Research Paper Engineering

Fatigue Analysis in Fuel Injection of Diesel Engine Based on FEA

Amit	Department of Mechanical Engineering, Shepherd School of Engineering and Technology, Sam Higginbottom Institute of Agriculture Technology And Sciences, Deemed University Allahabad (U.P) INDIA
Earnest vinay prakash	Department of Mechanical Engineering, Shepherd School of Engineering and Technology, Sam Higginbottom Institute of Agriculture Technology And Sciences, Deemed University Allahabad (U.P) INDIA
Prabhat kumar sinha	Department of Mechanical Engineering, Shepherd School of Engineering and Technology, Sam Higginbottom Institute of Agriculture Technology And Sciences, Deemed University Allahabad (U.P) INDIA
manish kumar	Department of Mechanical Engineering, Shepherd School of Engineering and Technology, Sam Higginbottom Institute of Agriculture Technology And Sciences, Deemed University Allahabad (U.P) INDIA

ABSTRACT

The Liquid load of the fuel injection in Diesel Engine was studied based on FEA. Seating stress of the fuel injection was calculated by using the theory of stress wave. Then fatigue strength analysis of the fuel injection was performed. Results show that: static strength of the fuel injection is adequate; stress is concentrated in the jet orifice, but its safety factor is 8.60 which is big enough; safety factor of the needle valve seat is on the low side which means that the needle valve seat will produce cracks first in the fuel injection. These results are very useful for further failure analysis of fuel injection and optimum design of its structure parameters.

KEYWORDS Diesel, Engine, Fuel injection, FEA, Stress, CFD, fatigue

INTRODUCTION: It is known that the fuel injection is the most important part of the fuel injection system which is the heart of the diesel engine. The fuel injection works under the very terrible conditions, alternate liquid load, mechanical load and thermal load included, over a long period of time. The fuel injection comes under the function of the high pressure of the fuel when the fuel injector is injecting, while the pressure of the fuel inside the fuel injection is very low when the injection is end. The liquid load the fuel injection receives is alternate. The needle valve strikes upon the valve seat periodicity when the injector is working normally which means the mechanical load the fuel injection endures is variable. Under long term exposure to the high-temperature fuel gas in the cylinder, the temperature of the external surface of the fuel injection always reaches up to 200-300 sometimes even higher. The fuel injection is cooled by the fresh air inhaled into the cylinder in the intake stroke. The thermal load the fuel injection bears is alternate. It is obviously significant to analysis the fatigue strength of the fuel injection working under the coupling of a lot of complex and alternating load. The fluctuation of the temperature of the cylinder wall reflects mainly in a very thin tier and the amplitude of the fluctuation declines exponentially so the thermal load the fuel injection receives can be regarded as stable and is not be involved in the study. The stress and the deformation of the fuel injection under the alternate liquid load relates to the coupling of the liquid and the solid which is difficult to solve using the traditional method. This problem was solved successfully based on FEA (Finite Element Analysis) methods.

Finite Element Analysis in Fuel injection under Liquid Load:

Computational Fluid Dynamics (CFD) Simulation of the Flow inside the Fuel injection: Simulation analysis of the three-dimension fluid flow in the fuel injection of a certain diesel engine was performed by computational fluid dynamics software package ANSYS-FLOTRAN and the hydraulic pressure distribution at the tip of the fuel injection was obtained which is one of the important boundary conditions when the finite element analysis of the fuel injection is performed under the liquid load.

According to the theory accurate flow features can be obtained through the steady flow analysis of the jet orifice of the fuel injection when the needle valve is at its maximum lift and the flow velocity in the entrance of the valve seat of the fuel injection is very low and the direction of the velocity is identical. Thus, the flow region at the tip of the fuel injection when the needle valve is in its maximum lift is selected as the computational domain. In consideration of the periodical symmetry of the fuel injection, 1/5 of the computational domain, from the entrance of the valve seat to the outlet of the jet orifice, was set up.

