An investigation into the interaction of closely-spaced starting jets

D. Rival *, G. Ciccarelli

Department of Mechanical and Materials Engineering, Queen’s University, Kingston, Canada

Received 24 July 2006; received in revised form 12 December 2006; accepted 18 December 2006

Abstract

The transient, three-dimensional scavenging flow inside a novel two-stroke engine has been investigated both experimentally in a scaled water model as well as numerically using a commercial CFD code incorporating an unsteady Reynolds averaged Navier–Stokes (URANS) formulation. The scavenging flow consists of 16 round jets in close proximity of each other and the cylinder wall, developing from the top of the combustion chamber down towards the exhaust ports located along the wall at the bottom of the cylinder. Flow visualization of the scavenging flow was performed using a scaled fixed-piston water model and was used as a means of validating the URANS simulations themselves. The flow visualization experiments provided insight into the complex jet–jet and jet–wall interactions within the engine cylinder. These interactions were not as well predicted by the CFD simulations. In fact, the CFD simulations were found to significantly under-predict the turbulent mixing between the jets. This suggests that unsteady-RANS formulations are incapable of reproducing the large-scale and unsteady mixing structures associated with the vortex shedding between the closely-spaced jets.

© 2006 Elsevier Inc. All rights reserved.

Keywords: Transient flows; Jet–jet interaction; Jet–wall interaction; Unsteady RANS

1. Introduction

In light of the recent success of URANS simulations in the prediction of periodic shedding flows, i.e. behind a square cylinder [1], there exists a strong level of optimism that robust and efficient URANS techniques can predict other such unsteady flows with reasonable accuracy. The effectiveness of such a numerical formulation in the prediction of jet–jet and jet–wall interaction in close proximity, strongly governed by a similar periodic shedding phenomenon [2], has been tested in context of a unique engine cylinder flow. A current push towards high-efficiency hybrid vehicles, has recently led to the development of a novel uniflow-type scavenged two-stroke engine in which the combustion products are purged with fresh intake air through a set of passive check valves situated in the cylinder head. The fresh intake air is driven by a constant-pressure intake plenum located above the combustion chamber. Prior to this scavenging process, blowdown is initiated when the piston uncovers the exhaust ports at the bottom of the cylinder. Immediately after, the intake check valves open as the cylinder pressure reaches a pressure similar to the air plenum pressure – marking the start of the scavenging process. Of interest to this particular investigation is this scavenging flow path, depicted in Fig. 1, in which the added fresh air displaces the exhaust products out of the cylinder via the exhaust ports. Mixing of fresh air and combustion products at the jet boundary occurs through the scavenging cycle resulting in small levels of intake air short-circuiting. The current configuration used in the developmental engine consists of 16 intake valves in close proximity. For this particular study, this intake valve configuration has been simplified to 16 equivalently-sized round jets, as shown in Fig. 2. During a scavenging period on the order of milliseconds (depending on engine speed), these 16 intake jets develop axially down through
the cylinder, purging the combustion products before the start of the next cycle. This flow field is unique and contains the superposition of many complex flows including: confined free jets and wall jets, the interaction of multiple axisymmetric jets and finally, the starting effects of axisymmetric jets.

