MARINE DIESEL ENGINE COMBUSTION INFLUENCED BY INJECTION NOZZLE AND PRIMARY FUEL ATOMIZATION

Tadeusz Borkowski

Maritime University
Waly Chrobrego Street ½, 70-500 Szczecin, Poland
tel: +48 91 4809400, fax: 48 91 4809575
e-mail: tborkowski@am.szczecin.pl

Abstract

In recent years modelling has been applied to investigation of mixture formation, fluid flow, combustion process and pollutant formation in diesel engines. In this study numerical simulation was carried out to investigate the effect of fuel spray characteristics in marine medium speed diesel engine – Sulzer 6A20/24D. 1-D and 3-D CFD fuel spray model was adopted for simulation of atomization and combustion process. For comparison purposes modelling outcome was set against experimental tests results. Operational validation of injection assembly model was performed according to standard marine engine test cycles. Injection pressure history for full load range of the engine has been experimentally established.

Fuel oil supply system, test engine cylinder experimental setup, 1-D engine fuel injection assembly model and geometry of cam profile, measured nominal injection duration, comparison of calculated and measured injection pressures, calculated nominal injection process components shape, the cross section of velocity flow field for nominal injection phase and for nominal injection phase, the cross sections of velocity flow field in hole channel for nominal injection phase after cut-off and for nominal injection phase after cut-of, the cavitation-induced spray formation process in modelled combustion chamber are presented in the paper.

Keywords: *marine diesel engine, injection assembly, combustion and injection modelling*

Introduction

A decade ago, the marine industry paid little attention to air pollution. That has changed when the amended IMO exhaust emission regulations were established for all engines installed on ships and all engine units that undergo major conversion. Customarily, before the IMO statement is issued, the ship-owner needs to provide a "NO_x Technical File" which should include, among others, injector and injection pump details. In view of evolving engine management systems, alteration of injection profile offers the flexibility of changing the exhaust emission characteristics during ship operation. This may prove helpful for vessels entering area where emission will be restricted by new legislation in the future. The injection nozzle operation largely controls the combustion process, and set of related issues, such as engine performance - specifically fuel efficiency and emission. This complex problem concerns a specific engine subsystem which can be modelled in computer simulations. Recently, simulation software have been developed that combines 1-D modules for hydraulic subsystem and 3-D CFD code for combustion and injector nozzle channel modelling. This paper focuses on the modelling of a research marine diesel engine fuel injection system and related combustion process. For that purpose a hydraulic model of the fuel injection system has been developed by means of Power GT-fuel software. The flow modelling utilizes one-dimensional flow elements to model piping and quasi-3D elements for arbitrarily-shaped volumes of the complete in-line pump and injector. The dependability of the numerical results was tested through a comparison with experimental tests on standard diesel fuel oil. Basis for this detailed comparison were the measurement results of fuel injection pressure in the injection pipe and in-cylinder combustion pressure for a single engine unit. The presented investigation combines a 1-D simulation tool for modelling of hydraulic and gas dynamics systems, with 3-D CFD code for modelling the in-cylinder combustion and emission. The 3-D CFD fuel nozzle flow completed with spray process has been performed by Vectis – Ricardo code. The goal of the study was to validate the hydraulic, one-dimensional, model of an in-line pump type injection system. The model can be further developed for standard diesel oil to analyze differences in the fuel injection spray behaviour related to differences in discharge coefficients and fluid properties. It is a common practice in spray modelling to specify few particular injection parameters at the nozzle exit and initial droplet distribution. The injection rate trough the injector nozzle flows can be either estimated or measured [1]. The velocity matrix can be created using the steady state flow through the nozzle, whereas droplet distribution data is typically obtained from experiments. In the study, parametric studies were carried out with varying nozzle pressure (in-sac) and exit pressure, defined as a boundary cylinder pressure. Both influences were examined in accordance with experimental engine load condition. The results of steady-state simulations performed were discussed.

1. Experiment details

The experimental efforts described below were an attempt to validate 1-D model of the in-line injection assembly and to perform associated CFD 3-D injection nozzle flow calculation. The test bed engine had the specification listed in Table 1. The fuel oil supply system used in the study consisted of a low-pressure fuel supply pump (engine driven), a cam-driven mechanical in-line high pressure pump and an injector – one per cylinder. The fuel oil supply system layout of the engine is presented in Figure 1.

