Computer modelling of isothermal compression in the reciprocating compressor of a complete isoengine

Stephenson P L and Coney M, Innogy plc., Windmill Hill Business Park, Whitehill Way, Swindon, Wiltshire SN5 6PB, Tel: 01793 877777
(Please email P L Stephenson at peter.stephenson@innogy.com)

Abstract

Innogy plc is developing a reciprocating piston compressor, into which water is injected in order to cool the air and make the compression as near to isothermal as possible. It forms a key part of the iso-engine, which is a novel high efficiency reciprocating engine. Making the compression near isothermal reduces the compression work and increases the overall cycle efficiency. The water is sprayed into the cylinder in a quantity that is sufficient to achieve the cooling without significant evaporation. The compressor and related CFD models are being developed in stages. The first two stages, namely detailed investigations of single sprays and the testing of a complete compressor have been reported at earlier ILASS conferences. This paper is concerned with the third stage, namely a complete 3 MW isoengine. It describes the application of the CFD model to the compressor of the isoengine, and its validation against rig measurements. A fourth stage with a 7 MW isoengine is planned.

The CFD simulations were performed using the commercial Star-CD code. The multiphase aspects were treated in a Lagrangian/Eulerian mode with a moving mesh. Standard droplet models in the code were used, together with some work, in conjunction with Computational Dynamics, on improved collision models. The spray model (specification of initial droplet locations, size and velocity) has been derived from the single spray tests.

Comparisons have been made with measurements for four tests on the Engineering Demonstrator. The agreement between measured and predicted pressures is always very good. The agreement between temperatures derived from measured pressures and predicted temperatures is reasonable, and this aspect of the predictions has been investigated.

Further validation of the model is planned, after which it will be used in the design of the Commercial Demonstrator.

1. Introduction

Innogy plc is developing a reciprocating piston compressor, into which water is injected in order to cool the air and make the compression as near to isothermal as possible [1],[2]. It forms a key part of the iso-engine, which is a novel high efficiency reciprocating engine. Making the compression near isothermal reduces the compression work and increases the overall cycle efficiency. The water is sprayed into the cylinder in a quantity that is sufficient to achieve the cooling without significant evaporation. The compressor and related CFD models are being developed in the following stages:

- Detailed experimental investigations of a number of single sprays from different nozzles were performed at Cranfield University for various ambient pressures and nozzle differential pressures. The findings of these experiments were used to develop and verify a CFD model of a single spray in a stationary mesh. [3]

- A complete compressor (called the Proof of Concept machine) of 200 mm diameter and with 18 spray nozzles was built and tested by Ricardo Consulting Engineers. Tests on this compressor have shown that near-isothermal compression can be achieved, with substantial energy saving. The CFD model used the spray models derived from the single spray studies. A representative range of cases has been simulated and the agreement with measurement was good in the majority of cases. [4]

- A complete isoengine (called the Engineering Demonstrator) has been built with a 385 mm diameter compressor cylinder equipped with 360 spray nozzles. The isoengine is complete and Ricardo have performed tests on it.

- Work has started on the design of a second and larger isoengine, called the Commercial Demonstrator.

This paper describes the application of the CFD model to the compressor of the Engineering Demonstrator, and its validation against rig measurements. CFD is being used as a tool to understand how the water droplets from the sprays are distributed in the cylinder at different stages of the compression and how it is affected by different parameters. This knowledge is important in the design and operation of the water injection system.

2. Experiments

The Engineering Demonstrator is currently being tested. It is a complete isoengine with one cylinder for compression and three for combustion. Fig 1 shows, diagrammatically, the spray arrangement in the compressor. Experimental data from its compressor are used to validate both the 3-D CFD model of the compressor (described here) and a 1-D model of the compressor and auxiliary components [5].

