Journal of Applied Sciences

ISSN 1812-5654

Journal of Applied Sciences 11 (1): 36-45, 2011
ISSN 1812-5654 / DOI: 10.3923/jas.2011.36.45

CFD Analysis of Gas Exchange Process in a Motored Small Two-stroke Engine

[1]Ramamurthy Hariharan, [1]N.V. Mahalakshmi and [2]Jeyachandran Krishnamoorthy
[1]Department of Mechanical Engineering, C.E.G. Anna University, Guindy, Chennai, 600 025, India
[2]Formerly of Department of Mechanical Engineering, C.E.G. Anna University, Chennai-25, India

Abstract: The major shortcoming namely short-circuiting of fresh charge into atmosphere of a two-stroke engine has been correlated to the spatially and temporally changing in-cylinder velocity patterns. In a small engine, modifications in the cylinder head for carrying out experiments to investigate the gas exchange phenomenon become extremely difficult. CFD study offers the effective alternative for such investigations and this study has been undertaken to develop a model using a commercial CFD code SRAR-CD and study the gas exchange process in a small 100 cc SUZUKI two-stroke engine. The geometric model has been created using GAMBIT, a preprocessor for mesh creation initially and importing it into the STAR-CD environment. Further development of the geometric and simulation model is carried out by the in-built facilities of STAR-CD. The comprehensive CFD model consists of the clearance and displacement volumes of the cylinder, a right-cylinder shaped equivalent crankcase volume, scavenge and booster and exhaust ports and passages. The model study has been successful in providing predictions for crankcase pressures and port passage velocities comparable to the experimental results obtained by FLDV measurement techniques on a similar production line 100 cc SUZUKI crankcase scavenged 2-stroke engine. The details of the comparison between CFD predictions and experimental results have been presented.

Key words: Small two stroke engines, scavenge ports, equivalent crankcase volume, short-circuiting, velocity patterns, CFD study

INTRODUCTION

In a conventional spark ignition crankcase scavenged two stroke engine the naturally aspirated air-fuel mixture is compressed in the crankcase and delivered to the engine cylinder through scavenge and booster ports which are uncovered by the piston in its expansion stroke. This process takes place simultaneously when the burned products in the cylinder are being discharged through the exhaust port which is uncovered earlier in the expansion stroke. The time period for this process of filling and emptying of cylinder contents gets shorter and shorter as the engine speed increases (Heywood and Sher, 1999). Mathematical description of this dynamic process has not been easy and many experimental and mathematical tools have been used in the past to understand this dynamic phenomenon. Jante (1968) was the first to develop a useful tool with the comb of pitot tubes to study and evaluate the scavenging process inside the cylinder of a 2-stroke engine. Many attempts have been made after the experiments of Jante. Sher (1985) has reported practical methods to study the scavenging process. Reddy *et al.* (1986) have studied the in-cylinder air movement in a 2-stroke engine using hot wire anemometer. Ikeda *et al.* (1991a) have conducted scavenging flow measurements in a 2-stroke engine by Fiber Laser Doppler Velocimeter (FLDV) on a SUZUKI 100 cc engine for Motored and Fired conditions. Hilbert and Falco (1991) have developed a new measurement technique using Laser Induced Photochemical Anemometry for evaluating velocity and velocity gradients over a chosen plane in a motored 2-stroke engine during scavenging. Apart from the above, many attempts, as chronicled by Heywood and Sher (1999) have been made to study in-cylinder parameters in an IC engine employing mathematical tools. Amsden *et al.* (1985) have developed a comprehensive model for 2D and 3D engine simulations using a KIVA code followed by Changyou and Wallace (1987), Raghunathan and Kenny (1997), Epstein *et al.* (1991) and Yang *et al.* (2005), have proposed the Reynolds Stress Turbulence Model (RSTM) for anisotropic turbulence engine flows and have modified the KIVA code to include RSTM for simulation of combustors and direct injection. Basha and Gopal (2009) have investigated the research undertaken between 1978 and 2008 in respect of in-cylinder fluid flow, turbulence and spray characteristics and have concluded that some scientists considered the Re-Normalized Group (RNG) k-ε model as the best suited for engine simulations.

Corresponding Author: Ramamurthy Hariharan, 23, SBO Colony, Maharaja Nagar Tirunelveli 627 011, Tamil Nadu, India
Tel: +91-462-257 2310

Verhelst and Sheppard (2009), have proposed a unified approach to sub-models involved in multi-zone modeling of engine simulation especially for turbulent combustion modeling.

Most of the previous investigations have yielded only limited data though reliable. In other words, the attempts have been to obtain a time history of the flow patterns as the cycle progresses. In investigative techniques, the advantages of a mathematical model are manifold. CFD analysis which is one of the widely accepted mathematical tools provides instantaneous temporal and spatial values of the various performance parameters influenced by the different operating variables of an engine. The CFD study is also very helpful when the engine is too small in which modifications/attachments to the engine head to facilitate conducting experimental studies becomes an onerous task. The objective of this study is to adopt the CFD analysis to a small crankcase scavenged 2-stroke engine and establish its suitability, dependability and effectiveness. The data and information available for comparison are the published results of the experiments on a SUZUKI 100cc engine by Ikeda *et al.* (1991a, b). The experimental values taken for comparison are the scavenge port velocity and the crankcase pressure for a motored engine. A calibrated CFD model of a 2-stroke engine provides a benchmark for further work especially for inlet and exhaust port design since the experimental setups are very expensive and time consuming, apart from being unviable for minor or major changes in design and operating parameters. The CFD study is also useful to analyze the in-cylinder flow pattern in the 2-stroke engine thereby eliminating the need for an expensive experimental set-up and to determine the adaptability of a CFD tool to the engine concerned. The inclusion of a representative crankcase volume in the model to make the analysis more meaningful and comprehensive is one of the unique features of this study. Further, no time dependent histories of boundary conditions have been externally imposed on the simulation model. It may also be mentioned that the initial uncertainties in the geometric model are resolved by parametric studies as a result of which the predictions and experimental results show close agreement. The study also aims to determine the start of crankcase compression specific to the two-stroke engine.

