Characteristics of helicopter engine exhaust through scaled experiments using stereoscopic particle image velocimetry

Zhen, Wei Teo; Wong, Wai Hou; Lee, Zhi Wen; New, Tze How; Ng, Bing Feng

2021

Zhen, W. T., Wong, W. H., Lee, Z. W., New, T. H. & Ng, B. F. (2021). Characteristics of helicopter engine exhaust through scaled experiments using stereoscopic particle image velocimetry. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, 235(9), 1091-1104. <https://dx.doi.org/10.1177/0954410020966471>

<https://hdl.handle.net/10356/155212>

<https://doi.org/10.1177/0954410020966471>

© 2020 The Author(s). All rights reserved. This paper was published by SAGE in *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering* and is made available with permission of The Author(s).

Downloaded on 26 Apr 2023 18:08:28 SGT

Characteristics of helicopter engine exhaust through scaled experiments using stereoscopic particle image velocimetry

Zhen Wei Teo¹, Wai Hou Wong¹, Zhi Wen Lee¹, Tze How New¹, Bing Feng Ng^{1*}

Abstract

Helicopter engines are often mounted atop the fuselage to keep the aircraft footprint small and optimal for operations. As a result, hot gases produced by the engines may inadvertently impinge upon the tail boom or dissipate inefficiently that compromises on operation safety. In this study, a scaled fuselage model with a hot air blower was used to simulate hot exhaust gases. The velocity field immediately outside the exhaust port was measured through stereoscopic particle image velocimetry to capture the trajectory and flow behaviour of the gases. Two cases were considered: freestream to exhaust velocity ratios of 0 and 0.46, respectively. The formation of a counter-rotating vortex pair was detected for both cases but were opposite in the rotational sense. For the case without freestream, the plume formed into a small “kidney” shape, before expanding and dissipating downstream. For the case with freestream, the plume formed into a slenderer and more elongated “reversed-C” shape as compared to the case without freestream. It also retained its shape further downstream and maintained its relative position. These observations on the trajectory and shape of plume provide basis to understanding the nature and interaction of the plume with its surroundings.

Keywords

Aerodynamics, Vortex, Counter-rotating, Helicopter, Crossflow, PIV

¹ School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798

Corresponding author: bingfeng@ntu.edu.sg

1 Introduction

The main motivation behind this study is to characterize the exhaust plumes, with and without forced convection, to aid in the understanding of tail boom impingement. During flight, the tail boom is subject to aerodynamic forces from the rotor downwash and the freestream. In addition, depending on the geometry of the exhaust duct, the tail boom may experience additional thermal stresses from the exhaust plume, leading to premature failure.¹ This is especially important, since recent improvements in power-to-weight ratios of turbo-shaft engines used in helicopters incurs higher exhaust temperatures.²

Investigations into helicopter exhaust plumes are highly complex and usually split into component studies. Barring the complex combustion process inside the turbo-shaft engines, to date, research is still being conducted on the lobed mixer, which is in the aft-section of the engine that encourages cold secondary flow to mix with the hot primary flow from the engine with low pressure losses.³ One of the more recent studies on the lobed mixer involved modifying its geometry to study its effects on heat and mass transfer.⁴ Lobed mixers are widely used on aerospace engine exhaust systems due to their ability to increase nozzle thrust and efficiently mix high temperature exhaust flow with cooler ambient air, thus lowering noise and plume IR radiation.^{2, 5-10}

The adoption of mixer-ejectors represented a paradigm shift in ejector design; previous designs utilised viscous forces to transfer energy from the hot stream to entrain a cooler secondary stream.⁸ Substantial axial length was required to ensure mixing since the method relied on viscous shear layer mixing of streams. In the lobed mixer-ejector, the device continues to carry out mixing with no moving parts, but makes use of a endplate with a convoluted trailing edge; these convoluted edges creates shear layers between the streams, eventually creating streamwise vortices which encourage a high degree of mixing.¹¹ This mixing process is enhanced due to the following effects: (1) spanwise vortices occurring in

free shear layers due to Kelvin-Helmholtz instability, (2) increased contact area due to the convoluted edges of the lobed-mixer and (3) streamwise vortices that are created due to the unique geometry of the lobed-mixer.¹²

In the study of helicopters aerodynamics, a lot of effort has been placed on understanding the flow over the helicopter fuselage and/or rotor downwash. One of the most comprehensive study was the particle image velocimetry (PIV) experiments that was performed in the Generation of Advanced Helicopter Experimental Aerodynamic Database for computational fluid dynamics (CFD) code validation (GOAHEAD) project.^{13, 14} The aim of the GOAHEAD project was to create a comprehensive experimental database for validation purposes, with the focus on unsteady viscous flow phenomena such as flow separation and transition, including rotor dynamics. PIV measurements of the velocity fields were done at various spanwise sections of the retreating and advancing rotor blades, along the tail boom as well as across the horizontal stabiliser. However, in that study, no engine exhaust was incorporated in the model.