2. Background

The aerodynamic flow pattern associated with an axisymmetric confined jet differs greatly from that of its equivalent free jet counterpart. The major difference between the two flows is the large recirculation zone developed around the confined jet, in which the entrained secondary flow is restricted by the geometry of the duct walls. Such steady confined flows are of great interest in the development of combustors and furnaces alike. An early experimental study undertaken by Becker et al. [3] on the mixing of a single confined jet showed that the turbulent mixing occurs in three distinct zones. The first zone, also found in an equivalent free jet, is termed the true jet-mixing zone in which the jet grows laterally, ending when the jet reattaches onto the duct wall. The second zone, downstream of the true jet-mixing zone, contains the lateral dissipation of the mean velocity gradients. Finally, in the third zone, the mixing is analogous to that of a fully-developed pipe flow. A more recent study by Gould et al. [4] examined in more detail the true jet-mixing zone using optical access ports and laser Doppler velocimetry (LDV), allowing for measurements of mean velocities and Reynolds stresses across a range of axial planes in the duct. Subsequently the data was used to validate a CFD simulation incorporating a $k-e$ turbulence model. Agreement was found to be rather good except in the region where the turbulent mixing layer reached the end of the potential core (at approximately three jet diameters). Discrepancies were caused by the Boussinesq approximation’s failure in predicting anisotropic turbulence, such that the axial turbulent normal stresses were under-predicted while the radial turbulent normal stresses were over-predicted. With regard to another aspect of the unique scavenging flow, Anderson et al. [2] investigated planar jet–jet interaction with extreme close-spacing ($1.5 < S/w < 2.67$), where $S$ is the jet spacing and $w$ is the jet exit width. For these configurations a periodic, bluff-body-type shedding behaviour was detected between the two jets, where the spacer, of width $d = S - w$, served as the bluff body. It was not surprising then that the near-field flow exhibited both jet-like and wake-like characteristics. Further measurement of the periodicity in the wake was made using hot-wire anemometry. Above a jet spacing of $S/w = 3$, no periodicity was detected – hence no vortex shedding. As the spacing was decreased, however, the Strouhal number itself decreased from 0.35 to an asymptote value of approximately 0.20–0.24, which agrees with vortex shedding Strouhal numbers corresponding to D-shaped bodies in wind tunnel experiments. As an extension to jet–jet interaction between two planar jets, studies investigating multiple-jet interactions exist in the literature, often as applied to cold-model aerodynamic simulations of industrial gas burners. One such study of particular interest to this research project, performed by Yimer et al. [5], examined the fields of mean velocity and velocity fluctuation intensity for a variety of close proximity, multiple-round-jet configurations. It was found that beyond the near field, the flow from all the burner configurations tested developed into an equivalent round-jet flow emitting from a single source. Only in the near field did unique structures associated with individual test configurations observed. Even with the jet configuration containing the closest jet spacing, the fully merged single jet did not form until approximately 16 individual jet diameters downstream of the burner. In other words, the merging of the jets requires that, even for the configuration with the closest jet spacing, a significant distance in the axial direction – a distance much larger than that of the cylinder stroke length associated with this study’s novel two-stroke engine.

As the scavenging flow of interest must develop in a very short time span associated with the engine’s scavenging
phase (as short as 2 ms at 9000 rpm), the scavenging process will be dominated by starting effects of axisymmetric jets. Unfortunately, little exists in the literature on the development of transient jets. By far the most comprehensive examination of a round jet starting vortex was performed by Maxworthy [6] in which he devised a flow visualization experiment where a small piston was used to generate a short fluidic pulse. Through dye injection, this short pulse provided sufficient momentum to develop a distinct, turbulent vortex ring which could then be tracked as it propagated down through the test section within the encompassing vortex bubble. This study provided insight into the physical mechanisms dictating the development and convection downstream of the round jet starting vortex. Specifically, the concentrated vortex ring was generated through the rolling-up of the jet exit shear layer within the separated flow region at the exit of the jet. The vortex ring was then subsequently observed to convect downstream, entrain outer fluid, mix with it, and deposit the majority of this entrained fluid into the wake, thus forming a turbulent starting vortex bubble. In a continuously developing unsteady jet, this starting vortex bubble is formed and then slowly overtaken by the following nozzle fluid. A handful of studies examining the relation between entrainment rates and unsteady jets have been performed. One such study, using pH indicators to determine jet mixing characteristics, was performed by Kato et al. [7], where the vortex bubble was found to enhance mixing of the jet both in the axial and radial directions. Another study, this time quantifying jet entrainment using LDV, was performed by Bremhorst and Hollis [8]. Here, the unsteady jet was pulsed with a significant no-flow period between pulses. The entrainment rate was found to be more than double that of an equivalent steady jet, primarily because of the higher production of Reynolds shear stresses at the leading vortex bubble.

