ISOTHERMAL COMPRESSION BY DENSE WATER SPRAYS IN A RECIPROCATING PISTON COMPRESSOR

Stephenson P L*, Malmgren A*, Coney M*, Huxley, R A,** and **Richards R A*****

Email peter.stephenson@innogy.com

***Innogy plc., Windmill Hill Business Park, Whitehill Way, Swindon, Wiltshire SN5 6PB, UK**

**** W S Atkins Consultants Ltd, 220 Aztec West Park Avenue, Almondsbury, Bristol, BS32 4SY, UK (previously with National Power/Innogy)**

***** Richards Engineering Consultants, April Cottage, Blackstone, Henfield, W.Sussex, BN5 9TF, UK (previously with Ricardo Consulting Engineers)**

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 associated CFD models are being developed in a number of stages:

- Detailed experimental investigations of a number of single sprays from different nozzles have been performed at Cranfield University.
- A complete compressor (called the 'Proof of Concept' machine) of 200 mm diameter and with 18 spray nozzles has been built and tested by Ricardo Consulting Engineers.
- A complete iso-engine (called the 'Engineering Demonstrator') with a 385 mm diameter compressor cylinder equipped with 360 spray nozzles is being built by Ricardo and the compressor is currently being tested by them.

An overall description of the project plus details of the single spray work were presented at last year's ILASS conference. Since then, a detailed validation has been made between CFD predictions and the measurements for the 'Proof of Concept' machine, and the work is described in this paper. The tests on this compressor have shown that near-isothermal compression can be achieved, with substantial energy saving. In addition, the measurements have been used to validate a CFD model which was initially validated against experiments for a single spray. The CFD model is now being used with greater confidence in the third stage of this work, namely the Engineering 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. 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 associated CFD models are being developed in a number of stages:

- Detailed experimental investigations of a number of single sprays from different nozzles have been 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. Some simulations were also carried out with the single nozzle replaced by a cluster of three nozzles

- A complete compressor (called the 'Proof of Concept' machine) of 200 mm diameter and with 18 spray nozzles has been 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 with a moving mesh and all 18 nozzles was developed, based on the spray models derived from the single spray studies. This model can simulate the compression stroke at two different speeds and with two different nozzles. A representative range of cases has been simulated and the agreement with measurement was good in the majority of cases.

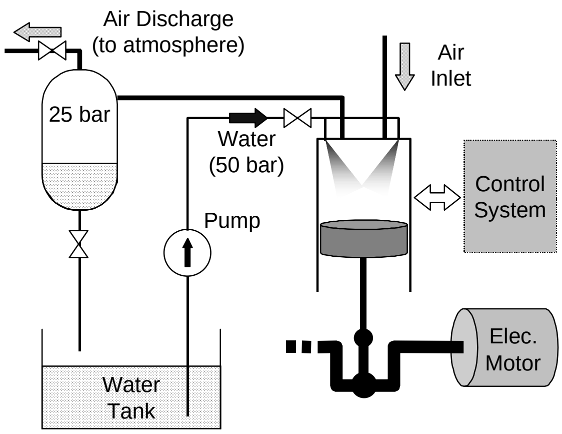
- A complete iso-engine (called the 'Engineering Demonstrator') with a 385 mm diameter compressor cylinder equipped with 360 spray nozzles is being built by Ricardo. The compressor is currently being tested by them.

The single spray work was presented at last year's ILASS conference [1]. An overall description of the project has been reported [2] [3]. Since then, a detailed validation has been made between CFD predictions and the measurements for the 'Proof of Concept' machine, and the work is described in this paper.

