A Study of Durability Analysis Methodology for Engine Valve Considering Head Thermal Deformation and Dynamic Behavior

Kum-Chul, Oh[1], Sang-Woo Cha[1] and Ji-Ho Kim[1]

[1] R&D Center, Hyundai Motor Company

Abstract: Automotive engine valves draw the air and fuel into the cylinders and allow the exhaust gas out. The valves are under various loads during the engine running: a combustion pressure, a spring force, thermal loads and the impact force due to valve closing speed. Their combined loads sometimes lead a valve fracture. In this paper, the failure behavior in the exhaust valve is investigated with the finite element methods. An implicit time integration method is used to simulate the transient dynamic behavior of the valve. The stresses results by FEA were in good agreement with the measured data under a valve train rig test. The weak position by FEA is exactly matched with the failure position in the engine durability test by considering the three dimensional thermal deformation of the cylinder head. This study showed the durability of the valve neck, the most common failure position, is subject to various factors such as the partial contact due to the thermal deformation of the valve seat, the degradation of material strength due to high temperature, the dynamic load when the valve closes.

Keywords: Valve, Engine, Failure, Crack, Finite Element Analysis

1. Introduction

The valve train of the automotive engine is a device to control the operation of valves. It is closely related to the flow efficiency and the performance of the engine.

Since the recent trend requires a higher performance for an automotive engine, a larger section area in the cylinder port to get more air and the larger valve face to cover the port are designed and it leads a higher force on the valves. And the inertia force of the valves increases dramatically as the engine speed increases. The valve durability is subject to a variety of complex conditions such as thermal loads, the temperature-dependent material properties, a valve impact closing velocity, a partial contact between thermal deformed valve seat and valve, combustion pressure, and so on [1].

The various loads are combined and make repeat stresses on the valve during the engine running. A valve failure occurs when the accumulated fatigue damage to the valve is over the fatigue strength of the valve material. If the valve breaks, the engine loses its function and stops. Thus it is important to check the durability of the valve in advance. However, in general, the dynamic analysis to valve train in the initial design stage helps to set the design parameters, e.g. a cam profile and a valve timing and to check the valve performance such as a jump or a bounce of valve movement, a valve spring surging, a maximum valve acceleration, and etc.

Since another main function of the valve is sealing the gas in the cylinder the valve and the valve seat are machined and installed in the engine to have a uniform contact each other. However the contact between the valve and the valve seat becomes uneven because of the distortion of the valve seat due to the thermal load. This partial contact causes the stress concentrations in the valve neck when the valve closes and the combustion pressure acts, which is the most common failure in the valves. The previous investigations have been mostly done by the valve suppliers [2,3]. They used a simple valve system to evaluate the durability. Concerned are few components such as a valve and a valve seat with the thermal loads, temperature dependant material properties, a combustion pressure, and so on. But the three dimensional thermal distortion of the system is not considered yet.

This study involved the thermal distortion and the valve train dynamics analysis to improve the correlation of the simulation and the actual failure. The distortion of the system based on the thermal deformation of the head, the valve seat, and the valve guide by using the result of cylinder head analysis is considered. And the dynamic analysis with the overall valve-train system model under the actual engine operating conditions is also performed. The valve neck crack during the engine operation was reproduced by this analysis procedure. Finally a systemic approach for the valve durability is built here.

