Technical Paper

Jig Rigidity Evaluation Technology by Vibration Analysis

Eiji Nabata

Yuji Terasaka

Large vibration (chatter vibration) can occur in tools and workpiece during machining. This vibration causes problems in machining accuracy, efficiency, tool life and safety. One of the causes for chatter vibration is lack of sufficient dynamic rigidity to stabilize parts in a dynamic cutting force. In this case, parts are reinforced by a jig to supplement inadequate rigidity. In the past, Komatsu has studied vibration analysis technology by real eigenvalue analysis using the finite element method. One problem in this study has been the creation of an analysis model due to computer performance and constraints in shape definition.

This report describes the development of vibration analysis technology for analyzing of an entire system including jigs achieved through the utilization of recent 3D-CAD, and through a new method to express clamps. Komatsu's activities in achieving high efficiency in machining and application examples are reported.

Key Words: *machining vibration, vibration suppression, modal analysis, fixture design, 3-Dimensional analysis, chatter*

1. Introduction

About a dozen years have passed since Komatsu first adopted 3D-CAD, which has now become a necessary tool in design work thanks to enhancement in its controllability and computer throughput. In manufacturing also, activities have been undertaken to enjoy its advantages. In the offline teaching of welding robots, in the evaluation of assemblability and in other phases of work, the advantage of being able to know the shapes of parts accurately in advance is large. In machining also, jig rigidity evaluation by vibration analysis that has been used for many years is now upgraded using a 3D-CAD model as reported below.

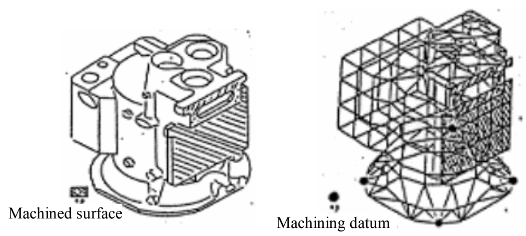
2. Jig Design by Vibration Analysis

Many of the parts in construction machinery are large and stock allowances in machining are also large, requiring high-efficiency machining to shorten machining time. For this reason, machine tools that feature high rigidity and a high feed rate and tools that allow fast cutting are introduced. Nevertheless, high-efficiency machining on which expectations are placed cannot be accomplished unless machined parts have stable dynamic rigidity enduring a dynamic cutting force. Parts are designed to meet the performance needed as products, but many parts do not have the rigidity that is needed for machining. Large transmission cases and casings of reduction gears in particular are made of plates or boxes of thin wall thickness and are typical of them. Particularly in milling, regular vibration is caused and regenerative chattering that triggers the natural mode of machined parts present problems. To avoid this, appropriate design of jigs has been necessary so as to supplement the low rigidity of machined parts. This design process has been implemented empirically and is validated by trial cutting after fabricating parts, resulting in problems due to a lack of rigidity and in fabricating expensive jigs with excessively high rigidity. Beginning around 1982, a vibration analysis technique was implemented to evaluate the dynamic rigidities of machined parts at the drawing-drafting stage using real eigenvalue analysis by the finite element method (FEM) [1), 2)].

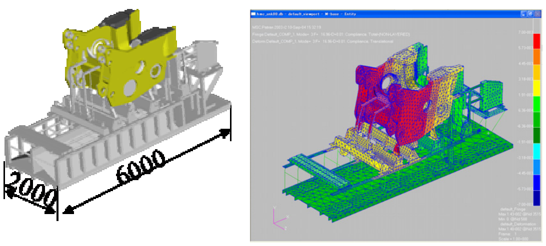
Jig design support by this technique was effective and jigs for the machining of parts could be rationalized. However, due to limited computer computation capability and constraints on modeling of part shapes at that time, simplified shell model diagrams (**Fig. 1(a)**) of only parts were used. Normally, problems are not found with the rigidity when one side of parts is less than 500 mm that assures sufficiently high jig rigidity. Nevertheless, when parts become large, jig rigidity requires consideration. Additionally, the time to build analysis models needs be shortened. This has resulted in the following tasks:

(1) Reading of 3D-CAD models into analysis software
(2) Application to solid models to solve shapes of thick wall thickness
(3) Modeling of contact areas between parts

These tasks could be accomplished and this technique could be applied to the large parts illustrated in **Fig. 1(b)** requiring jig rigidity.