ENGINE DETAILS

The engine specifications used in the experiment are as follows:

Type of engine	: Two-stroke spark ignition crankcase compression, single cylinder
Stroke volume	: 98.17cc
Bore×stroke	: 50×50 mm
Compression ratio	: 6.59
Exhaust port opening (EO)	: 95°ATDC
Exhaust port closing (EC)	: 265°ATDC
Scavenge port opening (SO)	: 120°ATDC
Scavenge port closing (SC)	: 240°ATDC
Scavenging type	: Schnurle

The following details were deduced from the data in the paper and details from similar engines:

Exhaust port size	: 25×30 mm
Scavenging ports	: 11×11 mm
Transfer ports	: 11×25 mm
Connecting rod length	: 100 mm
Crankcase volume	: 363 cc

CFD AND STAR CD

Figure 1 describes the CFD methodology in a nutshell (Gosman and Ideriah, 1983). The Navier-Stokes equations for conservation of mass, momentum and energy and standard turbulence forming the basis of the CFD methodology is used by the STAR processor and the methodology (CD Adapco Group, 2001) is outlined in the manuals. The k-ε model is the most widely used complete

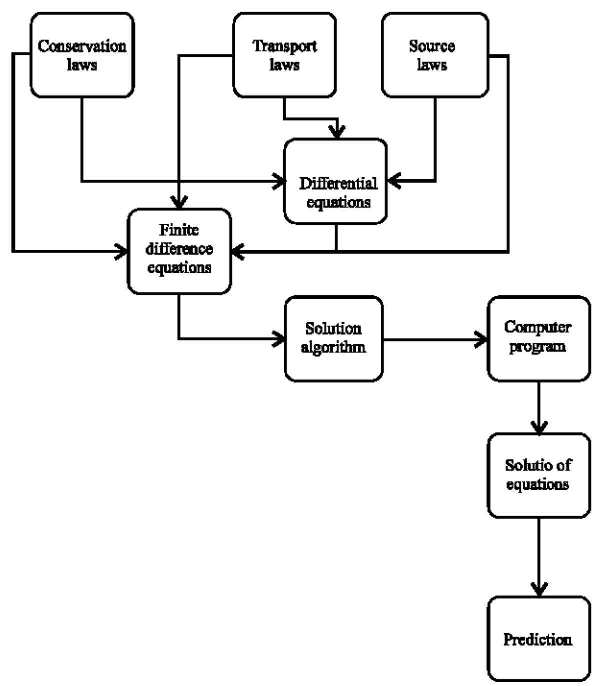

Fig. 1: Block diagram showing the basic methodology for CFD approach

turbulence model (Pope, 2005) and it is incorporated in STAR-CD like in most other commercial CFD codes.

CFD MODEL CREATION

In the CFD analysis the simulation model is created from the geometric model after imposition of initial and boundary conditions. The computing domain of the engine cylinder, ports and the crankcase are divided into a number of discrete grids or cells in the pre-processor in STARCD and a moving mesh feature simulates the changing positions of the piston top. The model of the SUZUKI 100 cc naturally aspirated carbureted spark ignition engine, used in the experiment, has been created in multiple stages and from multiple blocks taking into account the various actual dimensions of the production line engine as detailed in earlier. The combustion chamber of the engine is nearly a hemispherical dome of about 17.2 mm radius with a small flattened portion for fixing the spark plug at the top. This chamber has been created from GAMBIT grid generation package and imported to Pro-Star environment. The grid structure consists of body fitted non-orthogonal hexahedral meshes all connected in an arbitrary fashion. The cylinder portion of 50 mm diameter and about 51 mm height is then extruded. For the ease of analysis, the crankcase has been extruded as a right cylinder equivalent to the total volume of the crankcase at BDC plus 98 cc of piston displacement volume. The grids for scavenge ports on either side of the Exhaust port and those opposite to Exhaust port are created (Fig. 2) using the local coordinate systems at the respective port locations and then connected to the crankcase by merging the vertices at the connecting faces. The grid structure for the Exhaust port is then modeled. In an earlier CFD study (Hariharan and Krishnamoorthy, 2006) only a flat top piston has been used to study turbulence and its enhancement in small two stroke engines with the stroke divided into equivalent number of 0.25 mm layers for a 46 mm bore and 42 mm stroke engine. The computer time for analysis of the present engine of 50 mm stroke and 50 mm bore with a proportionately higher crankcase volume has been made shorter with the use of 1mm layer piston (Fig. 3) instead of a model with 0.25 mm layer. Grid independence analyses conducted separately have established that even a 2 mm layer does not affect the results. However for creating ports of 11 mm height, 1 mm layers have been used in this study.

