

Application of Finite Element Analysis Using 3D Auto-mesher for Transmission Development

Shigeru NAGAI*

Takanao UCHIDA*

ABSTRACT

In order to effectively utilize three-dimensional models, the quality of a 3D auto meshing function linked with a 3D CAD system and 3D models was improved. This was achieved with the aim of performing finite element analysis on, without modification of, detail models including detailed fillets, draft angles and other complex shapes which can be applied to die manufacturing. As a result, 3D detail models with a data volume in excess of 100 MB could be converted to CAE models by auto meshing, allowing designers to perform finite element analysis at the transmission design site.

1. Introduction

With the shift to 3D design tools, even cast parts such as the engine and transmission are now designed mainly using detail models including detailed fillets, draft angles and other complex shapes (hereafter “3D detail models”) which can be applied to die manufacturing. Design using these 3D detail models formerly required more process time than 2D drawing design due to the problems of workstation calculation time, insufficient 3D CAD system operation functions, a lack of modeling skills on the part of designers and other reasons. However, workstation performance, 3D CAD system functions, and designers’ skills are improving, and process time problems are gradually decreasing.

The main advantages of 3D design include being able to check interference with other parts using a 3D layout, to check actual shapes through rapid prototyping, and application of 3D detail models to die manufacturing, etc. However, for designers these merits have not offset the increase in process time occurring with the shift from 2D to 3D design.

Therefore, allowing designers themselves to perform finite element analysis directly using 3D detail models was considered as one merit of 3D design for designers.

Based on this background, tetrahedron element auto-mesher functions were improved to allow meshing of a large 3D detail transmission case model which was formerly considered difficult for an auto-mesher. The results showed that finite element analysis can be performed by designers even for large 3D detail models.

2. Aim of 3D Auto-mesher Improvement

For designers to perform finite element analysis, they must learn how to use finite element analysis software, and the 3D model data must be converted from the 3D design tool to that analysis software. The task of learning how to use these many types of software programs is a large burden for designers.

Therefore, if the finite element analysis functions that are a part of Honda’s standard 3D CAD system can be used for analysis, this would have the following merits with respect to the above two requirements. i) The basic operations and user interface are shared, so the finite element analysis functions can be easily learned. ii) Data conversion is not necessary.

For these reasons, using the 3D CAD system equipped with finite element analysis functions that is currently used in-house was thought to be an efficient way to introduce finite element analysis to designers.

For designers to perform finite element analysis on 3D detail models, the auto-mesher function that is a standard function of the 3D CAD system needed to be improved. In addition, changes were also necessary to enable designers to perform finite element analysis in a short time on the same 3D CAD system while performing design work with 3D models. (See Fig. 1.)

2.1 Conventional CAE Models

In the past, performing finite element analysis on 3D detail models required special 3D models for finite element

* Tochigi R&D Center

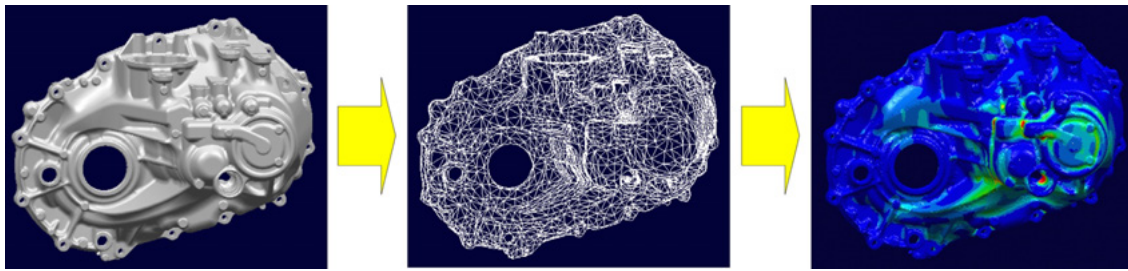


Fig. 1 Finite element analysis of a 3D detail model using the 3D auto-mesher

analysis which omitted detailed fillets, minute surfaces, minute holes and other detailed shapes. This was mainly for the following two reasons: i) It was difficult for the auto-mesher to generate a mesh following detailed shapes. ii) After mesh generation, the number of nodes and elements of the resulting CAE model exceeded the workstation calculation capacity.

