

Multi-Fidelity Total Integrated Simulation Technology for High Pressure Pump with Squeeze Film Effect

U Oh, Kazuya Kusano, Norihiko Nonaka, and Hironobu Yamakawa
 Hitachi, Ltd.

ABSTRACT

Automotive fuel can be efficiently combusted by injecting it into the cylinders at high pressure to atomize it to pass the regulations for exhaust gas and fuel economy. For this reason, automotive companies have developed direct injection engines, which can inject gasoline into the cylinders directly. Furthermore, the demand for lower-noise high pressure pumps is also increasing from the viewpoint of automotive comfort. Since the valve velocity and noise level will increase as the pressure in fuel pumps increases, noise problems need to be solved under the high pressure conditions. Accordingly, the valve motion should be predicted with high accuracy under operating conditions to evaluate the noise caused by valve impingement. In addition, the squeeze film effect phenomenon will occur in the physical fuel pumps affect the prediction of the noise level caused by valve impingement. Therefore, we focused on high pressure fuel pumps to develop multi-fidelity (MF) total integrated simulation technology. We couple 1D system analysis and 3D moving boundary analysis with the squeeze film effect to predict the valve motion under operating conditions in the MF simulation. The valve motion and the boundary conditions of a 3D moving boundary analysis are predicted by 1D system analysis. The fluid force that acts on the valve is predicted by 3D moving boundary analysis. Finally, we could predict the valve deceleration phenomena by applying the MF technology to the high pressure fuel pumps.

CITATION: Oh, U., Kusano, K., Nonaka, N., and Yamakawa, H., "Multi-Fidelity Total Integrated Simulation Technology for High Pressure Pump with Squeeze Film Effect," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1325.

INTRODUCTION

The regulations for exhaust gas and fuel economy are being strengthened to reduce the environmental load causing climate change. To pass these regulations the fuel could be efficiently combusted by injecting it into the cylinders at high pressure to atomize it. For these reasons, automotive companies have developed direct injection engines, which inject gasoline into the cylinders directly.

Furthermore, from the viewpoint of automotive comfort, the demand for lower-noise high-pressure pumps is also increasing. Since the valve velocity and noise level will increase as the pressure in fuel pumps increases, noise problems need to be solved under the high pressure conditions. To evaluate the noise caused by valve impingement, the valve motion under operation conditions must be predicted. However, the valve motion is difficult to measure since several components are located around the valve region. In addition, making a prototype and conducting an experiment take a long time. Therefore, a high-accuracy analysis method needs to be developed.

The prediction of valve motion has previously been studied with model based design (MBD) [1] and computer fluid dynamics (CFD) [2] [3]. MBD can predict the valve motion affected by the total fuel pump system, however it has difficulty for explaining exactly how the fluid affects the valve motion. On the other hand, CFD can grasp how the fluid influences the valve. It has difficulty for evaluating total fuel pump system. For solving the above issues, we developed

multi-fidelity (MF) total integrated simulation technology [4], which can predict the behaviour of a total product system by using coupled analysis models that change the fidelity depending on the required accuracy. In addition, the squeeze film effect phenomenon, which involves pressure increasing between two plates when the two plates become close enough, occurs in the physical fuel pumps. This phenomenon has effects on the prediction of the noise level caused by valve impinging.

Therefore, this research aims to predict of the valve motion with the squeeze film effect under the operating condition. This research focuses on valve closing motion in flow rate control valve region to develop MF total integrated simulation technology by directly coupling 1D system analysis and 3D moving boundary analysis.

STRUCTURE AND BASIC ACTIONS OF HIGH PRESSURE PUMPS AND FLOW RATE CONTROL VALVE

Figure 1 shows the overview of the direct injection fuel system and flow rate control valve region. The direct injection fuel system is mainly composed of a fuel tank, a feed pump, common rails and injectors. The fuel pump is composed of a plunger, a pump room, a flow rate control valve, an outlet check valve and solenoids. The flow rate control valve can roughly be divided into two parts: Solenoid and valve. The solenoid part is composed of a core and an anchor. The

valve part is composed of an inlet valve, a valve seat, and a stopper. The broken yellow lines show that springs are located between the anchor-core and inlet valve-stopper. The inlet valve is open (as illustrated in Fig. 1) when the fuel pump starts to suck the fuel.