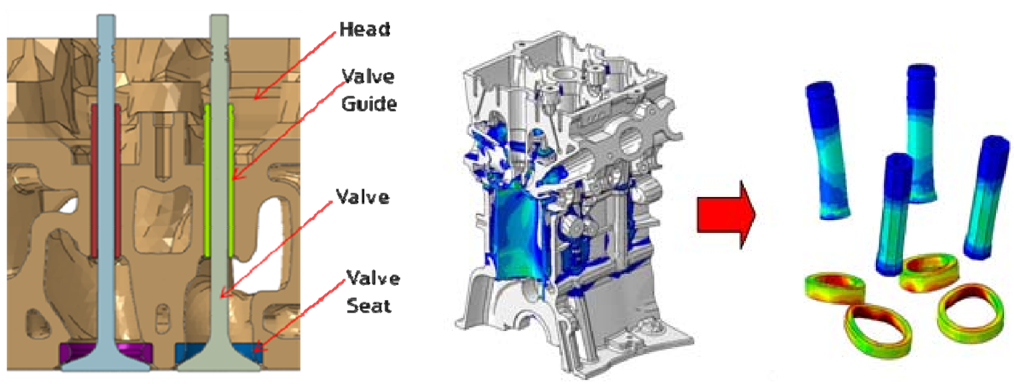


Figure 1. Thermal deformation of valve seat and valve guide

2. Engine Valve Durability Analysis

2.1 Engine Valve Neck Failure Mechanism

Figure 2 shows a typical valve neck crack reported in exhaust valves of a gasoline engine. The failure in valve neck of a gasoline engine is more likely than that of a diesel engine because of the higher engine speed and the higher exhaust gas temperature. A valve inertia force increases dramatically according to the engine speed and the material strengths of the valve are deteriorated rapidly in a high temperature condition [4,5].

When the exhaust valves open, the exhaust gas starts to flow out and hits the valve neck directly. The hot exhaust gas causes the oxidation in the valve neck and the degradation of material properties due to the high temperature accelerates the fatigue damage. If the valve is failed, the debris will travel in the combustion chamber or sometime go into the turbocharger and a secondary damage of the engine components (a piston, a head, a block, or a turbocharger) can be followed. The valve neck crack occurred in an engine durability test is displayed in Figure 2. The valves and the valve stems are divided and one of valve stems is bent. It is found that a micro crack is initiated in the valve neck surface due to the high-temperature oxidation and a fatigue crack propagates.

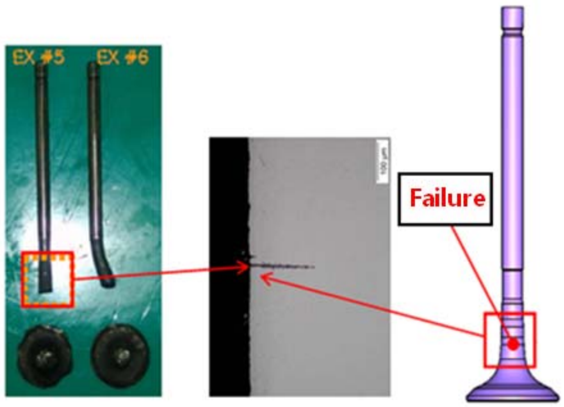


Figure 2. Exhaust valve neck failure

2.2 Model Configuration and Considerations

2.2.1 Material Properties

Exhaust valves are exposed in the high temperature corrosive environment more than intake valves. An austenitic steel SUH35 is mainly used for exhaust valves since it has material strength properties, a corrosion resistance, and a stable metal structure in high temperatures. In case of a very high performance engine, a nickel-base superalloy Inconel 1751 is sometimes used because of a better high temperature properties then SUH35.

In general a material strength becomes lower as a temperature goes up. After the temperature analysis, the valve neck which the exhaust gas flows though is calculated as a maximum temperature point. Hence the valve neck is under the worst condition (the lowest material properties and the highest oxidation) and a crack is easy to initiate than other area.

The valve durability in this paper is evaluated in terms of safety factors with the material properties shown in Figure 3 [4,5].

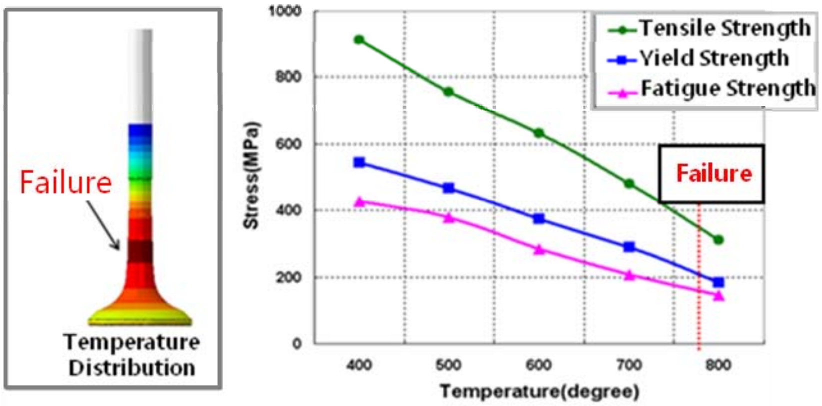


Figure 3. Temperature dependent material strength : SUH35 [4,5]

2.2.2 Dynamic Durability Analysis

To consider the dynamic behavior such as the inertia of the valve and the impact of the valve to the valve seat when the valve closes, a dynamic analysis is performed. Figure 4 shows the finite element model for the dynamic durability analysis of the valve. An implicit time integration method is used with the commercial software Abaqus in the dynamic analysis [6].

