

Development of a Strength Prediction Technique for Automotive Exhaust Systems

Takumi TAKAKAMO*

Masahiko HIGUCHI*

Hiroshi NAKAGAWA*

Youtarou UKAI*

Ayumu ISHIZUKA*

ABSTRACT

A technique has been devised for predicting the strength of an automotive exhaust system in relation to the rotational force input from the torque reaction force of the engine.

Conventional static analysis conducted using the Finite Element Method (FEM) has been replaced by techniques for dynamic analysis that treat the force input accompanying large transient displacement as a forced displacement, and techniques that take into account the nonlinear nature of mounting rubbers, flexible joints, and so on. As a result, it has become possible to predict the strength of exhaust systems with greater accuracy, and to apply these techniques as development support tools.

1. Introduction

Although the development period for new model vehicles is decreasing, there are major demands for improved quality, decreased weight, lower cost, etc. Utilizing CAE is an effective means for meeting these needs in the development process, and the Finite Element Method (FEM) is generally used as a method for analyzing the vibration and strength of automotive components.

The exhaust system is subject to vibration from the road surface, vibration from the engine, and engine rotation while the engine is running; in the front wheel drive vehicle, the source of the greatest stress input is this engine rotation while the engine is running. Further, because the input displacement from this stress is large, the response of the exhaust system exhibits nonlinear characteristics. The input mode also has transient characteristics. These points are extremely important in predicting the strength of the exhaust system.

This paper describes how we applied NASTRAN transient response analysis and expressed component characteristics in terms of two opposing forces equivalent to action and reaction, developed a means for predicting transient stresses on an automotive exhaust system by applying nonlinear characteristics as a displacement/load map, and how we were able to apply these technologies as strength development tools.

2. Modeling a Nonlinear Exhaust System

The means for calculating the nonlinear strength of an exhaust system used in this paper is based on a linear vibration model, with the addition of select items that are essential for strength analysis. Table 1 describes the differences between our model and the base model.

Since it is necessary to determine the engine rotational

Table 1 Differences between vibration prediction FE model and nonlinear stress prediction FE model

		Vibration FE model	Stress FE model
Estimation value		Eigen value MAC value	Displacement response Stress
FE model basic element		Shell	Shell+solid
Displacement area of input		Small	Large
Restriction condition	Mounting rubber	Elastic numerical value of spring	Displacement-Load nonlinear MAP
	Flexible joint	Elastic numerical value + of spring Shape model	Displacement-Load + nonlinear MAP Shape model

input in the time domain, we elected to use transient response analysis. For the input conditions, we used engine rotational input equivalent to what would be generated while driving for the forced displacement, derived using the Lagrange multiplier method.

In the model, two opposing forces are represented in an action/reaction relationship. Because components that have a displacement-limiting structure, such as flexible joints and mounting rubbers as shown in Fig. 1, exhibit nonlinear characteristics in response to a large displacement input, the spring characteristics of such components are represented by two relative forces in an action/reaction relationship based on a linear numeric spring (vibration model). In addition, the model uses a pseudo-nonlinear spring with the nonlinear characteristics given as a displacement/load map ("nonlinear map," hereafter).

One feature of this method is that the pseudoforce method is used to model components that have a nonlinear spring. This makes it possible to represent the nonlinearity of the exhaust system materials within the scope of linear analysis without a major increase in analysis resources.

* Tochigi R&D Center

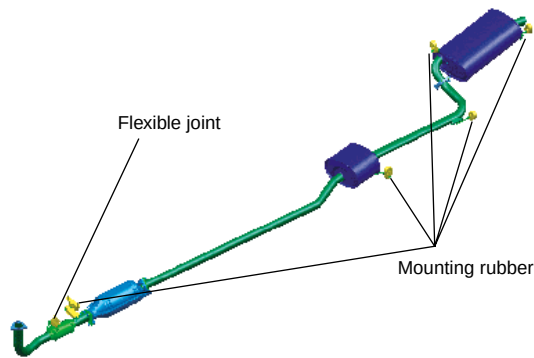


Fig.1 Nonlinear components of an exhaust system

3. Nonlinear Phenomena in an Exhaust System

3.1. Nonlinear Components in an Exhaust System

Two components in an exhaust system that exhibit nonlinear characteristics are shown in Fig. 2: the mounting rubbers and the flexible joint.

