Design optimization of the A320 engine inlet cowl

Bruno Santos da Conceição
brunoconceicao29@gmail.com

Instituto Superior Técnico, Universidade de Lisboa, Portugal

November 2016

Abstract

Aircraft operate in environments in which the components are subject to large temperature and pressure variations. In structures as the engine nacelles, composed by several components and materials, the presence of wear and corrosion becomes noticeable, due to the their operation in environments as the one foremost described. Corrective actions must be employed to the components which present this kind of problems. The acoustic panels of the inlet cowl of the Airbus A320/A321, present corrosion problems on the aluminium doublers of the joints. In order to develop a corrective action to the joint of the acoustic panels, the analysis of the mechanical behaviour and forces acting on the joint must be made. In this work, a methodology involving Computational Fluid Dynamics (CFD), Finite Element Method (FEM) and Computer Aided Design (CAD) tools is developed in order to analyse the mechanical behaviour of the acoustic panels joint. The geometry of the inlet cowl and of its components are in Solidworks. The assessment of the aerodynamic loads acting on the inlet cowl is made with CFD tools, with STAR CCM+ software. The structural analysis of the members of the joint of the acoustic panels is made with the use of FEM tools in ANSYS Workbench software. The steps involved in the analysis are explained and the result are presented.

Keywords: Inlet cowl, acoustic panel, aerodynamic loads, mechanical behaviour, joints

1. Introduction

Commercial aircraft engines are invariably external pod-mounted and they are usually attached to the wing. The engines are enclosed in a structural housing, called nacelle. The main goal of a nacelle is to reduce the drag associated to airflow passing around the engine, minimize engine noise propagation and to provide a smooth airflow to the engine [1]. The nacelle configuration varies with the engine type [2].

TAP, the Portuguese airline has a fleet of 80 aircraft. From the 80 aircraft fleet, 19 aircraft are Airbus A320 and 3 are Airbus A321. At TAP, both the A320 and A321 aircraft share the same engine and the same inlet cowl. The engine of these aircraft is the CFM56-5B. Despite the use of the same engine, the engine is installed on each aircraft type with different rates. The inlet cowl of the A320/A321 is composed of three acoustic panels, the upper, the lower and side panels. The acoustic panels form a diffuser that provide an uniform airflow to the engine and also have acoustic properties, that partially cancel the engine’s noise [3, 4]. The inlet cowl of the A320/A321 is presented in Fig.1.

During the aircraft operation, the inlet cowl is exposed to large pressures and temperature ranges. Atmospheric temperatures can range from -65°C to 50°C, and some components of the inlet cowl can reach even higher temperatures due to the anti-icing system. In flight operation the inlet is exposed to rain, hail and birds. On ground operation, debris can be suctioned with the airflow into the inlet. All these factors contribute to the inlet degradation [5].

When aircraft components start to degrade, corrective actions must be employed. The corrective
actions depend on many factors, such as the severity of the problem and its location on the component. The corrective actions can be found in the repair manuals provided by the component manufacturer.

With regard to the acoustic panels, there are several repairs that are foreseen by the manufacturer. From time to time, aluminium corrosion appears on the doublers of the joint of the acoustic panels. Fig.2, shows a typical A320 acoustic panel and its components. A repair for the aft doubler of the panel (please see Fig.2) is foreseen by the manufacturer, however, no repair for the internal doubler is present in the repair manual.

TAP is certified with a Design Organization Approval (DOA). The certification is granted by EASA (European Aviation Safety Agency). This certificate grants to TAP the authorization to design changes for repairs, for some aircraft areas.

Since the repair of the corroded internal doubler of the acoustic panel is not foreseen in the repair manuals of the aircraft, making use of the certification to design a repair for this component can be beneficial for when such problems arise.

Hereupon, the work presented aims to support the approval of the repair with the analysis the mechanical behaviour of the joint where the doublers are applied.

2. Aerodynamic Loads Determination

2.1. CAD Modelling - A320/A321 Nacelle

Both aircraft models, A320 and A321, share the same inlet cowl and engine, the CFM56-5B. In order to correctly simulate the flow behaviour around the engine’s nacelle, a CAD model was created in Solidworks environment. Information gathered from Airbus website, Airbus A320 manual and measurements performed at TAP facilities, were used to create the CAD model, presented in Fig.3. A very important input for the CAD model was the mapped geometry of the lower acoustic panels. With the data of the mapped acoustic panel it was possible to determine the inlet curvature and length.