Tab. 1. Test engine details

Engine type	Sulzer 6A20/24, non-reverse.
Number of cylinders	6
Bore/Stroke [mm]	200/240
Rated engine speed [rpm]	720
Output [kW]	397
Compression ratio	14
Brake mean effective pressure [MPa]	1.47

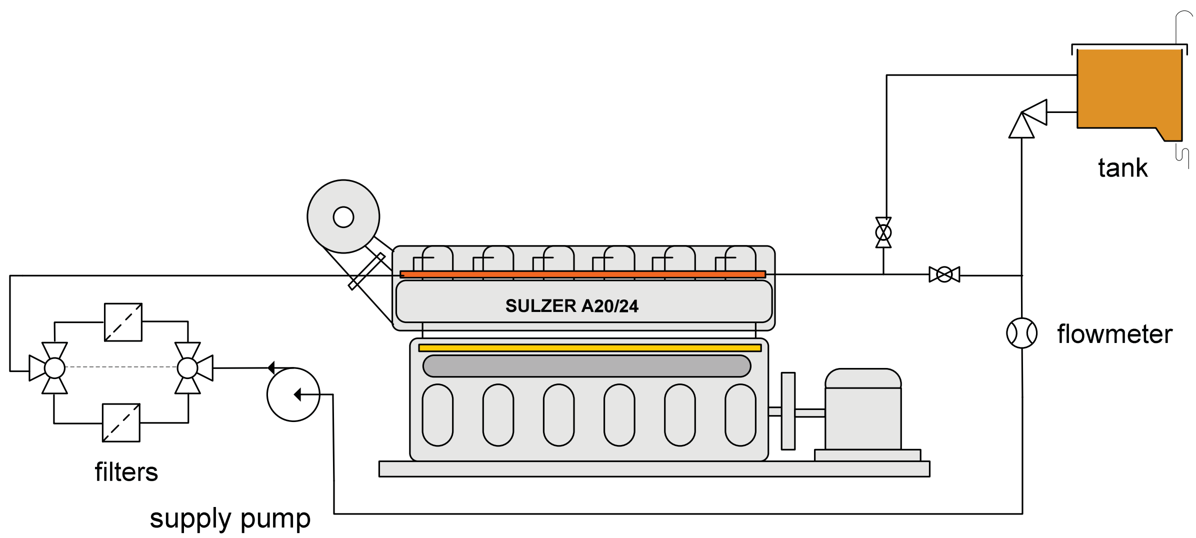

Fig. 1. Fuel oil supply system

The injector was continuously controlled by the pump plunger position and fill-spill port edge. In this experiment, a multiple-orifice mini sac nozzle (SAC) was used. The beginning of injection timing in the experiment was constant by engine design, while injected fuel (end of injection period) volume was controlled by means of the speed governor operation. Relying on factory adjustment characteristics, examination of the injection static settings and all geometrical properties of the fuel injection pump and injector was performed. Basic fuel equipment settings are presented in Table 2.

It is known that the injector nozzle (in-sac) chamber pressure (the space around needle valve seat) varies widely during the injection period, even when the injector valve is open, making the representative value difficult to define [2], [3]. A reasonable choice would involve monitoring the injection pressure that requires a use of only one pressure transducer located in the fuel line near the injector, which is easily set in practice. The parameters of injection characteristic investigated in this study were the injection pressure and injection rate. The combustion pressure as well as injection pressure was continuously monitored and recorded by a fast data acquisition system. Test engine experimental setup is presented in Figure 2, while experimental component details are shown in Table 3.

Tab. 2. Fuel oil injection equipment settings

Fuel pump		Setting
Delivery stroke	[m]	0.0042
Injection start	[deg]	-18.5
Effective stroke	[m]	0.0075
Delivery completion	[deg]	+18
Injection nozzle		
Opening pressure	[MPa]	40.0
Spray angle	[deg]	159
Number of spray holes	[-]	9
Spray hole diameter	[m]	0.00023
Needle lift	[m]	0.0005