All the components in a valve train system are involved: cam, follower, HLA (Hydraulic Lash Adjuster), valve, retainer, spring, valve guide, and valve seat. The dynamic behavior of the valve was simulated by the rotation of the cam with the engine rated speed and by the contact between the parts (cam and follower, follower and valve). The cam is modeled with the actual cam profile. The hydraulic characteristic of HLA is replaced with a spring-damper system. The stiffness of the valve spring is reflected by adjusting the elastic modulus of spring model. The initial coordinate valve guide and valve seat are modified by using the deformed shape from the result of cylinder head analysis that had been conducted in advance. The contact conditions are applied to contact surfaces between each part with the coefficient of friction 0.2. And the softening contact option of Abaqus is applied in order to improve the convergence of the dynamic analysis and also to consider lubrication effect between the parts.

2.2.3 Thermal Deformation of Cylinder Head

If the valve components (valve, valve guide, and valve seat) are exactly aligned in straight together at the engine operating conditions, the contact between the valve and the valve seat will be axisymmetrically equal. Then the highest stresses will be presented around the valve head and the distribution will be circumstantially uniform. But, the partial contact is not avoidable because of the thermal distortion of the head and the stress concentrations in the valve neck due to the bending when the valve closes and the combustion pressure acts. The consideration of the thermal deformation is quite important to evaluate the durability analysis. For that, not only the valve components but also the head and block system are modeled and their thermal deformations are calculated in this study. The FEA model used here and its temperature distribution and thermal deformations are displayed in Figure 1. Then the thermal deformation results of the valve components are transferred as the initial conditions of the valve components in the dynamic durability analysis, which is explained in the previous section.

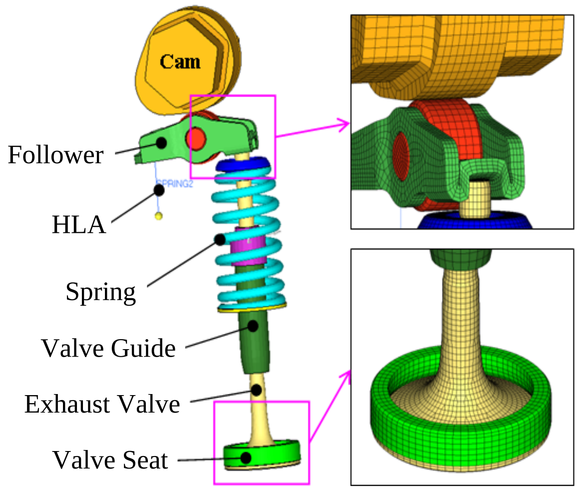


Figure 4. FE model of valve-train

2.2.4 Load Conditions and Analysis Procedures

The spring preload was adjusted by controlling the initial displacement boundary condition of the valve spring. The combustion pressure is applied to the bottom side of the valve head and the timing of combustion pressure was controlled by the cam angle data in the analysis. The thermal load of the valve due to the high temperature distribution is applied as well. The schematic procedure proposed here for the valve durability evaluation is shown in Figure 5.

At the first step, the temperature distribution of the exhaust valve is calculated by considering the thermal interaction between the valve and the valve seat with the exhaust gas temperature of the full load condition. And the thermal distortion of the valve components are prepared as explained before. These data is transferred to dynamic analysis model in the next step. The dynamic analysis is performed in the second step. Using Abaqus, the assembly analysis applying the preload of spring and the dynamic analysis of valve movement under the cam rotation and the combustion pressure are performed in sequence. As a result, the stress history of valve is obtained. Finally safety factors are calculated in the last step by using the combination of stress history of valve and temperature-dependent material strengths.

