# MECHANICAL FINDER

## version 13

## Extended Edition

## Standard Edition

## Bone Strength Evaluation Software

## Manual

# Contents

# Chapter 1 Introduction

As computer simulation has been widely used for analysis such as strength of structure or flow of fluid, it is indispensable in engineering. Simulation is also spreading in medical research, for example blood flow in the vessel or joint force during walking. With regard to bone, bone density has been used for evaluation of bone strength, especially osteoporosis risk. However it has been pointed out the limitation that bone density cannot remove the influence of bone spur or calcified vessel which don't contribute to strength and cannot consider bone structure. Therefore evaluation of bone strength by finite element analysis have attracted attention over the years. This method evaluates the bone strength by constructing patient-specific bone model from CT data, assigning bone density and material properties from CT value, and performing finite element analysis. Today, it is used for various analyses such as implant installation or osteotomy in research field.

Computer simulation has benefit that analysis or measurement which cannot be performed in vivo or experiment using cadaver is possible. However widely-used software has general versatility and high functionality, they make it difficult for non-engineer to learn the operation and the technique for an analysis. MECHANICAL FINDER is the finite element analysis software developed for medical doctors to analyze with no difficulty in clinical practice and a study.

MECHANICAL FINDER is the software to evaluate bone strength by considering the entire bone as a three-dimensional structure and applying finite element analysis.

## 1.1 Notations in the manual

**<u>Operation</u>**

| | |
|---|---|
| Mouse Drag | Release the button after specifying the range while holding down the mouse button. |
| Click<br>Pick | Press and release the mouse button. |
| Double Click | Click twice. |
| Pulldown | To display the menu, press the mouse button, then move the mouse to the point where the target process is described with the mouse pressed, and select.<br>In some pulldowns, an item can be selected with the mouse released after clicking the button. |
| Slider | This operation determines numerical value by moving left and right while holding down the slider with the mouse button pressed. |

**<u>Abbreviations</u>**

| | |
|---|---|
| MF | MECHANICAL FINDER |
| ROI | Region of interest (analysis region) |
| (EE) | Extended Edition |
| (SE) | Standard Edition |

## 1.2 Specifications

**Recommended Hardware Requirement**

| OS | Windows 8.1 / 10 (64bit)<br>Windows 11 |
|---|---|
| CPU | Intel® Core™ i7、Intel® Xeon® |
| Memory | Standard Edition: 8GB and over<br>Extended Edition: 16GB and over<br>*Analyzable model size depends on memory size |
| Graphics | Discrete: NVIDIA®, AMD<br>Integrated: Intel® Iris™ Graphics |
| Display | Resolution 1280x1024 and over |
| Storage | Disk space 100GB and over<br>*Particularly required for saving project<br>(Project data can be saved in external storage) |

## 1.3 Operational test

For this software, we performed operational tests using the sample Digital Imaging and Communications in Medicine (DICOM) data for the functions described in this document, and confirmed that it operates normally.

## 1.4 Version update history

- Version 13 (May 31, 2024)

  1) "ROI/Phantom"

     ✧ Drawing processes after AI bone auto-segmentation get faster. (SE, EE)

  2) "Mesh Generation"

     ✧ In fTetWild meshing, the function that specifies points to generate corresponding nodes is also enabled. The aspect ratio of the solid element can be worse by the function in fTetWild because the nearest generated node of the specified position is moved to the position. (EE)

     ✧ In "Internal Mesh", the mesher type (ANSYS ICEM CFD / fTetWild) can be selected. (EE)

     ✧ In "Mesh", the mesher type (fTetWild / MF mesher) can be selected. (SE)

     ✧ The display of the axial, coronal and sagittal direction is added to the 2D view of "Show CT around Read Geometry" in "Mesh". (EE)

  3) "Material Sorting"

