Journal of KONES Powertrain and Transport, Vol. 25, No. 4 2018

ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/01.3001.0012.4786

CFD SIMULATIONS AS A SUPPORT OF EXPERIMENTAL RESEARCH IN A RAPID COMPRESSION EXPANSION MACHINE FACILITY

Agnieszka Jach, Rafał Pyszczek, Łukasz Jan Kapusta, Andrzej Teodorczyk

Warsaw University of Technology, Institute of Heat Engineering
Nowowiejska Street 21/25, 00-665 Warsaw, Poland
tel.: +48 22 234 52 66, +48 22 234 52 41, +48 22 234 52 26
e-mail: ajach@itc.pw.edu.pl, rafal.pyszczek@itc.pw.edu.pl
lukasz.kapusta@itc.pw.edu.pl, ateod@itc.pw.edu.pl

Abstract

The main aim of this study to reproduce methane combustion experiment conducted in a rapid compression-expansion machine using AVL FIRE^{TM} software in order to shed more light on the in-cylinder processes. The piston movement profile, initial and boundary conditions as well as the geometry of the combustion chamber with a prechamber were the same as in the experiment. Authors by means of numerical simulations attempted to reproduce pressure profile from the experiment. As the first step, dead volume was tuned to match pressures for a non-combustion (air-only) case. Obtained pressure profile in air compression simulations was slightly wider (prolonged occurrence of high pressure) than in the experiment, what at this stage was assumed to have negligible significance. The next step after adjusting dead volume included combustion simulations. In the real test facility, the process of filling the combustion chamber with air-fuel mixture takes 15 s. In order to shorten computational time first combustion simulations were started after the chamber is already filled assuming uniform mixture. These simulations resulted in more than two times higher maximum pressure than recorded in experiments. It was concluded that turbulence decays quickly after filling process, what was also confirmed by next combustion simulations preceded by the filling process. Then the maximum pressure was significantly decreased but still it was higher than in the experiments. Based on the obtained results it was assumed that the discrepancy noticed in air cases is further increased when combustion is included. Moreover, the obtained results indicated that pre-combustion turbulence level is very low and suggested that either piston profile movement is not correct or there is high-pressure leak in the test facility.

Keywords: *AVL FIRE^{TM}, self-ignition, RCEM, CFD, knocking combustion*

1. Introduction

Computational fluid dynamics (CFD) and structural analysis software is widely used by engineering companies in the design, optimization and development processes of various industrial systems. Various models (fluid flow, turbulence, combustion, heat transfer) can be tuned and calibrated based on tests done on existing physical prototypes leading to simulations that are more accurate. By means of computations, a huge amount of time and money is saved and a need for building next prototypes is less frequent.

Currently, CFD simulations are an invaluable support of experimental researches and stands, especially those under development. They help to better understand given phenomena, boundary (BC) and initial conditions (IC), uncertainty and how these parameters influence an outcome and conclusions from the test. This article is devoted to reproduction of the methane combustion process-taking place in a rapid compression-expansion machine (RCEM) in order to shed more light on in-cylinder processes – combustion dynamics and knocking combustions.

Knocking combustion limits efficiency and output of reciprocating engines. It determines their durability, noise and emission performance [1-3]. Most widely, the research octane number (RON) and the motor octane number (MON) are used for ranking the fuel resistance to knocking combustion. RON and MON are tested in standardized conditions [4, 5]. However, they are not sufficient to describe the knock resistance of various fuels in modern engines, which are often highly boosted and cooled. This creates a need for tailored tests in specific conditions.

An RCEM is a test facility imitating compression and expansion strokes of a piston engine. It is used for detailed investigations in spray development and combustion processes. A driving concept in an RCEM used in this study is different from in an engine. A piston is hydraulically driven and it is not mechanically coupled with any crankshaft. The driving concept is based on two cylindrical and concentric driving pistons moving in the opposite direction. This helps in minimizing vibrations and improves optical measurements. A detailed description of the machine and experiment performance is given by Eisen [6] and Pöschl [7].

