Discretized Miller approach to assess effects on boundary layer ingestion induced distortion

Esteban Valencia a,*, Víctor Hidalgo a, Devaiah Nalianda b, Laskaridis Panagiotis b, Riti Singh b

a Departamento de Ingeniería Mecánica, Facultad de Ingeniería Mecánica, Escuela Politécnica Nacional, Ladron de Guevara E11-253, Quito 17-01-2759, Ecuador
b School of Engineering, Cranfield University, Bedford MK43 0AL, United Kingdom

Received 19 November 2015; revised 27 June 2016; accepted 19 October 2016

KEYWORDS
BLI; Correlations; Distortion; Embedded propulsors; Empirical; Mean-line analysis; Turbomachinery

Abstract The performance of propulsion configurations with boundary layer ingestion (BLI) is affected to a large extent by the level of distortion in the inlet flow field. Through flow methods and parallel compressor have been used in the past to calculate the effects of this aerodynamic integration issue on the fan performance; however high-fidelity through flow methods are computationally expensive, which limits their use at preliminary design stage. On the other hand, parallel compressor has been developed to assess only circumferential distortion. This paper introduces a discretized semi-empirical performance method, which uses empirical correlations for blade and performance calculations. This tool discretizes the inlet region in radial and circumferential directions enabling the assessment of deterioration in fan performance caused by the combined effect of both distortion patterns. This paper initially studies the accuracy and suitability of the semi-empirical discretized method by comparing its predictions with CFD and experimental data for a baseline case working under distorted and undistorted conditions. Then a test case is examined, which corresponds to the propulsor fan of a distributed propulsion system with BLI. The results obtained from the validation study show a good agreement with the experimental and CFD results under design point conditions.

© 2016 Chinese Society of Aeronautics and Astronautics. Production and hosting by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

1. Introduction

The performance of axial fans under distorted conditions has been studied extensively in the past. Different approaches and tools such as through flow methods,1 semi-empirical correlations,2 and fan map based methods (parallel compressor3) have been utilized to assess their performance. It has been found that even though through flow methods such as streamline curvature4,5 and CFD can predict fan performance with...
higher accuracy than other methods, they also require more resources in terms of computational power and time. For boundary layer ingestion (BLI) systems, streamline curvature and parallel compressor have been combined in order to assess circumferential and radial distortion.\(^4\) In the case of CFD, this tool enables the assessment of the fan performance radially and circumferentially, and for this reason CFD has also been utilized for the assessment of BLI distortion problems. This approach has been preferred for cases where the detailed geometry is known and accuracy is paramount. In this study, CFD has been used to compare some results obtained under uniform conditions with the discretized Miller approach. Some experimental test-rigs have been set for the assessment of BLI type distortion; however testing engines or in this case fans are expensive and the cost increases with advanced engine technology and designs. Furthermore, the cost of the energy required for each run and the possibility of damaging the tested equipment increase the difficulty of carrying out these procedures. Jerez et al.\(^1\) experimentally studied the validation of a CFD model for a fan working under distortion. Redmond\(^6\) described a test-rig and experimental results for BLI type distortion; however in this latter case, the uncertainties achieved are large and it is difficult to assess the real benefits of BLI.

At the preliminary design stage when the detailed geometry of a system/component is still undefined and several configurations have to be tested, reducing requirements of computational resources becomes imperative. Due to excessive simulation times, these methods have also been found unsuitable in cases where full annular simulation is required and circumferential distortion is present (such as in boundary layer ingesting systems \(^7\) \textit{et al.}). These reasons, therefore, make methods such as the parallel compressor\(^8\) more attractive for preliminary design at design point. However, this method has its limitations, as it only enables to assess circumferential distortion. Hence it may be considered to have limited accuracy in the case of BLI systems where a combination of circumferential and radial distortion of flow is observed.

This paper addresses this limitation of the method and proposes a novel approach to overcome it. Using a semi-empirical approach, the proposed method essentially utilizes empirical
data for the blade and performance calculations. Further by discretizing the inlet region in circumferential and radial directions, the proposed approach takes into account the combined effects of the distortion patterns on fan performance calculation.

In order to verify the approach and ensure that the set of empirical correlations predicts the fan performance accurately, a series of validation test cases are undertaken. The first stage of the validation is undertaken against data available in the study of Osborn et al.\textsuperscript{10} for the NASA axial flow fan stage 53 utilizing experimental data and CFD under similar operating conditions. It commences with the assessment of fan performance at design point under various off-design operating conditions, assuming a uniform inlet flow field. The next stage of validation examines the performance of the fan under distorted conditions. As no published data is available for the performance of the NASA axial flow fan stage 53, the data available for the NASA axial flow fan stage 67\textsuperscript{1} is considered. The validation studies under uniform and distorted conditions are then followed by assessment of the fan performance assuming boundary layer ingestion, which is similar to conditions faced by a propulsor fan unit in a distributed propulsion system with BLI.\textsuperscript{11,12} The operating conditions and propulsion system configuration for this final phase of validation is adapted from a previous parametric optimization study,\textsuperscript{9} using the airframe design and cruise conditions of the NASA N3-X aircraft concept.\textsuperscript{11}

Hierarchy of distortion methods

Fig. 1 shows the distortion methods that have been utilized for the assessment of BLI induced distortion. To illustrate where they are implemented in the propulsion system analysis, the input data and propulsor modules have been included in this diagram. As observed in this figure, the parametric method which was utilized by Valencia et al.\textsuperscript{9} is the most basic approach where the fan performance characteristics are assumed. The other two methods assess the fan performance characteristics based on the fan face flow properties. This enables to relate the inlet flow characteristics with the propulsor performance.