Fig. 3 shows the nonlinear map of a mounting rubber, and Fig. 4 shows the nonlinear map of a flexible joint. The flexible joint in particular exhibits strong nonlinearity.

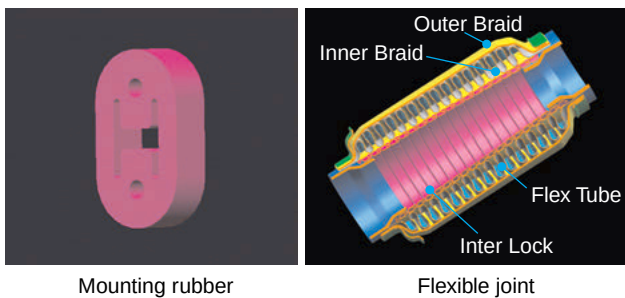


Fig.2 Nonlinear elements

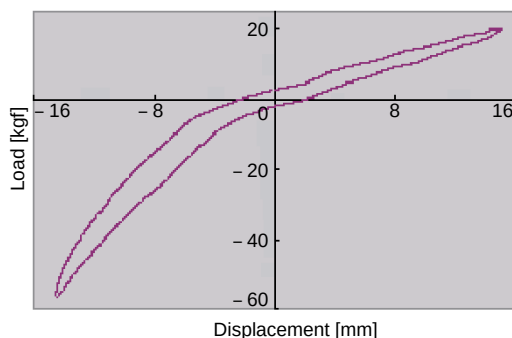


Fig.3 Hysteresis curve (Mounting rubber)

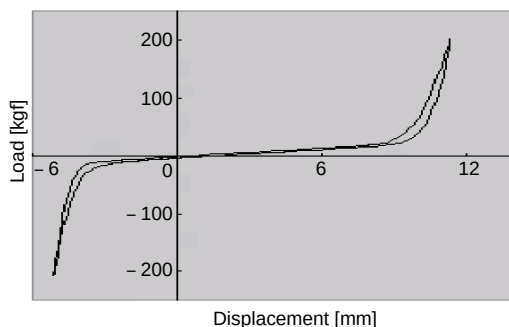


Fig.4 Hysteresis curve (Flexible joint)

3.2. Analysis of Nonlinear Phenomena

In order to understand the effect that the nonlinear characteristics of these components have on the behavior of the exhaust system, we conducted a vibration test in which we applied rotational input equivalent to that generated in an actual vehicle to the exhaust system through a hydraulic shaker, as shown in Fig. 5.

Fig. 6 shows the change in displacement behavior in the vertical direction of the end of the exhaust mounting points when the mounting rubber displacement stopper has been removed. Fig. 7 shows the horizontal displacement of the rearmost joint end when the outer braid and interlock components (the components that demonstrate nonlinear characteristics) are removed from the flexible joint, leaving only the tube.

These figures show that in large displacement areas, the nonlinear characteristics of components have a major effect on the displacement behavior of the exhaust system as a whole. Accordingly, the modeling of components that have nonlinear characteristics, such as mounting rubbers and flexible joints is an extremely important element in predicting the strength of an exhaust system.

