Improved Structure and Three-dimensional Numerical Analysis of Diesel Nozzle

LI Ming-hai, YIN Hai-ting*
School of Traffic and Transportation, Dalian Jiaotong University, Dalian 116028, China
*corresponding author, e-mail: dlinghai@vip.sina.com, a5458539@126.com*

Abstract
First the study shows flow area can enlarge and turbulence generated by energy loss can reduce as the fuel injection hole chamfering, when spray nozzles are designed in the cone by analyzing and calculating the parameters of a high-power diesel. Then the investigation is created that it is about the pressure maps, velocity maps and discharge maps after the models are imported into the fluid software. The results about the numerical simulation test and verify the validity of the improved structure providing a reference for domestic peers.

Key words: diesel engine, engine nozzle, three-dimensional fluid, improved design

Copyright © 2012 Universitas Ahmad Dahlan. All rights reserved.

1. Introduction
The internal combustion engine fuel injection system is one of the most important manufacturing and the most accurate regulation system in the internal combustion engine. Key performance indicators of its internal combustion engine are such as the power, fuel economy, emissions, noise and so on. Of course reliability and durability also has an important impact. As the heart of the internal combustion engine, the fuel injection system consists of three precision dual-fuel injection pump plunger and barrel assembly, the outlet valve and needle valve coupled parts. Usually in order to ensure the gap between the needle valve coupled parts of the seal, the interval between needle valve needle and valve body always is 1.5-3.0 $\mu$m. Among them, the worst work environment is needle valve coupled parts.

On one hand, it suffers high pressure oil pressure from the injection pump. On the other hand its head is inserted into the combustion chamber by high temperature gas [1]. In addition we do not only pay attention to the needle valve coupled parts, materials, heat treatment and processing technology, but also pay attention to the rational design of the structure. Test and calculation results show that domestic diesel injector needle even work when the parts temperature between 200 °C -300 °C; the needle valve needle and valve body impact stress gets up to 882-931MPa, but it has poor impact resistance. Statistical analysis showed that: the past three years, about 8 injectors per diesel locomotive overhaul in 40 000KM need replacement for atomization bad. In order to reduce the impact of stress, it need ensure that the fuel injection pump needle valve flow capacity is under the premise of minimizing needle lift, thereby increasing the service life [2]. The literature [3] shows that 90° needle valve seat is more effective than the 60° needle valve seat to improve nozzle flow characteristics, and literature [4] shows that nozzle at the surface of nozzle flow properties are better than at the lower pressure chamber. Then the pressure loss is reduced and the total flow coefficient is slightly increased. Thus to reduce the impact of needle lift the needle valve body ensuring the flow capacity is essentially the same case, by improving the original structure. However, due to the small volume within the nozzle, the flow field is difficult to get through experiments and CFD model can compensate for this shortcoming. The paper has the ideas to improve the structure of a high-power diesel engine nozzle flow field simulation distribution and analysis of the structural improvements of rationality.

2. Improvement and analysis of the structure of a nozzle position
Typically, in order to make a good co-ordination of the needle valve body cone angle difference of about 1°, it may ensure its general seal, between needle valve needle and valve
body cone angle, when the minimum flow area is showed in the II- II at the Figure 1. While the accurate flow area is calculated as follows [5]:

\[ f_1 = \pi \left( \frac{\sin(\alpha_1/2)}{\cos[(\alpha_1-\alpha)/4]} \times \left[ h + (d_1-d)/2 \right] \times \left[ (\alpha_1/2) \times \frac{\cos[\alpha_1/2]}{\tan(\alpha_1/2)} \right] \right) \times \frac{d_1 - \sin(\alpha_1/2) \times \cos[\alpha_1]}{2 \left[ \frac{\cos(\alpha_1/2)}{\tan(\alpha_1/2)} - 1 \right]} \times \left[ \frac{\alpha_1 - \alpha}{2} \right] \times \left[ h + (d_1-d)/2 \right] \times \left[ (\alpha_1/2) \times \frac{\cos[\alpha_1/2]}{\tan(\alpha_1/2)} \right] \]  

(1)

Where: \( h \), for the needle lift, mm; \( d_1 \), sealed diameter, mm; \( d \), the pressure chamber diameter, mm; \( \alpha_1 \), needle valve body cone angle, (°); \( \alpha \), needle valve end of the cone angle, (°).