The main structure parameters of the fuel injection are as follows:

The jet orifice diameter (*d)* =0.53 mm,

The jet orifice length (*l)* =1.55mm,

The jet orifice obliquity () =76°,

The maximum lift of the needle valve (*h)* =0.45 mm.

Three-dimension solid model of the fuel injection was set up by the software Pro/Engineer of the company PTC, as shown in figure 1. Lead the IGES file generated by Pro/Engineer to ANSYS-FLOTRAN. Select the tetrahedral element 3D FLOTRAN 142 provided by ANSYS-FLOTRAN to mesh the model. The grids formation of the computation model is shown in figure 2 where there are 20 455 elements and 4 702 nodes.

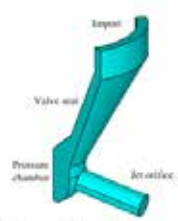


Fig.1 Geometrical model of computation region

Fig.2 Grids used for the CFD computations

The fuel the diesel engine uses under normal operating condition is 10 light diesel fuel, whose density ρ=850 kg/m^3, viscosity ν=4.25×10^{-3} Pas. It is assumed in the simulation that the liquid flow in the fuel injection is adiabatic, incompressible and viscous. The Reynolds number R in the entrance of the valve seat can be easily calculated which is much bigger than 2300, which means the flow inside the fuel injection is turbulent flow. The standard k-ρ model is the preferred model when analyze the turbulent flow. However, the velocity of flow at the tip of the fuel injection is very high and there is evident normal strain in the flow, the flow quantity of the turbulent flow estimated by the standard k-ρ model is greater than the fact, and so is the kinetic energy obtained. The RNG model adopted in this paper can make up for this shortage.

Based on the above CFD model, the simulation of the liquid flow inside the fuel injection was performed successfully. The hydraulic pressure distribution at the tip of the fuel injection was obtained when the needle valve is at its maximum lift, as shown in figure 3. Form the needle valve seat to the jet orifice, the hydraulic pressure distribution is not homogeneous but drop successively. The pressure in the valve seat is the maximum which is 27.3MPa, and the pressure in the jet orifice, which is 2.5MPa, is close to the pressure of the cylinder at the top dead centre in the compression stroke.

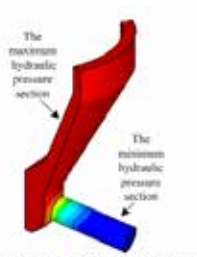


Fig.3 Distribution of hydraulic pressure

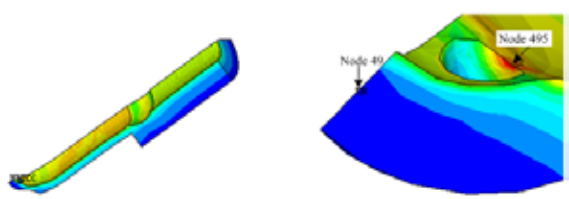
Fig.4 Finite element model of nozzle

Finite Element Analysis of Fuel injection:

One-fifth of the fuel injection was selected for the establishment of the FEA model of the fuel injection and its further finite element analysis. Set up the 3-D solid model of the fuel injection by Pro/Engineer then lead it to ANSYS. To reduce the quantity of the finite elements, some details, such as the chamfers, which have little effect on the local stress of the fuel injection, were ignored when modeling. Because of the existence of the oil groove and the jet orifice, the geometry of the needle valve is very irregular. Thus, the 3-D solid model was meshed by use of the 10-node linear tetrahedral entity element Solid 92 provided by ANSYS, according to Smart Manner and designing the size of the elements in different zone reasonably. There are 15569 elements and 24456 nodes in the finite element model.