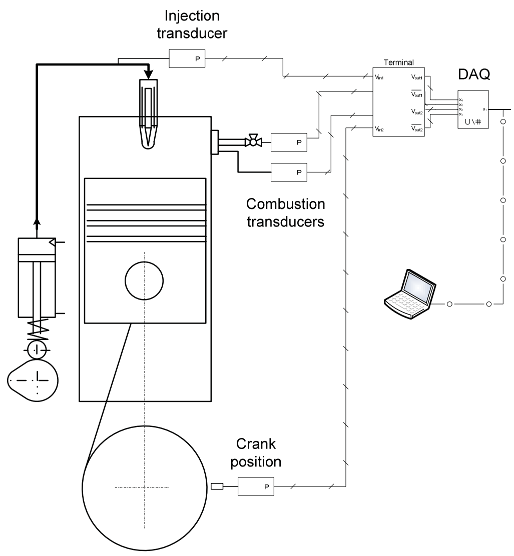

Fig. 2. Test engine cylinder experimental setup

Cylinder pressure measurement within the combustion chamber was uniquely recorded by means of two pressure transducers and was set against data processing system in time domain, which upon need is easily transformed onto the angular domain. All pressure signals were collected synchronously to speed signals. The established way to measure cylinder pressure in marine engine, followed in the experiment, is to use an indicator pipe with indicator cock. This in turn has an influence on the accuracy of the pressure measurement at the location where the sensor is mounted. The basic pressure sensor (Optrand) was mounted on the indicator cock, while piezoelectric sensor assembly inserted in pressure channel close to cylinder head. A LabVIEW Data acquisition system was utilized to operate the system during the trial and to record and analyze the test data.

Tab. 3. Engine trial basic equipment details

Parameter	Transducer - part	Description
Cylinder pressure	Optoelectronic Optrand, F532A8-ACu	Range: 0 – 34 MPa, 0,85 mV/psi @ 200°C
Cylinder pressure	Piezoelectric PCB 12B10	Range: 0 – 20.68 MPa, 0.145 pC/kPa
Injection pressure	Piezoelectric PCB 113A03	Range: 0 – 137.9 MPa, 0.064 pC/kPa
Crank position and speed	Fiber optic Keyence, type FU-85Z	1 pulse/1.0°
Signal	Acquisition board	PCMCIA - AD126, 12 bit, sampling rate 175 kHz

The engine trial was performed to assess engine operational profiles with measurement carried out at steady-state engine operation. All engine parameters were recorded on continuous basis. The engine test adhered to test-cycles procedure D-2 and E-2 (ISO 8178), which include generator and pitch propeller drive. Throughout the test engine was supplied with selected DMX marine distillate fuel in accordance to ISO standard. An evaluation of this fuel is given in Table 4.

Tab. 4. Fuel oil characteristic

Determination			Test results
1	Density @ 15 °C	kg/m^3	853
2	Viscosity @ 40 °C	mm^2/s	5.6
3	Flash point	°C	61
4	Conradson Carbon (CCR)	%	0.003
5	Calorific value	MJ/kg	42.70

2. Calculation method

The first stage of high pressure fuel injection system calculations were performed using GT software module for hydraulic system simulation. The flow modelling utilizes one-dimensional flow elements to model piping and quasi-3D elements for arbitrarily-shaped volumes. The module contains libraries for mechanical and thermal modelling. The code employs a density equation of state model that exhibits real trends of measurable fluid properties such as wave propagation speed. Cavitation is allowed to occur in the piping and other injection part sub-volumes, respectively. The formulation is based on a homogenous equilibrium model. The application of the code to a typical marine fuel injection system is described below. The tool contains higher-level models of injectors and pumps that are assembled from numerous primitive models. These higher-level models account for the complexities of the modelling through encapsulation of particular details. Dedicated models are necessary when the tool is utilized in pump-line and nozzle fuel injection system modelling. For example, one of these models is used to predict the fuel leakage rate draining of the high-pressure pump plunger. The high-pressure pump plunger leakage affects the predicted peak pressure. The leakage model estimates the fluid leakage through the annular gap between piston type parts like plunger or needle guide and bush. Figure 3 shows the created in-line marine cylinder fuel injection assembly model that is assembled from primitive models.