Fuel is supplied to the fuel pump by a low pressure feed pump located in the fuel tank. The supplied fuel is sucked by the inlet valve as the plunger descends. When the plunger moves up, fuel flows backward through the inlet valve, and the inlet valve is closed by control signals. After the inlet valve closes, the backward flow stops and the pressure of the pump room increases. When the pressure level of the pump room exceeds that of high pressure fuel piping downstream of the outlet check valve, the outlet check valve push the spring to open and the fuel is supplied to the common rails. Also, when the pressure level of pump room decreases, the outlet check valve is closed by spring force, the supply of fuel finishes, and the plunger starts to move down.

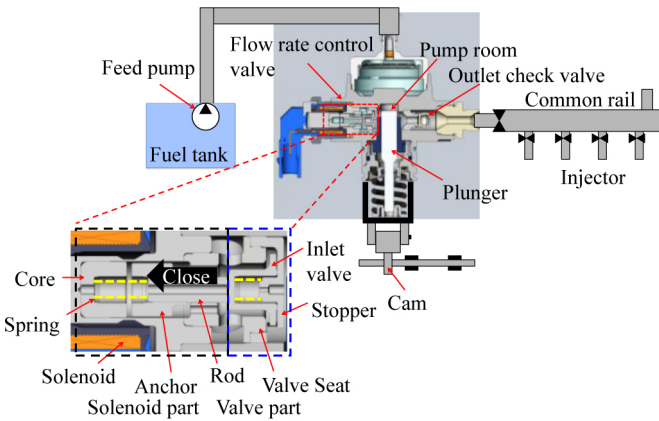


Figure 1. Overview of direct injection fuel system and flow rate control valve region

MF TOTAL INTEGRATED SIMULATION TECHNOLOGY

Overview of MF Total Integrated Simulation Technology

In conventional product design, component phenomena are modeled by 3D analyses such as structural analysis and fluid analysis are performed predict performance with high accuracy. On the other hand, component phenomena are modeled by 1D analyses such as mechanism analysis and system analysis, are performed roughly estimate the total performance of subsystems or the total system. However, 3D analysis requires a huge amount of processing time, and it was applied to only a part of the product. On the other hand, 1D analysis has difficulty accurately predicting performance because simplified modeling is used. Therefore, the conventional approach using 3D analysis for components or using 1D analysis has had difficulty accurately predicting the total product performance in a short time. Thus, we developed a multi-fidelity total integrated simulation technology that makes it possible to analyze total systems

by selecting analysis models in accordance with the required accuracy. 3D and 1D analyses are defined as high and low fidelity, respectively, in MF total integrated simulation technology.

Overview of MF Total Integrated Simulation Platform for the Fuel Pump

Figure 2 shows the overview of the MF total integrated simulation platform for the fuel pump. The system analysis model is constructed by using commercial software, MATLAB/Simulink. This analysis model is developed to simulate the behavior of the total fuel pump system under the operating conditions, and the flow channels are modeled by pipe/orifice models in the system analysis. The inlet valve motion is determined by the 1D motion equation (Eq. (1)), and the fluid force that acts on the inlet valve is considered as external force in the system analysis model and the fluid force is calculated by the pressure difference in the inlet valve (Eq. (2)).

$$m_{valve}\ddot{x}_{valve} + k_{valve}x_{valve} + c\dot{x}_{valve} = -F_{f_valve} \quad (1)$$

$$F_{f_valve} = \int \Delta P \cdot dS_{valve} \quad (2)$$

m_{valve} is the mass of the inlet valve, k_{valve} is the spring coefficient located between the inlet valve and the stopper, c is the damper coefficient, x_{valve} is the valve displacement, F_{f_valve} is the fluid force acting on the inlet valve, ΔP is the pressure difference in the inlet valve, and dS_{valve} is the surface area of the inlet valve.