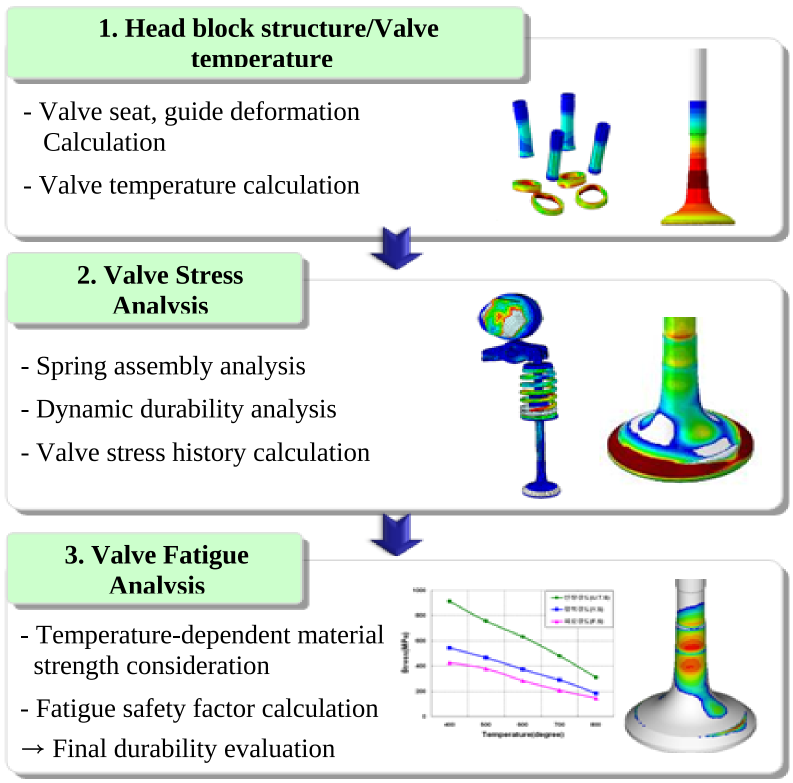


Figure 5. Procedure of valve durability analysis

2.3 Procedure Verification

A verification test was performed to check the reliability of the analysis proposed here. Figure 6 shows the motoring rig test device and the location of the strain gauge for strain measurement. The strains on the valve neck and the follower bottom were measured. The test was conducted under the motoring rig test without engine firing because of the difficulty of measurement in the actual full load endurance test. That means that the verification test is done without the thermal effect such as the thermal deformation of the valve seat and valve guide, and combustion pressure. Even though the thermal effect is excluded, it is enough to check the reliability of the dynamic analysis results. The strain comparisons of the analysis to the measurements are shown in Figure 7. At the follower bottom, two strain plots in the cam ramp region are different (Figure 7a). The reason is expected that the mesh of finite element is failed to describe the actual cam profile in the ramp section. But, the trend and magnitude of the strain of the analysis are quite similar to the measurement results in the valve opening and valve closing regions. And, the trend and magnitude of the strain of the analysis are quite similar to the measurement results in the valve neck too (Figure 7b). It means that this analysis is a reliable.

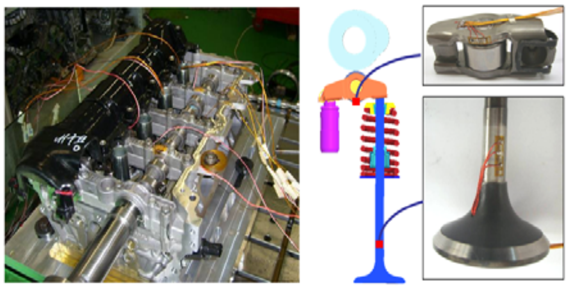
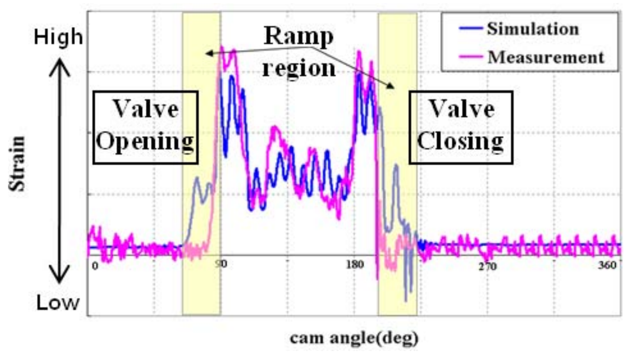
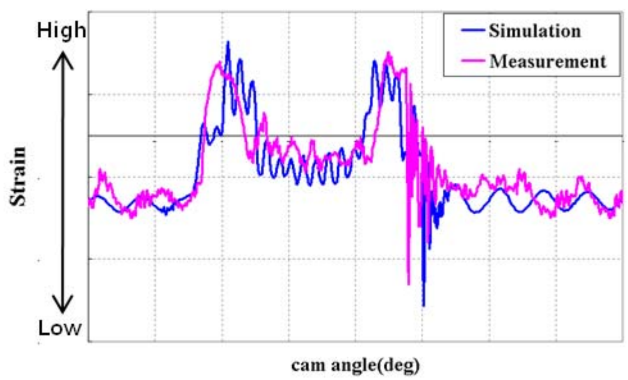