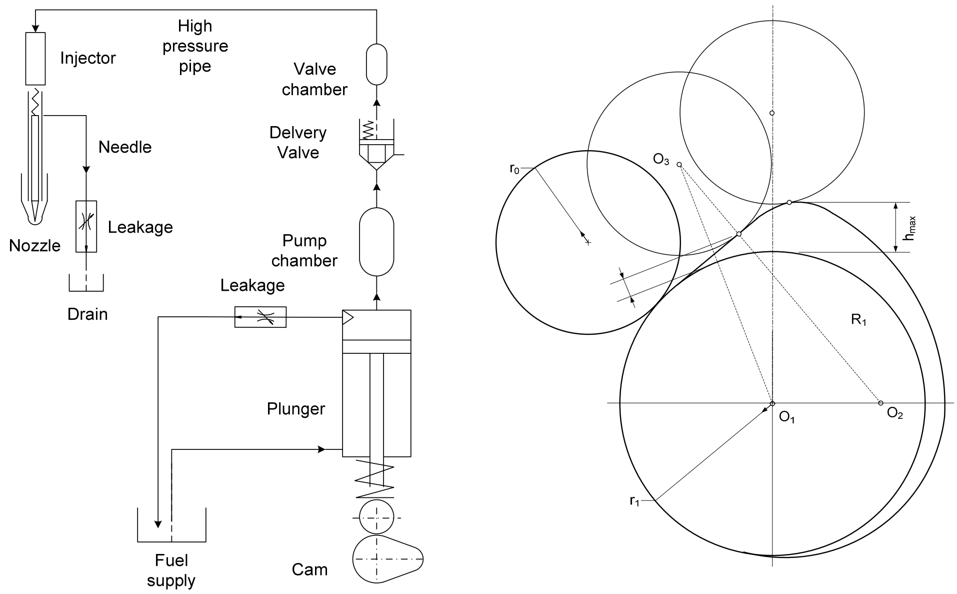

Fig. 3. 1-D engine fuel injection assembly model and geometry of cam profile

The model can be seen as a chain of restrictive and capacitive elements. In the capacitive elements, such as the nozzle chamber and valve chamber, fluid pressure and temperature were calculated. In the resistive elements, such as the valve-throttle, only the mass flow rate was calculated from Bernoulli equation [4]:

$$\dot{m} = \rho \cdot C_d(\lambda) \cdot A \cdot \sqrt{\frac{|2\Delta p|}{\rho}} , \tag{1}$$

where:

p – fluid density,

C_d – discharge coefficient,

A – cross-sectional area.

For non-cavitating flow, C_d is dependent on flow velocity, fluid density and viscosity. This dependency is taken into account by introducing a flow number - λ. To make correction for cavitation the cavitation number is defined as the ratio of the pressure difference over the channel restriction and downstream pressure [4], [5]. If the cavitation number is higher than pre-defined critical cavitation number – $CN_{critical}$, equation (1) changes to:

$$\dot{m} = \rho \cdot C_d^c(\lambda) \cdot A \cdot \sqrt{\frac{|2\Delta p|}{\rho}} \cdot \sqrt{\frac{1+CN}{CN}} , \tag{2}$$

where:

C_d^c - discharge coefficient at the cavitation limit (CN→∞).

Mechanical input cam profile data defines the lift and acceleration of the roller and pump plunger as well. Cam profile data was created for plunger dynamic calculations (Figure 3). Plunger input – output area was calculated from diameter. Furthermore, plunger velocity is subject to flow-rate that will enter continuity equation of the attached chamber volume. All capacitive and restrictive components were connected by high pressure fuel pipes. In them 1-D hydraulic channel pressure wave dynamics was taken into account. Dynamics of motion was evaluated for mass components. Other components in the injection system were modelled in analogous manner. The 3-D CFD nozzle and cylinder flow and combustion calculations were performed with Ricardo-Vectis, complemented with spray and combustion models. The model library includes advanced multi-dimensional models specifically developed for diesel engine simulation. Three-dimensional steady-state simulation method was adopted. The structure of the made flow passage was based on injector nozzle. For calculation convenience the volume mesh was divided according to symmetry feature and ½ of the flow field from one nozzle hole was selected for calculation. Injector nozzle details and part of the mesh structure used in calculation is shown in Figure 4.