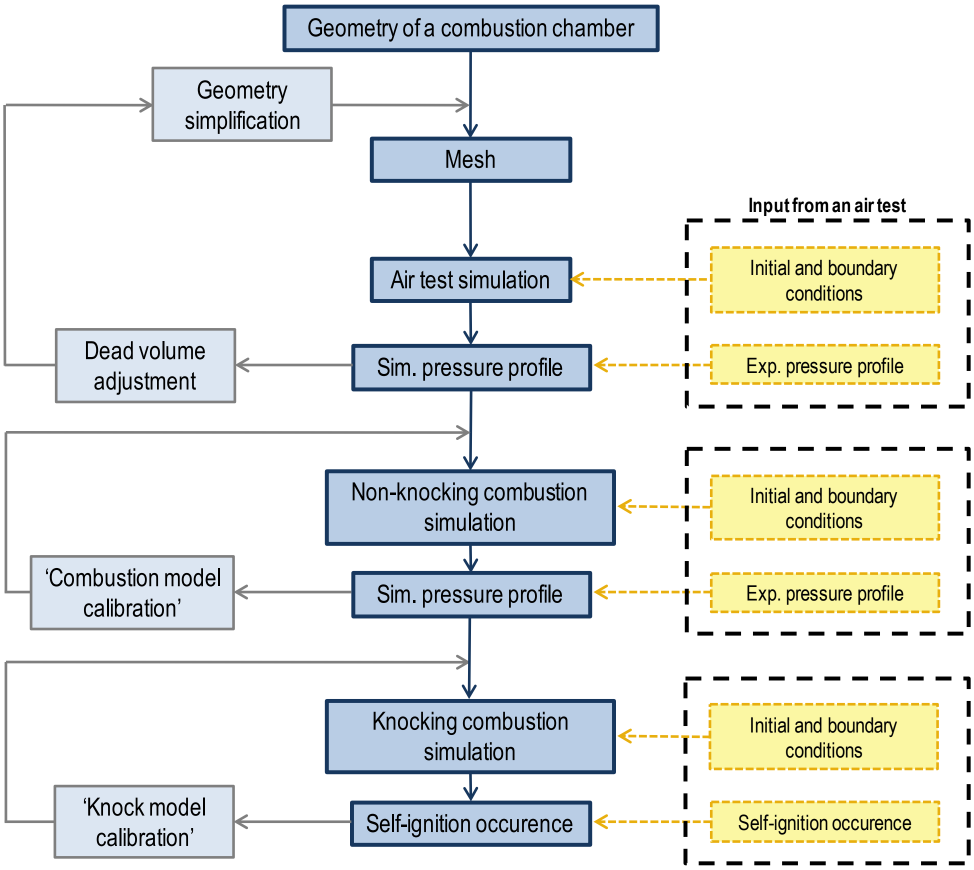

Fig. 1. Workflow of CFD simulations

2. Workflow

The article is devoted to combustion of methane. The aim is to compare experimental and simulated pressure profiles and learn more about conditions in the RCEM. The experiment is performed for lean mixture (air excess ratio 1.2) and 5 kJ of energy closed in the cylinder, what corresponds to initial pressure of 0.23 MPa. CFD simulations are performed in the AVL FIRE^{TM} software and following models are used: turbulence model k-ζ-f, Standard Wall Function, combustion model ECFM-3Z [8]. Simulations are divided into several steps according to the workflow presented in Fig. 1. Firstly, a mesh is created and a dead volume is adjusted base on air tests pressure profiles. Then, combustion simulations are performed preceded with a filling of the combustion chamber.

2.1. Geometry, mesh and air test simulation

The RCEM has a combustion chamber with a prechamber. The piston bore is 85 mm and the maximum stroke is 180 mm. The investigated geometry is presented in Fig. 2 a. It is simplified in air test and combustion simulations – pipes used for charging are removed and the corresponding volume is located under the piston as shown in Fig. 2 c. Fig. 2 b presents the mesh at the top dead centre (TDC) without any dead volume. A moving mesh is used with variable number of elements – from ~370k at TDC up to ~850k at BDC (bottom dead centre).