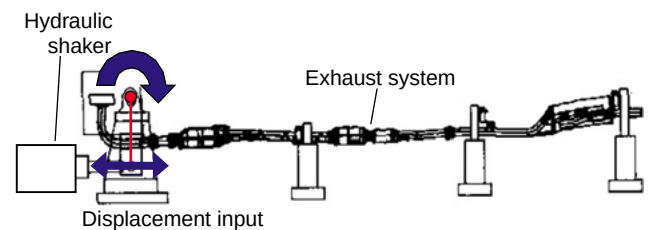


Fig.5 Rig test

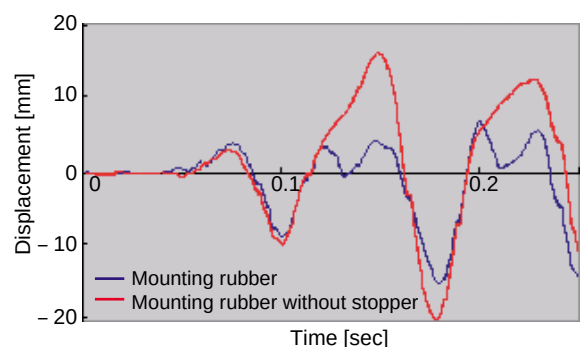


Fig.6 Influence of mounting rubber stopper

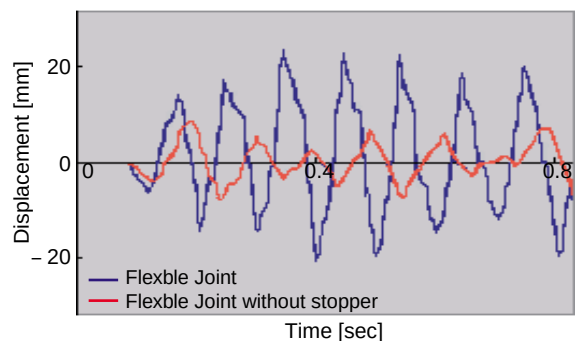


Fig.7 Influence of flexible joint stopper

4. Modeling Techniques

4.1. Overview of the Linear Model and Its Precision

Fig. 8 shows the Finite Element (FE) model that we used for the basic exhaust system (vibration model).

We used shell elements to model all exhaust system components that permitted shape modeling, such as the pipes and mufflers, and used solid elements only in the vicinity of welds.

Fig. 9 shows a comparison of the model with the frequency response level (up to 100Hz) generated by shaker input obtained in experimental results. The model matches the experimental results closely.

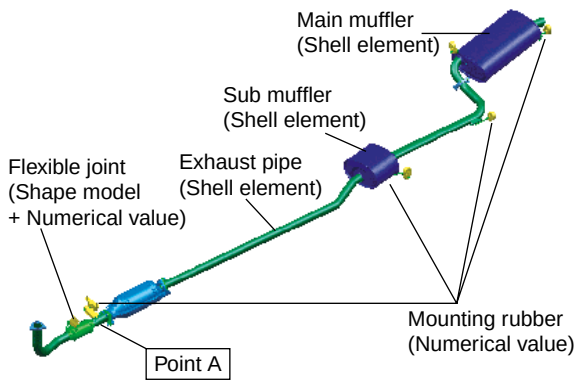


Fig. 8 Exhaust vibration prediction model in linear area

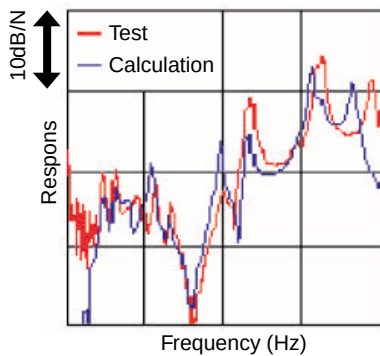


Fig. 9 Results of the frequency response test (Point A)

4.2. Mounting Rubber Modeling

4.2.1. Overview of the Model

In order to understand the characteristics in the strength range, the mounting rubber modeling technique involved testing the spring characteristics in a wide displacement

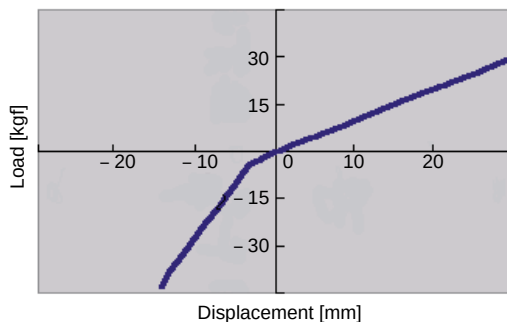


Fig. 10 Nonlinear map of mounting rubber

amplitude area. We then created a spring characteristics map (Fig. 10) by performing linear approximation for both the linear area and the nonlinear area of the displacement-load curve that was obtained from testing (Fig. 3). Attenuation characteristics were also identified from the curve in Fig. 3.

4.2.2. Model Verification Results

Fig. 11 shows the results calculated for the standalone characteristics produced by the above models. The calculated results closely match the standalone test results.

Dynamic verification was performed by setting one mounting point at the end of the system and creating a straight pipe exhaust model (Fig. 12) supported by mounting rubbers, and then comparing the calculated results with the results produced by a rig test. Fig. 13 shows the results of a comparison of vertical displacement in the mounts as a time series. The calculated results correspond closely with the results of the rig test.

