APPLICATION OF FEM IN PRESSURE PULSE TEST SIMULATION FOR CHARGE AIR COOLER

Piotr TUTAK, Marcin KUBIAK

Czestochowa University of Technology, Institute of Mechanics and Machine Design Foundations, Czestochowa, Poland, EU, piotrtutak@wp.pl, kubiak@imipkm.pcz.pl

Abstract

This work presents an application of Abaqus simulation software based on Finite Element Method in pressure pulse test simulation to optimize design of Charge Air Cooler and prepare it for serial production release. The work explains step by step how the pressure pulse test simulation of Charge Air Cooler design is performed on a level of virtual validation and what kind of useful information it can provide to the development team. Most of all this work, based on the example of Charge Air Cooler, shows how a big role plays simulation software today and how it can improve the product and reduce development costs.

Keywords: FEM, simulation, charge air cooler, pressure pulse test

1. INTRODUCTION

Finite Element Method/Analysis [1] (FEM/FEA) is widely used in numerical modelling to solve simple and complex issues [2]. The pressure pulse test is the key test verifying the durability of products which will be in contact with the variable pressure of hot working medium during use. For a charge air cooler, the pressure pulse test consists of a cyclically varying pressure of air supplied to its inlet at a specific temperature and frequency. The test is considered as passed if the charge air cooler performs the number of cycles defined in the test specification, remains leak free, or depending on the specification with the leakage level within the permissible limit. In this study the model of the charge air cooler is designed in the NX Siemens software (**Figure 1**). It consists of an aluminum core to which plastic tanks are attached. The tightness between them is ensured by rubber gaskets. Tanks play a function of interface with other components of the car heat exchange system and its structural frame. The hot charge air is supplied to the CAC through inlet tank pipe.

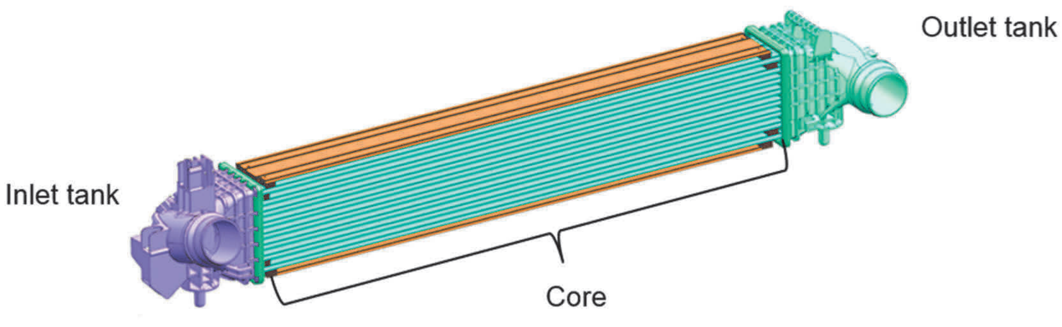

Figure 1 Charge Air Cooler CAD model

2. PRESSURE PULSE TEST SIMULATION

For pressure pulse test simulation several simplifications are implemented [3]. To minimize the computational complexity, a half symmetry model is considered built [2]. CAD model of charge air cooler is divided into two parts: inlet and outlet side and both sides contained only part of the core because all possible damages occurring during pressure pulse test take place in this specific area. Created CAD model is then exported to the Abaqus FEA software [4], where mechanical properties corresponding to working temperature of 200°C for each component of the charge air cooler are considered. The aluminum core is meshed with quadrilateral shell elements for tube, turbulator, air center, reinforcement and solid tetrahedral elements for the header. Plastic tanks are discretized by tetrahedron elements [2]. In the FEM model the degrees of freedom are assumed in the X, Y, Z axes for the core and in the Z axis for the tank pins which position the exchanger in the car frame (**Figure 2**).