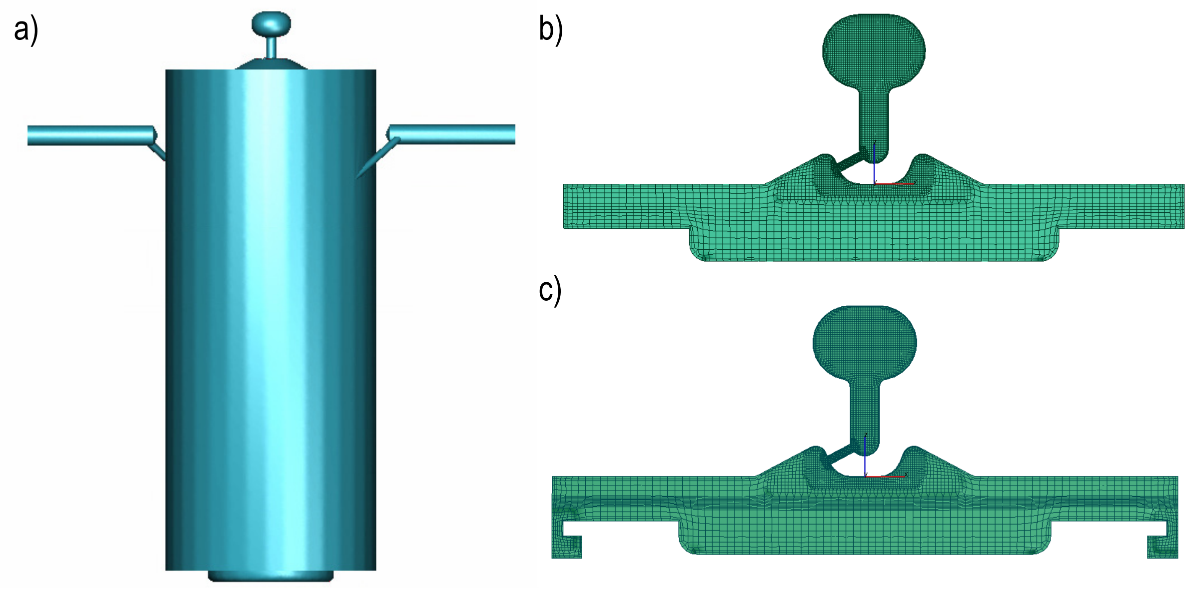

Fig. 2. A model and meshes of the combustion chamber

The initial and boundary temperature is 358 K. The movement of the piston is case-related and differs between simulations. Firstly, an air test simulation is run with the dead volume equal to the real one measured on the test bed (corresponding to 20 cm of 6 mm diameter pipe). The dead volume is closed when the piston passes the channels (Fig. 2 b). The obtained pressure is shown in Fig. 3 left. A peak pressure in simulation is 5 bar higher than in the experiment. Therefore, the dead volume is gradually increased until the point when pressure peaks match. This last case is presented in Fig. 3 right. The simulated pressure profile is wider than the experimental one. Simulations indicate a 3.5 times bigger dead volume than initially assumed.

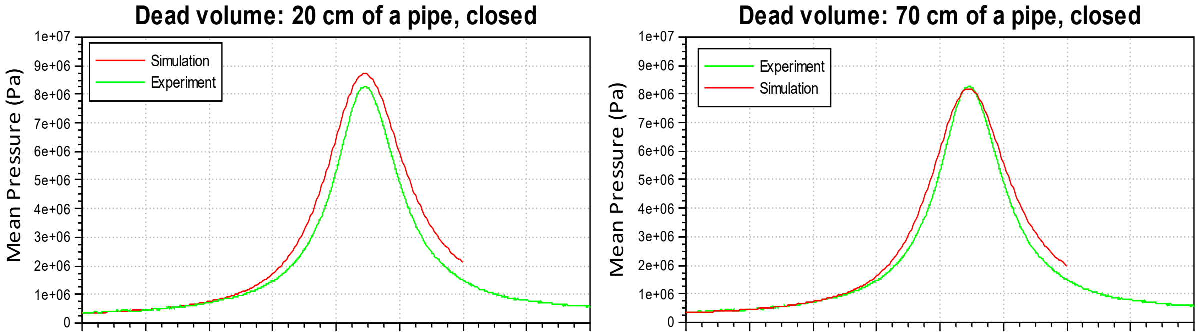

Fig. 3. Pressure profiles [Pa]. Left: a real dead volume. Right: a dead volume 3.5 times bigger