The anchor motion is determined by the force balance of the spring force and solenoid force in the system analysis (as shown in Eq. (3)), and the solenoid force that acts on the anchor is used in a data table. This data table is composed of the solenoid forces calculated by the static magnetic analysis.

$$m_{anchor}\ddot{x}_{anchor} - k_{anchor}x_{anchor} + c\dot{x}_{anchor} = F_{solenoid} \quad (3)$$

m_{anchor} is the mass of the anchor, k_{anchor} is the spring coefficient located between the anchor and the core, x_{anchor} is the anchor displacement, and $F_{solenoid}$ is the solenoid force acting on the anchor.

As shown in Eq. (3), the fluid force that acts on the anchor is not considered in the system analysis. Therefore, the anchor and the inlet valve motion are difficult to predict with high accuracy by only using the system analysis. Thus, we start to develop the total analysis platform for high pressure fuel pumps with MF total integrated simulation technology that couples directly the system analysis and the fluid analysis. In addition, the 1D motion equation of the anchor in the system analysis is modified as follows:

$$m_{anchor}\ddot{x}_{anchor} - k_{anchor}x_{anchor} + c\dot{x}_{anchor} = F_{solenoid} - F_{f_anchor} \quad (4)$$

where F_{f_anchor} is the fluid force acting on the anchor.

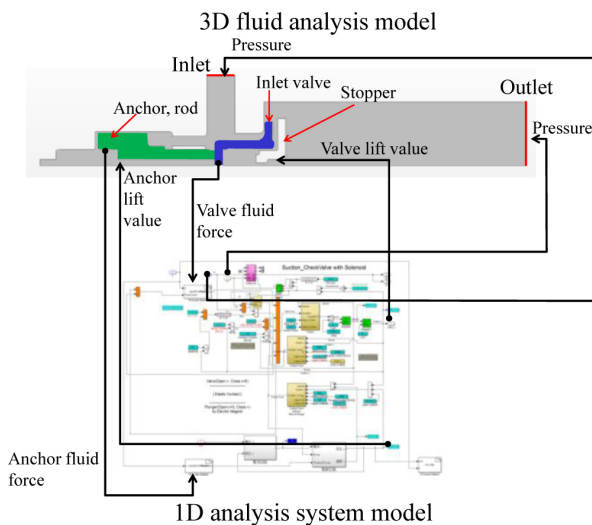


Figure 2. Overview of high pressure MF total integrated simulation platform for fuel pump

