Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 45 (2014) 701 – 710
Energy Procedia

68th Conference of the Italian Thermal Machines Engineering Association, ATI2013

3D CFD analysis of the influence of some geometrical engine parameters on small PFI engine performances – the effects on the tumble motion and the mean turbulent intensity distribution

Stefania Falfari[a],*, Federico Brusiani[a], Piero Pelloni[a]

[a]*Department DIN, University of Bologna, Bologna, 40136, Italy*

Abstract

In scooter/motorbike engines coherent and stable tumble motion generation is still considered an effective mean in order to both reduce engine emissions and promote higher levels of combustion efficiency. The promotion of a stable and coherent tumble structure is largely believed in literature to enhance in-cylinder turbulence accelerating combustion process. In small PFI engine layout and weight constraints limit the adoption of more advanced concepts. In previous technical papers the authors demonstrated the influence of head shape and squish area on tumble vortex formation, development, breakdown and on final value of turbulence close to spark plug for small PFI engines. The main result of the this research was that the combustion chamber having the less squish area resulted to have the highest level of turbulence close to spark plug at ignition time. The geometry under analysis in the current paper is a 3-valves pent-roof motorcycle engine. 3D CFD simulations were ran at 6500 rpm with AVL FIRE code. The chosen engine geometry was the geometry found to be the best set-up in terms of turbulence and combustion performances in the previous paper. In the present paper the head shape and the squish area were kept constant and the following engine parameters were varied: the intake duct angle (the angle of the intake duct entering the head was reduced of 6%, i.e. it was more directed toward the exhaust side of the chamber), the piston shape, and finally the compression ratio (it was reduced of 9%). The main goal of the current analysis is to understand which of these parameters is predominant in accelerating combustion for directing engine design toward the best set-up.

Selection and peer-review under responsibility of ATI NAZIONALE

Keywords: Small PFI Engine; Tumble Motion; Engine Geometrical Parameters; Engine Design;

---

* Corresponding author. Tel.: +39-051-209.3316; fax: +39-051-209.3313.
E-mail address: stefania.falfari@unibo.it

Selection and peer-review under responsibility of ATI NAZIONALE
doi:10.1016/j.egypro.2014.01.075

1. Introduction

Nowadays the European Commission and the Association des Constructeurs Europeens de Motocycles (ACEM) have issued the decision to the motorcycle industry that would slowly migrate cycles through the Euro 4 requirements in 2014 and up to Euro 5 in 2020. The continuing demand of engines having low environmental impact and high efficiency requires the necessity of fast combustion phase and reduced cyclic variability. In modern PFI engines, which are still the most used, it is mandatory to obtain in the suction phase a coherent and structured in-cylinder charge motion in order to increase the efficiency. A tumble structured vortex having the dimensions comparable with the piston stroke promotes the formation of an high turbulence level close to the spark plug at the end of the compression stroke, which speed the combustion up and it allows the adoption of lean burn strategies, that otherwise are characterized by high cyclic variability because of the combustion instabilities.

Motorcycle industries have to promote the research in order to head the engine design toward the technological solutions capable to further reduce the raw emissions, the fuel consumption and to improve the performances. In fact the necessity of the carbon dioxide reduction implies the design of engines with higher compression ratio (RC), which needs to be associated with a faster combustion rate to keep the detonation risk away.

2. Literature summary on the tumble motion

Literature on tumble and squish motions analysis is quite wide, especially for tumble. In particular the research activity carried out by the authors on the tumble and the squish motions analysis is quite wide.

In [1] the authors carried out an analysis of a small 3-valves PFI engine characterized by a small bore and a high stroke-to-bore ratio. They compared three geometries differing only in the squish area distribution and values. They carried out a deep insight of the in-cylinder flow motion for highlighting the possible mutual influence between the squish and the tumble motions. Moreover it was highlighted that the squish motion interacts strongly with the tumble motion.

In [2] the authors presented a theoretical model capable of describing the interaction between the squish velocity and the tumble velocity depending on the engine class.

Authors in [3] focused on the effect of the joint use of the tumble and the squish flows in promoting the in-cylinder turbulence for fast combustion purposes. The squish velocity to the tumble velocity ratio at TDC was used to evaluate the effectiveness of the joint use of the tumble and the squish flow.

