 6, the difference in δ resulted in a +8.1% error in height on the higher side and -1.3% on the lower side. In the center, it was +4.9% total error in boundary layer height.

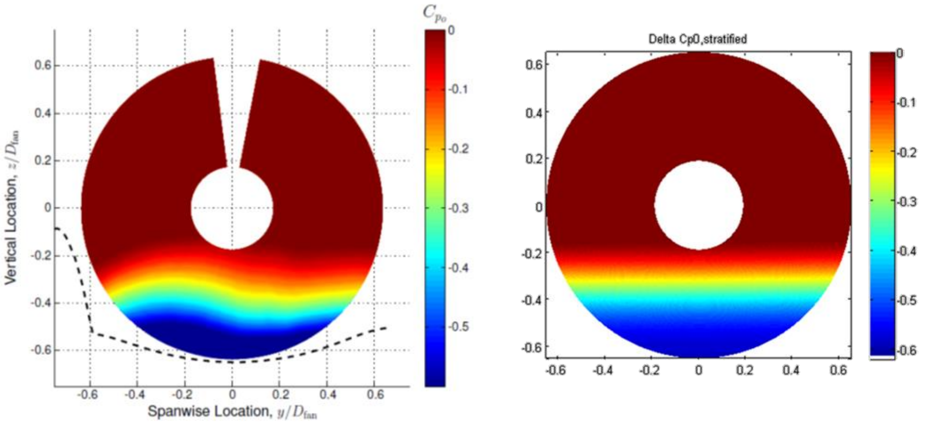

Figure 6 - Comparison of total pressure losses presented in Figure 3.3 by Lieu [10] (left) and the PAIL tool equivalent model (right)

Additionally, the boundary layer shapes, even though PAIL uses an analytical Coles profile, appear to correspond well as shown in Figure 7. Using Equation (5), an estimate of the two kinetic energy defects, one from PAIL and other from Figure 3.3a of Lieu (Figure 7) resulted in a 2.3% difference.

$$\mathcal{K}(x) = \iint \frac{1}{2}\left(u_e^2 - u^2\right)\rho u \, dS(x) \tag{5}$$

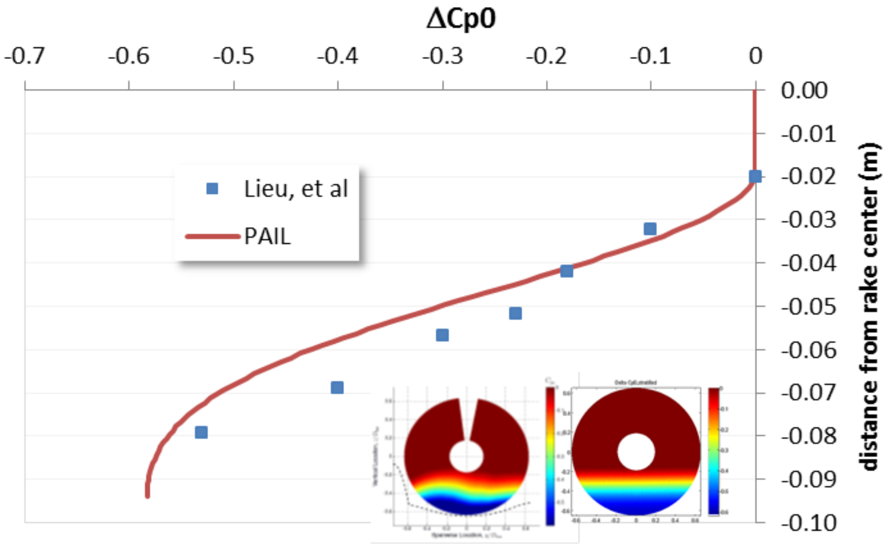

Figure 7 - Comparison of MIT D8 wind-tunnel rake and PAIL in terms of ΔC_{p0} on the engine centerline

Finally, Lieu calculates the mechanical flow power (Equations 6,7) of the flow going through the rake, both upstream and downstream of the propulsor. The PAIL results are also from the pressure rake location and size and not the engine inlet itself therefore it is appropriate to compare values of C_{Pk}. Lieu defines the C_{Pk} as

$$C_{Pk} = \frac{\mathrm{P_k}}{\mathrm{q_\infty V_\infty S_{ref}}} \tag{6}$$

Where the total C_{Pk} is the sum of the inlet and exit values:

$$C_{Pk} = C_{Pk,inlet} + C_{Pk,exit} \tag{7}$$

Lieu quotes the $C_{Pk,inlet}$ of the left propulsor as 0.0015, whereas, PAIL calculates $C_{Pk,\ inlet}$ as 0.00121, around a 17% error (dependent on rounding).

