13th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2009, Hammamet, Tunisia, 16-21 October 2009

A FINITE ELEMENTS PROCEDURE FOR EVALUATING INJECTORS RELIABILITY OF MEDIUM SPEED DIESEL ENGINES

Roberto Muscia
Dipartimento di Ingegneria Meccanica, Università di Trieste Via Alfonso Valerio n.10 – 34100 Trieste, Italia

Francesco Campagnolo
Wärtsilä Italia S.p.A., Bagnoli della Rosandra 334, 34018 San Dorligo della Valle, Trieste, Italia

Franco Fabio
Wärtsilä Italia S.p.A. Consulting Engineer

ABSTRACT
A kind of injection nozzle manufactured since 1990 and utilized for ten thousands of motors, has recently showed a series of failures. These failures have happened only in one naval equipment and during a short period of time. In any case an uncertainty relative to sizing criteria arose. With reference to this problem, in the present work a methodology for performing a more careful structural analysis of a nozzles injection class is suggested. This procedure is based on the finite element method and allows an accurate evaluation of the stress in the region where the failure has happened. Particular care has been taken to understand where singular and non singular stresses are superimposed. The methodology suggested is illustrated with reference to an injector that has shown a failure in a characteristic part of the nozzle. The conclusion of the analysis shows that this failure has been caused by a poor quality fuel supply.
Keywords: FEM, injector, diesel engine

1. INTRODUCTION
The injection system of Diesel engines is very important for increasing the thermodynamical performances of the system. At present the conventional injection system with Bosch pump [1] used for big size motors with low rpm's still has some advantages in comparison with the system adopted for the smaller Diesel engines (common rail). The need to decrease fuel consumption, emission of air pollutants, and the advantage of the utilization of less refined fuels have imposed a progressive increasing of the injection pressure. Consequently, the components of the injection device are much stressed. In particular the injection nozzle is an important part by which the fuel is injected in the combustion chamber. The working condition of this nozzle is the most critical of the whole system. The reasons are the following: i) the nozzle is placed in the cylinder head, therefore its dimensions are small, ii) its end is exposed to high temperature, in the combustion chamber, iii) it is exposed to the highest pressure of the injection system, iv) the fuel has the highest velocity in the injection holes, and v) the section changes of the fuel conduits in the nozzle can easily cause fuel cavitation in certain points. Usually, in relation to the phenomena complexity, sizing and design solution are experimentally tested on the basis of previous checks performed by simple computations. Since some nozzles manufactured since 1990 and utilized for thousands motors have recently showed a series of failures only in one naval equipment during a short period of time, it has been thought suitable to

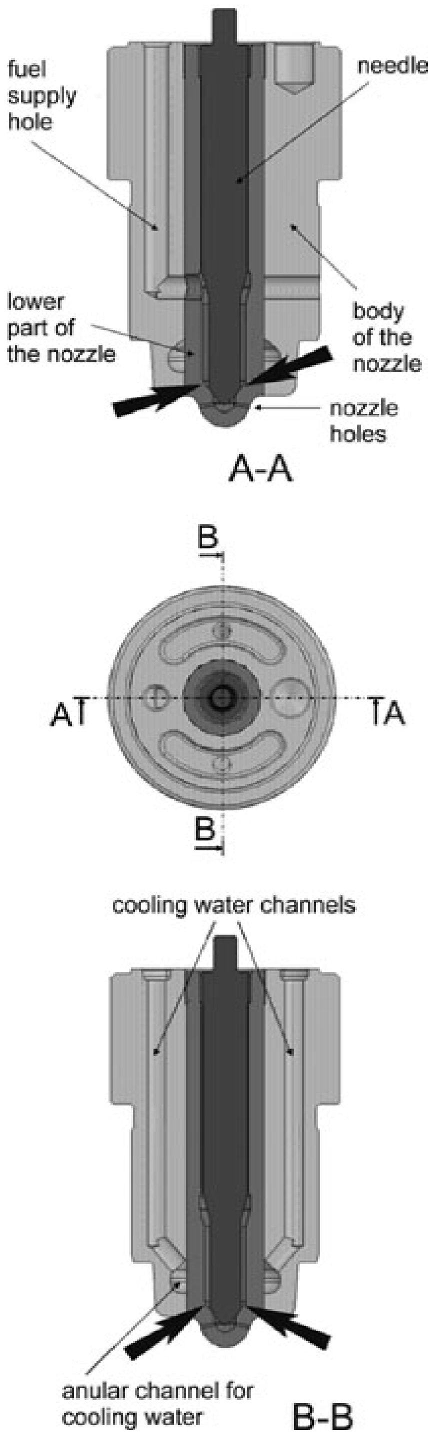
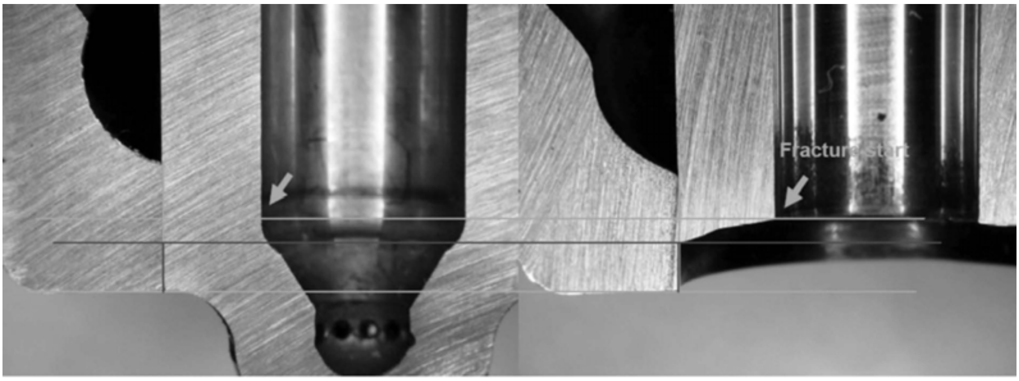