In [4] authors used the numerical simulation to assess the influence of some intake duct geometrical parameters on the tumble motion generation during both the intake and the compression strokes to highlight the turbulence production process. For a fixed engine class and varying the engine speed the analysis was carried out. In particular the authors tried to understand the importance of the geometrical duct parameters by means the use of the tumble torque parameter. This analysis was deepened in [5]: the tumble torque parameter was used for comparing in details the different results obtained in changing the intake duct parameters highlighted in [4]. Moreover a probability distribution function (pdf) of the turbulence intensity close to the spark plug was introduced.

Bianchi et al. [6] analyzed the vortex formation for small C/D ratio engines. Blaxill et al. [7] presented a parametric analysis carried out detecting six fundamental geometric parameters capable to describe the intake duct with a fixed inlet valve lift, while Gaikwad et al. [8] carried out an optimization process on an intake duct. In [9] results obtained performing experimental tests on three different intake ducts were reported. In Ramajo et al. [10] results obtained running a mono-dimensional (1D) model developed for predicting in-cylinder tumble motion formation and breakdown till the appearance of high turbulence level close to TDC were described.

Laget et al. [11] presented an innovative methodology based on the relationship between the charge motion and the combustion process. The analysis was performed using a DOE technique.

Ahmad Amer et al. [12] found that the large scale tumble structures are essential in enhancing the turbulence inside the cylinder but only if these structures collapse just before TDC, which allows them to control the initial kernel development, as also reported in [13, 14]. Moreover the upper part of the piston is detrimental for tumble breakdown, so they stated the importance of the piston presence in all the tumble motion analysis, as also underlined in [15].

Arcoumanis et al. [16] showed that a consistent and structured tumble vortex doesn't involve directly a proper turbulence level because it could collapsed before reaching TDC.

Mittal et al. [17] studied the combined effect of the cyclic variability and the control system on the incoming fluid, while Huang et al. [18] presented some experimental results in terms of the tumble ratio and the turbulence level on a motorcycle engine equipped by the intake ducts having or circular shape or elliptical shape.

In [19] the authors analyzed the effect of five piston shapes on the tumble motion and they experimentally measured the tumble ratio and the mean value of turbulent kinetic energy during the intake and the compression strokes.

Lee et al. [20] experimentally studied the effect of the tumble and the swirl flows on the turbulence near TDC for correlating the flow field generated in the intake stroke with the turbulence intensity near the spark timing. In Achuth et al. [21] it was highlighted as the tumble breakdown and the consequent turbulence are affected by the chamber shape: for the same piston, they depend on the head shape and in particular on the pentroof angle.

Starting from above reported literature results and generally speaking it is possible to state that in the open literature the parameters which are believed to be the most significant in affecting the tumble vortex formation, stabilization, breakdown and of consequence the final level of turbulence at the end of the compression stroke are:

- The intake duct shape and most of all the duct angle entering the head;
- The compression ratio;
- The piston shape;
- The squish area.

Thus the industrial problem of determining the tumble vortex formation, stabilization, breakdown involves a multivariable issue.

3. Present paper target and originality

As in the papers [1, 2], the present paper deals with PFI motorcycle engines characterized by having high stroke-to-bore ratio (C/D) and small bore (D), as summarized in Table 1. In [1] simulations on three different engine industrial configurations (Table 2) were ran by means a 3D CFD code: the simulation results showed that the configuration having the highest percentage squish area and the intake duct most capable of generating high tumble ratio at IVC (namely C3_original) was characterized by the lowest combustion velocity. This was the evidence of a not coherent tumble structure related to an increase of the squish area which in turn results in a weaker tumble breakdown during the compression stroke.

Table 1. Engine data.

Bore [mm]	52.0
Stroke [mm]	58.6
Conrod length [mm]	98.0
C/D	1.12
Compression ratio (RC)	11.0
Valve number	3.0
% squish area	10.8

Table 2. Geometric parameters of the geometries under analysis – step 1

Engine configuration	% squish area	Head type	Piston Type	Intake duct type
C1	10.18	A	P	T
C2	12.20	B	P	T
C3_original	21.00	C	PP	TT

In order to explain the last result a scientific approach was adopted, turning the industrial multivariable problem into a single-variable problem, focused only on the different percentage squish area. CFD results performed on three engines having only different percentage squish area (Table 3) showed that increasing it there is a worsening of the in-cylinder fluidynamic.

The geometry named C1, that is the geometry having the less percentage squish area, was established to be the best configuration.

Starting from this C1 geometry and once assessed the effect of the squish area on these type of engines, in the present paper the further step toward a fast engine design was to investigate the effect on combustion speed of each single parameter of the multivariable problem not investigated in paper [1]: the piston shape, the compression ratio and the intake duct.

