Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 101 (2016) 622 – 629

Energy Procedia

71st Conference of the Italian Thermal Machines Engineering Association, ATI2016, 14-16 September 2016, Turin, Italy

Large Eddy simulation of a steady flow test bench using OpenFOAM®

Cristian Catellani[a], Giulio Cazzoli[a]*, Stefania Falfari[a], Claudio Forte[a], Gian Marco Bianchi[a]

[a]*Department of Industrial Engineering, University of Bologna,V.le del Risorgimento 2, Bologna 40136, Italy*

Abstract

Stationary flow bench testing is a standard experimental methodology used by the automotive industry to characterize a cylinder head. In order to reduce the development time, the use of a CFD-based virtual test bench is nowadays a standard practice too. The use of a conventional RANS methodology for the simulation of the flow through the ducts of an engine head allows to get only the mean flow variables distributions because the time average of the generic flow variable fluctuation is zero by definition, but the fluid-dynamics of a stationary flow bench is not really stationary due to the flow instability induced by the duct design and the interaction between valve jets in a multi-duct head. In order to obtain an in-depth knowledge of the fluid-dynamics of a stationary flow bench test rig a LES simulation of a heavy duty DI diesel engine head with two intake ducts, for which experimental data was available, has been carried out using OpenFOAM®. The comparison between LES, experimental and conventional RANS results widened the understanding of the test-bench fluid-dynamics and of the swirl generation process. Due to the high computational cost of the LES approach, the outcomes of this latter have been also used to evaluate potential accuracy improvements of the RANS simulation, namely using a model sensible to flow anisotropies and curvatures such as a RSTM model. The simulation with the new turbulence model has been carried out and compared with the previous results demonstrating predictive improvements with an affordable computational cost for industrial routine usage.

Peer-review under responsibility of the Scientific Committee of ATI 2016.

Keywords: RANS; LES; Stationary flow; Test bench; OpenFOAM; Swirl; Cylinder head.

---

* Corresponding author. Tel.: +39-0512093316; fax: +39-0512093314.
E-mail address: Giulio.cazzoli@unibo.it

Peer-review under responsibility of the Scientific Committee of ATI 2016.
doi:10.1016/j.egypro.2016.11.079

1. Introduction

Among the key factors that severely influence both fuel efficiency and pollutant emissions, engine coherent flow structures such as swirl and tumble motions are of primary importance. These unsteady flow structures depend on the geometry of the engine and its operating conditions, so that some synthesis parameters, namely the permeability and the Swirl and Tumble ratio, are required [1]. The synthesis parameters are of paramount importance during design stage in order to predict and optimize the engine performance [2-7].

Since direct measurement of the flow structures in dynamic engine operation is currently feasible only with test engines equipped with optical access, these parameters are indirectly evaluated using stationary flow benches. This experimental equipment measure the flow rate and the charge motion across the engine head without taking into account the real engine geometry. In order to describe the dynamic of the fluid of a real engine from the synthesis parameters, some assumptions must be made. Despite no standardized testing methodology exists, the industrial de-facto standards, are the methods proposed by AVL [8] and Ricardo [9]. As observed by Li [10], the difference between the estimated swirl ratio can be up to 50%, so that great care has to be taken when comparing data coming from different sources, a comprehensive review of the most widely adopted techniques can be found in [11].

In the last decade the virtual approach based on the computational fluid dynamics has gained popularity. This technique allows to reach a level of detail that is difficult to obtain experimentally and to evaluate the performance of the engine components at an early design stage, thus reducing the prototyping effort with tremendous benefits in terms of cost savings. In an author's previous work [12] four different prototype engine heads have been simulated with a commercial code (AVL Fire 2010) adopting a Reynolds Averaged Navier-Stokes (RANS) modeling obtaining uneven results: for some heads a satisfactory agreement between experimental and numerical result has been reached, for other heads the results are not so good. This kind of results is not surprising since several scientific publications present similar findings: a reasonable accuracy is reached in the head permeability evaluation but there is pretty low reliability in swirl torque evaluation and mismatches up to 50% in the prediction of the latter can be found as reported by Yang [13] or Palumbo [14]. The adoption of more accurate predictive models, capable of reproducing most of the unsteady flow features characterizing turbulent flows, can reduce this mismatch: here is where Large Eddy Simulation (LES) can make the difference over RANS, albeit with an increased computational cost.