A high power diesel engine injector nozzle in the pressure chamber following improvement program can be the way to change the nozzle needle and valve body of the cone. As shown in Figure 1, it is the seat surface circulation area, according to equation (2) calculation. Calculation should be replaced by the pressure chamber diameter \( d_n \), the distance between the center of the nozzle. The flow area increases the injection pressure to the pressure loss is reduced by the throttle effect to improve the quality of atomization of the fuel and the fuel injection time is more reasonable. Although the formula is very accurate, it is rather cumbersome; the \( \alpha_1 - \alpha < 1 \) °, the needle lift \( h > 0.4 \) mm, formula (1) can simplify to be the following formula:

\[ f_2 = \pi \sin(\alpha/2) \times (d-h \sin(\alpha/2)) \times h \]  

(2)

According to Figure 2, the nozzle structure shows that the structure of the flow area is calculated as follows: \( f_a = \pi \sin(\alpha/2) \times (d-h \sin(\alpha/2)) \times h = 2.36 \text{ mm}^2 \). According to the structure in Figure 2(b) as shown, we can see that the flow of cross-sectional area of the structure is calculated as follows in the nozzle:

\[ f_b = \pi \sin(\alpha/2) \times (d_n-h \sin(\alpha/2)) \times h = 3.73 \text{ mm}^2 \]

The flow of cross-sectional area change before and after:

\[ \Delta f_k = f_b - f_a = 1.37 \text{ mm}^2 \]

The calculation of the initial significant increase in the flow area of the nozzle when the needle nozzle is set in valve body cone of the nozzle pressure chamber following. Through the improved nozzle structure in the cone of the needle valve body, and by reducing the diameter of the fuel injection pressure chamber it can reduce the fuel pressure loss. Analyze the needle valve seat surface cone angle, spray the nozzle location and nozzle angle and other factors, the author proposes four simulation scenarios. At the same time, in order to compare these four nozzle structure of three-dimensional flow field distribution, the author uses the fluid Software out of a comparative analysis of improvement before and after the needle valve flow field of the solid model.

3. Calculation Model
3.1 nozzle flow channel structure model

The first using of 3D modeling software, 3D graphics for the nozzle to the high pressure fuel within the fluid parameters of the nozzle of the major improvements are shown in Table 1.
All programs nozzle diameter is 0.42 mm and has 8 nozzles. Built by the simplified model are shown in Figure 3.

Figure 3. Four kinds of structure Three-dimensional model of the fuel fluid in the nozzle

<table>
<thead>
<tr>
<th>Structure</th>
<th>Seat Cone</th>
<th>Angle (°)</th>
<th>Needle Lift (mm)</th>
<th>Chamber Diameter (mm)</th>
<th>Nozzle Angle (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>60</td>
<td>0.7</td>
<td>0.25</td>
<td>146</td>
<td></td>
</tr>
<tr>
<td>B</td>
<td>60</td>
<td>0.6</td>
<td>0.2</td>
<td>120</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>90</td>
<td>0.7</td>
<td>0.25</td>
<td>146</td>
<td></td>
</tr>
<tr>
<td>D</td>
<td>90</td>
<td>0.6</td>
<td>0.2</td>
<td>120</td>
<td></td>
</tr>
</tbody>
</table>

3.2 Meshing
The three-dimensional solid models need to import the Gambit mesh before using the fluid Software computation. According to the model structure characteristics in the nozzle near the oil duct shape rules hexahedral grid is selected, while the rest adopt tetrahedral mesh.

3.3 Numerical Methods
In this paper, three-dimensional steady-state simulation is adopted [6]. And make the following assumptions and simplifications are made: (1) no chemical reaction in the flow process. (2) The temperature gradient is ignored in the flow process. (3) The flow of fuel has no incompressibility; density is assumed that density change with pressure by D.Dowson and GR Higginson proposed function of the dimensionless.

\[
\rho = \rho_0 \left(1 + \frac{0.6 \times 10^{-0.9} p}{1 + 1.7 \times 10^{0.5} p}\right)
\]

Where: \(\rho\) is the density; \(p\) the pressure.

3.4 Fuel fluid properties
Desirable oil fluid properties contain that fuel mixture for No. 0 light diesel oil, density of 850 kg/m³ at 20 °C, the effective viscosity coefficient: 0.05 kg/(m s), which are from [7].

3.5 The boundary conditions
Reference of (DF4 and DF11 locomotive diesel quality sampling rules), in order to ensure the needle lift reaches its maximum, selected the inlet pressure is 120MPa; The outlet pressure is taken when the cylinder piston is at compression top dead center when the pressure in accordance is calculated as 4.4MPa for the gas adiabatic compression. At the same time the wall roughness of 0.1mm.3 numerical results are taken.

4. Numerical Results
4.1 Different structures of static pressure
In figure 4 the comparison from static pressure of A and C structures and B and D structures of the cloud images can be found between the needle valve and seat surface pressure did not change significantly between the needle valve; seat does not play a role in the closure. This is the reason why the flow of cross-sectional area of flow area and needle valves and needle valve between the oil passages is basically the same. So the 90 ° seat needle valve
does not play the role of the diffuser. Contrast to the static pressure structures of the A and B and structures of C and D cloud, it can be found with the fuel injection nozzle location on the shift: the fuel injection nozzle flow area increases. Because the nozzle pressure chamber volume decreases, the nozzle internal fuel turbulence intensity increases, so the total pressure remained unchanged. Obviously C and D programs can be excluded, because the needle valve coupled parts is a precision when the needle valve seat cone angle is processed into 90 °. And no doubt that it will be a significant increase in manufacturing costs.