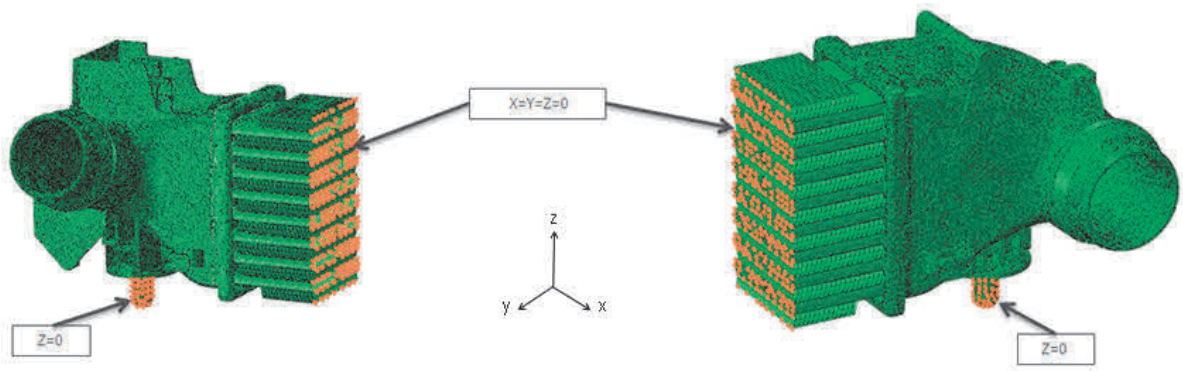

Figure 2 Charge Air Cooler FEM model

Load in the form of a constant pressure with an amplitude of 0.2 MPa 2 bar (g) is assigned inside each element of charge air cooler (**Figure 3**).

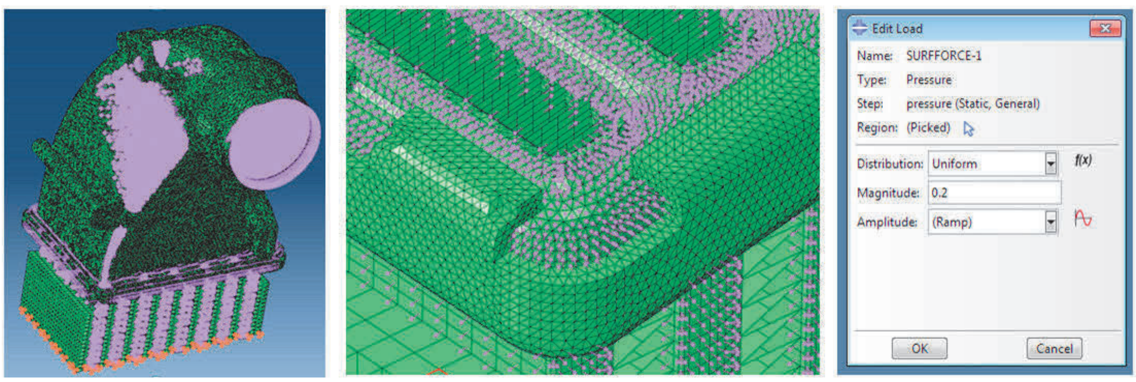

Figure 3 Boundary condition: pressure load

3. RESULTS AND DISCUSSION

Obtained results include principal stresses and average displacements in the system. The simulation indicates the location of maximum principle stresses [5], which are present on the radius of the upper tube of the inlet tank on the engine side (**Figure 4**) and the upper tube of the outlet tank on the cooling air side (**Figure 5**). According to product specifications the displacement of each tank should not exceed 0.5 mm. The limit of maximum principal stresses for the tube is set at the level of 65 MPa as a result of calculations and data from the real pressure pulse test which was performed for similar charge air cooler. In this case this limit has been exceeded for both sides of analyzed design of charge air cooler. **Figures 6** and **7** show the level of displacement on both tanks, respectively on inlet and outlet tank. On the inlet tank the maximum displacement level is 0.97 mm. The highest value of displacement on the level of 1.82 mm is present on the outlet tank. The level of maximum principle stresses in the case of the inlet tank exceeded the permissible limit of 45 MPa, reaching a maximum value of 55 MPa (**Figure 8**). A similar issue occurred also on the outlet tank where the maximum stress level calculated by program was 77 MPa (**Figure 9**).

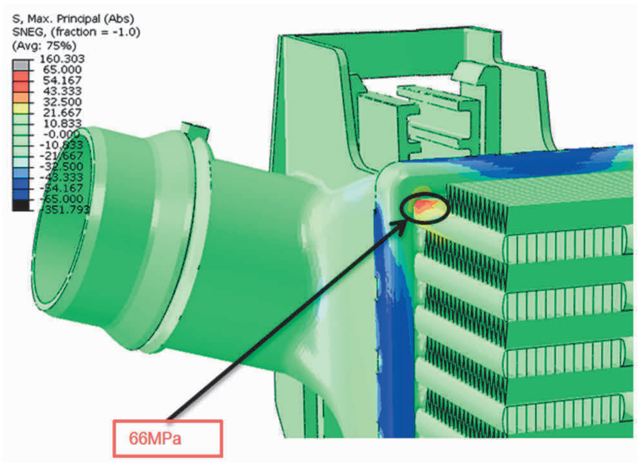

Figure 4 Inlet side: Max principal stress (MPa)