2. Experiments

2.1 Description of the 'Proof of concept' compressor

Proof-of-concept tests were conducted in 1995 to determine the feasibility of quasi-isothermal compression of air by means of water injection. One cylinder of a Bolnes three-cylinder low-speed two-stroke marine diesel engine with a 200 mm bore and a 350 mm stroke was converted into a compressor. A variable speed electric motor was used to drive the compressor. The cylinder head contained one inlet valve (suction valve) in the centre and four discharge valves, all hydraulically operated. Additionally, two relief valves were accommodated on the cylinder head as a safety feature. Six cylindrical inserts (each containing three commercial hollow-cone atomisers) were mounted in the cylinder head. Air was taken from the test shop though a filter. The compressor discharged the air/water mixture into a pressurised water separator, which was normally maintained at 25 bar pressure using a control valve. A purpose-designed hydraulic piston pump capable of up to 50 bar injection pressure was used to inject de-ionised water. The hydraulic pump was fed from a supply loop at 10 bar, which could also provide some low pressure injection at an early stage of compression. It was possible to control the start and end of the injection period and to adjust the ramp rate of the high pressure injection pump. The high pressure piston pump could inject up to 65 grams per stroke. It was possible to achieve 100 grams per stroke if both high and low pressure injection was used. After discharge of the compressed two-phase mixture air to the pressurised separator, the water was fed back into the main atmospheric water tank as shown in Figure 1.



Rig Test Parameters

- Rig running speed (50 - 200 rpm)
- Atomizer type and set up
- Timing of main water addition
- Mass of water added
- Cylinder clearance volume
- Intake and discharge valve timings

Figure 1: Rig test schematic diagram and test programme

A high-speed system recorded transient data including cylinder pressure and atomiser pressures at half-degree crank intervals. The atomiser differential pressure allowed calculation of the transient water flow into the cylinder. A low speed data recording system was used for measuring parameters which changed slowly. More than 350 tests were performed, with the key operational parameters included in Figure 1. Examples of typical water flow rate profiles through the nozzles are shown in fig 2. The compression stroke starts at a crank angle of 180° and ends at 360°.

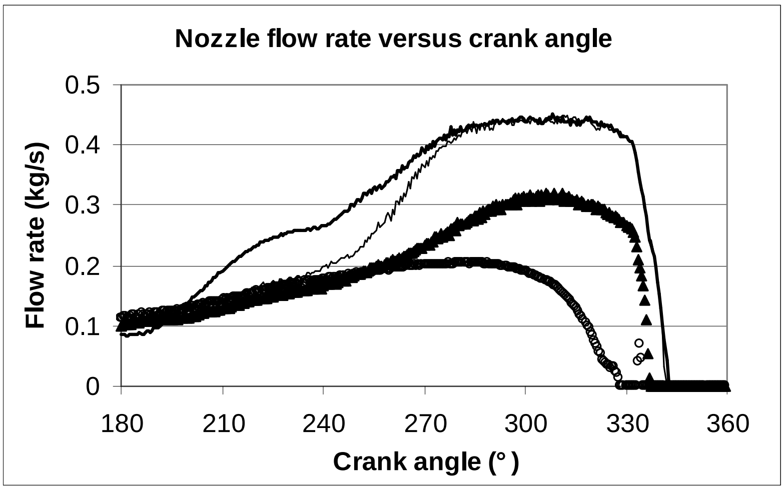


Figure 2: Typical water flow rate profiles through the nozzles.
(Compression stroke is from 180° to 360°)

2.2 Results from tests on the 'Proof of concept' compressor

The best experimental results were obtained at 100 rpm rig running speed using Spraying Systems N4 nozzles, which have a flow number of $2.5\times10^{-7}m^2$. In this case the air temperature in the cylinder (calculated from the measured cylinder pressure) was held below 100˚C for the whole transient and the work saving was 28% relative to the reference adiabatic case. Figure 3 shows some results at 200 rpm using N10 nozzles, which have a flow number of $6.3\times10^{-7}m^2$.