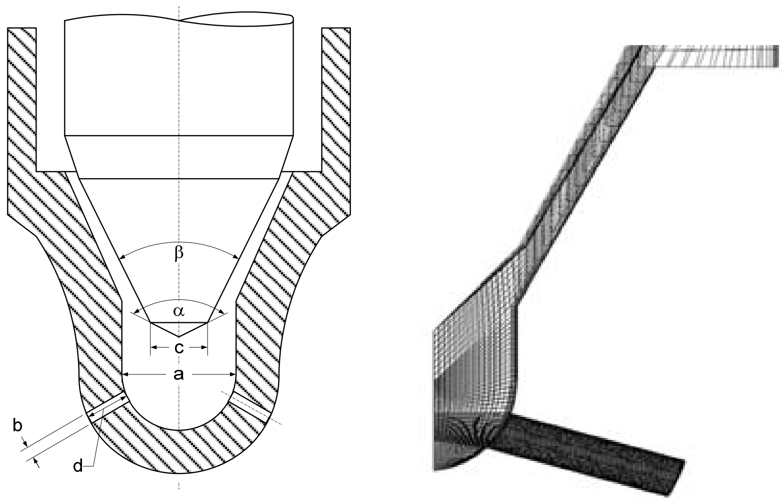

Fig. 4. Injector nozzle and part of the volume mesh adopted for calculation

Nozzle entrance hydraulic pressure and exit combustion pressure were chosen as boundary conditions [6]. The initial conditions at the entrance of the nozzle guide passage resulted from 1-D pressure wave calculation. Nozzle exit pressure conformed to measured cylinder pressure obtained through experimental tests. Simple pressure field and effective algorithm was adopted taking into account turbulent intensity of the flow field in the nozzle and k-ε flow turbulent model.

3. Results and discussion

The injection pressure of a mechanical driven in-line pump system, maintained in engine constant speed drive, mainly depends on injection duration. As shown in Figure 5, the injection pressure reached approximately 104 MPa with nominal injection duration. Measured injection pressure traces have shown a noticeable slip in dynamic operation against measured static adjustments. The injector response time, the time between injection command and start-of-injection (SOI), was in all cases ~ 3.1 deg CA. The injector response time consisted of two time increments. The first time increment is required to lift the needle, while the second one is required for the full needle stroke- to move to the open position. End-of-injection (EOI) is characterized by a longer delay, under nominal load it is ~ 7.7 deg CA. Finally, the controlled injection period equals ~ 26.7 deg CA, while measured dynamic injection period was ~ 31.0 deg CA. For the injection characterization study, the injection pressure was measured at 25 cm upstream of the fuel inlet of the injection nozzle holder. To correlate the upstream pressure and the sac pressure the pressure in the sac chamber of the nozzle tip was calculated using 1-D hydraulic model. The test example case and results are summarized in Figure 5, which shows pressure traces for two of the calculated injector system points – high pressure pipe (at the measurement sensor placement) and nozzle sac chamber.

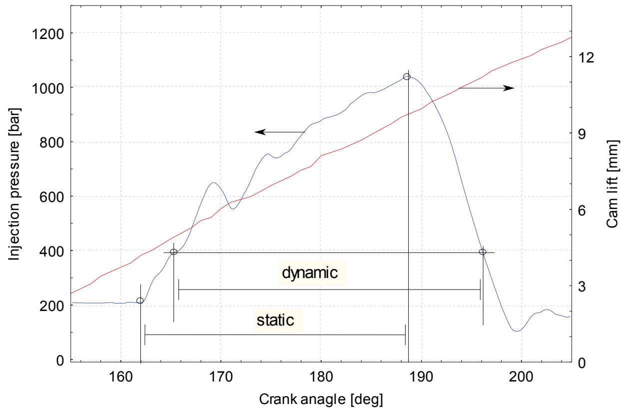

Fig. 5. Measured nominal injection duration qualified for calculation

The calculated pressure dropped between the upstream location and the sac chamber and was in the range of 3 to 9 MPa. It is known [3] that the pressure drop occurred at the needle seat region when high-pressure and high-velocity fuel flowed through it. The calculated injection pressure trends depended on the presence of capacitive elements, such as the nozzle chamber and valve chamber and have shown slightly different increment to experimental data. As shown in Figure 6 calculated start of injection coincided with full open needle position at experimental data. Both pressure traces are similar in terms of specific trends, especially during the needle opening phase. Anyhow, significant difference can be seen after injection cut-off and ~10 deg CA delay period would be encountered as a consequence of delivery valve stringent operation. Also, observed opening delay on calculated pressure traces needs to be investigated. The history of calculated injection pressure in nozzle sac, corrected to experimental time was adopted to create fuel flow rate file with boundary conditions set for 3-D CFD model.

