INJECTION CHARACTERISTICS SIMULATION AND ANALYSIS IN DIESEL ENGINES*

A. FICARELLA and D. LA FORGIA
Politecnico di Bari, Istituto di Macchine ed Energetica, Via Re David, 200, 70125 Bari, Italy

(Received: 30 January 1991; accepted in revised form: 18 November 1992)

ABSTRACT. By means of a purposely developed numerical code an investigation of injection systems in diesel engines has been carried out. Cavitation at low pressure was simulated. The experimental results obtained were compared with the numerical ones. Data on cavitation waves, in the pipe between pump and injector, were obtained during that simulation. In particular, the influence of the relief volume on cavitation volume and injected fuel rate were computed. Unstable working conditions, characterized by a large variation of the injected fuel, at the same operating point, have been experimentally investigated and simulated.

SOMMARIO. È stato sviluppato un modello di calcolo per la simulazione degli apparati di iniezione, sia con pompa rotativa che alternativa, per motori Diesel comprendente il fenomeno della cavitazione. Parallelamente, è stato appositamente strumentato un apparato di iniezione per investigare sia gli andamenti della pressione sia quelli di altri parametri fondamentali in diverse sezioni dell'apparato. Il confronto dei risultati numerici e sperimentali ha dato una buona corrispondenza. € stato, inoltre, approfondita l'influenza della cavitazione e dei gas disciolti nel combustibile sul funzionamento dell'apparato evidenziandone la grande variabilit€ causata nella portata di iniezione.

KEY WORDS: Cavitation, Injection simulation, Injection instability, Injected flow-rate variability.

1. INTRODUCTION

The need for maximum fuel economy and minimum pollution and noise in diesel engines requires a detailed investigation – both experimental and numerical – on the characteristics of the fuel injection system [1–4], but much must be done to develop very accurate mathematical models to reduce the long and expensive experimental testings.

The earliest mathematical models simplified the system in two phases: pressurization of the volume determined by the pump, the pipe and the nozzle; and injection through the nozzle orifices. In this way the phase lag between the pressurization of the pump and the nozzle was neglected [5]. Nevertheless, the slow engines of that period, the short delivery pipes and the low pressures made such a hypothesis acceptable. After this approach, equations of pressure wave propagation inside the pump-injector pipe [5] were used.

The possibility of taking into account the effects of the elastic concentrated volumes, the counterpressure of fuel system and cylinder, and the inertia of different moving elements allowed parametric studies of the system [6–8]. Also the possibility of describing precisely the influence of the various volumes on injection permitted the simulation of new systems such as the compact injector-pump units [9]. The use of simplifying hypotheses (neglecting friction, assuming fluid velocity and density constant with pressure) permitted a theory of formulation of the propagation in pipes based on small acoustic perturbation [10]. This is characterized by its simplicity and a sense of physics. Nevertheless, in the new systems the high pressures (more than 100 MPa) contradict the hypothesis of a constant sound velocity and call attention to phenomena like blowby losses in the gap between plunger and needle, pipe elasticity, and variation of the flow characteristics with pressure [10–13].

Afterwards, interest was directed to the spray and the interrelation between its characteristics and the geometric configuration of the system [14–16]. The influence of different injector geometries was studied as well as the detailed simulation of the moving parts allowing the calculation of the mechanical stress. Cavitation began to be investigated first in a rather primitive way [17, 18] and later with a more scientific approach [19, 20]. The vapour wave formation in the apparatus following the fuel flow-back phase was analysed; the effects of some apparatus and fuel characteristics, as the air dissolved in the fuel and the relief volume of the delivery valve, were studied.

The model used here is mainly based on the model presented in [20] and further developed in [21] for an in-line injection pump; later, for the first time, in [22, 24] the theory was applied to a rotating one. The last model required the simulation of the complex geometry of the rotation–translation motion of the pumping element and the simulation of several orifice flow areas varying with time. The method of characteristics has been applied, allowing for the variation of sound velocity in the different pipe sections. Pischinger et al. [25] had already demonstrated that the variation of sound velocity with pressure cannot be neglected due to the small amount of air in the solution which can remarkably reduce the sound velocity at low pressure (from 1400 m/s to 850 m/s). Therefore, the method of characteristics had to be used to simulate the strong influence of the air in the solution contained by the fuel.

The quantity of vapour formed in a certain section of the pipe is directly calculated by means of the mass continuity
2. INJECTION SYSTEM SIMULATION

To investigate fluid-dynamic phenomena in injection systems a mathematical model for the simulation of pump, nozzle and connecting pipe is to be developed. It must predict the behaviour of the pressure and the fluid velocity in many points of the system. As for the fuel pump, the model has to define behaviour in time of the pressure waves starting from it, which affect tip lift and injection.

The developed mathematical model, as for the fuel pump, can consider different types of valves, as delivery valve, constant pressure valve, RSD valve, dash-pot valve. The pump can be divided into several characteristic volumes: intake chamber, plunger chamber, chamber upstream of delivery valve, and the chamber downstream of the valve. The mathematical model is based on the continuity equation between the volumes mentioned \([10, 20]\). The calculation of flow velocity in the various contractions of the section of the chambers can be determined using the classic formulation of steady flow suggested by Schmitt \([26]\). The area of leakage losses, due to the plunger backlash-gap, and the flow coefficient can be modelled assuming a flow in the laminar regime through the plunger gap, as suggested by Gibson \([12]\).

Cavitation phenomena have been treated starting from Garro’s suggestions, applying the continuity equation to all the considered volumes. To determine the valve movement, the balance between the pressure forces on the valve surface, the elastic reaction of the spring, the friction and inertia forces have been considered.

The basic conditions for the injector model are flow rate continuity and equilibrium of the forces applied to the nozzle needle. In particular, a detailed mechanical model was used to simulate the dynamics of the spring as well as of the moving elements. The nozzle needle, the spring plate and the spring have been divided into a series of masses connected by springs and dampers, which simulate the stiffness of such elements and the damping phenomena inside the material. Moreover, the possibility that those elements would not touch each other has been taken into account. The needle impact on the superior stop was modelled considering contact stiffness; the same scheme was used for the needle impact on the sac, which has been regarded as a mass elastically connected with the whole structure. In addition to the internal forces, the forces due to fluid pressure on the needle and sac, as well as the friction between a moving mass and the fluid around it, were considered; consequently, the general equation of dynamics was applied to each single mass. This model allows the simulation of injectors with complex geometry, as the two spring injector.

The injector was modelled as appointed by several communicating chambers; there are at least two chambers: the delivery chamber and the sac. Since the pressure drop in the section between the needle and the seat depends on (besides the needle lift) the geometry of the section itself, a