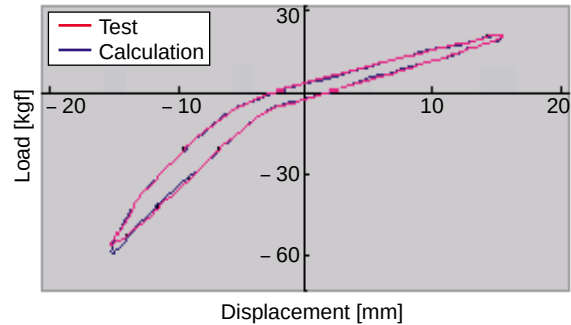


Fig. 11 Hysteresis curve (Mounting rubber)

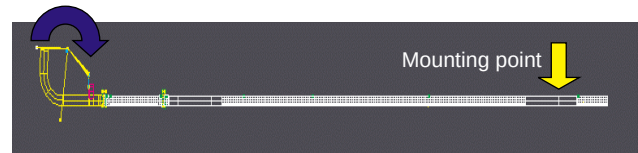


Fig. 12 Simple exhaust FE model

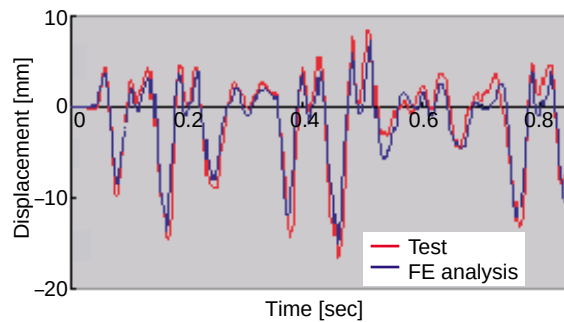


Fig. 13 Response in the vertical direction at the mounting point

4.3. Flexible Joint Modeling

4.3.1. Overview of the Model

In modeling the flexible joint, we used a shape model for the flexible tube as shown in Fig. 14, and we used a nonlinear map similar to that used for the mounting rubbers for other elements.

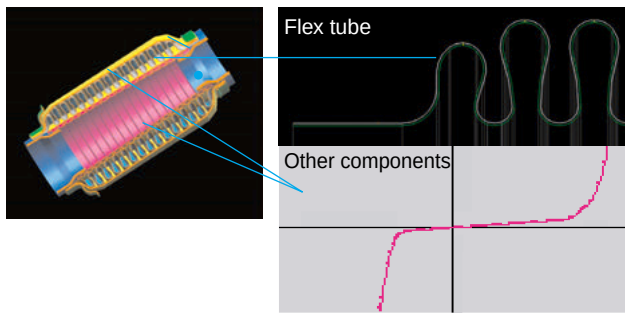


Fig.14 Flexible joint model

4.3.2. Flexible Joint Relative Displacement Calculation Technique

In order to use a nonlinear map to identify counterforces that are generated in the flexible joint and mounting rubbers, it is necessary to derive the amount of relative displacement of each component corresponding to the nonlinear map.

In the case of a mounting rubber, because the body side is regarded as a fixed point, as shown in Fig. 15, the displacement of the movable point of the exhaust side becomes the relative displacement of the mounting rubber.

On the other hand, in the case of a flexible joint, because both ends are movable points, it is necessary to derive the