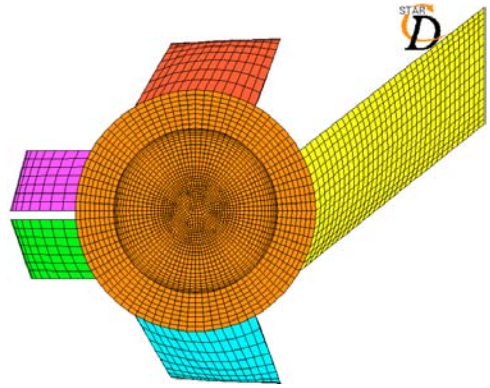

Fig. 2: Plan View of the model showing the port orientations of the SUZUKI 100 cc 2-stroke engine

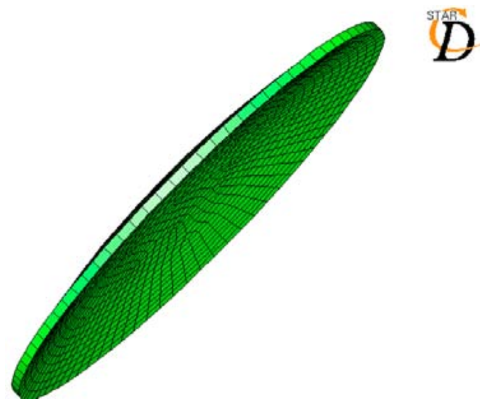

Fig. 3: A layer of piston with a crowned top of the 100 cc model

Model features: STAR-CD, the fluid dynamic based computational code, solves the partial differential equations for the conservation of mass, momentum, energy and species concentrations and uses numerical methods to solve the finite difference equations arrived at from the partial difference equations after integration at the control volumes. Ideal gas laws have been applied for the fluids being analyzed namely the cylinder contents, crankcase contents and the exhaust fluids and solved for static enthalpy using thermal option for cold flow analysis. Specific heat is defined as a polynomial function of temperature for N2 which is used as the background fluid for all the fluids. The piston movement is simulated by the moving mesh feature of the software. Layers of the piston are reduced to zero volume cells, termed as deactivation of cells, in a compression process. The movement of the piston is decided by the in-built programme in the software based on the connecting rod length, speed of rotation, stroke and crank radius of the engine. The contents of a layer so removed are added into its neighboring layer and a modification of cell geometry is executed in which the bottom vertices of the layer in question are shifted up by the EGRID change grid commands. In the expansion process cells are restored to their original volumes and the vertices are shifted to their original position by the activation of cells. The unsteady gas exchange process during port openings and closings is simulated by the facility of ASI (Arbitrary Sliding Interface) commands. Both the transfer and scavenge ports have been named as scavenge ports in keeping with the terminology used in the paper referred. The layers of the port cells and the cylinder cells are either connected or disconnected depending upon the direction of piston movement by the ASI commands thereby allowing for the fluid flow to take place between the cylinder and the crankcase and exhaust port.

The model contains finite volume cells as follows:

Scavenger passages	:	17556
Clearance volume	:	17094
Cylinder	:	64700
Crankcase	:	219980
Exhaust passage	:	9800

Simulation model: The next step in the setting up of the model is assigning different boundaries for the cylinder and crankcase body, inlet into the crankcase and the exhaust followed by writing the code for the mesh movement. The moving mesh code consists of number of events specifying the piston and crankcase positions with respect to time or the crank angle as the crank shaft rotates. An event is a collection of elementary mesh operations which occur at a given time and an example of such an event is the cell layer Deactivation or Activation as explained earlier.

Ideally throttle is best defined by the changing density of the charge for the different throttle openings by specifying an appropriate initial density (based on the mass of air filling the cylinder at BDC). Inlet is designated as a WALL boundary on the surface of the crankcase after other options have been tried.

The boundary options for the exhaust port are an Outlet, Wall or a Pressure boundary. In the case of an Outlet boundary a not split option has been used. In the Pressure boundary option either a positive or negative pressure has been tried. In the wall boundary, among many options an adiabatic wall is chosen.

The following assumptions have been made while creating the model.

Assumptions: (1) For the crown piston model the curvature has been designed using the available data. (2) To a great extent the skews in the ports have been modeled to represent the actual engine. However, small differences are bound to exist between model and the actual engine since the detailed drawings of the engine are not available. (3) An adiabatic wall boundary in which no exchange of energy or mass takes place across the wall is chosen for the cylinder and crankcase body. (4) A cylindrical Crankcase has been modeled assuming the diameter as that of the piston for ease of analysis. (5) The port connections with the cylinder have been styled to conform as far as possible to the actual orientations and positions in the engine. (6) The port sizes, connecting rod length and crankcase volume have also been assumed as detailed earlier. (7) In the absence of any details the exhaust system has been built on the basis of a similar engine.

PROCEDURE

Having created the model, the following procedures have been adopted to calibrate the model by comparing the predictions with the results of the experiment and thereby resolve the uncertainties. The analysis runs have been carried out at 3000 rpm and data collected for reporting the velocity components, U, V, W, relative and absolute pressure P and temperature T, etc. The collection of all data has been restricted to every 5° CA since, the size of the data file is likely to be too large for shorter intervals.