(a) Shell element model simplifying parts only (around 1985)



(b) Analysis model incorporating part and entire jig (2005)

Fig. 1 Analysis model used in vibration analysis

3. Development of Analysis Software of 3D-CAD Models

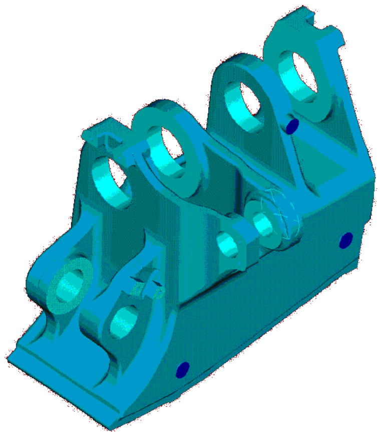
The analysis by a shell model developed earlier required the fabrication of simplified analysis models for analysis from design drawings, and the time needed for fabricating models was a problem.

At present, 3D shapes can be expressed in a computer thanks to 3D-CAD, enabling the validation of mountability and the checking of interference with tools, and other applications are implemented using 3D-CAD models in the design of parts and jigs.

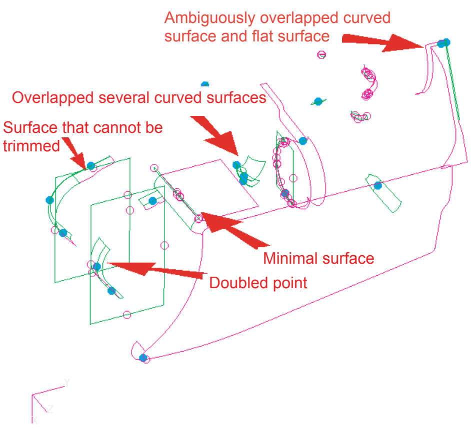
The time needed for the modeling of part shapes, which was a problem, could be shortened significantly by incorporating 3D-CAD models into analysis software. Additionally, shapes were simplified by 3D-CAD, accomplishing both a reduction in automatic mesh-splitting time and an enhancement in mesh quality.

Analysis software and 3D-CAD for design are separate software and analysis models cannot be created using 3D-CAD models in analysis software as they are. An intermediate file format used in data interchange between different CADs such as IGES is therefore used. In other cases, a direct interface (a feature to read the data of different CAD software by converting it) is used for each 3D-CAD type. Nevertheless, model shapes are sometimes incorrectly recognized depending on how 3D-CAD models are built and the model shape if data are merely read in analysis software, disabling the delivery of cubic shapes.

An example of a conversion problem experienced in reading the model shape of a construction machinery part from 3D-CAD into analysis software by a direct interface is illustrated in **Fig. 2**. The model as it is cannot be recognized as a perfected model. This problem was corrected as described in Section 3.1. Shapes were simplified tailored to analysis purposes to shorten the analysis time (Section 3.2).



(a) 3D-CAD model



(b) Error locations when data were read into analysis software (A direct interface was used)

Fig. 2 Example of problem in model reading

3.1 Avoidance of Data Conversion Problem

3.1.1 Transferring IGES as Intermediate File

IGES is the most commonly used transfer file format and is supported by almost all analysis software. However, its plane definition method is not perfect and accurate transfer is precluded in many cases.