2.2. Simulated Flight Conditions

In order to structurally analyse the joint of the acoustic panel, the critical aerodynamic load must be determined. The takeoff and cruise phases were considered the most critical flight phases. In the takeoff phase engines are pushed close to their maximum power, in turn, the maximum air mass flow is suctioned by the engine. In the cruise phase maximum aircraft speed is attained. The landing phase wasn’t considered critical due to the low speeds of the aircraft and low power employed.

Four flight conditions were analysed. Three for takeoff with different angles of attack, at sea level. The fourth condition, for cruise at 11000 m (36000ft). The flight conditions are summarized in Table 1.

<table>
<thead>
<tr>
<th>Flight Conditions</th>
<th>Altitude [m]</th>
<th>Mach Number</th>
<th>Mass Flow [kg/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Takeoff 0°</td>
<td>0</td>
<td>0.438</td>
<td>400</td>
</tr>
<tr>
<td>Takeoff 9°</td>
<td>0</td>
<td>0.438</td>
<td>400</td>
</tr>
<tr>
<td>Takeoff 16°</td>
<td>0</td>
<td>0.438</td>
<td>400</td>
</tr>
<tr>
<td>Cruise</td>
<td>11000</td>
<td>0.847</td>
<td>110</td>
</tr>
</tbody>
</table>

2.3. CFD Approaches

Two different methodologies were implemented in order to obtain the aerodynamic load on the engine nacelle. As the engine fan blades are constantly rotating, real boundary conditions are complex and difficult to reproduce. Some simplifications to the
real model had to be done. Both implemented methodologies intended to reproduce the engine’s operation.

The first approach intended to simulate the engine fan operation. In order to simulate the fan a STAR-CCM+ inbuilt fan simulator was used. Some simulations were performed in order to understand the fan simulator’s reaction to different inputs. A porous media was used at the engine core in order to simulate flow dissipation and to generate a uniform flow at the engine exit. Due to poor results, unrealistic flow behaviour and lack of information about the engine’s fan, this methodology was dropped.

To simplify the complexity of the real engine model and to avoid the problems associated to the fan modelling, an alternative methodology was introduced. This methodology was the adopted one to perform all CFD calculations. In order to simplify the real engine model the following simplifications were made: the fan geometry was replaced by a section with the same diameter of the real fan. Boundary conditions were applied to the fan section to simulate the fan flow suction; No engine core components were modelled. A “black box” mechanism was used to simulate the engine core; as the focus of this work is to analyse the effect of the flow at the engine inlet, no exhaust flow simulation was performed. Instead, a “Sting” was created at the end of the nacelle. With the introduction of a sting the interaction of both hot and cold jets was avoided, reducing the simulation complexity. The sting has a length 5 times the fan diameter to minimize the wake effect on the area closer to the engine, as made in [6].

The final engine configuration for the CFD simulations is presented in Fig.4.

2.4 The CFD Methodology

The methodology used to simulate the flight conditions was based on the second approach previously presented. All the CFD simulations were made using an isolated STAR CCM+ engine model.

2.4.1 Computational Domain

A rectangular prism with typical dimension was used to defined the computational domain. All the prism dimensions were set as function of the fan diameter $D_{fan}$, starting at a referential with origin at the center of the fan. Both the computational domain and the boundaries used in the CFD simulations are presented in Fig.5.

![Computational domain and boundaries.](image)

2.4.2 Generated Mesh

The mesh used in the present work was composed by a trimmed mesh combined with a prismatic layer. The prismatic layer was only generated on the surfaces of interest, that is, on the surfaces where the wall shear stress was to be calculated. A few layers of prismatic cells are needed, typically 5 to 8 to determine the above properties. Ten layers were used in the computational simulations. The number of layers was also taken into account in order to obtain a $y^+ < 200$. Due to computational limitations, a mesh with a maximum number of 1.61 million cell was created. In order to obtain better accuracy in the regions of interest a block refinement was created. The mesh and its different levels of refinement can be seen in Fig.6.

![Mesh at the geometry mid-plane.](image)

2.4.3 The Problem Physics

In order to correctly represent the engine a tri-dimensional model was used to account for the en-
engine asymmetries and to allow the determination of the pressure and wall shear stress on the engine’s nacelle. All simulations were performed in steady state.