2. Methodology

This section describes the methodology used to develop the discretized Miller approach. As the discretized Miller approach is based on the mean-line design and semi-empirical performance approaches, the first part of this section describes the conventional mean-line approach and the empirical correlations utilized for fan performance analysis, while the second part explains the discretized Miller approach. In order to carry out the fan performance calculation, a computer program (MATLAB platform) is developed.

The discretized Miller approach comprises several subroutines which follow a process as shown in Fig. 2. The input variables (in Fig. 2) are set by the inlet flow properties, fan geometric characteristics and fan operating conditions (as summarized in Table 1). The collected input data is used by the blade design subroutine, where the mean-line design approach allows defining the fan annular section and the air angles along the blade span.\textsuperscript{13}

The blade design module is not necessary if the geometry is known, as the air angles and other parameters can be defined. The incorporation of the blade design module enables us to make the method independent of the fan geometry for the cases where this is unknown. This is an important feature for preliminary design of distributed propulsion systems, as different propulsor arrangements and hence fan configurations need to be tested. However, it is important to note that the

![Fig. 1 Road map of distortion models.](image-url)
generation of the blade geometry following the method
described in the next section (free vortex design) is basic and
hence it will present discrepancy with the actual blade design.
This aspect will reduce the accuracy of the method when the
blade design module is utilized. The effects of using the actual
and designed geometry will be examined later.

2.1. Mean-line design method

The mean-line method is a design approach, which uses the
Euler equations and velocity triangles to calculate the air exit
angles. These angles and empirical correlations for blade
gеometrь and pressure losses allow us to calculate the fan per-
f ormance. The blade arrangement studied in this work com-
prised a rotor and stator. A schematic representation of the
fan stage is shown in Fig. 3.

For the calculation of the air exit angles, a free vortex con-
t ioni on and a constant axial velocity through the fan assembly are
assumed. These assumptions allow simplification of the model at an expense of loss in accuracy (Section 3.1.2). Since
the method to calculate the pitch to chord ratio shown in the
study of Saravanamuttoo et al.13 presents some discrepancies
with the actual pitch to chord ratio of the fan examined, this
parameter was assumed the same as the reported one in the
work of Osborn et al.10 for the validation studies. The blade
gеomet rу in the proposed method is determined using Carter’s
rule2 for deviation angle $\delta$. This approach was preferred over
the Miller’s14 definition due to its simplicity and better predic-
t ion capability for the uniform case. For the non-uniform
cases, the deviation angle, calculated as a function of the stag-
ger and camber angles through empirical charts, is based on
the methodology as described by Howell15 and Miller.14 This
parameter then allows determining the minimum loss (ml),
 optimum stall and choke incidence angles. The loss coefficient
calculati on in the method follows the approach as described in
the work of Howell15, Miller14 and White16 and uses the sets of
empirical correlations as summarized in Table 2.

The loss coefficient definition used in the fan performance
calculation is given by Eq. (1), and is based on studies
conducted by Howell15, Miller14 and Osborn.10 The total loss
coefficient is given by

![Fig. 3](image-url)
where \( \omega_p \) and \( \omega_{sec} \) stands for profile and secondary losses respectively. The secondary losses implemented in the model correspond to the end wall and shock wave effects. The ideal static pressure increment is calculated based on the assumption of constant relative total pressure across the rotor or constant total pressure for the case of the stator. As the fan assembly for the test case presents one stage and is expected to operate at low pressure ratio, the density variation and blockage effects across the blade arrangement are neglected. This assumption essentially simplify the model by enabling the use of incompressible flow equations. These equations can be used to calculate the static pressure increment, which for the case of constant axial speed is equal to the total pressure increment in the absolute frame of reference. This latter parameter together with the total loss coefficient is then used to determine the isentropic efficiency. In the case of multi-stage configurations and high pressure ratios, the aforementioned assumptions reduce the accuracy of the method to a large extent.

This is observed in the validation studies, and is discussed in the results section. A synthesized algorithm for the fan performance calculation encompassing the uniform and distorted flow cases is shown in Fig. 4.

