Improvement of Simulation Technology for Analysis of Hub Unit Bearing

K. KAJIHARA

Recently, severe development competition, a development process reform aiming for shorter development period and reduced development cost has been actively performed. To realize such a reformation in our development process, highly matured designing is required to be performed from the initial development stage. As a result, CAE is indispensable for clarifying the design bases.

From the above viewpoints, an approach for providing a development method using CAE has been performed. This paper introduces the outline of the development process with regards to bearings for automobile wheel.

***Key Words**: FEM, hub unit bearing, development process, simulation*

1. Introduction

Under recent severe development competition by manufacturers, development process reform has been performed actively for the purpose of shortening development period and reducing development cost. In order to reform the development process, the concept of front-loading must be adopted for providing highly matured design from the initial development stage (**Fig. 1**).

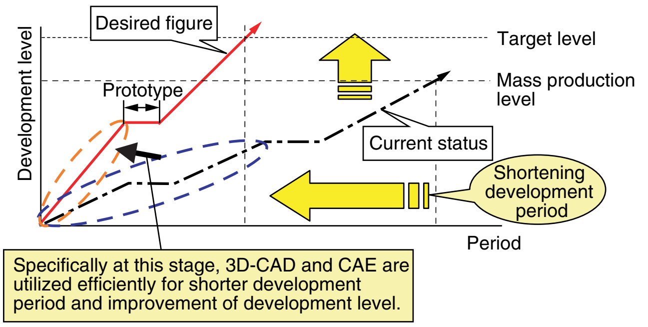


Fig. 1 Development by front-loading

For providing the development process by the front-loading, 3D-CAD and CAE techniques must be actively used from the initial stage of the development and the product performance must be promptly evaluated at the desk. The development based on the front-loading also contributes to clarifying the basis of the design and improves the design quality.

From the above viewpoint, an effort for establishing the evaluation method using CAE for the purpose of reforming the conventional development process has been made. Here introduce the details of our effort on the development of automobile wheel bearings.

2. Current Status and Problems in CAE-used Analysis of Wheel Bearings

Automobile wheel bearings are being required to satisfy higher reliability and performance requirements from various automakers at the same time as having smaller size, lighter weight, higher rigidity, and longer life. Recently, installations of hub unit bearings (hereinafter referred to as hub unit) as wheel bearings have been increasing, requiring design as a structural body and bearing combination. Therefore, the needs for CAE are increasing.

Figure 2 shows a representative structure of a hub unit.

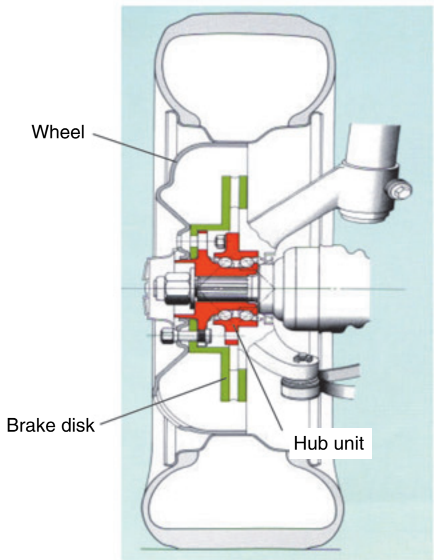


Fig. 2 Structure of hub unit

Conventional development evaluation frequently depended on experiments, therefore, when an experimental prototype shows a problem in an evaluation result, the design to prototype cycle must be repeated, so increasing the development period.

In view of this situation, CAE development based on front-loading has been performed for the purpose of shortening the development period and reducing the prototyping cost, the final objective of which is to provide the development process through which a designer himself can evaluate the product by CAE.

The following section describes a representative analysis example of a hub unit.

3. Strength

About 20 years have passed since CAE tools were introduced to Koyo (**Fig. 3** shows an FEM analysis model at the introduction). FEM analysis technique for a hub unit has been significantly improved with the evolution of the analysis tools.

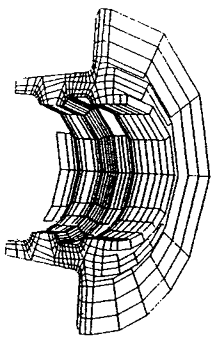


Fig. 3 FEM analysis model in 1987

With computers having higher performance, it is now possible to analyze larger size models. Current models can be calculated with remarkably higher accuracy than that used at the initial introduction of FEM. Application software used for the analysis also has been tremendously advanced, thus allowing current models to be analyzed under more complicated conditions that were impossible in conventional analysis.