To define both air dynamic viscosity and thermal conductivity Sutherland’s law was used. Since all simulations are performed for Mach numbers above 0.3, compressible effects had to be taken into account. The air density was modelled with the ideal gas model.

RANS equations were used along with SST K-Omega turbulence model to perform the simulations. SST K-Omega model utilization is recommended by the software for transonic simulations and simulations in which recirculation regions are present.

The Coupled Flow Model was used. This model solves simultaneously the conservation for mass, momentum and energy. The model uses an implicit spatial integration along with the multigrid method. A second order upwind discretization scheme was used.

2.5. Boundary Conditions

Boundary conditions (BC) have a very important role in CFD simulations as they have a direct impact on the flow behaviour and on its interactions with the components involved in the simulation. Boundary conditions were divided into domain BC and engine BC. All BC are assign to the boundaries presented in Fig.5.

For the domain boundary conditions, three types of BC were used: free stream, pressure outlet and symmetry. Free stream BC were used to simulate the incoming airflow into the domain, simulating the aircraft velocity and angle of attack. The free stream BC was assigned to CV_INLET boundary, SYMMETRY_2 and SYMMETRY_4. The last two, only for angles of attack other than zero. A pressure outlet BC was assigned to the CV_OUTLET boundary. The symmetry plane BC was used in all SYMMETRY boundaries, except for the mentioned case explained above, where free stream conditions are used.

When selecting the BC for the engine model special care must be taken, since a complex model is intended to be simulated. In [7] simulations of powered engines are performed and different types of boundary conditions are presented. Three types of boundary conditions can be specified at the fan. The mass flow, the pressure or the velocity can be defined for a specific engine working condition. In the present work a pressure boundary condition was used i.e. a pressure variation ∆p, relative to the reference pressure value, is applied to the fan face. In [8], both mass flow and pressure boundary conditions are analysed. Results showed that pressure based boundary conditions have better convergence. Adiabatic wall boundary conditions were used on the rigid sections. A pressure outlet BC was used at the fan face with a defined pressure difference relative to the reference pressure.

2.5.1 The Fan BC Calculation

In order to determine the desired ∆p, to define the fan boundary condition, the isentropic relations were used together with the mass flow function. The information available to perform these calculations is: the mass flow (\( \dot{m} \)), the atmospheric static pressure (\( p_{atm} \)), the atmospheric static temperature (\( T_{atm} \)), the free stream velocity (\( V_\infty \)) and the free stream speed of sound (\( a_\infty \)). The free stream Mach number can be simply obtain with the expression represented in Eq.(1).

\[
M_\infty = \frac{V_\infty}{a_\infty}
\]  

(1)

Combining this information with the isentropic relations and mass flow function (Eq.(2)), a starting value for the ∆p was defined. From this point the ∆p was corrected until the desired mass flow was achieved in the simulation. The isentropic relations can be use in this approximation since no shock waves are expected to occur in the inlet duct.

\[
\frac{\dot{m}_{fan}}{A_{fan}} = \frac{\rho_0 \sqrt{\gamma}}{\sqrt{R T_0}} M_{fan} \left( \frac{1}{1 + \gamma \frac{1}{2} M_{fan}^2} \right)^{\frac{\gamma+1}{\gamma-1}}
\]  

(2)

In Eq.(2), \( \dot{m}_{fan} \), \( A_{fan} \) and \( M_{fan} \), are respectively the mass flow at the fan, the area of the fan and Mach number at the fan: \( \rho_0 \) and \( T_0 \), are the stagnation pressure and temperature, respectively. \( R \) is the universal gas constant and \( \gamma \) is the ratio of specific heats for a perfect gas [4].

2.6. Methodology Suitability

Since no experimental data of the A320/321 was available to figure out the suitability of the computational result, a different engine with experimental data was analysed. The NAL-AERO-02-01 T.P.S. (Turbine Powered Simulation) wind tunnel experimental model was used. This model represents an axisymmetric turbofan engine from the Japanese Aerospace Technology Research Institute [9]. With the available information, it was possible to create a three-dimensional CAD model of the T.P.S.. In order to analyse the suitability of the computational model, all simulation parameters used in the A320/A321 model were maintained for the T.P.S. analysis. Despite the difference in sizes between models, the proportions in the computational domain were maintained, since the computational domain is a function of the fan diameter. The mesh
cells density was also maintained between simulations, please see Fig.7.

