Strojniški vestnik - Journal of Mechanical Engineering 51(2005)12, 744-756
UDK - UDC 621.43.01:004.94
Izvirni znanstveni članek - Original scientific paper (1.01)

Analiza tokovnih struktur v vstopnem sistemu motorjev - primerjava rezultatov meritev in numerične simulacije

Analysis of the In-Cylinder Flow Structures Generated by the Engine Induction System – a Comparison of Experimental and Simulated Data

Samuel Rodman Oprešnik - Sašo Prijatelj - Tomaž Katrašnik - Ferdinand Trenc - Frančišek Bizjan

V prispevku so prikazani rezultati numerične analize - simulacij na osnovi računalniške dinamike tekočin (RDT) in meritev hitrostnega polja v vstopnem sistemu motorja z notranjim zgorevanjem z uporabo laserskega Dopplerjevega anemometra (LDA). V ta namen je bil izdelan stekleni osmerokotni model valja, ki je po geometrijski obliki podoben valju resničnega motorja. Premičen bat je tokovne razmere na modelu še bolj približal resničnim razmeram pri pravem motorju. Ugotovljeno je dobro ujemanje izračunanih in izmerjenih rezultatov pri določitvi tokovnih struktur, hitrostnega polja v valju in pri določitvi pretočnega koeficienta v vstopnem ventilu motorja. Raziskava je tudi pokazala, da je mogoče z numerično analizo RDT-jem razmeroma dobro napovedati dogajanje v valju motorja in izboljšati razumevanje pojavov, ki spremljajo polnjenje valja motorja.
© 2005 Strojniški vestnik. Vse pravice pridržane.
(Ključne besede: motorji ZNZ, računalniška dinamika tekočin, CFD, polja hitrostna, anemometri Dopplerjevi laserski)

A comparison and analysis of laser Doppler anemometry (LDA) measurements and a computed fluid dynamics (CFD) simulation of the in-cylinder flow structures generated by an engine induction system is presented in this paper. An octagonal glass model with a moving piston complementary to a real engine cylinder was applied in the analysis. Good agreement between the measured and simulated patterns of the flow structures and the inlet-valve discharge coefficients was obtained. In general, this study shows that in-cylinder CFD predictions yield reasonably accurate results that help to improve the knowledge of the air-flow characteristics during the intake stroke. CFD therefore represents an effective design tool for developing less-polluting and more efficient internal combustion engines.
© 2005 Journal of Mechanical Engineering. All rights reserved.
(Keywords: internal combustion engine, computational fluid dynamics, CFD, air flow characteristics, laser Doppler anemometry)

0 UVOD

Kakovost zgorevanja v valju ottovega in dizelskega motorja z notranjim zgorevanjem močno vpliva na nastanek škodljivih emisij v ostankih zgorevanja, zato je dobro poznavanje tokovnih razmer v valju motorja z notranjim zgorevanjem neobhodno. Strukture turbulentnega toka v valju motorja so pomembne tako za začetek kakor tudi za celotno obdobje trajanja zgorevanja in njegovo učinkovitost [1]. Velikostni red turbulentnih struktur je različen: od največjega, ki ga omejuje geometrijska oblika zgorevalnega prostora, pa vse do najmanjših, pri katerih prevladuje molekulska difuzija. Vrtinčni tok zgorevalne zmesi, ki ima poudarjen vrtinec v obodni in/ali vzdolžni

0 INTRODUCTION

In order to reduce harmful exhaust emissions and increase the efficiency of internal combustion engines, fundamental knowledge of in-cylinder fluid motion is required, since it significantly affects the combustion process in both SI and CI engines. It is well known that the structure of the turbulent flow field of the fresh charge is a determining factor in the initiation, the rate of propagation, and the efficiency of the combustion process in an internal combustion engine [1]. The turbulent length scales range from the largest, which are limited by the physical constraints of the combustion chamber, to the smallest, which are governed by molecular diffusion. A smeri valja je stabilnejši, razpada počasneje in zato koristno pomaga pri učinkovitem mešanju goriva in zraka ter zgorevanja tudi v kasnejših obdobjih zgorevanja ([2] in [3]). Zato je za razvoj sodobnih motorjev, še posebno glede zmanjšanja škodljivih emisij v izpušnih plinih, izredno pomembno dobro razumevanje časovnega in krajevnega razvoja tokovnih struktur med vtekanjem in stiskanjem delovne snovi v valju [4].

Pri določanju oblike toka v valju, ki je predvsem posledica oblike vstopnega kanala in pogojev delovanja motorja, so bili do sedaj največkrat uporabljeni rezultati preizkusov na mirujočem valju oziroma modelu valja (npr. v objavah [5] in [6]). Kasneje so bili osnovnim podatkom dodani še dodatni parametri, ki so bolje opisovali integralni tok v valju [7]. Z rezultati eksperimentalnega dela, ki je bilo podprto s preprostimi fenomenološkimi modeli, podobne sta razvila za potrebe svojih raziskav Davis in soavtorji [8] in Murakami in soavtorji [9], je bilo mogoče določiti osnovne parametre, ki določajo tokovno polje v valju motorja. Pomembno prednost omenjenega načina kombiniranega reševanja odlikuje enostavnost in majhna poraba raziskovalnega časa, medtem ko pomeni manjša podrobnost opisa tokovnih struktur večjo pomanjkljivost, ki nas lahko pripelje tudi do napačnih rezultatov in sklepov. Velik korak pri natančnem preizkusnem določanju hitrostnega polja v mirujočem valju pomeni uvedba laserske Dopplerjeve anemometrije (v nadaljevanju LDA) ([10] in [11]). Rezultati so bistveno natančnejši od rezulatov, ki so bili izmerjeni s starejšimi ustaljenimi preizkusnimi metodami. Vpogled v tokovno dogajanje v valju omogočijo tudi sodobne 3D numerične simulacije (v nadaljevanju modeli RDT), s katerimi lahko rešimo ustrezne tokovne enačbe in izračunamo podrobne vrednosti povprečnih hitrosti toka in značilnosti turbulentnega hitrostnega polja. Pri uporabi modelov RDT moramo upoštevati posebnosti nestalnega toka, vpliv visokega Reynoldsovega števila toka in izjemno zamotane oblike trdnin - sten pretočnih kanalov, ventilov in valja.

V zadnjem času lahko v strokovni literaturi za potrebe določitve tokovnega polja v valju (tudi delujočega motorja) zasledimo (npr. [12] in [13]) tudi pogosto uporabo metode določanja hitrosti delcev toka z analizo optičnih posnetkov - DHD (PIV). Njena prednost je, v primerjavi z drugimi preizkusnimi metodami, v trenutnem in ne samo časovno povprečenem prikazu celotnega tokovnega stanja v valju. Ima pa metoda DHD tudi pomanjkljivosti: zahteva velike moči laserjev, sorazmerno velike dodatne delce za sipanje svetlobe, ki zaradi svoje

well-defined swirl and/or tumble flow structure is more stable than other large-scale in-cylinder flows and, therefore, it can break up later during the cycle, giving higher turbulence during combustion ([2] and [3]). Therefore, a good understanding of the evolution process of fluid motion in internal combustion engines is critical when the development of advanced engines with the most attractive operating and emission characteristics is concerned [4].