This work follows the aforementioned work of Forte et al. [12] choosing the head that presented the poorest agreement in terms of predicted swirl torque with a twofold objective: firstly applying the defined virtual flow bench methodology using OpenFOAM® v 2.3.0 [15] and a standard RANS approach to compare the predictive capabilities of the open-source code against the commercial one. Then, the LES methodology has been applied in order to better understand the system fluid dynamics and to investigate the causes of potential RANS predictive deficiencies. The quality of the analyses has been assessed through specific LES quality estimators and the computational results have been validated against measurements, showing pretty good agreement. Finally, the insights obtained through LES have been employed to propose a different RANS approach (Reynolds Stress Tensor Modelling RSTM) overcoming or, at least, alleviating the aforementioned predictive deficit.

2. Virtual steady flow bench

The prototype head object of the present study belongs to a 4 valves per cylinder heavy duty Diesel engine. In order to reproduce the effect of the impulse swirl meter with acceptable computational cost, the meter has been substituted inside the cylinder with a porous medium, modelled using the Darcy-Forchaimmer model, as it can be seen in fig. 1. Geometric details of the head, of the experimental methodology, obtained accordingly to Ricardo methodology [9], and of the porosity parameters are in Forte et al. [12] and are not repeated here for sake of brevity.

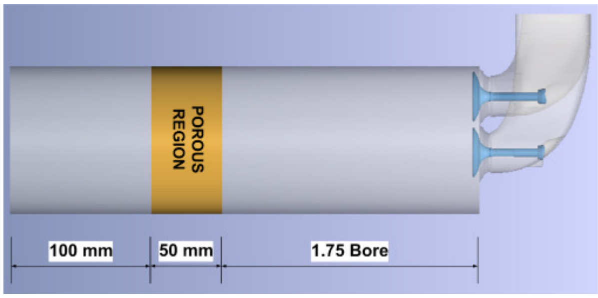


Fig. 1. Computational domain for the virtual steady flow bench and porous domain location

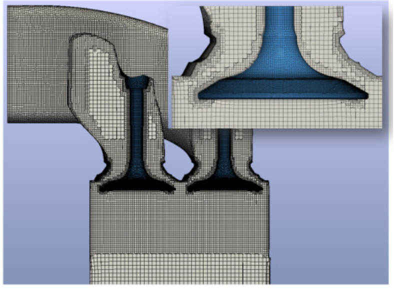


Fig. 2. RANS grid for the 5 mm valve lift

2.1. Geometry discretization

The domain discretization has been performed using cfMesh® [16], an open-source meshing program that produces hexa-dominant computational grids with optional local refinement and boundary layers.

The computational domain, reported in fig. 1, comprises the intake ducts, the engine head, the intake valves, the dummy cylinder up to the flowmeter, a cylindrical part with the same height of the flowmeter and an additional cylindrical part long enough to stabilize the outflow conditions. The reference geometry has been obtained assembling the individual STL files describing the aforementioned parts up to the flowmeter, then the valves are positioned at the required lift height with a simple translation operation.

2.2. Numerical setup: RANS

The mesh used for the RANS simulations (fig. 2) reaches approximately 2.1 million of cells with a bulk size (the maximum cell size) of 1.8 mm. In order to properly resolve the flow features, the ducts region and the one around the valves have been subject to local refinements up to 1/4 of the bulk size. To keep under control the number of cells, only in selected regions a 1/8 refinement has been used. At walls, two extruded layers have been used to better capture boundary layers.