### 2.2. Discretization Miller approach

In order to assess the radial and circumferential effects of ingesting a distorted flow field on fan performance, the mean-line design as previously described is modified. This modification uniformly discretizes the inlet rotor area into a number of blade passages and radial stations. After establishing the blade design with the mean-line approach, the inlet area is discretized in a number of streams. Fig. 5 shows a schematic representation of the fan face flow properties for uniform and distorted cases and the discretization of the areas for each configuration. As observed, the model discretizes in the circumferential region based on the number of blade passages, whilst for the spanwise blade direction (radial) the number of streams is a user’s input. In order to define an optimal set of values for this latter parameter, several

#### Table 2   Empirical approaches examined in this paper.

<table>
<thead>
<tr>
<th>Item</th>
<th>Model</th>
<th>Set 1</th>
<th>Set 2</th>
<th>Set 3</th>
<th>Set 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \omega_{ml} )</td>
<td>Miller(^{14})</td>
<td>Howell(^{15})</td>
<td>Miller(^{14})</td>
<td>Miller(^{14})</td>
<td></td>
</tr>
<tr>
<td>( \delta_{OD} )</td>
<td>Miller(^{14})</td>
<td>Howell(^{15})</td>
<td>Miller(^{14})</td>
<td>Miller(^{14})</td>
<td></td>
</tr>
<tr>
<td>( \phi_{OD} )</td>
<td>Miller(^{14})</td>
<td>Howell(^{15})</td>
<td>Wright(^{17})</td>
<td>Wright(^{17})</td>
<td></td>
</tr>
<tr>
<td>( \omega_{p} )</td>
<td>-</td>
<td>-</td>
<td>Wright(^{17})</td>
<td>Wright(^{17})</td>
<td></td>
</tr>
<tr>
<td>( \omega_{sw} )</td>
<td>-</td>
<td>-</td>
<td>Schwenk(^{18})</td>
<td>Schwenk(^{18})</td>
<td></td>
</tr>
</tbody>
</table>

* This case utilizes the approaches in Refs. 14, 17, 18 depending on the blade span section.

\[
\omega = \frac{\Delta P_{ideal} - \Delta P_{real}}{P_{LE} - P_{LE}} = \omega_p - \omega_{sec}
\]
cases are run and based on the accuracy needed and the computational resources this variable is defined. Although the present model is not computationally expensive compared with higher-fidelity tools such as CFD, the iterative calculation required can substantially increase the computational time. For instance, an increment in 5 streams in the spanwise direction would mean 150 points for a 30 blade passage case and hence the iterative calculation within the matrix arrangement would need to run for 150 times more. The parametric nature of the model and semi-empirical correlations for losses constrain up to certain extent the accuracy improvements and hence it is found in the present study that more than 10 streams do not enhance the trends obtained. Therefore, a range of discretized streams in the spanwise direction between 5 and 10 is found appropriate. To show better in detail how the trends of the flow performance are captured by the model, 10 streams have been selected in this work from hub to tip.

For each discretized stream, the air exit angle can be calculated in function of the speed triangles. This then enables the calculation of stall/choke angles and minimum loss for each stream in the discretized area, analogously to the mean-line approach.\textsuperscript{15,19} The flow and blade characteristics at each position are then used to calculate their empirical losses utilizing different semi-empirical approaches based on compressor performance. Table 2 describes the semi-empirical approaches investigated in this work. The definition of the loss coefficient for each discretized stream using the aforementioned correlations allows us to calculate the stagnation pressure losses through the blade passage and hence the static pressure increments and fan efficiency for each stream. The matrix of values obtained by using the discretized model enables us to calculate the three-dimensional flow behavior across the fan.

Since the aim of this model is to be implemented in the parametric assessment of the propulsion performance of BLI systems,\textsuperscript{7} the data obtained from the fan performance matrix is parametrized using the mass averaged values for static pressure increment and efficiency. These values then are used in the propulsion performance module as shown in Fig. 1.

It is pertinent for the reader to note that in this analysis the discretized area at the rotor face is considered constant till the stator exit. It is also important to note that as the empirical correlations cited by Schwenk and Lewis\textsuperscript{18} and Wright and Miller\textsuperscript{17} are developed for mean-line analysis, the variables in their definitions are not a function of the blade span and angular position. For the discretized Miller analysis undertaken in this study, the same expressions are used whilst angular and radial positions are taken into account. The loss coefficient definitions for profile and end wall losses in the discretized model are

$$\omega_{h}(r, \theta) = \frac{\omega_{h_{par}}(r, \theta)}{0.5 + \left(\frac{r_{\text{TE}(r, \theta)}}{r_{\text{LE}(r, \theta)}}\right) \cos(\beta_{\text{TE}(r, \theta)})}$$  \hspace{1cm} (2)$$

$$\omega_{\text{par}}(r, \theta) = \frac{\omega_{\text{par}_{\text{par}}}(r, \theta)}{0.5 + \left(\frac{r_{\text{TE}(r, \theta)}}{r_{\text{LE}(r, \theta)}}\right)^2 \frac{\omega_{l/c}}{h/c}}$$  \hspace{1cm} (3)$$

where $\omega_{par}$ (for end wall and profile losses) corresponds to the total loss parameter and depends on the diffusion factor.\textsuperscript{17} Similar to equations for calculating loss coefficients, the diffusion factors are also calculated as a function of the position in the flow. It is clarified that in Eqs. (2) and (3) the radius $r$ is defined by the number of stations along the blade span and the angular position $\theta$ is defined by the number of blade passages. For the uniform flow case, as there is no circumferential distortion, assessment of one blade passage is adequate. It is also important to note that the blade aspect ratio $h/c$ is assumed constant at any span and circumferential location in Eq. (3). This assumption is observed to cause some discrepancies between the calculated and experimental values (Fig. 9). The shock wave loss coefficient is determined analogously to the profile and end wall loss coefficients. This means that the supersonic turning angle, relative inlet Mach number and peak suction surface Mach number\textsuperscript{18} are defined for the radial and circumferential stations which are previously mentioned.
Because of the complexity of the flow in hub and tip regions, the empirical correlations presented by Wright and Miller have certain discrepancies compared with the experimental data. In order to improve the prediction capability of the loss models in these regions, different sets of correlations are assessed and compared. In the case of the stator, the loss coefficient model presented by Miller predicts with good accuracy the loss coefficients in all blade regions.