Figure 6. Motoring rig test device and locations of strain gauge for measurement



(a) The bottom of follower



(b) Valve neck

Figure 7. Comparison of strain analysis and measurement

2.4 Analysis Result

2.4.1 Valve Temperature Distribution

The temperature field analysis of the exhaust valve is carried out to apply the thermal load to the valve for dynamic stress analysis. As a result, the heat transfer percentages of each part are shown in Figure 8. The heat generated by exhaust gas that is the main heat source to the valve. The 59% of the heat generated is absorbed at flame contact side (❶) i.e. the bottom of valve. When the valve is open, the exhaust gas passes thorough the valve. In case, the 24% of the heat is absorbed at the upper surface (❷) of the valve and 16% is at the valve neck (❸). Absorbed heat is cooled by the valve seat and valve guide that contact to the valve, but the most of the heat is released through the valve seat. Because of the heat release to the valve seat, the temperature level at the seat contact (A) is lower than the valve seat even though they face directly the exhaust gas.

It is easily seen in Figure 9 that the maximum temperature is indicated at the valve neck where the failure is occurred. This high temperature deteriorates the material strength, displayed in Figure 3. So, it is quite possible that weak point is formed. The results of exhaust valve temperature analysis were applied to thermal load of stress analysis.

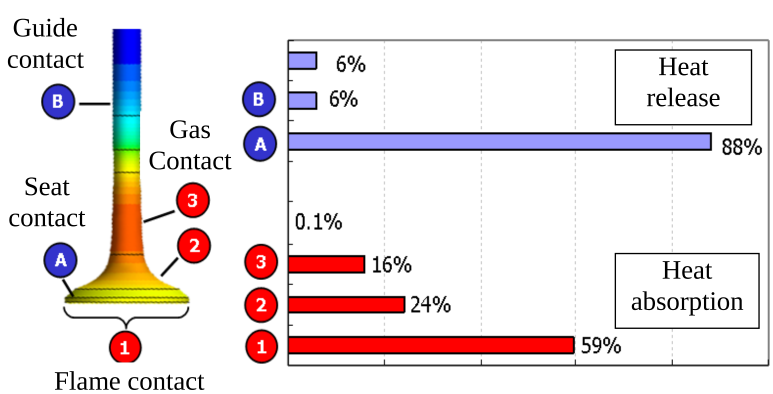


Figure 8. Heat transfer percentages at each parts of exhaust valve

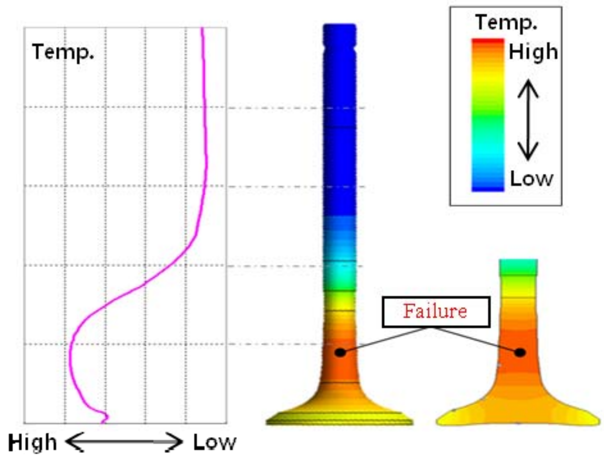


Figure 9. Temperature distribution of exhaust valve

2.4.2 Valve Stress Distribution and Comparison

Dynamic stress analysis was carried out for two conditions (with and without thermal deformation of valve seat and valve guide) to understand the effect of the partial contact between the valve and the valve seat. The maximum principal stress distributions of two cases are summarized in Figure 10. The very basic state is that the cam profile is a circle and only thermal load is applied to the valve. In this case, the circumferential stress distribution is the same regardless of the seat deformation. And the stress on the valve neck is a low level, which the fail is occurred at. The expansion of the flame contact area makes a high tension stress in the seat contact area since the temperature of the flame contact is much higher than that of the seat contact area.