Simple experimental tests in steady-state flow rigs have been widely used to determine the flow motion generated by induction systems (see, for example, [5] and [6]). Some supplementary parameters to evaluate the bulk motion of in-cylinder flow were defined more recently [7]. In conjunction with simple phenomenological models, such as those developed by Davis et al [8] and Murakami et al [9], these experimental procedures helped to determine several fundamental variables that define the flow inside the cylinder. Simplicity and a moderate consumption of time are the advantages of this type of approach. However, the information provided is not detailed enough, and the assumptions made can lead to inaccurate results. A more accurate experimental technique is the measurement of the velocity field in a steady-flow test rig using laser Doppler velocimetry (LDV) ([10] and [11]). This method provides high-quality results and more information than other conventional methods. Another approach for getting an insight into the in-cylinder flow is the application of three-dimensional numerical simulation codes (CFD models), that are efficient in solving the governing flow equations, and thus provide a detailed description of the mean velocity and the turbulent velocity fields. When applied to internal combustion engines, the CFD models have to consider the specific problems linked to the unsteady flows, the high Reynolds numbers involved, and the highly complex geometry of the solid boundaries.

Many authors (see, for example, [12] and [13]) have applied the PIV (Particle Image Velocimetry) method to determine the flow patterns in cylinders, since it produces whole field data relating to a particular instant of the observed time rather than a phase-average. However, high laser-power levels and relatively large seeding particles are required to achieve satisfactory results; on the other hand, large particles introduce measurement errors due to inertial effects [14].

The aim of this study was to validate the accuracy of the CFD simulations of the in-cylinder vztrajnosti ne sledijo toku in tako vnašajo večjo merilno negotovost [14].

Namen predstavljenega dela je, da ovrednotimo natančnost uporabljene simulacije RDT in raziščemo njeno uporabnost ter omejitve pri določitvi tokovnega polja v valju motorja z uporabo preizkusne metode LDA. Zahtevnost in različnost oblike vstopnega sistema za različne motorje onemogoča preprost prenos ekstrapolacije dobljenih rezultatov opravljenih preizkusov na druge motorje [15], pač pa je zelo primeren in pogosto uporabljan pri optimizaciji oblike vstopnega, izpušnega sistema in zgorevalnega prostora opazovanega motorja. Zato je bil izdelani model steklenega valja motorja ustrezno obdelan in pripravljen za uporabo v simulacijski kodi CFX [16]. Opravljena in prikazana analiza ima dva cilja: 1) da preveri uporabnost in natančnost izračunanega pretočnega koeficienta in 2) da preveri uporabnost rezultatov računske simulacije RDT tokovnih struktur v valju motoja.

Pretočni koeficient vstopnega ventila motorja pomembno vpliva na količino v valj motorja vnesene sveže snovi in s tem tudi na moč in emisijo škodljivih snovi v izpušnih plinih. Obenem je tudi pomemben vstopni podatek pri računanju z nič in enorazsežnimi simulacijskimi modeli. Žal je njegova računska določitev praktično nemogoča zaradi izredne zapletenosti toka skozi ventil.

1 PREGLED UPORABLJENIH PREIZKUSNIH METOD

1.1 Opis preizkusne naprave

Merilna naprava, katere posnetek je prikazan na sliki 1 in s katero so bili opravljeni vsi predstavljeni preizkusi, sestoji iz pet pomembnejših sestavnih delov:
- sistema LDA za merjenje hitrosti v modelu valja,
- računalniško vodenega položajnega sistema za uporabljeno merilno lečo LDA,
- sistema merilnikov diferenčnega tlaka v modelu valja,
- enote sesalnika za prah z elektronsko nastavljivo frekvenco vrtenja - nastavljivim tokom zraka in nastavljivim poljubnim tlačnim padcem v vstopnem ventilu,
- kovinske - izvirne glave primerjalnega motorja, ki je bila pritrjena na stekleni model valja; omogočena je poljubna nastavitev pretočne špranje v ventilu.

Za potrebe raziskav je bil iz optičnega stekla izdelan osmerostrani model valja dizelskega tlačno polnjenega motorja MAN D0926 LOH15 z delovno

flows with laser Doppler anemometry (LDA) measurements and explore the applicability and the limitations of the CFD representation of the in-cylinder flow pattern. The complexity of the fluid motion in IC engines makes it difficult to extrapolate experimental results from one engine geometry to another [15]; therefore, validated and calibrated CFD simulation codes are gaining popularity when the optimization of the combustion-chamber geometry, the induction and exhaust system geometry, etc., is concerned. Therefore, a cylinder model with optical access was built, and its geometry was simultaneously implemented into the CFX (AEA Technology) simulation code [16]. The presented analysis has two goals: 1) to analyze the accuracy and applicability of the simulated intake-valve discharge coefficient and 2) to analyze and explore the applicability of the CFD simulations to predict the in-cylinder flow structures.

The intake-valve discharge coefficient is a fundamental variable that substantially influences the quantity of fresh air in the cylinder, and therefore the engine performance and the emission of pollutants. It is also an important input parameter for all zero- and one-dimensional engine-simulation models. An analytical determination of the flow coefficients is not possible due to the complexity of the fluid flow.

1 EXPERIMENTAL METHODS

1.1 Experimental equipment

The measurement equipment used for all the experiments is presented in Fig.1; it consists of five general components:
- an LDA system for in-cylinder velocity measurements
- a computer-controlled positioning system for an applied optical probe
- a gauge-pressure measurement system
- a vacuum-cleaner suction unit to set the appropriate air flows and pressure drops across the cylinder and cylinder head
- a real engine cylinder head attached to the glass model of the cylinder; the setting of an arbitrary intake-valve clearance and any piston-crown position is possible.

