# CT-FEM System (Ver. 3.0)

## 0. Introduction

Since its initial release in September 2004, the CT-FEM System has been praised by many specialists involved in the limit design of gears because of the product's well-matched laboratory results against pre-analysis simulation for gear pairs with bias modification or axis angle errors. The product's high reputation is partly because its operation is similar to ordinary gear designing.

The CT-FEM System (Ver. 3) includes a number of new features that respond to our customer needs. For details, refer to the following sections.

## 1. Summary

Stress analysis using the finite element method (FEM) is a strength analysis on the order of millimeters. On the other hand, the analysis of meshed tooth profiles is on the order of micrometers. Moreover, FEM in general does not produce good results in contact problem analyses and it has been considered that analyzing stresses on a pair of meshed gears, taking into account the tooth profile shape, is not possible using FEM.

The CT-FEM System's newly developed FEM gear stress analysis software is capable of handling gear contact problems. It features flexible operation for machine designers. Fig. 1.1 shows the application window of the CT-FEM System.

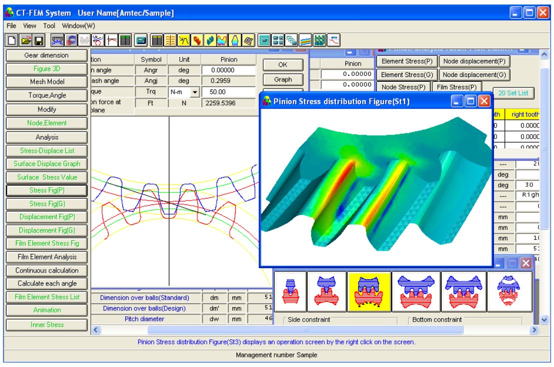


Fig. 1.1 CT-FEM System

## 2. Initial Setting

2.1 Applicable gear pair: Involute gear

- External gear × External gear
- External gear × Internal gear

2.2 Basic rack: Full depth tooth, stub gear tooth, and special tooth

The gear property and mesh setting windows are shown in Fig.s 2.1 and 2.2, respectively.

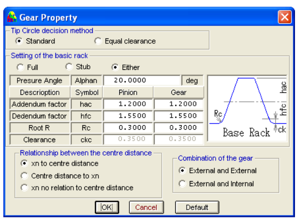


Fig. 2.1 Initial Gear Property Setting

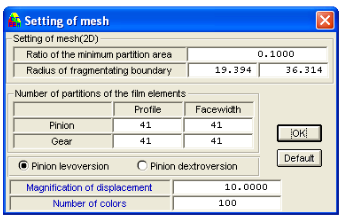
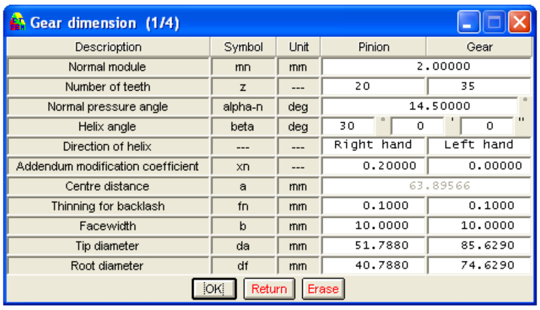


Fig. 2.2 Mesh Setting

## 3. Gear Dimension Entry (1/4)

An example of FEM analysis of a pinion/gear pair is shown below. As shown in Fig. 3.1, gear dimension entry is a straightforward process; the user simply specifies gear dimensions from "Normal module" up to "Root diameter" in sequence. To provide a backlash, specify a longer center distance. Fig.s 3.2 and 3.3 show the 3D rendering of the pinion/gear under discussion. The CT-FEM System is capable of showing only the film elements of the tooth faces. This feature facilitates the observation of the face-to-face contact when an intersecting/parallelism angle error and a tooth profile modification are specified.



Gear dimension (1/4)

| Description | Symbol | Unit | Pinion | Gear |
|---|---|---|---|---|
| Normal module | mn | mm | 2.00000 | |
| Number of teeth | z | --- | 20 | 35 |
| Normal pressure angle | alpha-n | deg | 14.50000 | |
| Helix angle | beta | deg | 30 ° 0 ' 0 " | |
| Direction of helix | --- | --- | Right hand | Left hand |
| Addendum modification coefficient | xn | --- | 0.20000 | 0.00000 |
| Centre distance | a | mm | 63.89566 | |
| Thinning for backlash | fn | mm | 0.1000 | 0.1000 |
| Facewidth | b | mm | 10.0000 | 10.0000 |
| Tip diameter | da | mm | 51.7880 | 85.6290 |
| Root diameter | df | mm | 40.7880 | 74.6290 |

OK Return Erase

Fig. 3.1 Gear Dimension Settings

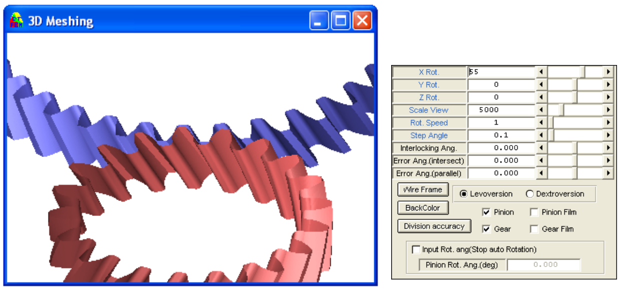


Fig. 3.2 Tooth Profile Rendering

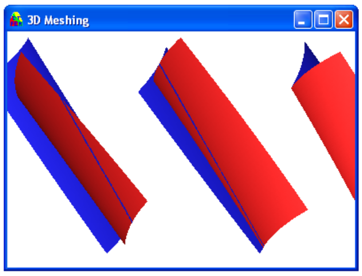


Fig. 3.3 Film Element Profile Rendering

## 4. Mesh Generation (2/4)

For FEM analysis, the CT-FEM System uses the secondary tetrahedral elements (see Fig. 4.1) to divide gears into meshes automatically. Since, in the mesh division condition settings, the division accuracy can be specified separately at the root, face, and tip of the pinion/gear, as shown in Fig. 4.2, you may choose to generate finer meshes for portions subjected to stress concentration and coarse meshes for the remainder.

The generated meshes can be viewed as shown in Fig. 4.3 and Fig. 4.4. In these figures, the mesh with the maximum flat degree among the generated meshes can be viewed as well. If you are not satisfied with the generated meshes, you can regenerate meshes by changing the division accuracy and number of nodes settings in Fig. 4.2.

