Aerodynamics of a Powered Lift F35-B Aircraft in Ground Effect

Barata, J.M.M.¹, Cerdeira, A.T.S.¹, Durão, D.F.G.², Neves, F.M.S.P.¹, Silva, A.R. R.¹, Vieira, D.F.C.¹
1: Aerospace Science Department, Universidade da Beira Interior, Covilhã, 6201-001, Portugal
2: Universidade Lusíada, Lisboa, 1349-001, Portugal
* Correspondent author: jbarata@ubi.pt

Keywords: Impinging Jets, Ground Vortex, Fountain Upwash Flow

ABSTRACT

The complex flow field generated by the impact of twin impinging jets in tandem through a low velocity crossflow was experimental and numerically studied to represent aerodynamically the powered lift of the F35-B aircraft when it operate with ground effect. A wind-tunnel investigation has been conducted in the AeroG V/STOL tunnel with a vectored-thrust F35-B fighter configuration to the pressure measurement on the body and on the wing in the transition-speed range. The Reynolds number based on the jet exit conditions was 43,000, the jet-to-crossflow velocity ratio from 15 to 33.7, and an inter-jet spacing of 5=6Dmean, where Dmean = (D+D)/2. The impingement height used was 3 diameters. The mathematical model used is based on the solution of the continuity and momentum equations. A RANS formulation was adopted with the “k-ε” turbulent model to represent the turbulent stresses. The experimental results were used to make a more complete analysis of the flow field using a computational method, and revealed that the deflection of the rear jet is due to the competing influences the wake, the shear layer, the downstream wall jet of the first jet and the crossflow. The numerical results showed the influence of the impingement height on the ground vortex location, size and interaction with the surrounding flow, but new aspects of this type of flows were found for the present case of a tandem configuration. In the region between the jets the usual fountain upwash flow does not occur, but a second small ground vortex was detected, due to the interaction between the wall jets of each impinging jet. To our knowledge this is a new phenomenon that is being reported in the literature for the very first time. These studies have shown a complex flow field with regions of strong curvature and vigorous velocity variations that may be associated with important negative pressures which are of major importance for a V/STOL aircraft in ground vicinity.

1. Introduction

Turbulent jets impinging on flat surfaces through a low velocity crossflow are typical of the flow beneath of VSTOL aircrafts. When a VSTOL aircraft operates there are three different stages during the flight, being all different, with its own flow characteristics and with different kind of problems associated. A VSTOL flight comprises the hover phase, the transition to forward flight phase and the forward flight operation. The hover phase is the most complex phase during the flight and can be further divided into two more phases because the flowfield associated in each
phase are completely different. So, the hover phase can be subdivided into out-of-ground-operation and hover in the vicinity of the ground. During its landing or near ground hovering phase, the VSTOL aircraft creates a complex three dimensional flow field between the jet streams, the airframe surface and the ground. When ground effect occurs, the lift forces on the aircraft changes, cause hot gas re-ingestion into the engine intake and due to the fountain upwash and ground flows, the fuselage skin temperature rises. The unsteadiness of the flow and raise of the temperature cause several problems in the engine performance, such as, compressor surge or even stall and thrust reduction. In respect to the intake ingestion phenomenon, it is very complex and can be associated with the design and operational parameters, such as, jet configuration, head wind velocity, jet impingement height or intake configuration. In the case of the hot gas ingestion problem, there are three mechanism involved, i.e., far field ingestion, near field ingestion and ground vortex ingestion. The first mechanism is results of the forward away initially movement of the ground sheet wall jet due to the aircraft movement. This happened because the hot gases after some distance lose its momentum, rising and separating from the ground. The portion of the hot gases that separating from the ground, mixes with the surrounding air and backs again to the intake. The second mechanism, near field ingestion, has a much greater impact on hot gas ingestion compared to the first, because it directly affected the lift nozzle exits into the surrounding area of the intake, being that when exits multiple impinging jets, its impact on the ground plane create a fan shape up wash fountain beneath the aircraft. When the fountain impinges on the underside of the fuselage, flowing from the fuselage to the intake, the engine may sucks the flow to the intake, creating severe temperature distortion to the intake, since, these gases are much hotter than those from the far field ingestion. The latter mechanism is due to the presence of a ground vortex. During a landing or hover the impingement of each downward-directed jet on the ground results in the formation of a wall jet which flows radially from the impinging point along the ground surface. The interaction of this wall jet with the free stream results in the formation of a ground vortex far upstream of the impinging jet. This flow field transports exhaust gases away from the ground and up toward the intake region. The level and intensity of the ingestion resulting from this mechanism depends critically on the forward velocity. If there are two or more adjacent jets, the resulting wall jets meet, and a fan-shaped upwash, or “fountain”, is normally formed between the jets. The fountain upwash flow depending on its strength and direction affects the forces and moments induced in the aircraft when operating in ground effect. The resulting ground vortex shape is strongly affected and the corresponding induced suckdown effect tends to be reduced by the upload produced by the fountain. The improvements of the knowledge are ever required because the aircraft design has been
changed since its first design, and some problems still persists. One of them is the negative pressure coefficient region on the lower surface of the wing and on the bottom of the fuselage induced by the vectored-thrust jet. The region is larger and the pressure coefficients are more negative for the front vectored-thrust nozzles than for the rear vectored-thrust nozzles. The jet exhaust also induces a region of negative pressure coefficients on the bottom of the fuselage. The induced pressure effects are larger at the larger velocity ratios and at the location nearest to the jet. For the next generation of VSTOL aircrafts F-35 no relevant studies can be found, because the impinging jets are aligned with the crossflow and this geometry has not yet been considered.