![Figure 4. Four options symmetry plane static pressure contour contrast cloud map](image)

As can be seen from Figure 5, compared to A and B structure, the pressure of the fluid region above the nozzle is basically the same, indicating that the tapered channel has no effect on the pressure between the needle and needle valve at the maximum needle lift. In the B structure of the nozzle at the pressure drop is significantly larger than the structure of A. This is because the more flow, the more pressure of the body can be transformed into a fluid kinetic energy, increasing the velocity of the fluid while the fluid quickly ejected. The A and C structure are compared with the cloud image. This is the same reason that the channel section area of the needle valve needle and valve body is essentially the same, and thus the needle valve of the size of the chamfer on the overall flow properties become smaller. At the same time, the high pressure fluid flow into the spray hole without chamber’s buffer directly in the B structure. Part of the fluid flow line of dramatic changes along to the nozzle that it will produce the negative pressure.

![Figure 5. Four options symmetry plane total pressure contour contrast cloud map](image)
Total pressure cloud of B and D structure is basically the same, where the needle valve angle is changed at the low end, while the fluid directly flow into the nozzle, without the pressure chamber. Obviously, the C and D structure has the same results as well.

As can be seen from Figure 6, the structure A performances at the larger peak velocity, but the uniformity of its speed is less than B structure compared to A and B structure. Apparently because the nozzles are close to the pressure chamber low-end, high-speed fluid in the pressure chamber will generate the larger turbulence. And the diameter is smaller in the pressure chamber, which may produce the fluid expansion. At the same time the pressure decreases. The transfer of turbulent energy will cause part of the fluid velocity decreases, while another part of the fluid velocity increases. This effect shows a certain degree of randomness and unpredictable. At the nozzle located on the cone, the high-speed fluid outflow of directly is less volatile, and the natural rate peak is relatively smaller. Contrast with the A and C structure, the 90° needle valve structure has the peak speed than 60° needle valve structure where the fluid in the needle valve head has diffuser effect. In general, the larger the needle valve angle, the greater the fluid velocity but there will be the large decline pressure. What can be seen is that the peak velocity distribution is significantly less than the A and B structures also the speed of the structure at the B peak distribution, which decides the more stable high-speed fluid. Comparison between B and D structures shows the speed value peak is basically the same, obviously D structure nozzle overall speed peak will be much smaller, and the chamfer D structure manufacturing costs are higher than the B structure. The numerical design is performed to find the optimum design [8]. To compare the speed of the four structural cloud images, we can assume that B structure more advantages, more fuel atomization.

![Figure 6. Four options symmetry plane velocity contour contrast cloud map](image)

### 4.2 The impact for chamfer of the flow field

In the Figure 7, it can be seen that the speed of the spray nozzle is more evenly distributed after the chamfering of structure B; the speed value is greater. The analysis results are showed that the higher the lower the pressure along the flow line speed of the incompressible fluid Bernoulli equation; the structure B, chamfer weakened the sharp edges of the closure of the high-pressure fuel, makes the downstream flow more conducive to the atomization of fuel in the combustion chamber. The visible chamfer inside the nozzle can also improve the fuel injection quantity and flow coefficient. However, the fillet radius is not easy too large, otherwise it will cause the spray holes through each other to reduce the ratio of nozzle length and diameter. And it will directly affect the spray penetration. When the nozzle is located in the needle valve body seat surface, chamfered the nozzle can reduce the turbulence generated by energy loss and improve fuel flow rate. At the same time due to the decrease of the needle lift, it can reduce the impact of the needle valve needle valve body. And improved the pressure chamber volume is small, and thus can reduce the operating mode of the HC emissions value in order to reduce the pollution of the environment. Therefore it will more consider the advantages of structure B after the chamfering.
5. Conclusion

Under the same conditions, the nozzle design in the conical seat surface can compensate for the turbulence generated by energy loss when the needle lift and nozzle pressure chamber volume decreases in order to improve liquidity by chamfering the nozzle. The structure of the high-power diesel engine fuel injectors of 90° needle valve seat cannot effectively improve the nozzle flow characteristics, and thus the change does not apply to all the needle valve seat cone angle nozzle structure. According to different literature on the nozzle structure optimization, it were summarized and combined with the specific circumstances of the diesel engine simulation, which can be found is not a simple way that all the improvements piled together in the optimization but the need for further analysis.

References