The fluid used has been air at ambient conditions modeled as ideal gas. On the ducts inlet a fixed total pressure condition has been prescribed, while at the dummy cylinder outlet a static pressure value has been set along with a zero-gradient velocity condition. In order to smoothly develop the fluid flow, the domain has been initialized with an uniform static pressure field close to the inlet static pressure value, then the outlet pressure has been progressively ramped-down until the pressure difference applied to the system matches the value suggested by Ricardo and equals to 500 $\mathrm{mmH_2O}$. The turbulence has been modeled employing a high-Re RNG k-ε model with standard wall functions at solid surfaces. On the ducts inlet, turbulence variables have been set through the duct hydraulic diameter and prescribing a turbulent intensity of 5% while a zero-gradient outflow is specified at the cylinder outlet. Finally a steady compressible formulation has been adopted with a second order upwinded scheme for the convection and diffusion terms and employing the SIMPLEC algorithm for the pressure-velocity coupling.

2.3. Numerical setup: LES

For the LES cases a more refined grid is required, so the bulk mesh size has been reduced to 1 mm Local refinements are still applied, but no wall layer has been extruded and the mesh has been isotropically refined up to 1/8 of the bulk size at solid surfaces. The meshes so obtained were nearly 7 million of cells. Despite the significant refinements applied to the walls, it has not been possible to reach y+ values close to unity, therefore wall functions have been applied to all solid boundaries.

A transient formulation has been adopted, using a blended first/second order implicit scheme (Crank-Nicolson) and a second order TVD Gamma [17] scheme for the advection, as a reasonable trade-off between accuracy and

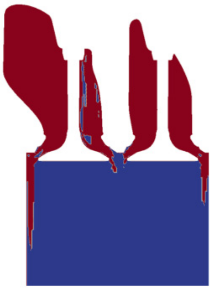
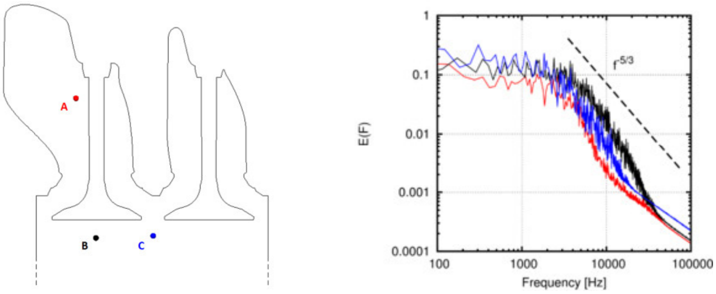


Fig. 3: Mean M parameter close to valve ports for valve lift 5 mm. In red the regions with M>0.2



a) Spectra probes locations

b) Turbulent kinetic energy spectra

Fig. 4. Spectral analysis for valve lift 5 for point A (red), point B (black), point C (blue)

stability. The pressure-velocity coupling has been performed using the PIMPLEC algorithm. Due to the transient formulation the outlet condition has been changed to a wave-transmissive condition to allow the proper treatment of the pressure waves. No explicit synthetic turbulence has been fed at inflows, since a natural onset of turbulence is expected before reaching the valve. The LES setup has been completed with the SGS WALE model, which, according to the previous authors' findings [19], is a good compromise between accuracy and computational cost. To be noticed that being these simulation compressible, the WALE model here used has been implemented in a compressible formulation as done by Catellani in his PhD thesis [19].

In order to save computing time, for each valve lift the domain has been initialized with the corresponding RANS results. The simulations have been then run for a total of 0.06 s and statistics have been accumulated after 0.025 s in order to allow the flow bench to reach a statistically steady regime. The time step has been fixed at 0.2 μs, this assure a Courant number less than 0.5 for the 99.9% of the computational cells. Due to the high computational cost involved, only three different valve lifts have been analyzed. The lifts are chosen based on the RANS simulation:

- 2 mm, as representative of a lift with reasonable agreement between computational results and experiments;
- 5 mm, where the maximum relative error between computations and experiments occurs;
- 10 mm, where the maximum absolute error between computations and experiments occurs.