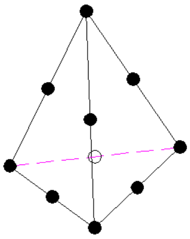


Fig. 4.1 Secondary Tetrahedral Elements

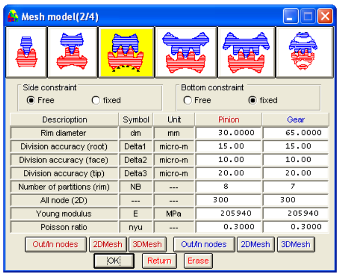


Fig. 4.2 Mesh Generation Conditions

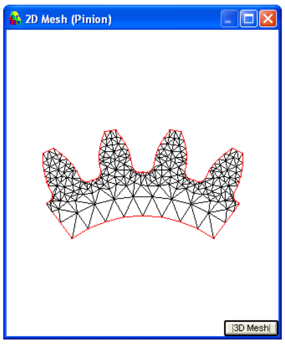


Fig. 4.3 2D Mesh

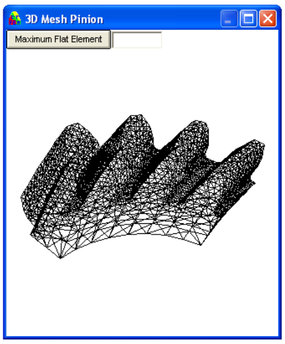
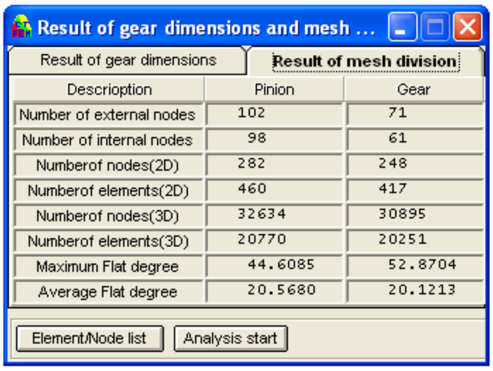


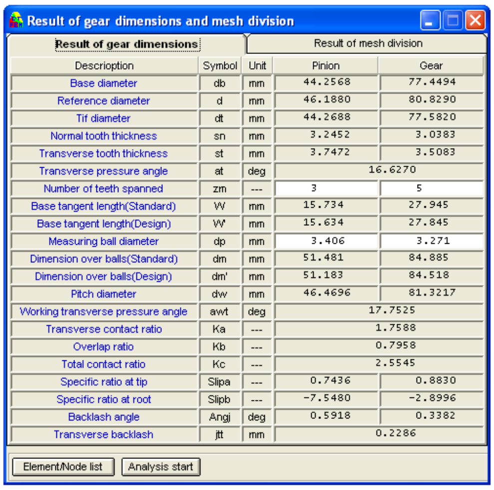
Fig. 4.4 3D Mesh

Fig. 4.5 shows the result of automatic mesh generation based on the mesh generation conditions shown in Fig. 4.2. As you can see, for the pinion, the number of 3D nodes is 32634 and the number of 3D elements is 20770. Fig. 4.6 shows the dimensions of the pinion/gear under discussion.



| Descrioption | Pinion | Gear |
|---|---|---|
| Number of external nodes | 102 | 71 |
| Number of internal nodes | 98 | 61 |
| Numberof nodes(2D) | 282 | 248 |
| Numberof elements(2D) | 460 | 417 |
| Numberof nodes(3D) | 32634 | 30895 |
| Numberof elements(3D) | 20770 | 20251 |
| Maximum Flat degree | 44.6085 | 52.8704 |
| Average Flat degree | 20.5680 | 20.1213 |

Fig. 4.5 Result of Mesh Division



| Descrioption | Symbol | Unit | Pinion | Gear |
|---|---|---|---|---|
| Base diameter | db | mm | 44.2568 | 77.4494 |
| Reference diameter | d | mm | 46.1880 | 80.8290 |
| Tif diameter | dt | mm | 44.2688 | 77.5820 |
| Normal tooth thickness | sn | mm | 3.2452 | 3.0383 |
| Transverse tooth thickness | st | mm | 3.7472 | 3.5083 |
| Transverse pressure angle | at | deg | 16.6270 | |
| Number of teeth spanned | zm | --- | 3 | 5 |
| Base tangent length(Standard) | W | mm | 15.734 | 27.945 |
| Base tangent length(Design) | W' | mm | 15.634 | 27.845 |
| Measuring ball diameter | dp | mm | 3.406 | 3.271 |
| Dimension over balls(Standard) | dm | mm | 51.481 | 84.885 |
| Dimension over balls(Design) | dm' | mm | 51.183 | 84.518 |
| Pitch diameter | dw | mm | 46.4696 | 81.3217 |
| Working transverse pressure angle | awt | deg | 17.7525 | |
| Transverse contact ratio | Ka | --- | 1.7588 | |
| Overlap ratio | Kb | --- | 0.7958 | |
| Total contact ratio | Kc | --- | 2.5545 | |
| Specific ratio at tip | Slipa | --- | 0.7436 | 0.8830 |
| Specific ratio at root | Slipb | --- | -7.5480 | -2.8996 |
| Backlash angle | Angj | deg | 0.5918 | 0.3382 |
| Transverse backlash | jt | mm | 0.2286 | |

Fig. 4.6 Results of Gear Dimensions

## 5. Rotation Angle and Torque Setting (3/4)

As shown in Fig. 5.1 and 5.2, the contact point(s) of the meshed gears can be adjusted by changing the pinion rotation angle. The graph in Fig. 5.3 shows the relationship between the contact diameter and roll length. The pinion and gear are in two-point contact where the roll length matches and in single-point contact otherwise. The rotation angle and the contact diameter at these contact points are indicated. The CT-FEM System supports up to three-point contact analysis; the user will be prompted to choose the number of faces to be analyzed between 3 and 1. The Rotation angle reference window shown in Fig. 5.4 may be used to obtain the pinion/gear diameter of the contact point, pinion/gear roll length, and pinion rotation angle by specifying only one of these five elements. If you want to know the rotation angle when the pitch circle diameter of the pinion makes contact, specify the "Diameter of the contact point" for the pinion in Fig. 5.4. Then, the pinion rotation angle required in Fig. 5.1 will be known.

If the contact ratio is between 1 and 2 inclusive, when the tip of the pinion makes contact, the adjacent tooth profile is also in contact (two-point contact). However, if the pinion makes contact at the center of its whole depth, it is in a single-point contact state. This means it is not possible to determine which rotation angle causes the maximum stress unless you collect multiple design data. Also, it should be noted that the rotation angle causing the maximum tooth face stress and the rotation angle causing the maximum tooth root stress do not always match. In the CT-FEM System, you specify the pinion rotation angle measured at the center of the face width. For helical gears, this may be difficult to view in the 2D mesh model. If so, use the 3D meshing model (see Fig.3.2 and Fig. 3.3).