The main focus of this analysis was to direct the engine design toward the solutions characterized by the fastest combustion velocity.

In order to do it the flow field of geometry C1 was frozen and it was used as a comparison. The above parameters characterizing the multivariable problem were varied one-by-one in order to fix a sort of ‘value scale’ for obtaining an enhancement of the turbulent kinetic energy level.

3D CFD simulations were ran changing with respect to the configuration C1 the following parameters singularly:

- The piston shape: it was believed to be interesting to evaluate the effective importance of this parameter on the tumble motion and if there is a particular range of the stroke where it more affects the vortex formation/breakdown. On geometry C1 the piston equipping the configuration named C3_original in paper (1) was mounted (Table 2).
- The compression ratio RC: it was decreased of 9% equipping the geometry C1 with a head having a different shape from the head A of Table 2, but the same squish area;
- The tumble ratio: on geometry C1 the intake duct TT of Table 2 is a duct more oriented toward the exhaust side of the cylinder than the intake duct T, i.e. it is a duct capable of generating a stronger tumble ratio at IVC . In particular the angle of the intake duct entering the head was reduced of 6%, i.e. it was more directed toward the exhaust side of the chamber.

The parameters that were identified by the authors as the most significant in evaluating the different behavior of the three geometries are: the tumble ratio and the mean turbulent intensity (evaluated as the ratio between the turbulent fluctuating energy and the mean piston velocity).

The originality of the work consists in establishing the effect of each single parameter on the tumble formation/breakdown estimating the weight of each one compared to the original geometry. The focus is to write a ‘value scale’ for fast addressing the engine design.

4. Engine characteristics

In Table 1 were summarized the engine data. The engine is the 3-valves PFI pentroof engine in the configuration C1, which was found to generate the more structured and coherent in-cylinder charge motion in the previous paper [1]. The geometries under analysis were the following, as summarized in Table 3:

- Geometry C1_PP: it is equipped by a different piston but the same compression ratio RC of the geometry C1. The goal was to fix the importance of the piston shape.
- Geometry C1_TT with an intake duct more oriented toward the exhaust side of the cylinder than the original intake duct of the geometry C1, i.e. with a duct capable of generating a stronger tumble ratio at IVC. The aim was to investigate its global effect with respects to the other parameters. It is well known that it should be the most important parameter in affecting tumble ratio and the focus was to establish how much is its weight if compared to the other parameters, especially in generating turbulence at TDC.
- Geometry C1_RC with a reduced compression ratio RC: the main focus was to establish how the RC could affect the in-cylinder flow structure. It was reduced of about the 9% by means a slight modification of the head shape, without changing the squish percentage area.

Table 3. Summary of the analyzed geometries

Engine configuration	RC	% squish area	Head type	Piston Type	Intake duct type
C1	RC	10.18	A	P	T
C1_PP	RC	10.18	A	PP	T
C1_TT	RC	10.18	A	P	TT
C1_RC	1_RC	10.18	A_1	P	T

5. Methodology of the analysis

For performing the analysis of the proposed geometries it was decided to adopt the following methodology:

- 3D CFD simulations;
- 0D analysis of the results obtained at the previous step.

In summary the methodological approach consists in a combined use of 3D CFD simulations and 0D analysis: the 0D model could allow one to investigate the inner process at the basis of the tumble vortex formation using the results coming from the CFD simulations. In fact it is reasonable to consider that a simply 0D model could not reproduce the tumble motion as well as a 3D code. Besides the 3D code in not suitable for separating the single inputs that contribute to the tumble structure.

5.1. 3D CFD simulation set-up

In order to assess the different behavior of the proposed engine geometries the simulation of the intake and the compression phases with the 3D code FIRE v. 2010.1 by AVL was performed. Boundary conditions for the intake process was set on the basis of the data from a proper 1D engine model [1]. Simulations were ran at full load and 6500 rpm with a computational domain composed by exhaust system, intake system and cylinder. The code solves momentum, continuity, turbulence and energy equations. The turbulent model was set to k-ε. The differencing schemes adopted were MINMOD relaxed for momentum equation, central differencing for continuity equation and upwind for turbulence and energy equations. At inlet and outlet boundaries mass flows boundary conditions computed by means a proper 1D model of the engine were imposed. The computational grid was an unstructured grid with hexahedral cells. The results will refer to the fluid-dynamic cycle-converged engine simulation. The methodological approach based on CFD simulations was tested in the last years by the research group the authors belong to, as reported in [22-30].