However, actual situation of analysis process by Koyo was that the consistency between the analytic value and the experimental value was not sufficiently examined, thus causing the accuracy to be unstable. An approach to improve the consistency between an experimental value and an FEM analytic value in the stress analysis was started to accumulate the results of the developments for providing an analysis of a hub unit with higher accuracy. The objective of this approach is to reduce the stress error between experimental values and FEM analytic values to within 10% (in consideration with variation in experiments).

3. 1 Consistency with Experiment

In order to achieve high consistency between FEM and an experiment, a plurality of factors including a method for preparing a model, load conditions, and restriction conditions must be considered. For this reason, in the first stage of the development, for a complicated model it is hardly possible to determine the cause of stress error which is observed in achieving consistency. Thus, in order to estimate the consistency with an experiment, consistency with a simple model was first obtained and then actual shape and conditions were gradually used. **Figure 4** shows the outlines of models in the respective steps actually used in the stress analysis.

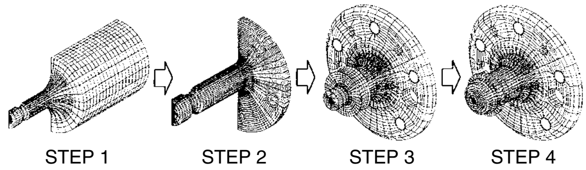


Fig. 4 Approach by analysis model

As shown in **Fig. 4**, the estimation of the consistency with an experiment was performed in the total of four steps of STEP 1 to STEP 4 in order to examine whether an experiment evaluation can be substituted by FEM.

< STEP 1 >

· Analysis with simple cantilever model

In STEP 1, following process and factor were determined.

① Method for dividing mesh for an FEM model; and

② The number of divided meshes at each part

< STEP 2 >

· Analysis with flange shape

In STEP 2, a method for determining a model (boundary conditions) was selected.

< STEP 3 >

· Examination based on flange shaft shape of actual machine

In STEP 3, a model having a shape of an actual machine was used to see how much consistency is obtained between an experiment and the model when the model mesh shape determined in STEP 1 and the boundary conditions determined in STEP 2 were used.

< STEP 4 >

· Examination with shaft assembly model

In STEP 4, based on the result of STEP 1 to STEP 3, an analysis model assuming actual test conditions was prepared to check the consistency in stress between the model and experimental values.

Figure 5 shows the consistency in stress between the experiment and FEM in two representative load conditions (for the load directions, see the schematic in **Fig. 5**). As can be seen in **Fig. 5**, every measurement position both in the load conditions ① and ② shows high consistency (error within 10%).

As can be seen from the result as described above, the present analysis method can reproduce the stress status in an actual machine with high accuracy.

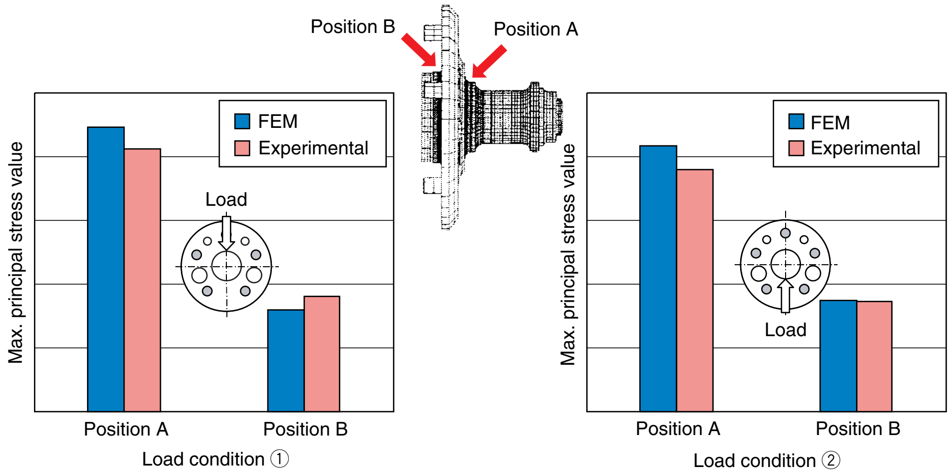


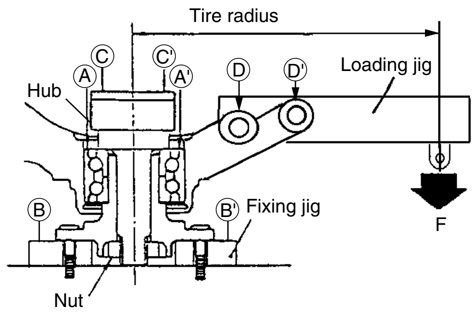
Fig. 5 Consistency in stress among measurement portions

4. Moment Rigidity

Rigidity of an automobile wheel bearing has a significant influence on the ride comfort and the steering feeling during the driving and thus has been required to satisfy severe requirements from automakers. Rigidity of a wheel bearing is generally evaluated by measuring the inclination angle (hereinafter referred to as moment rigidity) of each part to a specified moment.