Figure 1. CAD Modelization of the fuel nozzle case study.

check the criteria used for the structural verification of these devices. Consequently, in the present paper a methodology for the evaluation of the stress that affects this kind of nozzles is illustrated. In the conclusions, the results obtained, with reference to the particular rupture cases that have happened, are also reported.

2. METHODOLOGY FOR STRESS EVALUATION

The methodology proposed can be summarized by the following basic steps: i) identification of the best nominal model of structural computation for the stress evaluation [2-9], ii) virtual modelling of each part of the injector [10], iii) exportation of each part modeled towards FEM softwares (Finite Elements Method) [11-13], iv) study of the contact conditions among the various parts of the injector, v) identification of a correct constraint model in FEM context and the external forces applied, vi) study of the virtual model simplifications in order to reduce the computation time, vii) execution of the FEM calculation and identification of singular and spurious stresses, viii) check that the above-mentioned stresses are not affecting the region where the fracture has happened, ix) comparison between the stress computed by FEM and the nominal structural models (only with reference to the regions where the shape allows us to apply these simple kinds of models), x) reliability study of the results obtained by the FEM software that has been used versus geometries and constraints/forces considered, xi) check of the displacements consistency relative to the contact surfaces of each part when the external forces are applied, and xii) execution of a fatigue computation based on Goodman-Smith diagram concerning the region where the rupture has happened. Fig. 1 shows a CAD modeling of the injectors kind that has been studied in detail. The black arrows point out where the fracture is happened. In Fig. 2 a picture of the fracture section is reported. We observe that the shape of fracture surface is similar to a circular crown. Consequently, after the rupture, the complete detachment of the nozzle end is happened and the working of the correspondent cylinder has been interrupted. Fig. 3 shows one of the final nozzle FEM models with the relative stress mapping. Since the system of forces applied to the body of nozzle and also the same nozzle are practically axial-symmetric, the structural FEM models have been developed with reference to angular sectors whose amplitude was equal to 10°. The validity of this approach has been proved by comparing the results obtained by computations relative to whole and half injector models. In the cases studied by the methodology proposed, the simulations have been performed by using the Young moduli E of the injector body and nozzle materials, respectively. All the simulations were of static type. Usually, the injector body is manufactured by a hardened and tempered steel 32CrMoV12-10 (E=210 Gpa, *HRC* 40). The nozzle is made by a sintered steel Böhler M390 (*E*=220 Gpa, *HRC* 60). This sintered steel is used because of its high hardness that assures a longer life to the injector.

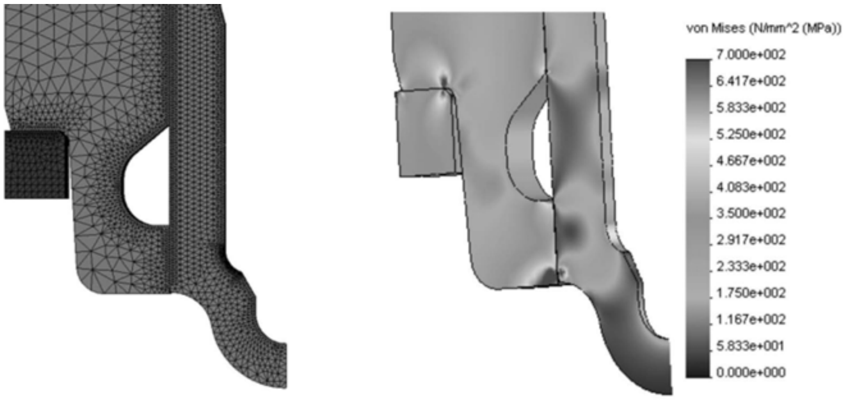


(a) (b)

Figure 2. Stereoscopic images of meridian sections (a) before and (b) after the fracture of the lower part of a real nozzle.

