APPLICATION OF LDA AND PIV TECHNIQUES TO THE VALIDATION OF VECTIS USING BOUNDARY MESH MOTION

S M Sapsford
Ricardo Consulting Engineers Ltd.

Computational fluid dynamics (CFD) is being increasingly used to analyse gasoline direct injection (G-DI) combustion systems in the early stages of design and development. Consequently, it is important to ensure that the CFD codes are both fast and accurate. The objective of the work presented in this article was to validate the new, advanced boundary motion capability of VECTIS which is built on it's industry-proven automatic meshing capabilities. Two experimental techniques were used to investigate the flow pattern generated within the combustion chamber of G-DI engine. The first was Particle Image Velocimetry (PIV) using a water-analogy Dynamic Flow Visualisation Rig (DFVR). The second technique was Laser Doppler Anemometry (LDA) in a motored engine. Finally, the results were compared directly to CFD predictions.

INTRODUCTION

The design of a direct injection gasoline combustion system, compatible with both early and late injection operation, creates a difficult challenge. There are many interacting variables, including fuel injection characteristics, fuel spray targeting, injection timing, ignition timing, combustion chamber geometry and charge motion, which are particularly hard to optimise. To design a G-DI combustion system efficiently, it is therefore necessary to understand the complex interactions and influences of all of the parameters.

One such parameter, charge motion, has a critical impact on fuel transport and mixing within the combustion chamber. Furthermore, it depends on the shape of both the inlet ports and the combustion chamber, both of which must be defined early in the design process. Hence, rapid and cost effective tools are required to study the in-cylinder flow generated during the intake stroke.

CFD is increasingly being used as one of these tools but has traditionally suffered from long lead times for mesh generation and mesh motion definition. Recently, automatic mesh generation has been the subject of intense development. Products based on both tetrahedral and hexahedral cells have become available and whilst the former tend to be less accurate and experience convergence problems, it is very difficult for the latter, with the exception of VECTIS, to generate a complete mesh, requiring manual generation of a significant number of cells. More importantly, it is very difficult, if not impossible, to use these meshes for moving mesh problems.

Ricardo have been carrying out such moving mesh analyses for several years using the industry-proven automatic mesh generation capabilities of VECTIS, but with the increase in geometrical complexity typically demanded by G-DI combustion systems, an even more advanced system for mesh generation and motion is required.

Consequently, Ricardo have developed a new boundary motion feature for VECTIS which enables the rapid treatment of time dependent flow domain problems involving complex geometry. The objective of the work presented in this article was to investigate the in-cylinder flow of a G-DI engine using two experimental methods and hence to assess the accuracy of the CFD engine simulations carried out using the new mesh features.
Two techniques, as described below, were used to provide data for comparison with the charge motion results from the CFD analyses:

- Particle Image Velocimetry in a dynamic water-analogy rig. The use of a water-analogy rig is limited to the induction stroke and relies on the assumption that the flow behaviour is preserved when matching the Reynolds number. Nevertheless, such a rig enables good optical access and slower flow velocities which, combined with the PIV technique, can provide measurement data even with early cylinder head and piston prototypes.

- Laser Doppler Anemometry in a motored engine with optical access through a window mounted in the spark plug hole. Two components of velocity can be measured throughout the engine cycle at a number of locations.

G-DI ENGINE CONFIGURATION

The engine configuration used throughout the studies described in this article was the Ricardo top entry G-DI combustion system previously described in [1]Ref. [1]. The general features of the engine are as follows:

Bore	74.0 mm	
Stroke	75.5 mm	
Swept volume	325 cm^3	
Cylinder head	RCE161 Top entry Port	
No. of valves	2 Inlet, 2 Exhaust	
Valve timings	IVO	16° BTDC
	IVC	48° ABDC
	EVO	46° BBDC
	EVC	18° ATDC.

ANALYSIS PREPARATION

A complete surface model of the combustion chamber and piston crown was passed directly to VECTIS from a Computer Aided Design (CAD) system in STL file format. The port geometry was generated by reverse engineering the test components to take account of modifications made during testing and ensure geometrical consistency between the analytical and experimental programmes. The individual computational meshes were then generated automatically by VECTIS with local mesh refinement near the boundaries in order to capture the surface definition accurately. Figure 1 shows the surface geometry sectioned to show the computational mesh. In addition to local boundary mesh refinement, block refinement (where cells are forced to be refined irrespective of their proximity to boundaries) was applied in the region of the valve gaps to ensure accuracy of solution. The mesh sizes ranged from 315,000 cells at TDC (valves open), through 400,000 at BDC to 120,000 at TDC firing.