- BDC start for a flat top piston model for a full 360° cycle of crank rotation (Fig. 4)
- BDC start for the crown top piston model for a full 360° cycle (Fig. 5) without simulating crankcase compression. Inlet to crankcase is designated as a wall boundary. Cylinder compression has been assumed from EC to TDC
- BDC start for crown top piston model for a full 360° cycle with simultaneous simulation of crankcase compression (Fig. 6). Inlet to crankcase has been assumed as a wall boundary. Compression of the cylinder contents has been assumed from EC to TDC

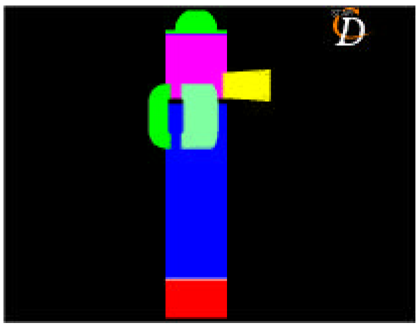

Fig. 4: Flat top Piston Model

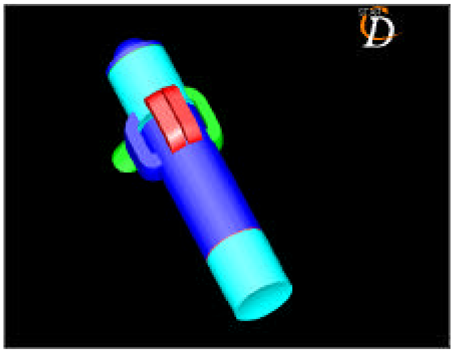

Fig. 5: View of the model with Piston at BDC showing full crankcase volume without crankcase compression

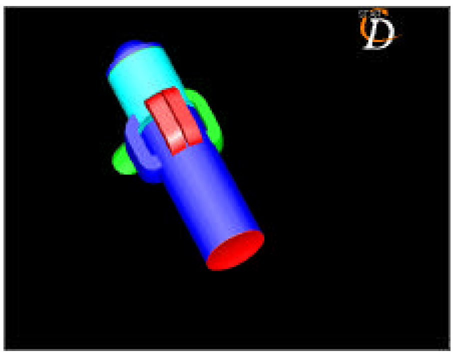

Fig. 6: View of model with the Piston at BDC and crankcase Compressed at time t = 0.0

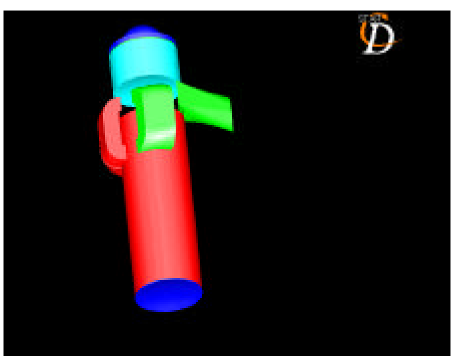

Fig. 7: View of the model with the Piston top just above Exhaust port at time t = 0.0

- TDC start for crown top piston model for a full 360° cycle with simultaneous simulation of crankcase compression. Inlet to crankcase is treated as a wall boundary. Compression of the cylinder contents has been assumed from EC to TDC
- EC (exhaust port closing) start for crown top piston model for full cycle (Fig. 7) with simultaneous simulation of crankcase compression. Inlet to crankcase has again been treated as a wall boundary. Effective cylinder compression has been assumed from EC to TDC, i.e., when the volume is totally isolated when all the port openings are closed
- SC (Scavenge port closing) start for the model with crank case volume in a fully compressed state
- Start at inlet reed valve (IO) opening (at about 60° BTDC) imposing atmospheric conditions for the crankcase contents
- SC start with a crankcase volume of 325 cc and crankcase simulation
- BDC start for the crown top piston model for a full cycle with crankcase simulation and with Inlet to crankcase as a wall boundary assuming that compression of cylinder contents starts at BDC and continues upto TDC, the run split into two parts, from BDC to TDC and from TDC to BDC. For the run from TDC to BDC, the initial conditions for the cylinder have been those obtained at the end of the first part while for the crankcase contents atmospheric conditions have been adopted to reproduce actual engine conditions

RESULTS AND DISCUSSION

Predictions obtained from CFD analysis of the SUZUKI 100 cc engine running at 3000 rpm without reaction flow (combustion of the charge) and the results of the experiment conducted on the same engine under motored conditions at full throttle and run at 3000 rpm are compared in the following paragraphs. It has not been possible to reproduce the exact curve obtained by the experiment for comparison since the values have been scaled from the available plots in the referred papers in the absence of tabulated data. In the experiment, the ensemble-averaged mean velocity values obtained by FLDV technique have been shown to be having good agreement with raw data and therefore, Ikeda *et al*. (1991a, b) have concluded that ensemble averaged mean velocity can describe the major features of the scavenging flow significantly. Moreover the values from CFD analysis have been obtained with the interval of 5° CA since smaller intervals are bound to increase computer storage space for each analysis run. It is to be noted that the curves of the predictions are from raw data and no curve fitting has been attempted. A noticeable feature in the experimental results is the negative crankcase pressure recorded prior to the SC. This negative pressure value is the lowest just prior to SC and increases as the piston moves towards TDC. This aspect of the result is in total variance with the analysis runs in the CFD.

