Investigation on the removal of the cavitation erosion risk in a control orifice inside a prototype diesel injector

Maxwell Brunhart[1,2], Celia Soteriou[1], Christian Daveau[1], Manolis Gavaises[2]
Phoevos Koukouvinis[2] & Mark Winterbourn[1]

[1]*Delphi Technologies, Gillingham, UK:* [2]*City, University of London, London, UK*

Abstract

A CFD investigation is in progress to study the cavitation characteristics and potential erosion risks of a control orifice in a prototype injector. An early design of the orifice resulted in cavitation erosion after endurance testing. A design modification eliminated the erosion and subsequent prototypes were free from damage. Initial results for the two designs using different simulation methods are discussed, along with the effects of different rates of evaporating and condensing mass transfer. Preliminary findings on possible erosion risk indicators comparing the eroding with the non-eroding design are presented.

Keywords: cavitation, erosion risk, control orifice

Introduction

In some cases, the collapse of cavitation can lead to erosion of high strength metals, and thereby, damage system components. Being able to predict if and where such erosion is likely to occur would be beneficial to many fields, like automotive and naval. Currently, there is no accurate and consistent method to predict cavitation erosion. The diesel injector is a common subject in cavitation-based research as cavitation in the flow cannot be avoided, but it does not necessarily cause erosion. Moreover, the amount of cavitation does not necessarily indicate the level of erosion risk: internal work in Delphi Technologies has shown that in some cases increasing the amount of cavitation can eliminate cavitation erosion. This complexity underlines the need for the cavitation phenomenon to be understood and managed.

The injector is of critical importance to the performance of engines. It is an area for continued development to meet future environmental and performance standards. Hence, an understanding of the fundamental behavior of hydraulic control orifices is needed to optimize the performance and production while avoiding any potential complications due to cavitation or cavitation erosion.

The control orifice of interest in this work is a spill orifice (Fig. 1) in an automotive high-pressure fuel injector. It is one of the orifices involved in controlling the rate of opening and closing the needle valve during an injection. All

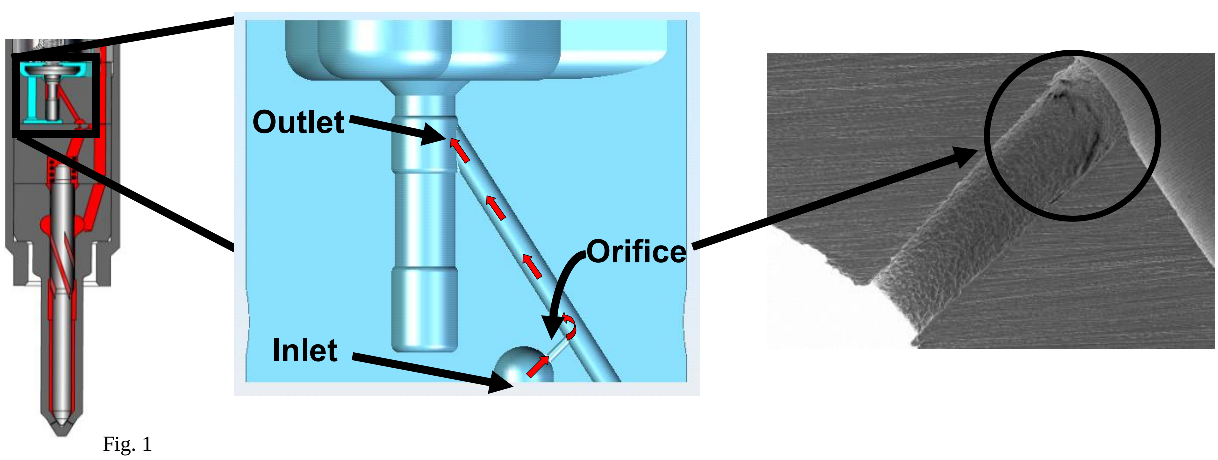

Fig. 1
Left: Location of the control orifice in a sketch of a diesel injector.
Center: General geometry used in the simulation showing flow direction and regions of interest.
Right: An early prototype after an endurance test. Specimen shows signs of cavitation erosion in the region circled.