Given the fact that the kinetic energy defect and boundary layer height correspond well, the larger difference in C_{Pk} doesn't have an obvious source. It is postulated that some of the discrepancy can be traced to a mis-match in defining the wind tunnel operating environment. PAIL, in general, and the C_{Pk} calculation, in particular, are sensitive to ambient density and altitude as well as non-standard temperatures. Since this study is matching wind tunnel data, knowledge of the tunnel conditions is important in order to justifiably validate a computational comparison. A clean-sheet design effort using PAIL would simply use standard temperature and altitude but for validation, there is a combination of a few known variables and a few assumptions that can combine to better model the wind tunnel test in PAIL. The combination of boundary layer height, kinetic energy defect, and C_{Pk} require good estimates of edge velocity, ambient and wall temperatures and the boundary layer shape. From Lieu [10], the Re, tunnel velocity and chord are known, therefore the kinematic viscosity is known. With a value of $1.55\text{x}10^{-5}$ m^2/s, the kinematic viscosity was not at sea level standard conditions but no value for the density was given. Since NASA Langley's 14x22 wind tunnel is physically a few feet above sea level, a choice exists as to whether the temperature is standard and it's a straightforward density altitude correction or whether there is a non-standard temperature and a lower density altitude. In the first runs of PAIL with a standard temperature density altitude correction, the boundary layer height and kinetic energy defect matched even closer than the final values but the C_{Pk} had an error greater than 30%. While the tunnel temperature is still not known, the wind tunnel tests were conducted in the summer and from the author's recollection, it was hot in the tunnel during the first test. Therefore, it was reasonable to believe that the tunnel and model were above standard temperature. A survey of combination of non-standard temperatures and density altitudes that correspond to the known kinematic viscosity was conducted in order to find the best match against all three test results (boundary layer height, kinetic energy defect and C_{Pk}).

The combination that worked best assumes a slightly lower density altitude (~1500 ft) than standard temperature density altitude, a +9° C deviation in standard temperature and a model about 10° C hotter than the tunnel. While these inputs are reasonable, they are nonetheless fits to available data and it's likely that the C_p is also higher than the wind tunnel results, possibly requiring a higher than reality temperature correction. There is also some uncertainty inherent in each chosen streamline with the current version of the tool chain that contributes to the error. Planned development work, as also mentioned in the following section, should result in a smaller uncertainty in streamline velocities and allow for even better matching of wind tunnel results.

3.3 Other Validation

In addition to the MIT D8 case study, there have been other validation cases, which are not presented in this technical paper due to proprietary restrictions, against which PAIL results have been compared. These cases were RANS CFD runs of working concepts using BLI. The comparisons have proved to be sufficiently accurate and there is on-going internal work planned to refine the method further, notably by including more streamlines in the construction of the inlet distortion map in order to capture more 3D effects (absent from these results as well) while improving separation prediction and significantly reducing person-hours and setup time.

4.0 CONCLUSIONS

PAIL demonstrates the ability to model, at an essentials-only level, a configuration like the MIT D8 concept in order to produce an estimate of the flow at the engine inlet. PAIL demonstrated sufficient accuracy in reproducing a distortion map corresponding to a particular cruise flight condition while taking a fraction of the setup and computational time required of an equivalent CFD modelling effort.

ACKNOWLEDGMENTS

The authors would like to thank Pierre-Alain Lambert and Clement Pornet of Safran Tech and Nicolas Tantot of Safran Aircraft Engines for their support and feedback on the development of PAIL.

REFERENCES

[1] J. Gladin, "A Sizing and Vehicle Matching Methodology for Boundary Layer Ingesting Propulsion Systems," PhD Dissertation, Georgia Institute of Technology, Atlanta, GA, 2015.

[2] D. K. Hall, A. C. Huang, A. Uranga, E. M. Greitzer, M. Drela and S. Sato, "Boundary Layer Ingestion Propulsion Benefit for Transport Aircraft," *Journal of Propulsion and Power,* 2017.