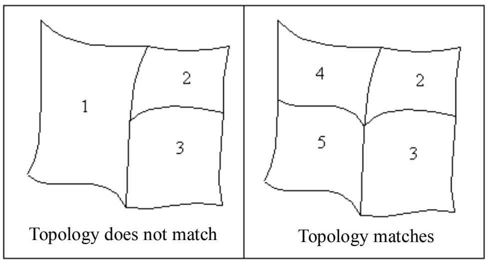
The following problems are experienced during transfer.

(1) Topology faults (See **Fig. 3**)
- Nodes and edges of adjoining surfaces are not shared.

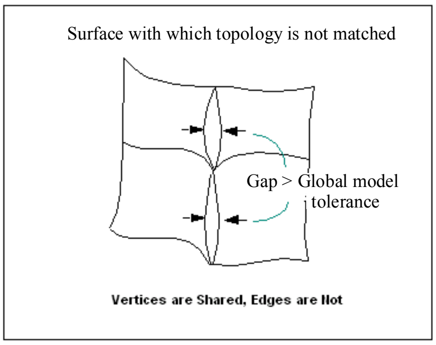
(2) Geometry faults (See **Fig. 4**)
- Bent edges
- Bent surfaces
- Distorted surfaces
- Overlapped surfaces
- Stuck-at (degeneracy) of base surfaces
- Separation from generating curved surface-trim line

These faults are caused by the plotting accuracy and differences in the internal expression method and are corrected as follows.

(a) Matching plotting accuracies (number of effective digits of the coordinates) between 3D-CAD and analysis software.
(b) Remaking surfaces within the analysis software.
(c) Modifying 3D-CAD models so as to avoid producing fine surfaces and edges.
(d) Modifying problematic surfaces by CAD data modification software (CADfix and others).

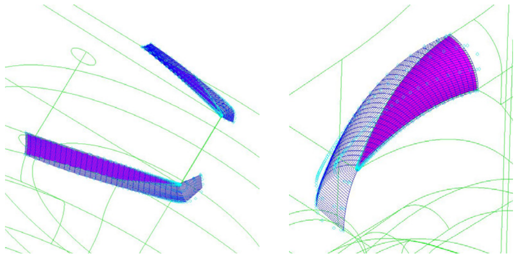


(a) Example of unshared nodes



(b) Example of unshared edges

Fig. 3 Examples of topology faults



(a) Bent surfaces (b) Distorted surfaces

Fig. 4 Examples of geometry faults

3.1.2 Conversion by Direct Interfaces

Conversion is performed less frequently as 3D-CAD data files are directly read. Test adjustment is made to avoid problems and this method is more advantageous compared with the method that uses intermediate files such as IGES.

However, problems are likely to occur when the plotting accuracy is different as in the transfer of IGES files.

The following two methods are used to correct faults.

(a) Changing model shapes by 3D-CAD.
(b) Remaking surfaces through modification of the analysis software.

3.2 Simplification of Undesired Shapes for Analysis

Design models incorporate screw holes meeting the functions of parts, tiny steps on machined and unmachined surfaces, corner radii, chamfering and other elements. In the case of vibration analysis for the evaluation of dynamic rigidity, these fine shapes do not affect the analysis results in many cases. Conversely, these fine shapes cause data conversion faults when converting from 3D-CAD models to analysis software, prevent mesh splitting and lengthen analysis time.

At present, simplification is modified with the following locations. A future task will be automation of the simplification of these elements if there are many locations to be modified.

(1) Elimination of small parts other than structural strength members (piping seats, brackets and other parts)
(2) Filling of small holes such as bolt holes
(3) Elimination of steps and corner radii that are not included in evaluation
(4) Drawing up welding bead shapes

4. Application to Solid Elements

A match between analysis and actual measurement has been confirmed with shell elements. However, the fabrication of shell elements using shapes that are read from 3D-CAD is time consuming. As well, errors are caused in expressing shapes with a thick wall thickness. The use of 3D solid elements was therefore studied. Great differences in analysis results were, however, confirmed depending on the type of shell element used. Phenomena of the following elements were studied and appropriate elements were selected. In this study, automatic meshing of 3D-CAD models described in the preceding chapter was taken into consideration.