An octagonal model cylinder made of optical glass (similar to the cylinder of the four-stroke 6.871 l turbocharged and aftercooled MAN D0826 LOH15 prostornino 6,871 dm^3. Podrobnosti lahko najdemo v magistrskem delu Prijatelja [17]. Natančna računalniško vodena položajna naprava s koračnimi motorji (natančnost ± 0,01 mm) je omogočala natančno nastavitev, ponovljivost lege izbranih merilnih točk in meritev tretje komponente prostorskega vektorja hitrosti. Trorazsežno hitrostno polje je bilo določeno v 165 izbranih točkah notranjosti modela valja za različne nastavitve odprtja vstopnega ventila. Z uporabo elektronsko krmiljenega ventilatorja sesalnega stroja je bilo mogoče nastaviti različne pretočne količine zraka pri izbranem tlačnem padcu v vstopnem ventilu in tako umetno ustvariti razmere, kakršne so pri dejanskem motorju. Za merjenje hitrosti toka je bil uporabljen dvožarkovni LDA za sočasno merjenje dveh komponent hitrosti izdelovalca TSI z imensko močjo 4 W, ki pa je deloval dejansko z 1 W. S pomočjo položajne naprave in zavrtitvijo optične leče za 90° je bilo mogoče določiti tudi tretjo komponento hitrosti v isti merilni točki. Dodatno sipanje svetlobe in obravnavo večjega vzorca podatkov v vsaki merilni točki je zagotavljalo dodajanje drobnih kapljic glicerina v vstopni zračni tok. Natančnost LDA je bila preverjena pri ustaljenem načinu delovanja z vrtljivo ploščo; pri hitrosti 25 m/s je bil odstopek manjši od 0,2 %. Razporeditev merilnih mest v opazovanih vodoravnih prereznih ravninah modela valja bo prikazana v poglavju 3.

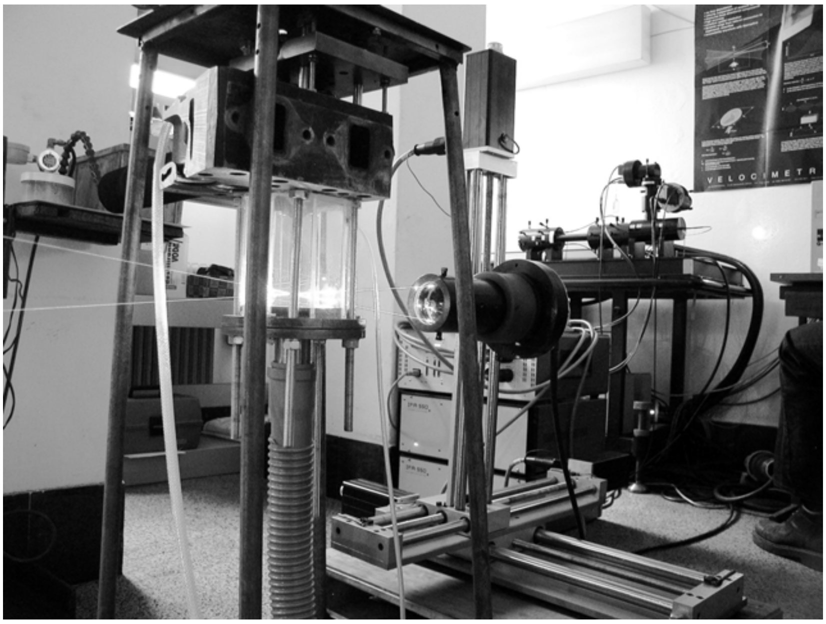
Payri in soavtorji [18] so ugotovili, da oblika čela bata, oziroma oblika zgorevalnega prostora v batu, nimata pomembnega vpliva na obliko toka v času celotnega sesalnega in vsaj v začetku takta stiskanja. O podobnih ugotovitvah poročajo tudi Zhu in soavtorji [19], ki poleg omenjenega tudi ugotavljajo, da je vpliv oblike kotanje v batu pri hitrotekočih dizelskih motorjih pomemben šele v zadnji fazi postopka stiskanja delovnega zraka, ko se pojavi prečno iztiskanje zraka iz čela bata v notranjost zgorevalne kotanje v batu. Tudi Arcoumanis in soavtorji [20] ter Wakisaka in soavtorji [21] ugotavljajo, da oblika zgorevalnega prostora nima vpliva na obliko tokovnega polja pri polnjenju valja motorja. Tako lahko ugotovimo, da zaradi poenostavitve oblike čela bata (ravna površina) pri opravljenih preizkusih ni trpela kakovost dobljenih rezultatov numerične simulacije RDT niti rezultati preizkusnih raziskav v modelu valja.

V tem prispevku je bil pretočni koeficient polnilnega ventila izmerjen v pogojih ustaljenega toka in mirujočega bata. Podobni način so uporabili

diesel engine) with a moving piston and with an attached real metal cylinder head was applied for the experimental analysis. Details of the design and the experimental procedure can be found in the work of Prijatelj [17]. An accurate (accuracy ± 0.01 mm) positioning system controlled by a PC with stepper motors ensured the repeatability of the selected measurement points and the measurement of the third flow-velocity component. The velocity field in the whole cylinder was determined for selected intake-valve lifts. The speed-controlled fan of a vacuum cleaner made possible settings of the pressure drop in the inlet valve as well as air mass-flows that corresponded to realistic engine operation under diverse running conditions. A two-component laser Doppler anemometer (LDA) for simultaneous measurement of two velocity components (blue and green beams), manufactured by TSI with a rated power of 4 W, but operating at approximately 1 W, was applied for the experiments. The third velocity component was measured by a simple turning of the optical probe by 90 degrees. Glycerin drops were introduced in the air-flow as seeding particles to ensure an appropriate number of sampled data to determine the flow-velocity components. The accuracy of the applied LDA was previously tested with a rotating disc: at a velocity of 25 m/s the error was smaller than 0.2 %. The determination of the observed horizontal measurement planes in the cylinder and the position of the measurement points will be given in the 3. section.

According to Payri et al [18], the piston-crown geometry (facing the combustion chamber) has little influence on the in-cylinder flow during the entire intake stroke and during the first part of the compression stroke; this conclusion also corresponds to the results published by Zhu et al [19], who reported that the effects of the geometry of the piston bowl pip for a high-speed direct-injection diesel engine become more pronounced late in the compression stroke due to the squish. Furthermore, Arcoumanis et al [20] and Wakisaka et al [21] reported that the geometry of the combustion chamber is not significant when the air flow inside the cylinder during the intake stroke is analyzed. Hence, it can be concluded that the quality of the results obtained with the simplified geometrical model of the piston (flat piston crown) applied for CFD and experimental analyses are not affected by this simplification.

The results of the measurements and computations of the steady-flow discharge coefficient are presented in the second part of the presented



Sl. 1. *Preizkusna naprava za meritev hitrostnega polja v modelu valja*
Fig.1. *Experimental setup for in-cylinder velocity measurements*

tudi drugi avtorji ([22] do [24]), medtem ko sta Fukutani in Watanabe [25] potrdila dobro ujemanje med statičnimi in dinamičnimi vrednostmi pretočnega koeficienta.

