

Analysis of Suspension Characteristics Considering the Elastic Deformation of Parts

Makoto TSUKAGOSHI*

Takuya SHUDO*

ABSTRACT

Taking the elastic deformation of parts into account in calculations to simulate the characteristics of suspension alignment was confirmed to improve prediction accuracy. Taking the elastic deformation of the main elements of the overall suspension, the sub-frame, and the body into account made it possible to accurately determine the side force compliance at the tire contact point with the ground, and also to clarify the contribution made by each of those parts. Development of an elastic body modeling method that accounts for significant deformation made it possible to improve the accuracy of alignment prediction, even in parts such as the twist beam axle that undergo significant deformation. Software was developed to assist with design analysis so that these prediction methods could be utilized in practical design work.

1. Introduction

Multi-body simulation software used to be employed for analysis of alignment characteristics when designing suspension geometry. Since the use of multi-body simulation software by designers imposed a heavy data origination load, a simple data entry tool for use by designers was developed. This made it a simple matter to perform analysis of alignment characteristics by inputting the coordinates for the main suspension points and the non-linear spring characteristics of the bushings.

The model for multi-body calculation up to this time, however, considered all suspension parts to be rigid bodies. Thus the elastic deformation of parts was not taken into consideration, and this is known to result in discrepancies between alignment calculation results and actual measurements.

Here, an elastic body model of suspension parts, sub-frame, and body have been incorporated into a multi-body model, and analysis of alignment characteristics has been carried out that

takes elastic deformation into account. The prediction accuracy and the contribution of each part have also been verified.

This article reports on modeling techniques developed for practical application of these methods in vehicle development and software developed to assist in design through improved analysis and easier modeling.

2. Coupled Simulation of Multi-Body and Structure

2.1. Overview

Deformation and stress calculations for an elastic body utilize the finite element method, which segments the model geometry into small areas and couples together the load and the deformation for all adjacent small areas. However, the application of this finite element method just as it is for elastic bodies in a multi-body model introduces practical disadvantages. That is, the multi-body model calculates objects that move along a time history, regardless of whether

* Tochigi R&D Center

those objects are static or dynamic. Consequently, this model must calculate the deformation state of an elastic body at every step in the calculations. The amount of calculation becomes enormous, making practical turn-around impossible. Current multi-body simulation software therefore uses the modal synthesis method as a means of incorporating an elastic body into a multi-body model.

2.2. Modal Synthesis Method

The elastic deformation of members can generally be expressed by superposing the eigen modes of those members. In order to express local deformation, however, it is necessary to incorporate an extremely large number of eigen modes, including even high-frequency components. When the load input points are clearly defined, as in the case of suspension members, it is known that accurate expression, even down to local deformation, is possible by superpositioning deformation modes for input to those points (in other words, the static correction mode) with a relatively small number of eigen modes(1). This method corresponds to the constrained mode method, one of the sub structuring methods used primarily for vibration analysis. Use of this method makes it possible to reduce the number of degrees of freedom significantly, from the several tens to hundreds of thousands of degrees of freedom found in an FEM model, to a number of degrees of freedom equal to just the number of modes employed in the model.

2.3. Mode Selection

Free oscillation caused by inertia does not occur in the static calculation of dynamic equilibrium used for analysis of suspension characteristics. It was expected, therefore, that the elastic deformation of members could be fully expressed by the superposition of static correction modes for all load input points. Two analyses were carried out in order to confirm this. First, an analysis was done of the knuckle and lower arm of a double-wishbone suspension modeled as an elastic body using only the static correction mode. This was compared with analysis using the normal mode from the lowest to the third order simultaneously to determine whether any difference appeared in the alignment variation. Figure 1 shows the change in the toe angle when lateral force is applied at the ground contact point. It was confirmed that there was no difference in either case, and the same was also true of other characteristics.