The complexity of the flow makes it difficult to capture the effect of the radial and circumferential distortion interactions between the discrete streams using empirical correlations and basic fan geometries, and therefore these have been usually studied with high-fidelity methods using detailed fan designs. As the developed model aims to be simple and accurate enough for preliminary design, these interactions are neglected.

2.3. Model validation

To examine the accuracy of the discretized Miller approach, a baseline case which ingests a uniform flow field is studied. As discussed earlier, for the validation study under uniform conditions, the NASA axial flow fan stage 53 operating at similar pressure ratios as the test case is selected. For the validation under distorted conditions, CFD studies are carried out and comparisons are made with data available for NASA axial flow fan stage 67.1

Uniform inlet flow field is assumed for the baseline case, and hence the radial fluctuations in the flow field are neglected. The uniform velocity profile used for this case is based on the mass averaged flow properties under the test conditions for the NASA axial flow fan stage 53. Table 1 lists some of the flow and fan characteristics.

2.3.1. CFD approach

In order to assess the balance between complexity demanded and accuracy provided using a high-fidelity approach, a CFD simulation is carried out to assess the fan performance. The results from this simulation then have been compared with the predictions of the discretized Miller approach. For this, the NASA axial flow fan stage 53 and a commercially available turbo-machinery performance analysis tool (ANSYS CFX) are utilized. The CFD study is carried out for the uniform case using the flow and component characteristics reported by Osborn.

The CFD analysis of the NASA axial flow fan stage 53 is carried out using total pressure and static pressure as inlet and outlet boundary conditions respectively. This approach is selected as it offers a reduced time for convergence and its results are found to be close to the experimental data. The values used for these boundary conditions at design point are described in Table 3.

It is important to mention that during the simulation the solution cannot converge when this starts from the design point conditions (Table 3). This is because if the downstream static pressure is highly enough, the intense flow separation will cause high instability and impede the solution to converge. The method applied to solve the problem is to start simulation with relatively low back static pressure till initial convergence is achieved, and then increase it gradually by manipulating the outlet boundary condition.

(1) Meshing

For the creation of the 3D model, the program ANSYS bladeigen and the data from report have been used. The blade model was then implemented in ANSYS turbogrid for meshing. ATM Optimized Mesh Topology has been selected due to its capacity of facilitating the creation of the high-quality mesh. Fig. 6 presents the meshes of rotor and stator generated according to default ATM setting. Tip-Clearance was set at 0.4% of the blade height.

To refine the mesh depending on the turbulence model, different levels of mesh refinement near the wall (blade, hub and shroud) have been considered. is the dimensionless distance of the node from the wall. This value should be refined consistently in order to resolve properly the viscous sub-layer. This study considers a value of 40 for the - model and value of 2 for the model. In order to have a good accuracy in the boundary layer region at hub and blade tip, the turbulence model was chosen.

(2) Grid independence study

The accuracy and precision of the simulation depend largely upon the mesh type and size. An adequate mesh should present a good compromise between simulation accuracy and computational time. A large mesh could present accurate results, but at the same time be highly expensive in computational resources.

The quality of the mesh relies heavily on the grid type (H, C or O). In this case, as it is a preliminary study, the ATM Mesh Type in Turbogrid presents good qualities as it delivers an optimized mesh structure divided according to the curvature of the blade (Fig. 6). This simplifies the mesh independence study to mesh size study. For this purpose, four mesh sizes have been investigated as shown in Table 4. They present approximately the same refinement for rotor and stator; however the presence of tip clearance for the rotor case contributes to a larger number of elements for the rotor mesh.

The key parameters used for mesh-independence study are the spanwise distribution of pressure ratio, temperature ratio and the averaged stage efficiency. For space reasons, here only the results obtained for the stage efficiency are shown. Fig. 7 shows how the mesh size increment improves the accuracy of

![Fig. 6 Rotor and stator grids at midspan.](image-url)
the results. However it is observed that the substantial improvement is obtained in the refinement from Mesh 1 to Mesh 3, whilst from Mesh 3 to Mesh 4 the improvement is less and the computational time is much longer. The good compromise between accuracy and computational time presented by Mesh 3 is the reason for it to be implemented in this CFD study.

The computational domain is shown in Fig. 8 and this reproduces the rotor and stator components. In order to ensure a uniform static pressure under the exit boundary condition, the exit stator component is assumed to have a length of three times the rotor radius.