Aside from GOAHEAD, there were also studies that focused on the immediate region aft of the exhaust nozzle, ignoring any effects of cold secondary flow mixing.^{15, 16} The goal was to understand and solve the overheating issue at the exhaust nozzle. Tufts in the exhaust revealed a clockwise (CW) swirl on the left and a counter-clockwise (CCW) swirl on the right when viewed from the top of the exhaust.¹⁵ Although not specifically mentioned, this secondary flow pattern observed was a counter-rotating vortex pair (CVP) and was formed due to the curvature of the exhaust. Helium-filled soap bubbles were used to visualize the flow around the exhaust. The flow was observed to roll over the edge of the exhaust lip upon exit, being drawn down and impinging on the side of the system,¹⁶ thus causing overheating. This was observed to happen at freestream to exhaust velocity ratios (FER) of between 0.6 to 1. The cruising conditions for this model occurred at a FER of 0.7 and having the correct

FER ensured that the wind tunnel testing would be representative of the operating conditions should the actual velocities be impossible to replicate.¹⁷ It is also worth noting that no rotor downwash was simulated in the aforementioned studies.^{15, 16} This gave confidence, at least in the regions near the exhaust nozzle, that wind tunnel testing without simulating the downwash would still be applicable and relevant. The slight difference would be the angle between the exhaust and the interacting flow.

This same phenomenon was also reported in a later CFD simulation and was attributed to the jet in cross-flow effect.¹⁸ In that simulation, only the exhaust, rotor downwash and fuselage were modelled, neglecting the effects of the tail rotor and vertical tail. The fuselage modelled was the Rotor Body Interference (ROBIN).^{19, 20} At low and intermediate speeds, characterized by advance ratios of 0.048 and 0.167, respectively, the exhaust plume intersected the main rotor disk, before being drawn back down by the rotor and impinging on the tail boom. This also caused a reduction in lift of the rotor due to the lower density of the heated plume, increasing the power required to maintain trim conditions. The exhaust plume does not intersect with the tail boom at a higher advance ratio of 0.287. For the case of hovering, Pan et al.⁶ examined the effects of the exhaust orientation and the rotor downwash on exhaust plume flow and IR signature numerically. In the case of the upward exhaust, the plume impinges on the tail boom, as observed by the streamlines and temperature distribution on the tail boom. This was not observed for the lateral and oblique exhausts. At a higher rotor downwash speed, the extent of heating was reduced and a more pronounced asymmetry in the temperature distribution was observed. This asymmetry was most likely caused by the velocity field induced by rotor downwash.

To date, most of these studies focused on the flow over a relatively larger timescale and were concerned with the overall trajectory of exhaust plumes. There is limited focus on the vortex created by the jet in crossflow effect which this paper aims to address. It seeks to examine the

exhaust flow behaviour over a smaller timescale, specifically the vortex formed immediately adjacent and downstream to the engine exhaust area. As such, a scaled mock-up of the engine cowling was used to better understand the near field flow behaviour of the exhaust in an open-loop wind tunnel.

2 Experimental Setup

2.1 Wind Tunnel

An open-loop wind tunnel with test section 1.1 m (width) $\times$ 0.9 m (height) $\times$ 2 m (length) was adopted in this study. The wind tunnel was able to achieve freestream velocities of up to 30 m/s. In this study, the operating freestream velocity was set at 21 m/s corresponding to a FER of 0.46. Through the use of a honeycomb flow straightener and anti-turbulence screens, the wind tunnel had a manufacturer reported turbulence intensity of less than 1% during operation. To ascertain the turbulence intensity at the operating speed of 21 m/s, a Dantec Dynamics 54T30 miniCTA system was used together with a 55P16 type straight, unidirectional hotwire probe. The output from the miniCTA system was recorded with a NI USB-6008 data acquisition unit, which can sample at rates up to 10 kHz. The turbulence intensity was determined from a 60 s sampling time at 10 kHz. For the operating freestream velocity of 21 m/s, the turbulence intensity was found to be 0.47%. Although this value was slightly higher than typical wind tunnels, this was still deemed acceptable as the focus of the study was on the exhaust plume, which itself is highly turbulent. The overall uncertainty for the turbulence intensity was no less than 3.3% based on procedures recommended by Moffat.²¹

2.2 Fuselage Model

A typical civilian helicopter fuselage was used in this study. To maximize the size of the test model, only a part of the cowling was selected to be scaled down for testing by a factor of

4.26. The simplified engine cowling model was 1.5 m long. Due to the internal structures and size, the test model was manufactured with nylon by additive manufacturing in seven separate pieces and fastened together mechanically. The parts were originally printed with dark grey colour and matt black spray paint was used in order to reduce laser reflection. In addition, the nylon parts were smoothed down by hand with 2000 grit sandpaper till they were smooth to touch. Previous experience²² with this method of fabrication justifies its use in this study. Minor gaps between the fastened pieces were filled with plasticine to ensure a smooth continuous surface. The test model was held up by 15 mm thick acrylic plates. The model in the test section is shown in Figure 1(a), with a close-up view of the model in Figure 1(b). To replicate the effects of the engine, a blower with heating coils was used. The operating speed of the blower was controlled through a frequency inverter. A type K thermocouple was placed at the end of the heating section to monitor and control the amount of power supplied to the heating coils to achieve the desired temperature through a feedback loop. The blower used is shown in Figure 1(c).