A comparative study by real eigenvalue analysis of the FEM elements shown in **Table 1** was conducted with the cantilever illustrated in **Fig. 5** using element shape, element degree, element size, element aspect ratio (ratio between the long side and short side) and other parameters as study items. The theoretical solution of natural frequency of this beam problem was obtained through the bending vibration formula[3] of a beam with a uniform cross section.

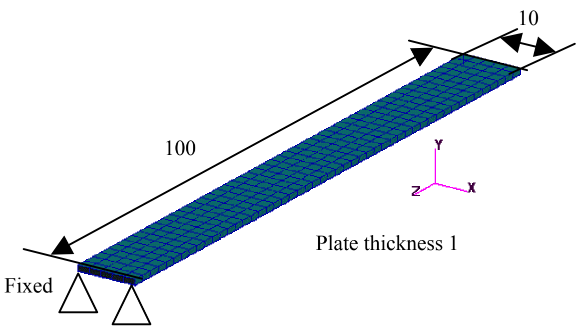


Fig. 5 Shape of model used in analysis

Table 1 FEM elements used in real eigenvalue analysis

	Element Type	Element Characteristic	Standard length of side of 1 element	Number of elements	Number of nodes
Theoretical solution					
Beam element	Beam	Beam	2.0	50	52
Tetrahedron primary element - 0.5mm	Tetra-hedron	Primary element	0.5	40,000	12,663
Tetrahedron primary element - 1mm			1.0	5,000	2,222
Tetrahedron primary element - 2mm			2.0	1,250	612
Tetrahedron secondary element - 1mm		Secondary element	1.0	5,000	11,663
Tetrahedron secondary element - 2mm			2.0	1,250	3,083
Tetrahedron secondary element - 10mm			10.0	50	179
Hexahedron primary element - 2mm	Hexa-hedron	Primary element	2.0	250	612
Hexahedron secondary element - 2mm		Secondary element	2.0	250	2,028

Natural frequencies when the standard lengths of one side of an element are made even to 2 mm are shown in **Fig. 6**. Secondary, primary and secondary natural frequencies of a tetrahedron, hexahedron and hexahedron coincided well with the theoretical values with errors of less than 5%. An error of about 120% was found in each natural mode of the tetrahedron primary element.

A comparison of errors in dynamic rigidity between beam elements and element shapes is shown in **Fig. 7**. In dynamic rigidity also, only the tetrahedron primary element produced an error of about 80%, and the analysis results of the other three element shapes were within 10%.

Fig. 8 compares natural frequencies by differences in the element size and aspect ratio of the tetrahedron elements. Errors of the tetrahedron secondary element with a theoretical solution were less than 10% even with a distortional element with a standard length of the long side of 10 mm (aspect ratio of 10). On the other hand, the accuracy of a tetrahedron primary element improves the finer the element is. However, the error remains 26% even though the element is made finer to 0.5 mm.

The analysis accuracy is considered affected by the ratio with the plate thickness and by the aspect ratio (long side/short side) of the element size and is summed up as shown in **Table 2**.

In accuracy, the hexahedron element is superior, but cannot interact with automatic meshing, leaving no choice but to use a tetrahedron element. By making a secondary element with one side, which is less than 10 times the plate thickness, and with an aspect ratio of 10 or less, errors of natural frequency are reduced to less than 10%. Judging from the dispersions of the measured values of natural frequency, the calculation accuracy can be tolerated at 10%.