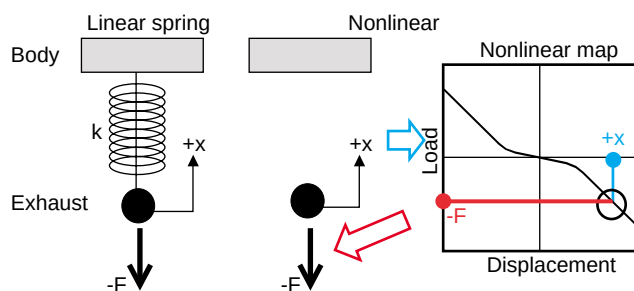


Fig.15 Load reference method for mounting rubber

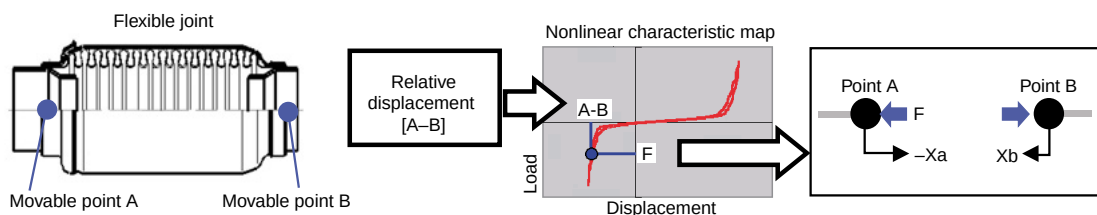


Fig.16 Relative displacement in the flexible joint

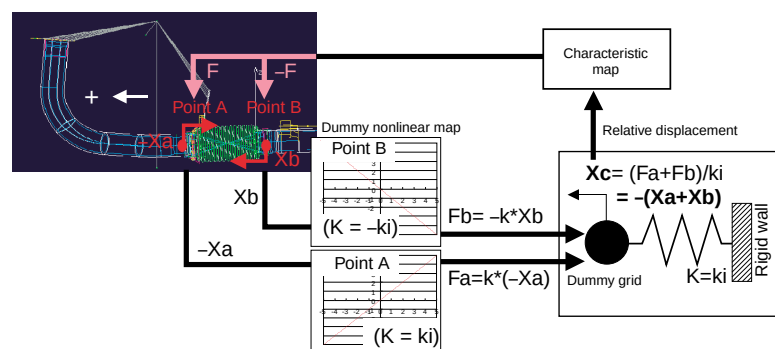


Fig.17 Dummy grid method

relative displacement from the difference in movement between the two points. However, it is difficult in NASTRAN to extract the relative displacement directly and to mathematically calculate the difference in movement from the amount of displacement of each point. Therefore, we proposed a method (the “dummy grid technique”) in which we would establish suspended structural nodes in the same analysis space as the exhaust system analysis model in order to calculate the relative displacement of both ends of the flexible joint, and then extract the relative displacement through the suspended structure linked to the ground by linear springs.

Fig. 17 shows an overview of the dummy grid technique. First, we created a dummy grid with a single degree of freedom that was independent from the exhaust system analysis model, and then linked that grid to the ground via a linear spring. Furthermore, the forces corresponding to the displacement of the movable points in front of and behind the flexible joint were applied to this system simultaneously. As a result, the displacement of the dummy grid becomes proportional to the relative displacement of the movable points in front of and behind the flexible joint, and generating forces in accordance with the dummy grid displacement become equivalent to generating forces in accordance with the relative displacement of the movable points in front of and behind the flexible joint.

4.3.4. Flexible Joint Model Verification Results

For the purpose of dynamic verification, we created a simple model (Fig. 18) that includes a flexible joint and then compared the results produced by a rig test and by calculation. Fig. 19 shows the results of a time series comparison of the relative displacement (in the vertical and horizontal directions) of the front and rear ends of the flexible joint. The results produced by calculation compare well to the results of the rig test, which means that it is possible to predict complicated flexible joint behavior through this modeling technique.

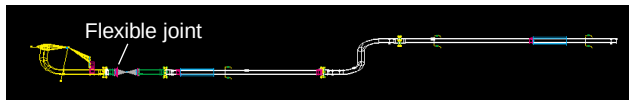


Fig.18 Simple exhaust FE model with flexible joint

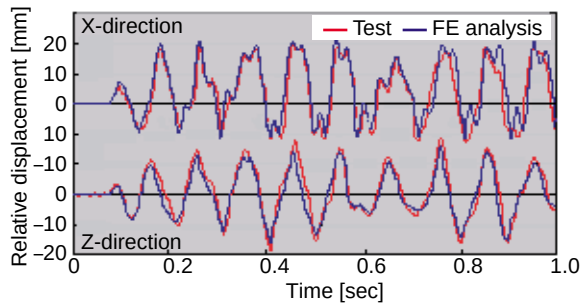


Fig.19 Relative displacement of flexible joint

5. Comparison of Results with Actual Tests

We applied all of the modeling techniques for each of the elements described above to an overall model of the exhaust system, then used the rig test apparatus shown in Fig. 20 to apply rotational input equivalent to that which would be generated in a real car, and compared the rig test results with the calculated results.

Fig. 21 shows a point-by-point comparison of the stress values predicted by the calculations and the stress values actually measured in the rig test. The trends in stress distribution obtained by the two methods match, with a high correlation coefficient of 0.96 obtained.