The stainless-steel pipes used to direct the flow output from the blower through the lobed mixer and test model, shown in Figure 1(d) and 1(e), had an outer diameter of 88.9 mm and an inner diameter of 78 mm. The pipes were assembled in two parts for ease of installation. This also allowed for planar measurements of the flow field to ensure that the flow after the bend had reattached. The location is indicated in Figure 1(b). The lobed mixer was also fabricated through additive manufacturing with stainless steel due to its geometric complexity and welded to the pipe. A pitot-static tube was used together with a Setra 265 low differential pressure transducer to measure the output of the blower. This pressure transducer is rated at 6.225 kPa at full-scale (FS) and $\pm 0.25\%$ FS accuracy. This procedure was different from the use of a handheld anemometer²³ as the pitot-static tube allowed for a higher sampling rate. The data acquisition system extracted readings from the transducer at a sampling rate of 2000

Hz. The same setup used to measure the turbulence intensity of the wind tunnel was also used for the pipe flow. The turbulence intensity determined for the pipe flow was 3.9%. The overall uncertainty for the turbulence intensity was 1% based on procedures recommended by Moffat.²¹

Figure 1. (a) Image of fuselage model in wind tunnel test section, inset showing region where the model is from on the actual geometry. (b) Internal structure of the fuselage model. (c) Centrifugal blower with heating coils used in this study. (d) Internal piping within the

fuselage model. (e) Close-up view of lobed mixer. (f) Fog generator used for visualizations and PIV.

The temperature field was measured using a platinum (Pt) resistance temperature detector (RTD). RTDs are known for their reliability, accuracy and stability as compared to other sensors such as thermocouples or thermistors.²⁴ The RTD used in this study was of class A accuracy as specified in the European standard DIN 60751: $\pm (0.15 + 0.002 \times T)$ °C, where T is the operating temperature in degrees Celsius. The RTD was connected to a National Instruments (NI) PXIe-4357 temperature input module which had twenty input channels and could record at rates up to 100 Hz per channel. The PXIe module supplied a 0.9 mA excitation current to each channel, where the resistance across each RTD was recorded and processed. The RTD was connected in a 4-wire configuration to compensate for the resistance of the lead wires connecting the RTD to minimize errors. The accuracy of the PXIe module, neglecting the RTD, was ± 0.4 °C for a temperature of 100 °C. The combined uncertainty²¹ of the RTD and PXIe module was then estimated to be no less than 0.82%. To minimize heat loss from the pipes, an elastomeric thermal insulation, 13 mm thick and rated at 0.0036 W/(m.K) at 20 °C, was used. Two Oriental Motor EZSM4D040K linear actuators were used in conjunction to accurately position the pitot-static tube and RTD within the inner diameter of the pipe. The linear actuators had a motion resolution of 0.01 mm, and repetitive positioning accuracy of ± 0.02 mm.

2.3 Surface Oil Flow Visualization Setup

Surface oil flow visualization method²⁵ was adopted to grasp qualitative information of the near-surface flow fields around the model engine cowling. The scoops, National Advisory Committee for Aeronautics (NACA) duct and exhaust outlet were covered up as the intention was to study only the external aerodynamics. Previous surface oil flow visualizations utilized a suspension of fine pigment in a mixture of mineral oil, for which the pigment is dependent

on the colour of the surface being applied.²⁶⁻²⁸ The pigments in the oil film applied on the model would leave streaks behind as the flow develops, and the remaining patterns were photographed for interpretation.

For the mineral oil mixture, the composition depends on the test conditions and varies from case to case. Ideally, the oil should only start to flow when the air flow reaches the desired testing conditions. In order to achieve that, the viscosity of the oil can be adjusted by either adding additives, such as naphtha, to thin it or by using a more viscous oil as the base solution. Hence, some trial and error were required to obtain the optimal mixture. In addition to viscosity, an additional safety consideration of low flammability was taken into account as the oil might be carried downwind to the motor, which can get hot after prolonged usage. A mixture of white oil and Carlupe™ oil treatment, a commercially available fuel additive, was used in the ratio of 4:1 in this study. The pigment used was a luminescent zinc sulphide, which glows a neon green in the absence of light. A digital single-lens reflex (DSLR) Canon EOS 550D camera equipped with an 18-135 mm, $f3.5-5.6$ lens was used to capture the images of the surface oil patterns. In order to capture the entire pattern in one image, this smaller focal length lens was required due to the size of the model. The camera was placed at a 45° angle in order to maximize the coverage of the top and the side of the model.