5.2. 0D model

The 0D model was developed with the Matlab code. The main model inputs are the tumble ratio RT and the incoming flow rate computed by 3D CFD analysis. The underlying hypothesis for the model development was to consider that the mass incoming into the cylinder rotates as an equivalent solid body around an axis perpendicular to the cylinder axis and centered in the center line between the head and the instantaneous piston position. The model part concerning the momentum rate source term evaluation was derived by the work of Ramajo et al. [10]. Zero-dimensional tumble formulation assumes that the inlet flow has the angular momentum with respect to an axis normal to the cylinder axis, which contributes to a tumble vortex placed in the middle between the piston and the head of the cylinder.

A formulation capable to process the CFD results and to extract the single contribution to the tumble vortex formation was derived by the authors. The tumble ratio trend can be loaded by CFD computations for computing the vortex angular velocity:

$$\omega = RT_{CFD} \cdot \omega_{ENGINE} \tag{2}$$

Where ω is the vortex angular velocity, RT_{CFD} is the tumble ratio from CFD computations and ω_{ENGINE} is the engine angular speed. The equivalent vortex inertia momentum I, computed with respect of an axis perpendicular to the cylinder axis and centered between the head and the piston, can be used in combination with the vortex angular velocity for computing the net momentum M:

$$M = \omega \cdot \frac{\Delta I}{\Delta t} + I \cdot \frac{\Delta \omega}{\Delta t} \tag{3}$$

Then it was possible to estimate the total contribution of the shear stress and the wall friction:

$$\frac{\mathrm{dJ_{IN}}}{\mathrm{dt}} - \mathrm{M} = T_W + T_S \tag{4}$$

where T_S and T_W are the momentum losses due to the internal shear stresses and the wall friction respectively, while the term J_{IN} represents the torque source given by the incoming flow through the inlet valves. In particular the term dJ_{IN}/dt represents the momentum applied by the incoming fluid and it can be evaluated as follows:

$$\frac{\mathrm{dJ_{IN}}}{\mathrm{dt}} = \dot{m}_{IN} \cdot U_0 \cdot h \tag{5}$$

The term dJ_{IN}/dt was computed considering that the inlet flow enters with a velocity U_0 transporting a positive angular momentum with respect to a theoretical tumble vortex center. The $\dot{m}_{IN}$ term is the mass flow rate through the inlet valves, while the term h states for the main vortex dimension. Starting from it the mean inlet velocity U_{IN} expression and then the mean inlet tangent velocity U_0 were computed:

$$U_0 = U_{IN} \cdot \cos\theta \cdot c_{IN} \tag{6}$$

Where the term c_{IN} represents a coefficient for taking into account the vena slowdown when the fluid goes across the inlet valves toward the cylinder. The angle θ is the pentroof angle.

By means this model formulation it was possible to estimate the single contribution of each parameter to the tumble motion formation.

6. Results analysis

The results from the 3D CFD simulations were reported in Table 4 for the geometries in analysis. In particular the data in terms of the tumble ratio RT at IVC, the mean in-cylinder turbulent intensity and close to the spark plug were reported. It is to note that the strongest tumble ratio was the one of the geometry C1_TT, followed by the other geometries. In Table 5 the percentage variation of the mean turbulent intensity close to spark plug in respect to geometry C1 was reported: the geometry C1_TT shows a percentage variation of mean turbulent intensity close to spark plug of about 34%. The variation between the geometry C1_PP and the geometry C1 was 4.2%, while the mean turbulent intensity of the configuration C1_RC was lower than the mean turbulent intensity of the configuration C1 of about 5.5%.

In Fig. 1 and 2 respectively the tumble ratio trend during the intake (Fig. 1a) and the compression phases (Fig. 1b), the mean in-cylinder (Fig. 2a) and close to the spark plug (Fig. 2b) turbulent intensity were reported.

During the intake stroke (Fig. 1a) the tumble ratio is stronger for configuration C1_TT, while it is just the same for the other geometries. In the compression phase (Fig. 1b) the tumble ratio is always higher for the geometry C1_TT. It is to note that the breakdown phase for the geometry C1_RC is less effective because of its reduced compression ratio RC: this is visible looking at its higher value in Fig. 1b after 650 CA. The tumble ratio for the geometries C1_PP and C1_RC decreases only after 630 CA, sign that the piston shape has effect only in the breakdown phase, independently by the compression ratio value.