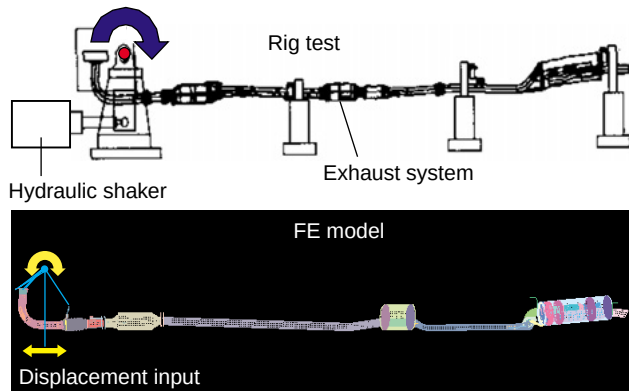


Fig.20 Rig test and FE model

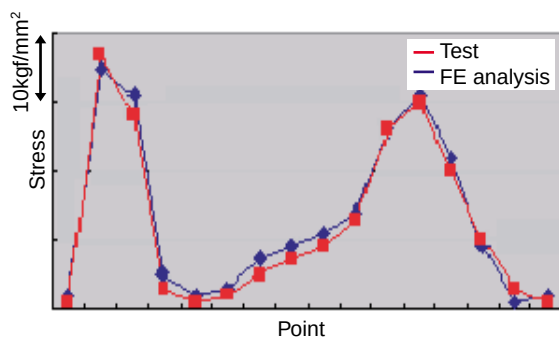


Fig.21 Exhaust system stress distribution

Figs. 22 and 23 show examples of the calculated results. The calculated results make it possible to understand the stress behavior corresponding to the displacement behavior of the flexible tube and the exhaust system as a whole, an understanding that is difficult to obtain experimentally.

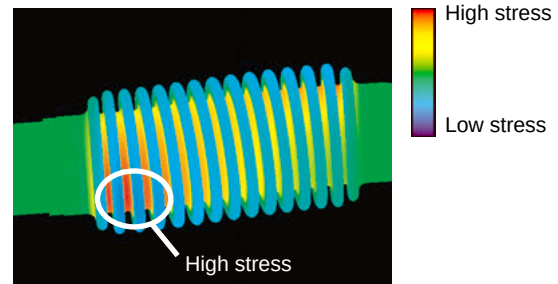


Fig.22 Flexible tube behavior

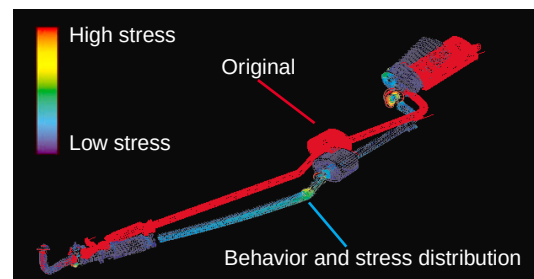


Fig.23 Exhaust pipe behavior

6. Case Study of Application

CAE is an effective tool for predicting stress, making small layout changes, conducting parameter studies, etc., during the drawing stage.

We predicted the differences in the stresses generated in the bellows with the flexible joint positioned in two different arrangements, Type A and Type B, as shown in Fig. 24.

The calculated results are shown in Figs. 25, 26, and 27. Figs. 25 and 26 show the behavior when the maximum stress is generated in the flexible joint, and Fig. 27 plots the stress values in that situation. These results show that Type B, with the flexible joint positioned in the front, is superior to Type A, which exhibited high stress due to the shearing mode. Fig. 28 shows that similar trends were obtained through rig tests, verifying the validity of the calculations.