2.4 Stereoscopic PIV (SPIV) Setup

A SAFEX fog generator 2010, shown in Figure 1(f), was used to generate the tracer particles to visualize and estimate the trajectory of the exhaust. The mean droplet size produced by the fog generator was approximately 1 μm . The resulting fog was directed into the intake of the blower. This visualization would aid in the SPIV experiments as the PIV window is limited in size and prior knowledge of the location of the flow to be measured would be beneficial. Images and videos were captured with the same camera as the one used in the surface oil flow visualizations.

Figure 2. (a) Top-view schematic of experimental setup. (b) Placement of camera in experimental setup. (c) Close-up view of calibration board. Calibration image (d) before dewarping procedure and (e) after dewarping procedure (Note the change in size and perspective of the image).

For a SPIV setup, at least two cameras are required to compute the third out-of-plane component of the flow. Figure 2(a) shows the top-view schematic of the stereoscopic setup used in this study. The difference in the views of the right and left cameras allowed for the calculation of the third out-of-plane of component as two geometric relations could be constructed from each camera (4 equations for the 3 displacements in the x, y and z direction based on the measured displacements from each camera); the extra equation can be used to increase the accuracy by taking the average.²⁹ Since the cameras (image planes) are set at an angle to the object plane, the Scheimpflug condition,^{30,31} must be met in order for the entire image to be in focus. This can be achieved through a tilt-shift of the lens with respect to the

image plane. This arrangement also causes the image to deform into a trapezoidal shape after mapping it back onto the original object plane due to the perspective, as seen in Figure 2(d) and (e). This reduces the effective overlap area between cameras, especially at the corners.

The SAFEX fog generator used for the visualization of the plume was also used to seed the flow for the SPIV measurements. The outlet of the generator was pointed directly into the intake of the blower to seed the flow coming out through the exhaust. A Quantel EverGreen 200 mJ double-pulsed Nd:YAG laser was used for the SPIV measurements. It had a maximum repetition rate of 15 Hz. A simple planoconcave lens was used to expand the beam into a sheet of roughly 6 mm thick. For the stereoscopic setup, two TSI PowerView™ Plus 4MP Camera were used together with Scheimpflug adaptors and Nikon 105 mm, $f/2$ lenses. The camera resolution is 2048 by 2048 pixels, with a pixel size of 7.4 μm , and a maximum capturing speed of 7.25 Hz. Six measurement planes were taken, spaced at intervals of 2.5% of the length of the model, downstream of the exhaust. The orientation and location of these planes are marked out in Figure 3(a) and (b), respectively. Optimally, to reduce errors in the out-of-plane component, the two cameras should form a right angle between them, or 45° between the imaging plane and each camera. However, due to restricted optical access for this setup, both cameras were placed on the same side of the wind tunnel with 20° between them for a backscattering setup.³²

Figure 3. (a) Orientation of axes used in this study. (b) Location of SPIV measurement planes.

The calibration process and PIV analysis were carried out with both UVMAT³³ and PIVlab,³⁴ open source MATLAB codes. The time intervals used between the laser pulse were 25 and 40 μ s for the case with and without freestream, respectively. These values were checked for after each run to ensure that the average pixel shift was approximately less than 25% of the final interrogation window based on best practices previously published.³⁵ To calibrate the cameras, a precisely manufactured calibration board was used. The calibration board forms a grid of 0.19 m by 0.19 m with white circular markers spaced 0.01 m apart. The calibration board also had an elevation 0.001 m for the out-of-plane component for half of the markers. A total of 1000 image pairs were captured to compute the average velocity field. To improve the quality of the results, the images were pre-processed based on a proper orthogonal decomposition (POD) approach to remove the background and stray reflections³⁶. The multi-grid approach was used, from an interrogation window size of 128 by 128 pixels, to 64 by 64 pixels and finally 48 by 48 pixels. A Gaussian 2-3-point fit algorithm was used to achieve sub-pixel precision.³⁴ A 50% overlap was used for a higher spatial resolution. The final resulting vector map had a resolution of 2 mm. Vectors larger than half the final interrogation window size were rejected and replaced by interpolation³⁴ via a boundary value solver. The estimated uncertainty of the results was 4.1% based on procedures previously published.²¹ The main contributor to the uncertainty was due to the measurement of the freestream of the wind tunnel. Other factors considered included the calibration errors and the uncertainty in the firing timing of the laser.

3 Results

In the following section, the pipe flow measurement results will be presented first. This shows the nature of the flow after the bend before entering the lobed mixer and the exhaust duct. Next, the surface oil flow visualizations will highlight any significant flow structures that would arise from the interaction with the freestream and the model. Finally, the plume trajectory and velocity field are presented.