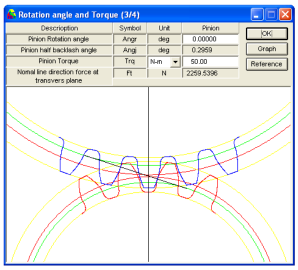


Fig. 5.1 Rotation Angle Setting 1

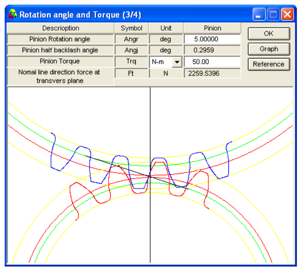


Fig. 5.2 Rotation Angle Setting 2

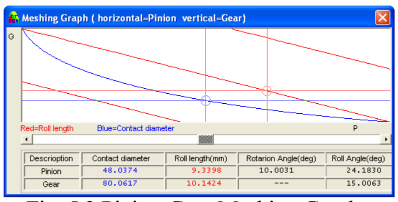


Fig. 5.3 Pinion-Gear Meshing Graph

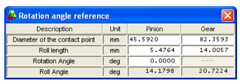


Fig. 5.4 Rotation Angle Calculations

## 6. Tooth Modification (4/4)

### 6.1 Tooth Modification (Standard)

Fig. 6.1 shows the setting window for the tooth modification and gearshaft mounting error data. The CT-FEM System provides four kinds of standard-form profile/flank modifications. As shown in Fig. 6.2 and 6.3, tooth modifications can be specified by the roll length, modification amount, and profile radius, which can be viewed as graphs in Fig.6.4 and 6.5. The system also supports the tooth profile, face, and flank modifications. These custom tooth modifications may be specified in the window shown in Fig. 6.6.

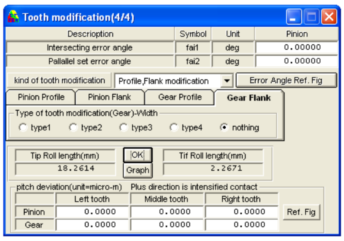


Fig. 6.1 Profile and Flank Modification Settings (Standard)

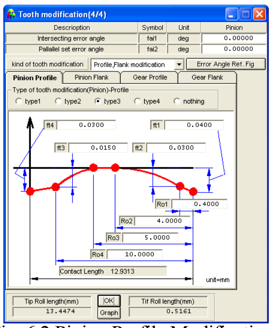


Fig. 6.2 Pinion Profile Modifications (Type3)

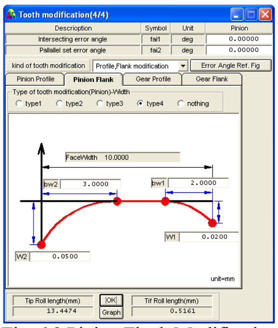


Fig. 6.3 Pinion Flank Modifications (Type4)

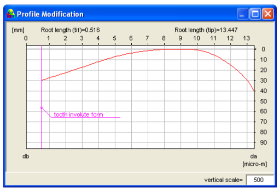


Fig. 6.4 Profile Modifications

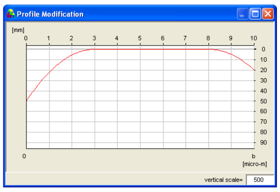


Fig. 6.5 Flank Modifications

## 6.2 Tooth Modification (Biased) - Optional

As a custom modification, the CT-FEM System provides profile modifications (see Fig. 6.7), flank modifications (see Fig. 6.8), and profile-based or flank-based face modifications (see Fig.6.9 and 6.10). The profile modification graph can be divided into up to 20 graphs, providing a simple error setting method as shown in Fig. 6.11. Moreover, topography can be used for checking how the modifications occur.

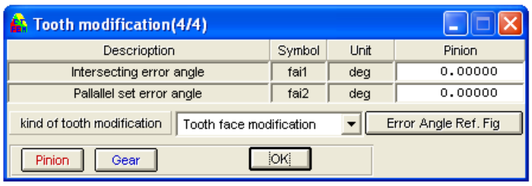



Fig. 6.6 Profile and Flank Modification Settings (Custom)

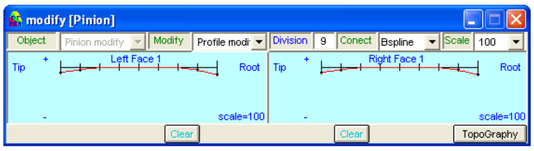


Fig. 6.7 Profile Modifications

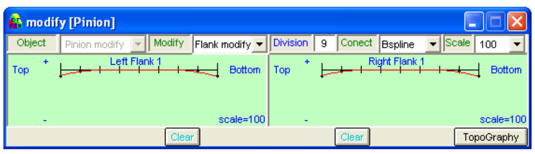


Fig. 6.8 Flank Modifications

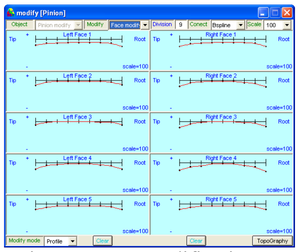


Fig. 6.9 Face Modifications (Profile-Based)

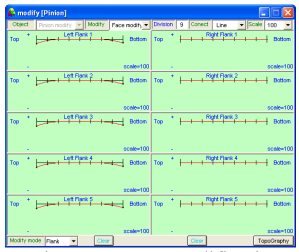


Fig. 6.10 Face Modifications (Flank-Based)

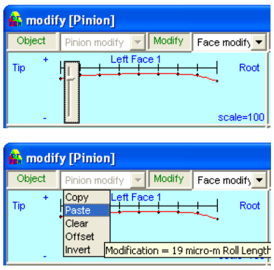


Fig. 6.11 Error Setting Method

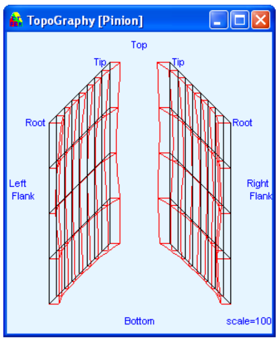



Fig. 6.12 Topography

After specifying an error angle and modification amount, you can view the face-to-face contact state in the Tooth Profile Rendering window shown in Fig. 3.3. This feature is useful when checking the design data for validity and entry errors prior to the analysis stage.

## 6.3 Gearshaft Mounting Error Angle - Optional

Fig. 6.13 shows the window for setting the gearshaft intersecting and parallelism error angles.