The analyses have been carried out with the new boundary motion feature of VECTIS which allows much greater movement of the flow domain boundaries before a solution re-zone becomes necessary. During the solution, as the boundaries move (e.g. the piston crown and valves), the internal mesh structure automatically deforms in order to minimise the distortion of each individual cell (Figure 2). This is performed by requiring that the displacement of any given point in the mesh is the mean of the displacements of surrounding points:

[1] Numbers in parentheses indicate references at the end of the article

mathematically, this means that the displacement's Laplacian is zero. This is exact in the limit of a dense mesh with large numbers of nearest neighbours. Discretization errors can arise because of the finite spacing of the mesh but these are resolved by iterative application of the mesh no-crossover condition to yield the final legal mesh. The calculation is automatically monitored continuously for excessive cell distortion (which could lead to mesh inversion) so that the user can re-zone the calculation onto a new mesh when required.

In addition, meshes can be moved in both directions; distorting from an initially Cartesian grid and also progressing from a distorted mesh to a Cartesian. The latter feature, known as reverse boundary mesh motion is useful for the compression stroke, particularly valve closure and piston approach to TDC.

In all, the elapsed time from receiving the geometry, generating the computational meshes and starting the analysis was less than 2 man days.

CFD ANALYSIS

CFD analysis of the engine was carried out for comparison with the measurements. The intake and compression strokes were modelled at Wide Open Throttle (WOT) to match the LDA measurement conditions and an intake stroke was analysed with water to simulate the DFVR conditions.

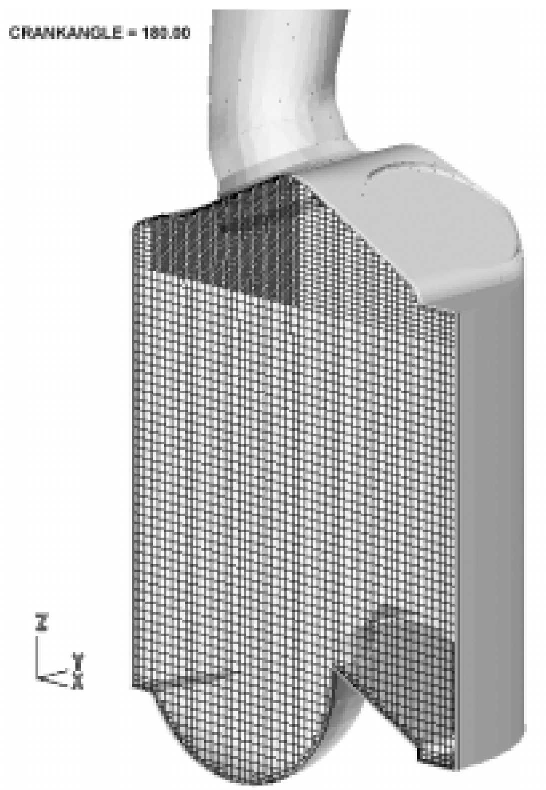

Figure 1: CFD Surface geometry and computational mesh

a) Water Simulation - The simulation of the DFVR, using water as the working fluid, was run with a constant pressure head of 1.08 bar to simulate the header tank on the rig. The modelled rig speed was 13.59 rev/min. A 3-D incompressible, unsteady analysis was carried out on the intake stroke only to BDC, solving for three momentum equations, continuity and the k-ε model for turbulence.

b) Air Simulation - The boundary conditions for the LDA simulation, using air as the working fluid, were obtained from a complete model of the engine, intake and exhaust systems created using the Ricardo engine performance simulation program, WAVE [2]. The model was analysed at a motored engine operating condition of 1500 rev/min, WOT. Unsteady boundary conditions at the cylinder head/inlet manifold face were generated and applied to the VECTIS model. A full 3-D unsteady analysis was performed solving for three momentum equations, continuity, energy and the k-ε model for turbulence. The analysis was carried out for the intake and compression strokes from TDC to TDC.

