Assessment of modelling capability for numerical simulations for designing higher efficiency and lower emission systems

F. Gerbino, R. Morgan, P. Atkins, K. Vogiatzaki

University of Brighton, Mithras House - Lewes Road - Brighton - BN2 4AT, UK

F.Gerbino@brighton.ac.uk

Abstract

As the concern for the environment is increasing and the regulations around harmful emissions are becoming more strict, novel Internal Combustion Engine (ICE) concepts are becoming an absolute necessity in order to keep high thermal efficiencies while reducing the pollutant emissions. Moving towards more radical engine designs based on novel thermodynamic cycles comes also with the lack of knowledge and understanding of the physical phenomena involved in the combustion process. As experimental activities have time and cost implication, Computational Fluid Dynamics (CFD) is becoming a promising tool for virtual design of future energy systems. In this paper we demonstrate the role CFD can play in future engine design, using as a basis the Split Cycle engine, a novel ICE concept based on separating the compression and the combustion stages. A prototype, under development by Ricardo Innovation, offers a potential breakthrough reducing both CO_2 emissions and the fuel consumption. The CFD analysis presented here focuses on the injection strategy of such engine, as it differs from conventional Diesel engines and potentially holds the key of the performances of this device.

Keywords: CFD, Split Cycle Engine, Impinging Jet, Crossflow

1. Introduction

During the last two decades, there is an increasing concern in power and transportation sector regarding environmental and energy issues. ICEs are still the main devices used in land transportation, however they are slowly being replaced by batteries in small vehicles. Though, low power densities typical of batteries are not suitable for long range mobile applications, as trucking industry. Due to this, engine manufacturers are focusing on new strategies and design, in order to reduce the pollutant emissions to near zero levels, keeping high thermal efficiency [1]. Concerning the energy balance of ICE, more than half of the input energy is lost as waste heat. Hence, recovering such heat and converting it to useful work would improve the overall engine efficiency up to $45 - 50\%$ [2]. Further improvements to efficiencies require a fundamental change to the ICE cycle. The Recuperated Split Cycle Engine (RSCE) is a concept developed initially at Ricardo in 1908 and the subject of active R&D by Ricardo Innovation for approximately ten years. Thanks to its entirely new approach to combustion, it is a potentially game-changing engine technology, offering significant reductions in engine-out emissions without compromising efficiency . The prototype currently under development by Ricardo Innovation offers a potential breakthrough, reducing CO_2 emissions by approximately 30% and saving 20% in operating fuel costs[3]. Figure 1 shows the split cycle schematic and its stages of operation.

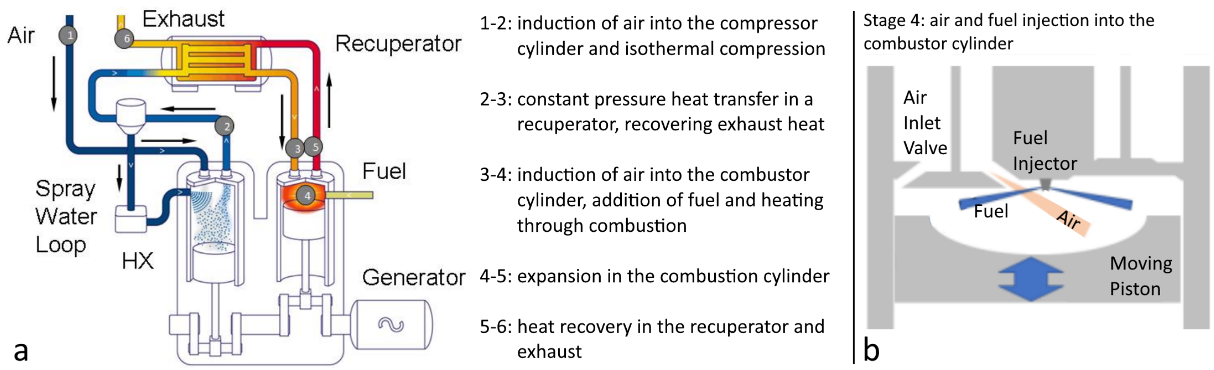

Figure 1: Schematic of the split cycle engine and its stages (a); conceptual representation of the air and fuel injection into the cylinder (b) (adapted from[3]).

Experimental tests for RSCEs are in their early stages. Hence, the actual physical phenomena involved within the combustor, affecting the fuel injection and mixing, have not been widely investigated and understood. Moreover, because of the harsh and extreme environment within the combustion chamber, experimental tests and direct observations are not always feasible. Thanks to the increasing computing capabilities, numerical simulations are the most promising way to study the flows within ICEs, being more cost and time-effective than experimental facilities.