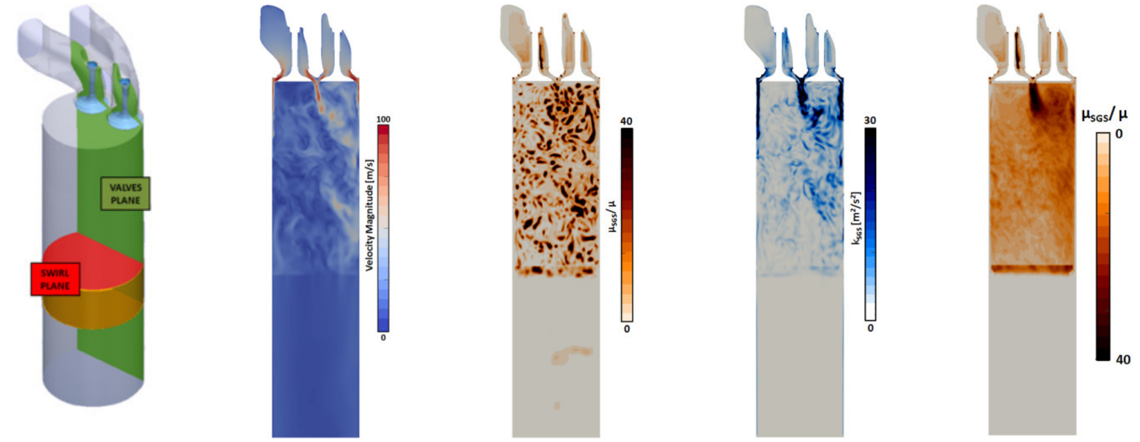
3. LES Result

The results of the LES simulation of the 5 mm valve lift case are presented and will be here discussed, the other lifts exhibited a similar behavior so, for sake of brevity, are not reported.

In order to assess the quality of the LES results, in analogy with the methodology adopted in [19], three probe locations have been chosen (fig. 4a): probe A lies close to the valve guide of duct 1, probe B is located underneath valve 1 head and probe C is located in between the two valves, where strong jets interactions are expected. The turbulent kinetic energy spectra for each probe are reported in fig. 4b. Point A exhibits the most noticeable deviation from the -5/3 theoretical slope, probably due to an insufficient resolution of the turbulent energy content in the ducts, Point B is characterized by a pretty neat spectrum with a trend quite close to the theoretical one. Point C exhibits a resolution level in between the other points.

The observed deviation is remarked by the Pope's M parameter [19] (fig. 3) that is pretty low in the whole domain, but show a resolution threshold violation almost uniform upstream of the valves and close to the liner near the valves. This highlights that a local mesh refinement comprising at least the valves region would be beneficial. Since for industrial (and academic) applications a reasonable compromise between accuracy and computational effort is required, the authors have chosen not to perform such refinement.

As expected, the LES simulations have shown the flow characteristics detectable also by RANS simulations, such as the local fluid recirculations past the valves and the jets attachment to the liner walls after impingement, as suggested by the instantaneous velocity magnitude map field fig. 5b. Furthermore, the chaotic nature of the flow is now revealed thanks to the capture of small flow structures as shown in fig. 5c. The valves' jet mutual interaction



a) b) c) d) e)

Fig. 5: LES results on the valves plane for valve lift 5: a) plane location, b) instantaneous velocity magnitude, c) mean viscosity ratio, d) instantaneous SGS turbulent kinetic energy, e) instantaneous viscosity ratio

and the jets impingement to the liner generate a relevant sub-grid activity as shown in fig. 5d. Fig. 5e shows that across the porous region , after an initial rise due to flow straightening, the SGS activity is greatly reduced due to the damping effect induced by the Darcy's Model.