2.3.2. Case studied: BLI configuration

The N3-X NASA aircraft concept is selected as the baseline architecture for the test case as it was designed to incorporate the turboelectric distributed propulsion system (TeDP) with boundary layer ingestion. As this work focuses on the fan performance assessment under distorted flow, the design point conditions and airframe configuration are assumed to be the same as those of the N3-X. Table 5 summarizes the design point conditions and fan design characteristics utilized for this study.

In order to calculate the flow properties at fan face, several assumptions are made. First, it is assumed that the inlet duct produces a drop in total pressure of 2%; second, the gradient of the total pressure and Mach number profiles remains the same until the fan face. To define the Mach number profile at fan face for the distorted case, an average Mach number of 0.5 is assumed. Based on this, the inlet Mach number profile is scaled up. The propulsor inlet total pressure and Mach number profiles are defined as a function of height and calculated using Eqs. (4) and (5) respectively.

\[ Ma_{BL} = Ma_{\infty} \left( \frac{y}{0.371 c_d \sqrt{Re_{cd}}} \right)^{1/11} - 0.14 \]  
\[ P_{BL} = P_{\infty} \left( \frac{y}{0.371 c_d \sqrt{Re_{cd}}} \right)^{1/15} - 0.075 \]

where \( c_d \) and \( Re_{cd} \) are the airframe length and Reynolds number respectively.

It is important to note that this study aims to assess the performance of a propulsor fan primarily designed for uniform conditions, but under the effects of distortion due to BLI. Under this premise, the distorted case starts with the design of the fan assuming the ingestion of uniform flow, which will not be the case in reality. However, this has been considered primarily to enable the assessment of fan performance and consequent deterioration, when a distorted flow field is ingested. After the fan geometry for the uniform flow is defined, the distorted condition is then assessed as a particular off-design case, where the flow properties vary in radial and circumferential locations.

In the test case, the flow coefficient (\( \psi = C_d / U \)) varies in radial and circumferential directions (depending on the three-dimensional axial velocity profile at the rotor inlet station, as observed in Fig. 5, due to the non-uniform axial velocity (\( C_a \)) that the ingested boundary layer presents. Therefore in this case, each discrete stream works under different operating conditions. For this reason, in this case, all the blade passages are assessed and then a radial and circumferential mass

<table>
<thead>
<tr>
<th>Table 5</th>
<th>Operating conditions at design point for BLI case.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
<td>Value</td>
</tr>
<tr>
<td>Intrinsic thrust at cruise (12192m) (N)</td>
<td>73952.6</td>
</tr>
<tr>
<td>Thrust split</td>
<td>95%</td>
</tr>
<tr>
<td>Mach number at cruise</td>
<td>0.84</td>
</tr>
<tr>
<td>Number of propulsors</td>
<td>15</td>
</tr>
<tr>
<td>Propulsor mass flow (kg/s)</td>
<td>55.36</td>
</tr>
<tr>
<td>Pressure ratio</td>
<td>1.25</td>
</tr>
<tr>
<td>Fan face Mach number (0.6)</td>
<td>Uniform</td>
</tr>
<tr>
<td>Inlet total pressure (kPa)</td>
<td>29.67 (Uniform)</td>
</tr>
<tr>
<td>Fan face Mach number (0.5)</td>
<td>Distorted</td>
</tr>
<tr>
<td>Inlet total pressure (kPa)</td>
<td>28.87 (Distorted)</td>
</tr>
<tr>
<td>Inlet total temperature (K)</td>
<td>244.9</td>
</tr>
<tr>
<td>Tip velocity (m/s)</td>
<td>244.2</td>
</tr>
<tr>
<td>Tip radius ( r_t ) (m)</td>
<td>0.6519</td>
</tr>
</tbody>
</table>

Notes: The remaining blade geometric properties are the same as those of the fan assembly 53.

\* These properties correspond to the mass averaged values.
average is used to define the average pressure ratio and fan stage efficiency.

3. Results and discussion

In this section, the results and discussion for the validation studies and the case studied (BLI configuration) are demonstrated.

3.1. Uniform case

The experimental data for the NASA axial flow fan stage 53 is used to select the most accurate set of empirical correlations for the loss coefficient. The selected set is then used to validate the discretized Miller approach.

3.1.1. Empirical correlations for loss coefficients

Since the loss prediction models have been developed for mean-line analysis, it is necessary to assess their suitability when they are used at different blade span locations. In this study, the loss prediction models described in Table 2 are investigated. In this table, Sets 1, 2 and 3 correspond to the approaches presented by Miller, Howell and Wright and Miller. Howell’s work is one of the earliest studies regarding the effect of losses on turbomachinery performance. For this approach, the total loss coefficient is determined directly from charts presented in his work. This aspect makes its implementation simple, nevertheless it does not enable assessing the effects of contributors to the total loss coefficient such as profile, end wall, shockwaves, and others. Another important work is presented by Miller and it focuses on the off-design performance of turbomachinery and analogously to Howell, it introduces semi-empirical charts in order to calculate the off-design performance. However, this work presents the same problem as Howell’s approach, as it only enables one to calculate the total loss coefficient. For this reason, Set 1 and Set 2 in Table 2 do not require the calculation of the loss contributors. In order to improve the performance prediction, Wright and Miller and Miller developed a model based on the individual source of losses in compressor performance and hence this approach allows one to individually calculate end wall, profile and shock wave losses. Fig. 9 collects the loss coefficient prediction using all the loss approaches aforementioned, which are described in Table 2.