When the injector is injecting, the mechanical loads the fuel injection endures are listed as follows: the stress the screw-down nut of the injector impresses on the shoulder of the needle valve, the high pressure inside the valve applied by the high-pressure fuel and the pressure impressed on the external surface of the fuel injection inside the cylinder by the working substance in the cylinder. According to the moment of force and the size of the nut, the stress impressed on the shoulder of the needle valve by the cap nut of the fuel injector can be easily got which is 314 MPa. The pressure impressed on the external surface of the fuel injection inside the cylinder by the working substance in the cylinder is close to the pressure when the combustion just starts which is 3 MPa. The hydraulic pressure distribution inside the fuel injection under the high-pressure fuel is given by the CFD simulation results, When the injection is end, the pressure of the fuel inside the fuel injection and at the oil inlet is 3.6 MPa, the stress impressed on the shoulder of the needle valve by the screw-down nut of the injector is still 314 MPa, the pressure impressed on the external surface of the fuel injection inside the cylinder is the intake pressure which is so low that it can be ignored.

The elastic modulus and the poisons ratio of the fuel injection are *E*=205 GPa, *μ*=0.3, respectively



(a) (b)

Fig.5 Distribution of Von Mises stress in nozzle

With the FEA model and the boundary conditions mentioned above, the finite element calculation of the fuel injection can be executed automatically. The Von Mises stress in the different positions of the fuel injection when the injector is injecting. From the figure we can see that the stress distribution in different positions of the fuel injection is not homogeneous but increases gradually from the external surface to the internal surface and from the top to the bottom of the fuel injection. The maximum tensile stress of the fuel injection is 81.80 MPa which appears at the 495 node in the upper end inside the jet orifice and the stress is concentrated in the small area, as shown in close-up view. The minimum tensile stress of the fuel injection appears at the 49 node. The boundary conditions changed, we can get the stress at the needle valve seat and the stress at the 495 node in the upper end inside the jet orifice, which is 5.83 MPa and 8.73 MPa, respectively, when the injection is end.

Calculation of Needle Valve Seating Stress:

At present, there are two formulas to calculate the needle valve seating stress under the impact of the needle valve: the empirical formula recommended and the formula deducted by Prof. Yao Chunde [5] from Tianjin University basing on the

theory of stress wave. As a matter of fact, the latter is more accurate and more reasonable than the former.

(1) There is no friction when the needle valve strikes on the needle valve seat under the spring force.
(2) The pressure inside the pressure chamber drops to zero instantaneously as soon as the injection is end.
(3) The collision between the needle valve and the needle valve seat is the collision between pure metals and there is no other substance between them.
Thus, the impact stress the needle valve seat endures is

$$\sigma = \frac{2K}{(d/D)^2+1}\sqrt{\frac{E\rho K_p}{M}\left(\frac{2F_s L}{K_p}+L^2\right)}$$

In which,

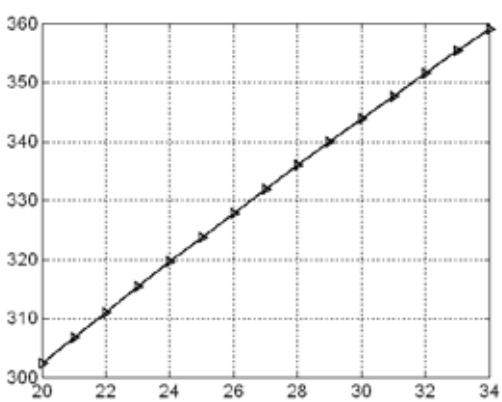
$$F_s = \frac{\pi}{4}(D^2-d^2)P_0$$

And,

$$K = \frac{d^2}{d_1^2-d_2^2}$$

Where,

K = section factor,
E = modulus of elasticity,
P = Density of the valve seat,
= stiffness coefficient of the spring,
M = summation of the mass of the needle valve and 1/3 the mass of the spring,
F_s = spring force,
L = lift of the needle valve,
P_0 = needle-opening pressure,
d = diameter of the valve seat,
D = pilot diameter of the needle valve,
d_1 = maximum seating diameter and
d_2 = diameter of the pressure chamber.



The relation between the needle-opening pressure and the impact stress the needle valve seating endures is basically linear. The impact stress the needle valve seating endures under the needle-opening pressure of 27 MPa is 332 MPa.