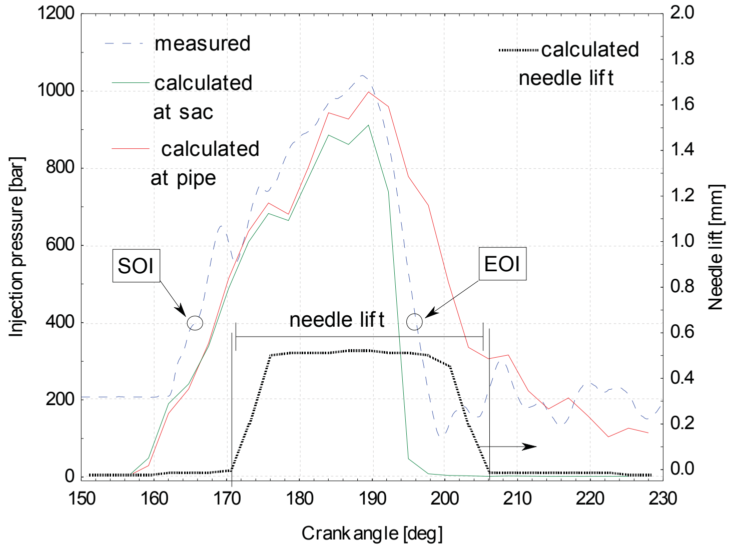

Fig. 6. Comparison of calculated and measured injection pressures

The parameters of spray characterization calculated in this study include penetration, overall structure, initial spray break-up, spray at end of injection, spray at peak-pressure injection, and near-nozzle-exit spray cone angle. For modelling of the injection rate, discharge coefficients of the nozzle holes and injector needle tip passage are the key parameters. The primary values taken for steady-state discharge coefficients of the nozzle were calculated from the following formula:

$$C_d = \frac{Q}{A \cdot \sqrt{\frac{2 \cdot p}{\rho}}}, \tag{3}$$

where:

Q – volume flow rate.

The dynamic discharge coefficients were calculated similarly. However, the calculated upstream pressure and injection rate were used instead of the measured injection pressure and average injection rate. Tuned discharge coefficient for the injector nozzle holes was 0.64, which is a relatively low value. A critical cavitation number of 0.93 has been used. For the length/diameter ratio of the used nozzle hole geometry fully cavitating flows were found. The calculated injection rate depended on the needle lift, the nozzle hole area, and the length of the rate-shaping pipe. Figure 7 provides an overview of basic injection components shape. When the increasing injection pressure was maintained steadily the injection volumetric rate and spray geometry became stable.

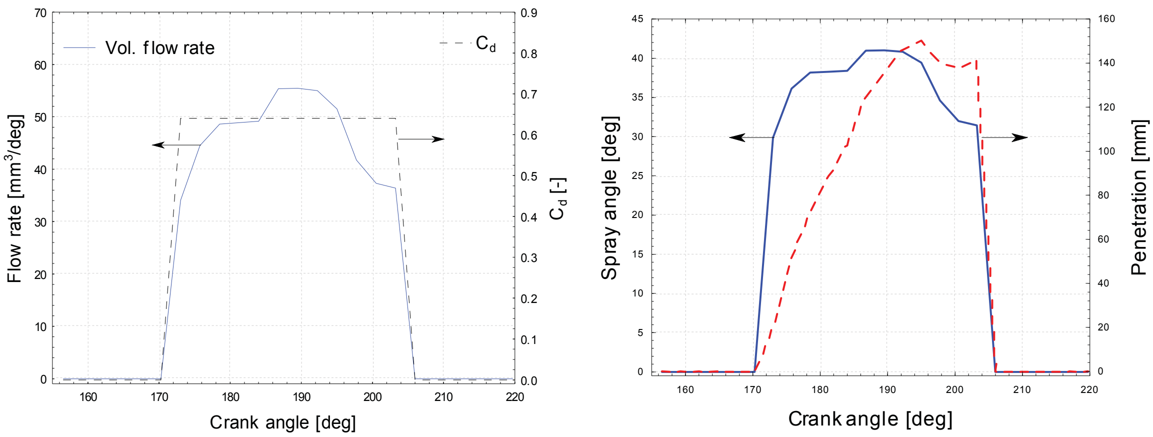

Fig. 7. Calculated nominal injection process components shape

Besides these hydraulic categories, the knowledge of the pressure in the nozzle sac is very important. In most cases the sac pressure close to the injector hole is used to evaluate the injection process. The preliminary 1-D calculations provided set of data containing the fuel pressure field along injector passage and in nozzle sac. Obtained results from 3D CFD calculations proved that needle chamber pressure is approximately constant and the pressure preceding the injection holes corresponds to the injection rate. Figure 8 shows exemplarily the velocity flow field of a needle controlled injection. The two presented cross-sections belong to injection phase at maximum pressure (~ 188 deg CA – left diagram) and after delivery cut-off (~ 202 deg CA – right diagram).