Figure 3a shows the plot of the in-cylinder pressure together with the pressure traces calculated for an ideal isothermal and an ideal adiabatic compression, both calculated with the same mass of gas and initial conditions as in the actual test. As expected, the experimental trace lies between the two ideal traces but is closer to the isothermal condition. The actual cylinder pressure exceeds 25 bar at the end of compression, but in calculating the work saving only the part of the compression up to this pressure was examined.

Figure 3b shows the calculated gas temperature and water flow rate for the same test run. In this example, the plot shows the gas temperature rising slowly as compression begins with a low level of water addition. When the main water spray begins, the gas temperature rises more slowly in spite of the increasing heat generation. Towards the end of the stroke, the heat generation rate increases rapidly and the water spray is unable to remove all of the heat. The maximum temperature occurs immediately before the delivery valve opens.

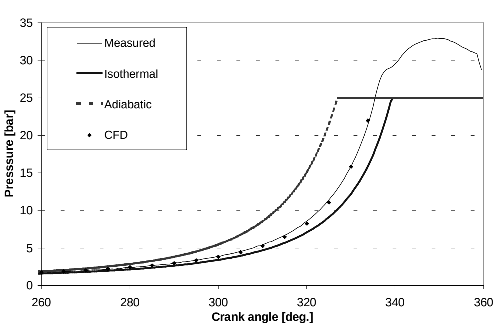


Figure 3a: Pressures

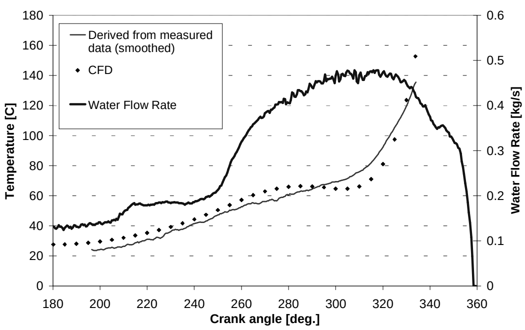


Figure 3b: Temperatures

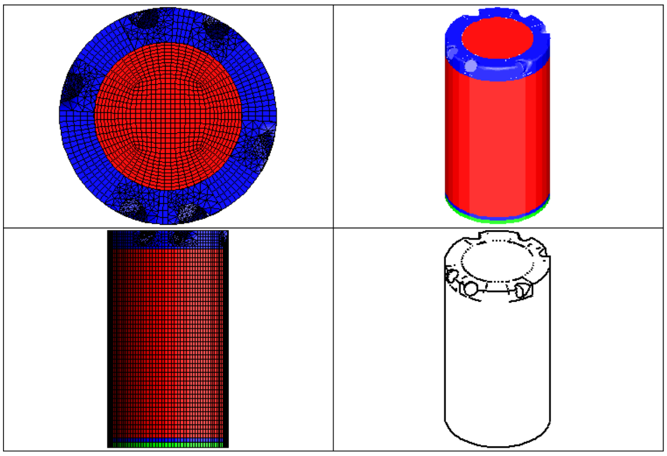
Figure 3: Pressures and temperatures during compression (rig test)

3. CFD Investigation

3.1 CFD methodology

The commercial CFD code Star-CD [4] was used to represent the transient development of the spray. A Lagrangian model is used for the liquid droplets, whereas an Eulerian model is used for the gas phase. Droplet break-up is modelled using the Reitz-Diwakar method [5]. Droplet collision was included in some of the calculations, using the method of O'Rourke [6] and the probability of collision leading to coalescence was calculated using the method of Brazier-Smith et al [7]. Droplet evaporation was assumed to be negligible. There is no representation of the nozzles themselves or of the primary atomisation process. Instead, a specified number of droplet "parcels" is injected at each time step, a few millimetres downstream from the nozzle tip. Each parcel consists of a number of droplets with the same properties (including diameter, location and velocity). The development and verification of a detailed CFD model of a single hollow cone spray is described by Stephenson et al. [1], and this model is used in the current work.