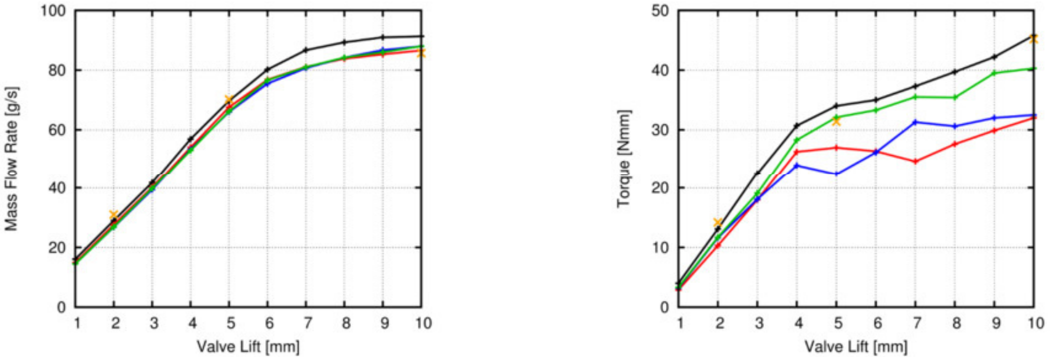
4. Simulation methodologies comparison

Flow bench analysis has dealt primarily with the evaluation of mass flow rates and swirl torque. Both quantities have been evaluated, accordingly to the methodology proposed by Catellani [19], through a cut plane (fig. 5a) perpendicular to cylinder axis and located 5 mm upstream the porous region to avoid possible disturbances in the sampling. For the steady state cases, since some runs have exhibited a mass flow rate slightly oscillating about a mean value, the average over the last 8000 iterations has been taken. For the LES cases, since dealing with transient simulations, proper time averaging has been performed.

In fig. 6 the values of mass flow rates (fig. 6a) and swirl torque (fig. 6b) obtained using OpenFOAM® with different turbulence models, namely eddy-viscosity RANS (EV-RANS), Reynolds Stress Tensor Model (RSTM-RANS) and Large Eddy Simulation (LES), and the one obtained with AVL Fire 2010 using an eddy-viscosity RANS model as presented in [12] are reported with the experimental results. Comparing the OpenFOAM® and AVL Fire 2010 EV-RANS results, is evident how the two codes perform in a quite similar fashion in terms of predictive capabilities.

The mass flow rate predictions depicted in fig. 6a demonstrate that all the simulation methodologies have comparable accuracy and are in good agreement with the experimental data, reporting a maximum relative error of 10% at lift 1 mm. The torque results obtained by the two codes using the EV-RANS models show a trend (fig. 6b) that is roughly similar to the experimental one, increasing with the lift even if not monotonically, but with a quite poor value prediction. The mismatch with experimental data grows with the valve lift with a maximum relative error of about 30%. Conversely, the LES results in terms of swirl are in decent agreement with the experiments and definitely outperform the EV-RANS results, but their cost is not negligible for a routine industrial usage. A potential improvement over the standard EV-RANS modeling, with a small increment in computational cost, could be obtained adopting a Reynolds Stress Tensor Model that has intrinsic capabilities to resolve flow anisotropic features and curvature effects typical of the vortical structures that characterize this kind of devices.

In order to assess the effectiveness of such approach, the cases have been run using the Launder-Rodi-Reece (LRR) RSTM model, already available in the standard libraries of OpenFOAM®. All the remaining setup and boundary conditions have been the same as the ones adopted for the EV-RANS. As expected, the results have shown a definitely better agreement with experiments for what concerns the swirl torques (fig. 6b). The deviations are contained in a 10÷15% range, with a maximum relative error of 17.5% for the 1 mm valve lift.



a) Mass flow rate b) Swirl torque

Fig. 6: Comparison between experimental and numerical results as a function of valve lift: experiments (black), AVL Fire 2010 RANS (red), OpenFOAM® EV-RANS (blue), OpenFOAM®RSTM EV-RANS (green), OpenFOAM® LES (orange).