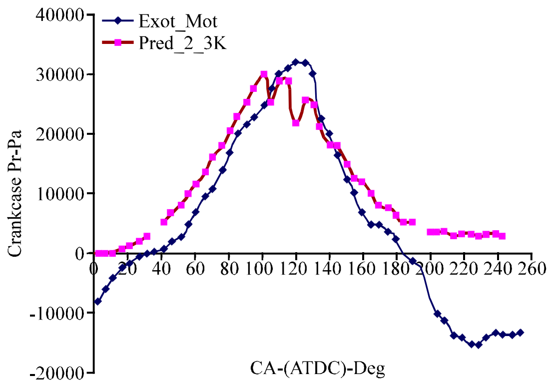

Fig. 8: Crankcase pressure at 3000 rpm and cylinder compression from EC to TDC

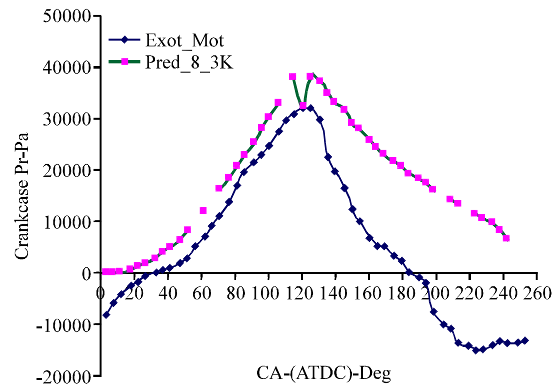

Fig. 9: Crankcase pressure at 3000 rpm cylinder and crankcase comprn from BDC to TDC

When, the inlet reed valve opens the crankcase is exposed to atmosphere and the inside pressure tends to become atmospheric about 30° to 40° BTDC and continues to remain atmospheric even beyond few degrees ATDC. However, in the curve reported by experiment the crankcase pressure remains below atmosphere even at the TDC and the pressure inside the crankcase is seen to slightly increase as the piston approaches TDC. This particular phenomenon of the experiment has not been possible to be simulated in the CFD analysis. In the STARCD code, in the table for user defined conditions, options of relevance of independent variable available for the Inlet boundary is Time (t) and the dependent variables are the velocity components (U, V, W) and not the pressure. In the pressure boundary the independent variables are only the X, Y, Z coordinates and the dependent variable is the pressure varying with respect to time and hence the time variant crankcase pressure for the period during which the crankcase is connected to inlet manifold becomes difficult to be simulated in this CFD study.

The results of the CFD analysis of some of the select models and under different conditions are plotted below as crankcase pressure vs. CA (Fig. 8-11) and port passage velocity vs. CA (Fig. 12-15).

Figure 8 presents the predictions of a crown top piston model along with the experimental results. This model had TDC start and a crankcase compression initialized from TDC itself and terminated at the exhaust port opening. The peak value of the pressure is seen to be lower and the intermediate values are higher. Since, the exact crankcase volume was not known while creating the model a value of 363 cc was extrapolated from available information. The peak value is lower because the assumed crankcase volume is probably higher. The other contributing factor is the uncertainty about the exact

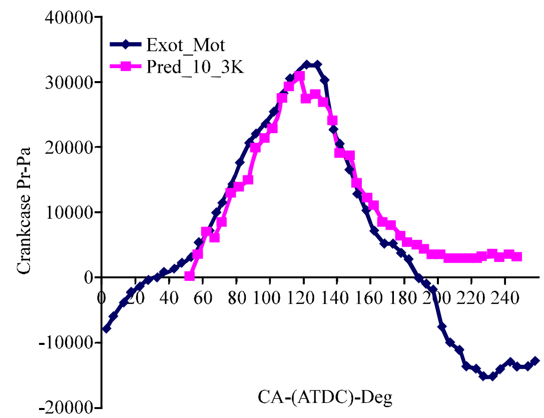

Fig. 10: Crankcase pressure at 3000 rpm cylinder compression from EC to TDC

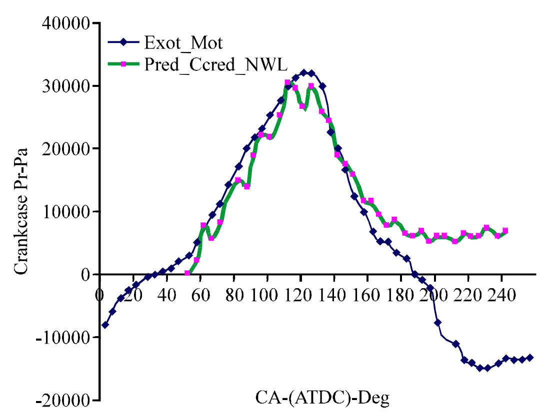

Fig. 11: Crankcase Pressure at 3000 rpm cylinder, crankcase compression from BDC to TDC and 325 cc Vol

position of isolation and start of crankcase compression in the expansion stroke of the piston. The intermediate