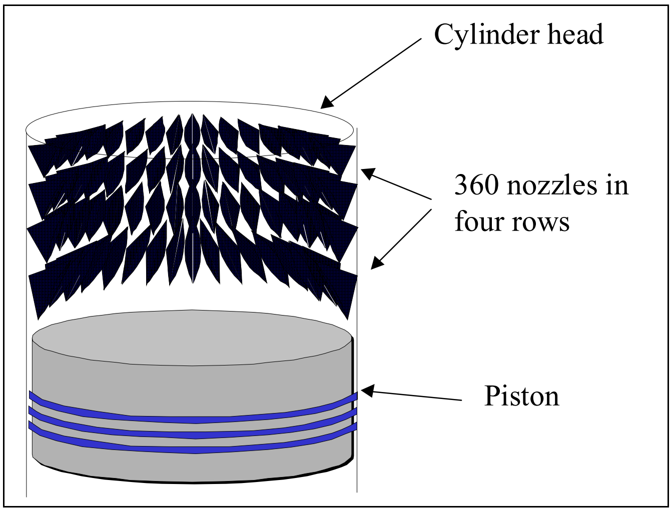

Fig 1. Diagram showing the arrangement of the 360 spray nozzles in the cylinder

The main emphasis with the Engineering Demonstrator has been to demonstrate that it can be made to work as designed. As a result, it has been possible so far to obtain only four cases suitable for comparison with CFD. Comparisons have been made for all four. During tests on the Engineering Demonstrator, the cylinder and other key pressures are recorded using a high-speed data collection system, and other relevant parameters recorded. The nozzle water flow rates are calculated from the measured pressure difference between the spray rings and the cylinder itself. As it was not possible to measure the cylinder air temperature directly, it was inferred from the measured cylinder pressures.

3. CFD approach

The CFD simulations were performed using the commercial Star CD code [6]. The multiphase aspects were treated in a Lagrangian/Eulerian mode. The Lagrangian method tracks 'parcels' of droplets with all the droplets in a given 'parcel' having the same properties (size, temperature, velocity, etc). A moving mesh is used to represent cylinder motion. Droplet break-up was modelled with the Reitz and Diwakar model [7]. The spray was modelled by introducing droplets a short distance downstream from the position of the nozzle. The spray model (specification of initial droplet locations, size and velocity) has been derived from the single spray tests. Droplet coalescence modelling was not activated in the predictions. Early predictions showed that the existing collision model in Star-CD (based on the method of O'Rourke [8] with the probability of collision leading to coalescence calculated using the method of Brazier-Smith et al [9]) could lead to the creation of a few unrealistically large droplets. To help resolve this, an improved collision model was developed and tested in conjunction with Computational Dynamics (the developers of Star-CD). This new model, based on work by [10], [11] and [12], had three aims, namely to reduce computing time, include more realistic tests for collision and coalescence, and to be less mesh dependent. Runs with this new model did not predict significant droplet collision for the type of calculations reported here. It was therefore concluded that it was reasonable to neglect droplet collision and thereby reduce computing time.

The Engineering Demonstrator compressor is more challenging to model than the Proof of Concept compressor as it has more nozzles (360 instead of 18) and runs faster (200 to 600 rpm, against 100 to 200 rpm). As its nozzles are arranged uniformly round the cylinder periphery, the initial modelling represented only a 4 or 8 degree sector, together with cyclic boundary conditions (Fig 2).

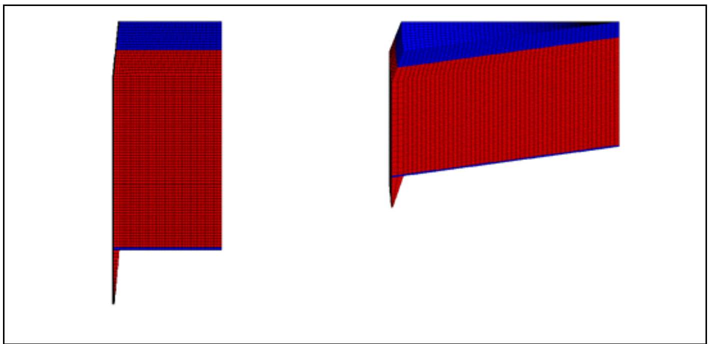

Fig 2. Original 8 degree sector mesh

This was done to reduce the size of the model and hence the computer requirements. However, these models used a sector of a polar coordinate mesh and this naturally involved very high aspect ratio cells near the cylinder axis and this resulted in major numerical problems. As a result, the current model uses a 32 degree sector mesh which is quasi-rectangular near the axis and quasi-polar near the cylinder wall (Fig 3). This has given far fewer numerical problems and the run times are acceptable.