In addition, in many cases even with a small number of nodes and elements, hexahedron elements with high accuracy of calculation results had to be used, or meshing had to be performed manually using shell elements in some parts to obtain appropriate results in a short process time. This was mainly done to effectively utilize scarce workstation system resources and to obtain accurate results.

For reference, a full-time CAE staff member sometimes required over 200 hours to create a CAE model of the cylinder block and other engine parts. In addition, decisions such as whether ribs created to disperse stress should be simplified or eliminated required considerable know-how.

2.2 Finite Element Analysis Performed by Designers

For reasons given in the previous section, finite element analysis was generally performed by full-time CAE staff members rather than designers, from the viewpoint of process time. In addition, designers generally followed a process flow in which shapes were determined based on intuition and experience, prototypes were manufactured, and design specifications were determined from actual measurement test results.

However, auto-mesher performance has improved in recent years, and it has become possible to generate meshes using the auto-mesher for palm-sized parts having few small fillets or other minute surfaces and then to perform calculations using this data. It is actually becoming possible for designers to perform finite element analysis for small parts during the design work. There are even some parts for which actual measurement tests are omitted and specifications are determined using only the results of finite element analysis.

The merits of designers performing finite element analysis by themselves include: i) Test results can be simulated at the design site even without the actual part, so the validity of ideas can be confirmed in real time. ii) The surrounding layout, product cost, die configuration and other limitations for that part are easily available, so countermeasures can be formulated in an efficient manner.

iii) Designers can confirm their own design ideas repeatedly in a short time, leading to improvements in their design sense.

The transmission case, which is one of the large parts introduced here, is comprised overall of curved surfaces and fillets; it has almost no flat surfaces, which made auto-meshing difficult thus far. In addition, manual meshing not only took time, but there was also the possibility of analysis being performed which omitted the parts where the maximum stress occurred. Thus, fillet removal and other shape simplifications required considerable know-how. Therefore, finite element analysis was almost never performed for the transmission case due to problems related to analysis time and accuracy. On the other hand, an auto-mesher capable of applying finite element analysis even to large, complex parts such as the transmission case was greatly desired, as this would make it possible to perform accurate finite element analysis in a short time.

2.3 Auto-mesher Direction

Hardware has made remarkable advances in recent years, with speeds increasing and prices falling. Workstations now feature performance on a level formerly only available in mainframe computers.

The conventional approach for creating CAE models was to i) construct individual know-how including the removal of minute surfaces and fillets, ii) perform time-consuming manual meshing, and iii) reduce the numbers of nodes and elements. However, in consideration of these hardware advances, it is clearly more efficient to mesh a 3D detail model with the auto-mesher, create a CAE model and then perform finite element analysis.

In addition, if auto-meshing is possible for 3D detail models, then not only the finite element analysis solver accuracy, but also the 3D detail model shape accuracy and the shape accuracy of the CAE model after meshing can also be taken into account when discussing accuracy of analysis results. Thus, accuracy of results can be pursued from various angles.

Based on the above, the three functions demanded of the auto-mesher are as follows. i) Auto-meshing of 3D detail models should be possible. ii) It should be possible to control the numbers of nodes and elements to optimize the calculation time and overall calculation accuracy. iii) Definition of local detailed element sizes should be possible to improve the accuracy of specific parts.

3. 3D Auto-mesher Improvements

In order for designers to perform finite element analysis during transmission development, the functions required of this 3D auto-mesher are the following two items.