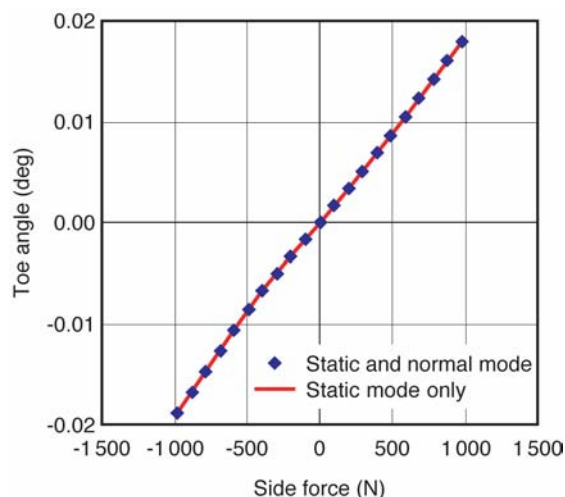


Fig. 1 Comparison of alignment

Next, an analysis was done using only a static correction mode with a four-wheel model that had a double-wishbone front and multi-link rear suspension. The body, sub-frame, and main suspension were treated as members of an elastic body. This was compared with a simultaneous analysis using eigen modes up to 100 Hz to determine whether there would be any differences in the displacement at the points where suspension members were attached to the body. Figure 2 shows the displacement at attachment points when lateral force is applied at ground contact points. It was confirmed that there was no difference in either case.

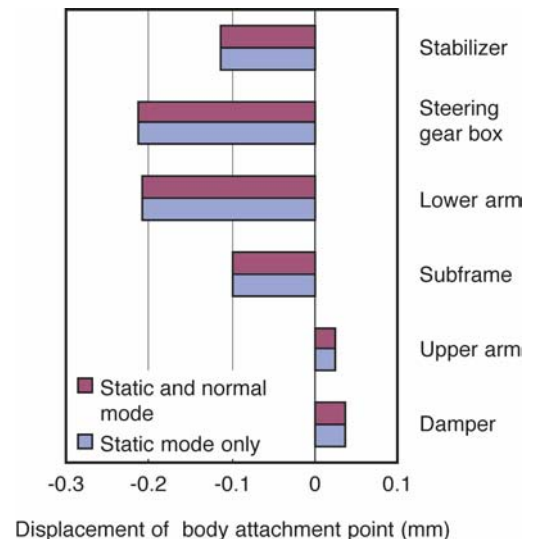


Fig. 2 Comparison of displacement

3. Verification of Prediction Accuracy

3.1. Comparison of Prediction With and Without an Elastic Body

Alignment characteristics were verified using a suspension that included arms functioning as elastic bodies. Figure 3 shows a comparison of the longitudinal force against toe angle when the arms are modeled as elastic bodies versus being modeled as rigid bodies. Likewise, Fig. 4 shows a comparison of the lateral force against toe angle.