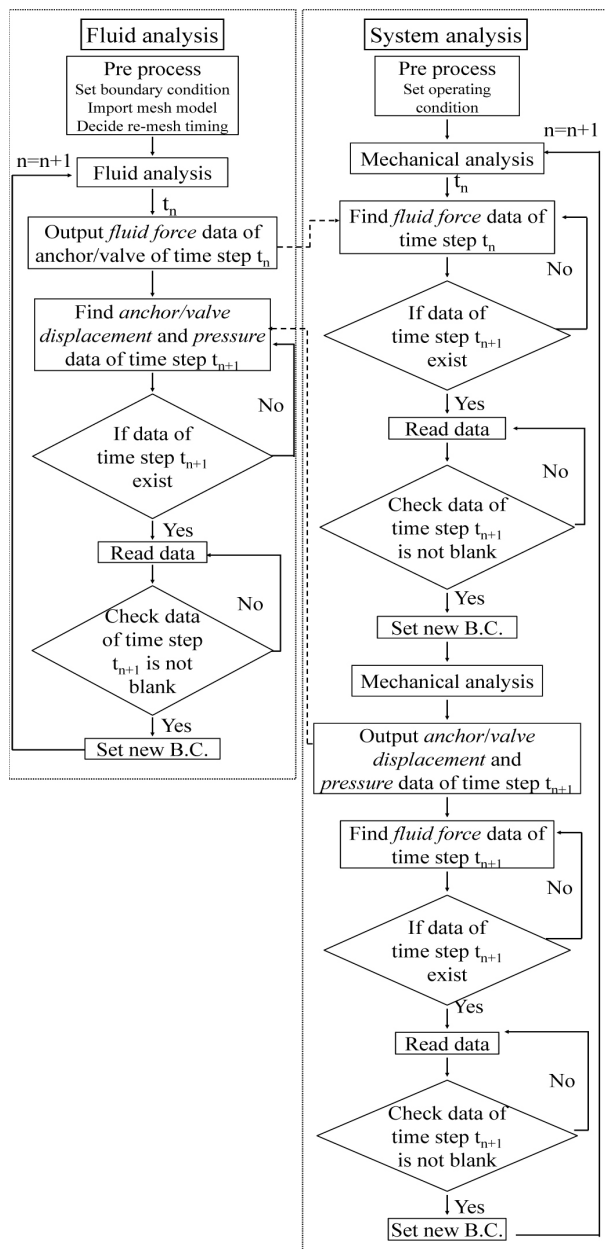


Figure 3. Flow chart of MF total integrated simulation platform for fuel pump

Figure 3 shows the flow chart of the MF total integrated simulation platform. t_n shows the time step. The fluid analysis first calculates the fluid force of t_n , and then inputs the force data into the system analysis to calculate the displacement data and the pressure data at $t_n + 1$. The system analysis is in a waiting state while the system analysis obtains the fluid force data at t_n . Next, the fluid analysis moves to the process of reading/setting new boundary condition data at $t_n + 1$ after it outputs the force data at t_n and the fluid analysis is in a waiting state while the system analysis calculates the new boundary condition data at $t_n + 1$. Finally, the fluid analysis uses the new boundary condition data to calculate the fluid force at $t_n + 1$ again. The MF total integrated simulation platform conducts these processes repeatedly to calculate the anchor/valve motion.

Fluid Analysis Model

Figure 4 shows the fluid analysis model of total analysis. The total analysis platform uses the commercial software Star-CCM+ to conduct the fluid analysis and the analysis model is an axially symmetric model. The initial gap h_{valve} between the inlet valve and the stopper and the h_{gap} between the anchor rod and the inlet valve are set to $0.5\mu\text{m}$ in the fluid analysis model. The minimum gap h_{valve} , h_{gap} , h_{anchor} between the anchor and the core and $h_{stopper}$ between the inlet valve and the valve seat are also set to $0.5\mu\text{m}$ in the fluid analysis model. The fluid analysis uses 3D moving boundary analysis to calculate the fluid force in the total integrated simulation platform. Moving boundary analysis uses the mesh morphing method [5] [6], which changes the initial shape of mesh to simulate the anchor/valve motion. Therefore, the deformation of the mesh shape will increase as valve displacements increase. Finally, the negative volume mesh will be generated, and the simulation will stop. For this reason, the fluid analysis needs to conduct re-meshing and keep the mesh quality high. We conduct re-meshing every 10 steps in the fluid analysis.

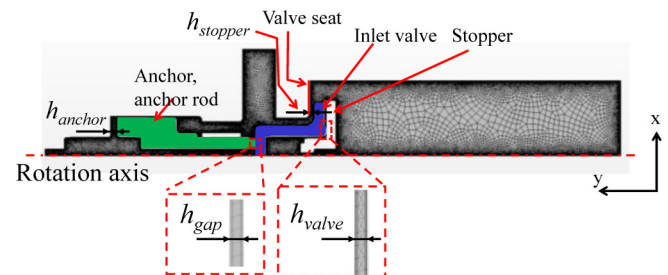


Figure 4. Fluid analysis model

SIMULATION RESULTS AND DISCUSSION

Confirmation of Squeeze Film Effect in Fluid Analysis

Figure 5 explains the squeeze film effect. The fluid is squeezed out when the two plates become close, and the flow between two plates is called squeeze flow. However, the fluid located at the center of the plate region is difficult to squeeze out due to fluid friction. Accordingly, pressure in that region increases, and the moving velocity of plates decreases (this phenomenon is commonly called the squeeze film effect).