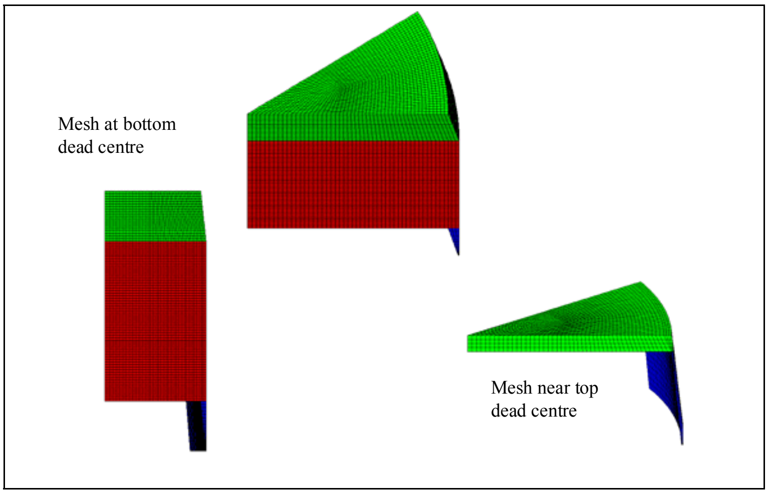

Fig 3. Details of new 32 degree sector mesh

The profiles for the variation of water flow rate and nozzle differential pressure with crank angle were taken from the measurements, together with the initial pressure and temperature in the cylinder.

In the CFD modelling of the Proof of Concept compressor [4], the volume predicted for the air within the cylinder was not identical with that in the actual experiments, for a number of reasons. The model did not include the water left in the cylinder from previous strokes. Predictions started at bottom dead centre (BDC) and this was slightly after the start of water injection, and this water was also ignored. The Eulerian part of the calculation assumed that the void fraction never exceeded 0.4 in any cell (the void fraction is defined here as the fraction of the cell occupied by the second, ie water, phase on a volume basis). A shortcoming in the version of Star-CD used at the time meant that droplets had to be deleted when they struck a wall. Finally, the clearance volume in the CFD model was not identical with that in the actual compressor. As a result, all the Proof of Concept predictions had to be corrected; this was done by assuming isentropic compression from the air volume assumed in the CFD to the actual volume.

To give greater confidence in the CFD predictions for the Engineering Demonstrator, much effort was devoted to reducing the size of the corrections, and to improving the correction method. Around 50,000 droplet ‘parcels’ (see section 3) are now created during the first time step, to represent the water left over from previous strokes. They are given a uniform random distribution over the cylinder, with an initial diameter of 2 mm (based on the clearance available at top dead centre, TDC). The mass of such water and its initial temperature are estimated from detailed analysis of the experimental results. The calculations start at 20 degrees before BDC and therefore before any water injection. Also, with the version of Star-CD used for this work, it is no longer necessary to delete droplets which hit a wall. They are retained in the calculation and are assumed to 'stick' on the walls. Two new methods have been developed to correct the results. In the first, a polytropic index is estimated from the uncorrected mean cylinder pressures and temperatures, and this is used in the corrections, instead of the earlier isentropic assumption. The second new method uses a simplified analytical model of the compressor, based on key parameters whose values are derived from the uncorrected CFD predictions. Both the polytropic index and these key parameters can vary during the compression stroke.

4. Results

Although the Engineering Demonstrator can operate at up to 600 rpm, validation against measurement has been limited so far to tests at 200 and 400 rpm. Typical predictions at a crank angle of 260 degrees are shown in Fig 4. Here, the black dots show the droplet 'parcel' locations and the colours show the air temperatures within the cylinder. It can be seen that, at this stage in the compression, the droplets have already penetrated across most of the cylinder.

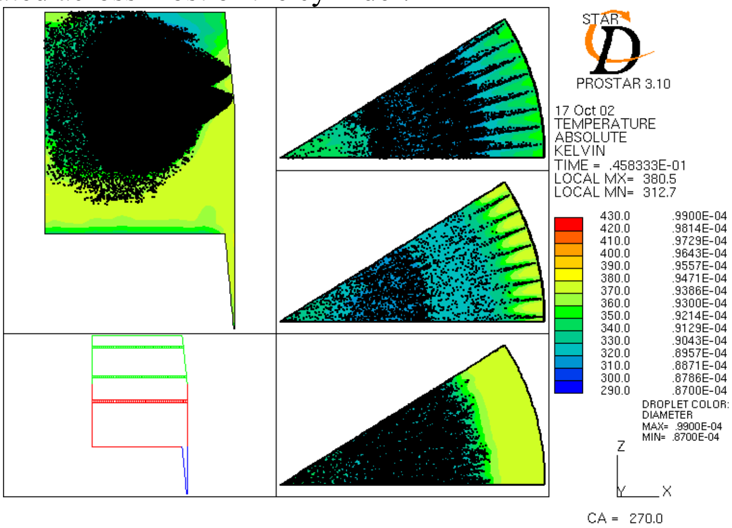

Fig 4. Typical CFD predictions showing droplet locations and air temperatures for a vertical slice (top left) and three horizontal slices (right) through the cylinder.

The predicted temperatures and pressures are compared with measurements in Fig 5 and 6. In each case, three sets of predictions are shown, namely the uncorrected values taken from the CFD and the corrected values obtained via the two methods summarised in section 3.