As it is observed that depending on the region on the blade (root, mid and tip), some approaches are able to provide better predictions than others, a loss model that uses a suitable and best approach for each blade region is assessed. This approach is denominated Set 4 (Table 2) and it uses, for the tip region of the blade, the end wall and shock wave losses predicted by combining methods described by Wright and Miller and those defined by the conventional Miller approach. For the root and mid regions, the losses are predicted using methods described by Wright and Miller and Miller respectively. As Set 4 demonstrates the best results, it will be used in further analysis. This selected approach is from this point onwards referred to as the discretized Miller (DM) approach.

3.1.2. Stage performance of NASA’s rotor 53

In this section, the stage performance of the NASA’s fan 53 is assessed with the discretized Miller approach. This validation is only carried out for the uniform case.

1 Case studied using report data

To illustrate the effects of some assumptions, such as uniform inlet velocity, free vortex design (including blade design module) and constant axial velocity through the fan assembly, on the predictions of the discretized Miller approach, a case that neglects these assumptions and takes these data from the report is examined. Fig. 10 shows the fan performance characteristics when the actual data (from report) is used.

Fig. 10 shows that the discretized Miller approach captures the trend of the pressure increment and values of isentropic efficiency. It is observed that Set 4 of empirical correlations predicts the loss coefficients and isentropic efficiencies along the blade span with good accuracy. There are only some discrepancies in the tip and hub region, which can be attributed to the basic loss coefficient models and the uniform discretizing step utilized, and make the accurate assessment of the flow in these regions difficult. In the case of the pressure increment, an offset between the experimental and calculated values is observed, which can be attributed to the assumption of incompressible flow through the stage.

2 Case studied using blade design module and constant velocity assumption

Fig. 10 shows the isentropic fan efficiency and pressure increment in a meridional section for the uniform case using the blade design module and the assumptions of constant axial speed and negligible radial distortion. As observed in Fig. 10, the implementation of the aforementioned assumptions produces a larger offset in the values predicted. Since the geometry predicted by free vortex and the blade angles are affected mainly in the tip and hub regions, it can be deduced that the increment in the gap in the midspan region is mainly produced by the assumption of constant velocity through the fan assembly. The experimental data shows that there is certain difference in the axial velocities after the rotor, and if this is considered as in Fig. 10 (designed), the static pressure increment which is directly related to these velocities would match the experimental results better.

![Fig. 9 Loss coefficients according to different empirical correlations.](image-url)
The difference in the axial velocities after the rotor was found to present a similar trend to the circumferential distortion cases assessed with the parallel compressor approach,\textsuperscript{22} where the high velocity streams are less accelerated due to the lower pressure increment of these streams. In the parallel compressor approach, this is achieved by the assumption of equality of static pressures between clean and distorted streams at the back of the fan. For the discretized Miller approach, this is difficult to assume a similar static pressure at the back of the rotor, due to the difference in pressure increment along the blade span, an adaptation was implemented. For the discretized Miller approach, the axial speed at the rotor exit is iterated till the difference between the mean line static pressure of the blade passage operating at the fan top (the highest axial velocities) and bottom (the lowest axial velocities) is less than 2%.

Since in the test case the fan geometry and axial velocities distribution at the exit of the rotor are unknown, the approach which incorporates the blade design module and the aforementioned assumptions has been used in the following analysis. It is opined that for the level of fidelity utilized in this study for the preliminary analysis of BLI systems with TeDP, the loss in accuracy due to the previous assumptions will affect the trends of fan performance under distorted conditions to small extent. This is important as it is more relevant to capture the trend of the phenomena rather than to give accurate performance predictions at preliminary design stage when several configurations need to be tested.

3.2. Distorted case

The results obtained for the case of non-uniform inlet conditions are shown in this section. First, a comparison of the predictions between the discretized Miller and CFD approach is presented. The computational study from Jerez et al.\textsuperscript{1} examines the NASA axial flow fan stage 67 using CFD and experimental approaches under total pressure distortion covering 1/3rd sector of the inlet flow field. Fig. 11 shows the domain used for the analysis of the fan under distorted conditions examined by Jerez et al.\textsuperscript{1} The operating conditions for this analysis are taken according to the study of Jerez et al.\textsuperscript{1} and they correspond to 90% of the design speed, corrected mass flow of 32 kg/s based on inlet, mass-averaged stagnation quantities and a DC\textsubscript{120} of 83%.

To verify the discretized Miller approach, the clean conditions of the single stage NASA axial flow fan stage 67\textsuperscript{23} and the distorted conditions studied by Jerez et al.\textsuperscript{1} are utilized. The clean conditions in the cases presented in Fig. 12 are assumed to be equal to the undistorted sector of the incoming flow (upper fan or 0°), and the angle measurement takes a different angular position rather than the assumed one in this work. This can be observed in Fig. 5. Due to the fan’s ability to impart more energy to the lower momentum flow region, the ratio presented in the aforementioned figure is larger than the one for some radial positions.