     ✧ When the nodes are separated by "Separate Node" or "Contact", the nodes number of truss elements belonging to the separated material is also updated to the new nodes number. Set the truss elements again if in the project created with the older MF versions it would to be considered whether the nodes of truss elements are on the separated side. (EE)

     ✧ The nodes of truss elements can be specified by the coordinate value. (EE)

  4) "Material Property"

     ✧ "Suzuki(2020) Callus" is added to the preset formulas to convert from density to Young's modulus and Yield stress and "Minamisawa(1981) is deleted. Check "Appendix 3.1 About Material Properties of Inhomogeneous Material" about the detail of conversion formulas. (SE, EE)

     ✧ It is changed the material property of shell elements to be set by not CT value but density value. Accordingly, CT value to density value conversion function is added. Pay attention in the case that the material properties of the project with the older MF versions, which was specified by CT value, is re-set. (SE, EE)

     ✧ The truss elements tension multiplier can be set. (EE)

  5) "Analysis"

     ✧ Considering to the contact force attenuation using the friction force calculation, it can be prevented that the contact analysis convergence is worsened by friction. "Deceleration Coefficient of Friction Force" can be set from "Detailed Parameter" in "Analysis" display. (EE)

     ✧ "Solver V1" is deleted. (SE, EE)

  6) "Display" and "Graph"

     ✧ In "Load View", display / undisplay of load condition can be set as groups. (SE, EE)

     ✧ The process of "Data Processing and Visualize" gets faster. (EE)

     ✧ Truss tension display is added in "Data Processing and Visualize". It can be colored and displayed according to tension value. (EE)

     ✧ In "Failure View", the color of plastic and crushed elements can be arbitrary changed. (SE, EE)

     ✧ In "Material/STL View", the model after analysis can be output as a STL file. (SE, EE)

     ✧ In "Graph" screen, when selecting Displacement, Load, Contact Force or Constraint Force as Y axis, the graph can be generated by selecting any of X/Y/Z-component in addition to the conventional value: "Magnitude". (SE, EE)

  7) "Tool"

- ✧ In "Inhomogeneous Material Editor", parameters of conversion formulas from density to physical property values can be defined. Defined formulas can be selected from "User-established" of bone material in Material Property display. (SE, EE)

- ✧ In "Remote Batch Program (Analysis)", the analysis process can be executed while switching between multiple server connections. (SE, EE)

8) Others

- ✧ By inputting the coordinate values, the axis can be set in "2-point Pick" of "Setting of Axis" in "Mesh", "Material Sorting" and "Boundary Condition" display. (SE, EE)

- ✧ In "Mesh" and "Boundary Condition" display, the center of rotation can be changed. (SE, EE)

- ✧ In Material Property of "Analysis Setting" display, the information of contact elements and truss elements is included. (SE, EE)

- ✧ Fixed UI partly. (SE, EE)

- ✧ In addition, fixed minor bugs. (SE, EE)


- Version 12 (August 31, 2022)

1) "ROI / Phantom"

- ✧ "Effect ROI" is deleted. (SE, EE)

- ✧ AI bone auto-segmentation function is available. (SE, EE)

2) "Mesh Generation"

- ✧ In mesh generation, "fTetWild" mounted on MF or "ANSYS ICEM CFD" as an external software can be selected. If using "ANSYS ICEM CFD", ANSYS products is necessary to be installed. "ANSYS ICEM CFD" used in the older version of MF can continue to be used. (EE)

3) "Display" and "Graph"

- ✧ In "Result", the function that "Contact Pressure, Contact Area" and "Relative Displacement" can be calculated, output, and displayed as contour is added. It is used in the analysis with contact setting. (EE)

- ✧ Slice information set in slice contour function can be saved to a file, and it can be read. Using this file, the same slice information can be used in slice contour function of other displays. (SE, EE)

4) Others

- ✧ "Master" and "Slave" in contact setting are modified to "Primary" and "Secondary". In addition, some UI is modified. (SE, EE)