2 NUMERIČNE SIMULACIJE VTOKA V VALJ

Natančnost izračunanih rezultatov je močno odvisna od geometrijske podobnosti modela pretočnih kanalov, ventila in valja, ki so bili uporabljeni pri računskem modelu. Z uporabo Microscribe 3 D kopirne roke in programom računalniško podprtega načrtovanja (RPN) so bili obrisi pretočnih površin notranjosti valjeve glave označeni in kasneje s programom za 3 D modeliranje Mechanical Desktop ponovno spremenjeni v dejansko obliko pretočnih kanalov motorja. ki je bila primerna za obravnavo v programskem paketu RDT. Ta omogoča ob upoštevanju predpisanih robnih pogojev rešitev gibalnih enačb in izračun značilnosti dinamike zračnega toka (RDT). Natančnost izračunanih rezultatov je odvisna od gostote računske mreže in izbranega turbulentnega modela. Primer izbrane oblike in gostote računske mreže v izbranem vzdolžnem prerezu modela vtočnega kanala in valja, je prikazan na sliki 2. Payri in soavtorji [18] so na podlagi preizkusnega dela ugotovili, da predpostavka izotropne turbulence v običajnem

study. A similar experimental procedure was also proposed by other authors ([22] to [24]), while Fukutani and Watanabe [25] confirmed good agreement between the static and dynamic discharge coefficient.

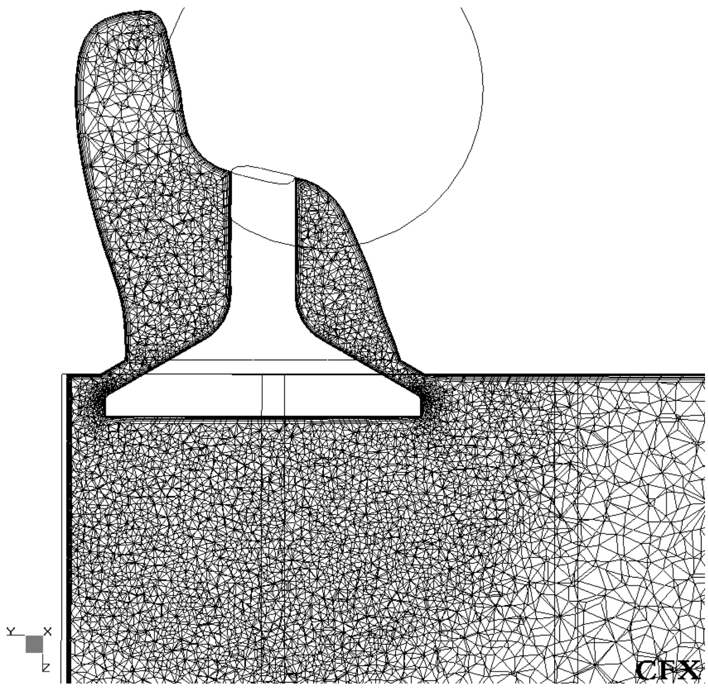
2 NUMERICAL SIMULATIONS OF THE INTAKE FLOW

The quality of the computed results strongly depends on the applied geometry of the engine intake channel and the cylinder. An appropriate geometry of the engine intake channel was therefore transferred to the 3D model, applying the CAD code. A 3D contour of the real engine system was precisely copied by the measurement arm Microscribe 3D and then transformed into a realistic engine geometry by applying a computer program for 3D modeling - Mechanical Desktop. The channel, cylinder, valve etc., geometry model was finally transferred into the CFX program package applied for the numerical evaluation of the flow dynamics (CFD) by solving momentum equations and by considering the prescribed boundary conditions. The accuracy of the computed results mostly depends on the density of the computational mesh and on the suitability of the applied turbulent model. The details of the applied mesh configuration in one longitudinal section of the observed inlet channel and cylinder are presented in Fig.2. The standard k–ε turbulence modelu k–ε ne prinaša večjih napak pri izračunu tokovnih razmer v valju motorja – posebno primerna je za modeliranje toka v zgorevalnih prostorih brez poudarjenega vrtinčnega toka oziroma brez izrazite prečne komponente hitrosti. S tem je bila potrjena tudi upravičenost uporabe turbulentnega modela k–ε tudi v obravnavanem primeru pri reševanju diskretiziranih Navier-Stokesovih enačb.

Tudi izbira robnih pogojev je bila v prikazanem primeru podobna ustrezni izbiri drugih avtorjev (npr. Payri [18]): robni pogoj nespremenljivega tlaka je bil predpisan na vstopu in izstopu zračnega toka v model valja, medtem ko sta bila za stene valja in vstopnih kanalov predpisana adiabatni in robni pogoj brez zdrsa (obodna hitrost tekočine ob steni je enaka 0). Poleg tega je bila predpisana tudi stalna vrednost temperature zraka na vstopu in predpostavka nestisljivega sredstva. Ta predpostavka ni vnesla v izračun gostote zraka pomembnejše napake, ker so bile tudi ustrezne spremembe tlaka (podatek) v pretočnem prerezu ventila zelo majhne (tlačno razmerje v ventilu znaša 0,97 do 0,99). Običajni turbulentni model k–ε je bil uporabljen tudi pri izračunu (modeliranju) pretočnega koeficienta. Vrednosti za turbulentno kinetično energijo k in raztrosno hitrost kinetične energije ε je

model was applied to solve the discretized Navier-Stokes equations, since Payri et al [18] experimentally verified that the isotropic turbulence assumption of the standard k–ε model is reasonably accurate for calculations of the in-cylinder flow, particularly in quiescent combustion chambers where there is no strong interaction between swirl and squish and, more generally, in zones of the cylinder where the radial flow does not dominate the flow pattern.

In accordance with other publications (see also [18]), constant pressure boundaries were assigned to the flow inlet and outlet, whereas no-slip and adiabatic boundary conditions were applied on solid boundaries. The measured temperature was assigned to the intake-flow boundary condition, and the non-compressible flow was assumed during the computation procedure due to a very small air density change being the consequence of very small pressure fluctuations in the intake valve (the pressure ratio in the inlet valve was 0.97 to 0.99). The values for the turbulent kinetic energy k and the velocity dissipation of the kinetic energy ε should also be prescribed at the flow inlet boundary condition. The option "default intensity and autocompute length scale" was assumed for the performed computations due



Sl. 2. *Primer uporabljene računske mreže v vzdolžnem prerezu modela vstopnega kanala in valja*
Fig.2. *Details of the computational mesh in the longitudinal section of the inlet channel and cylinder*

bilo treba določiti na vstopnem robnem pogoju. Zaradi nepoznavanja teh vrednosti je bila privzeta intenziteta in izračunana dolžinska skala, kar pa na srečo ni imelo bistvenega vpliva na izračunano strukturo toka.

3 PRIKAZ IN OBRAVNAVA REZULTATOV

3.1 Tokovno polje in potek hitrosti v valju motorja

V sklopu preizkusnih raziskav so bili na 33 različnih merilnih mestih v petih vodoravnih – prečnih ravninah osmerostranega valja (torej skupno v 165 točkah) izmerjene po tri komponente vektorja hitrosti toka. Več točk v vsaki ravnini ni bilo mogoče izmeriti zaradi poševne postavitve steklenih sten modela. Na sliki 3 je prikazan potek oziroma primerjava izmerjenih in izračunanih hitrosti v prečni ravnini $x-y$, ki je oddaljena 120 mm od vrha valja. Iz diagrama lahko povzamemo, da:

- se lega središča izračunanega vrtinca le malo razlikuje od lege središča vrtinca toka, ki je bil izmerjen z meritvami LDA,
- se absolutna velikost izračunanih in izmerjenih vrednosti vektorjev hitrosti v opazovani ravnini precrej dobro ujema z vrednostmi meritev.