3.2.1. Validation of discretized Miller method for circumferential distortion

The study presented by Jerez et al.\textsuperscript{1} examines the NASA axial flow fan stage 67 using CFD and experimental approaches under total pressure distortion covering 1/3rd sector of the inlet flow field. Fig. 11 shows the domain used for the analysis of the fan under distorted conditions examined by Jerez et al.\textsuperscript{1} The operating conditions for this analysis are taken according to the study of Jerez et al.\textsuperscript{1} and they correspond to 90% of the design speed, corrected mass flow of 32 kg/s based on inlet, mass-averaged stagnation quantities and a DC\textsubscript{120} of 83%.

To verify the discretized Miller approach, the clean conditions of the single stage NASA axial flow fan stage 67\textsuperscript{23} and the distorted conditions studied by Jerez et al.\textsuperscript{1} are utilized. The clean conditions in the cases presented in Fig. 12 are assumed to be equal to the undistorted sector of the incoming flow (upper fan or 0°), and the angle measurement takes a different angular position rather than the assumed one in this work. This can be observed in Fig. 5. Due to the fan’s ability to impart more energy to the lower momentum flow region, the ratio presented in the aforementioned figure is larger than the one for some radial positions.

The comparison of total pressure increment through the fan assembly predicted by the discretized Miller method and the values reported by Jerez et al.\textsuperscript{1} are shown in Fig. 12.
The results from Jerez 1 are included.

Total pressure increment along blade span corrected for Fig. 12

Effects on boundary layer ingestion induced distortion 11
improve the accuracy of the method. However, for the level
due to BLI. Further refinement would be recommended to
enable the assessment of the detriment in fan performance
method captures the trend of the values and hence it could
ratios.

NASA axial flow fan stage 53, as it operates at higher pressure
and non-separation in regions of high distortion.

observed are negligible radial and circumferential interactions
and non-separation in regions of high distortion.

These effects are more severe for this case than for the
NASA axial flow fan stage 53, as it operates at higher pressure
ratios.

To summarize, it is opined that the discretized Miller
method captures the trend of the values and hence it could
enable the assessment of the detriment in fan performance
due to BLI. Further refinement would be recommended to
improve the accuracy of the method. However, for the level
of fidelity required at the preliminary design stage, the trends
of the values for the fan performance characteristics are impor-
tant to highlight potential benefits/drawbacks in the configura-
tion of BLI systems. 9 The discretized Miller method presents a
good balance between accuracy and complexity. The simula-
tion carried out by Jerez et al. 1 for circumferential distortion
can illustrate the high computational demanding that assessing
BLI induced distortion is for the case of CFD. For instance, in
that study the simulation of the total domain (rotor, stator and
duct), which has a size of approximately $42.5 \times 10^6$ elements,
took two months on a 128 CPUs cluster. 1

3.2.2. Case studied: BLI configuration

Since the aim of the present method is to be implemented in
the analysis of TeDP systems with BLI at component and sys-
tem levels, the BL flow properties can be included in the
assessment.

This part of the work assesses the performance of the fan
propulser on the N3-X aircraft concept, operating as a part of
the distributed propulsion system with BLI. Some of the
characteristics for this case are described in Table 5. For this
case, the total pressure distortion in the inlet flow field is de-
veloped using the procedure explained in Section 2.3.2. The mass
averaged inlet total pressure distortion used for the BLI case is
shown in Fig. 13. The meridional performance characteristics
at different circumferential positions are shown in Fig. 14
(a) and (b). Fig. 14(c) and (d) shows the mass averaged values
for these parameters at different circumferential positions.
Table 6 provides the mass averaged values for the fan perfor-
ance under clean and distorted conditions.

Fig. 14 shows that the pressure increment for the distorted
case is larger than that for the clean case. This is attributed to
the lower axial velocity that the distorted profile presents. This
reduction in axial velocity at the distorted locations (Fig. 5)
produces a lower relative velocity at the inlet, which con-
tributes to the increase of the pressure increment through the
blade. For this reason, the blade passage located at 180° where
the axial velocity is the lowest (according to the distorted pro-
file assumed) presents the highest pressure increment. As
expected, in the case of the isentropic efficiency, it is observed
that the sectors experiencing lower distortion present higher
efficiency due to the lower pressure loss coefficients. Also it
is observed that in the tip region for the sectors less distorted
60° the isentropic efficiency is higher than the clean case. This
is related to the aforementioned reason, whereas the sectors
experiencing lower distortion and relative high pressure incre-
ment present less sensitivity to the pressure losses. In the region
of high distortion (fan bottom), the pressure losses increase
and the efficiency is affected more severely.