The CFD model consists of 160 000 computational nodes in a moving mesh. The transient simulations were carried out from a crank angle of 180° (bottom dead centre) to 333.8°, with 0.2° crank angle per step, i.e. a total of 769 steps for one case.



Geometry of the compressor

Figure 4: Diagram of CFD mesh at bottom dead centre (BDC)

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

18 different cases have been modelled, covering the following parameters:

- Spraying Systems Inc. nozzles type N10 and N4, producing hollow cone sprays
- The widest possible range of different water flow profiles through the nozzles.
- Two running speeds, i.e. 100 rpm and 200 rpm

The N10 parameters were derived directly from the single spray measurements. The N4 parameters were derived theoretically as no measurements were available.

The CFD predictions did not include all the water within the cylinder, for three reasons.

1. Some water was left in the cylinder after the previous compression stroke and was not included in the CFD.
2. Some water was injected just before BDC for this compression stroke and also was not included.
3. In view of a minor shortcoming in the version of Star-CD used for most of this work, droplets that struck a wall had to be deleted from any subsequent time steps

The correction for the missing water involved calculating, firstly, the volume occupied by air in the CFD model, and, secondly, the volume that would have been occupied had all the water been included in the CFD. Then the corrected mean pressure and temperature were found by assuming isentropic compression from the CFD volume to the actual volume.

3.2 Comparisons between CFD and experiment

Typical predictions for cylinder temperature and droplet locations are shown in fig 5. The contrast is clear between the cooler regions within the sprays and the hotter region towards the cylinder axis where the spray has not yet penetrated. Fig 6 compares measured and predicted pressures and temperatures in the cylinder at a crank angle of 333.8°C, which is as far as the CFD simulations were taken.

In general, the difference between measurement and prediction was greater for tests with lower than average water flow rate (ramp rate). In these cases, the nozzle differential pressure was lower than that in the single nozzle experiments. There was no significant variation in droplet size for the range of differential pressures used in the single nozzle experiments. However, it is possible that the droplet size increases at lower differential pressures than those used in the single nozzle tests. Assuming larger drop sizes in such cases would lower the heat transfer and increase the predicted final pressure and temperature, and the predictions would then be closer to the measurements.

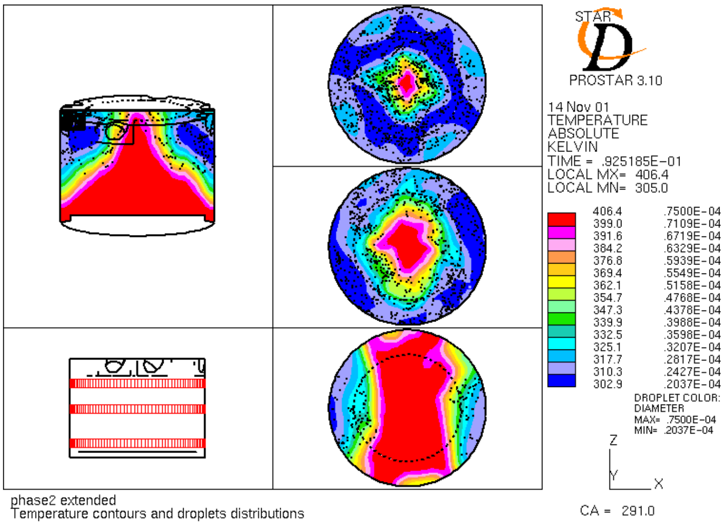


Figure 5: Temperature and droplet distribution in different cross sections through the cylinder at a crank angle of 291°.