4. 1 Actual Measurement Method

Figure 6 shows the outline of the actual measurement method.



[Calculation method of inclination] $l_{X\text{-}X'}$: distance between X-X'
Measuring criteria: *B–B'*

· Inclination of hub shaft

$$\theta_S = \tan^{-1}\frac{C'\text{–}C}{l_{C\text{–}C'}}$$

· Inclination of bearing

$$\theta_B = \tan^{-1}\frac{A'\text{–}A}{l_{A\text{–}A'}} - \theta_S$$

· Inclination of knuckle

$$\theta_K = \tan^{-1}\frac{D'\text{–}D}{l_{D\text{–}D'}} - \theta_S - \theta_B$$

Fig. 6 Outline of actual measurement method

4. 2 Moment Rigidity Analysis Method by CAE

Moment rigidity was calculated by general-purpose FEM codes and a technical calculation program which was developed by Koyo. In the FEM analysis, the analysis method shown in the above section **3** was used.

Although FEM can simultaneously calculate the rigidity of a knuckle, bearing, hub, and drive shaft, rigidity of each part was calculated individually due to the reasons as shown below.

- · Excessive increased number of nodes causes high work load for the computer.
- · Structuring of the model is complicated (i.e. designer has difficulty in handling the model).

In other words, the development shown below used a method in which rigidity of each part was separately calculated and subsequently adding the calculation values to calculate the moment rigidity of the entire structure.

4. 3 Consistency between FEM Analytic Result and Actual Measurement Result

Figure 7 shows the comparison between the actual measurement result and the FEM analytic result.

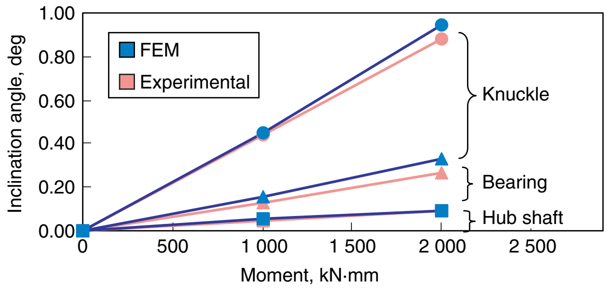


Fig. 7 Comparison between actual measurement result and analytic result

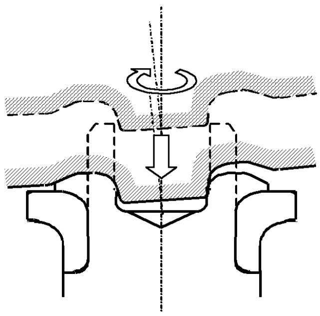
The difference in the values of moment rigidity between CAE and the actual measurement was 8% at the maximum, showing relatively small difference. The difference with regards to a hub shaft inclination angle and a knuckle inclination angle showed favorable consistency.

5. Shaft-end Clinching

A shaft-end clinching method has been currently developed for tightening a flange shaft of a hub unit with the bearing inner ring as a substitute for the conventional nut tightening method. This method has been used for mass production.

The clinching method, which does not use nuts, is advantageous because it does not require the control of preloading in assembling a bearing and can reduce the number of components so that further adoption is expected. The following introduces a simulation for finding an optimal design including the production engineering technique.

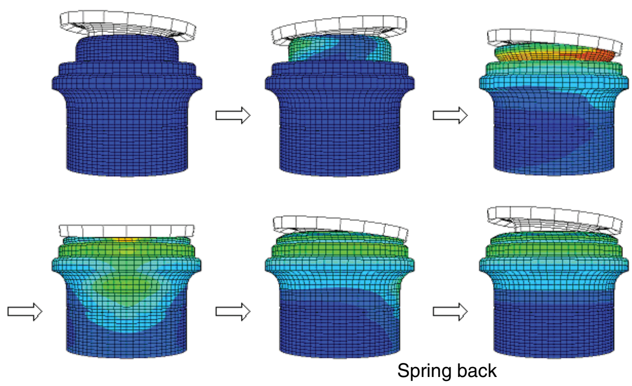
Figure 8 shows the outline of an analysis model.



Simulation of plastic processing is carried out with processed surface set to axial direction in oscillating punch (rigid surface).