Obodno gibanje (kroženje v prečnih ravninah) sveže polnitve v valju je zelo pomembno za pravilno

to a lack of any appropriate information. Fortunately, their choice has little effect on the flow structure for the observed case. The standard turbulent k–ε model was also applied to compute the flow-discharge coefficient.

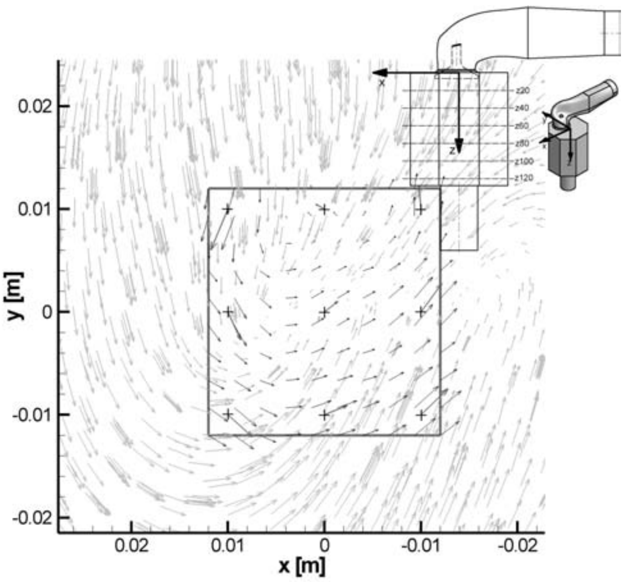
3 RESULTS AND DISCUSSION

3.1 Flow pattern and velocity field within the cylinder

Three components of the velocity vectors were measured in the octagonal glass cylinder model at 33 different measurement points and in 5 different cylinder transverse planes (altogether at 165 locations). Fig. 3 represents the computed and measured velocity vectors in the *x-y* plane, 120 mm from the cylinder top (z – axis). It can be concluded from this diagram that:

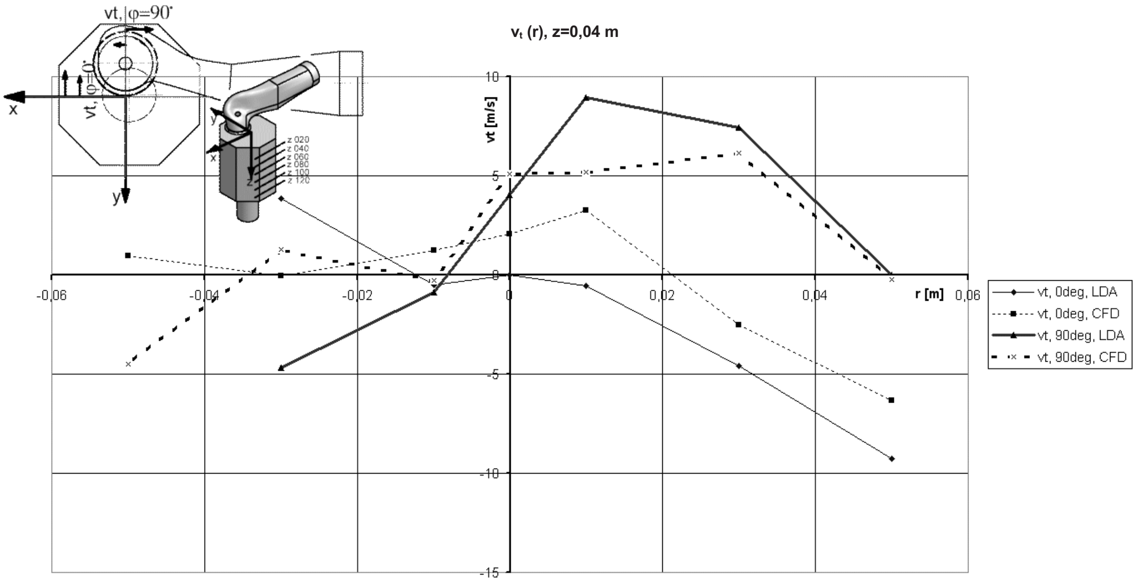
- the positions of the calculated and measured vortex center nearly coincide
- the size of the computed and measured velocity vectors (length of particular indicated velocity vector) are similar within the observed plane

Tangential motion (in the transverse cylinder planes) of the fresh charge is very important for appropriate air-fuel mixing, and therefore for effective combustion. Thus, the swirl ratio or the swirl



Sl. 3. *Primerjava izmerjenih in izračunanih vektorjev hitrosti v prečni ravnini valja 120 mm od vrha (smer z); temnejše puščice – meritev, svetlejše puščice – računska simulacija*

Fig. 3. *Comparison of measured and computed velocity vectors in the cylinder transverse section 120 mm from the top (direction z); black arrows – experiment, light arrows – numerical simulations*



Sl. 4. *Primerjava izmerjenih in izračunanih obodnih hitrosti v dveh izbranih legah v prečni ravnini valja 40 mm od vrha (smer z)*

Fig. 4. *Comparison of the measured and computed tangential velocity components in the cylinder transverse section 40 mm from the top (direction z)*

mešanje zgorevalnega zraka z gorivom in posledično za čim popolnejše zgorevanje. Velikost in porazdelitev obodne hitrosti toka v valju močno vplivata na vrtinčno število toka v valju, ugotavljajo Payri in soavtorji [18]. Zato je na sliki 4 prikazana tudi primerjava poteka in velikosti obodne komponente vektorja hitrosti v prečni ravnini valja $x-y$ 40 mm od vrha valja. Potek in smer opazovanih hitrosti sta podobna za oba primera. Velikost izračunanih vrednosti, posebno blizu stene valja (ni prikazano na tej sliki) in v prečnih ravninah na polovici višine valja, je večja od izmerjenih vrednosti. Vzrok tiči verjetno v naslednjih pomanjkljivostih: 1) neupoštevanju spremembe lastnosti toka pri pretakanju ob steni osmerostranega modela valja in 2) zaradi nenatančnih meritev padca tlaka med opazovanim vstopnim in izstopnim robom toka zraka, ki je rabil tudi kot robni pogoj pri numeričnem modeliranju.