The mean turbulent intensity spread in the whole cylinder (Fig. 2a) is higher for the configuration C1_TT, followed by the geometry C1. The other geometries have a mean turbulent intensity value close to each other and very close to the value of geometry C1. The difference between the geometries is more visible in Fig. 2b, where the turbulent intensity close to spark plug was depicted: it is visible the trend highlighted by the summary of Table 5. The geometry C1_RC has the lowest value because its tumble breakdown was less effective than for the other configurations due to its reduced RC.

In summary it is to highlight that in respect to geometry C1 changing the intake duct the difference was about 34%, changing the piston the difference was about 4%, while reducing the squish velocity the difference was about 10%. This led to the consideration that the ‘value scale’ in terms of turbulent kinetic energy close to spark plug at the ignition timing was structured as follows: the intake duct, the squish area and last of all the piston shape.

Table 4. 3D CFD results summary for the geometries

Engine configuration	TR@ IVC	Mean u’/Vp @ 690 CA	u’/Vp @ 690 CA close to spark
C1	0.90	0.52	0.49
C1_PP	0.90	0.51	0.51
C1_TT	1.10	0.58	0.70
C1_RC	0.89	0.50	0.47

Table 5. Percentage variation of turbulence intensity at 720 CA close to spark plug in respect to geometry C1

Engine configuration	Percentage variation of u’/Vp @ 720 CA close to spark in respect to geometry C1
C1	-
C1_PP	4.2%
C1_TT	34.0%
C1_RC	-5.5%

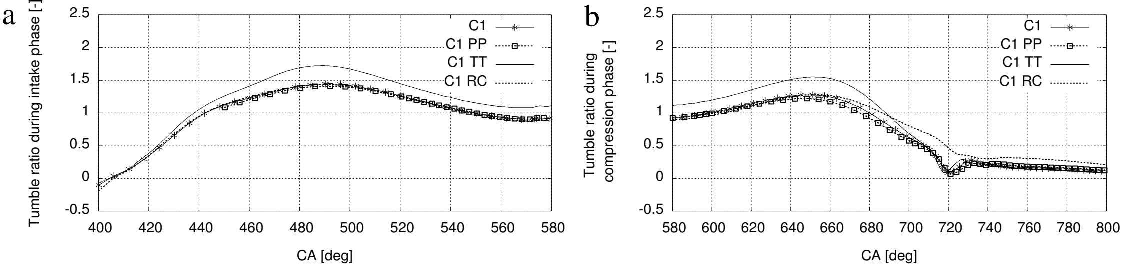

Fig. 1: In-cylinder tumble ratio trend during (a) intake phase; (b) compression phase.

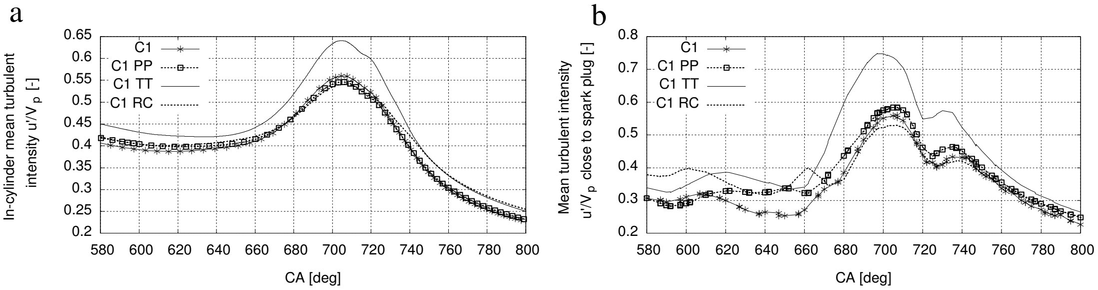

Fig. 2: Mean turbulent intensity trend (a) in the whole cylinder domain; (b) close to spark plug.

6.1. 0D model

After the analysis of the 3D CFD results the 0D model was ran in order to study in detail the tumble vortex formation/breakdown for the different geometries.