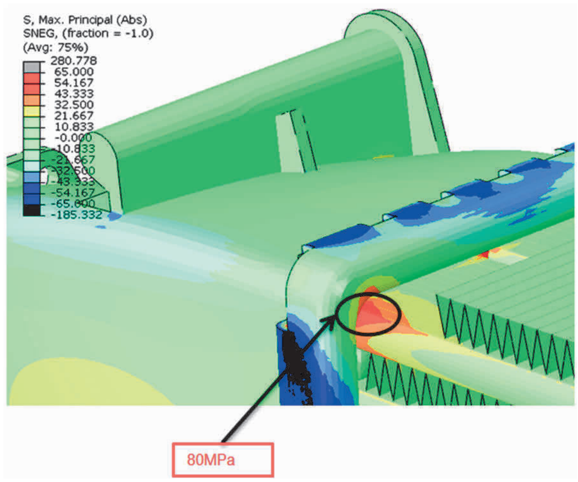

Figure 5 Outlet side: Max principal stress (MPa)

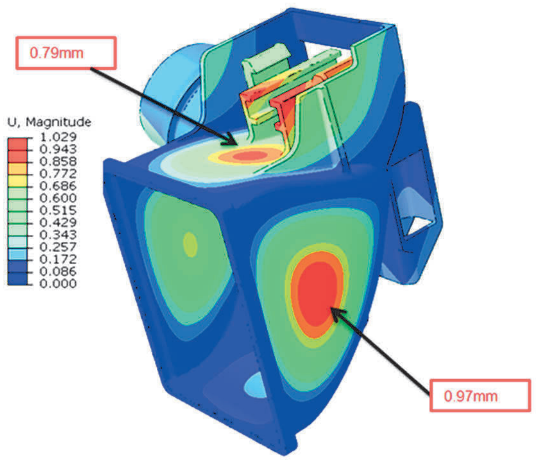

Figure 6 Displacement of inlet tank (mm)

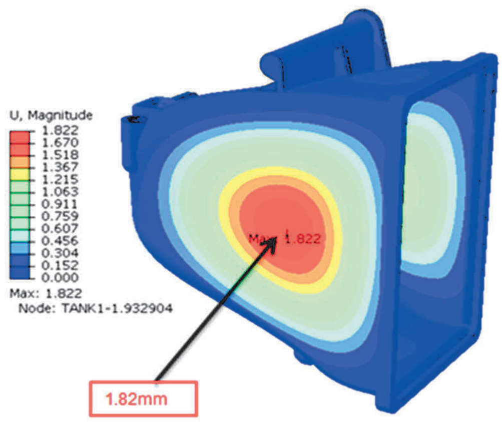

Figure 7 Displacement of outlet tank (mm)

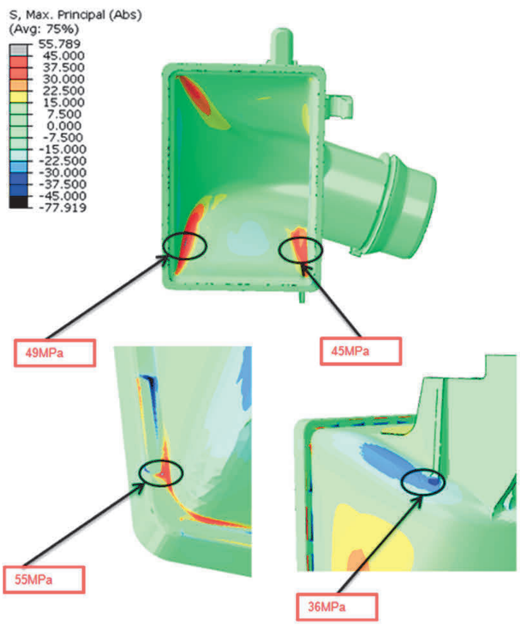

Figure 8 Inlet tank: Max principal stress distribution (MPa)