- ✧ In DICOM setting, "Coord. System" option is added. With "DICOM Coord. System" option, the zero position of the DICOM data is set to the origin in MF. In this case, the coordinate system is different to that made in the older version of MF. Set "Old Ver. Compatible" if the same system as the older one would like to be used. (SE, EE)

- ✧ The main menu is wider than that of version 11.0. Click "MF" in the title bar and drag, when the position of main menu would like to be moved. (SE, EE)

- ✧ Window 11 is supported. (SE, EE)

- ✧ In addition, some minor problems are corrected. (SE, EE)


- Version 11.0 (May 7, 2020)

1) "ROI / Phantom"

- ✧ Keyboard shortcuts are available in 2D ROI and 3D ROI screen. The key binds are listed in "5.3.8 Keyboard shortcut". (SE, EE)

2) "Mesh Generation"

✧ In import setting of "Surface Mesh and Import" screen, the initial coordinate of import geometry is memorized. Even after releasing the import geometry from Import Info. list and re-add to the list, the geometry can be set as the inhomogeneous material in "Reference CT" of Material Property screen. (EE)

✧ "Auto Size" is deleted from "Settings of Internal Mesh Size" in "Setting of Internal Mesh". (EE)

3) "Material Sorting"

✧ The processing of "Separate Node" is improved in Special Material setting. (EE)

4) "Material Property"

✧ Truss elements can be defined and set as a group. (EE)

✧ Friction coefficient can be set over 1. (EE)

5) "Boundary Condition"

✧ Load condition can be defined until 9999. In addition, each condition can be named. (SE, EE)

✧ If slave nodes of the contact element are set as constraint nodes, when pushing the "Accept" button, the warning message is displayed. (EE)

✧ In forced displacement to nodes, degree of freedom by X, Y and Z direction can be set. This enables the object to move freely in a part of direction. (SE, EE)

6) "Analysis"

✧ Fixed: when the outsteps interval was over 1 and the solver V2 analysis finished in 1 step, the analysis result is not output. (SE, EE)

7) "Display" and "Graph"

✧ The calculation method of "Average" processing in Extraction of Result is changed to volume weighted average. The previous method is "Ave. (Arithmetic)" processing. (SE, EE)

✧ In linear analysis, the upper limit of display of tensile and compressive stress strength ratio in contour display and data extraction is removed, previously 100%. Thus, the load value in the case without deformation is easy to be considered. (SE, EE)

✧ The function to save and read a slice setting is added in the following screen: Contour Display, Material Contour Display, Extraction of Result, Extraction of Material Property and Vector Display. (SE, EE)

✧ Fixed: unit conversion is invalid in the data extraction function of "Graph" screen. (SE, EE)

8) Others

✧ The confirmed dialog is displayed when the "Cancel" button is pushed in the following screen: "CT Range", "ROI / Phantom", "Mesh Generation", "Material Sorting", "Material Property", "Boundary Condition" and "Analysis". (SE, EE)

✧ UI is updated. (SE, EE)