3.2 Pretočni koeficient vstopnega ventila

3.2.1 Določitev pretočnega koeficienta z meritvami in računsko simulacijo

Najprej je bil pretočni koeficient določen z uporabo rezultatov meritev. Potem je bila izvedena računska simulacija masnega toka zraka skozi vstopni

number is strongly affected by the size and the distribution of the tangential velocity, F.Payri et al [18]. The distribution of the calculated and measured tangential velocity component in the transverse plane 40 mm from the cylinder top (z = 40 mm), which is perpendicular to the x and y axes is presented in Fig. 4. It can be concluded that the orientation of the tangential velocities is similar for both cases. The calculated values are larger in comparison with the measured ones, especially in the transverse planes half-way from the cylinder top and in the vicinity of the cylinder wall (not presented in this figure). The reason for this can be found in: 1) a probably incorrect consideration of the flow characteristic when the computation of the flow along the octagonal cylinder wall is considered, and 2) incorrect measurement of the pressure drop, which represents the boundary condition for the performed computations.

3.2 Discharge coefficient

3.2.1 Experimental and numerical determination of the discharge coefficient

The discharge coefficient was first determined experimentally, i.e., it was calculated from the measured in-cylinder mass-flow data by applying ventil, izračunani ustrezni pretočni koeficienti in rezultati primerjani z rezultati preizkusov. Pretočni koeficient μ je bil določen z enačbo:

the equation of compressible flow through the nozzle, see also [17]. The discharge coefficient is determined by the equation:

$$\mu = \frac{A_{eff}}{A_g} \quad (1),$$

kjer pomeni A_g najmanjšo geometrijsko pretočno površino vstopnega ventila, A_{eff} pa dejanski pretočni prerez v vstopnem ventilu:

A_{eff} je bil potem izračunan z enačbo (2), ki opredeljuje masni tok (podrobnosti najdemo v viru [17]) stisljivega medija skozi zaslonko pri znanih podatkih: tlačnem razmerju v ventilu p_2/p_1=0,97, ter temperaturi in tlaku okolice T_1 = 295K in p_1 = 100 kPa:

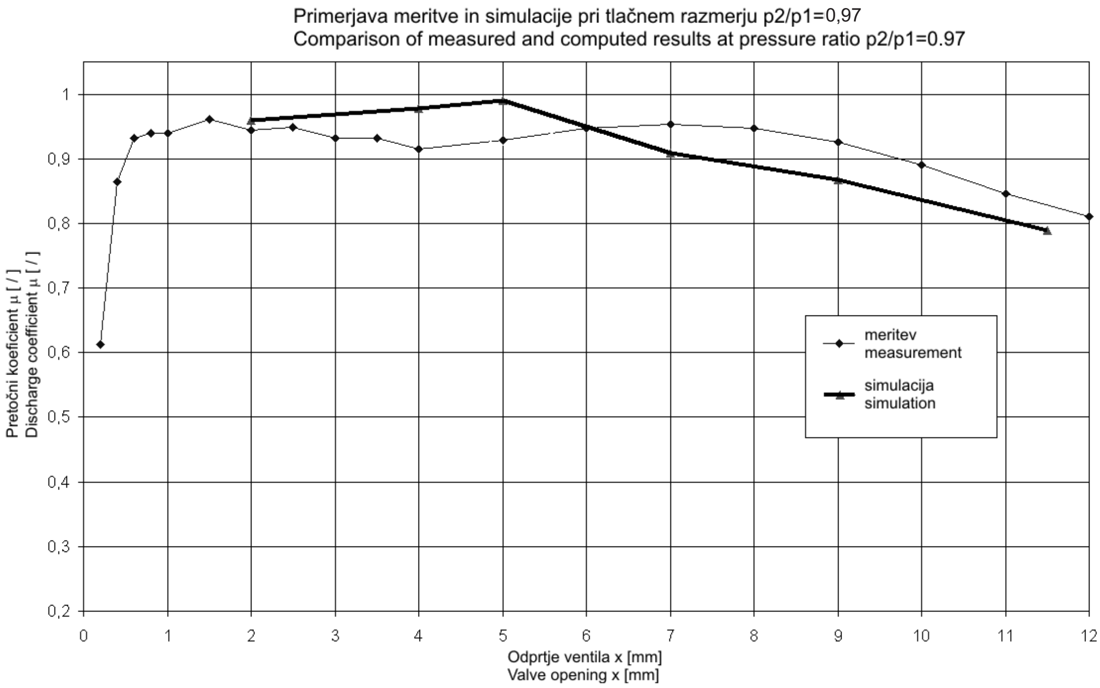
where A_{eff} denotes the effective cross-section through the inlet valve and A_g the minimum geometrical cross-section of the inlet valve.

A_{eff} was then calculated from Equation (2) by applying the results of the simulation for the mass-flow $\dot{m}$, from the known pressure ratio across the valve p_2/p_1=0.97 and from the ambient parameters T_1 = 295K and p_1 = 100 kPa:

$$A_{eff} = \dot{m} \cdot \left(\cdot \frac{p_1}{\sqrt{T_1}} \cdot \sqrt{\frac{2\kappa}{\kappa\text{-}1} \cdot \frac{1}{R} \cdot \left(\left(\frac{p_2}{p_1}\right)^{\frac{2}{\kappa}} - \left(\frac{p_2}{p_1}\right)^{\frac{\kappa+1}{\kappa}} \right)} \right)^{-1} \quad (2).$$

Primerjava izmerjenih in izračunanih vrednosti koeficienta μ je prikazana na sliki 5. Rezultati se zelo dobro ujemajo pri odprtju ventila za 2 mm. Potem je opazno hitrejše večanje in manjšanje (odmik) izračunanih vrednosti koeficienta μ. V celoti se vrednosti meritev in izračunov sorazmerno dobro ujemajo. Tudi oblika spremembe poteka izmerjene kriulje μ s spremembo odprtja vstopnega ventila se ujema z napovedmi drugih avtorjev ([22] do [24]) in je potrjena s spremembami toka v špranji ventila. Pri majhnih odprtjih ventila, pri katerih ne prihaja do odcepitev toka od trdne stene, vpliv mejne plasti pa je tudi zanemarljiv, smer toka sledi obliki ventilne glave in sedeža ventila. Z večanjem odprtja ventila se zato v prvi fazi odpiranja ventila intenzivno povečuje tudi vrednost μ. Pri nekoliko večjih tokovih, večjih odprtjih ventila, se najprej pojavi odcepitev toka od vodilne površine ventilne glave; dejanska pretočna površina se zmanjša in z njo tudi vrednost μ. Z nadaljnjim povečevanjem odprtja ventila opazimo odcepitev toka tudi na strani sedeža ventila, kar vpliva na nadaljnje zmanjšanje vrednosti μ. Potem se usmeritev zmanjševanja vrednosti μ nekoliko umiri zaradi ponovnega prilepljenja toka ob vodilno površino ventilne glave in s tem nekoliko povečanega dejanskega pretočnega prereza ventila. Pri velikih vrednostih odprtja ventila opazimo nadaljnje, vendar zmerno zmanjševanje pretočnega koeficienta.