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk

prototype designs are subjected to extensive testing during product development, including endurance tests. During development one such endurance test resulted in significant cavitation erosion of the orifice. Further development of the component eliminated the erosion and subsequent prototypes were free from damage.

A CFD investigation is in progress to understand the flow-field and cavitation characteristics in and around this orifice, and to explore possible indicators for cavitation erosion risk. The analyses used two design levels: the original geometry which suffered from erosion (E – eroding geometry) and a subsequent prototype design which eliminated the erosion (NE – non-eroding geometry).

The CFD study includes initial simulations using Reynolds-Averaged Navier-Stokes (RANS) turbulence modelling and the standard Zwart-Gerber-Belamri (ZGB) cavitation model. For increased accuracy and detail, this work was then expanded upon with a hybrid LES-RANS turbulence model: Detached Eddy Simulation (DES). This type of model has shown improvement over RANS models [1] [2] [3]. However, in industry the RANS method is still a useful CFD investigation tool, especially considering the heavy runtime cost of LES/DES methods. The DES simulations were first run using the standard ZGB model. Then a modified ZGB method was used, by means of a UDF (user-defined function) which enabled significant changes to be specified to the rate of mass transfer [4]. In this work, the magnitude and location of peak pressures and peak rates of pressure change (material derivative of pressure: DP/Dt) resulting from cavitation collapse are examined.

The paper provides some preliminary results of this on-going project, commenting on the effects of the different simulation methods and of the two different designs. The CFD investigation is backed up with photographic evidence of damage on an early prototype component. Finally, some initial findings on indicators for cavitation erosion risk are presented. This work forms part of the EU funded Marie Sklodowska-Curie Innovative Training Network (ITN) MSCA-ITN-2014-ETN.

CFD Simulation

To simulate cavitation in the flow, a mixture model, which assumes that the two fluids travel as a homogeneous multiphase flow [5], was utilized. The mass transfer between liquid and vapor is governed by the vapor transport equation [5]:

$$\frac{\partial(\alpha\rho_v)}{\partial t} + \nabla \cdot (\alpha\rho_v \boldsymbol{U}_v) = R_e - R_c \qquad \text{Eq. 1}$$

where ρ is density, α is vapor volume fraction, $\boldsymbol{U}$ is velocity field, v denotes vapor and R_e and R_c denote rates of evaporating and condensing mass transfer, respectively. First a standard, and then a modified form of the ZGB cavitation model was applied in the simulations. The ZGB model uses the following rates of mass transfer, first described by Zwart et al [6]:

$$R_e = F_e \frac{3\alpha_{nuc}(1-\alpha)\rho_v}{\Re_B} \sqrt{\frac{2}{3}\frac{(P_v - P)}{\rho_l}} \qquad \text{Eq. 2}$$

$$R_c = F_c \frac{3\alpha\rho_v}{\Re_B} \sqrt{\frac{2}{3}\frac{(P - P_v)}{\rho_l}} \qquad \text{Eq. 3}$$

where α_{nuc} is nucleation site volume faction, $\Re_B$ is bubble radius, ρ is density, P is pressure, F_e and F_c are constants and v and l denote vapor and liquid respectively. In Fluent the default values for the ZGB model constants are 1e-6 m for $\Re_B$, α_{nuc} is 5e-4, F_c is 0.01 and F_e is 50. These constants can be grouped as follows:

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk

$$F_{evap} = F_e \frac{3\alpha_{nuc}}{\Re_B} \quad \text{Eq. 4}$$

$$F_{cond} = F_c \frac{3}{\Re_B} \quad \text{Eq. 5}$$

resulting in F_{evap} equalling 75,000 m^{-1} and F_{cond} equalling 30,000 m^{-1} with the default values.

The modified cavitation method was implemented in a UDF which comes from the work of Koukouvinis et al [4]. For this, the ZGB model is implemented with the value of the mass transfer constants defined to create an effect closer to that of a barotropic model. A barotropic model is approached asymptotically as the mass transfer rates tend towards infinity. For this work, a large but practical value of 1e8 for both F_{evap} and F_{cond} was used.