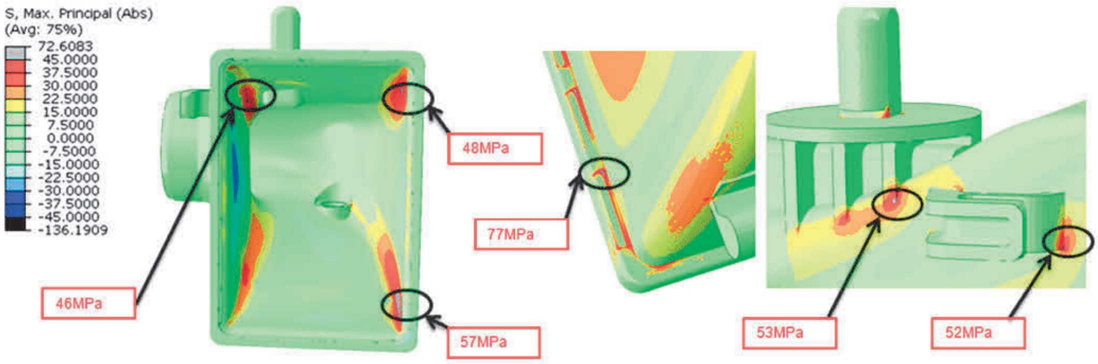

Figure 9 Outlet tank: Max principal stress distribution (MPa)

For all types of charge air coolers design like the analyzed one in this study, the main reason for failing the pressure pulse test is leakage from the tube as a result of the crack. The reason for the tubes cracking and the occurring leaks is the rotation of the header with which the tubes are connected by brazing. In turn, the movements of the header are the result of the tank pulsing due to the varying sinusoidal pressure of the hot air. Therefore, to prevent cracks and leakages of the tubes before reaching the required number of cycles, the inlet and outlet tank deformation need to be reduced by adding ribbing on their walls. The following example shows the simulation of the pressure pulse test for the analyzed charge air cooler, in which ribs on the external walls of tanks were introduced to reduce their deformation.

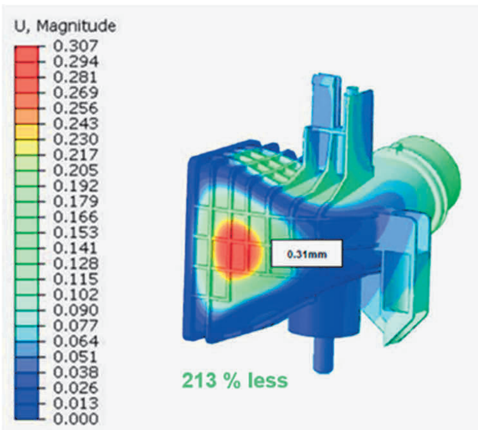

Figure 10 Displacement of inlet CAC tank (mm)

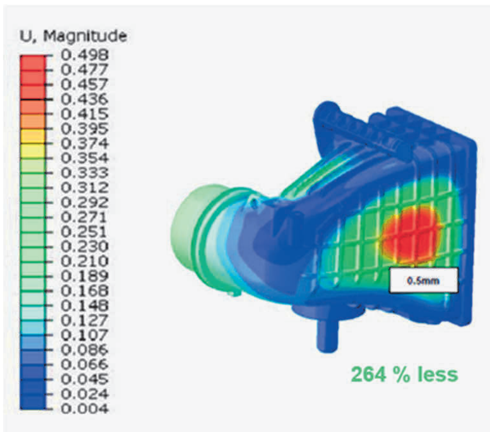

Figure 11 Displacement of outlet CAC tank (mm)

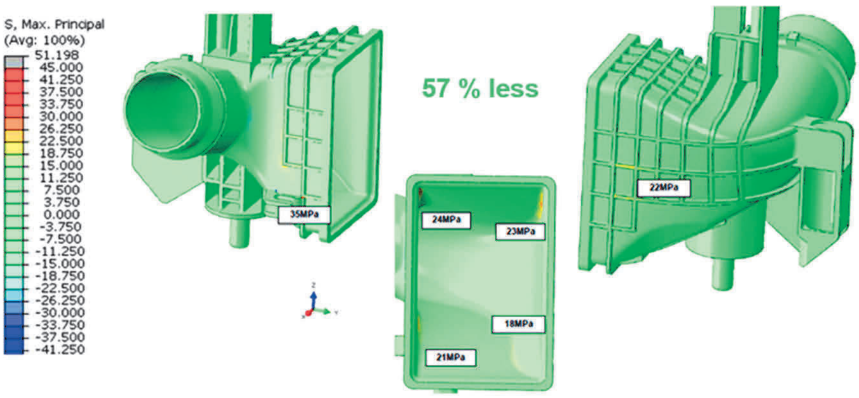

Figure 12 Inlet tank: Max principal stress distribution (MPa)