The results of the numerical simulations of the discharge coefficient in the inlet valve were compared to those obtained by experiments. The results of the comparisons are presented in Fig. 5. The results coincide very well for a 2-mm valve lift, whereas it can be seen that the numerical simulation predicts a gradual increasing and a subsequent decreasing of the discharge coefficient earlier than the experimental data for approximately 2 mm of the valve lift. Overall, the measured and computed results are in relatively good agreement. The shape of the discharge coefficient curve also coincides well with the results reported by other authors ([22] to [24]). From the analysis of the flow structure it can be seen that at very small valve openings the air flow follows the shape of the valve seat and the valve head – no flow separation occurs and due to the negligible effects of the boundary layer the discharge coefficient increases with increasing valve lift. At larger airflows the boundary layer breaks away from the valve head, reduces the effective flow area and thus reduces the discharge coefficient. Further opening of the valve gives rise to the flow separation, even at the side of the valve seat, and substantially decreases the discharge coefficient. The decrease of the discharge coefficient is less pronounced with still further opening of the valve, due to the re-attachment of the flow to the valve head – the effective flow area is, therefore, relatively larger. The discharge coefficient then continues to decrease at the largest valve openings, but at a lower rate.



Sl. 5. *Potek pretočnega koeficienta v odvisnosti od odprtja ventila*
Fig.5. *Discharge coefficient for the intake valve, depending on the valve lift*

3.2.2 Ocena merilne negotovosti pri določitvi pretočnega koeficienta

Pri oceni skupne merilne negotovosti za pretočni koeficient ě, so bili v skladu z enačbama 1 in 2 in ob upoštevanju ustrezne literature [26] upoštevane naslednje merilne negotovosti:

- ocenjena merilna negotovost izmerjenih statičnih tlakov $u_r(p1) = \pm 1\%$
- merilna negotovost izmerjene temperature $u_r(T) = \pm 0,17\%$
- merilna negotovost izmerjenega masnega toka zraka $u_r(\dot{m}) = \pm 0,8\ \%$
- merilna negotovost izmerjenega masnega toka zraka $u_r() = \pm 0,8\ \%$
- pri določitvi merilne negotovosti geometrijskega pretočnega preseka A_g so bile ocenjeni merilni pogreški: pri določitvi premera ventila ± 1 mm in pri dvigu ventila ± 0,05 mm; ocenjena merilna negotovost $u_r(A_g)$ je torej znašala ±2,5%.
- Izračunana je bila tudi merilna negotovost dejanske - efektivne pretočne površine ventila $u_r(A_{eff})$ v merilnem območju masnih tokov zraka (= 0,1 kg/s), ki ne presega ± 1,38%

Končno je bila izračunana še skupna merilna negotovost pretočnega koeficienta:

3.2.2 Assessment of the measurement uncertainty for the determination of the discharge coefficient

Taking into account measurement parameters in Equations 1 and 2 and in corresponding literature [26] when determining the total collective measurement uncertainty the following individual uncertainties were considered:

- Of the measured absolute pressures $u_r(p1) = \pm 1\%$
- Of the measured temperatures $u_r(T) = \pm 0.17\%$
- Measurement uncertainty of the measured air mass-flow $u_r(\dot{m}) = \pm 0.8\ \%$
- The following measurement errors were estimated when assessing the measurement uncertainty of the geometrical valve flow area A_g: ± 1 mm for the valve diameter and ± 0.05 mm for the valve lift; the assessed measurement uncertainty $u_r(A_g)$ was therefore ±2.5%.
- The measurement uncertainty of the valve effective flow area $u_r(A_{eff})$ was then determined for the observed mass-flow range of 0.1 kg/s and amounted to ±1.38%.

Finaly the total measurement uncertainty of the discharge coefficient was determined for the observed air mass-flow applying the formula:

$$u_r(\mu) = \sqrt{(u_r^2(A_{eff}) + u_r^2(A_g)} = \sqrt{0{,}0138^2 + 0{,}0253^2} = \pm 0{,}029 = 2{,}9\% \quad (3),$$

ki v opazovanem merilnem območju masnih tokov zraka okrog 0,1 kg/s ne presega ±2,9%

4 SKLEPI

Iz predstavljenih rezultatov raziskave lahko povzamemo, da je uporaba metode RDT primerna in da zagotavlja prepričljive rezultate, ki izboljšujejo razumevanje tokovnih pojavov, ki spremljajo vtok medija v valj motorja. RDT je torej učinkovito orodje pri snovanju sodobnih motorjev. Omogoča dobro napoved pojava, mesta in velikosti vrtinčnih struktur v valju. Primerjava med izmerjenimi in izračunanimi vrednostmi obodnih komponent hitrosti toka v valju modela motorja je pokazala, da so izračunane vrednosti večje predvsem v bližini stene valja in v prečnih ravninah, ki ležijo v področju polovice višine valja. Na to so verjetno vplivali: netočni izmerjeni podatki padca tlaka v ventilu, ki so obenem pomenili tudi predpisani robni pogoj pri opravljenih računskih simulacijah in nepoznavanje tokovnih razmer pri kroženju zraka ob osmerostranih stenah modela valja.

Dobro ujemanje med izmerjenimi in izračunanimi vrednostmi lahko opazimo tudi pri rezultatih pretočnega koeficienta v vstopnem ventilu. Iz rezultatov izračunov izhaja, da je izračun zanesljivejši pri večjih odprtjih ventila, ko je vpliv mejne plasti na tok zraka v špranji med glavo in sedežem ventila manjši. Gostota računske mreže je lahko zaradi tega pri večjih odprtjih ventila ustrezno manjša, s tem pa se ustrezno izboljša tudi razmerje med kakovostjo rezultatov in zmanjšano potrebno porabo računalniškega časa. Analiza dobljenih rezultatov in primerjava z ustreznimi rezultati drugih avtorjev je tudi pokazala, da je vpliv oblike čela bata (oziroma oblike zgorevalnega prostora v batu) in lege bata v valju na velikost in potek pretočnega koeficienta majhna.

that never exceeded ±2.9%.

4 CONCLUSIONS

The presented study shows that in-cylinder CFD predictions ensure credible results that help improve the knowledge of the air-flow characteristics during the intake stroke. CFD therefore represents an efficient design tool for developing less-polluting and more efficient internal combustion engines. Good prediction of the vortex center, direction and even the size of the flow field was made possible by applying the described numerical method. A comparison between the measured and computed tangential in-cylinder velocities showed that the relatively larger computed values – especially close to the cylinder wall – were probably affected by the incorrect measurement of the pressure drop across the intake valve; it was applied as a prescribed boundary condition for the performed computations. On the other hand, it is also difficult to precisely determine the flow field close to the octagonal walls of the model cylinder.

Good agreement between the measured and calculated inlet-valve discharge coefficients was obtained. It can be concluded from the performed computations that it is was relatively convenient to calculate the discharge coefficients at larger inlet valve openings, since the boundary layer has a smaller influence on the air-flow within the restricted area between the valve head and the valve seat. A good compromise between the quality of the results and the computational times could also be achieved for larger valve openings, since accurate results could be obtained with a reasonable computation-mesh density. The analysis and comparisons with the appropriate results of other authors showed that that the piston position in the model cylinder has little effect on the size of the discharge coefficient.