The initial RANS CFD simulations used the k-epsilon method alongside the standard ZGB cavitation model. The RANS simulations were used for the low computational costs and previously seen accuracy in predicting the Cd (coefficient of discharge) when implemented with due care ([7] & internal documents). Initial flow details were noted with the RANS simulations, but they were unable to display detailed transient behaviour that is likely needed in developing cavitation erosion risk assessment.

Further to that, the entire injection cycle experienced by the orifice was modelled in URANS (Unsteady RANS) simulations. This was achieved by inputting the time-varying upstream and downstream pressure traces for the whole injection cycle. These traces were obtained from an established 1D hydraulic model of the injector of which the overall performance had been well validated with measurements from hardware. The URANS results did show some oscillation movement in the region of cavitation but did not add significantly to the steady state simulations. A full injection cycle with DES would have an extremely long runtime but may be done later when more processing power is available. Hence, with the current computational power, transient DES simulations using constant pressure conditions were carried out and are reported here.

The DES technique is an LES-RANS hybrid, implementing RANS at the boundaries and an LES model elsewhere [8]. This makes for a simulation that is less computationally intensive than full LES while still producing the higher level of detail at the areas of interest. The simulations were implemented in ANSYS Fluent v17 as IDDES (Improved Delayed Detached Eddy Simulation), as first proposed by Shur et al [8]. IDDES is a more recent version of DES that provides more flexibility and convenience for high Reynolds number flows. This setup was previously validated with experiments for a similar operating environment by Bush et al [1] and is also congruent with Koukouvinis et al [4].

The simulations were run with constant boundary conditions using an upstream pressure of 1380 bar and downstream pressure of 340 bar. These values provided the highest pressure difference across this orifice during a 2200 bar injection cycle. As the pressure drop is relatively high and the flow velocity was expected to be high, a compressible fluid was used. The fluid is representative of Normafluid (ISO4113), a standard fluid used in automotive testing. The vapor is assumed incompressible for computational simplicity.

For both designs, the entrance to the orifice is slightly rounded, the diameter is in the region of 0.22mm and there is a small divergent taper along the length of the hole. Although highly significant in terms of design, the differences between the E and NE prototype geometries are not great, with the taper for the NE geometry being slightly less, that is, closer to zero. To start with, CFD simulations were tested against manufacturing pass-off flow tests to ensure the CFD geometry of each design was an accurate replica of the hardware, along with a mesh study that found that further mesh refinement had little effect.

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk

Experimental Tests

Newly proposed injector designs must all go through a rigorous testing process. One of these tests is an endurance test, which consists of exposing the prototype injector to a high typical load cycle (2200 bar injection cycle) for x-many cycles. This type of test is essential in determining if a prototype can withstand normal use and is ready for serial production. Examples of the damage following endurance testing can be seen in Fig. 1 and Fig. 2. The material used was automotive high strength steel with surface treatment.

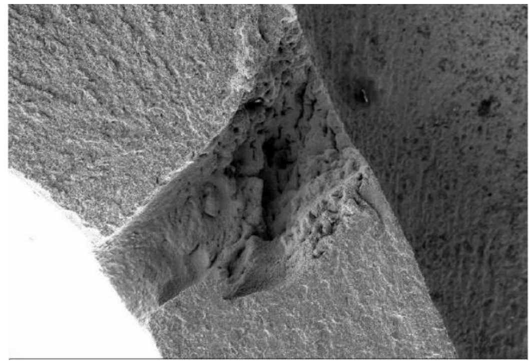

Fig. 2- Another early prototype sample showing further signs of cavitation erosion after an endurance test. Subsequent design modifications eliminated the erosion.