In order to shed some light on the accuracy deficiencies of the swirl torques predicted by the EV-RANS simulations, the flow field for the 5 mm valve lift has been compared to the ones obtained by the RSTM-RANS and LES simulations. Fig. 7 depicts the streamlines on planes located at 0.8 (TOP row, fig. 7a, 7b, 7c), 1.2 (MID row, fig. 7d, 7e, 7f) and 1.6 (BOT row, fig. 7g, 7h, 7i) bores from the head. The TOP slice presents two distinct counter rotating vortexes of similar extent promoted by the ducts arrangement. Proceeding to the mid-section (MID), while in LES results only the main clockwise rotating vortex is present (fig. 7f), in EV-RANS results the secondary counter-rotating vortex still exists (fig. 7d). Such vortex, despite its small size and its weakness, strongly affects the development of the main swirling structure, which is still squished in nearly half of the cylinder section. The bottom plane (BOT) reveals a well-shaped and centered main vortex for LES (fig. 7i), while, for RANS the main vortex remains significantly off-axis (fig. 7d). Moreover, for the latter case it can be speculatively presumed that the prolonged interaction of the two counter-rotating vortexes has dissipated a greater fraction of the swirling strength of the main vortex respect to the LES case.

The RSTM-RANS simulation results are much closer to LES at least at the MID and BOT plane confirming the increased predictive accuracy.

4.1. Computational cost

A final note relates to the computational cost of the analyses performed in this work. Obviously, the steady-state RANS analyses are much cheaper than the LES simulations. To give a rough idea of this, the simulation cost for a single valve lift can be estimated to be:

- EV-RANS: 1 day on a 2 x Intel Xeon E5-2609 @ 2.40GHz workstation (8 cores total), Approximately 3 GB RAM needed.
- RSTM-RANS: 1.5 days on a 2 x Intel Xeon E5-2609 @ 2.40GHz workstation (8 cores total). Approximately 3.5 GB RAM needed.
- LES: 35 days on a 4 x AMD Opteron 6212 @ 2.60GHz blade (32 cores total). Approximately 15 GB RAM needed.

5. Conclusions

The present paper has presented an application of the LES simulation methodology to a steady flow bench for engine head performance evaluation. The goal of the study has been twofold: to gain better insights of the system fluid dynamics and to investigate the potential causes of predictive deficiencies emerged when a standard RANS simulation approach is used.

The conclusions that can be drawn from this study are summarized as follows:

- The common eddy-viscosity RANS methodology applied to steady flow bench simulation could sometimes return significant errors in the swirl torque predictions. Such poor accuracy, not uncommon in the scientific literature, could compromise development and optimization processes of engine heads and air induction systems.

- The LES methodology applied to steady flow benches allows the accurate prediction of the swirl torque, so that the virtual flow bench can be used with the same capabilities of the physical one.
- The LES methodology also allows the accurate resolution of the unsteady behavior that characterizes these devices that is not appraised using the physical bench. Without claiming to have comparable fidelity to experiments, this methodology can be used in substitution of PIV or LDV measurements when these are not available or as a valuable integration capable of providing full 3D flow fields of the device under investigation.
- Discrepancies between experiments and LES results for both mass flow rate and swirl torque remain still sensible at low lift. Possible improvements could be obtained by grid refinements in the valves region, where some resolution deficiencies have been highlighted by the quality assessment. However, it must be noted that while it is always advisable to reach the higher (reasonable) accuracy, from an industrial perspective, the results accuracy related to low valve lifts has the least importance for Swirl Ratio evaluation purposes. In fact, these lifts are characterized by low flow rates and provide therefore marginal contributions to the in-cylinder angular momentum during intake stroke.
- A viable solution to overcome these predictive issues while still adopting an inexpensive steady-state RANS modeling approach is the choice of turbulence models sensible to streamline curvature, such as a Reynolds Stress Transport Model (RSTM).