5 LITERATURA
5 LITERATURE

[1] Rask, R.B. (1979) Laser Doppler anemometer measurements in an internal combustion engine. *SAE Paper 790094*, 1979.

[2] Heywood, J.B. (1987) Fluid motion within the cylinder of internal combustion engines. *Journal of Fluids Engineering*, 109 (1987), pp.3-35.

[3] Heywood, J.B. (1988) Internal combustion engine fundamentals. *McGraw-Hill*, New York.

[4] Huang, R.F., Huang, C.W., Chang, S.B., Yang, H.S., Lin, T.W., Hsu, W.Y. (2005) Topological flow evolutions in cylinder of a motored engine during intake and compression strokes. *Journal of Fluids and Structures*, 20 (2005), pp.105-127

[5] Kang, K.Y., Reitz, R.D. (1999) Intake flow structure and swirl generation in a four-valve diesel engine. *Spring Technical Conference, ASME 1999*, ICE-vol. 32-2 (1999) , Paper no. 99-ICE-182

[6] Snauwaert, P., Sierens, R. (1986) Experimental study of the swirl motion in the direct injection diesel engines under steady state flow conditions (by LDA). *SAE Paper 860026*.

[7] Yun, J.E. (2002) New evaluation indices for bulk motion of in – cylinder flow through intake port system in cylinder head. *Proc. Instn. Mech. Engrs.* Part D, 216 (2002), pp. 513-521

[8] Davis, G.C., Kent, J.C. (1979) Comparison of model calculations and experimental measurements of the bulk cylinder flow processes in a motored PROCO engine. *SAE Paper 790290*, 1979.

[9] Murakami, A., Arai, M., Hiroyasu, H. (1988) Swirl measurements and modeling in direct injection diesel engines. *SAE Paper 880385*, 1988.

[10] Desantes, J.M., Rastor, J.V., Doudou, A. (2001) Study of the steady flow produced by direct injection diesel engine intake ports. *Proc. Instn. Mech. Engrs.* Part D, 215 (2001), pp. 285-298

[11] Uzkan, T., Borgnakke, C., Morel, T. (1983) Characterisation of the flow produced by a high-swirl inlet port. *SAE Paper 830266*, 1983.

[12] Haste, M.J., Garner, C.P., Halliwell, N.A. (1999) A PIV study of the inlet port deactivation on the in-cylinder flow structure in an SI engine. *Fall Technical Conference, ASME 1999*, ICE-vol. 33-3 (1999), Paper no. 99-ICE-240

[13] Li, Y., Zhao, H., Peng, Z. Ladommatos, N. (2002) Particle image velocimetry measurement of in-cylinder flow in internal combustion engines-experiment and flow structure analysis. *Proc. Instn. Mech. Engrs.* Part D, 216 (2002), pp. 65-81

[14] Chan, V.S.S., Turner, J.T. (2000) Velocity measurement inside a motored internal combustion engine using three-component laser Doppler anemometry. *Optics & Laser Technology*, 32 (2000), pp. 557-566

[15] Payri, F., Desantes, J.M., Pastor, J.V. (1996) LDV measurements of the flow inside the combustion chamber of a 4-valve D.I. diesel engine with axisymetric piston-bowls. *Experiments in Fluids*, 22 (1996), pp. 118-128

[16] User manual for the program package CFX 5.5

[17] Prijatelj, S. (2004) Analysis of flow phenomena in a valve of internal combustion engine. M.SC. Thesis No. M/1277 (2004). *Faculty of Mechanical Engineering, Ljubljana, UL*

[18] Payri, F., Benajes, J., Margot, X., Gil, A. (2004) CFD modeling of the in-cylinder flow in direct-injection diesel engine. *Computers & Fluids*, 33 (2004), pp.995-1021

[19] Zhu, Y., Zhao, H., Melas, D.A., Ladommatos, N. (2004) Computational study of the effects of the geometry of the piston bowl pip for a high-speed direct injection diesel engine. *Proc. Instn. Mech. Engrs.* Part D, 218 (2004), pp. 875-890

[20] Arcoumanis, C., Begleris, P., Gosman, A.D., Whitelaw, J.H. (1986) Measurements and calculations of the flow in a research diesel engine. *SAE Paper 861564*, 1986.

[21] Wakisaka, T., Shimamoto, Y., Issihiki, Y. (1986) Three-dimensional numerical analysis of in-cylinder flows in reciprocating engines. *SAE Paper 860464*, 1986.

[22] Tsui, Y.Y., Lee, S.Y. (1992) Calculation of turbulent flow through engine inlet ports. *Int. J. Heat and Fluid Flow*, 13 (3) (1992), pp. 232-240

[23] Arcooumanis, C., Whitelaw, J.H. (1987) Fluid mechanics of internal combustion engines – a review. *Proc. Instn. Mech. Engrs.*, 201 (1987), pp. 57-74

[24] Baumruk, P., Hatschbach, P., Mareš, B. (2003) Simulation of steady flow through engine inlet port to cylinder. *2003 EAEC European Automotive Congress A*, pp. 229-237.

[25] Fukutani, I., Watanabe, E. (1995) Air flow through poppet inlet valves – analysis of static and dynamic flow coefficients (820154); Design of racing and high performance engines PT-53, *SAE* 1995, pp.111-121

[26] Coleman, H.W., Steele, W.G.. (1998) Experimentation and uncertainity analysis for engineers, 2nd edition. *John Wiley & Sons, Inc.*, New York.

Naslovi avtorjev: dr. Samuel Rodman Oprešnik
dr. Tomaž Katrašnik
prof.dr. Ferdinand Trenc
dr. Frančišek Bizjan
Univerza v Ljubljani
Fakulteta za strojništvo
Aškerčeva 6
1000 Ljubljana
samuel.rodman@fs.uni-lj.si
tomaz.katrasnik@fs.uni-lj.si
ferdinand.trenc@fs.uni-lj.si
francisek.bizjan@fs.uni-lj.si

mag. Sašo Prijatelj
saso@basic.si

Authors' Addresses: Dr. Samuel Rodman Oprešnik
Dr. Tomaž Katrašnik
Prof.Dr. Ferdinand Trenc
Dr. Frančišek Bizjan
University of Ljubljana
Faculty of Mechanical Eng.
Aškerčeva 6
SI-1000 Ljubljana, Slovenia
samuel.rodman@fs.uni-lj.si
tomaz.katrasnik@fs.uni-lj.si
ferdinand.trenc@fs.uni-lj.si
francisek.bizjan@fs.uni-lj.si

Mag. Sašo Prijatelj
saso@basic.si

Prejeto:
Received: 26.8.2005

Sprejeto:
Accepted: 16.11.2005

Odprto za diskusijo: 1 leto
Open for discussion: 1 year