3.1 Pipe Flow Measurements

The measurement points constituted a grid spaced evenly, from the centre of the pipe, with interval sizes of approximately 10% of the internal radius of the pipe. The pitot-static tube was shifted close to the walls of the pipe. As the L-shaped pitot-static tube was held horizontally, the measurements were done twice from both sides of the pipe to the centre, to limit the blockage within the pipe, as shown in Figure 4(a). The resulting halves were then stitched together to form the entire field and are shown in Figure 4(b) and (c), with the “×” symbols denoting the measurement locations. The contours were generated in MATLAB through the biharmonic spline interpolation.³⁷ This method was used for its consistency, as it was not based on triangulation, and extrapolation beyond the measurement points. This can provide necessary information for readers to perform numerical simulations and validations. The velocity was normalized by the area-averaged velocity distribution of 45.5 m/s. The temperature was normalized by the room temperature of 300 K.

The velocity and temperature distributions were slightly skewed at a small angle CCW and not symmetric about the centreline. This was most likely due to the swirl generated by the blower’s impeller and the pipe bend. There is a larger concentration of high-speed region in the upper half of the pipe where y/r is positive. This distribution was the result a set of helical CVP. This CVP was caused by the pressure gradient induced through the centrifugal force experienced by the flow across the bend.³⁸ This pressure gradient caused an acceleration in

the flow near the outer radius of the bend and a deceleration in the flow near the inner radius of the bend, resulting in secondary flow structures. This feature is also known as Dean's vortices, attributed to the first person who introduced an analytical solution to such a flow scenario.³⁹ As the Dean number, a function of the Reynolds number and curvature ratio (ratio of the cross-sectional radius to the radius of the curvature) increases, this pressure gradient increased and the speed up in axial velocity in the outer radius of the bend became more pronounced. The observed measurements in Figure 4(b) and (c) were in agreement with the previous literature^{38, 40} for the given geometry as seen in Figure 1(b).

Figure 4. (a) Setup used to measure the velocity and temperature distribution within pipe section, interpolated and extrapolated contour for (b) normalized velocity distribution and (c) normalized temperature distribution of pipe with (x) indicating measurement locations.

3.2 Surface Oil Flow Visualization Results

Prior to running the stereoscopic PIV experiments, surface oil flow visualization was first conducted on a clean configuration (with scoops, NACA duct and exhaust covered) without the blower at a FER of 0.46. This was done to get a preliminary understanding of how the flow around the clean configuration behaved and if it would significantly impact on the flow and trajectory of the exhaust plume. The oil-and-pigment mixture was stirred and applied evenly on the surface manually with a fine brush. The wind tunnel was left running until no changes in the oil streaks could be observed.

Figure 5. Surface oil flow visualizations on model with NACA duct, scoops and exhaust covered with FER of 0.46.

Overall, there were no significant areas of flow separation on the cowlings, suggesting a streamlined body shape without regions of significant adverse pressure gradients, as seen in Figure 5. Oil would tend to accumulate in regions of flow separation.⁴¹ Moreover, this was confirmed through recorded footage of the development of the flow patterns on the surface. There were two spots of oil accumulating directly behind the two scoops. This recirculating region was expected due to the wake forming behind the protruding scoops along the cowlings. Another interesting effect was the upwards trajectory of the oil near the floor after the lower scoop. This was probably due to the boundary layer growing along the floor interacting with

the wake after the scoop. However, this would likely not happen on the actual helicopter cowlings as there is no ‘floor’ for the boundary layer to grow. The other slight smudges were mainly due to gravity acting on the oil, causing it to drip downwards. Overall, it can be concluded that there would be minimal impact caused by the shape of the fuselage model on the freestream which would interact with the exhaust plume.

3.3 Smoke Visualization Results

A total of 2 cases were examined: with and without the freestream ($FER = 0.46$ and 0 , respectively). The flow was allowed to stabilise by running the blower for 5 minutes prior to the visualizations. Two views, from the top and the side, were captured in order to appreciate the three-dimensional shape of the exhaust.

Figure 6. Smoke visualization of exhaust plume from the side at (a) $FER = 0$, (b) $FER = 0.46$ and from the top at (c) $FER = 0$, (d) $FER = 0.46$.

For the case without freestream, a linear trajectory was observed for the smoke, presented in Figure 6(a) and (c). The trajectory followed closely the angle at which the smoke departed from the curvature of the exhaust duct. The trajectory was observed to follow a linear path as there were no other sources of interaction. As for the case with freestream, from the side view presented in Figure 6(b), the smoke did not travel as high up as compared to the case without freestream. This was because the smoke was restricted and advected downstream by the incoming freestream. In the vertical direction, the exhaust trajectory resembled that of a curve approaching an asymptote. It was also brought downwards closer to the floor. From the top view presented in Figure 6(d), the spanwise expansion was also much smaller compared to the case without freestream.