Fatigue Analysis in Fuel injection:
The danger areas and the vulnerable spots of the fuel injection can be found out if we associate the calculation results with the mechanical character of the material of the fuel injection which is shown in table.1

Table 1 Mechanical Behaviour of the fuel injection.

Limit of Yielding /Mpa	Fracture Limitation /MPa	Coefficient of Extension (%)	Reduction of Area (%)	Elastic Modulus /GPa
835	1175	10	45	205

From the analysis above, we can get the following results: the stress in the fuel injection is alternative generated under the liquid load and the strike of the needle valve, when the injector is injecting the maximum stress appears at the jet orifice, under the shock of the needle valve the maximum stress appears at the needle valve seating, when the injection is end the stress at the valve seating and at the jet orifice are both the minimum, as shown in table 2. The danger areas of the fuel injection are the valve seating and the jet orifice, but the values of stress in both areas are less than the fracture limitation of the material of the fuel injection, which means that the static strength of the fuel injection is adequate.

Table 2 Stress and Safe Coefficient of danger area of the nozzle

Parameter / Position	The maximum stress /MPa	The minimum stress /MPa	Stress amplitude /MPa	Mean Stress /MPa	Safety factor
Jet Orifice	81.80	8.73	36.54	45.27	8.60
Value Seat	332	5.83	163.09	168.92	2.02

Working under the alternate load over a long period of time, cracks will come into being easily at the weak points of the nozzle, fracture may even happen sometimes. The fatigue strength of the nozzle is studied according to Götaverkren-Soderberg method.

The calculation formula is given as follows:

$$n = \frac{1}{\sigma_a/\sigma_{-1} + \sigma_m/\sigma_b}$$

Where:

σ_a = Stress amplitude,
σ_{-1} = Fatigue limitation of the material,
σ_m = mean stress and
σ_b = fracture limitation of the material

The safety factors which are listed in table 2 at different positions of the nozzle under the alternative liquid, mechanical load can be obtained using the above formula. The change of the cross section around the jet orifice is eminent and the stress is concentrated here and here is regarded as the position where the cracks generates first in the nozzle, but the results of the fatigue strength analysis of the nozzle shows that the safety factor here is 8.60 which is big enough, while the safety factor at the needle valve seating is smaller which means that the valve seating is the weak point of the nozzle where the cracks generate first.

Conclusions:
The results of the static stress and the fatigue strength analysis in the nozzle of a certain diesel show:

(1) The needle valve seating is the position where the stress is the maximum of the nozzle. The stress is concentrated at the upper end inside the jet orifice. The valve seating and the jet orifice are the weak areas of the nozzle.
(2) The static strength of the nozzle is adequate.
(3) The stress is concentrated at the jet orifice but the safety factor is adequate here. The safety factor at the needle valve seating is smaller which means the valve seating is the position where the cracks generate first in the nozzle.
(4) These results are very useful for further fracture and failure analysis in nozzle of diesel engine.

REFERENCES

[1] Wang Xizhen. Analysis of Temperature Field and Heat Deformation for the Cylinder Liner of 195 diesel Engine by Using 3-dimensional FEM. Ms D Thesis. Hefei: Hefei University of Technology, 2000(in Chinese) | [2] Ju Yusheng. Application of CFD in research and development of products. Technology of Oil Pump and Nozzle, 1995, (4): 20-24(in Chinese) | [3] Reitz R D, Rutland C J. Development and testing of diesel engine CFD models. Progress in Energy and Combustion Science, 1995, 21(2): 173-196 | [4] Institute of Tractor of Luoyang. Spraying and Adjustment of Fuel in I.C. Engine. Beijing: China Machine Press, 1977(in Chinese) | [5] Yao Chunde. Calculation of needle valve seating stress for fuel injection of diesel engine. Mechanics and Practice, 1999, (4): 56-59(in Chinese) | [6] Wang Changyi, Chang Hanbao. Internal Combustion Engine. Changsha: Publishing House of Defence University of Science and Technology, 2001, 321-316(in Chinese)