Fig. 8 Outline of analysis model

Figure 9 is a VonMises stress diagram showing the dynamic analysis of oscillation clinching in chronological order (Deformation is represented by same magnification). As can be seen from **Fig. 9**, a shaft is caulked while receiving punch rotation and push. The sample shows a uniform stress distribution after spring back, showing that the punch is removed securely.



Vonmises stress + deformation by oscillation clinching (dynamic analysis)

Fig. 9 Simulation of shaft-end clinching

The simulation result with regards to plastic region of the shaft and the punch reaction force was also consistent with the actual measurement.

6. Fretting

Fretting has been caused at the joint face of a hub unit, brake disc, and wheel, causing a problem of stick-slip noise and thus requiring a method to reduce fretting[1]. Although fretting has been conventionally evaluated by an experiment, fretting analysis by CAE as a prior examination is necessary for an examination of the design.

Thus, a new analysis method by CAE was examined by using FEM to extract fretting-related parameters based on factors such as contact surface pressure applied between a hub unit and a brake rotor.

6. 1 Distribution of Fretting Contact Surface Pressure

Fretting is caused and progressed due to generation of abrasion powders, repeated small relative slip and hardening of abrasion powders by oxidation. It is considered that the relative slip is caused at the part at which joint faces contact and separate each other during the operation and is hardly caused at the part at which joint faces always contact each other or always have no contact. In other words, surface stress at the part at which joint faces always contact each other or always have no contact is not directly linked to fretting. Thus, the contact surface pressure at the part at which the contact status is changed when the load direction is changed was defined as a fretting contact surface pressure and was calculated. The analysis result of it is shown in **Fig. 10**. **Figure 11** shows the fretting status of the flange face after the test.

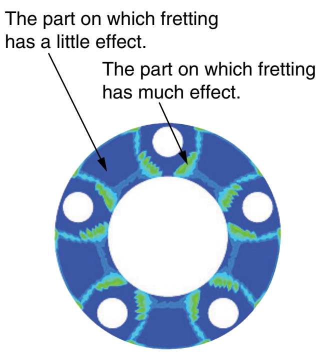


Fig. 10 FEM analysis result

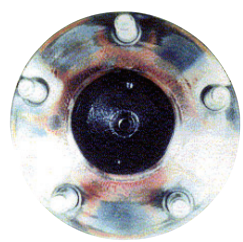


Fig. 11 Fretting status after test

The distribution diagram of the fretting contact surface pressures and the fretting status after the test showed good correspondence with regards to the qualitative aspects such as a fretting region and an abrasion level.

This method can allow the user to examine the design for suppressing fretting.

7. Weight Reduction

A hub unit has been recently required to have a smaller size and lighter weight due to factors such as improved fuel efficiency by reduced unsprung weight, improved vehicle movement performance, and expanded freedom in size of peripheral components. Reduction of such basic performance as strength, rigidity or the like due to reduced weight must be of course avoided.

Therefore, the method for reducing the weight of a hub unit in consideration of the analysis of strength and rigidity has been used to optimize the design, a specific example of which is shown in **Fig. 12**. This model is an example that achieved a significant weight reduction while maintaining the strength and the rigidity equal to those of current mass-produced products. Recently, such approaches for providing the design of "light and strong" bearings have been increasingly used.

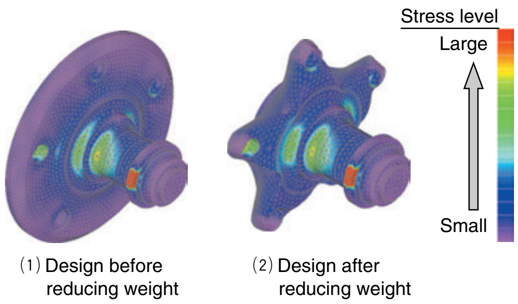


Fig. 12 Example of light-weight hub shaft

8. Conclusion

In the above sections, our approaches using CAE for reforming conventional development processes have been described by taking an example of an automobile wheel bearing. When looking back at the situation when our approaches were started, items that can be employed for the analysis have been remarkably increased.

However, it is considered that the verification by experiments has been and will be indispensable no matter how the future CAE techniques will be evolved. Thus, the author will make further efforts for achieving further sophisticated CAE analysis techniques including efforts for securing the consistency between experiments and the analysis.

Reference

1) K. Yamakawa and K. Kajihara: Koyo Engineering Journal, 165E (2004) 41.

K. KAJIHARA*

* *Analysis Engineering Department, Bearing Business Operations Headquarters*