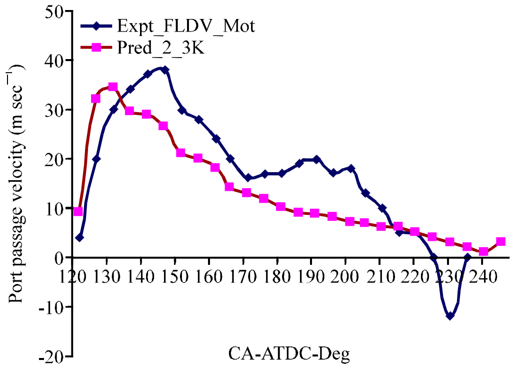

Fig. 12: Port passage velocity at 3000 rpm cylinder compression from EC to TDC

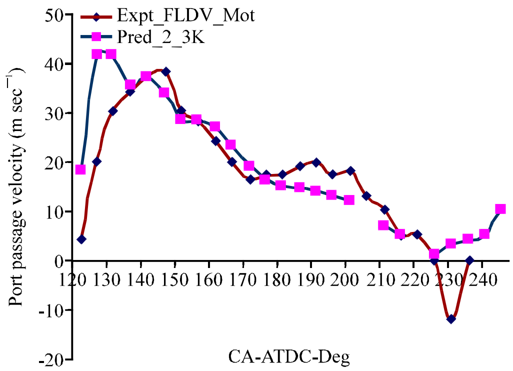

Fig. 13: Port Passage Velocity at 3000 rpm, cylinder and crankcase compression from BDC to TDC

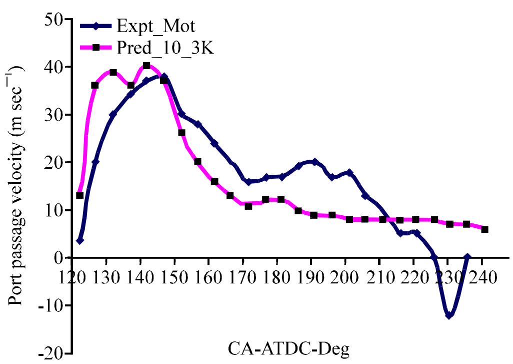

Fig. 14: Port passage velocities at 3000 rpm Cylinder compression from EC to TDC

crankcase pressure values seen in the graph may be higher since the compression does not probably start at TDC.

The predictions of the model with crankcase compression from TDC to BDC of the expansion stroke of the piston and a volume of 363 cc are shown in Fig. 9. It is seen that trend is reflected well in the predictions but both

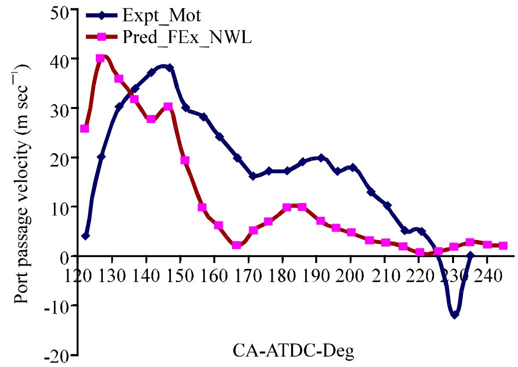

Fig. 15: Port passage velocity at 3000 rpm, cylinder, crankcase compression from BDC to TDC and 325 cc Vol

the intermediate values and peak values are higher. It has to be therefore inferred that the start and extent of compression have to be corrected to achieve closer agreement with the experimental results.

The predictions presented in Fig. 10 are for a model with 325 cc of crankcase volume. The moving mesh code was written for this model specifying crankcase compression to start from 45° ATDC in between TDC and Inlet Valve closing (IC). It is seen that the trend as well as the peak value has good agreement with the experimental results. Deficiency is still noticed since the values beyond BDC do not quite reflect the experimental results. Inefficient exhaust modeling probably caused this discrepancy. Attention to this aspect of the modeling is therefore focused in later models.

In Fig. 11, crankcase pressure curves for the experimental results and the CFD model analysis for the 100 cc Suzuki engine are presented. In this model, crankcase compression was initialized at 45° ATDC and allowed to continue till SO . The exit of the exhaust duct was imposed with an OUTLET boundary. Exhaust was designed for the full length of 1000 mm. At the connection to the short duct it was created with 24 mm diameter and flared to 32 mm at the middle. It was then tapered to 24 mm diameter at the end. Turbulence model was maintained as the near wall treatment model with Norris and Reynolds coefficients. However, in this curve also the trend beyond 180° is not projected satisfactorily compared to the experimental results. The discrepancy is attributed to the diffuser section in the exhaust system.

Figure 12 shows the predictions of the port passage velocities in a crown to piston model initialized with crankcase compression from TDC and continued upto EO of the stroke. It was seen in Fig. 8 showing crankcase pressure predictions that the model had deficiencies and needed improvement.

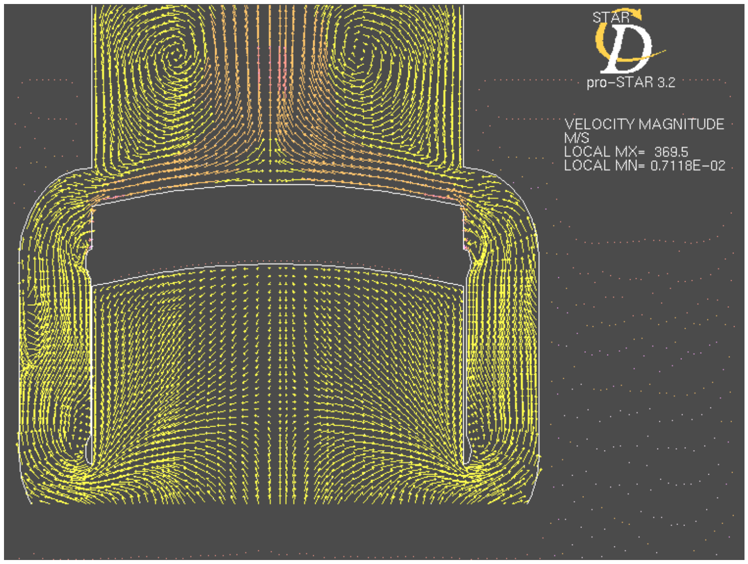

Fig. 16: Velocity pattern after scavenge port opening in section of the cylinder with scavenge ports on either side of exhaust port at 135° ATDC

The fact that the initial peak velocity scales with the peak pressure attained by the crankcase is demonstrated remarkably well by the Fig. 13. The model used for this analysis was with the crankcase volume of 363 cc and the crankcase compression was started from the TDC itself. It is seen that the peak in the velocity attained in the model is higher than the experimental values. It is seen that the predictions mimic the experimental values reasonably well upto 170° ATDC.