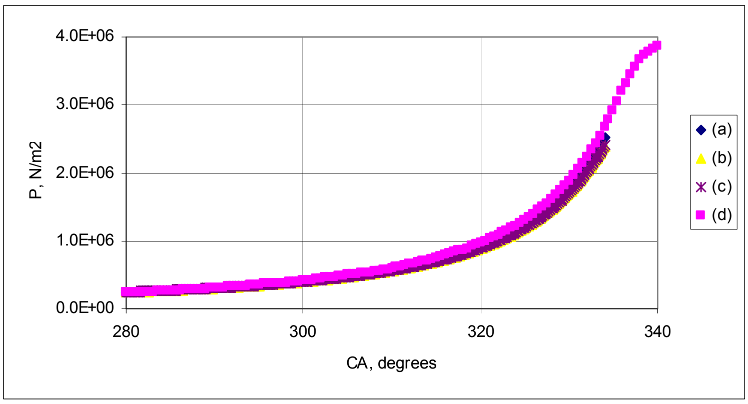

Fig 5. Comparison between predicted and measured cylinder pressures for test at 400 rpm. (a)=uncorrected CFD, (b)=polytropic correction (c)=detailed correction (d)=measurement

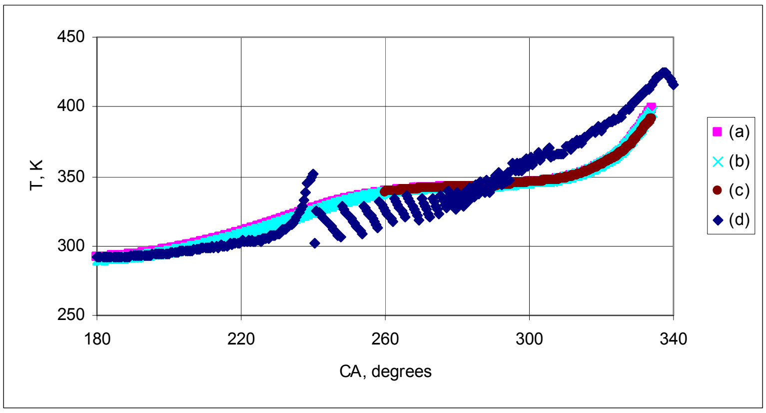

Fig 6. Comparison between predicted and inferred cylinder temperatures for test at 400 rpm. (a)=uncorrected CFD, (b)=polytropic correction (c)=detailed correction (d)=inferred

It is encouraging that the differences between the uncorrected and the two sets of corrected values are small, for both pressure and temperature. The agreement between measured and predicted pressures is very encouraging. The agreement between temperatures inferred from measured pressures and predicted temperatures as reasonable but not as good as for pressure. This was a bit surprising, as the measured temperatures are, in fact, deduced from the measured pressures and are not an independent result. As a check, the temperatures have been calculated from the corrected pressures, using the method used to derive temperatures from the measured pressures. These temperatures have always proved to be close to the corrected temperatures derived directly from the CFD results. It is thought that that the temperatures deduced from the pressure are sensitive to minor errors in the pressures. For example, the small oscillations in the temperatures inferred from measured pressures (Fig 6) are related to the sensitivity of the pressure transducer.

Similar results for one of the 200 rpm cases are shown in Figs 7 and 8; agreement between measurement and prediction is at least as good as for Figs 5 and 6.

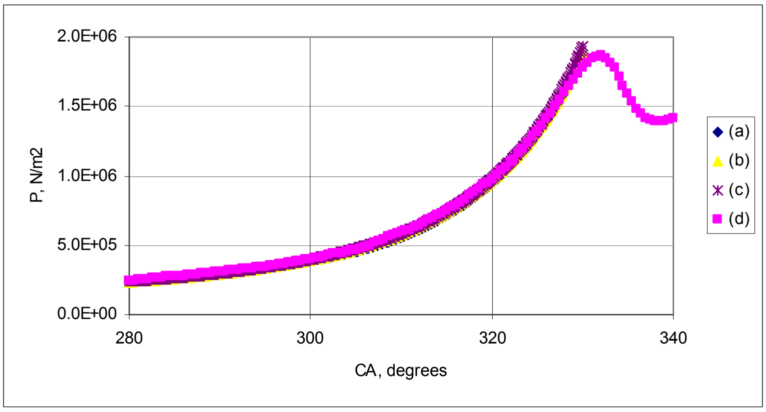

Fig 7. Comparison between predicted and measured cylinder pressures for test at 200 rpm. (a)=uncorrected CFD, (b)=polytropic correction (c)=detailed correction (d)=measurement

