

# Dynamic cavitation inside a high performance diesel injector – an experimental and CFD investigation

**Daniel Bush, Celia Soteriou, Mark Winterbourn & Christian Daveau**

Delphi Diesel Systems, Gillingham, UK

daniel.bush@delphi.com

**Abstract.** A combination of simulation and special experimental techniques has been used to investigate the transient flow and cavitation phenomena of a control device inside a high performance diesel injector. Dynamic cavitation behaviour was captured on a large scale transparent model, which was then used to develop and validate an advanced turbulence CFD model with Large Eddy Simulation. These techniques are used within Delphi to gain insight and optimise injector performance at real-size.

## 1. Introduction

The diesel injector is arguably the most important element in achieving premium engine performance with low emissions. It therefore follows that injectors for high performance engines must be controlled with precision. As a result, the hydraulic control valves and orifices must be designed and manufactured with high accuracy, and understanding their fundamental behaviour will provide an important contribution to the optimisation and fine-tuning process. Delphi uses a combination of simulation and special experimental techniques to investigate the flow characteristics of these devices.

As with all new injectors, a High Performance Diesel Injector under development at Delphi has been the subject of much simulation activity. Of particular interest to this paper, are the CFD investigations of the hydraulic control functions.

Flow cavitation is a familiar topic in diesel injection equipment; previous experimental experience showed that the industry standard Reynolds-Averaged Navier-Stokes (RANS) modelling was unable to give accurate results for dynamic cavitation behaviour. Hence in addition to the CFD work, large scale models (LSM) of different parts of the hydraulic control functions were commissioned. The use of transparent models of control valves and orifices at greatly increased scales enables a developed understanding of hydraulic performance [1] that is not easily achievable with other techniques. The hydraulic performance is sufficiently independent across all scales to be useful as a development tool [2].

In CFD simulation, resolving large scale turbulent structures using Large Eddy Simulation (LES) based models shows transient modelling benefit over RANS simulations [3] [4] [5]. The serious disadvantage of LES based CFD in an industrial environment is the huge computing resource required,



of the order of several hundred dedicated cores. Without such a resource, the run time for a single case can take months, making serious investigations impossible in a sensible time frame.

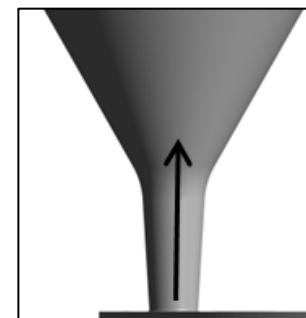
Significant developments have been made within CFD modelling which attempt to combine the benefits of both LES and RANS turbulence modelling [6]. These models are often termed hybrid models and give greater potential for LES to be utilised within an industrial environment.

A hybrid LES-RANS CFD model, using a mixture multi-phase method for cavitation was created to simulate the performance of the control orifice. Turbulence modelling using a hybrid LES-RANS method allows significant computational gains over full LES, especially in the high Reynolds number (Re) wall-bounded flow geometries within a diesel injector.