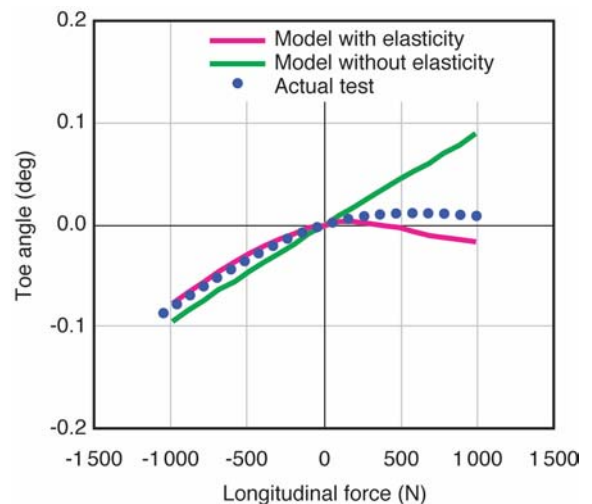


Fig. 3 Comparison of toe angle vs. longitudinal force

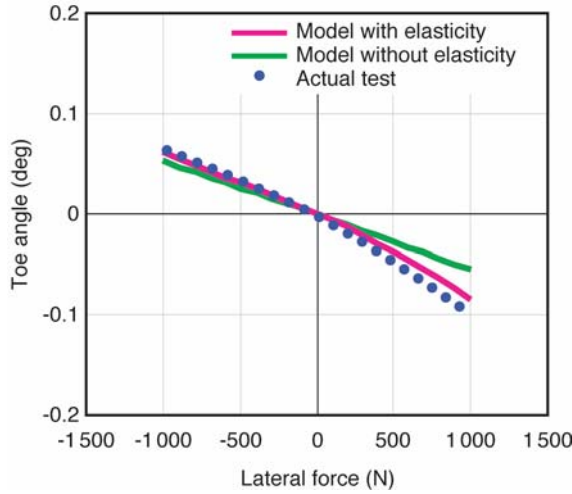


Fig. 4 Comparison of toe angle vs. lateral force

The model with arms as rigid bodies shows a straight-line rate of change. By comparison, the model that incorporates elastic bodies shows the characteristic varying with the direction of the longitudinal force, just as seen in the actual measurements. This is thought to result from the change in the balance between the bushing deflection and the variation in the elastic body caused by the difference in the spring stiffness when the bushings are pulled forward and backward.

3.2. Comparison of Differences in Elastic Body Rigidity

It is known that weight reduction measures for suspension parts, through change in thickness or shape, affect the alignment characteristics. Therefore the rigidity of the elastic bodies was varied and the changes in the characteristics were compared and verified. Modifications were made so that the change in rigidity would be significant in order to produce a distinct variation relative to the base. Comparison of alignment characteristics was performed with similar modifications also made to the model. Figure 5 shows the comparison of longitudinal force against toe angle. Figure 6 shows the comparison of lateral force against toe angle.

The results confirmed that the degree of effect on alignment characteristics, caused by different rigidity in elastic parts, was faithfully reproduced.

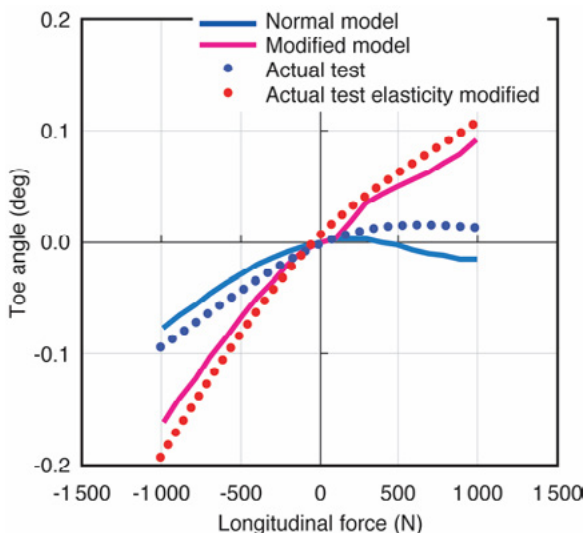


Fig. 5 Comparison of toe angle vs. longitudinal force

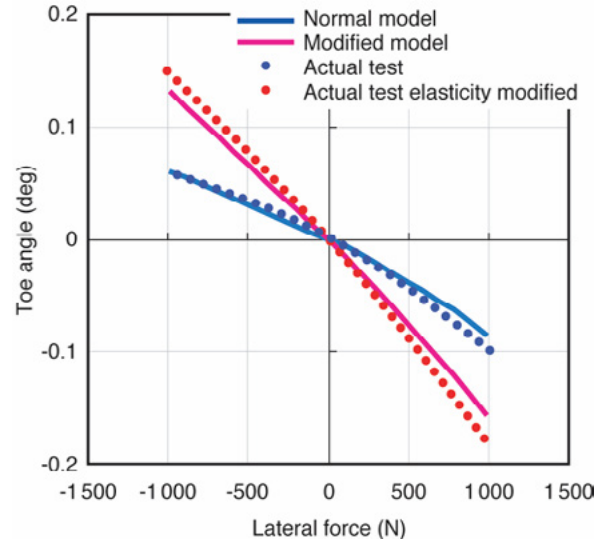


Fig. 6 Comparison of toe angle vs. lateral force

3.3. Verification of Prediction Accuracy for Lateral Stiffness at Tire Contact Point

The lateral stiffness at the tire contact point is considered affected by elastic deformation in the body and sub-frame. The accuracy of calculation of this stiffness was verified and the contribution made by the rigidity of the different parts was analyzed.