In Fig. 3a and 3b the net momentum during the intake phase and the compression phase was reported. During the intake phase (Fig. 3a) the net momentum applied to the in-cylinder fluid is higher for the geometry C1_TT in the range 440-500 CA, that is during the vortex generation phase (IVO-BDC). This is due to the intake duct angle in respect to the head entering: the duct TT more oriented toward the exhaust side of the chamber is capable of generating an incoming flux having a momentum stronger that the intake duct T (Table 3). During the compression stroke (Fig. 3b) and in particular in the breakdown phase (after 650 CA) the net momentum related to the residual tumble vortex is less for the geometry C1_TT than for the geometry C1, while the geometry C1_RC has the highest value. It is to highlight that the minimum is the net momentum value, the maximum is the turbulent kinetic energy (Fig. 2 and 3) and the most effective was the tumble breakdown (Fig. 1b). In Fig. 4 the zoom of the net momentum during the compression phase was reported: it is to note the piston shape effect. In fact the configurations C1, C1_PP and C1_RC have a trend close to each other but not coincident. The highest net momentum is for the geometry C1_RC, while the geometry C1 has a net momentum medium between the configurations C1_PP and C1_RC. At the end of the compression stroke the less effective breakdown phase was for the geometry C1_RC, followed by the configurations C1_PP and then C1. This trend was a sign of the relevance of the piston shape during the breakdown phase. The main conclusions were the followings:

- Changing the piston shape, for the same RC, the breakdown phase could lower or higher its efficiency;
- Lowering the RC value, the breakdown efficiency is decreased.

In summary the net momentum trend is a picture of the efficiency of respectively the generation and the breakdown phases of the tumble motion. In Fig. 5 the resistant torque due to both the wall friction and the shear stress was depicted. Between 440 and 500 CA the resistant torque is less for the geometry C1_TT than for the other configurations: this is due to the better flux path related to the intake duct TT.

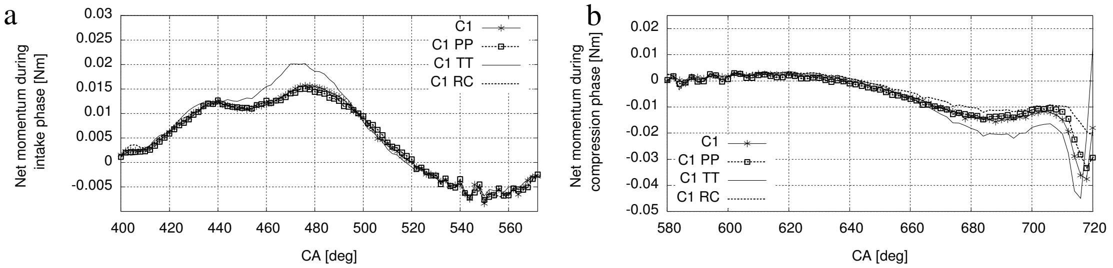

Fig. 3: Net momentum during (a) intake phase; (b) compression phase.

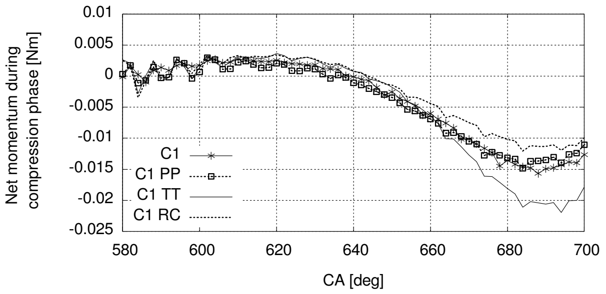

Fig. 4: Zoom of net momentum during compression phase.

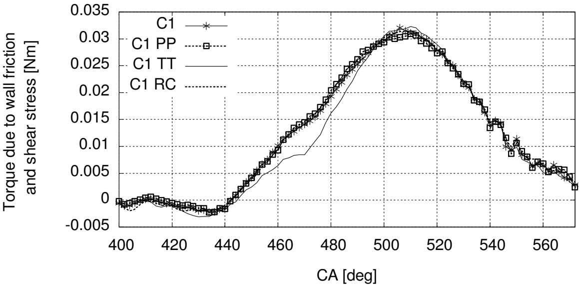

Fig.5: Torque due to the wall friction and the shear stress.

7. Conclusions

The present paper deals with PFI motorcycle engines characterized by having a high stroke to bore ratio (C/D) and a small bore (D). The focus of the paper was to demonstrate how to speed the combustion velocity up turning a multivariable problem into a single-variable problem. In particular the parameters most significant in affecting the tumble vortex formation, stabilization, breakdown and of consequence the final level of turbulence at the end of the compression stroke are: the intake duct shape, the compression ratio, the piston shape and the squish area.