# Chapter 2 Overview

This SOFTWARE performs 3D bone strength analysis from multiple DICOM data. The process is described below.
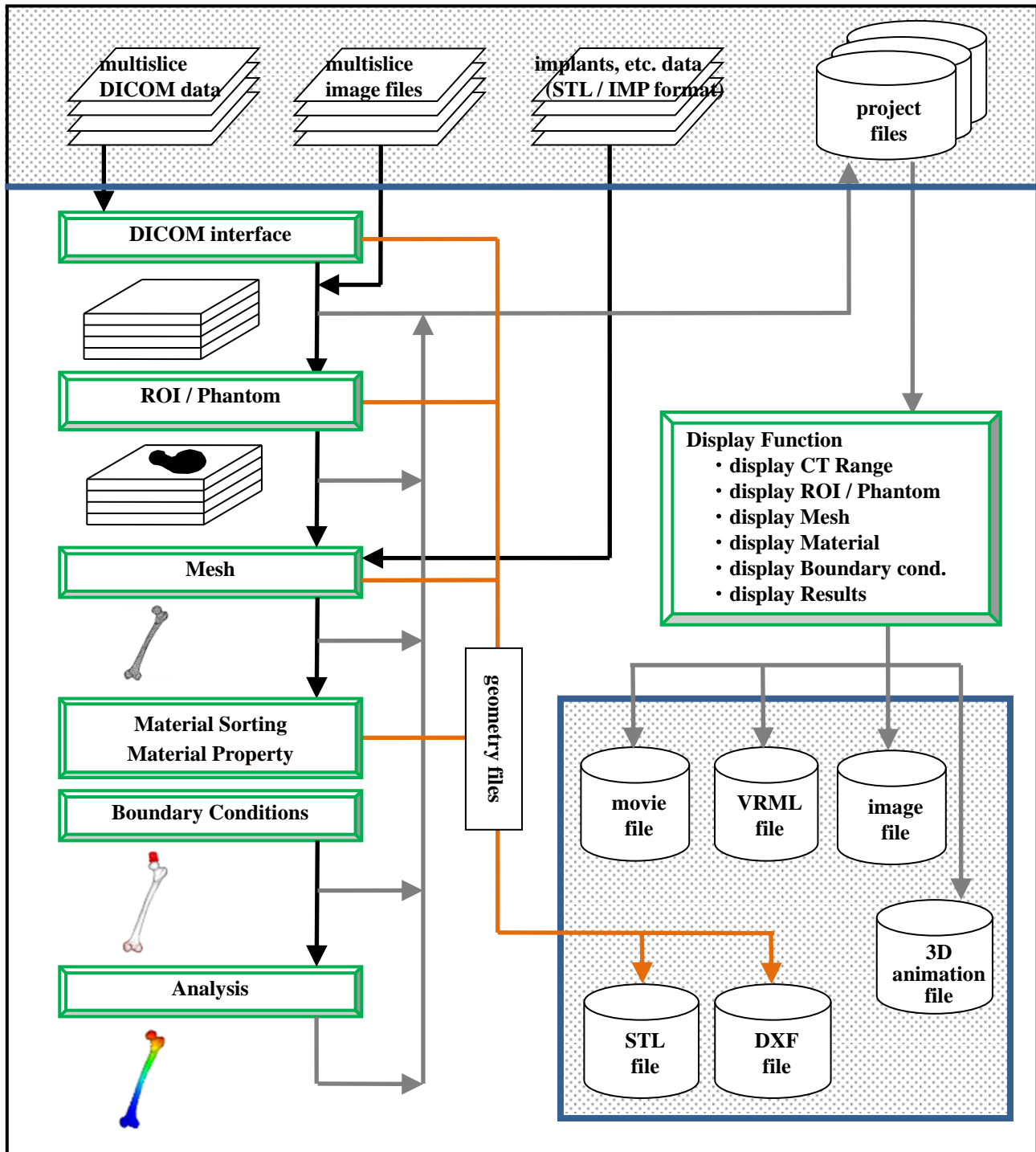
## 2.1 Overview of each function

### (1) Organization of functions

In this software, the followings can be operated by operations from the "Main Menu."

- "DICOM interface"
- "CT Range"
- "ROI / Phantom"
- "Mesh"
- "Material Sorting"
- "Material Property"
- "Boundary Condition"
- "Analysis"
- "Output"
- "Display"
- "Graph"

### (2) Main Menu

This is the main screen of MECHANICAL FINDER for processing [File], [Option] menu items and opening "CT range," "ROI extraction," "mesh generation,"…

### (3) DICOM interface

By selecting [File] → [New Project], you can convert DICOM data consisting of multiple images taken by the X-ray CT imaging device into a file of this software format. You can also convert image files (BMP/JPEG) to project files for this software by using the "Construct Project by Image" of the tool group.

### (4) CT Range

By limiting the effective range of the CT volume, the operation load of the subsequent process can be reduced.