There are two main objectives to this study. The first and foremost goal is to qualitatively understand the mixing process associated with multiple, confined round jets in close proximity. The second objective is to verify the effectiveness of a URANS formulation in the prediction of such a complex scavenging flow associated with the novel two-stroke engine described in Fig. 1. An experimental water model has been constructed to conduct flow visualization. In order to verify the effectiveness of the CFD simulations, validation tests of the complex scavenging flow have been performed using the aforementioned experimental water model.

3. Experimental tests and numerical simulations

Transient and isothermal (room temperature), fixed-piston CFD simulations were performed and compared with equivalent experimental water-model tests; both in turn replicating the actual engine scavenging flow. As described by Stuecke et al. [9,10], through careful use of similitude, simple water models can provide qualitative data for simulating engine flows. Operating conditions used in these tests were specifically chosen to correspond with actual operating conditions used in the prototype engine, i.e. engine speeds between 1000 and 9000 rpm and an intake plenum pressure of 2 kPa gauge. By matching the Reynolds, Strouhal and Euler numbers the scavenging period and inlet head for both CFD and experiment were determined to be 0.6 s and 0.245 m of water (2.4 kPa gauge), respectively.

3.1. Experimental tests

An experimental water model was manufactured to geometrically simulate the actual fixed-piston scavenging process. The experimental water model consisted of several transparent acrylic pieces machined on a CNC-mill, which were subsequently bonded together and sealed. In Fig. 3, the cylinder model (17.2-cm bore, 15.3-cm stroke) is shown sitting in an aquarium, which during the course of the scavenging process is also filled with water. The purpose of incorporating the water-filled aquarium around the cylinder was to remove any light refraction associated with the curved surface of the cylinder walls when viewing the scavenging flow normal to the aquarium’s surfaces. A schematic describing the operation of the water-model cylinder is shown in Fig. 4. The cylinder is separated from the inlet water plenum by a plate with 16 3-cm diameter holes (A) distributed in a pattern consistent with the prototype engine, as shown previously in Fig. 2. The seeding liquid representing the scavenging fluid, i.e. water seeded with talcum powder, was isolated from the rest of the water model by a tube above one of the 16 valves using rubber plugs (B), as shown in the schematic. To start the scavenging process these rubber plugs were pulled into the ‘open’ position from above. Subsequently the exhaust duct doors (C), retained in the closed position with springs, sealed with rubber and vacuum grease, and hinged at the bottom of the aquarium, were pulled open via high-strength lines driven by a pair of Bimba Corp. 092-DP double-acting air-cylinders (D). Control of the air-actuators themselves was made through the activation of solenoid-driven air valves (E) supplied by compressed air at roughly 600 kPa (F).
Finally, constant intake plenum pressure was maintained via a larger reservoir of water (G). This larger water reservoir was supported above the over-flow line of the aquarium at a height $H$ such that a constant head (pressure) could be maintained throughout the scavenging period. The timing of the scavenging process, which was held constant at 0.6 s for all tests, was controlled via an IOTech DataShuttle A/D board. The digital 12 V square-wave signal to the solenoid-driven air valves was produced using a signal generator within the DaisyLab 5.6 control and data-acquisition software package.

Flow visualization of the jet-mixing process for Valve 1 (V1) and Valve 4 (V4) was performed for each valve individually so as to compare the development of the transient intake jets to those predicted by the CFD simulations. Valve 2 (V2) and Valve 3 (V3) were not examined due to high levels of light-reflection associated with their location in the cylinder. As a means of capturing quasi two-dimensional images through a particular intake jet of interest, a 4-mm thick light sheet was created through the model devoid of background light while a single Sony DCR-TRV140 NTSC camera acquired pictures normal to this light sheet as shown in Fig. 5. The light sheet itself was created by passing the collimated light from a NEC LT260 light projector with a 2100 Lumens rating through a slit 4 mm in width. The blocking-out of background light was possible by covering the aquarium with layers of black construction paper and then draping black cloth over top. All flow visualization was performed at a 15-frame-per-second (fps) sampling rate, which provided adequate information regarding the development of the intake jets.