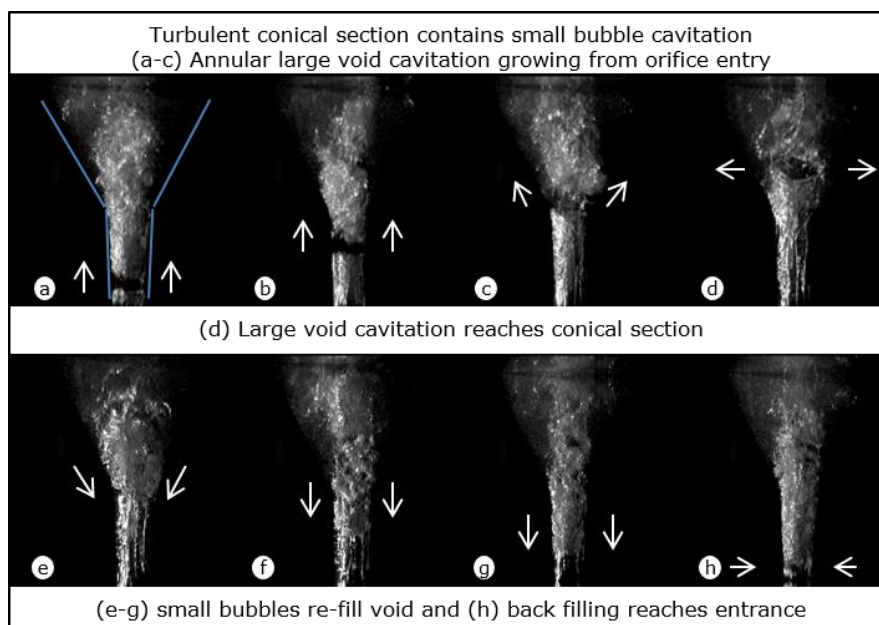
## 2. Experimental Work

Transparent models were created of hydraulic control parts from within a diesel injector, including a high pressure control orifice, between the needle top and control valve. Due to the small scale of real injectors the model was created at 40 times real-size, as it has been shown in previous studies that orifice flow performance is reasonably independent of scale [1]. This increased scale allowed much greater ease of visualisation using a high speed camera, a Vision Research Phantom V1210. Such a camera with appropriate lighting captures detail of the flow not visible to the human eye, and allows the hydraulic behaviour to be tracked via the cavitation process.

The orifice diameter is around  $100\mu\text{m}$  and tapers slightly before a larger conical section. Flow is from bottom to top through the orifice (Figure 1). Pressure measurements are taken upstream and downstream. Scaling calculations from accurate 1D simulation results allow the LSMs to be run under conditions which are equivalent to those of the real-scale component in normal operation [7]. The LSM was setup with scaled running conditions to match those of an injector operating at 1000, 1500 & 2000 bar rail pressure. As well as the large void and small bubble cavitation, as seen in previous orifice studies, the imaging revealed a new effect not previously seen, where the cavitation showed a pulsing motion along the orifice.



**Figure 1 Orifice geometry and flow**



**Figure 2 Transient effect of the pulsing cavitation seen on LSM - 1000bar equivalent injection pressure**

Figure 2 shows a complete pulse of the cavitation in 8 images. The frequency was consistent over a significant number of events. The three injection pressures gave results which were visually similar but had different frequencies. For the 1000bar case the frequency was 160Hz and the Coefficient of Discharge calculated from the flow rate and pressure drop was 0.71.

### 3. CFD Simulation

The standard CFD modelling uses RANS turbulence modelling due to the low computational requirements and accuracy of Cd prediction. It was obvious that the RANS modelling could not predict the transient behaviour seen on the Large Scale Model from nominally steady pressure conditions.

A turbulence simulation method was chosen which combined both the LES and RANS techniques. This was set up using the commercial code ANSYS Fluent v15, with the intention of developing a simulation technique that could be used on a wide range of geometries. The hybrid LES-RANS technique was chosen to allow the model to run in RANS mode on a coarser mesh in areas of lesser interest, while still achieving the required detail within the orifice.

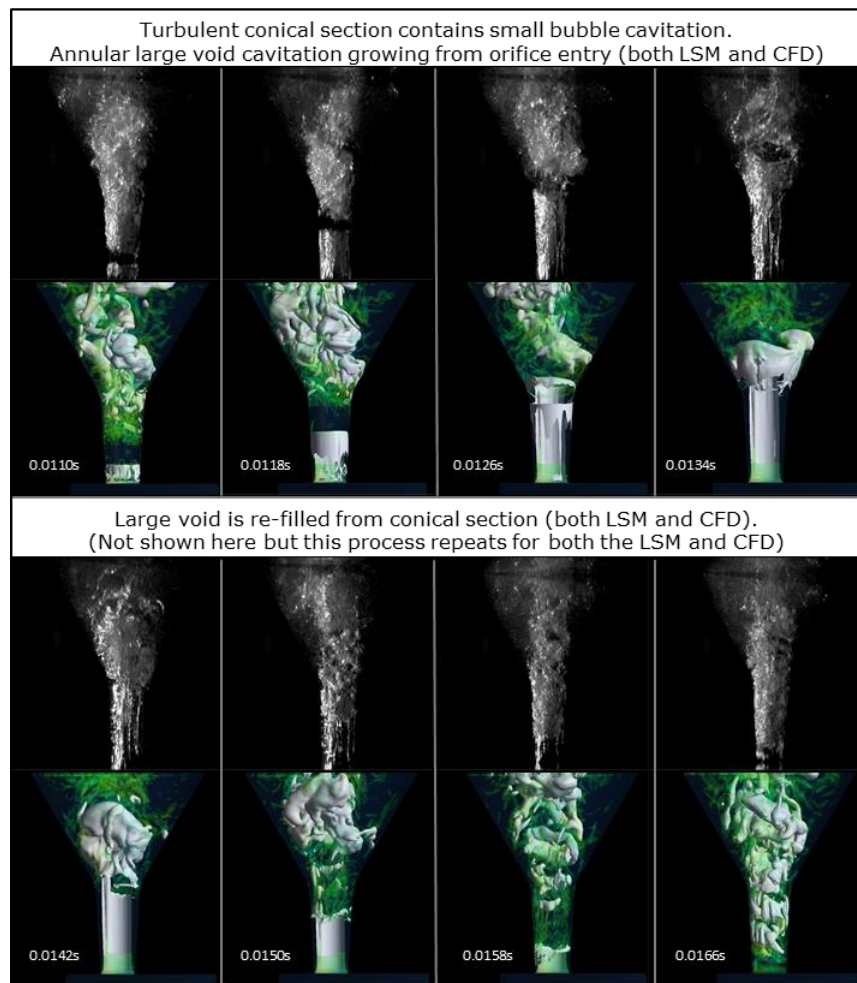
The 1000Bar equivalent case was simulated using the same scale pressure drop as calculated for the Large Scale Model. For the cavitation, a mixture model was used. This assumes that both liquid and vapour travel in a homogeneous multiphase flow; although not without limitations, it is a valid assumption for cavitating flows. Two incompressible phases are modelled, which is reasonable at the low pressure drops used in the large scale modelling.