- i) Reliable mesh generation should be possible (= efficient use of time).
- ii) Element quality should be good (= ensuring accurate analysis results).

The specific targets for these requirements when improving 3D auto-mesher functions are the following two items.

With respect to all 3D detail models,

- i) The mesh success rate should be 90% or more.
- ii) The length ratio should be 10 or less.

(The mesh success rate and length ratio are described below.)

3.1 Specific Evaluation Methods and Conditions

In order to achieve these targets, specific methods were determined for evaluating the mesh success rate and length ratio.

3.1.1 Evaluation Methods

- i) Mesh success rate

Using the total number of times meshing was performed as the denominator, the ratio of the number of times a mesh was obtained which satisfied the element quality standards of the finite element analysis solver integrated into the 3D CAD system was defined as the “mesh success rate” as shown in equation (1).

$$MS = VM / MT \quad (1)$$

Here, MS is the mesh success rate, VM is the number of times a calculable mesh was obtained, and MT is the number of times meshing was performed.

- ii) Length ratio

The worst length ratio value was checked to ensure calculation results accuracy.

Here, “length ratio” is the parameter generally referred to as “aspect ratio.” (See Fig. 2.)

In addition to the length ratio, other element quality items such as the Jacobian and shape factor were also checked at the same time to see whether they could be accommodated by the finite element analysis solver.

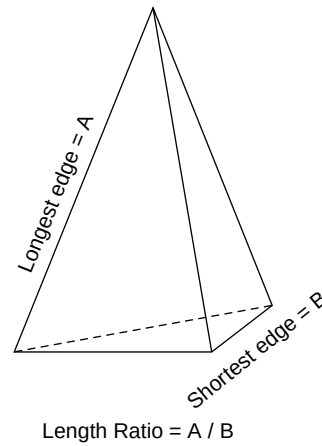


Fig. 2 Length ratio

Of the various element quality standards, the length ratio was used as the standard for the following three reasons. i) When checking the element quality of the generated mesh, the length ratio was the only item that frequently exceeded the finite element analysis solver standard value. ii) The length ratio directly influences the calculation results accuracy. iii) The length ratio is one of the element quality items that is appropriate as an index for ensuring calculation results accuracy.

The calculation fails if even a single element in the mesh exceeds the element quality standard values. It is said that elements with a length ratio of 15 or more should generally be avoided.

3.1.2 Evaluation Conditions

- i) Evaluation model

Five 3D detail models in existence at the time evaluation started were arbitrarily selected as evaluation models.

Fig. 3 shows some of these models. The numbers in the figures indicate the size of that model.

- ii) Meshing conditions

At this point, tests were conducted using a combination of the two meshing parameters of size and absolute sag. (See Fig. 4 and Fig. 5.)

These two parameters were set so that there were approximately 150,000 nodes (approximately 80,000 to 90,000 elements) after meshing the 3D detail model. This



Fig. 3 Evaluation models

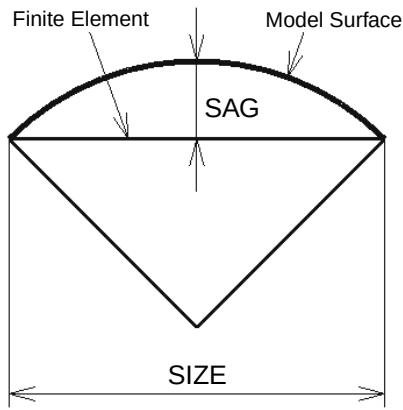


Fig. 4 Size & sag

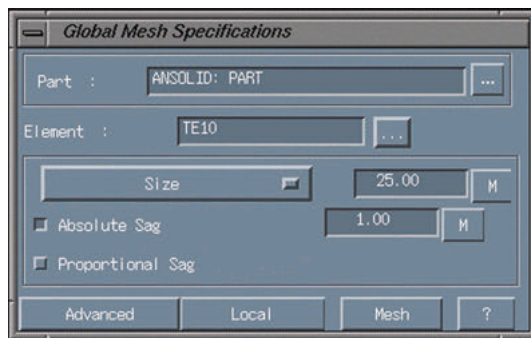


Fig. 5 Meshing parameter settings window

figure of 150,000 nodes is the number of nodes that can be, for the most part, reliably calculated with the 4 GB temporary storage area that can be used by the design workstations currently used for development.