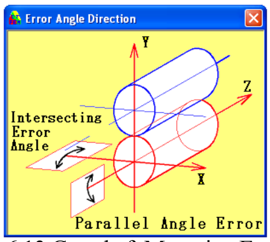



Fig. 6.13 Gearshaft Mounting Error

## 6.4 Pitch Error - Optional

In the pitch error setting, a separate value can be specified for each of the film elements, "Left tooth," "Middle tooth," and "Right tooth." For example, assume a two-point contact state in which the pinion and gear make contact at the "Left tooth" and "Middle tooth." In this case, specifying an error amount of 1 μm at the "Left tooth" will enlarge the contact area on the "Left tooth" accordingly because the profile will be shifted by the specified amount from the theoretical position.

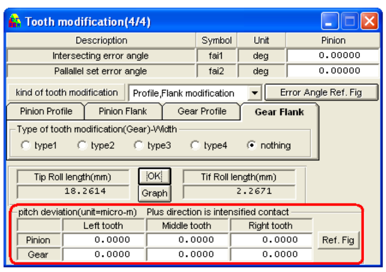


Fig. 6.14 Pitch Error Entry

## 7.1 Analysis

Once (1) gear dimension settings, (2) mesh generation settings, (3) rotation angle and torque settings, and (4) tooth modification/error angle settings have been completed, you can start an analysis. However, before doing so, your design data can be saved.

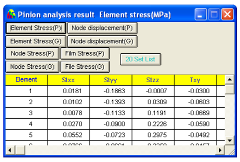
An analysis consists of (1) tooth face stress calculation and (2) FEM analysis of the mesh models, performed in this order. The CT-FEM System also provides options for running multiple analysis processes efficiently. These options may be used as required.

When the analysis is completed, the analysis results can be viewed using the following windows: (1) FEM Analysis Result List for showing the element stress, node displacement, node stress, and film stress values, (2)Stress Distribution Fig., (3) Displacement Distribution Fig., (4) Film Element Stress Distribution Fig., (5) External Surface Maximum Stress List, (6) Displacement of Face Graphs, (7) Film Element Stress Color Distribution List, and (8) Inner Stress List.

The analysis results will show a total of 10 types of stresses, that is, six stress components, three principal stresses, and one equivalent stress. On the other hand, displacement will be reported in four types: displacement in the x-, y-, and z-axis directions and the total displacement.

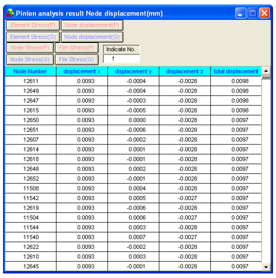
## 8 FEM Analysis Result (Stress/Displacement Value List)

Fig. 8.1 is a FEM Analysis Result list showing stress values. Clicking a column header sorts the stress values in that column in an ascending order. This feature is convenient when you check the node or element number where the stress becomes the maximum. After this check, you can set the element and node to flash in stress distribution figures, so that you can easily notice where in the figure the stress becomes the maximum.




| Element | Stxx | Styy | Stzz | Txy |
|---|---|---|---|---|
| 1 | 0.0181 | -0.1863 | -0.0007 | -0.0300 |
| 2 | 0.0102 | -0.1393 | 0.0309 | -0.0603 |
| 3 | 0.0078 | -0.1133 | 0.1191 | -0.0669 |
| 4 | 0.0270 | -0.0900 | 0.2226 | -0.0590 |
| 5 | 0.0552 | -0.0723 | 0.2975 | -0.0492 |

Fig. 8.1 Stress/Displacement Value List (showing 10 types of stress values)




| Node Number | displacement x | displacement y | displacement z | total displacement |
|---|---|---|---|---|
| 12611 | 0.0093 | -0.0004 | -0.0028 | 0.0098 |
| 12649 | 0.0093 | -0.0004 | -0.0028 | 0.0098 |
| 12647 | 0.0093 | -0.0003 | -0.0028 | 0.0098 |
| 12615 | 0.0093 | -0.0005 | -0.0028 | 0.0098 |
| 12650 | 0.0093 | 0.0000 | -0.0028 | 0.0097 |
| 12651 | 0.0093 | -0.0006 | -0.0028 | 0.0097 |
| 12607 | 0.0093 | -0.0002 | -0.0028 | 0.0097 |
| 12614 | 0.0093 | 0.0001 | -0.0028 | 0.0097 |
| 12618 | 0.0093 | -0.0001 | -0.0028 | 0.0097 |
| 12648 | 0.0093 | 0.0002 | -0.0028 | 0.0097 |
| 12652 | 0.0093 | -0.0001 | -0.0028 | 0.0097 |
| 11508 | 0.0093 | 0.0004 | -0.0028 | 0.0097 |
| 11542 | 0.0093 | 0.0005 | -0.0027 | 0.0097 |
| 12619 | 0.0093 | -0.0006 | -0.0028 | 0.0097 |
| 11504 | 0.0093 | 0.0006 | -0.0027 | 0.0097 |
| 11544 | 0.0093 | 0.0003 | -0.0028 | 0.0097 |
| 11540 | 0.0093 | 0.0007 | -0.0027 | 0.0097 |
| 12622 | 0.0093 | -0.0002 | -0.0028 | 0.0097 |
| 12610 | 0.0093 | 0.0003 | -0.0028 | 0.0097 |
| 12645 | 0.0093 | -0.0001 | -0.0028 | 0.0097 |

Fig. 8.2 Stress/Displacement Value List
(showing three types of displacement values)

## 9 FEM Analysis Result (Stress Distribution Fig.)

The Stress Distribution Fig. shows the distribution of the stress components σx, σy, σz, σm, and the principal stresses S1, S2, S3. Fig.9.1 to 9.7 show the stress distribution of the pinion for this example. The distribution figures for the example gear will be displayed in a similar fashion. These figures can be enlarged, shrunk, and rotated; and the desired element and node numbers can be set to flash on the display.

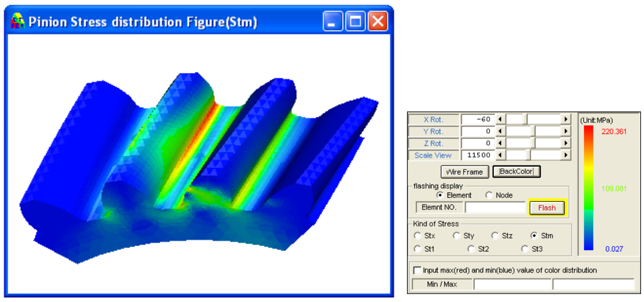



Fig. 9.1 Stress Distribution Fig. (σm)

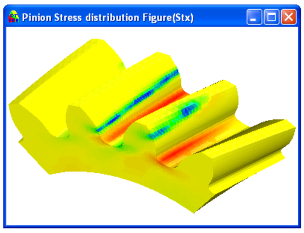



Fig. 9.2 Stress (σx)