A simulation to recreate the experimental data was performed, simulating the cruise condition. In Fig.8, a graphic with the experimental and computational pressure coefficient (Cp), is presented. In Fig.8 the good agreement between the experimental and computation result can be seen. Errors below 3% can be found in the region of interest (Inlet cowl). Since good results were obtained with this computational setting, it is expected to obtain result for the A320/A321 as suitable as the ones obtained with the T.P.S. model.

![Figure 7: T.P.S. model and mesh.](image1)

Figure 8: T.P.S. Cp distribution for computational and experimental results.

2.7. CFD Results and Discussion

Four flight conditions were simulated in order to determine the critical flight conditions to which the maximum flight aerodynamic load is presented. All simulations ran until the stopping criteria was attained. The control parameters in the present work were the Residuals, the Mass Flow and the $y^+$. The $y^+$ was used to control the mesh quality. Both the residuals and mass flow were used to control the solution convergence. In the present work the solution was considered to be converged when all the residual had values below $10^{-4}$. Although, the mass flow variation was analysed in parallel with the residual to ensure that this important quantity had also converged. In all flight conditions the residuals are below $10^{-4}$ and the mass flow converged to the desired values, approximately 400 kg/s for the takeoff conditions and approximately 110 kg/s for cruise.

From analysis of the velocity field it could be verified that flow is uniform for the lower takeoff angle conditions. For the takeoff 16°, flow separation is visible, accompanied with flow recirculation, that can affect the engine performance, please see Fig.9. At cruise condition, the flow is uniform in the inlet region, but the exterior of the nacelle is affected by shock waves.

The analysis of the Mach number at the fan, for the takeoff conditions, shows that the Mach number is in the typical range for turbofan engines, 0.4-0.7 [4]. For the takeoff 16° condition, the influence of the flow separation is visible on the Mach number distribution, please see Fig.10. The result for the cruise condition are slightly below the typical values, maybe due to the mass flow approximations.

The analysis of the pressure distribution shows that the maximum pressure has about the same magnitude for the three takeoff conditions. The minimum pressure increases in magnitude as the angle of attack increases, due to a further acceleration around the lip leading edge. For the cruise condition large pressure variations appear in the shock wave regions. The pressure result are presented in Table 2.

The analysis of the wall shear stress (WSS) shows that its magnitude is higher in regions of higher velocity, the inverse is also verified. Comparing the WSS magnitude to the pressure magnitude it can be seen that the WSS corresponds to about 1% of the pressure magnitude. The WSS results are presented in Table 3.

![Figure 9: Velocity field for takeoff 16°.](image2)
3. Analysis of the Joints of the Acoustic Panel - FEM Methodology

The methodology used to determine the mechanical behaviour of the joint of the acoustic panel and the fasteners of the joint is presented. The different approaches to the problem are explained. Results from the CFD methodology chapter were used, in order to correctly simulate the aerodynamic loading on the structure.

3.1. Preparing the FEM Simulations

The joint of the acoustic panel intended to be analysed is the joint presented in Fig.1. In order to understand the effect of the aerodynamic load on the acoustic panel’s joint and fasteners, several FEM simulations were performed.

3.1.1 Aerodynamic Load

All aerodynamic load obtained with Star CCM+ software, were imported to ANSYS Workbench. In order to preform the structural analysis both the pressure and wall shear stress were imported to ANSYS. The imported data correspond to the four flight conditions, takeoff 0, 9, 16 and cruise. Attention was made to verify if the imported coordinate system was coincident with the one found in ANSYS, and also to verify if the same units are used in both systems.

3.1.2 Geometry and Geometry Importation

In order to generate the structural analysis, a CAD model was created and imported into the ANSYS environment. All the geometries used in the analysis were created using Solidworks. Since the objective of the present work is to analyse the joint of the acoustic panel, special attention was made to correctly simulate the components in that region. In Fig.11, the external region of the inlet cowl are presented. The three sections corresponding to the acoustic panels are visible with three different colors. In Fig.12 the internal view of the inlet cowl with the joint components is presented. In addition to the components represented, there is an attachment ring that creates the connection between the inlet cowl joint and the engine.

