MULTI-DIMENSIONAL NUMERICAL SIMULATION OF SCAVENGE FLOW IN A SMALL TWO-STROKE ENGINE

Sameera Devsritha Wijeyakulasuriya

This thesis is submitted to the Department of Mechanical Engineering
of the University of Moratuwa
in partial fulfillment of the requirements for the Degree of
Master of Science

Department of Mechanical Engineering
University of Moratuwa, Sri Lanka
October 2007

ACKNOWLEDGMENT

I'm grateful to the National Science Foundation of Sri Lanka, which funded me for this research. The Guidance and help rendered at every stage of this research by my supervisors Prof. Razi Nalim, Associate Professor, School of Mechanical Engineering, IUPUI, USA and Dr. A.G.T. Sugathapala, Head of Department, Mechanical Engineering, University of Moratuwa are appreciated with heart felt gratitude.

The assistance of, Mr. Indika Perera, Lecturer, Department of Mechanical Engineering, University of Moratuwa and the staff of the CAD Centre at the University of Moratuwa, and the assistance of Dr. S. A. Khalid and the staff of the Computer Networking Centre at IUPUI, USA are also acknowledged.

The support received from es-ice support team of CD-adapco towards resolving moving grid issues are also appreciated. Computing resources from Indiana University's AVIDD supercomputing cluster, VLSI supercomputer and academic licensing of STAR-CD were valuable in this work.

Visits to vehicle manufacturers and dealers were also helpful, and the insights of David Peries Motor Company, Sri Lanka, LML Industries, India, and Mr. Narayan Iyer (formerly of Bajaj Auto, India) are particularly appreciated.

Finally I'd like to thank my family for all the support and understanding throughout this research.

ABSTRACT

Computational study of scavenging flow in a typical small vehicle engine is accomplished using multi-dimensional moving-mesh unsteady and quasi-steady models. A representative two-stroke 5 port engine was selected and sectioned to measure the geometrical features of the transfer ports. The commercial codes Auto CAD and Pro-Engineer was used to model the cylinder and ports. The commercial code Star-CD was used to analyze the unsteady gas exchange process within the engine combustion chamber soon after combustion. Pressure boundary conditions were used to represent crankcase conditions and exhaust tail pipe conditions at the scavenge port inlets and the exhaust port exit respectively. The simulation was stated at top-dead-centre (TDC) after combustion, assuming no exhaust gas remained in the engine from the previous cycle. For moving-mesh calculation, a cycle simulation is performed in two stages, separated by a theoretical constant-volume combustion calculation. The TDC calculation was initialized using a preliminary simulation staring at the bottom-dead-centre (BDC) assuming no exhaust gas remained in the cylinder at the start of simulation at Parallel computing was used to run the simulation.

Both quasi-steady and transient simulations indicate that typical multi-port inlet placement and timing allow significant short-circuiting of mixture, consistent with emissions observations. Residual gas is well scavenged from the hemi-spherical head cylinder. Backflow of combustion gasses into the transfer ports were observed upon opening of the transfer ports which indicate an inadequate blow down period. The inlet angles of the mixture from transfer ports are oriented upwards towards the dome of the cylinder and are satisfactory. The relative positioning and orientation of the scavenge ports promote good mixing and helps to reduce the inflow velocities, though considerable short-circuiting is still present in the engine.

TABLE OF CONTENTS

Page

Page

LIST OF TABLES

LIST OF FIGURES