



Julien Aymeric Beauquel

Computational Fluid Dynamics Modelling

Flow Behaviour in the Combustion
Chamber of a Spark Ignition Engine

Table of Content

Nomenclature, symbols and abbreviations

iv

Chapter 1: Current Status of CFD Modelling on Flows Inside the IC Engine

1.1 CFD Modelling on Flows Inside IC Engines and Methodologies 1-1

1.1.1 Experimental investigation and CFD modelling on flows in IC engines 1-1

1.1.2 Turbulence phenomena inside IC engines 1-5

1.1.3 CFD modelling considerations 1-7

1.1.4 Overview of simulation strategies 1-18

1.2 Recapitulation and Current Study Objectives 1-23

References 1-25

Figures 1-28

Chapter 2: Experimental Investigation of Steady Flow Inside an Open Cylinder

2.1 Introduction to the Devices for Flow Field Measurement 2-1

2.1.1 Thermal anemometry 2-1

2.1.2 Laser-Doppler anemometry and phase-Doppler anemometry 2-2

2.1.3 Pitot tubes 2-5

2.2 Experimental Setup 2-7

2.3 Experiment Method 2-12

2.4 Experimental Results and Analysis of Steady Flow in the Cylinder

2-13

2.4.1 Case 1: 3mm valve lift 2-15

2.4.2 Case 2: 6mm valve lift 2-16

2.4.3 Case 3: 9mm valve lift 2-17

2.5 Concluding Remarks on Experimental Work 2-18

References 2-19

Figures 2-20

Chapter 3: CFD Modelling of Steady Flow Inside a Cylinder

3.1 Introduction	3-1
3.2 Acquiring Cylinder Geometry by the Method of Digitalisation	3-2
3.2.1 Cylinder head scan	3-2
3.2.2 Inlet ports casting scan	3-4
3.3 CFD Modelling	3-7
3.3.1 Point data acquisition	3-7
3.3.2 Pre-CFD model creation	3-8
3.3.3 Full Gambit modelling (3 model valve lift)	3-12
3.3.4 Mesh quality	3-12
3.3.5 Numerical modelling	3-17
3.4 Results of Steady Flow Simulation and Discussion	3-20
3.4.1 Flow behaviour inside the cylinder	3-20
3.4.2 Y velocity component distribution inside the cylinder	3-23
3.4.3 Effect of valve lift	3-27
3.5 Conclusions	3-28
References	3-29
Figures	3-30

Chapter 4: CFD Modelling of Transient Flow Inside the Cylinder: Simulation with Dynamic Mesh

4.1 Introduction	4-1
4.2 Description of Piston Displacement	4-1
4.3 Valve Lift	4-5
4.4 Dynamic Mesh Modelling	4-11
4.4.1 Two-dimensional modelling of the flow inside a cylinder	4-11
4.4.2 Dynamic mesh type	4-12
4.4.3 Geometry modifications	4-14
4.4.4 Numerical details and boundary conditions	4-15
4.4.5 Animation of the flow behaviour	4-18

4.4.6 The use of cutting-planes for reporting the results	4-18
4.4.7 Mesh quality	4-21
4.5 Concluding Remarks on the Use of Dynamic Mesh	4-25
Figures	4-26

Chapter 5: CFD Modelling of Transient Flow Inside the Cylinder: Dynamic Flow Behaviour

5.1 Introduction	5-1
5.2 Transient Flow Behaviour Inside the Cylinder	5-2
5.2.1 Evolution of volume averaged static pressure in the cylinder	5-2
5.2.2 Evolution of tumble and swirl phenomena inside the cylinder	5-4
5.2.3 Turbulence field inside the cylinder	5-10
5.2.4 Mass flow rate evaluation for transient flow	5-10
5.3 Discussion	5-12
5.4 Conclusions	5-13
References	5-15
Figures	5-16

Chapter 6: Conclusions Derived From the Study

6.1 Conclusions	6-1
6.1.1 Effects of cylinder geometries on flow behaviour	6-2
6.1.2 CFD modelling of steady flow inside the cylinder	6-2
6.1.3 CFD modelling of transient flow inside the cylinder	6-3
6.2 Recommendations for Further Work	6-4

Appendices	A-1
-------------------	------------