3.1.3 Material and Mechanical Properties

Several materials are used in the construction of the inlet cowl and acoustic panel. All material properties were introduced in the software. The properties were obtained from CES EduPack 2015 Software, except those of the honeycomb core, which were calculated based on the geometrical parameters [10]. The Honeycomb was modelled as an orthotropic material.

3.1.4 Contact Between Components

Whenever two or more components surfaces touch each other, these components are said to be in contact. In the simulations of the present work the two types of contact used were Bonded and Frictionless.
When using bonded contact it is assumed that components in contact are glued together. No sliding or separation is allowed between components. Bonded contact was used in the acoustic panel components. The frictionless contact allows components to slide and to separate from each other. Unlike the bonded contact type, frictionless contact has nonlinear formulation that results in longer solutions time. This type of contact is used to simulate contact between the acoustic panel doublers and the attachments ring. It is also used to simulate the contact between the fasteners shank and the Joint’s holes.

### 3.1.5 Defining the Mesh

ANSYS Workbench presents several meshing methods to mesh solids components. Depending on the type of method selected, different building blocks will be used in the construction of the mesh. In the present work, the hex-dominant method was used. The hex-dominant method creates a mainly hexahedral element mesh. When the hexahedral mesh is compared to the tetrahedral it can be verified that the hexahedral mesh requires much less elements than the tetrahedral one to obtain the same solution accuracy. Quadratic elements were used in the analysis. As the computational resources are limited, a global mesh size was defined and local refinements were used to increase the mesh density in locals of interest.

### 3.2 Determination of the Critical Load Condition

In order to determine the critical loading condition, the pressure and wall shear stresses, determined in STAR CCM+, were applied to the structures in ANSYS environment. An analysis of the Force Reaction and Moment Reaction is made to determine the critical loading conditions.

#### 3.2.1 Model Considerations

The complete model of the inlet cowl was used in the analysis. In order to reduce the simulation complexity and the computational effort, no fasteners were simulated and bonded contact was used in all components. Since only reaction forces and moments are to be obtained, the use of Bonded contact doesn’t compromises the results.

All simulations were performed with the same geometry and mesh in order to have a base of comparison between simulations. The mesh used was of the Hex Dominant Type, with 305136 elements.

For each simulation, the respective aerodynamic loads, were imported into the model. To ensure that the geometry was held in place, a constraint was imposed at the Attachment Ring. The constrain was defined as a Fixed Support.

A simulation with the structural weight was also performed in order to understand its influence.

#### 3.2.2 Results and Discussion

Reaction forces and moments corresponding to the analysed loading conditions are presented in Table 4. As the angle of attack increases, both forces and moments reaction increase. It is possible to conclude that the loading corresponding to the Takeoff 16 flight condition corresponds to the critical loading condition. From the analysis of the results corresponding to the structural weight condition, it is visible that all components of the reaction force and moment reaction, have the opposite sign when compared to the other analysis made. This means that the addition of the structural weight reduces the global loading.

<table>
<thead>
<tr>
<th>Loading Condition</th>
<th>Force Reaction [N]</th>
<th>Moment Reaction [N.m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Takeoff 0</td>
<td>947.53</td>
<td>2128</td>
</tr>
<tr>
<td>Takeoff 9</td>
<td>21154</td>
<td>14476</td>
</tr>
<tr>
<td>Takeoff 16</td>
<td>34695</td>
<td>20236</td>
</tr>
<tr>
<td>Cruise</td>
<td>7371.7</td>
<td>6168.4</td>
</tr>
<tr>
<td>Structural Weight</td>
<td>1380</td>
<td>1490.4</td>
</tr>
</tbody>
</table>

### 3.3 Approach to Analyse the Acoustic Panel Joint

In order to obtain enough resolution to capture the desired properties, both for the fastener of the joint and the interface between the internal honeycomb and internal doubler, some simplifications had to be done to the model.

#### 3.3.1 Model Simplifications

In order to simplify the model, the equivalent stress of the complete bonded model was analysed. The loading conditions used in the simulations were the
critical ones, Takeoff 16. From these analysis it was possible to identify the region with higher stress levels. This region corresponds to the interface region between the Lower and Side Acoustic Panels. A section of the model comprising that region and a fastener pattern on each side of the interface was created. Please see Fig.13.