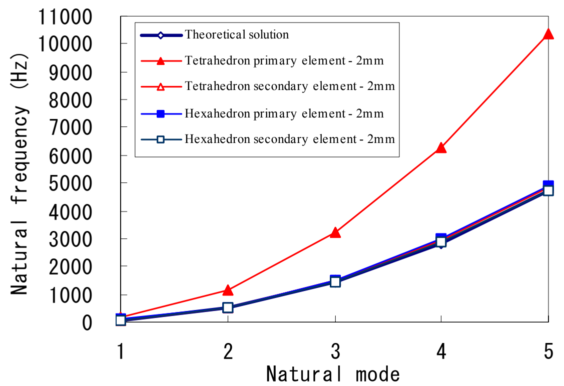


Fig. 6 Comparison of natural frequency by element shape
(A comparison of a standard length of 2 mm of one side is shown in Table 1 by the FEM element.)

Table 2 Comparison of tertiary FEM elements

		Tetrahedron element						Hexahedron element	
		Primary element			Secondary element			Primary element	Secondary element
Element size	Length of 1 side	1/2 of plate thickness	Plate thickness×1	Plate thickness×2	Plate thickness×1	Plate thickness×2	Plate thickness×10	Plate thickness×1	Plate thickness×2
	Aspect ratio	×1	×1	×2	×1	×2	×10	×2	×2
	No. of nodes (*1)	12,663	2,222	612	11,663	3,083	179	612	2,028
Analysis accuracy (error) (*2)		26%	73%	124%	5% or less	5% or less	10% or less	5% or less	1% or less
Auto mesh splitting		Possible	Possible	Possible	Possible	Possible	Possible	Not possible	Not possible
Degree of freedom		○	○	○	△	○	○	◎	△
Overall comment		×	×	×	○	◎	○	△	△
		Analysis accuracy is low. Recommend not to use	Analysis accuracy is low. Recommend not to use	Analysis accuracy is low. Recommend not to use	Many nodes and long analysis time, but can be used.	Auto mesh splitting is possible. High accuracy. Can be used as 3D solid element.	Fewer nodes and high analysis accuracy. Can be used as 3D solid elements.	No auto mesh splitting. Cannot actually be used.	No auto mesh splitting. Cannot actually be used.

×: Cannot be used △: Can be used ○: Good ◎: Excellent

*1 No. of nodes indicates the mesh splitting of plate, which is 10mm in width, 1mm in thickness, and 100mm in length.
*2 Maximum error from primary up to quinary mode

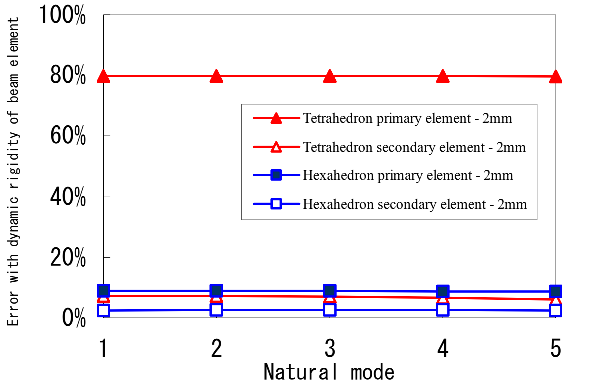


Fig. 7 Comparison of dynamic rigidities of beam element and element shapes
(A comparison by the FEM element with a standard length of 2 mm on one side is shown in Table 1 by the FEM element.)