References

[1] Heywood JB Internal Combustion Engine Fundamentals. Mcgraw-Hill; 1989

[2] Forte C, Bianchi GM, Corti E Multicycle simulation of the mixture formation process of a PFI gasoline engine. SAE Technical Papers 2011-01-2463 (2011);doi:104271/2011-01-2463

[3] Costa M, Bianchi GM, Forte C, Cazzoli G A numerical methodology for the multi-objective optimization of the DI diesel engine combustion. Energy Procedia 2014;45:711–720 doi:101016/jegypro201401076

[4] Bianchi GM, Cazzoli G, Forte C, Costa M, Oliva M Development of a emission compliant, high efficiency, two-valve DI diesel engine for off-road application. Energy Procedia 2014;45:1007–1016 doi:101016/jegypro201401106

[5] 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. Proceedings of the Spring Technical Conference of the ASME Internal Combustion Engine Division 2012, p 715–725 doi:101115/ICES2012-81095

[6] 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 Technical Papers 2012-32-0096 (2012)b;doi:104271/2012-32-0096

[7] Grimaldi C, Battistoni M, Mariani F Experimental and numerical analysis of charge motion characteristics depending on intake valves actuation strategies. SAE Technical Papers 2005-01-0242 (2005);doi:104271/2005-01-0242

[8] Port development - flow test bench. Tech Rep; AVL List GmbH; 1995

[9] Steady state flowbench port performance measurement and analysis techniques. Tech Rep DP93/0704; Ricardo 1993

[10] Li Y A new estimation of swirl ratio from steady flow rig testing. SAE Technical Papers 2014-01-2587 (2014);doi:104271/2014-01-2587

[11] Xu H Some critical technical issues on the steady flow testing of cylinder heads. SAE Technical Paper 2001-01-1308 (2001);doi:104271/2001-01-1308

[12] Forte C, Catellani C, Cazzoli G, Bianchi GM, Falfari S, Brusiani F, et al Numerical evaluation of the applicability of steady test bench swirl ratios to diesel engine dynamic conditions. Energy Procedia 2015;81:732–741 doi:101016/jegypro201512079

[13] Yang X, Chen Z, Kuo T Pitfalls for accurate steady-state port flow simulations. Journal of Engineering for Gas Turbines and Power 2013;135(6):061601–061601

[14] Palumbo MF CFD-analysis of intake-system performances of a small turbocharged spark-ignition engine SAE Technical Paper 2007-24-0047 (2007);doi:104271/2007-24-0047

[15] OpenFOAM-2.3.0 Programmer's Guide OpenFOAM Foundation; 2014

[16] cfmesh: A library for automatic mesh generation version 11; 2015

[17] Jasak H, Weller H, Gosman A High resolution NVD differencing scheme for arbitrarily unstructured meshes. Int. J. Numer. Meth. Fluids 1999; 31:431-449 doi:10.1002/(SICI)1097-0363(19990930)31:2<431::AID-FLD884>3.0.CO;2-T

[18] Catellani C, Bianchi G, Falfari S, Cazzoli G, Forte C Assessment of advanced SGS models for LES analysis of ICE wall-bounded flows - part I: Basic test case. SAE International Journal of Engines 2016;9(1):657–673 doi:104271/2016-01-9041

[19] Catellani C Development and assessment of large eddy simulation methodology for internal combustion engines. Alma Mater Studiorum Università di Bologna Dottorato di ricerca in Meccanica e scienze avanzate dell'ingegneria; 2016

[20] Pope S Ten questions concerning the large-eddy simulation of turbulent flows. New J. Phys., 2004, 6, 35 doi: 10.1088/1367-2630/6/1/035