As the inlet cowl was sectioned, new boundary conditions had to be implemented in order to simulate the complete structure behaviour. A spring model was used to simulate the rigidity of the complete model. Please see Fig.14.

3.4. Analysis of the fasteners of the Joint
In the present section the analysis of the fasteners of the joint is made in order to understand the mechanical behaviour of the fasteners in the Joint.

3.4.1 Components Contact and Mesh Refinement
The aerodynamic loads tend to bend the inlet, generating tension on the Hi-Loks. Bonded contact was used to connect the Hi-Lok head to the doubler and the collar to the attachment ring. Frictionless contact was used between the attachment ring and the doublers and between the Hi-Lok shank and holes. Due to computational limitation bolts were locally refined. For the convergence analysis only the fasteners were refined.

3.4.2 Convergence Analysis
For the convergence analysis the equivalent stress (von Mises) of the fasteners was analysed. In order to always analyse the convergence of the fastener at the same point, a fixed referential was created for each fastener, at the maximum stress node. The same referential was used at each refinement.

As it can be seen from Table 5, there still exist a fluctuation on the stress values. Although, the error between consecutive refinements is very small. Refinement 5 was used to obtain the result for the bolts analysis.

Table 5: Bolts convergence analysis

<table>
<thead>
<tr>
<th>Refinement</th>
<th>No Elements</th>
<th>von Mises [Pa]</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>132150</td>
<td>1.206E7</td>
<td>1.074</td>
</tr>
<tr>
<td>2</td>
<td>134838</td>
<td>1.219E7</td>
<td>6.084</td>
</tr>
<tr>
<td>3</td>
<td>143908</td>
<td>1.298E7</td>
<td>0.038</td>
</tr>
<tr>
<td>4</td>
<td>208108</td>
<td>1.299E7</td>
<td>0.964</td>
</tr>
<tr>
<td>5</td>
<td>274881</td>
<td>1.287E7</td>
<td>-</td>
</tr>
</tbody>
</table>

3.4.3 Result and Discussion of the Fasteners Analysis

Analysis of the Tension and Shear Stress
The results obtained from the fasteners analysis allowed to compare the maximum tension stress \( \sigma_{\text{max}} \), and the maximum shear stress \( \tau_{xy,\text{max}} \), of each Hi-Lok, with the material yield strength limits. From the analysis of the results a minimum safety factor of 47 and 246 were obtained for tension and shear respectively.

Pretension Results
Hi-Lok fastener with two different diameters were used in the main joint. The preload analysis was made to the fasteners, of each diameter, which presented highest normal stresses. The normal stress of each of these bolts, was converted into tension force in order to make a comparison between the resultant normal force and the theoretical preload. The preload \( F_i \), was calculated with the Eq.(3) [11].

\[
F_i = \frac{T}{Kd}
\]

\( T \) corresponds to the tightening torque applied to the fastener (obtained from the fastener data sheet) and \( d \) correspond to the major diameter of the fastener. A torque coefficient, \( K \), of 0.16 was used.

The results are presented in Table 6. As the maximum tension force on the fasteners is only about 5%
of the theoretical preload, it is possible to conclude that the joint is safe against flange separation.

Table 6: Comparison between the theoretical preload and resultant tensile force on the fasteners.

<table>
<thead>
<tr>
<th>Hi-Lok Diameter</th>
<th>Theoretical Preload[N]</th>
<th>Tension Force[N]</th>
<th>% of Preload</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.76 [mm]</td>
<td>5148.44 (Hi-Lok65,4)</td>
<td>227</td>
<td>4.41</td>
</tr>
<tr>
<td>6.35 [mm]</td>
<td>7784.03 (Hi-Lok32,1)</td>
<td>375</td>
<td>4.81</td>
</tr>
</tbody>
</table>

3.5. Analysis of the Interface between internal honeycomb core and internal doubler

In the present section the forces acting on the bonded interface between the internal honeycomb core and the internal doubler are determined.

3.5.1 Components Contact and Mesh Refinement

Bonded contact was used in all the acoustic panel components. The contact between the attachment ring and the doubler was frictionless contact. Frictionless contact was also used between the fasteners shank and the holes of the fasteners.