The simulation of the 1000Bar equivalent injection pressure gave transient behaviour which matched the experimental results, as shown in Figure 3 with the eight experimental images from Figure 2. The frequency is 156Hz, giving excellent correlation with the experimental value of 160Hz.

Movies of both the experimental flow and the CFD results show a striking similarity.

The average value of Cd from the hybrid LES-RANS model is 0.71. This gives excellent agreement with the value measured on the Large Scale Model.

These results give confidence in the modelling and strong validation for the combination of turbulence and multiphase CFD models. Full details are given in [8].



**Figure 3** CFD results correlate well with LSM experimental results. CFD (rows 2 & 4) cavitation (iso-surface of 90% liquid fraction - white) and vortex structures (iso-surface  $Q = 1e^7$  - green)

#### 4. Real-size CFD Modelling

The CFD model was run for the real size case and produced similar pulsation close to 150 kHz. The expected frequency from the scaling calculations is 160 kHz, which further confirms the method.

#### 5. Discussion

A realistic method has been demonstrated which enables the use of LES CFD modelling in product improvement. Some validation of the method has also been demonstrated, which is not common for LES CFD in this field.

In summary:

- High speed imaging of cavitation within a control orifice revealed a pulsing hydraulic effect not seen in previous studies. Large void cavitation was created along the orifice wall and once the conical section was reached, backfilling occurred upstream along the orifice. When the backfilling reached the orifice entrance, there was a flow structure transition, large void cavitation reformed and the process repeated
- Advanced turbulence modelling in the CFD simulations produced results that are hugely more accurate than standard models when compared with experimental results.
- The CFD matched the pulsing large void cavitation along the orifice and the back-filling event.
- The frequency of this behaviour on the CFD results also agreed with that of the experiment.
- The CFD revealed that the flow rate was also pulsating, linked with the cavitation behaviour described above. The fact that the average  $C_d$  in the CFD agreed with that measured in the experiment gave confidence that the CFD flow rate pulsation was accurate.
- Using LSMs enabled the development of the LES CFD models in a manageable time-frame. This saved considerable time in developing the models for real-size components.
- In the case of the real injector, the frequency of the pulsing cavitation was too high to influence the engine performance. The awareness of the phenomenon provided by this work will enable any potentially adverse effects to be avoided in future design levels.

#### References

- [1] C. Soteriou, M. Smith and R. Andrews, "Cavitation hydraulic flip and atomization in direct injection diesel sprays," in *IMEchE C465/051/93*, 1993.
- [2] 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.
- [3] 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.
- [4] 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.
- [5] C. Egerer, S. Hickel, S. Schmidt and N. Adams, "Analysis of turbulent cavitating flow in a micro-channel," in *SHF Conference on Hydraulic Machines and Cavitation / Air in Water Pipes*, Grenoble, France, 2013.
- [6] 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.
- [7] C. Arcoumanis, J. M. Nouri and R. J. Andrews, "Application of Refractive Index Matching to a Diesel Nozzle Internal Flow," in *6th International Symposium on Applications of Laser Techniques to Fluid Mechanics*, 1992.
- [8] D. Bush, C. C. E. Soteriou, M. Winterbourn and C. Daveau, "Investigating hydraulic control components in high performance injectors" in *Fuel Systems for IC Engines*, *IMEchE* 2015.