In addition, the same combination of parameters (size and absolute sag) set for the initial evaluation were also used for subsequent evaluations.

3.2 Evaluation Results

The 3D auto-mesher was evaluated in accordance with the above evaluation method and conditions.

3.2.1 Step 1 Test

- Aim: Detailed investigation of functions
- Results: The average mesh success rate for five models was approximately 15%.

This mesh success rate of 15% means that when meshing is performed ten times, the number of times a mesh that can

be used for calculations is obtained is less than two times. (See Table 1.) For designers who must carry out numerous tasks in a limited time, a mesh success rate of 15% makes the meshing work itself a burden.

Therefore, it was judged that this auto-mesher version could not be used for finite element analysis by designers without modification.

• Analysis of meshing results:

Analysis of the meshing failures (hereafter “meshing errors”) which accounted for 85% in the Step 1 evaluation showed that these meshing errors can be classified into two main types: i) Errors rooted in the 3D detail model, and ii) Errors rooted in the meshing program.

i) Errors rooted in the 3D detail model

The parts of 3D detail models that cause meshing errors are the minute portions which are no larger than the standard 3D CAD system tolerance (0.1 mm). Typical examples include the overlapped face and twisted face shown in Fig. 6.

These minute portions are circled in red when they become meshing errors, but the shapes are automatically created by the 3D CAD system, and these portions are extremely difficult to find before meshing starts.

Therefore, the quality of the 3D detail model itself must be improved by enhancing the modeling functions of the 3D CAD system in order to completely avoid errors rooted in the 3D detail model.

In addition to this fundamental solution, errors rooted in 3D detail models can also be avoided by the following two measures.

- Changing the mesher program – The mesher program can be optimized to ignore low quality portions of the model as appropriate.
- Modifying the 3D model – Errors can be avoided by having designers modify portions where problems occur as

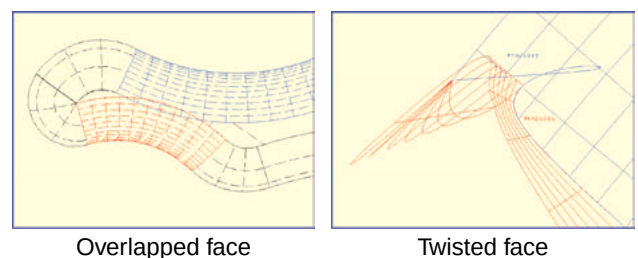


Fig. 6 Problematic geometry examples

Table 1 Step 1 result

Model	A	B	C	D	E	Total
No. of meshing trials	44	42	18	38	12	154
No. of successful meshes	5	2	0	9	7	23
Mesh success rate	11.36%	4.76%	0.00%	23.68%	58.33%	14.94%
Meshing failure details						
3D model quality	28	6	18	17	4	73
Element quality	8	32	0	10	1	51
Others	3	2	0	2	0	7

indicated by the red circles. This method is intertwined with 3D modeling, so it is omitted here.

ii) Errors rooted in the meshing program

Most of these errors are length ratio (element quality) problems. Even when meshing is completed normally, if there are elements with a length ratio value of 50 or more, the data cannot be used for calculations. To solve these length ratio problems, the mesher program must be improved to prevent inferior quality elements from being generated.

In addition, there are also program errors where the mesher program itself causes an error and meshing is completed abnormally.

• Measures for the next version:

Based on the Step 1 results, the following three changes were made to the mesher program.