This paper aims to present an experimental and numerical work of the pressure measurement on the body and on the wing of the vectored-thrust F35-B fighter configuration in the transition-speed range. The numerical work is also extended to a detailed analysis of the three dimensional flowfield for different velocity ratios. Both experimental and numerical are continuation of the numerical and experimental studies done early on side-by-side impinging jets or tandem impinging jets (Barata (2013) and Vieira et al. (2015)).

Experiments on the aerodynamics of jets through a crossflow have mostly been reported for large impingement heights, for low velocity ratios between the jet and the crossflow $V_j/U_0$ and the focus of the most studies are the velocity distribution on the flow field. In this Therefore these works have only peripheral relevance to the VSTOL F-35 ground effect problem, being the pressure distribution an important parameter to analyse this phenomenon. Until the early 80's most of the computational work published on jets with crossflow had been based on integral methods admitting simplified assumptions, which are only capable of predicting global effects such as trajectories and jet cross-section shapes. In the late 80's and 90's new developments emerged fostered by the need of improving the Harrier / AV-8B and several research took place especially funded by the UK and US. Barata (2013) presents a comprehensive bibliographic review of that era and also introduces the new age (of the JSF-Joint Strike Fighter) and the
relevant investigation including the aspects of the fountain upwash flows that emerge from multi-jet impingement.

![Fig. 2: Fountain upwash flow formation in the middle of the impingement jets](image)

In the present paper part of the attention is devoted to the flow between each impinging jet, which normally would give rise to an upwash flow and to the details of the present in tandem configuration. Previous detailed measurements of the flow properties for fountain upwash flow are scarce and have been presented essentially in the absence of a crossflow. The most relevant works have been reviewed by Barata et al. (1989a) and Saripalli (1983), showing high turbulence levels and spreading rates in the fountains (e.g. Gilbert (1983) and Nishino et al. (1996)). Barata (1996a) and Barata (1996b) extended their study to multi-jet impinging configurations for twin impingement jets, producing upwash fountain flows (Fig.2) which are the heart of the complex effects produced by VSTOL aircraft when they operate in ground proximity. Therefore, studies with the impingement jets aligned with the crossflow are scarce in the literature, but of great importance to understand the complexity involved in this type of flowfield.

The remainder of this paper is presented in four sections. Section II describes the experimental and numerical methodology. Section III presents the experimental and numerical results and discussion. The last section summarizes the main findings and conclusions of this work.

2. Methodology

**Experimental Method**

The AeroG V/STOL tunnel facility designed and constructed for the present work is schematically shown in Fig. 3. During all the design process, especially for the boundary layer part of the flow, were followed the recommendations for open circuit wind tunnels. A fan with 15KW nominal power drives a maximum flow of 3000m³/h through the boundary layer wind tunnel of 300x302mm exit section.
The test section used is an adaptation of the one used in previous experimental works for an impingement height equals to 20.1D (Barata et al. (2014)). The present experimental work is dedicated to the pressure measurement on the body and on the wing of the vectored-thrust F35-B fighter configuration in the transition-speed range for a low impingement height.