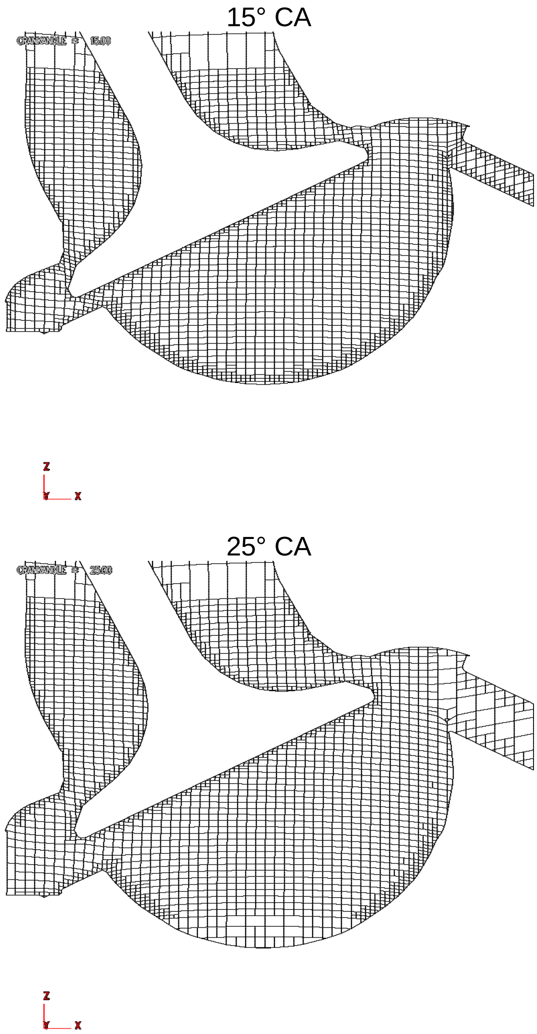

Figure 2: Examples of computational mesh during analysis

MEASUREMENTS

a) Dynamic Flow Visualisation Rig

Ricardo's Dynamic Flow Visualisation Rig consists of a water-analogy full scale model of the engine with optical access through both the liner and the piston (Figure 3). The rig was designed with the aim of creating a rapid, flexible and cost effective tool to investigate in cylinder charge motion. Velocity data can be extracted using a Particle Image Velocimetry cross-correlation technique at any crank angle during the induction stroke. Water is seeded with neutrally buoyant 70 μm polystyrene tracer particles and is contained in a header tank, providing a constant pressure head. Further details can be found in [3].

PIV results were obtained for the test engine within the mid-cylinder plane of the engine every ten degrees crank angle to study the development of the main tumbling vortex.

Figure 3: The dynamic flow visualisation rig

b) Laser Doppler Anemometry

LDA measurements were obtained through the spark plug orifice of the engine under motored conditions (Figure 4). The piston crown was anodised to minimise internal glare. The cylinder head was modified to accept a Kistler type 6121 pressure transducer for in-cylinder pressure measurement. Crank position was related through an optical shaft encoder, and a timing logic circuit was used to gate alternate TDC pulses with a timing pulse from an optical switch on the camshaft at 0° CA. The resultant signal of 2000 pulses per engine 4-stroke cycle, and a reference marker, was input to the Doppler signal processor and latched as a time event to a validated Doppler burst detection.