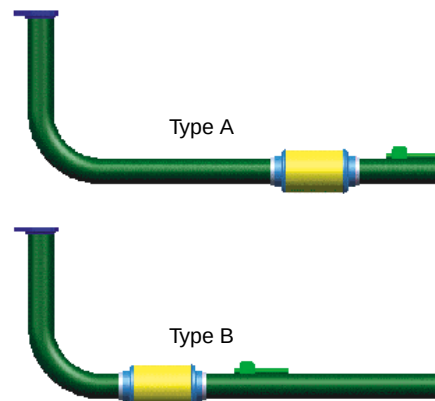


Fig.24 Comparative exhaust model

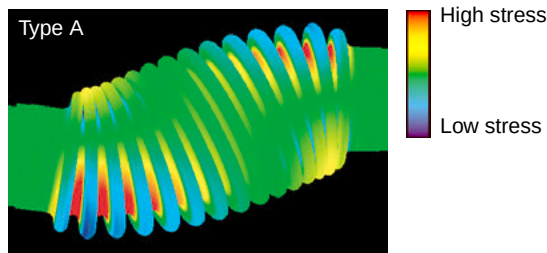


Fig.25 Flexible tube (typeA) stress distribution and behavior

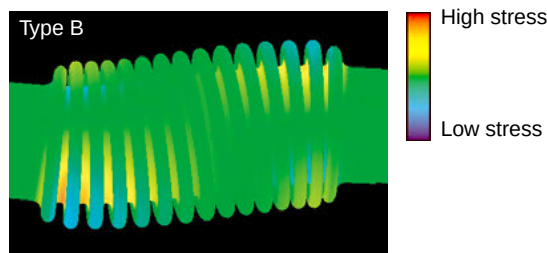


Fig.26 Flexible tube (typeB) stress distribution and behavior

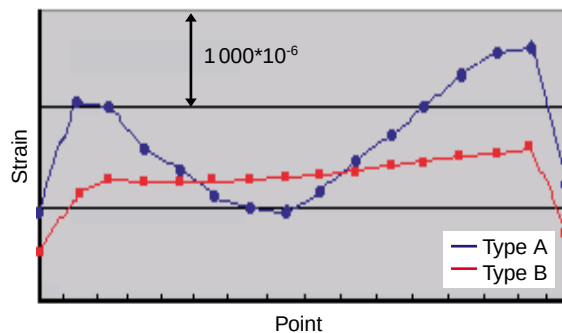


Fig.27 FE Analysis results

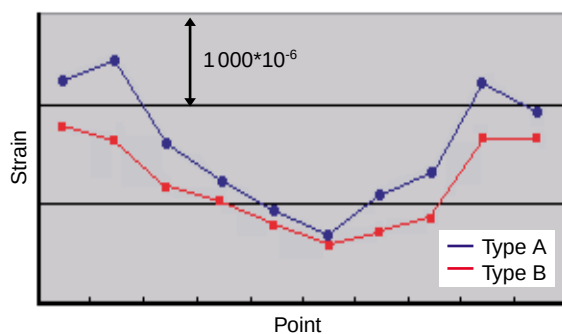


Fig.28 Rig test results

In this way, we were able, while in the drawing stage, to predict and analyze the behavior of dynamic stress in an exhaust system that includes a flexible joint, making efficient development possible.

7. Conclusions

- (1) By utilizing nonlinear maps and the proposed “dummy grid technique”, we were able to model nonlinear components, such as flexible joints and mounting rubbers, making it possible to predict stresses in a manner that takes the nonlinearity of the exhaust system

into account.

- (2) Detailed analysis of the locations where stress is concentrated was made possible by using shell elements or solid elements to represent all components that permitted modeling by shape, such as the flexible tube, pipes, and muffler.
- (3) By making nonlinear analysis possible for the FE model, we were also able to adapt certain parts of the model for a vibration analysis model.

References

- (1) Kihachirou Tanaka, Syouzou Saegusa : Vibration model and simulation, Sangyokagaku Systems Co., Ltd., p.130-137 (1995) (Written in Japanese)
- (2) Hisayoshi Satou : Basis and application of modal analysis, MARUZEN Co., Ltd., p.68-74, p.156-161 (1990) (Written in Japanese)
- (3) Hayato Togawa : Vibration analysis with finite element method, SCIENCE Co., Ltd., p.34-44 (1997) (Written in Japanese)
- (4) Toshio Miyoshi, Masaki Shiratori, Juhachi Oda : Academic basis material dynamics, Jikkkyou Publishing Co., Ltd., p.77-84 (1984) (Written in Japanese)
- (5) The Japan Society of Mechanical Engineers : JSME Handbook for mechanical engineers, The Japan Society of Mechanical Engineers, p.1-9, p.31-40, p.88-90, p.126-145 (1977) (Written in Japanese)
- (6) Handbook for strength calculation of mechanical element, Morikita Publishing Co., Ltd., p.109-169 (1978)
- (7) Hiroichi Fujita : Nonlinear problems, Corona Publishing Co., Ltd., p.39-46 (1984) (Written in Japanese)

authors



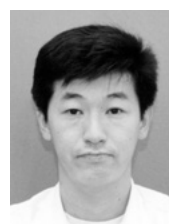
Takumi TAKAKAMO



Masahiko HIGUCHI



Hiroshi NAKAGAWA



Youtarou UKAI



Ayumu ISHIZUKA