Results

Table 1- List of the different simulations sets and results overview

Case	Description	Mass transfer constants	Negative pressure (bar)	General outcome (E vs. NE)
1	RANS with standard ZGB method and default values	Default F_{cond}=3e4, F_{evap}=7.5e4	-200	Region of recirculation and backflow in E geo
2	DES with standard ZGB and default values	Default F_{cond}=3e4, F_{evap}=7.5e4	-200	No obvious difference
3	DES with modified mass transfer UDF	UDF F_{cond}=F_{evap}=1e8	-8	E has significantly more activity in max P and max DP/Dt

Case 1 – RANS, standard ZGB The results for the eroding geometry showed a region of recirculation and backflow at the end of the topside of the control orifice, whereas the non-eroding geometry did not (Fig. 3). This region correlated with the location of erosion seen in the failed endurance test images (Fig. 1 & Fig. 2).

Case 2 – DES, standard ZGB Results showed no substantial differences between the E and NE geometries.

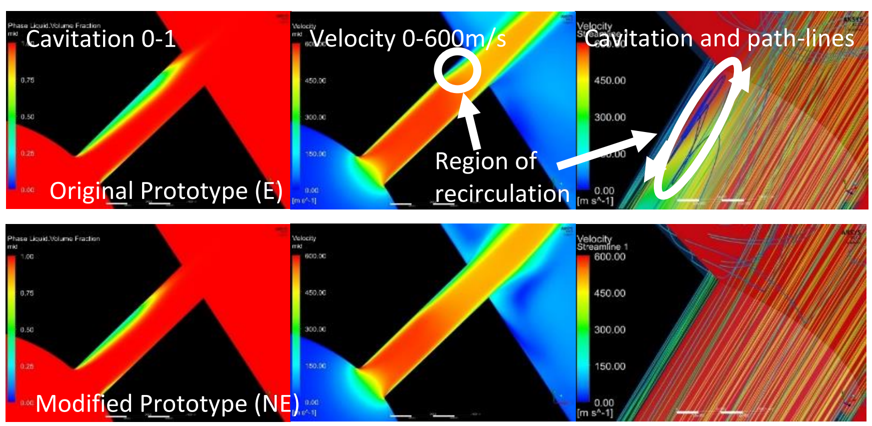

Fig. 3- Mid-plane section of the E and NE RANS simulations (Case 1, RANS with standard ZGB). Region of recirculation and backflow prominent in the E geometry.

Case 3 – DES, increased mass transfer rate - A key improvement using this method was the lack of negative pressure which commonly occurs in cavitating simulations and is a non-physical effect (Table 1). This negative pressure is a numerical inaccuracy and is generally capped at the vapor pressure by users during post-processing. Furthermore, the UDF enables converged solutions with the required increased mass transfer rates. However, the run time of case 3 compared to case 2 was noticeably increased.

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk

The collapse of cavitation vapor bubbles can cause an erosion process. During cavitation collapse near a surface, micro jets and shockwaves are produced. These can create unduly high pressures on the surface which can lead to progressive damage of the component surface and underlying material. Consequently, this work tracked the peak pressures that occur on the surface and throughout the domain of the CFD simulation. However, it should be noted that the limitations in modelling the physics of the cavitation and its collapse influences the accuracy of the actual levels of peak pressure calculated. Peaks in the rate of pressure change, related to power per unit volume, are also examined.