Lateral stiffness at the tire contact point is one index used in feeling tests that is considered affected by the vehicle cornering response. The measurement is made by attaching metal discs that can be treated as rigid bodies in place of the tires and measuring the lateral displacement that occurs when lateral force is applied at a location corresponding to the ground contact point. Therefore this only measures the vehicle characteristic and does not include the deformation of the tires.

Conceivable factors in displacement are the deflection of the suspension bushing and sub-frame mount bushing, elastic deformation of suspension members such as knuckles and arms, and local and overall elastic deformation of the sub-frame and body. The present model included all the suspension arms, the sub-frame, and the body as elastic bodies. Figure 7 shows a general view of the model.

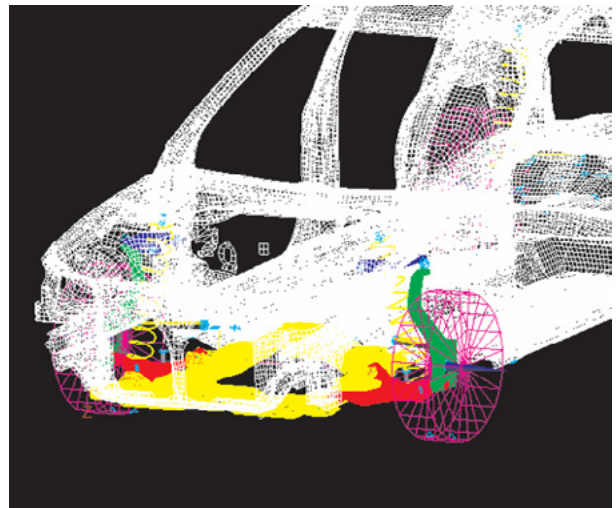


Fig. 7 Simulation model including elasticity of body, sub-frame and suspension

The verification confirmed that the results from the model incorporating the elastic body were much closer to the actual measurement than the model taken as a rigid body. Figure 8 shows the lateral displacement of the ground contact point in relation to the lateral force in the front suspension. The size of the contribution made by individual components could also be confirmed by adding elastic bodies, one by one, to a model that had contained no elastic bodies, then tracking the changes in displacement with each addition.

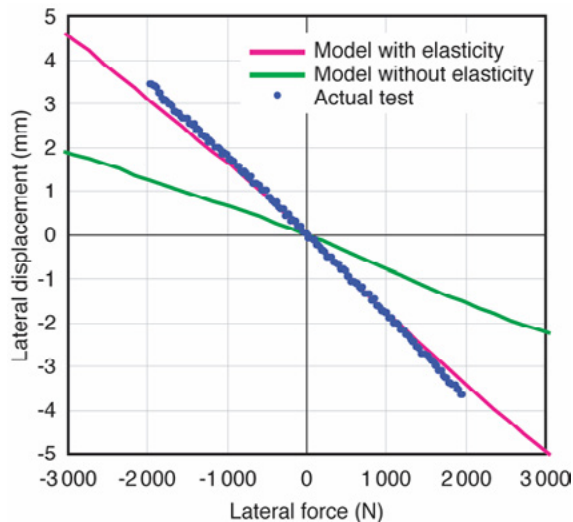


Fig. 8 Lateral compliance of front suspension

3.4. Modeling a Twist Beam Axle

When the left and right wheels stroke in alternate mode with a twist beam suspension, a large torsion deformation is caused in the beam components. The FEM model assumes a small and linear deformation, and in the case of a major deformation such as this, the small deformation is expanded linearly from its initial position. Therefore, as shown in Fig. 9, the resulting deformation mode differs radically from the actual deformation. Consequently, it is insufficient merely to model these components as elastic bodies. A method of modeling was therefore used that segments the FEM model of the twist beam axle into several parts, as shown in Fig. 10, which then rejoins them using multi-body simulation software. Figure 11 shows a general view of the model.