### 3.2. Numerical simulations

A commercial CFD package (Fluent 6.1) was used throughout the investigation. URANS-type CFD simulations of the transient scavenging process were performed on a grid- and domain-independent quarter-cylinder mesh consisting of approximately 450,000 cells. The clearance volume was constructed with unstructured cells (approximately 160,000 tetrahedral cells) while the rest of the cylinder, as well as the exhaust duct and exhaust plenum, was meshed with structured cells. All transient simulations used fine time stepping on the order of 0.1 ms in order to maintain solution stability. The PISO algorithm was used for pressure–velocity coupling. As well, the simulations incorporated a full first-order discretization scheme for both time and space.

A full second-order spatial discretization scheme was attempted, including a second-order pressure discretization and subsequently a second-order upwind discretization for all other equations such as continuity, momentum, $k$, $\varepsilon$, energy and scalar transport (species concentration). It was found that the solution would rapidly diverge due to numerical instabilities. In order to achieve a full second-
order spatially-discretized solution, further grid and
time-scale refinements were necessary. However, such
refinement was not feasible due to the limitations on com-
putational resources. Although first-order simulations on
coarse grids are notorious for high-levels of artificial dissi-
pation, first-order simulations incorporating a well refined
mesh (grid independent solution), such as the one used in
this study, will experience negligible levels of artificial
dissipation.

The standard $k$–$\varepsilon$ turbulence model was used for all sim-
ulations as it is considered the most robust turbulence
model for high-Reynolds number flows in complex internal
geometries, as reported by Nallasamy [11]. All aspects of
the transient simulations were configured such that they
matched as closely as possible the experimental water
model. In other words, the simulations used room temper-
ature water to represent both the scavenging fluid as well as
the combustion products. In so doing, no discrepancies
associated with small compressibility effects or similitude
would impede on the validation process. In Fig. 6, the
quarter-cylinder hybrid-mesh is shown with its respective
boundary conditions.

On the intake valve (pressure inlet) boundaries a prede-
efined turbulence intensity ($Tu = 5\%$) and turbulent charac-
teristic scale ($w_{valve} = 30$ mm) were prescribed in order to
compute the appropriate values of the turbulent kinetic
energy ($k$) and the turbulent kinetic energy dissipation rate
($\epsilon$) at the inlet surfaces. However, turbulence measurements
themselves could not be taken in the experimental water
model and so a set of scavenging simulations were per-
formed with different values of inlet turbulence intensity
in order to determine the sensitivity of this value on jet mix-
ing. The most appropriate method of quantifying the effect
of inlet turbulence intensity on jet mixing is by plotting the
scavenging efficiency as a function of time, defined as the
ratio of the mass of fresh charge retained over the total
mass of trapped cylinder charge. Since the scavenging
performance is a strong function of the mixing between
the intake jets and the ‘exhaust products’, accurate model-
ing of the turbulence intensity being convected into the
solution domain would be expected to play a role on the
subsequent mixing. As mentioned previously, the standard
turbulence intensity used in all simulations was $5\%$ while
for this particular sensitivity study a simulation with a
$20\%$ intensity was attempted, representing an extremely
high inlet turbulence condition. In the other extreme, a
completely laminar simulation was performed. In Fig. 7,
one can see that the high inlet turbulence intensity simula-
tion had virtually no effect on the scavenging performance,
thus showing that the bulk of the turbulence (mixing) was
generated in the multiple-jet shear layers. The laminar case
predicts a higher scavenging efficiency since less mixing
allows for more pure displacement of the ‘exhaust prod-
ucts’ out of the cylinder. One can also infer three distinct
scavenging periods from the shape of the scavenging
curves, showing that the scavenging process never reaches
quasi-steady state. The ramp-up constitutes the initial
start-up period of the inlet jets (in the first 0.1 s) followed
by a linear displacement/mixing period (between 0.1 s
and 0.4 s) during which the intake jets drive much of the
‘exhaust products’ out of the cylinder. The third and final
(logarithmic-shaped) scavenging period asymptotes
towards a maximum scavenging efficiency of roughly 95%
at 0.8 s. The beginning of this final period (at approxi-
mately 0.4 s) can be associated with the establishment of
short-circuiting, defined as the point in time when ‘fresh
charge’ flow begins exiting the exhaust ports at the bottom
of the cylinder. Note that in the following discussion the
comparison between experimental and CFD results are
made at two particular time periods, 0.2 s and 0.4 s, repre-
senting the linear displacement/mixing period and the
beginning of short-circuiting, respectively.