- The program was changed so that low quality portions on 3D detail models can be ignored.
- Length ratio values were improved.
- The program was debugged.

3.2.2 Step 2 Test

• Aim: To confirm the degree of mesher improvement target achievement.

• Results: A mesh success rate of 88% was achieved, which is a vast improvement. (See Table 2.)

The greatest change in this step was that the occurrence of length ratios of the standard value 50 or more was significantly reduced from 33% to 2% of the total number of meshings.

However, another problem occurred that elements with negative Jacobian values were generated. Negative Jacobian values are an impossible condition for finite element analysis calculations.

• Measures for the next version:

Based on the Step 2 results, the mesher program was changed to prevent elements having negative Jacobian values from being generated.

3.2.3 Step 3 Test

• Aim: To confirm the degree of mesher improvement target achievement.

• Results: No elements with negative Jacobian values were generated, but the mesh success rate dropped from 88% to 86%.

In addition, the meshing error rate due to length ratio also increased from 2% to 5%. (See Table 3.)

3.3 Evaluation Summary

Based on the above results, the improvement in mesher functions by Steps 1 to 3 is as follows. (See Fig. 7.)

• The average mesh success rate for the five models rose from 15% to 86%.

This improvement is broken down as follows:

• The meshing error rate due to a length ratio of 50 or more decreased from 33% to 5% (average value of the five models).

• The meshing error rate due to 3D detail model quality decreased from 47% to 8% (average value of the five models).

Thus, it can be said that the target was for the most part achieved.

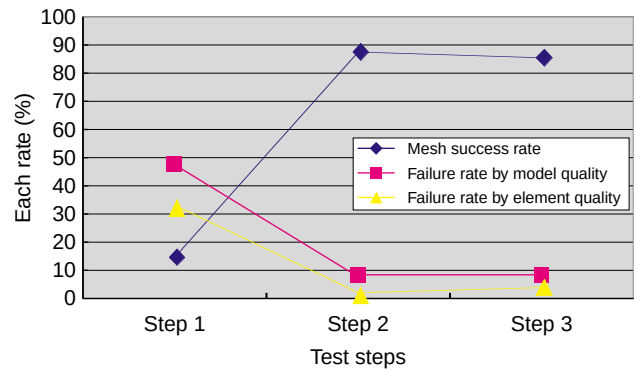


Fig. 7 Transition of results (5 model average)

Table 2 Step 2 result

Model	A	B	C	D	E	Total
No. of meshing trials	44	42	18	38	12	154
No. of successful meshes	37	41	10	36	11	135
Mesh success rate	84.09%	97.62%	55.56%	94.74%	91.67%	87.66%
Meshing failure details						
3D model quality	7	1	3	2	0	13
Element quality	0	0	2	0	1	3
Others	0	0	3	0	0	3

Table 3 Step 3 result

Model	A	B	C	D	E	Total
No. of meshing trials	44	42	18	38	12	154
No. of successful meshes	38	38	11	33	12	132
Mesh success rate	86.36%	90.48%	61.11%	86.84%	100.00%	85.71%
Meshing failure details						
3D model quality	6	0	2	5	0	13
Element quality	0	4	3	0	0	7
Others	0	0	2	0	0	2

4. Introduction of Actual Stress Analysis Examples

This section introduces some actual examples of unit part stress analysis carried out by designers using this auto-mesher.

Fig. 8 shows some actual stress analysis examples. The models used in these analyses are 3D detail models with absolutely no fillet elimination or model modification performed for finite element analysis.

These analyses use parabolic tetrahedron elements, and have approximately 150,000 nodes (approximately 90,000 elements).

It used to take a full-time staff member approximately 80 hours just to perform manual meshing (no fillets, approximately 2000 nodes) for the transmission case. However, boundary conditions can currently be added on a workstation in approximately 10 minutes, and calculations including auto-meshing can be completed in approximately 45 minutes. For reference, the time required for meshing alone is 7 or 8 minutes out of the approximately 45 minutes of calculation time.