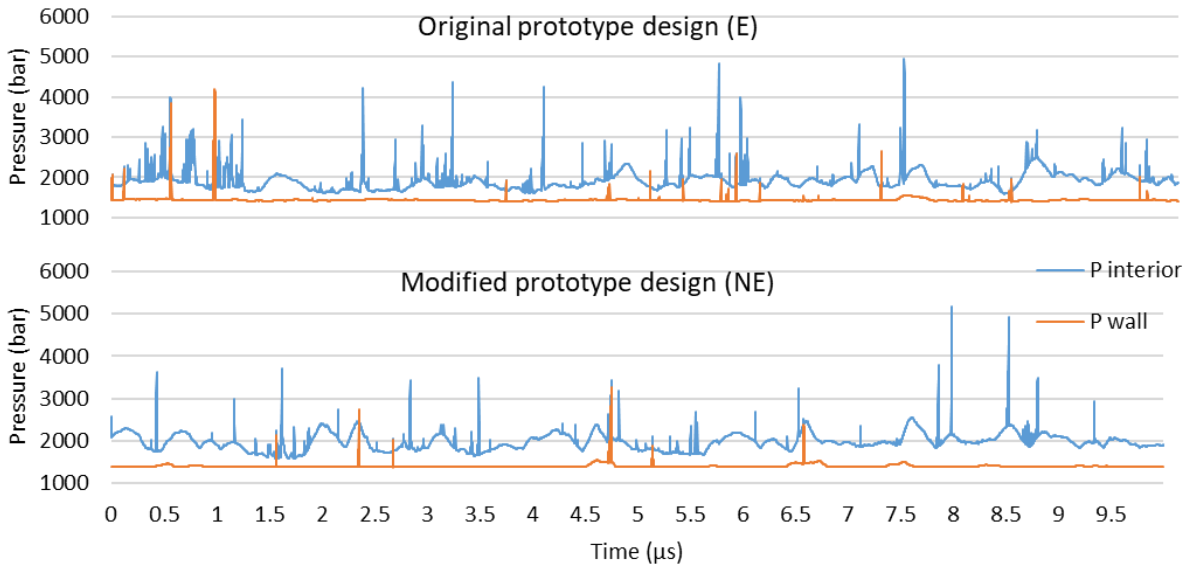

Fig. 4- Case 3 (DES with increased mass transfer rates). Maximum pressure in the entire domain (P interior) and on any surface (P wall) for 10 µs after initialization.

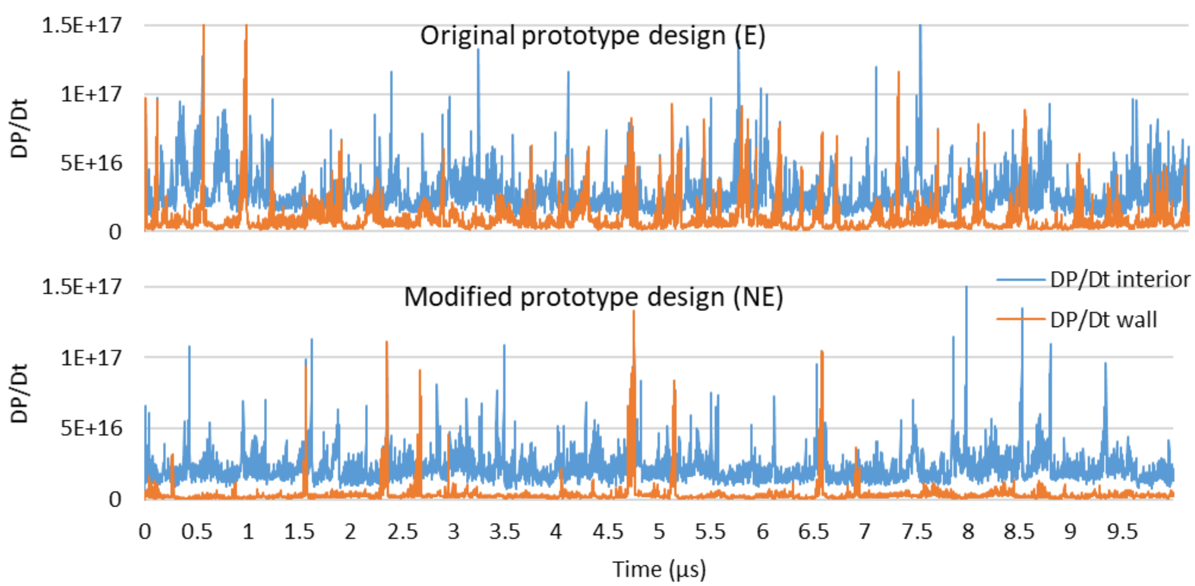

Fig. 5 - Case 3 (DES with increased mass transfer rates). Maximum total derivative of pressure in the entire domain (DP/Dt interior) and on any surface (DP/Dt wall) for 10 µs after initialization.