3.4 SPIV Results

The coordinate axes used for the presentation of the SPIV data were shown in Figure 3(b). The 2D planar PIV measurements were taken in the Y-Z plane, resulting in the v and w velocity components. The velocity component in the x direction, the out-of-plane component, u , was reconstructed from the two vector maps. The resultant velocity was denoted as V and normalized with the area-averaged velocity distribution from the pipe measurements, V_{ref} . Six different measurement planes were taken, with a gap of 0.0375 m between them, to study the evolution of the wake downstream. The distance was 2.5% of the length of the model and was based on a previous study.¹³

3.4.1 FER = 0

Figure 7 shows the normalized velocity magnitude, V/V_{ref} , obtained from the stereoscopic PIV and shows the progression from planes 1 to 6. The plume from the exhaust formed a “kidney”-shaped profile, with its highest velocity found in the centre of the plume. This “kidney”-shaped profile can be attributed to the formation of a CVP within the curve of the exhaust duct. The top one rotated CW, while the bottom one rotated CCW, when looking at it

from the upstream position, i.e. freestream direction is into the page. As it progressed downstream, the core was seen to elongate diagonally, forming a slenderer profile at plane 6. It also moved upwards and away from the model. As expected, with no freestream, the velocity of the plume decreased as it progressed downstream from planes 1 to 6.

Figure 7. Normalized velocity magnitude contour plots for FER = 0 for (a) plane 1, (b) plane 2, (c) plane 3, (d) plane 4, (e) plane 5, (f) plane 6.

3.4.2 $FER = 0.46$

Figure 8 shows the normalized velocity magnitude, V/V_{ref} , obtained for the case of $FER = 0.46$. In this case, the plume from the exhaust formed a slender reversed C-shaped profile, with some swirl at both its ends, although the bottom one is less prominent. The plume did not move as far out in the spanwise and vertical directions. This was due to the freestream magnitude being much larger than the spanwise and vertical components, thus suppressing displacements in those directions. A CVP was formed at either ends of the plume. The top one rotated CCW, while the bottom one rotated CW, when looked upon from the upstream position, i.e. freestream direction is into the page. Note that there were some erroneous high-speed regions found in the images in Figure 8(c) to (e) at the middle left region of the image. This was due to the laser reflection of the model surface and lack of seed particles in that region.

Figure 8. Normalized velocity magnitude contour plots for $\text{FER} = 0.46$ for (a) plane 1, (b) plane 2, (c) plane 3, (d) plane 4, (e) plane 5, (f) plane 6.

Figure 9. Interpreted flow structure for the case with (a) absence of freestream and (b) presence of freestream.

3.4.3 Comparison between absence and presence of freestream

On first inspection of the velocity contours, the direction in which the exhaust plume face was opposite as seen from the “kidney”-shaped profile and the reversed C-shaped profile. The directions in which the CVP formed and developed were opposite in direction. In the case of $FER = 0$, the CVP were formed due to the internal curvature of the exhaust. The bend in the exhaust duct caused the formation of the CVP. This phenomenon was the same as the one observed for the pipe measurements presented in the earlier section. For the case of $FER = 0.46$, the freestream interacting with the exhaust plume was not dissimilar to the case of a jet in cross-flow done by authors in previous studies.^{42, 43} In this case, the exhaust was an angled jet in cross-flow. As mentioned previously, notwithstanding the lack of downwash in their experimental setup, Shaw and Wilson^{15, 16} identified the flow features that caused heating issues in actual operating conditions. Moreover, the formation of the CVP was also identified in a prior simulation,¹⁸ inclusive of the downwash, was also present in this study. The main effect of the downwash would be to alter the trajectory of the CVP in a downwards direction, but would not change the nature of the exhaust plume. The exhaust to freestream velocity ratio was approximately 2, and this effect was more significant compared to the secondary flows formed due to the exhaust duct curvature. Upon exiting the exhaust duct, the initial CVP formed within the exhaust duct was suppressed by the CVP formed upstream of

the exhaust duct exit. The proposed large-scale flow structures were shown in Figure 9(a) and (b) for the case without and with freestream, respectively. The other flow structures that would have formed such as the leading- and trailing-edge vortices and jet shear layer were not drawn in this case as this study was more focused on the wake rather than the flow near mouth of the exhaust.

The λ_2 -criterion⁴⁴ was used to detect the vortex core location from the averaged velocity fields. The vorticity was calculated at the location of maximum λ_2 . The circulation was calculated by integrating the vorticity around the common area that satisfied both the λ_2 criterion and a normalized vorticity magnitude of more than 10 as the threshold. This would ensure that only the appropriate area was included in the calculations. It is interesting to note that no vortex cores were detected for the case with no freestream. It seems like the CVP formed inside the exhaust duct was weak in strength and dissipates quickly upon leaving the duct. The only remaining evidence of the CVP was the shape and velocity distribution of the plume.

Figure 10(a) and (b) show the trajectory of the cores for $FER = 0.46$ from planes 1 to 6 in the lateral and vertical directions, respectively, while Figure 10(c) and (d) show the normalized vorticity and circulation of the vortex pairs, respectively. In the spanwise direction, the CW rotating vortex appeared to move in the direction towards the model as it progressed downstream. This is the effect of the freestream and the model. At that location, due to the presence of the model, the flow was forced to curve and follow the shape of the model, hence inducing some spanwise component to the flow. As seen with the CCW rotating vortex, which was higher up off the floor, there was much less spanwise movement as the effect of the model on the freestream was much less pronounced. For the movement of the cores in the vertical direction, the CW rotating core was observed to move slightly upwards while the CCW rotating core was observed to move slightly downwards. These slight deviations were

probably also the effects of the freestream after interacting with the model. The vorticity and circulation calculated reduced as the cores progressed downstream from plane 1 to plane 6. The strength of the vortices decreased through dissipation due to viscous forces. The trajectory of the CVP is important as most of the heat would be transported by the CVP.