2.2. Filling process

First combustion case simulations performed with a default initial turbulence level showed that the computed peak pressure exceeds the experimental one over two times (30.5 MPa against 13.6 MPa). It was concluded that the level of turbulence is not correct and combustion is too *fast*. Combustion experiments start from filling the combustion chamber and the whole process takes 45 s until the start of the piston movement. Partially premixed fuel/air mixture is provided to the chamber through two pipes circumferentially located on the cylinder liner ca 5 cm from the TDC. The first stream consisting of premixed fuel and part of air is added slowly. Then the second stream consisting of air is added faster to improve the mixing. The simulation starts at time 0 (Fig. 4) with a homogenous distribution of the first stream. The experimental pressure profile from the filling process is recalculated to mass flows and Mass Flow BCs are used at inlets. The red line in Fig. 4 shows the pressure profile obtained in simulation. It is in very good agreement with the experimental pressure.

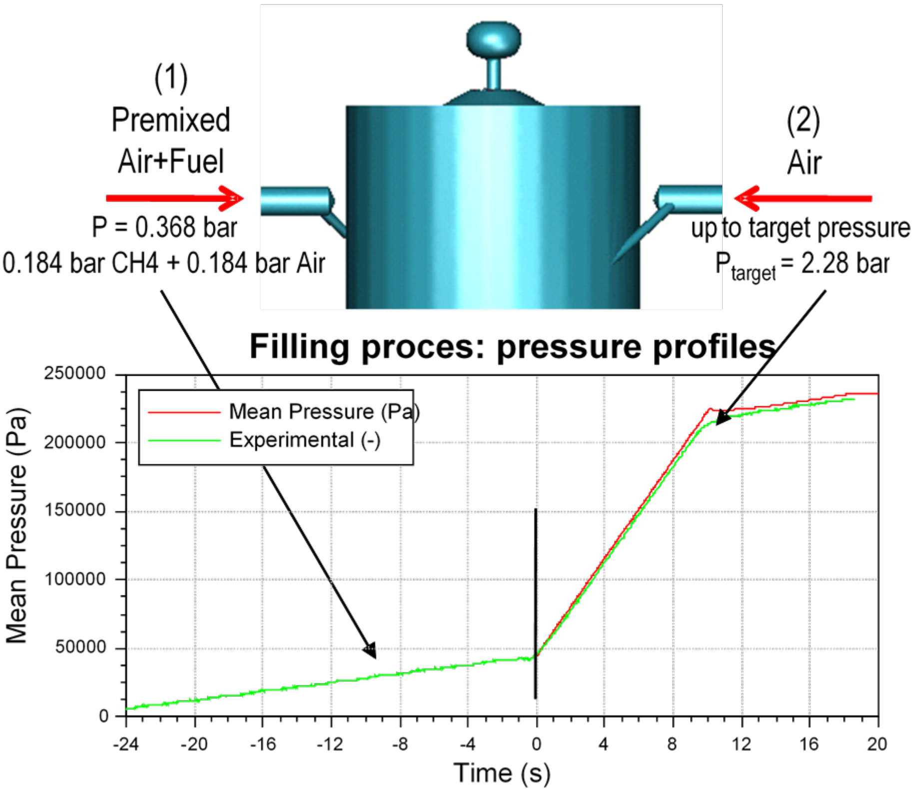

Fig. 4. Filling process of methane case

Figure 5 shows velocity magnitude and distribution of λ at the end of the filling process. The mixture is stratified – λ varies from 0.5 in the prechamber up to 1.29 close to the cylinder head and 1.08 close to the piston. The velocity magnitude is low, 5.5 mm/s, what corresponds to the Reynolds number ~40 and indicates a laminar flow. Some amount of the fuel is left in the first stream channel and as a results energy in the combustion chamber. It is smaller than target 5 kJ and the average λ is lowers than 1.2.