The challenge is that the simulation accuracy depends on the reliability of the virtual tools, requiring rigorous validation over a wide range of operating conditions. The currently available quantitative data are based on lab-scale experiments representing isolated operating conditions, at fixed pressures and temperatures. In a real engine, where the piston is moving, a range of these conditions is present simultaneously. As regards novel ICE concepts, further levels of complexity are present. In fact, within RSCE, an air jet is injected in the combustion cylinder at high pressure and velocity (up to $100-200m/s$ [3]) causing shock waves and impinging the fuel jet. Injecting the fuel against a high velocity air jet causes unusual fuel-air interactions, since in conventional ICEs the fuel is injected in a "static" environment (with very low air velocities). It is expected that the presence of shock waves affects both the mixing and the spray atomization. In the current work we use numerical simulations to provide insight on the physical phenomena involved in these condition. In the next sections we will briefly discuss the numerical set up and the CFD models used, then we will present the numerical results that show how the impinging jet dynamics affect the mixing process.

2. Numerical setup and validation

To keep an acceptable compromise in terms of computational time and accuracy (resembling industrial purpose CFD), the Reynolds Averaged Navier Stokes (RANS)/Eulerian-Lagrangian approach is used here. In such method, the liquid phase is treated using the Lagrangian particle tracking, while the gas phase is modelled using the Eulerian framework. Initially the code has been validated in a simpler configuration against the experimental cases documented by the Engine Combustion Network (ECN)[4] for Spray A in inert conditions. The input conditions are reported in table 1 and they correspond to conditions close to standard Diesel engine conditions. Figure 2a shows the setup for Spray A.

Table 1: Operating condition of spray A

Ambient properties		Spray properties	
Pressure:	$p_{amb} = 6MPa$	Single hole Injector:	$d_{inj} = 90\mu m$
Temperature:	$T_{amb} = 900K$	Fuel:	N-dodecane
Density:	$\rho_{amb} = 22.8kg/m^3$	Injection pressure:	$p_{inj} = 150MPa$
Inert condition:	$0\%O_2$	Inj. temperature:	$T_{inj} = 363K$

After a thorough sensitivity analysis, the optimum mesh cell size for RANS simulations is found to be $0.25mm$ [5]. As regards the spray droplets atomization, the Reitz-KHRT break-up model is used, as it is the most commonly used for diesel-like fuels as N-dodecane [6]. Both the optimum mesh size and the break-up model coefficients, have been selected, validating

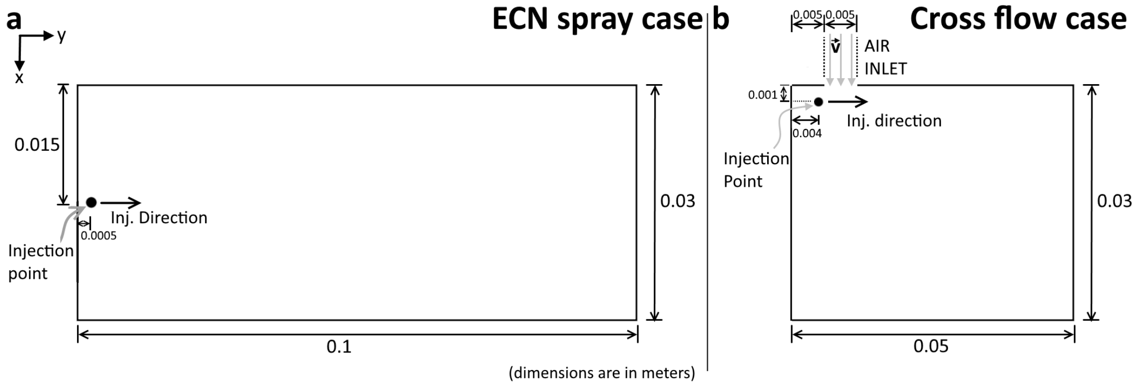

Figure 2: Schematics for the simulations of: ECN spray A case (a) and the crossflow case (b).

the results against the ECN experimental data. Such numerical set up is used here for the investigations on the impinging jet, as no experimental data are available for such case. The air inlet is represented by a square hole of $5\text{x}5mm$ and three different cases are compared varying the air inlet velocity, setting $\mathbf{v} = 0$ (no cross-flow, similar to ECN Spray A), 30 and $100m/s$. The injection point is moved closer to the air inlet. Figure 2b shows the schematics of the crossflow case. Notice that $\mathbf{v} = 10 - 30m/s$ are typical velocities observed in conventional ICEs, while in RSCE velocities in the range of 100 to $200m/s$ are predicted [3].