Figure 10. (a) Horizontal trajectory, (b) vertical trajectory, (c) normalized maximum vorticity and (d) normalized circulation of the CVP for FER = 0.46.

4 Conclusion

In this paper, stereoscopic PIV was carried out on a scaled helicopter engine cowling for the cases of FER = 0 and 0.46. Surface oil flow visualization was also performed on the scaled model without exhaust. The oil visualization showed that there were minimal effects on the exhaust plume due to the shape of the model interacting with the freestream. The pipe flow measurements showed the nature of the flow before reaching the lobed mixer and exhaust

duct. The formation of a counter-rotating vortex pair was detected for both cases but were opposite in the rotational sense. For $FER = 0$, although no vortex cores were identified in the plume, the curvature of the exhaust duct and shape of the plume suggested that the CVP was indeed formed inside the duct, but was weak and dissipated quickly upon leaving the duct. For $FER = 0$, the plume formed in a “kidney” shape, before expanding and dissipating downstream. For $FER = 0.46$, the plume formed into a slenderer and more elongated “reversed-C” shape as compared to the case when $FER = 0$. It also retained its shape further downstream and maintained its relative position. The flow structure changed significantly with and without the freestream. The presence of the CVP in the wake allows the plume to retain its shape further downstream. In comparison, the plume for the case with no freestream expanded continuously and moved away significantly as it progressed downstream. This would have implications on the trajectory of the plume, as well as the temperature distribution, and consequently the heating effect on the tail boom.

Acknowledgments

The authors would like to acknowledge Mr. Cheawchan for his MATLAB code and processing of the vortex core locations and strength.

Declaration of Conflicting Interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This work was supported by Leonard-Finmeccanica Helicopter division.

References

1. Yeter E. Thermal fatigue analyses of riveted structures. *Mechanika* 2018; 24: 689–694.
2. Pan C, Zhang J and Shan Y. Effects of exhaust temperature on helicopter infrared signature. *Appl Therm Eng* 2013; 51: 529-538.
3. Presz WM, Jr, Morin BL and Gousy RG. Forced mixer lobes in ejector designs. *J Propul Power* 1988; 4: 350-355.
4. Sheng Z, Liu J, Yao Y, et al. Mechanisms of lobed jet mixing: About circularly alternating-lobe mixers. *Aerosp Sci Technol* 2020; 98: Article 105660.
5. Pan C, Zhang J and Shan Y. Modeling and analysis of helicopter thermal and infrared radiation. *Chin J Aeronaut* 2011; 24: 558-567.
6. Pan C, Zhang J, Ren L, et al. Effects of rotor downwash on exhaust plume flow and helicopter infrared signature. *Appl Therm Eng* 2014; 65: 135-149.
7. Cheng W, Wang Z, Zhou L, et al. Infrared signature of serpentine nozzle with engine swirl. *Aerosp Sci Technol* 2019; 86: 794-804.
8. Shan Y, Pan C and Zhang J. Investigation on incompressible lobed mixer–ejector performance. *J Propul Power* 2014; 31: 265-277.
9. Toulmay F. Internal aerodynamics of infrared suppressors for helicopter engines. *J Am Helicopter Soc* 1988; 33: 4-14.
10. Zhang J, Pan C and Shan Y. Progress in helicopter infrared signature suppression. *Chin J Aeronaut* 2014; 27: 189-199.
11. McCormick DC and Bennett JC. Vortical and turbulent structure of a lobed mixer free shear layer. *AIAA J* 1994; 32: 1852-1859.
12. Elliott J, Manning T, Qiu Y, et al. Computational and experimental studies of flow in multi-lobed forced mixers. In: *28th joint propulsion conference and exhibit*, Nashville, TN, 6-8 July 1992, Virginia: American Institute of Aeronautics and Astronautics (AIAA).
13. De Gregorio F, Pengel K and Kindler K. A comprehensive PIV measurement campaign on a fully equipped helicopter model. *Exp Fluids* 2012; 53: 37-49.