Comparing Fig. 13 with Fig. 14 in which the crankcase volume was reduced to 325 cc and crankcase compression initialized from 45° ATDC, it is seen that the peak value has improved and matches the experimental results well. In this model even the slight increase in the velocity values beyond 170° ATDC found in the experimental results has been reflected in the predictions. It is to be inferred therefore that for best agreement improvement in exhaust modeling has to be attempted.

Figure 15 shows that the agreement between the predictions and the experimental values is maintained in this model also. The model used for obtaining the curve in Fig. 15 has all the conditions used in the model for Fig. 14, but with a full exhaust system of about 1000 mm long and a diffuser section created by increasing the diameter from 24 to 32 mm in the middle and tapering back to 24 mm at the end. The exhaust created in this model is nearly equivalent to that in the engine. Any further improvement in the predictions have to be by better turbulence models using correct constructional details of the exhaust in particular.

A very good agreement is noticed in the port velocity and crankcase pressure curves of the experiment and predictions of the later CFD models. The plots for the reduced crankcase volume of 325 cc are seen to have better agreement suggesting that the actual volume of crankcase must be between 363 and 325 cc. The crankcase pressure curves show some closeness to the experiment curves in the regions up to the scavenge port openings but show marked departure after the scavenge port openings probably due to the inaccurate simulation of the exhaust system. The exhaust system influences the scavenging process and affects the cylinder pressure and in turn introduces differences in the crankcase pressure thereby increasing the deviations from actual working engine values. The typical velocity patterns at some critical crank angle positions are presented in Fig. 16 and 17.

Exhaust pressure predictions from CFD study have not been presented in the results since invariably the values have all been flat with negligible variations with respect to the crank gangle. Ikedag *et al*. (1991a) have stated that port passage velocities scale with the differential pressure between the crankcase and the exhaust. It was seen in the velocity curves that the trend is almost reproduced in the CFD predictions in spite of the flat exhaust pressure curves. However, Haworth *et al*. (1993), have stated that the driving force for the scavenge flow is from the difference between crankcase and in-cylinder pressures while investigating the gas exchange process by imposing a history of the externally measured

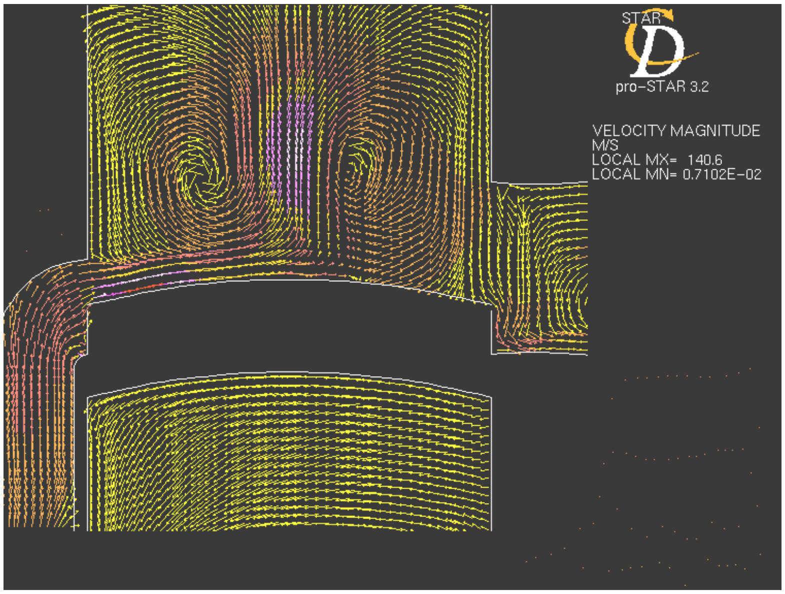

Fig. 17: Velocity pattern after scavenge port opening in a section of the cylinder showing the exhaust port and a scavenge port opposite to exhaust port at 135° ATDC

time-dependent boundary conditions in a mathematical model. The CFD predictions seem to substantiate this reasoning. Lai *et al*. (1993) have conducted a CFD study to demonstrate its capability as a predictive tool and have concluded that port passage velocities depended on the transfer port pressures which to certain extent coincides with the view expressed by Haworth *et al.* (1993). They have also stated that CFD studies were able to provide detailed information about relevant flow processes in motored two-stroke engines and have referred to the works of Epstein *et al*. (1991). Comparing the qualitative predictions of this study with the predictions of Epstein *et al.* (1991) it is seen that the present work has reinforced the earlier studies while at the same time has also presented spatial and temporal quantitative results. Further, Ikeda *et al*. (1991a, b) have explained the negative velocities as the consequence of the leakage past the pistons. This aspect of the experimental results has not been investigated since the crevices can not be generated easily in the CFD model.