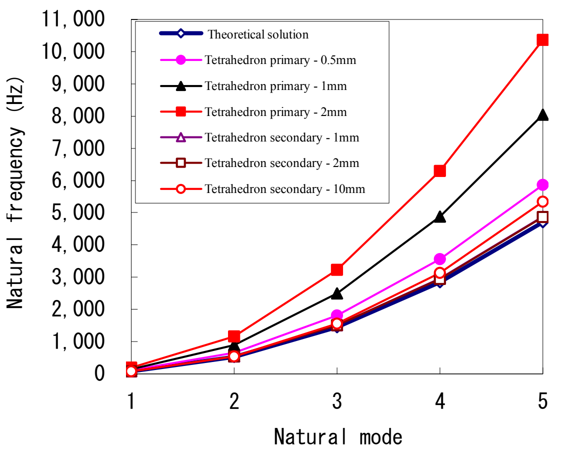


Fig. 8 Comparison of natural frequency by element size

5. Modeling of Clamp between Part and Jig

Modeling of an area where a jig and machined part make contact is important for analyzing large parts, which require consideration for jig rigidity, as shown in **Fig. 9**. When individual shapes are faithfully modeled as shown in **Fig. 10(a)**, contact areas approach point contacts, disabling mesh generation and requiring refabrication of the mesh due to a change in the clamp location. The rigidity of the contact area against a clamping force cannot be expressed correctly. Point contacts are therefore provided where a part and clamp make contact as shown in **Fig. 10(b)**, and modeling is performed using a spring (spring constant k and viscous damping coefficient) between these point contacts to express clamp rigidity.

It is to be remembered that degrees of freedom of a nodal element as a typical point of juncture and of a 3D solid element differ. The nodal element has six degrees of freedom (translational motions and rotational motions in the x, y and z directions) as variables, while the 3D solid element has only three degrees of freedom (translational motions in the x, y and z directions).

For this reason, constraint of a rotary component is needed to connect the two. Rotation was constrained by connecting the degree of freedom of a point element and the degrees of freedom of the plural nodes of a solid element as shown in **Fig. 11**.

The use of these clamp expression methods has enabled a clamp layout in any place without changing the mesh of the analysis model.

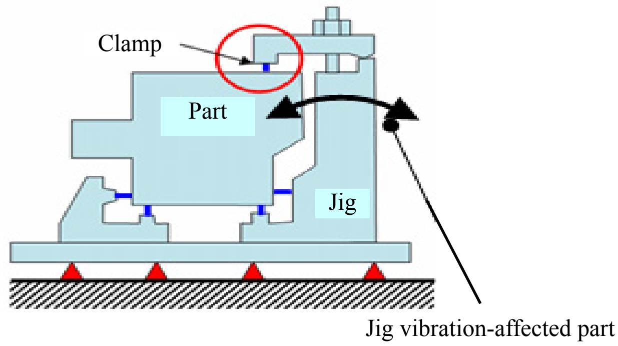


Fig. 9 Conceptual diagram of modeling of contact area

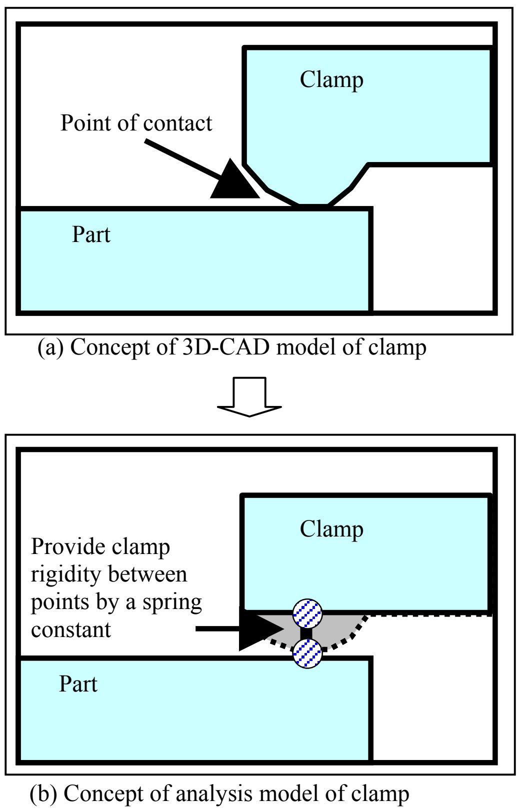


Fig. 10 Conceptual diagram of modeling of contact area

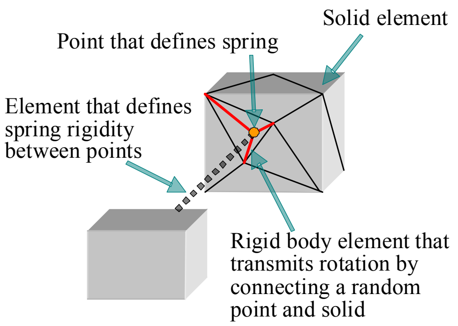


Fig. 11 Method of fixing degrees of freedom of rotation of nodal point and 3D solid element

6. Evaluation Example of Total Dynamic Rigidity of Part and Jig

An evaluation example of the total dynamic rigidity of a part and a jig is described.