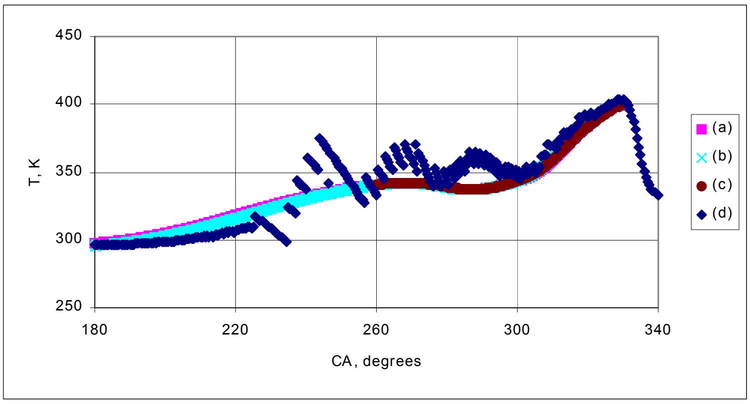

Fig 8. Comparison between predicted and inferred cylinder temperatures for test at 200 rpm. (a)=uncorrected CFD, (b)=polytropic correction (c)=detailed correction (d)=inferred

A similar level of agreement with measurement was obtained for the other two cases. Calculations have been made with time steps of 0.05 and 0.1 degrees crank angle per time step. Although the results are not significantly different, there are indications that the results for the lower time step are more stable numerically. Some sensitivity calculations have been made to check some of the parameters used to specify water at the start of the calculation, and more are planned.

5. Further work

It is planned to validate the model by comparing it against more test data, and make further sensitivity calculations, for example by varying the parameters used to define the water at start of calculation. Once this is done, the model will be used to help to optimise the performance of the Engineering Demonstrator and to design the proposed Commercial Demonstrator.

6. Conclusions

A CFD method initially validated against measurements for a single spray and measurements for a reciprocating engine with 18 nozzles has been successfully applied to the much more challenging case of a compressor with 360 nozzles. Various improvements have been made to the modelling technique to give greater confidence in the predictions. This, in turn, has enabled the CFD model to be used in the initial design studies for the proposed larger isoengine.

7. Acknowledgements

This paper is published with permission of Innogy plc.

8. References

[1] Coney, MW, Cross, AM, Linnemann, C, Morgan, RE, and Wilson, B, 2002, Engineering aspects of a novel high efficiency reciprocating internal combustion engine, paper IJPGC2002-26047, Proceedings of IJPGC’02, 2002 International Joint Power Generation Conference, Phoenix, AZ, USA, June 24-26

[2] Coney, M, Linneman, C, and Cross, AM, 2002, Isoengine promises 60 per cent efficiency, Modern Power Systems, December, p 37

[3] Stephenson PL, Abdallah H, Powell A, Jasuja A, and Ng K, 2001, Application of liquid sprays to achieve isothermal compression, Part 1, ILASS-Europe 2001, Zurich, Switzerland, 2-6 September
[4] Stephenson PL, Malmgren A, Coney M, Huxley, RA, and Richards RA, 2002, Isothermal compression by dense water sprays in a reciprocating piston compressor, ILASS-Europe 2002, Zaragoza, Spain, 9-11 September
[5] Malmgren A, Sidders J, Stephenson PL, and Coney M, 2003, Experiments and simulations of a quasi-isothermal compressor for a novel high efficiency engine, to be presented at International Conference on Compressors and their Systems, I Mech E, London, 8-10 September.
[6] Star-CD Methodology, Computational Dynamics, London, UK, 1999
[7] Reitz RD, and Diwakar R, Effect of drop break-up on fuel sprays, SAE Paper 860469, (1986)
[8] O'Rourke, PJ, 1981, Collective drop effects on vaporising sprays, Ph D Thesis, University of Princeton.
[9] Brazier-Smith PR, Jennings SG, and Latham J, 1972, The interaction of falling water drops: coalescence, Proceedings of Royal Society, London, Vol. A326 pp. 393 – 407
[10] Nordin, N, 2001, Complex chemistry modelling of diesel spray combustion, PhD thesis, Chalmers University of Technology, Sweden
[11] Aamir, MA and Watkins, AP, 1999, Dense propane spray analysis with a modified collision model, ILASS-Europe '99
[12] Schmidt, DP and Rutland, CJ, 2000, A new droplet collision algorithm,, J of Computational Physics, vol 164, pp 62-80.