3. Results and Discussion

Comparison between the three cases, varying the air inlet velocity, shows that although there aren't major differences between the case with 0 and $30m/s$, the effects of the crossflow became much more important as $\mathbf{v} = 100m/s$. Figure 3a demonstrates the fuel vapour distributions for $t = 0.0005s$ (i.e. as the liquid jet reach its stationary length). As $\mathbf{v} = 100m/s$ the vapour mass fraction gets more dispersed, and it is possible to observe some oscillations on the vapour front facing the air jet. These oscillations can be better explained observing the pressure field.

Looking at the pressure field in Fig. 3b, the first two cases are similar and pressure oscillations (potentially shock waves) appear around the injection point. On the other hand for $\mathbf{v} = 100m/s$ a noticeable pressure rise occurs along the entire leading edge of the liquid jet, followed by a much lower pressure region. Such a behaviour indicates that as the crossflow contacts the liquid jet a shock wave occurs in that region. Pressure waves can be observed as well further up the liquid jet tip, explaining the oscillations in the vapour mass fraction field. Another parameter shown here, relevant for the fuel-air mixing, is the vorticity. Once again, there aren't major differences between the case without crossflow and with $\mathbf{v} = 30m/s$. In fact, looking

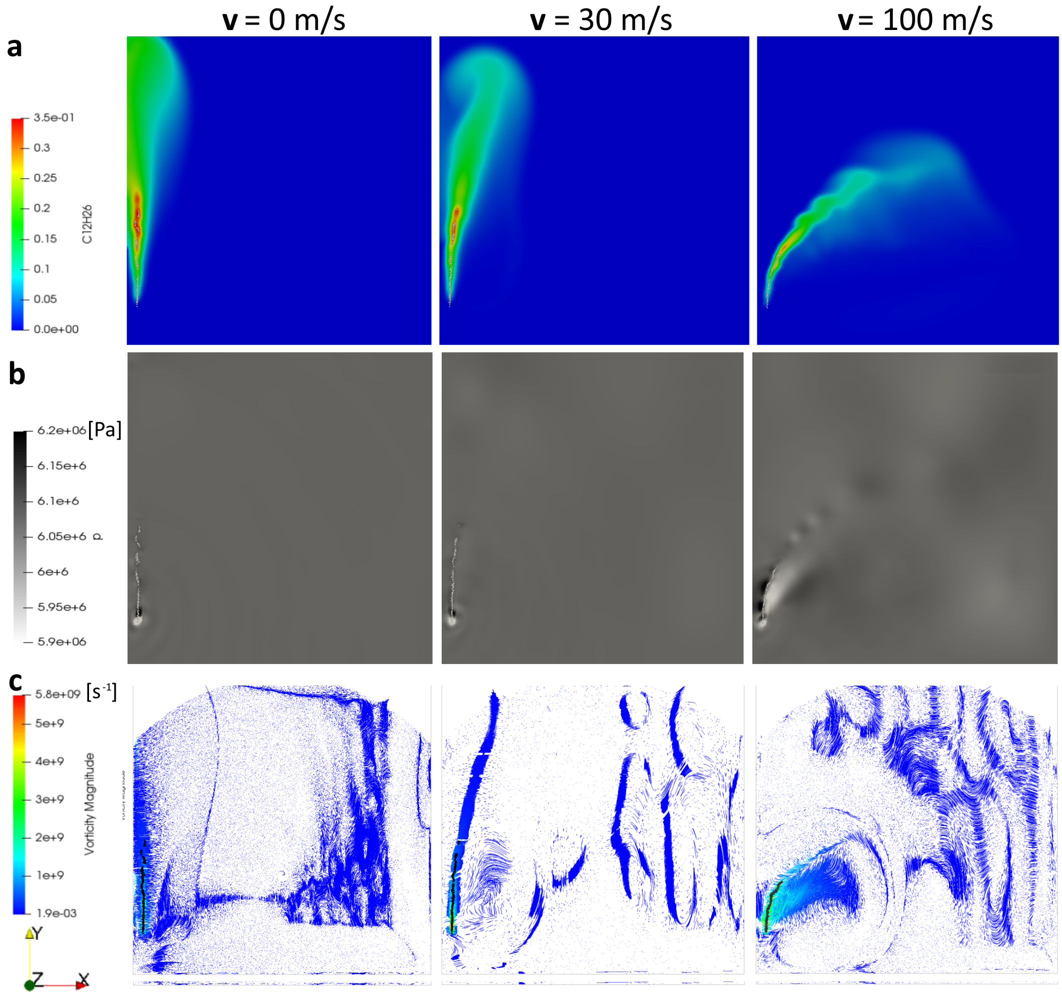

Figure 3: Simulations outcomes for different air inlet velocity at $t = 0.0005s$: fuel vapour mass fraction (a); pressure field (b); vorticity streamlines (c)

at Fig.3c which reports the vorticity streamline, it can be seen that as the crossflow velocity is 0 and $30m/s$ the vorticity is kept confined around the liquid jet (the black particles), while as $\mathbf{v} = 100m/s$ it extends downstream the jet, indicating a higher mixing occurring in that region.