Fig. 12(a) shows a large part of construction machinery that is machined by being fixed on a jig measuring 6000 mm in width and 2000 mm in length. The jig has a skeleton construction for moving of the clamp position by a linear guide and ball screw so that plural parts can be mounted and removed in automatic setup.

The design of jig constructions that accomplish jig rigidity, flexibility and controllability was targeted by pre-evaluation using vibration analysis.

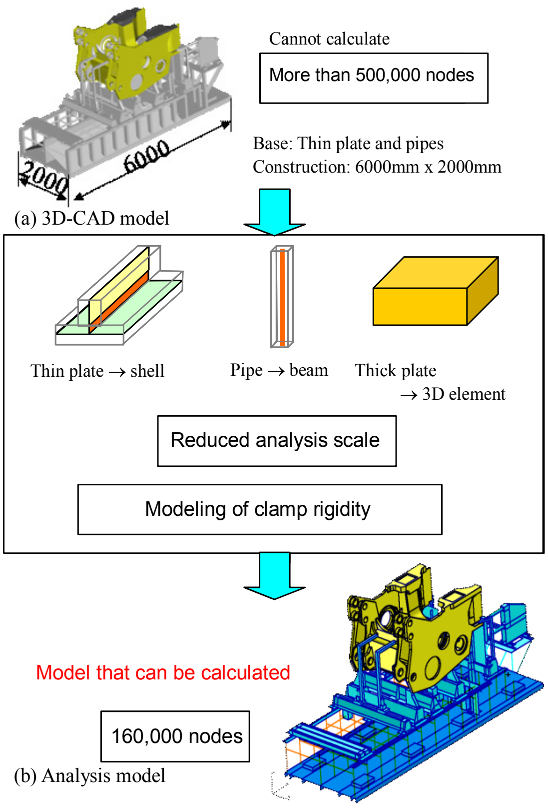


Fig. 12 Analysis Modeling of Entire Construction

The entire construction would become roughly 6000 mm x 2000 mm x 3000 mm, and the scale would exceed 500,000 nodes when the entire model is modeled with 3D solid elements. As shown in **Fig. 12(b)**, the skeleton construction comprising plates and pipes in the base was modeled by combining shell elements, beam elements and 3D solid elements, simplifying to an analysis model with about 160,000 nodes and affording calculations.

The clamp was modeled employing the clamping method described in section 5.

Through a comparison of the dynamic rigidities of a new-construction jig and an existing jig by analysis, the rigidities of the skeleton construction itself and of the clamp were evaluated before fabricating the jig. As a result, the skeleton construction was reinforced and the shape of the clamp was changed. These changes enabled smooth equipment servicing.

After fabricating the jig, vibration was measured and the measurement result was compared with the analysis. In both the measurement and analysis, the natural mode existed in the order of ABC in **Fig. 13** and error between the natural frequency measurements was about 17%.

The estimated value of dynamic rigidity on the machined surface matched the measured value well as plotted in **Fig. 14**.