### (5) ROI / Phantom

From the CT original image data, extract the part to be analyzed (bone). The extracted information is saved as binary data, expressed by the part to be analyzed as "true," and the part not subject to analysis as "false." By setting the phantom, calibration of density value can be performed at the material property setting.

### (6) Mesh

Three-dimensional meshes are automatically generated for the implanted shape (EE) and the CT data extracted as the ROI.

### (7) Material Sorting

The software is designed so that analysis material settings/axis setting/analysis coordinates can be set up.

### (8) Material Property

Set material properties such as Young's modulus and yield stress value.

### (9) Boundary Condition

Set the load and constraint conditions.

### (10) Analysis

Analysis is performed under the set conditions and the result is displayed.

### (11) Output

The outer shape (shell element), the part of the shape generated as a mesh, can be output as a file in STL format or DXF format. Output function is prepared at three stages: "Output ROI Surface," "Output Surface (before internal Meshing)," and "Output Analysis Surface."

**(12) Display**

This software has a display function suitable for each processing stage. It is also possible to save the display screen as an image file or movie file. It also has an animation display and VRML output.

**(13) Graph**

Graphs can be drawn using material properties and analysis results.

## 2.2 "Project"

**(1) Concept of "project"**

In this software, one analysis case is called a "project." Even when using the same CT image, if any setting value of "CT Range"/"ROI · Phantom"/"Mesh"/"Material Sorting"/"Material Property"/"Boundary Condition"/"Analysis" is different, it should be saved as a different project.

**(2) Saving**

• Data can be saved at the points shown below. It is also possible to resume processing from the point.

• Please pay attention to the saving method when reading the analysis carried out in the past and setting another condition and analyzing it.
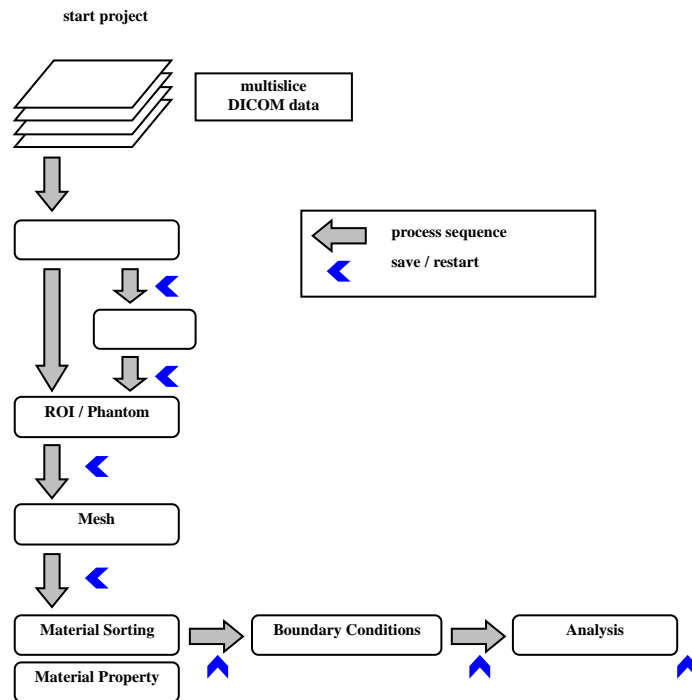


**(3) Project reprocessing**

• If you change the conditions of the working project, the settings in the steps after the changed point will be discarded. For example, if you change "ROI / Phantom" for a project that has been completed up to "Mesh," "Mesh" must be done again after changing "ROI / Phantom."

• If the dialog box shown below is displayed, the data file of the current project is discarded.

• In a new project (if the project has not been saved), the steps after the reprocessing point are discarded.

• In the case of existing project (if the project is saved), the project is copied to the temporary working directory. Therefore, in the project in that working directory, the steps after the reprocessing point are discarded, but the original project is never destroyed.