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk

Fig. 4 shows the maximum pressure computed in the entire domain and on the wall for every time-step for the E and NE geometries. Both geometries experienced pressure spikes in the 5,000 bar range. Other researchers have measured the pressure in the proximity of cavitation collapse and have recorded local peaks in the 10,000 bar range [9]. While both geometries showed high pressure peaks, the E geometry experienced many more. The first total derivative of pressure also shows much more activity for the E geometry, both in the interior and on the wall (Fig. 5). Again, the maxima hit similar values, but the frequency of occurrence is starkly different between the two designs. It should be kept in mind however, that the values shown in Figs 4 and 5 may have occurred anywhere in the CFD domain and on the surfaces. Hence further analysis regarding the location of the activity was necessary.

Differences between the geometries become more apparent when examining the surface of the orifices in the region of erosion. For greater emphasis in the region of interest, Fig. 6 displays peak values for both the square of the pressure and the square of the rate of pressure change. While there is activity in the original (E) geometry in terms of the squared pressure peaks, there is also some in the modified (NE) prototype geometry (Fig. 6). However, when considering the squared DP/Dt peaks, a significant difference is immediately apparent. The original prototype (E) design shows many more maxima occurring, even over the short time data was collected. Most importantly, these peak values are in the regions of erosion seen on the hardware that was damaged in the endurance tests.

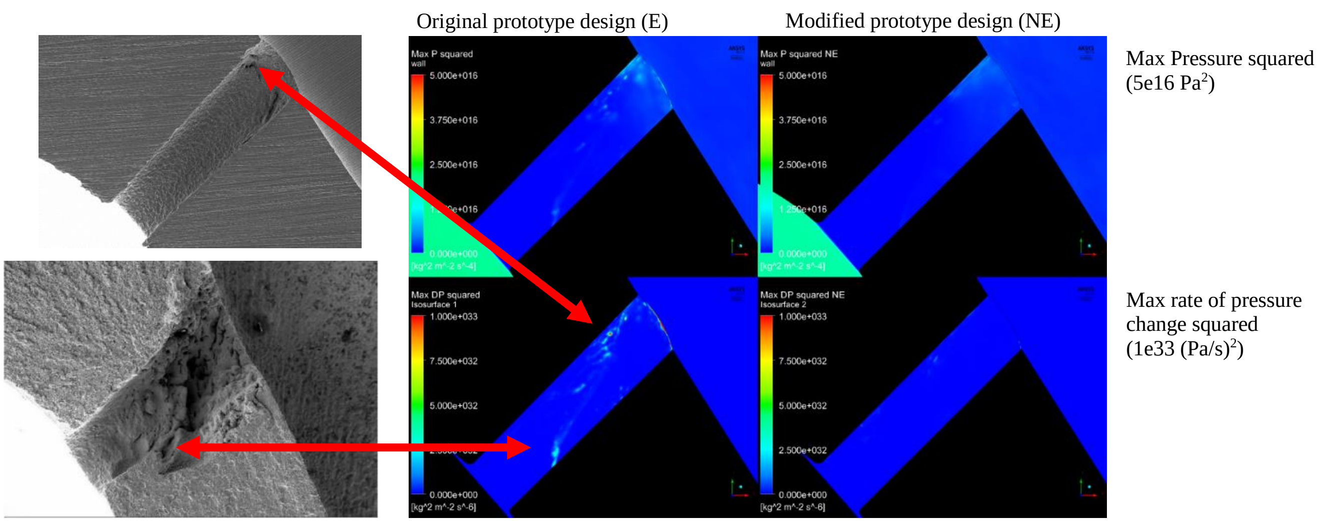

Fig. 6 – Maximum values raised to the second power for pressure and DP/Dt from Case 3 (DES with increased mass transfer rates) CFD simulation. Values recorded over 10µs. The left column shows the early prototype (E) hardware after endurance tests. Maxima locations correlate well with the location of damage on hardware.