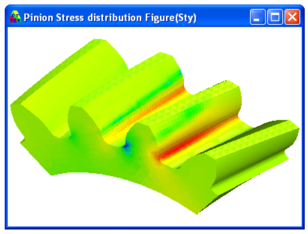



Fig. 9.3 Stress (σy)

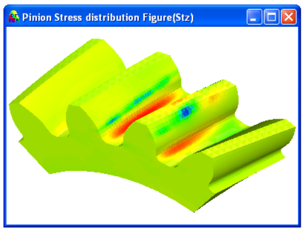



Fig. 9.4 Stress (σz)

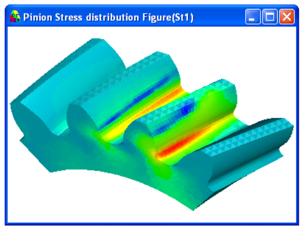



Fig. 9.5 Principal Stress (S1)

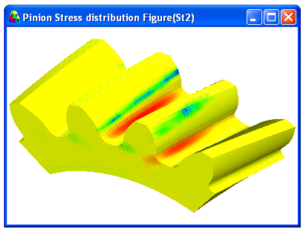



Fig. 9.6 Principal Stress (S2)

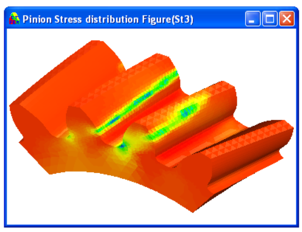



Fig. 9.7 Principal Stress (S3)

## 10 FEM Analysis Results (Displacement Distribution Fig.)

The Displacement Distribution Fig. is a colored representation of the displacement distribution in the x-, y-, and z- directions, and the overall displacement distribution. Fig. 10.1 shows a sample displacement distribution figure.

This figure can be zoomed in, zoomed out, and rotated; and the desired element and node numbers can be set to flash on the display.

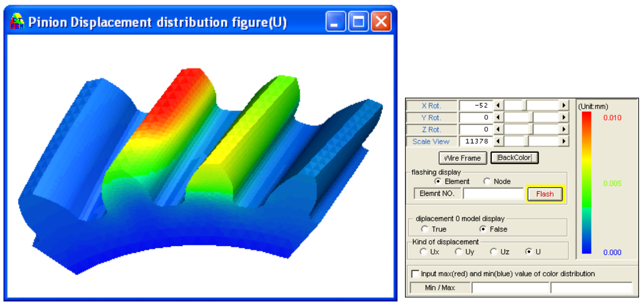
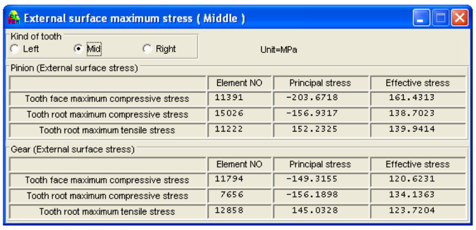



Fig. 10.1 Displacement Distribution Fig. and Control Form

## 11 FEM Analysis Result (External Surface Maximum Stress)

Fig. 11.1 shows an External Surface Maximum Stress list. The item "Tooth face maximum compress stress" indicates the stress applied to the external surface of the mesh model after FEM analysis. Check the element numbers where the items "tooth face compressive stress," "tooth root compressive stress," and "tooth root tensile stress" become the maximum. Then, set so that the points with the maximum stress flash in the Stress Distribution Fig..


External surface maximum stress ( Middle )

Kind of tooth: Left / Mid / Right — Unit=MPa

Pinion (External surface stress)

| | Element NO | Principal stress | Effective stress |
|---|---|---|---|
| Tooth face maximum compressive stress | 11391 | -203.6718 | 161.4313 |
| Tooth root maximum compressive stress | 15026 | -156.9317 | 138.7023 |
| Tooth root maximum tensile stress | 11222 | 152.2325 | 139.9414 |

Gear (External surface stress)

| | Element NO | Principal stress | Effective stress |
|---|---|---|---|
| Tooth face maximum compressive stress | 11794 | -149.3155 | 120.6231 |
| Tooth root maximum compressive stress | 7656 | -156.1898 | 134.1363 |
| Tooth root maximum tensile stress | 12858 | 145.0328 | 123.7204 |

Fig. 11.1 External Surface Maximum Stress List

## 12 FEM Analysis Result (Displacement of face graphs)

Each graph in Fig. 12.1 shows the displacement of the tooth face.

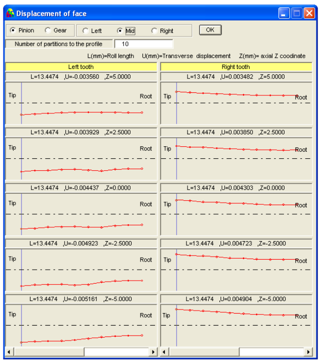


Fig. 12.1 Displacement of Face Graph

## 13. Tooth Face Stress Analysis Results (Film Element Stress Distribution Fig.)

The stress applied on the tooth faces will be displayed as shown in Fig. 13.1. By showing and hiding the tooth faces (film elements) of the pinion and gear, stress distribution on the pinion/gear can be viewed individually, or in an in-mesh state. The tooth face stress values may be checked on the Stress Value List (see Fig. 8.1) and the Film Element Stress Color Distribution List (see Fig. 14.1).

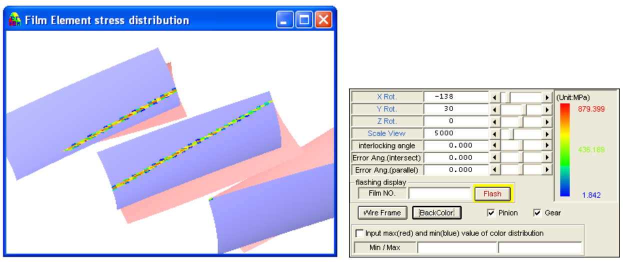


Fig. 13.1 Film Element Stress Distribution Fig.

## 14. Tooth Face Stress Analysis Result (Film Element Stress Color Distribution List)

Like the film elements shown in Fig. 13.1, the Film Element Stress Color Distribution List shows stress values across cells in 40 rows and columns (if the number of divided film elements is 41).

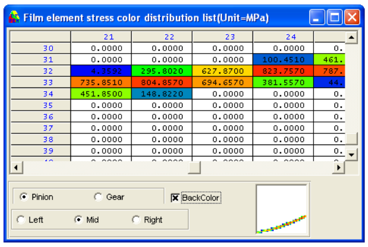


Fig. 14.1 Film Element Stress Color Distribution List

## 15. Analyze Film Stress Only - Optional

As shown in Fig. 15.1, the CT-FEM System runs an analysis sequence by first analyzing the face-to-face contact between the pinion and gear. It then performs a FEM analysis by simulating the tooth face stress distribution on each mesh model. This feature, when enabled, stops the analysis sequence as soon as the face stress calculation is completed. This leads to reduced analysis time because FEM analysis can be omitted if only the face stress analysis is required. Moreover, because whether or not FEM analysis is required can be judged based on the result of the tooth face stress analysis, if you eliminate toque unit and rotation angle setting errors at this point, the time for doing FEM analysis can be saved.