14. De Gregorio F. Flow field characterization and interactional aerodynamics analysis of a complete helicopter. *Aerosp Sci Technol* 2012; 19: 19-36.
15. Shaw CS and Wilson JC. *Wind tunnel investigation of simulated helicopter engine exhaust interacting with windstream*. Report no. NASA TM X-3016, 1974. Washington, D.C.: National Aeronautics and Space Administration (NASA).
16. Shaw CS and Wilson JC. *Wind tunnel investigation of a simulated gunship helicopter engine-exhaust-windstream interaction*. Report no. NASA TM X-3161, 1974. Washington, D.C.: National Aeronautics and Space Administration (NASA).
17. Williams JW, Maurice N. *Aerodynamic interference effects with jet-lift V/STOL aircraft under static and forward-speed conditions*. Report no. Tech. Rep. No. 66403, 1966. British Royal Aircraft Establishment (RAE).
18. O'Brien D, Calvert ME and Butler SL. An examination of engine effects on helicopter aeromechanics. In: *AHS specialist's conference on aeromechanics*, San Francisco, CA, 23-25 January 2008, Virginia: American Helicopter Society International.
19. Freeman CE and Mineck RE. *Fuselage surface pressure measurements of a helicopter wind-tunnel model with a 3.15-meter diameter single rotor*. Report no. NASA TM 80051, 1979. Washington, D.C.: National Aeronautics and Space Administration (NASA).
20. Mineck RE and Gorton SA. *Steady and periodic pressure measurements on a generic helicopter fuselage model in the presence of a rotor*. Report no. NASA TM-2000-210286, 2000. Washington, D.C.: National Aeronautics and Space Administration (NASA).
21. Moffat RJ. Describing the uncertainties in experimental results. *Exp Therm Fluid Sci* 1988; 1: 3-17.
22. Teo ZW, New TH, Li S, et al. Wind tunnel testing of additive manufactured aircraft components. *Rapid Prototyp J* 2018; 24: 886-893.
23. Teo ZW, Wong WH, Mohamed MA, et al. On the flow measurements of simulated helicopter exhausts. In: *AIAA Scitech 2020 forum*, Orlando, FL, 6-10 January 2020, American Institute of Aeronautics and Astronautics (AIAA).
24. Tong A. Improving the accuracy of temperature measurements. *Sens Rev* 2001; 21: 193-198.
25. Lu FK. Surface oil flow visualization. *Eur Phys J Spec Top* 2010; 182: 51-63.

26. Maltby RL. *The surface oil flow technique*. Report no. AGARD-AG-70, 1962. North Atlantic Treaty Organization (NATO), Advisory Group for Aeronautical Research and Development (AGARD).
27. Merzkirch W. *Surface flow patterns*. Report no. AGARD-AG-302, 1987. North Atlantic Treaty Organization (NATO), Advisory Group for Aeronautical Research and Development (AGARD).
28. Settles GS. Modern developments in flow visualization. *AIAA J* 1986; 24: 1313-1323.
29. Prasad AK. Stereoscopic particle image velocimetry. *Exp Fluids* 2000; 29: 103-116.
30. Scheimpflug T. *Improved method and apparatus for the systematic alteration or distortion of plane pictures and images by means of lenses and mirrors for photography and for other purposes*. Patent 1196, GB, 1904.
31. Carpentier J. *Improvements in enlarging or like cameras*. Patent 1139, GB, 1901.
32. Benedetti L, Chen W, Clelland D, et al. *Guideline on best practices for the applications of PIV/SPIV in towing tanks and cavitation tunnels*. Report no. 7.5-02-01-04, 2014. Zürich, Switzerland: International Towing Tank Conference (ITTC) Association.
33. Sommeria J. UVMAT: A MATLAB program for particle imaging velocimetry (PIV), <http://servforge.legi.grenoble-inp.fr/projects/soft-uvmat> (2018, accessed 30 April 2020).
34. Thielicke W and Stamhuis EJ. PIVlab – towards user-friendly, affordable and accurate digital particle image velocimetry in MATLAB. *J Open Res Software* 2014; 2: e30.
35. Keane RD and Adrian RJ. Optimization of particle image velocimeters. I. Double pulsed systems. *Meas Sci Technol* 1990; 1: 1202-1215
36. Mendez MA, Raiola M, Masullo A, et al. POD-based background removal for particle image velocimetry. *Exp Therm Fluid Sci* 2017; 80: 181-192.
37. Sandwell DT. Biharmonic spline interpolation of GEOS-3 and SEASAT altimeter data. *Geophys Res Lett* 1987; 14: 139-142.
38. Berger SA, Talbot L and Yao LS. Flow in curved pipes. *Annu Rev Fluid Mech* 1983; 15: 461-512.

39. Dean WR. XVI. Note on the motion of fluid in a curved pipe. *Philos Mag* 1927; 4: 208-223.
40. Rowe M. Measurements and computations of flow in pipe bends. *J Fluid Mech* 1970; 43: 771-783.
41. Keener ER. *Flow-separation patterns on symmetric forebodies*. Report no. NASA TM 86016, 1986. Washington, D.C.: National Aeronautics and Space Administration (NASA).
42. Lim TT, New TH and Luo SC. On the development of large-scale structures of a jet normal to a cross flow. *Phys Fluids* 2001; 13: 770-775.
43. New TH, Lim TT and Luo SC. Elliptic jets in cross-flow. *J Fluid Mech* 2003; 494: 119-140.
44. Jeong J and Hussain F. On the identification of a vortex. *J Fluid Mech* 1995; 285: 69-94.