Discussion

Currently, there is no accurate and consistent method to predict cavitation erosion in diesel injection components. Cavitation occurring in the flow does not mean there will be erosion. Moreover, the amount of cavitation does not necessarily indicate the level of erosion risk. For this initial study the magnitude and location of the pressure and rate of pressure change, that result from vapor collapses, are examined to define possible erosion risk indicators.

For much of the CFD work performed in industry, the RANS technique for modelling turbulence is still the method of choice due to its acceptable time scale compared with LES/DES techniques. Hence, it is worth noting that the RANS model results (Case 1) showed a significant difference between the E and NE geometries. It is possible this difference was only case specific. However, it is interesting that the difference was in the region of erosion which suggests that backflow and recirculation, close to collapsing vapor structures, may result in an increased risk of erosion.

Employing the UDF for modified mass transfer with the DES method (Case 3) enabled a more accurate and potentially useful result than the DES case without the UDF. The results showed notable differences between the

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk

original prototype design (E) and the modified design (NE). Specifically, the location of peaks in the rate of pressure change correlated well with images of cavitation erosion from endurance test components, more so than peak values of pressure. The results are encouraging, but further work is required before concluding the outcome. This work is in progress and includes running the simulations over longer periods of time, and further data analysis and interpretation. Future work will also include stress analysis using the actual hardware geometry to understand the effect on the metal. These stress analyses will explore the impact of pressure spike levels and associated frequencies, as well as the rate of pressure change.

Acknowledgment

This work is part of the CaFE (Cavitating Flows, surface Erosion damage and material loss) Project, which is sponsored the EU Marie Sklodowska-Curie Innovative Training Network (ITN) MSCA-ITN-2014-ETN, grant agreement number 642536. Furthermore, this work would not be possible without the help of Delphi Technologies and City, University of London. In addition, the contribution of Alexandre Coutton in sourcing information is much appreciated.

References

[1] D. Bush, C. Soteriou, M. Winterbourn and C. Daveau, “Investigating hydraulic control components in high performance injectors,” in *IMechE Fuel Injection*, London, 2015.

[2] B. Befrui, P. Spiekermann, M. A. Shost and M.-C. Lai, "VoF-LES Studies of GDi Multi-Hole Nozzle Plume Primary Breakup and Comparison with Imaging Data," in *ILASS Europe Conference on Liquid Atomization and Spray Systems*, Chania, Greece, 2013.

[3] R. E. Bensow, "Simulation of the unsteady cavitation on the Delft Twist11 foil using RANS, DES and LES," in *Second International Symposium on Marine Propulsors*, Hamburg, Germany, 2011.

[4] P. Koukouvinis, N. Mitroglou, M. Gavaises, M. Lorenzi and M. Santini, "Quantitative predictions of cavitation presence and erosion-prone locations in a high-pressure cavitation test rig," *J. Fluid Mech.*, vol. 819, pp. 21-57, 2017.

[5] Ansys, Inc., "Ansys Fluent Theory Guide," SAS IP, Canonsburg, PA., 2013.

[6] P. Zwart, A. Gerber and T. Belamri, "A Two-Phase Flow Model for Predicting Cavitation Dynamics," in *5th international Conference on Multiphase Flow*, Yokohama, Japan, 2004.

[7] C. Soteriou, M. Smith and R. Andrews, "Diesel injection - laser light sheet illumination of the development of cavitation in orifices," in *IMechE C529/018/98*, 1998.

[8] M. L. Shur, P. R. Spalart, M. K. Strelets and A. K. Travin, "A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities," in *International Journal of Heat and Fluid Flow*, 2008.

[9] T. Okada, Y. Iwai and K. Awazu, "A study of cavitation bubble collapse pressures and erosion part 1: A method for measurement of collapse pressures," *Wear*, vol. 133, no. 2, pp. 219-232, 1989.

[10] C. Soteriou, M. Lambert, S. Zuelch and D. Passerel, "The flow characteristics of high efficiency diesel nozzles with enhanced geometry holes," *Thiesel*, 2006.

Corresponding Author, Maxwell Brunhart: maxwell.brunhart@city.ac.uk