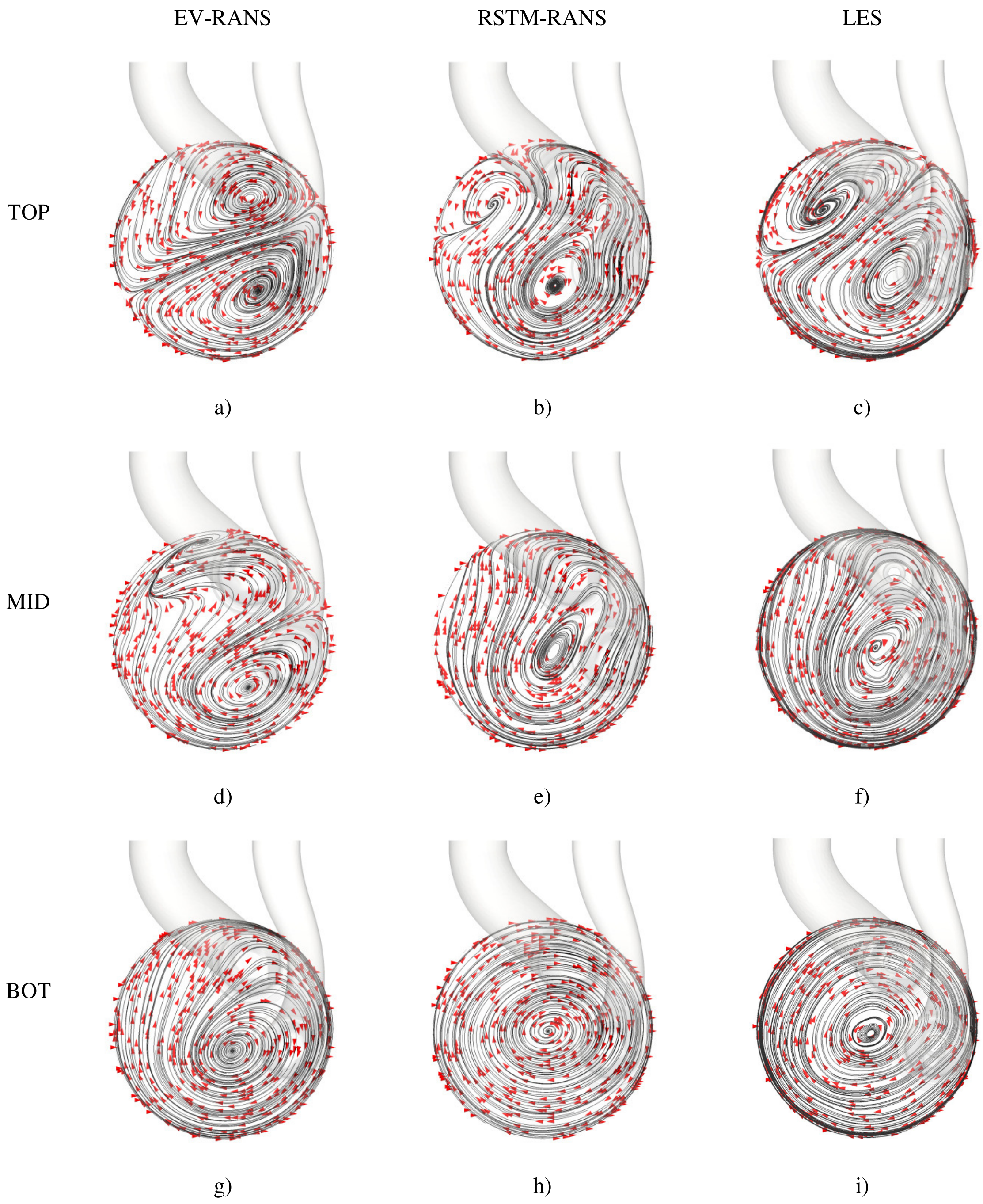


Fig. 7. Valve lift 5 mm. Planar streamlines. Left column: Eddy-vicosity RANS, middle column: RSTM RANS, right column: LES. First row: TOP plane, second row: MID plane, third row: BOT plane