2.3. Combustion simulations

In the next step, combustion of methane is simulated. The previously adjusted dead volume is used. The initial turbulent kinetic energy and dissipation rate are initialized from the end of the filling process. It is assumed that the inhomogeneity in distribution of λ has negligible influence on the combustion process and a uniform mixture with λ=1.2 is initialized.

Example results – the flame propagation represented by an iso-surface at 1400 K is shown in Fig. 6. Fig. 7 left presents the pressure profile obtained in simulations. The calculated peak pressure is smaller than previous 30.5 MPa, but still 2.5 MPa higher than in the experiment. In Fig. 7 right results of further adjustments of the dead volume is shown. The closest case is for the initially planned dead volume (20 cm of 6 mm diameter pipe) which is open during a whole process.

Nevertheless, the simulated pressure is *wider* than the real one (red solid line).

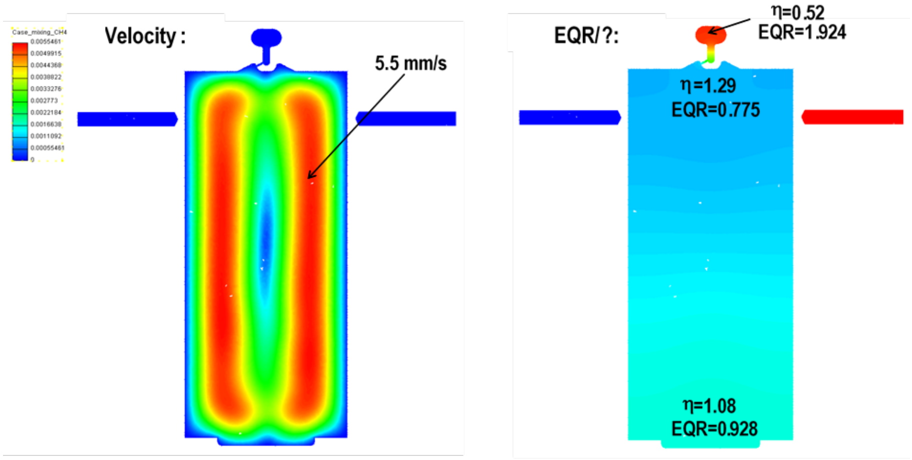

Fig. 5. Velocity magnitude and distribution of equivalence ratio (EQR) and air excess ratio λ at the end of filling process