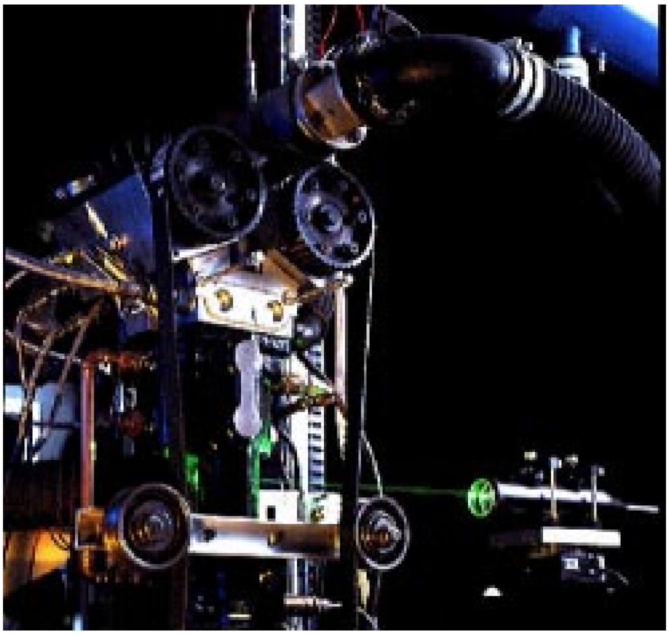

Figure 4: Typical single cylinder Ricardo "Hydra" optical research engine

Measurements of instantaneous velocity and RMS turbulence intensity were made in two orthogonal planes; the tumble and cross-tumble planes, co-linear with the axis of the spark plug. The field of view was restricted by the optical access through the spark plug hole. Data were recorded at seven points from the spark plug tip, in increments of 5 mm to a maximum depth of approximately 20 mm below the gas face, as shown in Figure 5. The engine was run at 1500 rev/min ±5 rev/min and WOT. Further details describing the analysis of the LDA data are described in [5], [6] and [7].

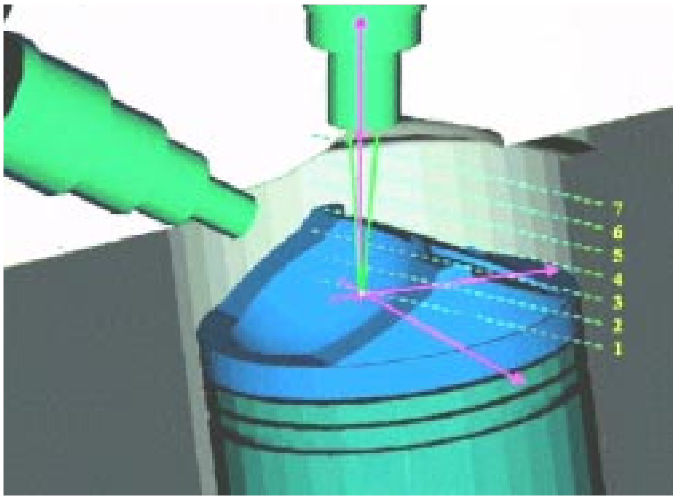

Figure 5: Measurement locations reached via the spark plug probe

RESULTS AND DISCUSSION

a) Water Simulation - The results showing the comparison between the DFVR and CFD results are presented in Figure 6. Note that a horizontal line is marked on the CFD plots to indicate the limit of visibility available on the DFVR.

120° CA ATDC - Both techniques show that most of the intake charge is forced to flow down the cylinder liner generating a strong reverse tumbling air motion. During intake, high velocity gradients are present in the mid-cylinder plane where the flows through the inlet valves interact. The piston bowl helps create an upward jet from the bowl lip towards the spark plug. The main difference between the results is the location of the edge of the band of high velocity flow coming from the inlet valve. The CFD simulation predicts a slightly stronger flow across the roof of the combustion chamber. This has the effect of extending the high flow band towards the exhaust side of the cylinder and therefore appears as an area of high difference. Other differences in velocity between the CFD prediction and the PIV measurements are found in areas such as the centre of the tumbling vortex and at the exit of the bowl. These differences can be explained by the vortex centres being at slightly different locations, but also by the fact that these areas correspond to high levels of cyclic variation in the measurements. The time averaging of the PIV velocity results tends to underestimate the velocity in such regions. This effect also occurs in the squish region on the exhaust side of the cylinder but is less visible due to the smaller velocity magnitudes and hence, smaller absolute differences.

180° CA ATDC - A relatively strong vortex is evident centred in the bowl region of the cylinder. The exhaust side of the cylinder is virtually quiescent. The same general comments which were made about the areas of high cyclic variation for the flow field at 120° CA ATDC apply for the velocities at BDC. Due to lower mean velocities, the absolute differences between CFD and PIV measurements are significantly smaller. It can be noted the difference in flow velocities coming from the inlet valves described for crank angle 120°, lead to the creation of a slightly more penetrating jet from the piston bowl lip in the case of the PIV measurements.