**Fig. 4**: Test section with pressure measurement system installed

**Fig. 5**: Mesh created on the plate to the pressure measurement

Therefore, it is design a plate with the same length and width of the test section with 76 pressure taps with 1 mm inner diameter and 2 nozzles that represents the impingement jets (Fig. 4). The mesh created with the pressure taps on the plate is showed in Fig. 5, corresponding to each of the nodes of the mesh to the location of a pressure tapping. Each jet unit have a different diameter, where the front/first jet ($D_1$) has 11 mm inner diameter and the rear/second jet ($D_2$) has 10.35 mm inner diameter, and both are mounted vertically in the
plate with the axis contained in the vertical plane of symmetry parallel to the crossflow. The inter-jet spacing used is $S=11.5D_{\text{mean}}$, where $D_{\text{mean}} = (D_1+D_2)/2$ and the impingement height used is 3 diameters.

The origin of the horizontal, $X$, and vertical, $Y$, coordinates is taken at the midpoint between the centres of the jets exit (Fig. 3). The $X$ coordinate is positive in the direction of the wind tunnel exit and $Y$ is positive upwards.

The present results were obtained at each location for jet mean velocities of $V_j=36\text{m/s}$ and mean crossflow velocities of $1.06 \text{m/s} < U_0 < 2.4 \text{m/s}$, corresponding to a velocity ratio of $15 < V_R = V_j/U_0 < 33.7$. These velocity ratios are used in order to continue and complete the experimental study begun by Barata et al. (2014).

![Fig. 6: Pressure gauge used in experiments](image)

The pressure measurements were done through a pressure gauge with 22 pressure taps (Fig. 6). The pressure gauge was inclined about 7 degrees with the horizontal plane in order to obtain more accuracy in the results.

**Numerical Method**

**Mathematical Model**

In order to better understand if the experimental results realistically portray the situation under study, a numerical simulation was performed for the same conditions presented experimentally.

** Governing differential equations**

This section presents a numerical analysis based on the experimental data presented by Barata et al. (2014) and Vieira et al. (2015). The mathematical model used in the numerical simulation is
based on the solution of the continuity and momentum equations. A Reynolds-Averaged Navier Stokes (RANS) formulation was adopted with the “k-ε” turbulence model described by Launder and Spalding (1974), to represent the turbulent stresses.

The governing equations are written in a similar form:

$$\frac{\partial}{\partial x}(\rho u \phi) + \frac{1}{r} \frac{\partial}{\partial r}(rp v \phi) = \frac{\partial}{\partial x}\left(\Gamma_\phi \frac{\partial \phi}{\partial x}\right) + \frac{1}{r} \frac{\partial}{\partial r}\left(r \Gamma_\phi \frac{\partial \phi}{\partial r}\right) + S_\phi$$  \hspace{1cm} (1)

Where the property $\phi$ represents the velocity, turbulent kinetic energy or dissipation while $S_\phi$ and $\Gamma_\phi$ assume different values related with $\phi$ as described in table 1.

<table>
<thead>
<tr>
<th>$\phi$</th>
<th>$\Gamma_\phi$</th>
<th>$S_\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U$</td>
<td>$\mu_r$</td>
<td>0</td>
</tr>
<tr>
<td>$V$</td>
<td>$\mu_r$</td>
<td>$\Phi - \rho \epsilon$</td>
</tr>
<tr>
<td>$k$</td>
<td>$\mu_r/\sigma_k$</td>
<td>$C_\epsilon \Phi k^{-1}$</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>$\mu_r/\sigma_k$</td>
<td>$C_\epsilon \Phi k^{-1}$</td>
</tr>
</tbody>
</table>

Table 1: Differential equation coefficients

The turbulent diffusion terms are approximated by two equations from “k-ε” turbulent model where the Reynolds tension is related with shear tension:

$$\rho \ddot{u}_i \ddot{u}_j = -\mu_r \left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) + \frac{2}{3} \delta_{ij} \rho k$$  \hspace{1cm} (2)