Adding ribs to its tanks gave the maximum displacement of the charge air cooler inlet tank on the level of 0.31 mm (**Figure 10**) and 0.50 mm of the outlet tank (**Figure 11**). Based on the results of the simulation, there is 213% decrease of displacement for the inlet tank and 264% for the outlet tank in relation to the maximum values of displacement in the baseline model. Adding ribs on the inlet tank reduced maximum principle stresses on inlet tank by 57% in relation to the maximum value obtained for the baseline design inlet tank (**Figure 12**). The stress reduction on the inlet tank automatically caused stress reduction occurring on the inlet side tubes by 164% (**Figure 13**).

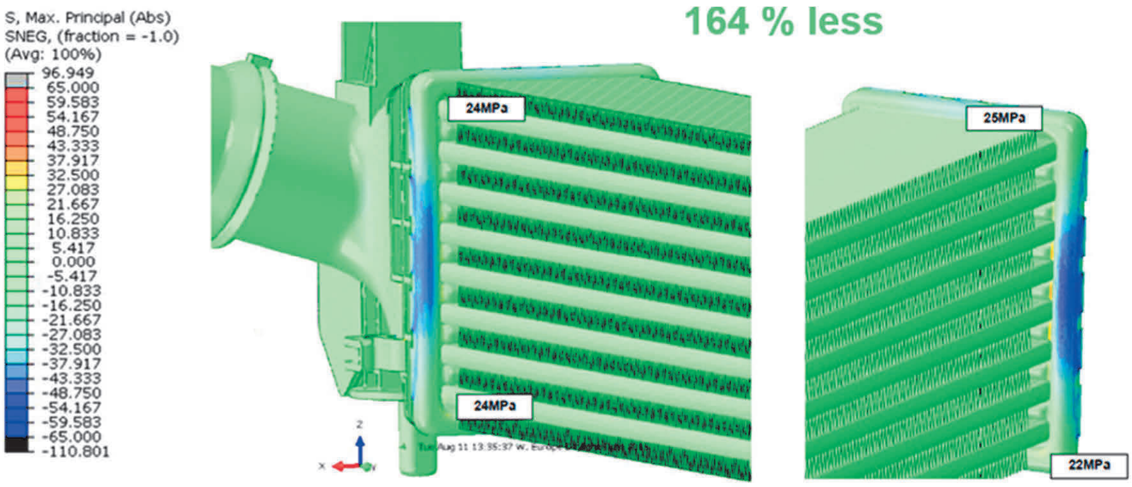

Figure 13 Inlet side tubes: Max principal stress distribution (MPa)

Figure 14 shows the calculated values of maximum principle stresses in the outlet tank, whose maximum value decreased by 40% in relation to the baseline model. The empirically determined stress limit for the material from which the tank was made is 45 MPa. This value was exceeded in some places of the tank. Therefore, the injection mold should be then modified as indicated by simulation. The area smooth radiuses and joint of ribs should be implemented.