At the end of film element analysis, the Film Element Stress Distribution Fig. and Film Element Stress Value List can be displayed. Because FEM analysis is not completed, analysis results using the mesh models cannot be displayed.

To perform a FEM analysis after the completion of the tooth face stress analysis, click the [Analysis Start] button. A FEM analysis will start immediately since the tooth face stress calculation has been completed.

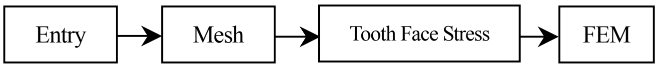


Fig. 15.1 Analysis Sequence

## 16. Continuous Calculation (Batch Processing) - Optional

When this option is enabled, analyzing multiple design data and saving the results will be completed by a single mouse click.
You can choose [Analyze FEM and Film stress] or [Analyze only Film stress]. This is a useful feature when analyzing multiple design data because it is an automated process.

## 17. Calculate Each Angle (Batch Processing) - Optional

This feature can be used to reanalyzes multiple design data by changing only the rotation angle data. Fig. 17.1 shows the window in which you enter rotation angle values.

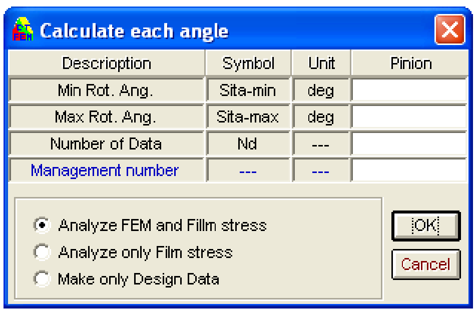


Fig. 17.1 Calculate Each Angle Window

## 18. Distribution Fig. Color Range Change - Optional

In the "Stress Distribution Fig.," "Displacement Distribution Fig.," and "Film Element Stress Distribution Fig.," the minimum and maximum value ranges are shown in blue and red, respectively. Using this feature, the values shown in blue or red can be changed as desired. Because the maximum and minimum values (that is, the values shown in red and blue) differ depending on the pinion rotation angle, it is useful when you compare multiple analysis results.

## 19. Animation Display - Optional

Using this feature, the "Stress Distribution Fig.," "Displacement Distribution Fig.," and "Film Element Stress Distribution Fig." from multiple design data can be viewed sequentially. This feature is effective only when all design data have the same gear dimensions and the same number of mesh divisions.

By running an animation of multiple analysis results including different rotation angles, you can view how the stress distribution changes with the rotation angle.

When animating Stress Distribution Figs, the stress distribution data displayed in the animation can be changed by changing the data number and change speed settings in the animation control window.

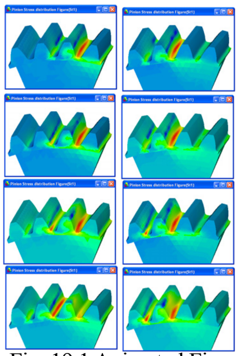


Fig. 19.1 Animated Fig

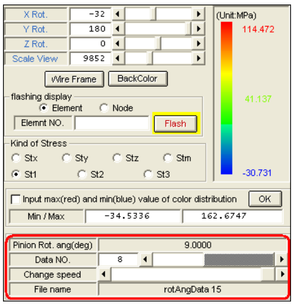


Fig. 19.2 Animation Control Window

## 20. Internal Stress - Optional

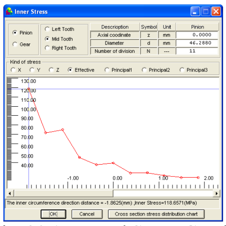


Fig. 20.1 Internal Stress Graph

This option is used to view the stress distribution within a mesh model, which cannot be viewed using a Stress Distribution Fig. that shows external stress on a mesh model. As shown in Fig. 20.1, stress values of different diameters along a point on the Z-axis (axial direction of gear) are plotted on a graph. Internal stress can also be viewed using Internal Stress Distribution Fig. shown in Fig. 20.2 and 20.3. These are cross-sectional views along a point on the Z-axis (axial direction of gear), which can be used to check the internal stress distribution and the stress value at the desired point.

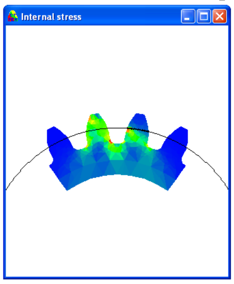


Fig. 20.2 Internal Stress Distribution Fig.

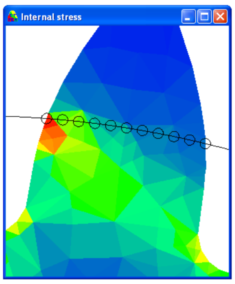


Fig. 20.3 Internal Stress Distribution Fig. (Enlarged View)

## 21. Mesh Model with Rim and Hub - Optional

Mesh models with a rim and a hub can be created. This feature is used when checking the relationship between the rim thickness and the root stress. Increasing the rim thickness will not affect the root stress greatly, once a certain rim thickness has been reached.

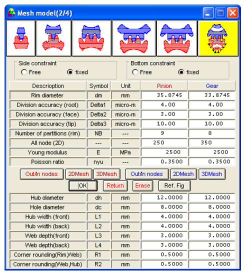


Fig. 21.1 Mesh Model Generation Window

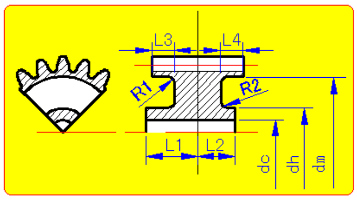



Fig. 21.2 Symbol Indications

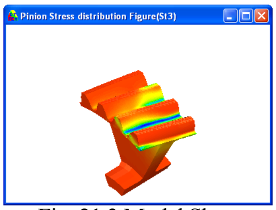


Fig. 21.3 Model Shape

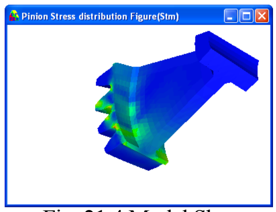


Fig. 21.4 Model Shape

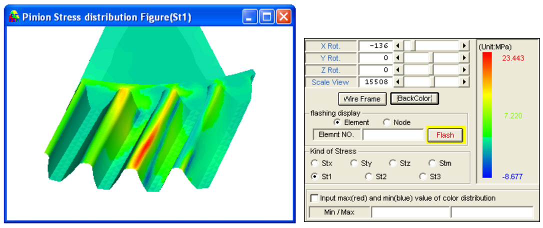


Fig. 21.5 Ex. Rim Thickness is 0.5 mm

(Tooth Root Tensile Stress σ1=23.4 MPa)