Where $\mu_r$ represent turbulent viscosity derivative from the turbulent model expressed by:

$$\Phi = \mu_r \left\{2 \left[ \left(\frac{\partial U}{\partial x}\right)^2 + \left(\frac{\partial V}{\partial r}\right)^2 + \left(\frac{V}{r}\right)^2 \right] + \left[ \frac{\partial U}{\partial r} + \frac{\partial V}{\partial x} \right]^2 \right\}$$  \hspace{1cm} (3)

The turbulence model constants which are used are those indicated by Launder and Spalding (1974):

<table>
<thead>
<tr>
<th>$C_\mu$</th>
<th>$C_1$</th>
<th>$C_2$</th>
<th>$\sigma_k$</th>
<th>$\sigma_E$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Table 2: Turbulent model constants
The solutions of the governing equations were obtained using a finite difference method. This method that uses the discretized algebraic equations deduced from the exact differential equations which they represent. The discretized equations are obtained integrating the flow equations on the control volume defined by the domain discretization.

Solution Procedure

The solution procedure is based on the SIMPLE algorithm widely used by several references (e.g. Patankar et al. (1977)). This algorithm it used to staggered grid arrangement and correct procedure to solve the problem of obtaining a pressure field such that the solution of the momentum equations satisfies the continuity equation.

Boundary conditions

The computational domain has six boundaries where dependent values are specified (Fig. 7). At the inlet boundary uniform profiles of all dependent variables are specified from the experimental conditions. At the outflow boundary the gradients of the dependent variables in the axial direction are set to zero. On the symmetry plane the normal velocity disappears and the normal derivate of the other variables are zero. At the solid walls the wall function method used by Launder and Spalding (1974) is used to prescribe the boundary conditions for the velocity and turbulence quantities. At the jet exit boundary the mass flow rates and the momentum are the same than those in the experimental study. The computational domain corresponds to the experimental conditions that are detailed by Barata (2013) and Barata et al. (2014).

3. Results and Discussion
In this section, experimental and numerical data obtained will be presented and discussed for the velocity ratios $15 < V_R < 33.7$ and an impingement height, $H/D$, equals to 3. Figures 8 to 13 show the pressure measurements obtained experimental and numerically.

**Fig. 8:** Experimental pressure distribution along the bottom of the fuselage and wing (i.e. $Y \approx 3$) for $V_j/U_0 = 15$, $H/D_m = 3$, and $L/D_m = 11.5$ (values non-dimensionalized by $1/2 \rho V_j^2$)

**Fig. 9:** Numerical pressure distribution along the bottom of the fuselage and wing (i.e. $Y \approx 3$) for $V_j/U_0 = 15$, $H/D = 3$, and $L/D = 6$ (values non-dimensionalized by $1/2 \rho V_j^2$)

**Fig. 10:** Experimental pressure distribution along the bottom of the fuselage and wing (i.e. $Y \approx 3$) for $V_j/U_0 = 22.5$, $H/D_m = 3$, and $L/D_m = 11.5$ (values non-dimensionalized by $1/2 \rho V_j^2$)
The impingement jet location is identified in the experimental figure through a circle, while in the numerical figures is identified by the intersection of the vertical line with the symmetry plane (Z/D=0).

Comparing the experimental results to the ones obtained numerically, it is evident that the experimental mesh used needed more points especially in the region between the impingement jets, -5.62<X/Dm<5.62, to increase the results resolution. Analyzing the results for all the velocity ratios studied, on the impingement jets location a red area is presented correspondent to a regions of high static pressure, due to the large positive values of the horizontal velocity...
component. The cold coloured regions around the impingement jet location on the numerical results correspond to low pressures and are associated with the core of the ground vortexes. As it is expected the location nearest to the jets are the region with larger induced pressure effects. These large negative pressure coefficients induced a suction region around the impingement location, corresponding to the phenomenon of the hot gas re-ingestion into the engine intake that causes several problems in the engine performance due to the unsteadiness of the flow and raise of the temperature.