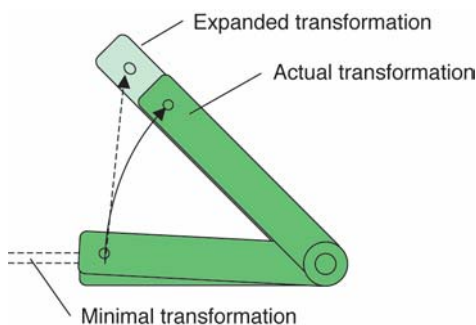


Fig. 9 Side view of twist beam axle

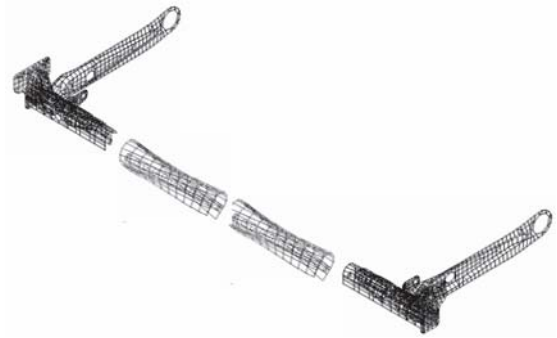


Fig. 10 Segmented FEM model of twist beam



Fig. 11 Suspension model with multi FEM segments

After calculating the static correction mode for the various segments of the FEM model, it was incorporated into the suspension model in the form of multiple separate elastic bodies. The elements on the segmented surface were then joined to each other using multi-body simulation software. This made it possible to simultaneously express the elastic deformation of components during major deformation, the independent movements of the segmented parts, and their rotational movement. It was confirmed that calculation accuracy was improved as a result. Figure 12 shows a comparison of the wheel center tracks that result with the suspension stroke in opposite phases laterally.

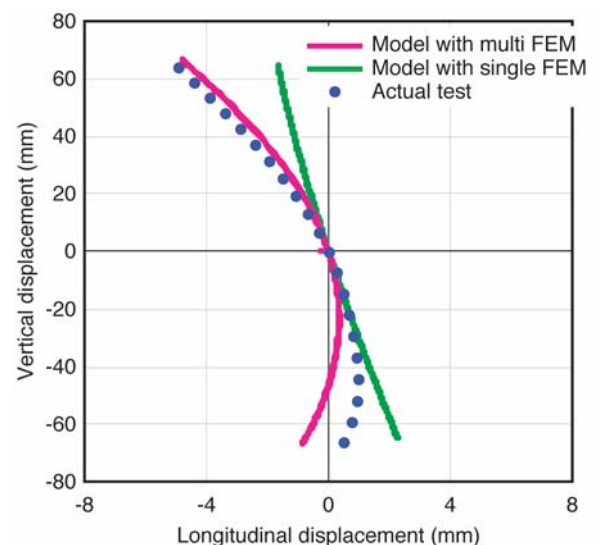


Fig. 12 Wheel center locus from side view

4. Design Assistance Software

4.1. Overview

The technique for analysis of a multi-body model that incorporates elastic bodies is extremely effective for chassis development. However, this requires the person performing the analysis to have skills in both multi-body simulation and structure analysis. Consequently, it is rather difficult for a designer without those skills to use. On the other hand, multi-body simulation of the suspension and FEM analysis of the suspension arm on its own are widely used techniques, and models using them are generated on an everyday basis. If it were possible to combine these and perform the calculations and post-processing simply and effectively, the result would be a practical analytical technique for the designer. Therefore software was developed that would automate the process of incorporating the FEM model into the multi-body model, and further carry out calculations and post-processing collectively. Figure 13 shows the overall flow of the processing. The processing flow has three major stages: (1) Pre-processing of the FEM model, (2) combination of the FEM calculations and the multi-body model, and (3) calculation and post-processing of the combined model. Figure 14 shows the software operation panel.