3. CONCLUSIONS

The results obtained by the FEM models managed as illustrated in the previous section [see steps i)-xii)] have proved that the more critical part of the nozzle is the lower fillet, just in the region where the fracture is happened. Nevertheless, we observe that the stress computed by utilizing the usual injection pressure of the fuel p_{max}= 1250 bar is noticeably lower than the yield stress σ_s relative to the sintered steel M390 (σ_s = 640 N/mm) by which the nozzle is manufactured. On the contrary, if p_{max} increases to 2000 bar, the equivalent Von Mises stress can attain the yield stress σ_s. All the simulations have been performed assuming that the nozzle is assembled in the injector body by an interference equal to 40 μm (an usual value for this kind of devices). Moreover, we really have to note that the pressure oscillates from the residue pressure p_r to the maximum one p_{max}. Therefore, by applying the procedure suggested, a fatigue stress computation has been performed. In relation to this aspect, we notice that the average life of the injectors considered must be equal to at least 5000 hours. After this time limit, a functional deterioration of the injector usually happens. This fact does not mean that the injector breaks. After 5000 hours of working it must be bench tested to check its fuel nebulization capability in the chamber of combustion. Consequently, in relation to the fatigue stress, Goodman-Smith diagrams have been drawn [3,8]. These diagrams considered a stress σ'_{fAF} = 455 N/mm^2 relative to a Wöhler test for 265000 cycles, a p_{max} = 1250 bar, and a stress preloading generated by assembling of the nozzle in the injector body with an interference equal to the above-mentioned 40 μm. The 265000 cycles corresponded with about 1020 minutes, the working time after which the fracture of the nozzle happened. With reference to a precautionary p_{max} = 1500 bar and p_r = 100 bar, it was clear that the working cycle in the Goodman-Smith diagram is well contained within the limits of the sub-diagram relative to a time life of the structure equal to 1020 minutes. In conclusion we can state that the region of the fillet certainly represents a critical part of the nozzle. Nevertheless, from the study performed we note that this stress concentration is not sufficient to produce a fracture after only 1020 minutes of working. Therefore it is reasonable to think that these ruptures are caused by a mechanical characteristic reduction of the material. This reduction could derive from an increasing of temperature associated to a too poorer flux of fuel. In fact the right fuel supply cools the end of the nozzle in the combustion chamber. When the “cold” fuel flow decreases, only a small quantity of heat can be removed. Consequently, the particular region of the fillet increases its temperature and the mechanical characteristic of the material decreases until the fracture happens. The fuel flow reduction has been certainly induced by the fuel filter obstruction that was noticed in the naval equipment where the fracture of nozzles happened. Therefore, the reason of the problem could be just a supply performed with a poor quality fuel.



(a) (b)

Figure 3.(a) Mesh of the final models, (b) Von Mises stress mapping.

4. REFERENCES

[1] Ferrari, G.: Motori a combustione interna, Edizioni Capitello, Torino, 2001.

[2] Nieman, G., Winter, H., Hörn, B.: Elementi di macchine, Est-Springer, 1983.

[3] Pilkey, W. D., Pilkey, D.F.: Peterson's Stress Concentration Factors, John Wiley & Sons, Inc., 2008.

[4] Strozzi, A.: Costruzione di macchine, Editore Pitagora, Italia, 1998.

[5] Lazzarin, P., Tovo, R., Filippi, S.: Campi di tensione da intagli laterali simmetrici in piastre di larghezza finita, IGF13, Cassino, 1997.

[6] Shigley, J.E.: Mechanical Engineering Design, McGraw-Hill, New York, 1986.

[7] Helsing, J., Jonsson, A.: On the computation of stress fields on polygonal domains with V-notches, International Journal for Numerical Methods in Engineering, (53) 433-453, 2002.

[8] Sors, L.: Fatigue Design of Machine Components, Pergamon Press, Oxford, 1971.

[9] *SolidWorks*, Dassault Systèmes SolidWorks Corporation.

[10] Zienkiewicz, O. C.: The Finite Element Method in Engineering Science, McGraw-Hill, London, 1971.

[11] Przemieniecki, J.S.: Theory of Matrix Structural Analysis, McGraw-Hill, NewYork, 1968.

[12] *CosmosWorks*, SRAC, SolidWorks Corporation.