In the previous paper [1] the focus was to analyze the influence of the squish area parameter and the geometry C1 was found to have the best configuration. Starting from this geometry and once assessed the effect of the squish area on these type of engines, in the present paper the further step toward a fast engine design was to investigate the effect on the combustion speed of each single parameter of the multivariable problem not investigated in the paper [1] and above listed.

The flow field of the geometry C1 was frozen and it was used as a comparison. The above parameters characterizing the multivariable problem were varied one-by-one in order to fix a sort of ‘value scale’ for obtaining an enhancement of the turbulent kinetic energy level.

The geometry C1_PP, C1_TT and C1_RC were analyzed: they were characterized by the same pent-roof angle, head shape and squish area. They differ in respect to the geometry C1 for respectively the piston shape, the intake duct shape and the compression ratio value.

The resulting ‘value scale’ was as follows:

- The most affecting parameter for the velocity combustion was the intake duct;
- The second parameter was the squish area. For engines having small bore the increase of the percentage squish area implied an intrusive geometric modification of the engine head, which worsened the in-cylinder fluidynamic and didn’t increase the combustion velocity. It was shown [1] that it is necessary to reduce the squish area, which is both detrimental for the tumble vortex formation/breakdown and doesn’t increase the combustion velocity;
- The third parameter was the piston shape: it plays a role about after 650 CA, independently by the compression ratio value;
- The compression ratio reduction influenced only the degree of deformation of the tumble vortex, leading to a lower level of the mean in-cylinder turbulence.

References

[1] Falfari S, Bianchi GM, Nuti L. Numerical Comparative Analysis of In-Cylinder Tumble Flow Structures in Small PFI Engines Equipped by Heads Having Different Shapes and Squish Areas. ICES2012-81095: Proceedings of ASME Internal Combustion Engine Division, 2012. Spring Technical Conference.

[2] Falfari S, Brusiani F, Bianchi GM. Numerical Analysis of In-Cylinder Tumble Flow Structures – Parametric 0D Model Development. ATI 2013 – 060-12805. ATI Congress 2013. Bologna, Italy. Accepted for publication on ATI 2013 Energy Procedia special issue.

[3] Falfari S, Bianchi GM, Nuti L. 3D CFD Analysis of the Influence of Some Geometrical Engine Parameters on Small PFI Engine Performances – The Effects on Tumble Motion and Mean Turbulent Intensity Distribution. SAE 2012-32-0096. SETC Congress, Madison, USA.

[4] Falfari S, Brusiani F, Bianchi GM. Assessment of the Influence of Intake Duct Geometrical Parameters on the Tumble Motion Generation in a Small Gasoline Engine. SAE 2012-32-0095. SETC Congress, Madison, USA.

[5] Falfari, Brusiani F, Cazzoli G. Tumble Motion Generation in Small Gasoline Engines: A New Methodological Approach for the Analysis of the Influence of the Intake Duct Geometrical Parameters", ATI 2013 – 060-12806. ATI Congess 2013, Bologna, Italy. Accepted for publication on ATI 2013 Energy Procedia special issue.

[6] Bianchi GM, Cantore G, Nicolini D, Sassi L, Fraboni L. Numerical Analysis of the Influence of the Stroke-to-Bore Ratio on Tumble Breakdown. Proceedings of the 4th International Conference ICE 1999. Capri.

[7] Blaxill H, Downing J, Seabrook J, Fry M. A Parametric Approach to Spark-Ignition Engine Inlet-Port Design. SAE paper 1999-01-0555. SAE International Congress, Detroit, USA, 1999.

[8] Gaikwad S, Arora K, Korivi V, Cho S.K. Steady and Transient CFD Approach for Port Optimization. SAE paper 2008-01-1430.

[9] Lee K, Bae C, Kang K. The effects of tumble and swirl flows on flame propagation in a four valve S.I. engine. Applied Thermal Engineering 27, 2007; 2122-2130.

[10] Ramajo D, Zanotti A, Nigro N. Assessment of a zero-dimensional model of tumble in four-valve high performance engine. International Journal of Numerical Methods for Heat & Fluid Flow, Vol. 17 No. 8, 2007, pp. 770-787.

[11] Laget O, Zaccardi J.M, Gautrot X, Mansion T, Cotte E. Establishing New Correlations Between In-Cylinder Charge Motion and Combustion Process in Gasoline Engine Through a Numerical DOE. SAE paper 2010-01-0349.

[12] Ahmad Amer A, Reddy T.N. Multidimensional Optimization of In-Cylinder Tumble Motion for the New Chrysler Hemi. SAE paper 2002-01-1732.