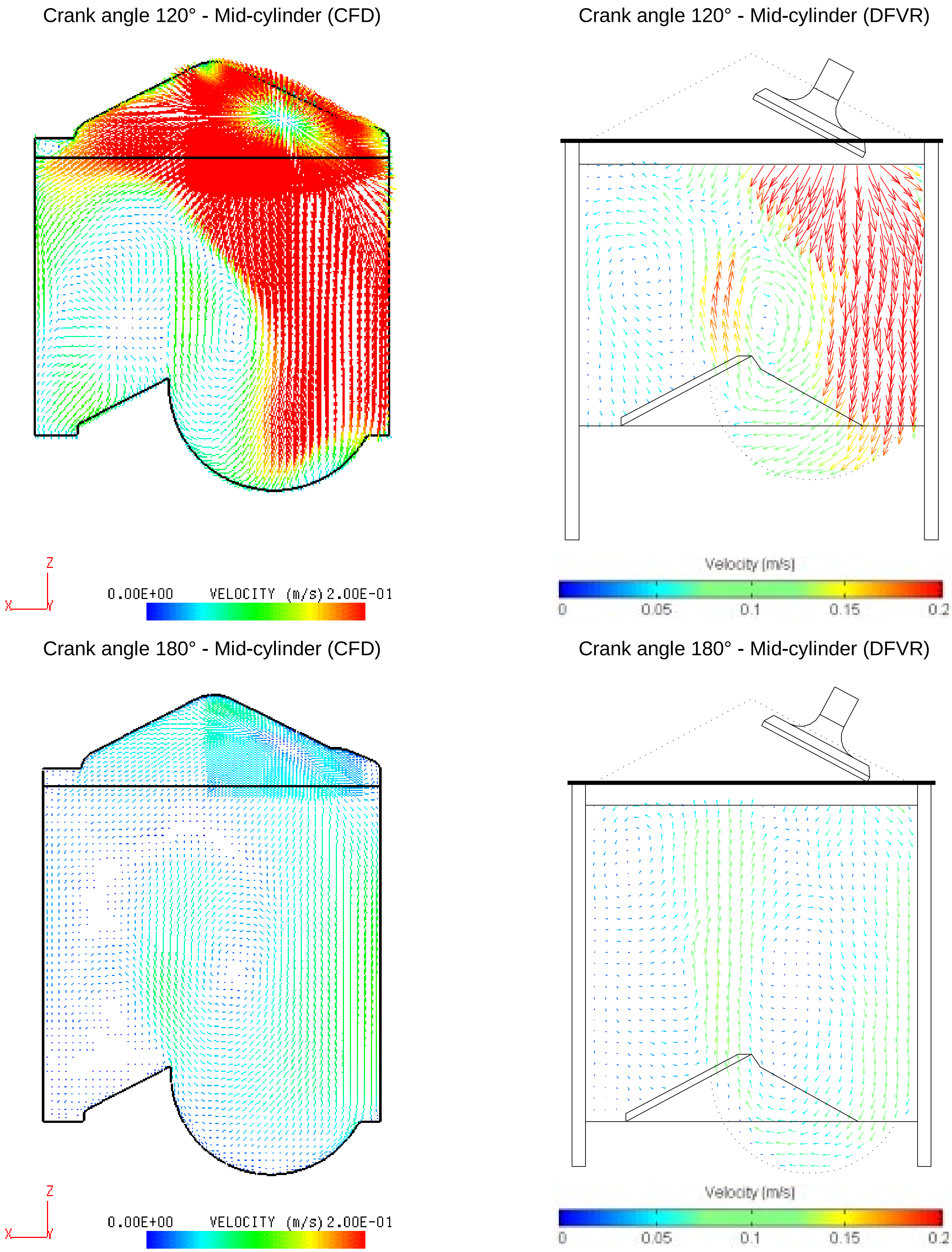

Figure 6: Water analysis - comparison of predicted (CFD) and measured (DFVR) results