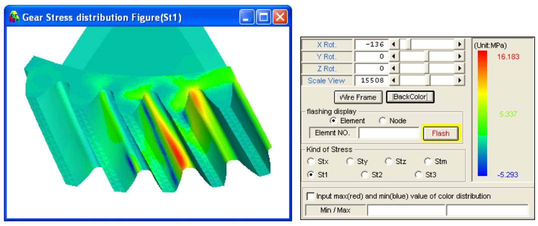


Fig. 21.6 Ex. Rim Thickness is 1.5 mm

(Tooth Root Tensile Stress σ1=16.8 MPa)

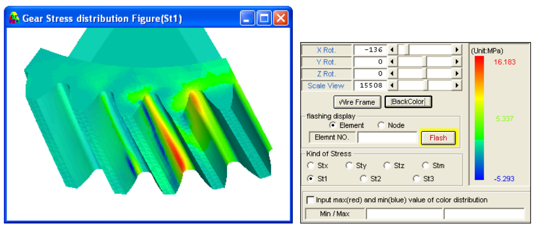
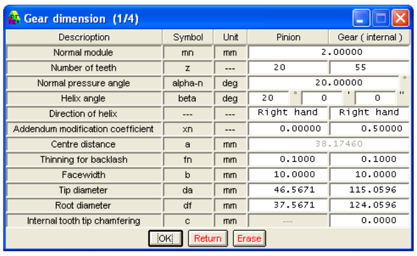


Fig. 21.7 Ex. Rim Thickness is 2.5 mm

(Tooth Root Tensile Stress σ1=16.2 MPa)

## 22. Internal Gear Calculation Example - Optional

The following shows an example of FEM analysis on an internal gear. The entry of internal gear data is similar to that of external gears. The user simply specifies gear dimensions from "Normal module" up to "Root diameter" in sequence. Example gear dimensions are shown in Fig. 22.1.



| Description | Symbol | Unit | Pinion | Gear ( internal ) |
|---|---|---|---|---|
| Normal module | mn | mm | 2.00000 | |
| Number of teeth | z | --- | 20 | 55 |
| Normal pressure angle | alpha-n | deg | 20.00000 | |
| Helix angle | beta | deg | 20 0 0 | |
| Direction of helix | --- | --- | Right hand | Right hand |
| Addendum modification coefficient | xn | --- | 0.00000 | 0.50000 |
| Centre distance | a | mm | 38.17460 | |
| Thinning for backlash | fn | mm | 0.1000 | 0.1000 |
| Facewidth | b | mm | 10.0000 | 10.0000 |
| Tip diameter | da | mm | 46.5671 | 115.0596 |
| Root diameter | df | mm | 37.5671 | 124.0596 |
| Internal tooth tip chamfering | c | mm | --- | 0.0000 |

Fig. 22.1 Gear Dimension Settings (Internal Gear)

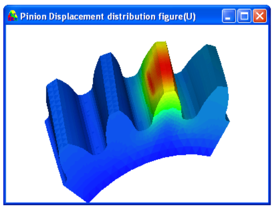


Fig. 22.8 Displacement Distribution Fig. (P)

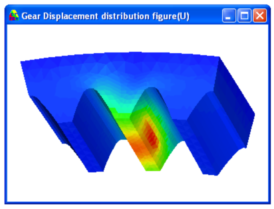
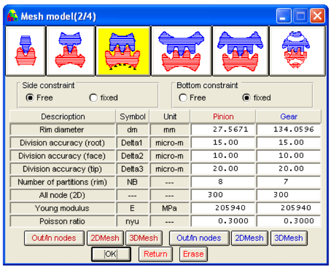


Fig. 22.9 Displacement Distribution Fig. (G)

Fig.s 22. 2 and 22.3 show the analysis conditions for this example. As shown in Fig. 22.4, the pinion will be given a crown of 20 μm in this analysis. Fig.s 22.5 to 22.9 show the results of this analysis.



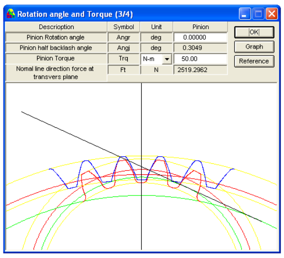


Fig. 22.2 Mesh Generation Settings Fig. 22.3 Rotation Angle Settings

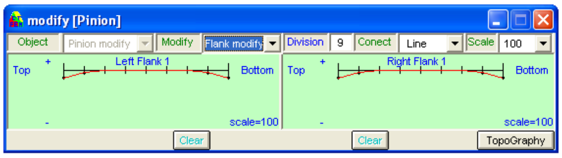


Fig. 22.4 Tooth Flank Modifications

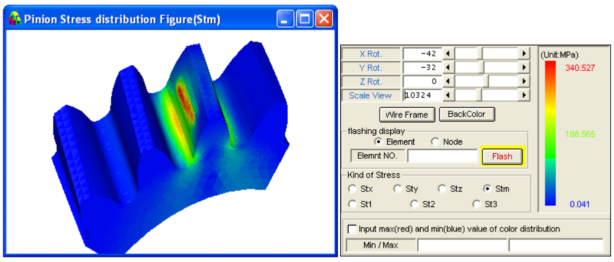


Fig. 22.5 Stress Distribution Fig. (σm)

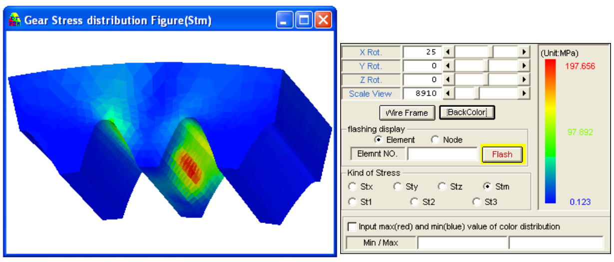


Fig. 22.6 Stress Distribution Fig. (σm)

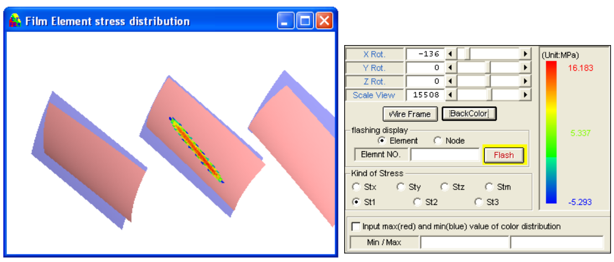


Fig. 22.7 Film Element Stress Distribution Fig.