[3] A. Isikveren and et.al., "Distributed Propulsion and Ultra-High Bypass Rotor Study at Aircraft Level," *The Aeronautical Journal,* vol. 119, no. 1221, pp. 1321-1375, 2015.

[4] J. Gray and J. Martins, "Approach to Modeling Boundary Layer Ingestion using a Fully Coupled Propulsion-RANS Model," AIAA 2017-1753, Cleveland, OH, 2017.

[5] M. Wintzer, "Optimization and Adjoint-Based CFD for the Conceptual Design of Low Sonic Boom Aircraft," PhD dissertation, Stanford University, Stanford CA, 2012.

[6] PACE Aerospace Engineering and Information Technology GmbH, *Pacelab APD 3.0.0 Release Note*, 2012.

[7] Empresarios Agrupados Internacional S.A, "PROOSIS," [Online]. Available: http://www.proosis.com.

[8] J. R. Welstead and J. L. Felder, "Conceptual Design of a Single-Aisle Turboelectric Commercial Transport with Fuselage Boundary Layer Ingestion," 2016 AIAA SciTech Conference, San Diego, CA, 2016.

[9] E. Greitzer and et.al., "N+3 Aircraft Concept Designs and Trade Studies, Final Report Volume 2," NASA, 2010.

[10] M. Lieu, "Quantification of the Boundary Layer Ingestion Benefit for the D8-series Aircraft Using a Pressure-Rake System," MSc Thesis, Massachusetts Institue of Technology, 2015.

[11] L. McCullers, "Aircraft Configuration Optimization Including Optimzed Flight Profiles," N87-11743, Kentron International, Inc, Hampton, VA, 1987.

[12] R. De Filippo, "ACSYNT User's Guide," Northrop Aircraft, 1983.

[13] D. Simos, "Project Interactive Analysis and Optimization-PIANO Version 3.8," Lissys Limited, 1990-2001.

[14] NASA, "Featured NASA Software: OpenVSP," 2015-2016. [Online]. Available: software.nasa.gov/featuredsoftware/openvsp. [Accessed 18 august 2016].

[15] R. Hartfield and V. Ahuja, "Aerodynamic Loads over Aribitrary Bodies by Method of Integrated Circulation," *Journal of Aircraft,* vol. 53, no. 6, pp. 1719-1730, 2016.

[16] V. Ahuja, "Aerodynamic Loads over Arbitrary Bodies by Method of Integrated Circulation," PhD Thesis, Auburn University, 2013.

[17] E. D. Olson and C. W. Albertson, "Aircraft High-Lift Aerodynamic Analysis Using a Surface-Vorticity Solver," AIAA 2016-0779, Hampton, VA, 2016.

[18] H. Glauert, "The Effects of Compressibility on the Lift of an Aerofoil," *Proceedings of the Royal Society of London, Series A: Mathematical and Physical Sciences,* vol. 118, no. 779, pp. 113-119, 1928.

[19] N. Standen, "Calculation of Integral Parameters of a Compressible Turbulent Boundary Layer using a Concept of Mass Entrainement," MSc Thesis, McGill Unversity, Montreal, 1964.

[20] P. Gelhausen and et.al., "A Systems Analysis and Design Approach to the Hybrid Wing/Body Aircraft," Yorktown, VA, 2011.

[21] D. Coles, "The Law of the Wake in the Turbulent Boundary Layer," *Journal of Fluid Mechanics,* vol. 1, no. 2, pp. 191-226, 1956.

[22] D. L. Rodriguez, "A Multdisciplinary Optimization Method for Designing Boundary Layer Ingesting Inlets," PhD Thesis, Stanford University, 2001.

[23] NASA Fundamental Aeronautics Program, "Advanced Concept Studies for Subsonic and Supersonic Commercial Transports Entering Service in the 2030-2035 Period," NASA, 2008.

[24] M. Drela, "Development of the D8 Transport Configuration," AIAA Applied Aerodynamics Conference, AIAA 2011-3970, Cambridge, MA, 2011.

[25] E. Greitzer and et.al., "N+3 Aircraft Concept Designs and Trade Studies: Volume 1," Massachusetts Institute of Technology, Aurora Flight Sciences, Pratt & Whitney, 2010.