[13] Church W, Farrell PV. Effects of Intake Port Geometry on Large Scale In-Cylinder Flows. SAE 980484.

[14] Miller R, Newman C, Dai W, Trigui N, Davis G, Trumpy D, Glidewell J. Up-Front Predictions of the Effects of Cylinder Head Design on Combustion Rates in SI Engines. SAE 981049.

[15] Xavier B, Floch A. Investigation of the In-Cylinder Tumble Motion in a Multi-Valve Engine: Effect of Piston Shape. SAE 971643.

[16] Arcoumanis, C, Godwin SN, Kim, JW. Effect of Tumble Strength on Combustion and Exhaust Emissions in a Single Cylinder, Four-Valve, Spark Ignition Engine. SAE 981044.

[17] Mittal M, Schock H. A Study of Cycle-to-Cycle Variations and the Influence of Charge Motion Control on In-Cylinder Flow in an IC Engine. ASME 2010, Journal of Fluids Engineering, Vol. 132, 051107-1.

[18] Huang RF, Lin KH, Yeh CN, Lan J. In-cylinder tumble flows and performance of a motorcycle engine with circular and elliptic intake ports. Journal of Experiments in Fluids, Vol. 46, No. 1, January 2009, pp. 165-179.

[19] Murali Krishna, Mallikarjuna B. Comparative study of in-cylinder tumble flows in an internal combustion engine using different piston shapes – an insight using particle image velocimetry. Journal of Experiments in Fluids, Vol. 48, No. 5, May 2010, pp. 863-874.

[20] Lee KH, Lee CS. Effects of tumble and swirl flows on turbulence scale near top dead centre in a four-valve spark ignition engine. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering 2003. 217:607.

[21] Achuth M, Metha PS. Predictions of tumble and turbulence in four-valve pentroof spark ignition engines. Int. Journal of Engine Research 2001. 2: 209.

[22] Bianchi GM, Cantore G, Fontanesi S. Turbulence Modeling in CFD Simulation of ICE Intake Flows: The Discharge Coefficient Prediction. SAE TRANSACTIONS-SAE Journal of Engines, Vol. 111, Sect. 3, pp. 1901-1918, ISBN 0-7680-1287-2. Warrendale (PA, USA), 2003.

[23] Bianchi GM, Brusiani F, Grimaldi CN, Postrioti L, Carmignani L, Di Palma S, Marcacci M, Matteucci L. CFD Analysis of Injection Timing Influence on Mixture Preparation in a PFI Motorcycle Engine. SAE TRANSACTIONS-SAE Journal of Engines, Vol. 115, Sect. 3, pp.984-998, ISBN 978-0-7680-1835-6. Warrendale (PA, USA), 2007.

[24] Cantore G, Fontanesi S, Mattarelli E, Bianchi GM. A methodology for the in-cylinder flow field evaluation in an ultra-low stroke-to-bore SI engine. SAE Paper 2002-01-1119. In: SAE International Congress, Detroit, 2002.

[25] Falfari S, Bianchi GM. Development of an Ignition Model for S.I. Engines Simulation. SAE Paper 2007-01-0148. In: SAE International Congress, Detroit, 2007.

[26] Bianchi GM, Brusiani F, Grimaldi CN, Postrioti L, Carmignani L, Marcacci M, Matteucci L. CFD Analysis of Injection Timing and Injector Geometry Influences on Mixture Preparation at Idle in a PFI Motorcycle Engine. SAE PAPER 2007-24-0041. In: SAE ICE2007 International Conference, 2007.

[27] Forte C, Bianchi GM, Corti E, Fantoni S. Combined experimental and numerical analysis of the influence of air-to-fuel ratio on cyclic variation of high performance engines. ICES2008-1668. In: ASME ICE DIVISION Spring Conference, 2008.

[28] Forte C, Bianchi GM, Corti E, Fantoni S. Analysis of cycle by cycle variation of an high performance engine: influence of mixture composition. 8th International Symposium on Combustion Diagnostics (AVL), Baden Baden, 2008.

[29] Bianchi GM, Forte C, Corti C. Combined experimental and numerical analysis of knock in spark Ignition engines. ICES2009-14102. In: ASME – ICEF 2009, Luzern (Switzerland), 2009.

[30] Bianchi GM, Forte C, Corti C. Validation of a Lagrangian Ignition Model in SI Engine Simulations. ICEF2010-35159,. In: ASME – ICEF 2010, San Antonio, Texas, 2010.