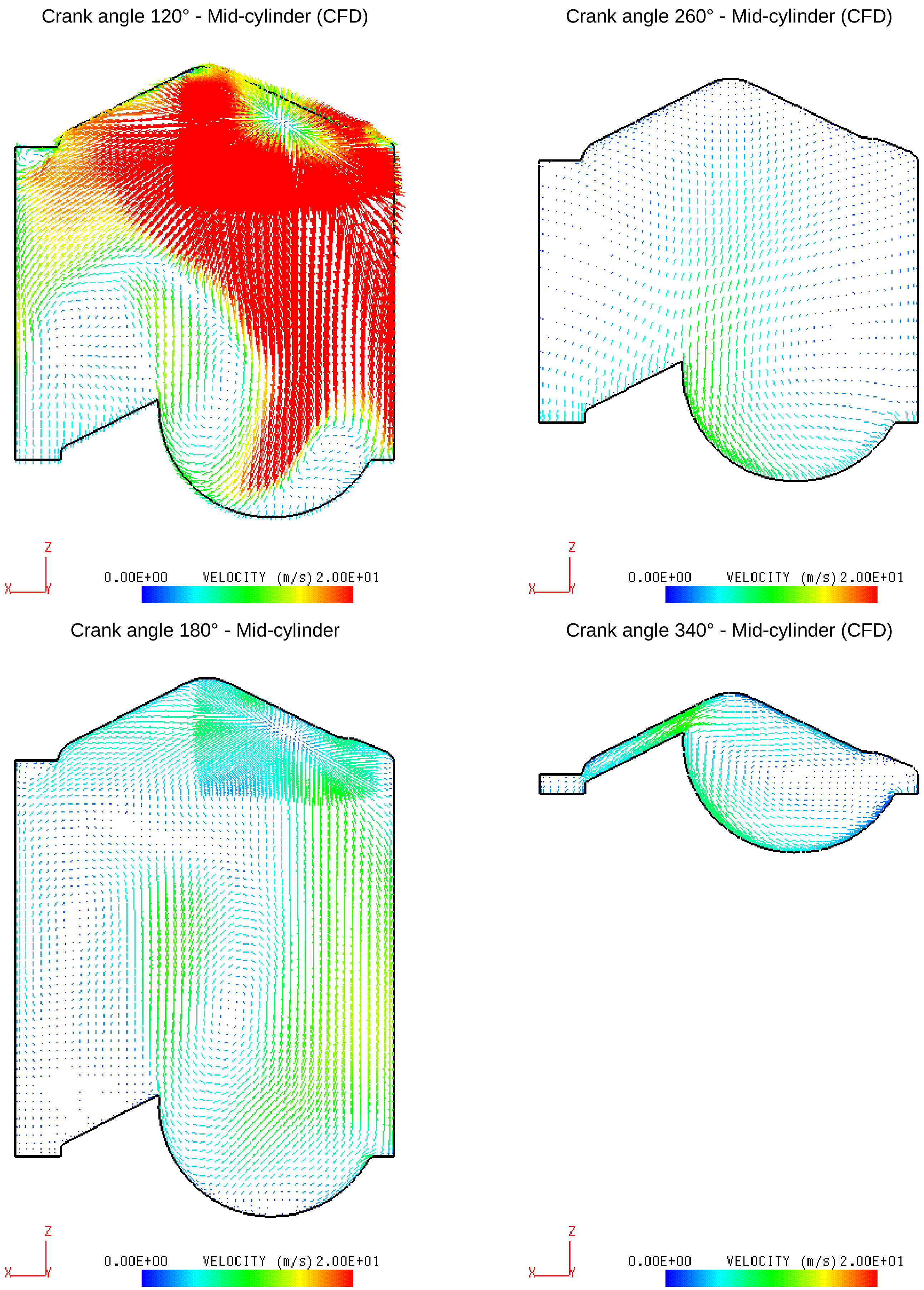

Figure7: CFD air simulation - Induction and compression strokes

b) Air Simulation - The results from the CFD prediction of the motored engine are shown in Figure 7. Many of the features in the induction stroke are the same as those seen in the water simulation including the stagnation zone between the inlet valves, the reverse tumbling vortex and the quiescent zones below the exhaust valves. However, as opposed to the water simulation, significant air enters the cylinder after BDC (due to compressibility) and contributes to the creation of the reverse tumbling vortex.