When the maximum combustion pressure is applied just after the valve closed, i.e. the moment of the explosion, the stress distribution under the thermal deformation shows a different tendency to the distribution not considering the thermal deformation. When the thermal deformation is not involved, there is no partial contact. Therefore the stress distribution under the thermal load and the combustion pressure is axisymmetric as similar as the stress under only the thermal load. But in case of the thermal deformation considered when the combustion pressure is added, the stresses at valve neck and valve head increase much more than the level under only the thermal load. Especially, a maximum stress occurs at the valve head. This indicates that a tensile stress due to a bending occurs at the valve head and the bending is from the partial contact of valve and the combustion pressure at the bottom of valve. Finally at the moment of the valve close under the consideration of the thermal deformation, the bending is much intensified by the inertia effect and the partial contact. The stress at the valve head is similar to the stress under the combustion pressure condition, but the stress on the valve neck goes up to high level where the failure occurred.

This phenomenon can be explained with the load type. In other words, the combustion pressure applied at the bottom of the valve makes the large bending at the valve head, which is the most nearest location. When the valve is closing the inertia force is applied upward to the whole valve. Then the large bending occurs at the valve neck having a smaller cross section than the valve head. From the results, these are found that the momentary maximum stress occurs at the valve neck and the maximum temperature presents at the same position. Consequently it is expected that the valve neck will be very weak in fatigue.