CONCLUSIONS

Since, the experimental methods are expensive and require elaborate testing and measuring techniques and are limited in their scope of predictions a CFD study has been undertaken. The following are some of the highlights of the investigation:

- CFD predictions were so far confined to profiles and patterns of the flow field. The comparison of quantitative values of CFD models for crankcase pressures and port passage velocity with experimental results, to be attempted for the first time in this investigation, was very successful
- The uncertainty about the actual crankcase volume of the Suzuki engine was resolved and it was concluded that it may lie between 325 and 363 cc. The exact start of crankcase compression in the expansion stroke was investigated at different crank positions and was found to be closer to 45° ATDC. The assumed port sizes were found to be correct since the port face areas were found to be in agreement with the reported values in the experiment. The port opening timings coinciding with the values reported in the experiment confirmed that the correct connecting rod length was assumed
- Designing the exhaust to its full length and changing the turbulence model to include the near wall treatment for the exhaust body produced good predictions
- The predictions have demonstrated that the CFD model study is a useful tool in understanding and visualizing the dynamic process of scavenging

However, newer turbulence models such as RSTM and LES have to be applied to determine whether advanced turbulence models can resolve the complexities of the exhaust modeling.

ACKNOWLEDGMENT

The above study was conducted in the simulation laboratory of the Internal Combustion Engines division of the Faculty of Mechanical Engineering in The College of Engineering, Guindy, Chennai, Tamil Nadu, India during the years 2006 to 2009. The study was initially conducted using STAR-CD v 3.15 and later using STAR-CD v 3.26.

REFERENCES

Amsden, A.A., T.D. Butler, P.J. O'Rourke and J.D. Ramshaw, 1985. KIVA-A comprehensive model for 2-D and 3-D engine simulations. Soc. Automot. Eng. Trans., 94: 4.1-4.15.

Basha, S.A. and K.R. Gopal, 2009. In-cylinder fluid flow, turbulence and spray models: A review. Renewable Sustainable Energy Rev., 13: 1620-1627.

CD Adapco Group, 2001. Methodology. Version 3.15, Computational Dynamics Ltd., London.

Changyou, C. and F.J. Wallace, 1987. A generalised isobaric and isochoric thermodynamic scavenging model. Soc. Automot. Eng. Trans., 96: 933-947.

Epstein, P.H., R.D. Reitz and D.E. Foster, 1991. Computations of a 2-Stroke engine cylinder and post scavenging flows. Soc. Automot. Eng. Trans., 100: 1014-1028.

Gosman, A.D. and F.J.K. Ideriah, 1983. A general computer program for two-dimensional turbulent, recirculating flows. Report No FM-83-2 University of California, Berkeley.

Hariharan, R. and J. Krishnamoorthy, 2006. CFD study of turbulence and its enhancement on a small two-stroke engine model. Proceedingsof the Scientific Computing to Computational Engineering, July 5-8, Atens, Greece, pp: 356-363.

Haworth, D.C., M.S. Huebler, S.H. El-Tahry and W.R. Matthes, 1993. Multidimensional calculations for a two-stroke-cycle engine: A detailed scavenging model validation. SAE Trans., 102: 1-26.

Heywood, J.B. and E. Sher, 1999. The Two-Stroke Cycle Engine Its Development, Operation and Design. Taylor and Francis, UK., pp: 451.

Hilbert, H.S. and R.E. Falco, 1991. Measurements of flows during scavenging in a Two-stroke engine. SAE Trans., 100: 999-1013.

Ikeda, Y., M. Hikosaka and T. Nakajima, 1991a. Scavenging flow measurements in a motored two-stroke engine by fiber LDV. Soc. Automot. Eng. Trans., 100: 981-989.

Ikeda, Y., M. Hikosaka, T. Nakajima and T. Ohhira, 1991b. Scavenging flow measurements in a fired two-stroke engine by fiber LDV. Soc. Automot. Eng. Trans., 100: 990-998.

Jante, A., 1968. Scavenging and other problems of two-stroke cycle sparkignition engines. Soc. Automot. Eng. Trans., 77: 1804-1824.

Lai, Y.G., A.J. Przekwas and R.L.T. Sun, 1993. Three-dimensional computation of the scavenging flow process in a motored two-stroke engine. SAE Trans., 102: 57-73.

Pope, S.B., 2005. Turbulent Flows. Cambridge University Press, Cambridge.

Raghunathan, B.D. and R.G. Kenny, 1997. CFD simulation and validation of flow within a motored two-stroke engine. Soc. Automot. Engineers Paper 970359

Reddy, K.V., V. Ganesan and K.V. Gopalakrishnan, 1986. Under the roof of the cylinder head experimental study of the air movement in a two-stroke engine. Soc. Automot. Eng. Trans., 95: 1894-1919.

Sher, E., 1985. A new practical model for the scavenging process in a 2-stroke cycle engine. SAE Trans., 94: 1485-1495.

Verhelst, S. and C.G.W. Sheppard, 2009. Multi-zone modeling of spark-ignition engine combustion: An overview. Energy Conser. Manage., 50: 1326-1335.

Yang, S.L., Y.K. Siow, C.Y. Teo and K. Hanjalic, 2005. A KIVA code with Reynolds-stress model for engine flow simulation. Energy, 30: 427-445.