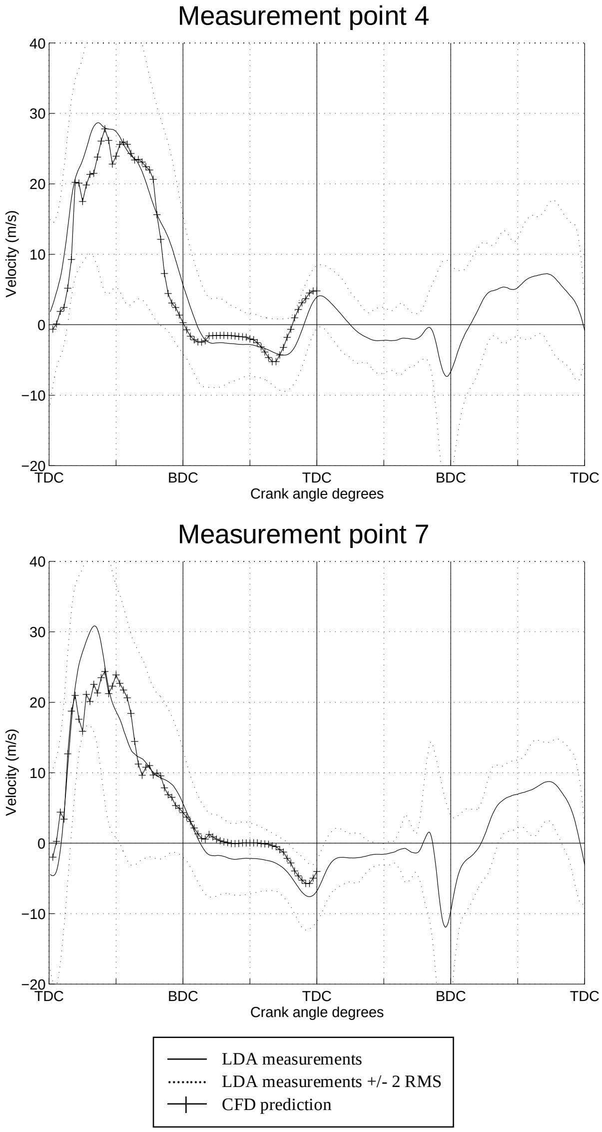

Figure 8: Comparison of CFD air simulation and LDA results

This vortex persists until top dead centre and does not break down into turbulent motion. The strong jet flow from the bowl lip also survives compression, although it is slightly deflected by the exhaust side squish flow.

The mean velocities measured by the LDA technique at points 4 and 7 are compared with the CFD predictions for the same points in Figure 8.

The CFD simulation of the motored G-DI engine predicts the same trends that were measured with the LDA technique. Good quantitative agreement is obtained with the CFD results being within the ensemble average plus or minus two RMS band for most of the cycle. The CFD results also confirm the fact that the vortex persists until the end of the compression stroke. Such differences as the maximum velocity during the intake stroke for point 7 can be explained by minor uncertainties in the location of the LDA measurement volume within the combustion chamber. Even though these results are only point measurements, they demonstrate the fact that VECTIS can be used to predict quickly and reliably the air motion created in the combustion chamber of a G-DI engine.

CONCLUSIONS

The new boundary motion feature of VECTIS, coupled with it's industry-proven automatic mesh generation capabilities is now the most advanced and easy to use method for CFD analysis of moving geometries and has allowed a further reduction in the lead times associated with in-cylinder analysis. Complete mesh generation and setup is achieved in less than two days, with analysis and post-processing completed within less than 2 weeks of receipt of geometry.

Excellent quantitative and qualitative agreement has been obtained between the CFD simulation results and the velocities measured by the two experimental techniques.

These demonstrate that VECTIS can be used to reliably predict the air motion created in the combustion chamber more quickly than ever before. This allows the designer to integrate the CFD results into his design within a timescale compatible with engine design and development processes.

REFERENCES

1. N. S. Jackson, J. Stokes, P. A. Whitaker, T. H. Lake, "Stratified and Homogeneous Charge Operation for the Direct Injection Gasoline Engine - High Power with Low Fuel Consumption and Emissions", SAE970543, 1997.
2. C. S. Wren, O. Johnson, "Gas Dynamics Simulation for the Design of Intake and Exhaust Systems - Latest Techniques", SAE951367, 1995.
3. N.S Jackson, J. Stokes, M.R. Heikal, J.H. Downie, "A Dynamic Flow Visualisation Rig for Automotive Combustion System Development", SAE 950728, 1995.
4. M.A. Faure, M.R. Heikal, N.S. Jackson, "PIV Measurements and Characterisation of In-Cylinder Flows in Combustion Engines", 8th International Symposium On Applications Of Laser Techniques To Fluid Mechanics, Lisbon, Portugal, 1996.
5. A.R. Glover, G.E. Hundleby, O. Hadded, "The Development of Scanning LDA for the Measurement of Turbulence in Engines", SAE 880378, 1988.
6. A.R. Glover, G.E. Hundleby, O. Hadded, "An Investigation Into Turbulence in Engines Using Scanning LDA", SAE 880379, 1988.
7. O. Hadded, I. Denbratt, "Turbulence Characteristics of Tumbling Air Motion in Four-Valve S.I. Engines and Their Correlation with Combustion Parameters", SAE 910478, 1991.