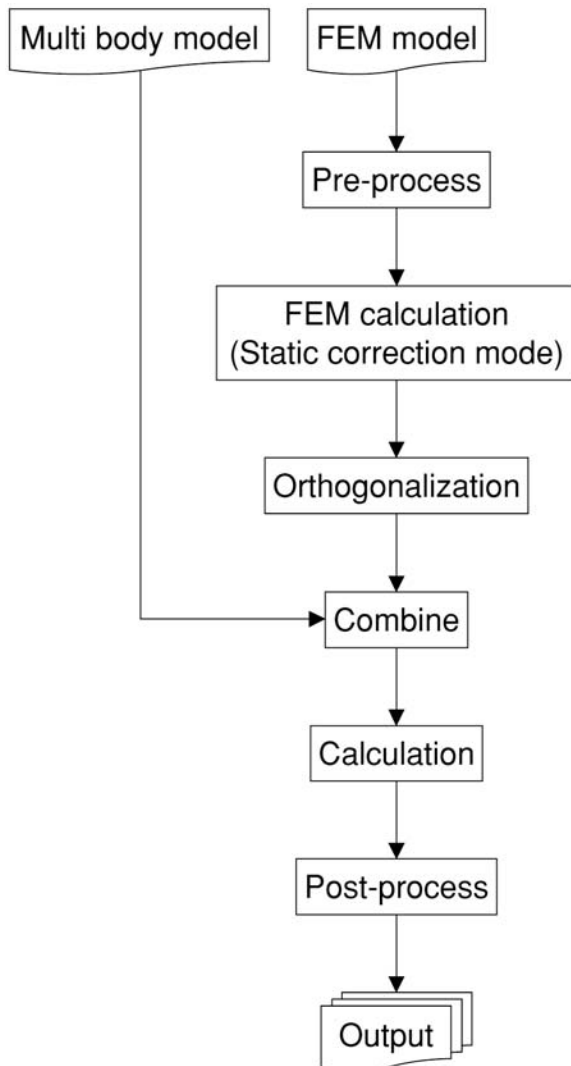


Fig. 13 Flow chart of automated process

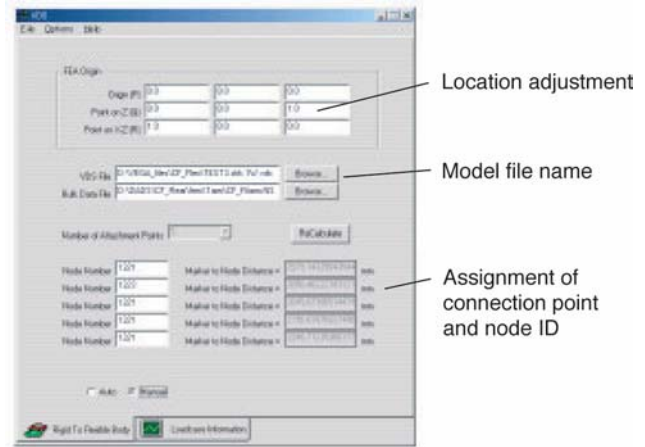


Fig. 14 Software operation panel

4.2. Coordinate Conversion and Mirror-Image Model Generation

When the multi-body model and the FEM model are combined, the reference axis systems of the original models may differ. In this case, there may be an issue if their attachment points do not match. Consequently, one or the other model must have its coordinates converted. The software converts the coordinates of the node positions of the FEM model to the coordinate system of the multi-body model. A four-wheel model will require symmetrical FEM models for the left and right pairs of wheels. Consequently, the FEM model for one side only is used to generate a mirror image FEM model for the other side.

4.3. Automatic Node ID Search and Boundary Condition Generation

When combining the multi-body model and the FEM model, the node ID information from the FEM model is required in order to define the attachment points (joints and bushings). In order to acquire these from an existing FEM model, it is ordinarily necessary to read them into FEM pre-processor and examine the node information. Some designers, however, are not familiar or comfortable with operations of FEM pre-processor. Therefore the software reads the node positions from the FEM model and compares them with the attachment point coordinate positions in the multi-body model. It automatically searches for the closest nodes, then informs the user and presents a list of them. Figure 15 shows an example of such a display. At this time, a setting is also simultaneously made for the boundary condition that is used to calculate the static correction mode from the FEM model, based on the node IDs determined here.