In order to capture the forces acting between the doubler and the honeycomb, a fixed joint connection was used between the components. Two joints are analysed, one joint correspond to the Lower Panel section and the other to the Side Panel section. The two joints and respective referential are presented in Fig.15. In this analysis the refinement was focus on the component of the joint.

A convergence analysis was made and the most refined results were used for the analysis.

Figure 15: Joint’s interfaces and referential.

3.5.2 Results and Discussion of the Joints Analysis

In order to determine the Joints properties, calculations to determine the flat tensile strength and shear stress are made. Formulation presented in standards ASTM C297-94 [12] and C273-00[13] were used. The forces used in the analysis are presented in Table 7.

Table 7: Joints forces.

<table>
<thead>
<tr>
<th>Component</th>
<th>Force [N]</th>
<th>Force [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lower Joint</td>
<td>160.15</td>
<td>-150.6</td>
</tr>
<tr>
<td>Side Joint</td>
<td>912.97</td>
<td>571.29</td>
</tr>
<tr>
<td>Z</td>
<td>450.22</td>
<td>365.37</td>
</tr>
<tr>
<td>Total</td>
<td>1030.5</td>
<td>694.66</td>
</tr>
</tbody>
</table>

The Flatwise Tensile Strength

For the flatwise tensile strength analysis the maximum Y component of the force was used, corresponding to the lower joint. Both joints have the same cross-sectional areas, $A = 9946\text{mm}^2$. The flatwise tensile strength was calculated with Eq.(4).

$$\sigma = \frac{P}{A} = 92kPa$$  (4)

In Eq.(4), $\sigma$ represents the flatwise tensile strength and $P$ the load. Comparing the obtained result with the experimental test results of the LOCTITE EA 9658 AERO[14], it can be verified that the obtained tension are equivalent to 4.38% of the minimum experimental strength. A force 22.82 times the force obtained in the simulation would be needed to attain the minimum experimental strength limit.

The Shear Stress

The shear stress resulting from the aerodynamic load on the joint corresponds to the X and Z components of the joint force. Since the Z component of the lower joint is the largest force component, this force was used to calculate the shear stress in the joint. The calculation is made as follows, Eq.(5).

$$\tau = \frac{P}{Lb} = 45kPa$$  (5)

In Eq.(5), $\tau$ represents the shear stress, $P$ the load, $L$ the length of specimen, and $b$ the width of specimen.

As in the previous analysis, the results were compared with experimental data. The simulation revealed that results correspond to about 0.4% of the limit strength.

4. Conclusions

The overall results of this work allowed to reach the following conclusions:

- Two CAD models of the A320/A321 inlet cowl were created for the analysis performed using CFD
and FEM. The correct modelling on the nacelle can be very challenging.

- A methodology using Computational Fluid Dynamics to determine the aerodynamic loads acting on the inlet cowl of an aircraft’s engine was developed.
  - The aerodynamics loads, pressure and wall shear stress, were obtained for four flight conditions.
  - The suitability of the methodology employed to determine the aerodynamic loads was verified.
  - A methodology to analyse the joints of the acoustic panels using FEM was developed.
  - The critical loading condition was determined.
  - It was possible to conclude that the joint and the fasteners are subjected to combined loads.
  - Stresses in the fasteners are far from their yield limits, both for tension and shear. This indicates that the fasteners do not present risks for the joint safety.
  - Loads on the fasteners far from the preload ones.
  - Forces acting on the interface between the internal honeycomb core and the internal doubler were determined.
  - The analysis of the flatwise tensile strength and shear stresses allowed to conclude that LOCTITE EA 9658 AERO can be used to connected both the internal honeycomb core and internal doubler, with a large safety margin.
  - Finally, the development of this project allowed to create a methodology that allows the analysis of aerodynamic and structural parameters of the inlet cowl of an aircraft engine. This methodology could be used and adapted to analyse other models involving aerodynamic loads and structural analysis.

5. Future Work

In the future it would be interesting to perform a similar analysis with more computational power. It would be interesting to analyse larger sections of the inlet, or even the complete model. An experimental analysis of the joint of the acoustic panel could be done in order to compare experimental and numerical results. Finally, the developed model could be used to approach different problem.

Acknowledgements

I would like to express my gratitude to Prof. Luis Reis and Vitor Anes for their support and recommendations through the development of this work. I would like to thank TAP Engines Engineering department for their availability and for the opportunity of having direct contact with the components at their facilities.

References