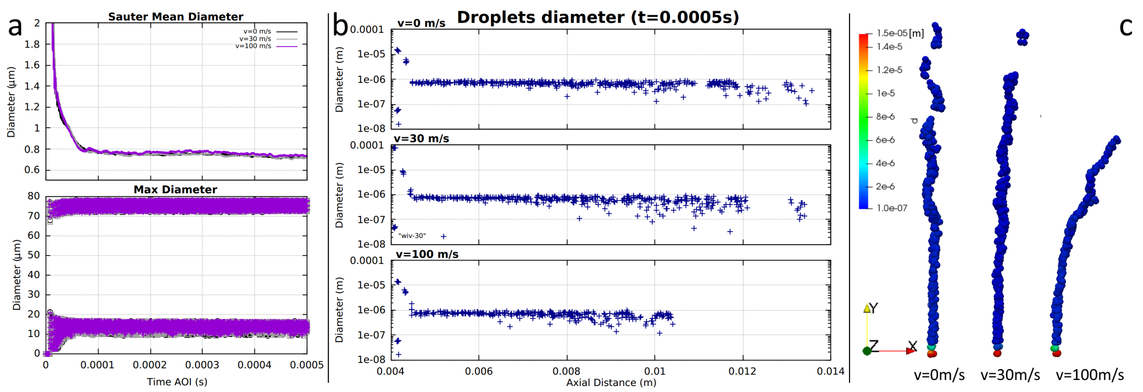

Figure 4: Simulations outcomes for the Lagrangian particle: Sauter mean diameter and maximum diameter as a function of time (a); droplets diameter distribution at $t = 0.0005s$ for different $\mathbf{v}$ (b); liquid jet morphology at $t = 0.0005s$ for different $\mathbf{v}$(c).

Notice that these effects are observed on the Eulerian field as the crossflow velocity gets higher, but there aren't major effect on the the liquid parcels, with the exception of their trajectory deviation, as shown in Fig.4. Analysing the spray droplets diameter, the droplets size distribution remain the same for all the cases (Fig.4a and b). Moreover Fig.4c shows there aren't ligaments or smaller droplets on the trail of the liquid jet, downstream the crossflow (as proposed in [7]). This indicates a certain inadequacy of the Lagrangian approach used here for the liquid jet characterization, in particular as regards the spray break-up model. In fact, the Lagrangian particle behave, to some extent, independently from the surrounding Eulerian field.

4. Conclusion

In this paper we demonstrate how a virtual design tools can support the development of future engine concepts, also when experimental data are not available, using a case relevant to industrial interest (a novel ICE concept). Numerical simulation performed here prove that, in conditions relevant to the RSCE, unusual physical phenomena may affect the fuel-air mixing and the spray atomisation process, with respect to conventional injection conditions. The presence of a high velocity impinging jet implies the occurrence of shock waves, as well as a more distributed viscosity field, which extends further downstream the jet region. The numerical coupling between the Lagrangian and the Eulerian framework has to be improved.

References

1 . Stanton DW (2013) Systematic Development of Highly Efficient and Clean Engines to Meet Future Commercial Vehicle Greenhouse Gas Regulations. SAE Int. J. Engines 6(3):1395-1480

2 . Sprouse C, Depcik (2013) Review of organic Rankine cycles for internal combustion engine exhaust waste heat recovery. Appl Therm Eng 51(1–2):711–22

3 . Morgan R et al. (2019) The Ultra Low Emissions Potential of the Recuperated Split Cycle Combustion System, Submitted to SAE ICE Conference, Capri

4 . Engine Combustion Network, http://www.sandia.gov/ecn/

5 . Gerbino F, Morgan R, Atkins P, Vogiatzaki K (2019) Quantifying the effect of the turbulence and break up modelling for spray simulations through a dimensionless number analysis, Submitted to International Journal of Engine Research, IJER-19-0097

6 . Baumgarten C, Mewes D, Maxinger F (2006) Mixture Formation in Internal Combustion Engines, Lehrbuch, Springer Verlag, Heidelberg

7 . Lia X, Gao H, Soteriou M C (2017) Investigation of the impact of high liquid viscosity on jet atomization in crossflow via high-fidelity simulations, Physics of Fluids 29, 082103