Point	Point Coordinate	Node Number	Distance (mm)
"A"	2188	8649	0.00
"E"	2744	8653	0.10
"F"	2578	8651	0.00
"D"	2719	8650	0.00
"S"	2521	8652	0.04

Fig. 15 Automatic search of node number connected to multi-body model

4.4. Adjustment of Attachment Point Coordinate Displacement

When the coordinate values for attachment points with the elastic bodies are to be changed during the process of design study, it is ordinarily necessary to change both the multi-body model and the FEM model. Therefore, as shown in Fig. 16, the node positions of the FEM model are automatically rewritten to match the node positions in the multi-body model by the software. As a result, it is also possible to generate models with different attachment points simply by changing the coordinates in the data input tool. This can be done without recreating the FEM model, if the range is small.

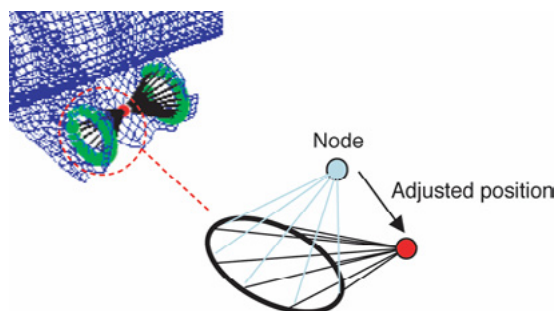


Fig. 16 Automatic adjustment of node's local position

4.5. Automatic Division of Elastic Body

When using twist beam FEM models, users specify the node IDs for the end points where segmenting begins. The software searches for adjacent nodes by following the segment surface from the end point, and takes those as the border for segmenting the model. In order to make it easier to recombine the segment surfaces afterward using the multi-body software, the nodes are organized into elements of several rigid bodies. Figure 17 shows the arrangement of the section surface.

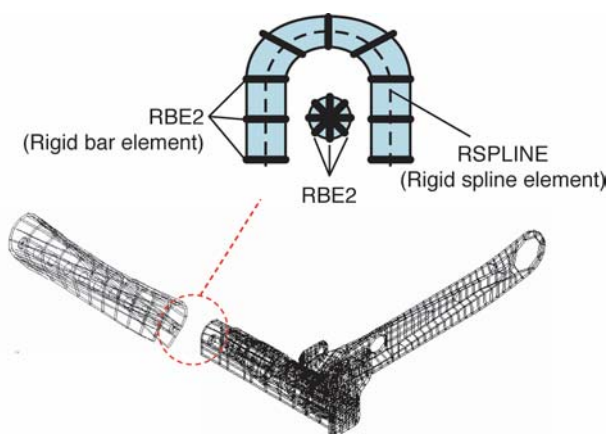


Fig. 17 Arrangement of elements at section

4.6. Automatic FEM Calculation and Model Combination

When the instruction is given to execute calculation at the stage when pre-processing has finished, operation is handed over to the FEM solver and the static correction mode is calculated. The file of calculation results is read into the multi-body software, the target rigid body components are replaced, and the combined multi-body and elastic body model is generated. Figure 18 shows an example of a combined model.