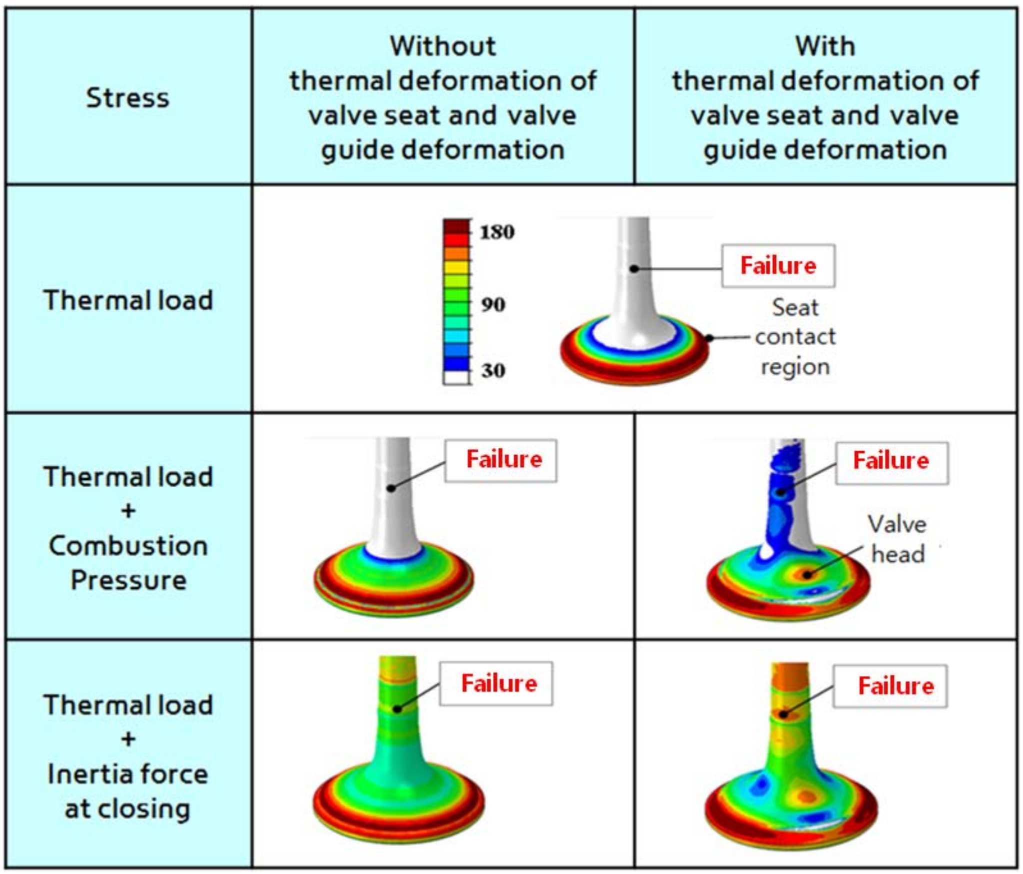


Figure 10. Stress distribution comparison

2.4.3 Fatigue Durability Evaluation: Safety Factor

The fatigue safety factors are calculated by using the stress history computed in the dynamic analysis and temperature-dependent material strength data of Figure 3. The safety factor is an indicator to evaluate the fatigue durability for an infinite life cycle. It can be expressed as the ratio of stress amplitude to the material fatigue strength. If the stress amplitude is larger or the temperature is higher, the fatigue durability becomes lower. The safety factors of the valve are displayed in Figure 11. Considering the partial contact, the valve neck shows a spot of low safety. It is lower about 21% than the condition without considering thermal deformation.

From the FE analysis procedure developed here, the weak position of the valve is expected in terms of safety factors considering the various factors such as a partial contact due to the thermal deformation of the valve seat, the degradation of material strength due to high temperature, and the dynamic load when the valve closes. The expected position exactly agrees with the failure position in the engine test.

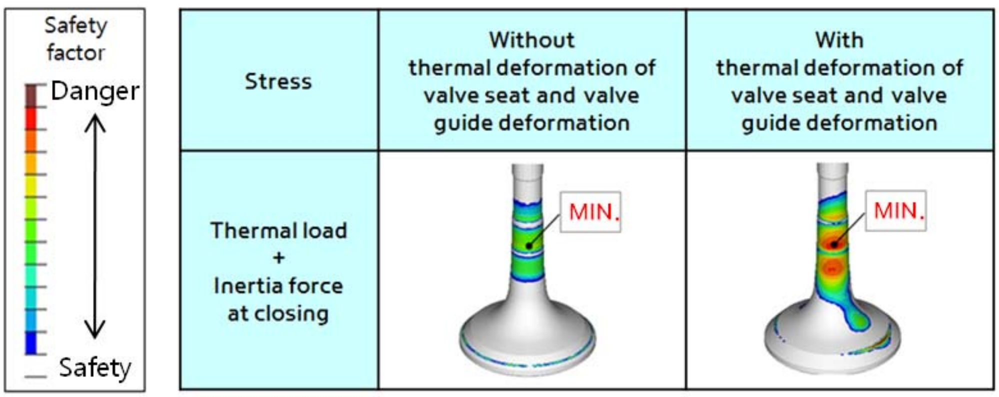


Figure 11. Fatigue Safety factor distribution

3. Conclusion

A durability analysis procedure of the valve was developed here with the study on the exhaust valve neck failure of a gasoline engine. The summarized are as follows:

1) In order to realize the actual condition of the partial contact between a valve and a valve seat in engine operation, the thermal distortions of the valve components (valve, valve seat, and valve guide) are considered in the valve durability analysis. The thermal deformations are obtained by a head and block FE analysis under high temperature condition.
2) The dynamic stress analysis of valve train is performed to involve the impact effect of the valve to the valve seat when the valve closes. The failure phenomenon is reproduced with the thermal deformation and the dynamic analysis.

3) The reliability of analysis model developed in this study was verified by the comparison between the strain measurement and the analysis results of the valve and the follower under motoring rig test.
4) It is found that the durability of the valve neck is subject to various factors such as a partial contact due to the thermal deformation of seat, the degradation of material strength due to high temperature, and the dynamic load when the valve moves.

4. References

1. G. Roth, "Fatigue Analysis Methodology for Predicting Engine Valve Life," SAE 2003-01-0726, 2003.
2. G. Roth, "Simulation of an Engine Valve Stress/Strain Response During a Closing Event," SAE 2003-01-0727, 2003.
3. M.L. Pang, S.P. Smith, and B. Buuck, "Stress Analysis of an Automotive Engine Valve by FEM," SAE 2006-01-0017. 2006.
4. K. Sato, T. Saka, T. Ohno, K. Kageyama, K. Sato, T. Noda, and M. Okabe "Development of Low-Nikel Superalloys for Exhaust Valves," SAE 980703, 1998
5. S. Umino, A. Hamada, T. Kenmoku, and Y. Nishizawa, "New Fe-base Exhaust Valve Material for Higher Heat Resistance," SAE 980704, 1998
6. Abaqus/CAE 6.11 User's Manual, Dassault Systèmes Simulia Corp., Providence, RI, USA.