Through the figures 14 to 15 (obtained numerically) it is observed that for a lower impingement height the formation of a ground vortex in the region between the jets. Taking into account studies referenced in the literature, it would be expected that for lower impingement height, the fountain upwash flow phenomenon was observed in the flowfield results. Instead of this, in the region between the jets, it is observed the formation of a second ground vortex that results of the

![Figure 14](image1.jpg)

**Fig. 14:** Predicted mean vertical velocity component distribution along the vertical plane of symmetry (i.e. Z=0) for $V_j/U_0=15$, $Re_j=43,000$, $H/D=3$ and $L/D=6$

![Figure 15](image2.jpg)

**Fig. 15:** Predicted mean vertical velocity component distribution along the vertical plane of symmetry (i.e. Z=0) for $V_j/U_0=22.5$, $Re_j=43,000$, $H/D=3$ and $L/D=6$

![Figure 16](image3.jpg)

**Fig. 16:** Predicted mean vertical velocity component distribution along the vertical plane of symmetry (i.e. Z=0) for $V_j/U_0=33.7$, $Re_j=43,000$, $H/D=3$ and $L/D=6
interaction of the first jet inner wall jet with the second inner wall jet that captured it and given rise to a clockwise recirculation close to the second jet. This result it is new and it has not yet reported in the literature. The ground vortex centre position is coming to upstream and the ground vortex becomes increasingly with the velocity ratio increase, as it is expected, traduced in an increased on the induced pressure effect. When compared this results with the ones obtained by Barata et al (2014), it is verified that for lower H/D the first jet practically does not suffer deflection caused by the crossflow interaction, due to the increase of the jet strength when it is close to the ground as the impingement height will be lower, and protects the rear jet of the crossflow influence. So, the rear jet is entrained by the upstream jet and not by the crossflow itself.

For all the velocity ratios, when the upstream ground vortex feels the presence of the ground vortex formed between the impingement jets, it becomes wider and closer to the lateral walls (identified by the green area). In the experimental results (fig. 8, 10 and 12) the development of the ground vortex in the region between the jets is not clearly identified due to the low number of pressure taps in this location. While the impingement jets location do not have any pressure tap, it is not possible to see the same pressure distribution that the one exposed numerically, being the results presented in this regions results of the interpolation made by the Tecplot. This interpolation assumed zero pressure in the location between the jets, breaking the upstream ground vortex flow, something that is merely an error evidenced by the results obtained numerically.

4. Conclusions

An experimental study and numerical study were done to provide information to the pressure measurement on the body and on the wing of the vectored-thrust F35-B fighter configuration in the transition-speed range. The experiments was carried out for a Reynolds number based on the jet exit conditions of Re=4.3x10⁴ with an impingement height of 3 jet diameters and for a velocities ratio between the jet exit and the crossflow, Vr= Vj/Ut of 15, 22.5 and 33.7 with and an interject spacing of S=11.5Dm. The jet exit conditions and velocity ratios were chosen in order to complete the experimental study initiated by Barata et al. (2014). To complete the investigation and compared the experimental results were also performed a numerical simulation for the same impingement height and velocity ratios.

Through the numerical results it is showed the formation of a ground vortex in the region between the impingement jets, a fact that has not been reported in the literature and it is surprising at the first sight, because it was expected to find an upwash fountain flow for lower
impingement heights. This structure results of the interaction of the first jet inner wall jet with the second inner wall jet that captured it and given rise to a clockwise recirculation close to the second jet. From the numerical results another important conclusion can be taken about the real role and contribution of the second jet, being this responsible for the first jet reinforce, influencing the size and location of the ground vortexes centres.

Comparing the experimental results obtained for the pressure measurements with the ones obtained numerically, it can be conclude that the experimental results do not have enough resolution in the region impingement jets region and in the region between them. In a general way the pressure distribution results agree with the expected for the situation on study, the aircraft hovering in the vicinity of the ground. The region where the vectored-thrust jets impinge presents the high static pressure, due to the large positive values of the horizontal velocity component, while the location nearest of them exhibit the larger negative pressure coefficients, being one of the causes for the hot gas re-ingestion into the engine intake phenomenon.

Acknowledgments

The present work was done in the scope of the activities of LAETA-Associated Laboratory of Energy, Transports and Aeronautics, and funded by FCT-Fundação para a Ciência e Tecnologia through contract PTDC/EME-MFE/102190/2008 and grant SFRH/BD/86434/2012.

References