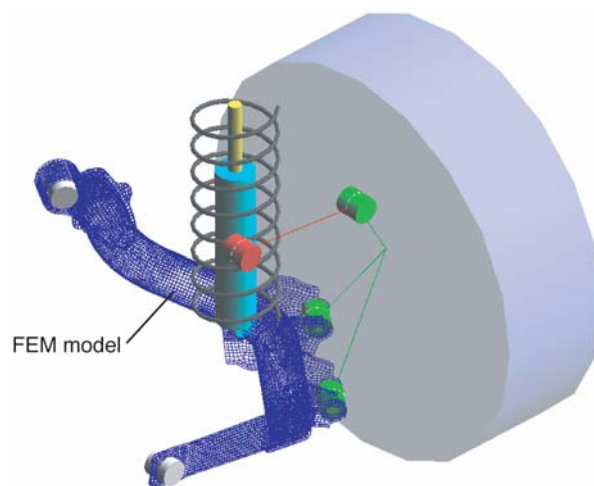


Fig. 18 Combined multi-body and FEM model

4.7. Combined Model Calculation and Post-Processing

The combined model is then handed over to the multi-body simulation solver for calculation of static time history behavior (quasi-static). After calculation, items that a designer wants to confirm can be extracted as necessary from the calculation results to generate a spreadsheet with a fixed format. This makes it easy to obtain the various kinds of tables and graphs needed for performance evaluation.

Figure 19 shows an example of the graph output.

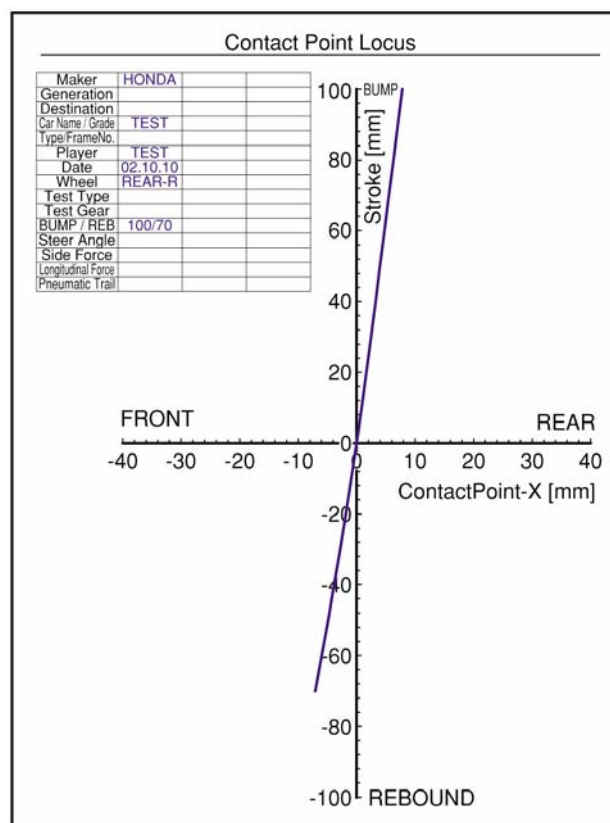


Fig. 19 Example of result output

5. Summary

It was found that incorporation of elastic bodies in the multi-body model to take elastic deformation into account significantly improves the accuracy of suspension and chassis analysis. By incorporating FEM models into the multi-body simulation software, it is possible to model characteristics contributed by lateral stiffness at tire contact point and other such rigidity of components such as the body, sub-frame, and suspension. The modeling extends to the non-linear spring characteristics of joint bushings. This makes it possible to perform calculations that simultaneously take account of the elastic deformation of the various individual elastic parts themselves, the movement of components as a whole caused by deflection of non-linear bushings, and their rotational movement. This indicates that more accurate results can be obtained by integrating multiple FEM models in a similar manner using FEM analysis software to perform calculations for a single large, linear FEM model.

A modeling technique was created that works by segmentation. It can also be used practically with conventional linear FEM models, without applying nonlinear FEM analysis techniques to the analysis of suspensions such as the twist beam type that involve significant elastic deformation. This necessitates complex, troublesome work to generate a model. However, this was made into a usable technique by developing a program that reads in the FEM models used for integration in advance, reads in and processes the attachment point information, and automatically generates a combined model.

Reference

- (1) Craig, R. R., and Bampton, M. C. C.: Coupling of Substructures for Dynamic Analyses, *AIAA Journal*, Vol. 6, No. 7, p. 1313–1319 (1968)

■ Authors ■



Makoto TSUKAGOSHI



Takuya SHUDO