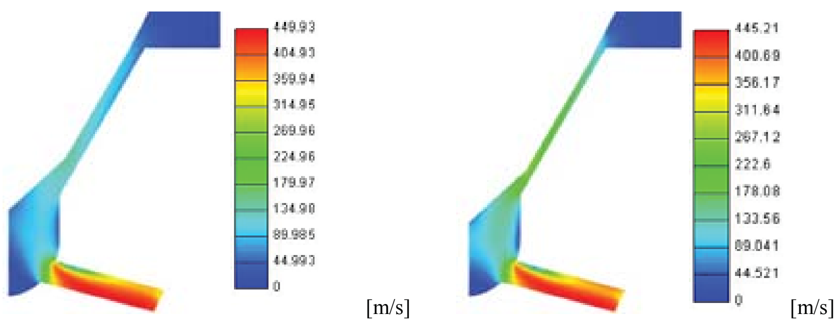

Fig. 8. The cross section of velocity flow field for nominal injection phase

It may be seen in Figure 8 that the speed distribution of flow field in nozzle chamfer volume is quite even at maximum flow rate stage. This weakens as the effect of a cut-off, thus the smaller chamfer volume in nozzle can concurrently keep oil injection amount by increased flow coefficient. Also, the pressure drop of nozzle flow field between needle valve and seat surface is negligible, shown in Figure 9.

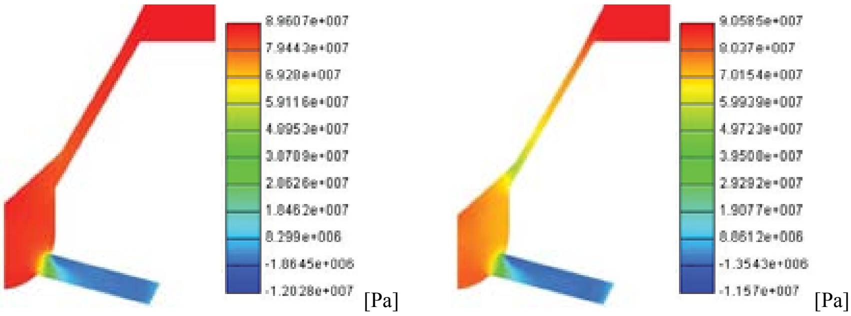

Fig. 9. The cross section of relative pressure flow field for nominal injection phase

The second diagram of the Figure 8 shows the sensitive area at seal seat surface where throttling effect occurs. There is a negative pressure region at the entrance of spray hole, and because of cavitation effect, in reality the negative pressure region is asymmetric. Accurate cavitation calculation could be solved in transient state. The figure also shows the pressure field in a time when the region inclined to cavitation effect is at the entrance of spray hole and adjoins downstream volume. It should be pointed out that cavitation were not observed between seal seat surface and needle valve. The figure 9 presents the cross sections of the hole channel in two places, A and B, and flow field distribution.

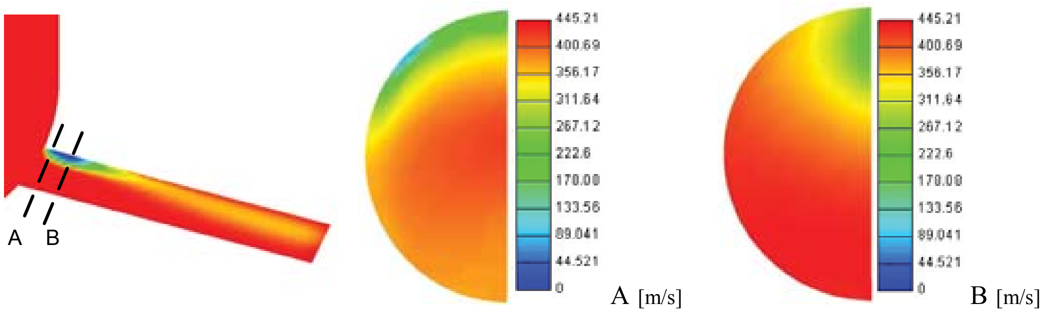

Fig. 10. The cross sections of velocity flow field in hole channel for nominal injection phase after cut-off