Fig. 9 shows a comparison of finite element analysis results for the model shown in Fig. 1 and the results of stress measurement using a strain gauge at each measurement point. These comparison results indicate that finite element analysis is effective for determining design specifications in terms of both trends and absolute values.

5. Conclusion

Based on the above,

- i) Auto-meshing is now possible for 3D detail models; this is something which used to be called impossible.
- ii) Finite element analysis can be performed by designers, and is being utilized at actual design sites.
- iii) The introduction of finite element analysis by designers has reduced the number of design changes and actual measurement tests.

6. Afterword

Finite element analysis was formerly requested of full-time analysis staff, and several days to several weeks were required to receive the results. However, finite element analysis of 3D detail models can now be finished in

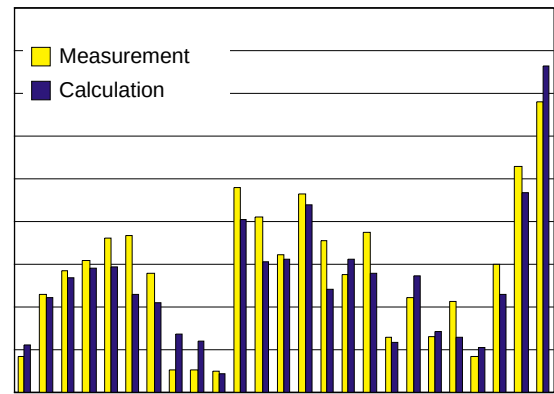


Fig. 9 Comparison of measurement and calculation results

approximately one hour using the improved auto-mesher function. Finite element analysis using the 3D auto-mesher dramatically reduces wasted time on the part of designers waiting for analysis results, and is becoming a useful tool that allows design specifications to be investigated in a short time.

The number of actual measurement tests has decreased and development efficiency is improving steadily by having designers determine specifications with high accuracy using finite element analysis.

7. Acknowledgements

The authors wish to express their heartfelt thanks to Dasseau Systems Inc. (France) for understanding the authors' concepts, working with the authors to set specific target values, and actively supporting improvements in 3D auto-mesher functions so that designers can perform finite element analysis.

8. References

- (1) Miyoshi, T.: Introduction to the Finite Element Method, Tokyo, Saifuukan, 1994, p.179-183
- (2) Higashimachi, T.: Finite Element Method Know-how, Tokyo, Morikita Shuppan, 1993, p.44-47
- (3) Takahashi, K., Takahashi, Y., Semba, K.: Utilization of Main Frame Computers for Analysis of Power Train

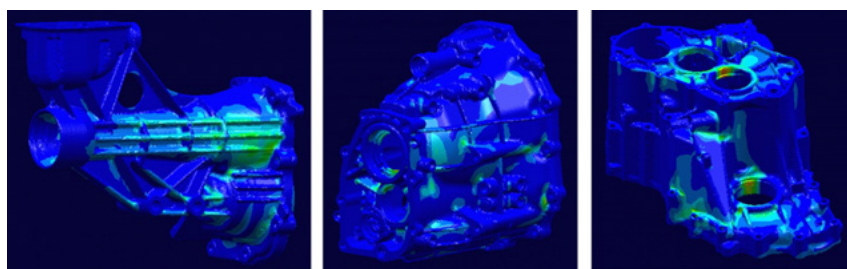


Fig. 8 Finite element analysis examples of 3D detail models

- Vibration, Automotive Technology Association,
Scientific Lecture Paper 901, p.65-68 (1990)
- (4) Semba, K., Takahashi, K., et al.: Finite Element Eigen
Analysis of Front-wheel Drive Power Train, 7th IMAC
Vol. 9 (1989)

■ authors ■



Shigeru NAGAI



Takanao UCHIDA