## 23. Comparison with Strength Calculation Result (Analysis Example)

A FEM analysis was made on the example gears shown in Fig. 3.1 by giving an intersecting error angle of 0.05° and tooth flank modification shown in Fig. 23.1. The following are a comparison between the analysis results (Fig.s 23.2 to 23.5) and the gear strength calculation results (Fig.s 23.6 to 23.9).

The strength calculation was made in accordance with two standards, JGMA401-01/402-01 and ANSI/AGMA2001-C95. The calculation results are summarized in Table 23.1. The comparison results show that giving an intersecting error and tooth flank modification as analysis conditions doubles the stress compared with the strength calculation results, but does not affect greatly the root bending stress.

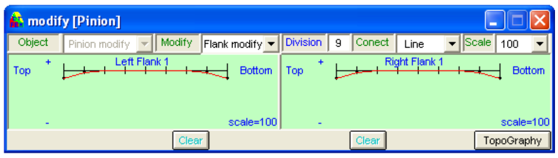


Fig. 23.1 Tooth Flank Modifications (20 μm from ends)

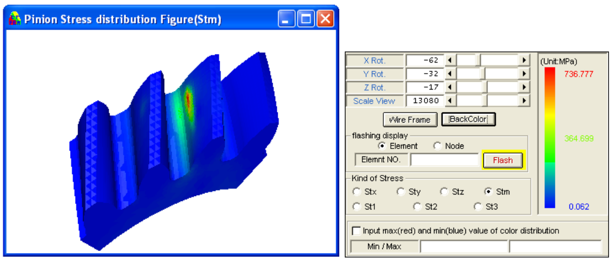


Fig. 23.2 Stress Distribution Fig. (σm)/Pinion

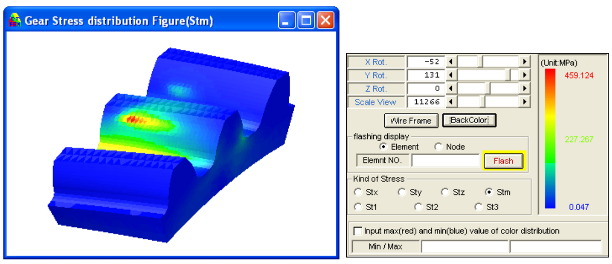


Fig. 23.3 Stress Distribution Fig. (σm)/Gear

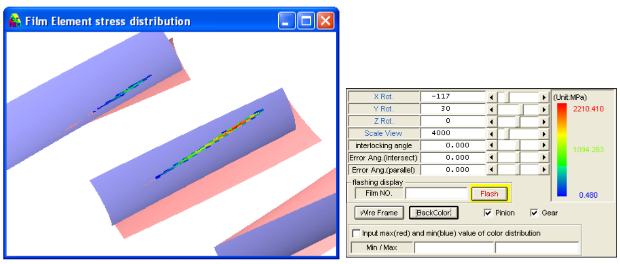


Fig. 23.4 Film Element Stress Distribution Fig.

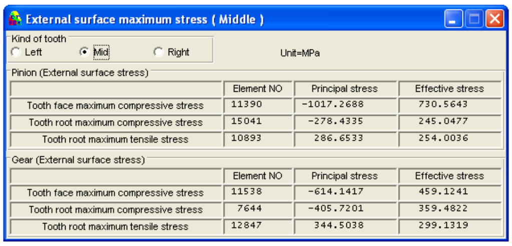


Fig. 23.5 External Surface Maximum Stress

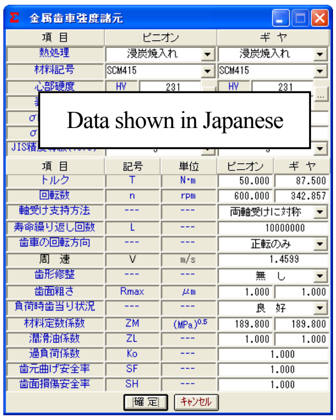


Fig. 23.6 Strength Element Settings by JGMA

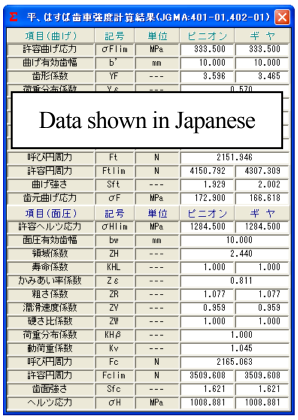


Fig. 23.7 Strength Results by JGMA

Fig. 23.8 Strength Element Setting by AGMA

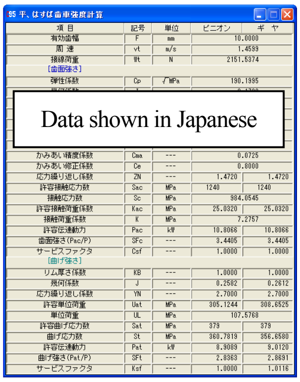


Fig. 23.9 Strength Results by AGMA

Table 23.1 Comparison between FEM Analysis and Strength Calculation Results

| Root Bending Stress (MPa) | | |
|---|---|---|
| | Pinion | Gear |
| CT-FEM | 286.7 | 344.5 |
| JGMA401-01 | 172.9 | 166.6 |
| AGMA2001-C95 | 360.8 | 356.7 |
| Tooth Face Stress (MPa) | | |
| CT-FEM (Film Element Force) | 2210.4 | |
| JGMA401-02 | 1008.9 | |
| AGMA2001-C95 | 984.1 | |

## 24. Other Features

24.1 Design Data Saving and Reading

24.2 Printing (Gear Dimensions, Mesh Division Result, Pinion -Gear Meshing Fig.)

24.3 Output of Stress/Displacement List (txt or csv)

24.4 Output of Node Coordinate/Element List (txt or csv)

## 25. Required System Configuration

(1) Operating system

Windows NT Workstation Ver 4.0, Windows 2000, Windows XP

(2) Computer

Personal computer with Pentium 1 GHz or faster capable of running Windows

(3) CD-ROM drive

(4) Windows-compatible monitor with 1024 × 768 or higher resolution

(5) 256 MB or more of available memory space

(6) 100 MB or more of available hard disk space

(7) Windows-compatible mouse or other pointing device

(8) Windows-compatible printer

## 26. Option

① Tooth Modification (Biased)

② Gear shaft Mounting Error Angle

③ Internal Gear

④ Batch Processing (Continuous Calculation, Calculate Each Angle)

⑤ Range Change

⑥ Analyze Film Stress Only

⑦ Animation Display

⑧ Pitch Error

⑨ Internal Stress

⑩ Mesh Model with Rim and Hub