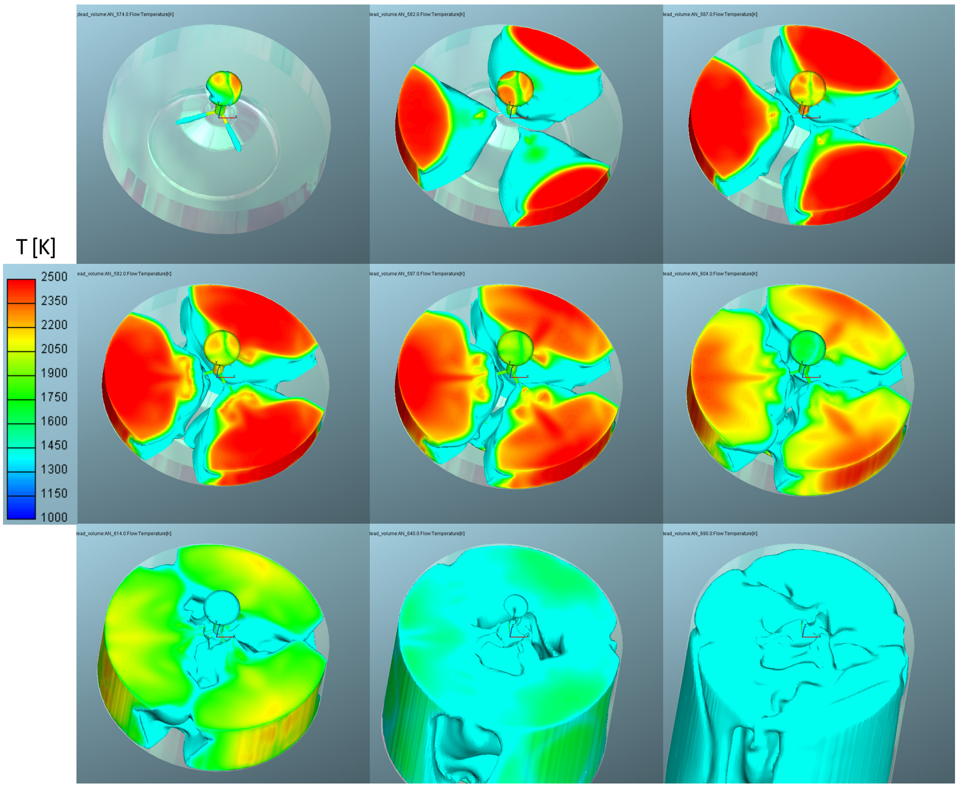

Fig. 6. Flame propagation in the investigated methane case; flame is represented by an iso-surface at 1400 K

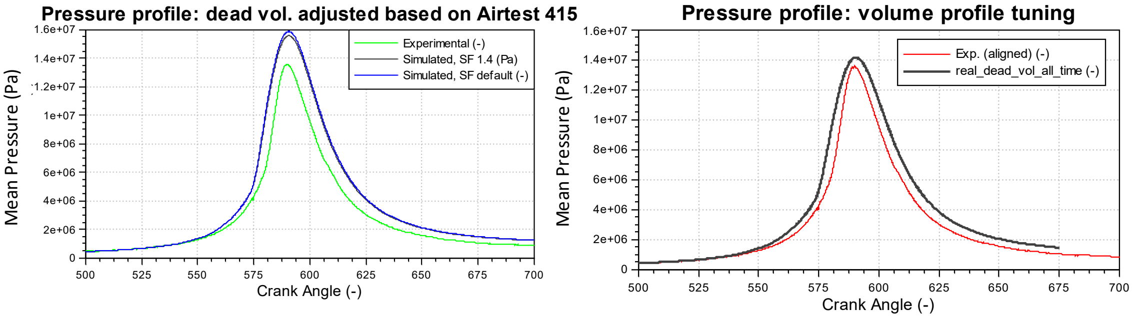

Fig. 7. Pressure profiles obtained in simulation of methane case; left: results for a mesh with a dead volume adjusted base on air test; right: results for a further adjusted dead volume

2.4. Further air test and combustion simulations

The discrepancy noticed in air cases is further increased when combustion is included. It was decided to come back to air test simulations to get a better match of the simulated and experimental pressure profiles. The volume profile constricted to 85% (Fig. 8 right) gives a reasonable match (Fig. 8 left). However, the same correction for the volume profile (Fig. 9) has not improved the match in the combustion case. It means that the correction is case-sensitive. Due to a poor match of pressures simulations of the knocking case are not performed.

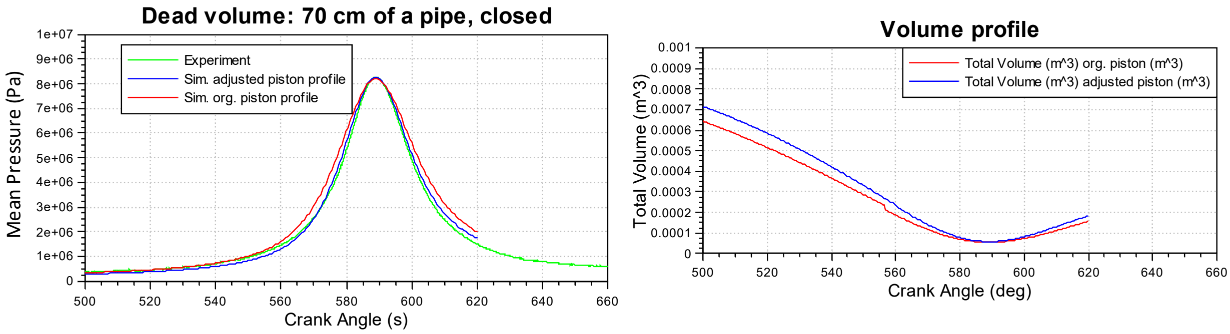

Fig. 8. Left: pressure profiles for air test with adjusted dead volume or adjusted piston profile. Right: volume profile – original from the experiment and adjusted with kf=0.85