![Fig. 6. Quarter-cylinder hybrid-mesh seen with piston fixed at bottom.](image-url)
4. Results

For each experimental test, eight consecutive flow visualization pictures of the jet development were captured. As discussed previously, the three-dimensional jet–jet interaction associated with this scavenging flow is complex. Therefore, as a means of benchmarking the flow visualization experiment, studies examining the single free-jet and wall–jet flows from Valve 1 and Valve 4 respectively, without the influence of the surrounding jets, were also performed. Refer to Fig. 2 for the valve number designations. In Fig. 8a, the left column shows the development of the jet flow from Valve 1 with the influence of the surrounding jets, i.e. all inlet ports are open. The right column shows the corresponding development of the free-jet flow from Valve 1, but with all the other ports blocked. Note that 'negative' images are shown in order to aid with the contrast of the flow visualization. Because of this, regions of over-exposure appear black – this does not necessarily represent any coherent dense structure within the jet flow. Using light intensity as a direct correlation of seeding concentration, as quantitatively performed in the sol-scattered-light technique developed by Becker et al. [3], one can make inferences into the jet development and mixing in the cylinder as a function of time. In the comparison between the multiple-jet flow (left columns) and the single free-jet flow
(right columns), a distinct difference in the jet development can be observed. Although the heads of both jets seem to move downwards at approximately the same speed, the multiple-jet flow shows no signs of the starting vortex bubble associated with free-jet development. In fact, the multiple-jet flow appears highly diffuse and thicker at the port inlet than at the head, contrary to the associated spreading of the single-jet flow. Note that for the single-jet flow, which appears to behave very much like an unconfined free jet, the exit port is close to the center of the cylinder such that very little jet–wall interaction is expected.

With regard to flow visualization of Valve 4, a similar comparison was made between the multiple-jet flow and the single-jet flow (Fig. 8b). Again, the multiple-jet flow is portrayed in the left column, whereas, the single-jet flow of Valve 4, without the influence of the surrounding jets, is shown in the right column. In this comparison, a much different behaviour was observed between the two seeded flows. Although the single-jet flow projects directly downwards along the wall towards the exhaust port, as would be expected from a normal wall jet, the interacting jet flow from Valve 4 is entrained inwards towards the center jet. As well, the development of the multiple-jet flow appears to accelerate much slower, suggesting that the total mass flow rate through the cylinder is limited by the size and geometry of the exhaust ports. In other words, when examining the single-jet flow, the cylinder mass flow rate is low and the viscous losses through the exhaust ports are minimal. However, with all 16 valves flowing, as is the case with the multiple-jet flow, viscous losses in the exhaust ducts are high. Because of this, the jets cannot accelerate as quickly due to the larger pressure drop across the exhaust ports.

Similar to the comparison between the interacting-jet flow from Valve 1 and its respective single-jet flow, the interacting-jet flow from Valve 4 is again much more diffuse and spreads significantly at the port entrance. This highlights the strong effect of the jet–jet interaction and the associated strong levels of turbulence generated by the shearing of multiple jets in close proximity. Also, there appear to be large turbulent structures, on the order of the jet diameters, that aid in the mixing of the multiple-jet flow. These are most likely fed by vortex shedding between the closely-spaced jets, as observed by Anderson et al. [2]. As is the nature of complex turbulent flows the evolution of the jet from experiment to experiment is found to be stochastic. By comparing a set of ensemble-averaged experimental flow visualization images with the corresponding water model CFD simulations, one can qualitatively validate the CFD model which is based on the ensemble-averaged (URANS) governing equations. Using particular reference times at one-third and two-thirds of the way through the scavenging process (0.2 s and 0.4 s respectively), the downward propagation and spreading of the transient jets can be compared. In Fig. 9a and b, the intake flow from Valve 1 is tagged with talcum powder while the rest of the ports emit non-tagged water – thus the full effect of jet–jet interaction is being examined. The same comparison is made in Fig. 9c and d, just with the tagged intake