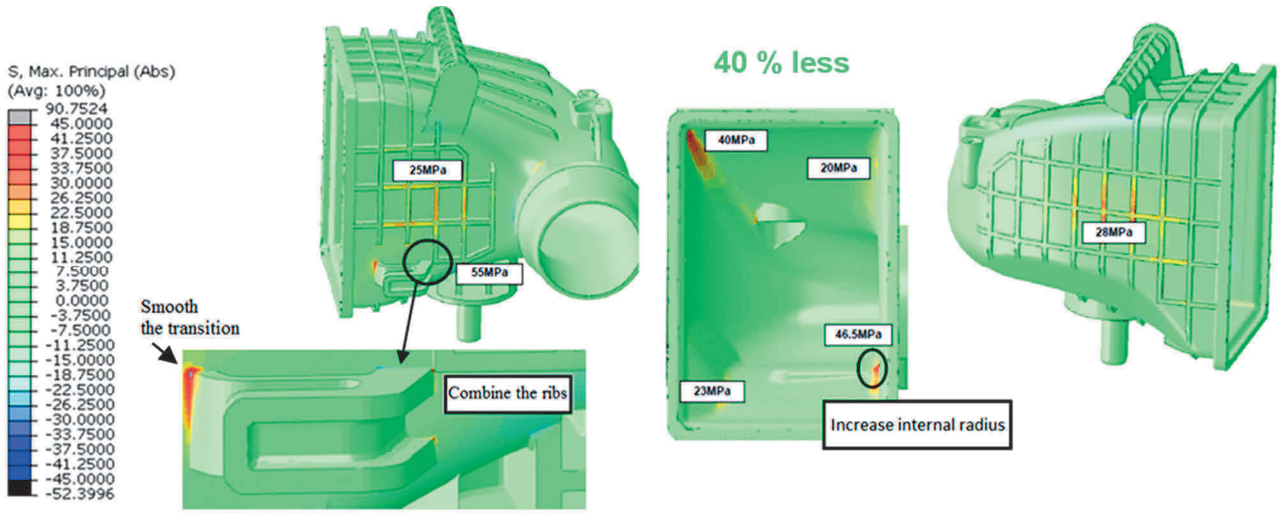

Figure 14 Outlet tank: Max principal stress distribution (MPa)

Identically as in the inlet tank, adding ribs to the outlet tank and fixing the radiator to charge air cooler positively affects the reduction of stress on the charge air cooler tubes on the outlet side by 135% in relation to the maximum value of the baseline design (**Figure 15**).

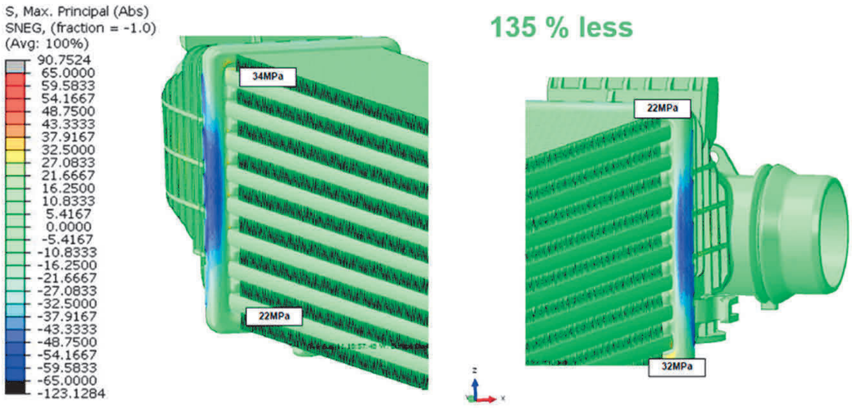

Figure 15 Outlet side tubes: Max principal stress distribution (MPa)

4. CONCLUSIONS

The presented work shows that structural calculations are rarely carried out once but this is a cycle of analyzes consisting of structural improvements introduced to construction based on previous simulation results, empirical material data and experience and consultation with the engineering team. For the presented pressure pulse test there is the possibility of implementing suggested changes in the mold and apply the optimal solution. Such activities are carried out until the result of the final simulation and then the physical test meet all set requirements. **Table 1** presents a comparison of stress level for charge air cooler components calculated for the baseline design and the design after implemented modification of tanks. A huge impact of the applied changes to the reduction of stresses in the design of analyzed charge air cooler can be observed.

Table 1 Impact of changes in design on the results of pressure pulse test simulation

CAC side	Component	Simulated physical quantity	Unit	Baseline CAC desing	Modified CAC design	Profit
Inlet	Tank	Displacement	(mm)	0,97	0,31	213%
		Max principal stress	(MPa)	55	35	57%
	Tube	Max principal stress	(MPa)	66	25	164%
Outlet	Tank	Displacement	(mm)	1,82	0,5	264%
		Max principal stress	(MPa)	77	55	40%
	Tube	Max principal stress	(MPa)	80	34	135%

REFERENCES

[1] BATHE, K. J. *Finite Element Procedures*. Prentice Hall, New Jersey, 1996.

[2] ROBIN, ROY P., HARIRAM, V. and SUBRAMANIAN, M. Fatigue life prediction of a commercial vehicle radiator under internal pressure cycling loading. *Indian Journal of Science and Technology.* 2016. vol. 9, p. 33.

[3] LI, C., FAN, S. and SHI, M. Preparation of CAD model for finite element analysis. In *International Conference on Computer, Mechatronics, Control and Electronic Engineering (CMCE).* Changchun, 2010. pp. 491-494.

[4] KHENNANE, A. *Introduction to Finite Element Analysis Using MATLAB and Abaqus*. CRS Press, 2013.

[5] MAOA, S. and CHENG, C., Thermal/structural analysis of radiators for heavy-duty trucks. *Applied Thermal Engineering.* 2010. vol. 30, pp. 1438-1446.