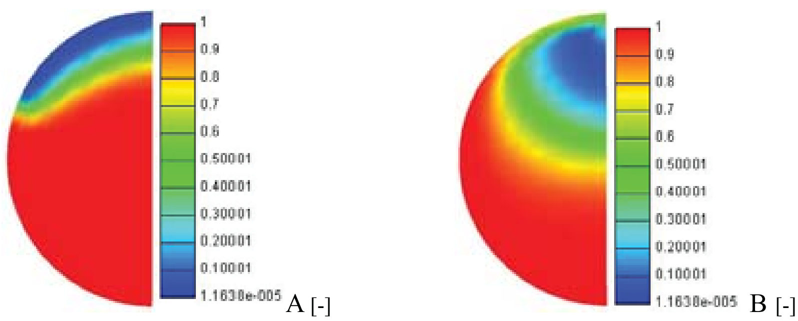

Fig. 11. The cross sections of volume fraction flow field in hole channel for nominal injection phase after cut-off

Cavitation bubbles are generated inside the injection hole. As cavitation induced bubbles travel downstream their diameter gets reduced, because the ambient pressure is higher than the vapour pressure. During the injection period, discrete fuel parcels enter the chamber with an initial diameter equal to the nozzle hole diameter. Each fuel parcel contains bubbles, according to the instantaneous volume fraction and size distribution computed from the phenomenological cavitation model. On the other hand, droplets penetrate excessively as parent parcels remain larger with a more slowly occurring primary break-up. This trend is illustrated in Figure 12 that compares spray snap shots after start of injection.

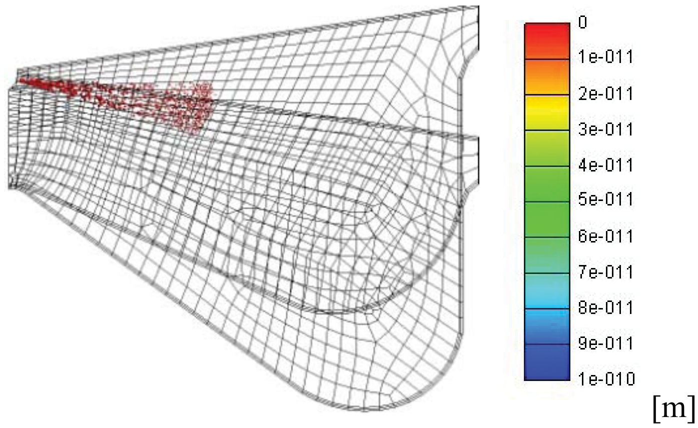

Fig. 12. The cavitation-induced spray formation process in modelled combustion chamber

4. Conclusions

An integrated simulation tool has been engaged for modelling the behaviour of a fuel injection system. For an effective injection simulation an experimental results are necessary to determine discharge coefficients. Having demonstrated good agreement between predictions and measurements at an injection pressure, discharge coefficients in 1-D injection system model were adopted. Predicted discharge coefficients for the injector nozzle holes and needle tip passage are found to be quite suitable, and resulted in acceptable calculation results. Based on 3D CFD code injection nozzle model created and as a support boundary set of data using 1-D model results produced. Part of the 3D CFD simulation results presented. In future work, the application of the integrated tool to different stages of fuel injection equipment will be presented.

References

[1] Heywood, J. B., *Internal combustion engine fundamentals*, McGraw-Hill, Inc. 1988.
[2] Stiesch, G., *Modeling Engine Spray and Combustion processes*, Springer Verlag 2003.
[3] Wang, T. C., Han, J. S., Xie, X. B., Lai1, M. C., Henein N. A., Schwarz E., Bryzik W., *Parametric Characterization of High-Pressure Diesel Fuel Injection Systems*, Transactions of the ASME, Vol. 125, April 2003, pp 412-426,2003.
[4] Seykens, X. L. J., Somers, L. M. T., Baert, R. S. G., *Modeling of Common Rail Fuel Injection System and Influence of Fluid Properties on Injection Process*, Proceedings of VAFSEP 2004, 6-9 July, Dublin, 2004.
[5] Nishimura, A., Assanis, D. N., *A Model for Primary Diesel Fuel Atomization Based on Cavitation Bubble Collapse Energy,* Proceedings of ICLASS 2000, 16-20 July, Pasadena, 2000.
[6] Minghai, Li., Xiaokun, Zhang, Hongjiang, Cui, Ying, Guan, *Numerical Simulation and Improvement of a Locomotive Diesel Nozzle*, CIMAC Congress, Vienna 2007.