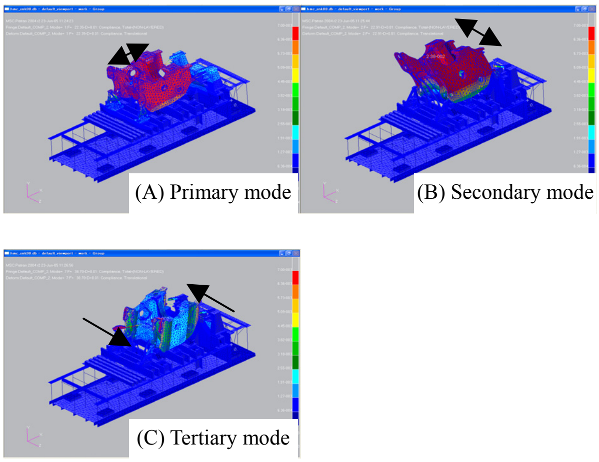


Fig. 13 Vibration mode shape

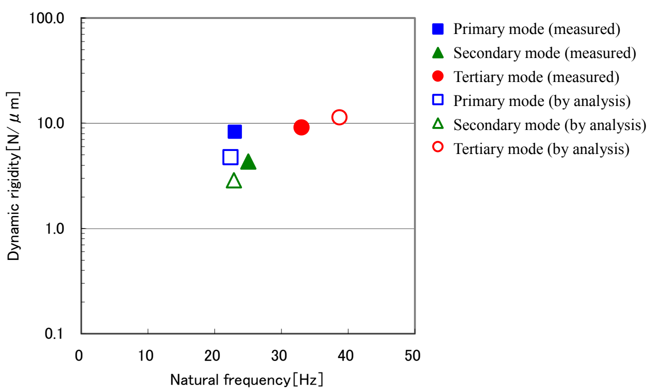


Fig. 14 Comparison of dynamic rigidity between measurement and analysis

7. Conclusion

A new analysis method including analysis of jig rigidity was developed unfolding the conventional dynamic rigidity evaluation technology of machined parts by shell elements into analysis by 3D solid elements utilizing a 3D-CAD model and by taking contact rigidity between a machined part and a jig into consideration. The new technique paves the way for application of vibration analysis technology to large parts with which jig rigidity affects machining.

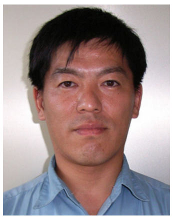
By using this technology at the design stage of jigs for large parts, the dynamic rigidities of the machining systems of parts, jigs and tools can be evaluated at the jig design stage, thereby affording rational jig design and fabrication.

Measurement data on the relationship between dynamic rigidity and chattering as a basis of dynamic rigidity evaluation criteria will be collected to enhance forecast accuracy. Spreading of this technology will be promoted to build an environment for the challenge of building jigs that incorporate new ideas free from conventional established notions.

References:

1) M. Daimon, et al.: Research of Dynamic Characteristic Design of Fixtures for Plate and Box Works, Seimitsu Kikai, vol. 51, No. 12 (1985)
2) Tetsutaro Hoshi: Vibration Analysis of Machining, Kogyo Chosakai, 1990
3) Osamu Taniguchi, et al.: Machine Vibration Theory, pp478, Corona-sha, 1976

Introduction to the writers



Eiji Nabata

Entered Komatsu in 1992.
Currently assigned to the Manufacturing Engineering Development Center, Production Division.

Yuji Terasaka

Entered Komatsu in 1983.
Currently assigned to the Manufacturing Engineering Development Center, Production Division.

[A few words from the writer]

Advances in computer technology are indeed breathtaking. In the past, our agony was how to express shapes and phenomena by simplifying them as best we could possible using various software applications. Now, all designers and engineers build shapes by 3D-CAD and split them into analysis meshes very easily and calculations can be finished at once. One is deluded to believe that correct answers will be obtained forthwith the moment analysis is made. This is a pitfall. One must not be deceived by the computer that provides an answer for the moment. Humans have eyes to search the essence of a problem, to wisely analyze the answer and to discern whether or not it is a fact or a falsehood. This is know-how. Analysis is a convenient tool that makes invisible things visible. We wish to accumulate know-how to discern facts and to spread it.