![Fig. 9. Multiple-jet flow validation for Valve 1 (a) and (b) and Valve 4 (c) and (d) between CFD (left-hand-side of columns) and experimental ensemble-averaged images (right-hand-side of columns) at two time intervals (mirrored over symmetry plane); for CFD images the red (black) and green (white) zones represent pure scavenging water and jet-mixing regions, respectively. (For interpretation of the references in colour in this figure legend, the reader is referred to the web version of this article.)](image-url)
flow from Valve 4 instead. The left-hand-sides shows contours of species concentration where red (black) represents scavenging regions from the RANS-type CFD simulations. Meanwhile, the right-hand-sides shows the ensemble-averaged negative image from the flow visualization of the same propagating jet, just mirrored across the symmetry plane.

Although one must be judicious when qualitatively comparing URANS-type CFD results with ensemble-averaged, instantaneous images from experiment, the comparison in Fig. 9a and b shows that the transient CFD simulations were capable of predicting the general mixing-shape of the inside jet, but not the proper distribution of ‘fresh’ intake charge. For example, the axial jet-propagation distance at both time steps is well predicted by the CFD. However, one can infer from the flow visualization images that there exists a large concentration gradient between the dark (concentrated) region near the jet exit and the grey (diffuse) region below it, whereas with the CFD, no large concentration gradient as such is predicted. Examining individually several runs of the flow visualization experiment for the same given valve showed that the scavenging process, which involved 16 interacting jets in close proximity, was in fact highly stochastic in nature – thus providing a great challenge for URANS formulations.

In Fig. 9c and d, on the other hand, a much larger disparity between CFD and experiment is observed. Again, the same multiple-jet flow occurs just that the intake flow from Valve 4 is tagged instead. In this case the CFD over-predicts the jet development observed experimentally such that the vortex head is well ahead for both time steps. Furthermore, in the flow visualization, the wall jet is entrained inwards away from the wall, almost as if a recirculation zone is situated just below the jet entrance. The CFD simulations are observed to under-predict the mixing between the interacting jets such that all jets develop in a much more structured, independent manner. This deficiency can be attributed to the unsteady-RANS formulation, which has difficulties simulating the vigorous mixing between the closely-spaced jets. Although the unsteady-RANS formulation is capable of capturing certain vertical structures, it cannot simulate the true turbulence in the flow but rather only its statistics. The flow visualization clearly depicts the evolution of turbulent structures on the order of the jet diameter (these are missing in the CFD simulations), which are responsible for most of the mixing between the jets.

Given the rapid availability of more powerful computational resources in the future, large Eddy simulations (LES) or hybrid techniques such as detached Eddy simulations (DES) would be the most appropriate choice to model such a transient, jet-interaction flow. The justification for this statement is that all dominant turbulent structures would be directly and accurately solved while small sub-grid eddies, generally isotropic in nature, would be modeled to conserve computational resources as with a RANS-type formulation. URANS-type formulations lose their competitive advantage over LES or DES when transient simulations dominated by large turbulent structures, as in the case of closely-spaced interacting jets, are being performed. More generally speaking, a steady-state CFD model incorporating a $k$-$\varepsilon$ turbulence model can predict the time-averaged spreading of a single round jet rather efficiently and with relative accurateness, as described by Pope [12], while for transient, multiple-jet flows dominated by large, time-dependent turbulent structures, an unsteady-RANS (URANS) formulation is much less effective.

5. Conclusions

In this study, flow visualization as well as CFD simulations were used to simulate the transient, three-dimensional development of the scavenging flow in a novel, uniflow-scavenged two-stroke engine. Flow visualization results for the development period of these interacting jets show that large turbulent structures, most likely fed by vortex shedding between the interacting jets, diffuse the jets much faster than in the case of their single free- or wall-jet counterparts. As well, the flow visualization tests showed that the leading front of the interacting jets do not contain any sign of a structured vortex bubble. Regarding the comparisons with CFD, strong limitations of the URANS formulation were uncovered in the prediction of interacting jets in close proximity.

References