The trends of the values for the case of the circumferential
fan performance characteristics (Fig. 14(c) and (d)) indicate a
greater drop in efficiency in the locations of low velocity, cor-
responding to what should be expected in BLI systems. 3,1

The fan performance characteristics for the clean and dis-
torted case are summarized in Table 6. In this table, the values
shown correspond to the discretized Miller approach assuming
that the propulsor unit delivers the same amount of thrust for
the clean and distorted cases. In this table, a 3% drop in mass
averaged fan efficiency is obtained and can be attributed to the
higher pressure ratio that the distorted case presents in order
to match the thrust of the clean case. This produces a reduc-
tion in the propulsors’ mass flow and consequently reduces the height of the capture sheet, which increases the rate of distortion (the low momentum region of the BL is ingested). The other reason for this large drop in fan efficiency is that neither the duct nor fan have been designed to deal with distortion. Therefore, if better designs for these components are implemented, this large detriment in performance could be reduced. In the work of Florea, an aerodynamic analysis of a tolerant fan to BLI distortion is assessed using CFD. However, the complexity of the flow requires that the layout and detailed geometry for these components are defined. As mentioned before, the purpose of the discretized Miller is to be adaptable for preliminary design and highlight main trends which will be helpful to lead to the detailed design of optimum propulsion architectures with high-fidelity approaches that require more computational resources.

### 4. Conclusions

This work highlights the computational and time limitations that CFD methods present for distortion problems in BLI systems. Furthermore, this work presents an alternative method that is computationally cheap based on well-known empirical correlations and mean-line design, which has the novelty of discretizing the inlet region in circumferential and radial streams, so that the combined distortion patterns that BLI presents can be assessed. For this method, which denominates the discretized Miller approach, a set of empirical correlations to predict loss coefficients at different blade span locations have been determined. This tool shows good agreement for cases considering uniform and distorted flow, and further due to its simplicity, it can be considered suitable for preliminary design of BLI systems at design point. The application of the discretized Miller approach enables assessment of the effects of BLI in a propulsor fan. The approach indicates that BLI may result in a decrease of 3% in fan efficiency for the configuration studied. Such deterioration in fan performance will consequently lead to an increase in TSFC and hence fuel burn. This will then necessitate redefinition of previously generated optimal (distributed propulsion) configurations. The suitability of new BLI propulsion architectures then could also be assessed by incorporating the tool developed in this work, and hence the effects of combined radial and circumferential

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Clean</th>
<th>Distorted</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow (kg/s)</td>
<td>55.26</td>
<td>48.84</td>
</tr>
<tr>
<td>Stage pressure ratio</td>
<td>1.19</td>
<td>1.25</td>
</tr>
<tr>
<td>Isentropic stage efficiency</td>
<td>91.25</td>
<td>88.15</td>
</tr>
</tbody>
</table>

**Table 6** Mass averaged fan characteristics for BLI case.

![Fig. 14](image_url)  
**Fig. 14** Meridional and circumferential fan performance characteristics for BLI configuration.

Please cite this article in press as: Valencia E et al. Discretized Miller approach to assess effects on boundary layer ingestion induced distortion, *Chin J Aeronaut* (2017), http://dx.doi.org/10.1016/j.cja.2016.12.005
distortion can be included in the performance calculation at component and system levels.

In addition, the method presented here is aimed at undertaking preliminary design of distributed propulsion systems with BLI and hence the simplifications utilized are at the expense of accuracy. Some of the modifications which could be implemented to improve the accuracy of the presented method are as follows:

1. Improve the pitch to chord ratio rule\(^5\), so that it will better match experimental values.
2. Improve the blade design code using more accurate blade design theories such as exponential blading.
3. Refine the sectors close to the tip and hub regions using a non-uniform discretization; however this will require further improvement of the actual empirical correlations so as to improve the accuracy of predictions in these sectors.
4. Require to take into account the compressibility effects while calculating the pressure increments over the flow. This is an important aspect to consider as these effects become more noticeable in fans operating at high pressure ratios.
5. The assumption of constant axial speed through the fan assembly could also be avoided.

Acknowledgement

The authors gratefully acknowledge the financial support provided by Escuela Politécnica Nacional for the development of the project PIMI 15-03 and PIJ 15-11. They also thank Cranfield Departmental Grant for the support provided.

References

Presently, he is an associate professor in mechanical engineering at EPN, Quito.

Devaiah Nalianda received the Ph.D. degree in techno-economic environmental risk assessment from Cranfield University in 2012 and the M.S. degree in thermal power (gas turbine technology) from Cranfield University in 2009. Devaiah’s research is focused on conceptual development and application of distributed propulsion systems for civil aircraft.

Laskaridis Panagiotis received the B.E. degree in aerospace systems engineering from Coventry University, the M.S. degree in aerospace vehicle design from Cranfield University and the Ph.D. degree in performance evaluation and systems integration for more electric aircraft from Cranfield University. During his Ph.D. student period, he worked closely with Goodrich Corporation on the subjects of all/more electric aircraft as well as use of an electric motor to drive an afterburner pump.

Riti Singh is an emeritus professor of Cranfield University. He leads the Gas Turbine Engineering and Technology Group within the Department of Power and Propulsion and is the director of the Rolls-Royce University Technology Centre in Performance Engineering. He was previously deputy head of School of Engineering and head of Department of Power, Propulsion and Aerospace Engineering.