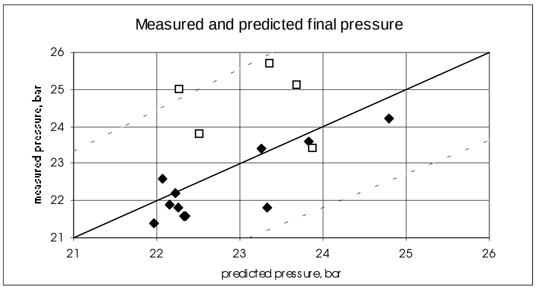


Fig 6a Comparison of predicted and measured pressures at CA=333.8°C

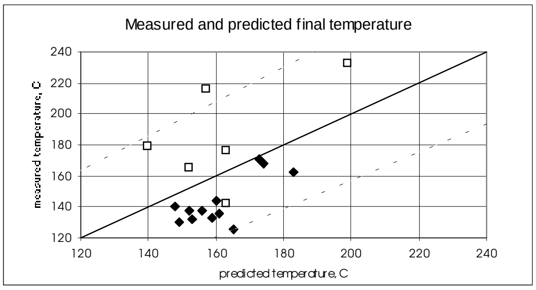


Fig 6b Comparison of predicted and measured temperatures at CA=333.8°C

Figure 6: Comparison of measured and predicted pressures and temperatures at CA=333.8°C.
Solid symbols=full ramp rate, open symbols=reduced ramp rate, ---=+ or -10% bands

Figure 7a and 7b also show results from CFD calculations of two different cases. Figure 7a has a smaller work saving compared to adiabatic compression than Figure 7b, which is the same case as shown in Figure 3. It is obvious that there is a strong correlation between the temperature distribution and the distribution of droplets (small black dots). It is also noted that a higher work saving is achieved in the case where the droplets have penetrated further. As in Fig 5, the central area without droplets is the area where the temperature is high.

21.1% work saving
[Case N10F230/05]

24.5% work saving
[Case N10F230/00]

a) b)

Figure 7: CFD plots of the temperature and droplet distribution in the cylinder at a crank angle of 291°.

A considerable amount of work has been undertaken to identify the factors causing the differences between simulations and experiments. For example, it is difficult to estimate accurately the amount of water in the cylinder at the start of the simulations. Also, the model assumes that the suction (inlet) valve has closed at the start of the simulation, whereas, in practice, it closes 14-20° after BDC, allowing some air to leave the cylinder during the early compression stage. The influence of various numerical parameters has also been investigated, included the number of droplet parcels that are assumed to be injected in each nozzle and time step.

4. Conclusions

The tests on the 'Proof of concept' compressor have shown that near-isothermal compression can be achieved, with substantial energy saving. A CFD method initially validated against experiments for a single spray has now been validated against measurements for a reciprocating engine with 18 nozzles. It is now being used with greater confidence in the third stage of this work, namely the Engineering Demonstrator.

5. Acknowledgements

This paper is published with permission of Innogy plc.

6. References

[1] Stephenson, PL., Abdallah, H., Powell, A., Jasuja, A. and Ng, K (2001): "Application of Liquid Sprays to Achieve Isothermal Compression, Part 1", Proc. 17th Annual Conference on Liquid Atomization & Spray Systems (ILASS), 2.- 6. September 2001, Zurich, Switzerland.

[2] Abdallah, H., Coney, M. and Stephenson, P. (2001): "Application of Liquid Sprays to Achieve Isothermal Compression, Part 2", Proc. 17th Annual Conference on Liquid Atomization & Spray Systems (ILASS), 2.- 6. September 2001, Zurich, Switzerland.

[3] Coney, MW, Stephenson, PL, Malmgren, A, Linnemann, C, Morgan, RE, Richards, RA, Huxley, R, and Abdallah, H, 2002, Development Of A Reciprocating Compressor Using Water Injection To Achieve Quasi-Isothermal Compression, to be presented at the International Compressor Engineering Conference, Purdue University, July

[4] Star-CD Methodology, Computational Dynamics, London, UK, 1999

[5] Reitz, R.D., and Diwakar, R., Effect of drop break-up on fuel sprays, SAE Paper 860469, (1986)

[6] O'Rourke, PJ, 1981, Collective drop effects on vaporising sprays, Ph D Thesis, University of Princeton.

[7] 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