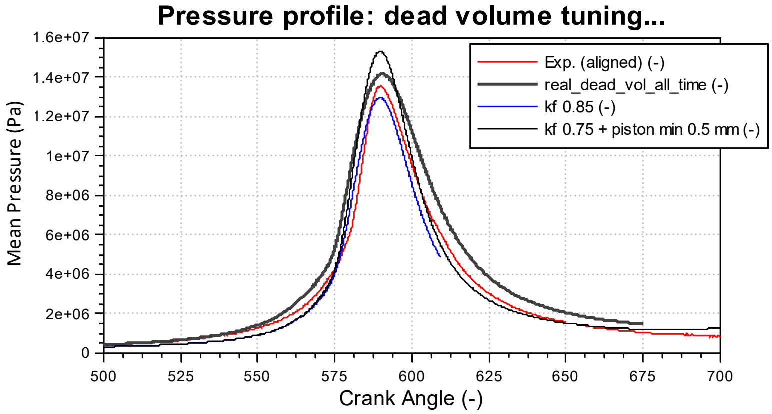

Fig. 9. Pressure profiles from the experiment and simulations with tuned volume profile (kf informs on how much a volume profile is constricted)

3. Conclusions

Conclusions from the performed simulations are meaningful. To match peak pressures in the air case, a 3.5 time bigger dead volume is needed than the dead volume measured on the RCEM. The analysis of the filling process showed that the mixture is stratified – λ varies from 0.5 up to 1.3. A part of the fuel is left in the channel what results in a lower energy closed in the cylinder than the target 5 kJ and lower average λ than planned (1.2). The velocity magnitude and the turbulence level at the end of the filling process are low. The Reynolds number is ~40 and the flow in the cylinder is laminar. Simulations of the methane case with settings resulting from previous simulations do not give a reasonable match of pressure profiles. Further adjustment of the dead volume gives a better match of peak pressures, but the profile is *wider* than in the experiment. Next simulations with the tuned volume profile show that all corrections are case-related. Either the piston profile movement is not correct or there is a high-pressure leak in the test facility. Simulations of knocking cases are not performed due to poor match of experimental and simulated pressures. History of the pressure and temperature determines the probability and strength of knocking combustion. To be able to predict knock in the future the non-knocking combustion simulation have to match very well with its experimental equivalents.

Acknowledgments

The AVL FIRE^{TM} calculation code was used as per the AVL AST University Partnership Program.

The project leading to this application has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement No. 691232.

References

[1] Wang, Y., Liu, H., Reitz, R. D., *Knocking combustion in spark-ignition engines*, Progress in Energy and Combustion Science, Vol. 61, pp. 78-112, 2017.

[2] Ceschini, L., Morri, A., Balducci, E., Cavina, N., Rojo, N., Calogero, L., Poggio, L., *Experimental observations of engine piston damage induced by knocking combustion,* Materials & Design, Vol. 114, pp. 312-325, 2017.

[3] Karvountzis-Kontakiotis, A., Vafamehr, H., Cairns, A., Peckham, M., *Study on pollutants formation under knocking combustion conditions using an optical single cylinder SI research engine*, Energy, Vol. 158, pp. 899-910, 2018.

[4] ASTM, *Standard test method for research octane number of spark-ignition engine fuel*, ASTM D2699-16, 2016.

[5] ASTM, *Standard test method for motor octane number of spark-ignition engine fuel*, ASTM D2700-16a, 2016.

[6] Eisen, S., *Visualisierung der dieselmotorischen Verbrennung in einer schnellen Kompressionsmaschine*, Lehrstuhl A für Thermodynamik, Technische Universität München, Germany 2003.

[7] Pöschl, M., Sattelmayer, T., *Influence of temperature inhomogeneities on knocking combustion,* Combustion and Flame, Vol. 153, Iss. 4, pp. 562-573, 2008.

[8] AVL FIRE^{TM} Version 2014.2 – The FIRE Combustion Module Manual, 2014.

Manuscript received 17 July 2018; approved for printing 21 September 2018