11
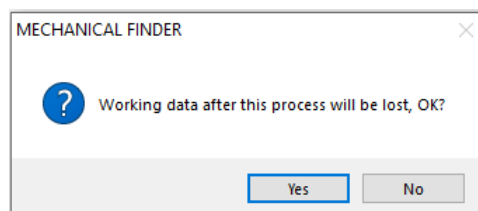
## 2.3 How to launch

| What to launch | How to launch |
|---|---|
| MECHANICAL FINDER | Select the following items in the Start Menu.<br>[Start]→<br>[Program]→<br>[MECHANICAL FINDER V13 EE]→<br>[MECHANICAL FINDER V13 EE] |
| Help | Select the following items in the Start Menu.<br>[Start]→<br>[Program]→<br>[MECHANICAL FINDER V13 EE]→<br>[Help] |
| Tools | Select the following items in the Start Menu.<br>[Start]→<br>[Program]→<br>Select the tool to use from [MECHANICAL FINDER V13 EE] |

## 2.4 Settings after initial launch

Since the setting is the initial value at the first startup after installation, it is necessary to change to a setting suitable for the user.
Select "Options" → "Default Settings" → "Application" from the menu and set it to suit the user.

# Chapter 3 Main Menu

This chapter explains how to use the main menu for each item. All operations performed in this software will start from the main screen. This section explains the operations that can be done on the main screen.



| Pulldown menu | Read and save files, and various settings are performed here. |
|---|---|
| Process buttons | Perform each process according to the procedure up to analysis. The process buttons vary in size depending on their functions.<br>• Large buttons: Perform parameter settings (actual work).<br>• Small buttons: Confirm display and output shapes of actual work. |
| Progress bar | Displays processing progress. |

## 3.1 Pulldown menu

The structure of the [File] item in the pull-down menu is as follows.
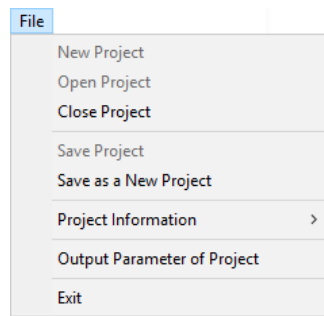


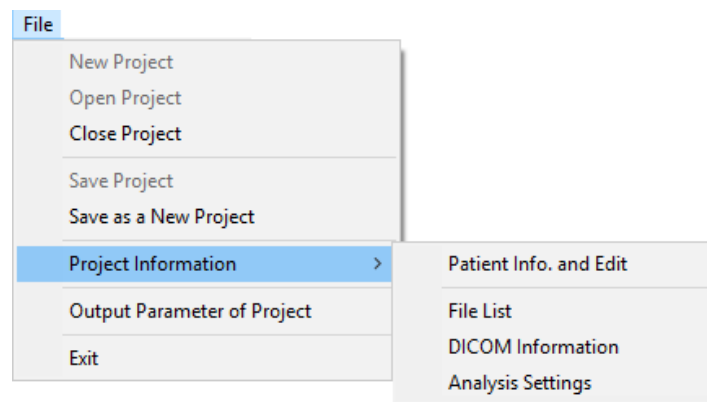| Menu items | Function |
|---|---|
| New Project | Read the DICOM files and create a new project. |
| Open Project | Open an existing project. |
| Close Project | Close the working project. |
| Save Project | Overwrite and save the project. |
| Save as a New Project | Save the project as another project. |
| Project Information | Display a variety of information about the working project. |
| Output Parameter of Project | Output the project setting value to a text file. |
| Exit | Close software session. |



| Menu items | Function |
|---|---|
| Patient Information and Edit | Display and change the patient information of DICOM data. |
| File List | Display the file list of the working project. |
| DICOM Information | Display the input information of the working project. |
| Analysis Settings | Display the information set by the user in the working project. |

15

3.2 Process Buttons

- Below the menu bar of the main screen, there are process buttons such as "CT Range," "ROI /Phantom," "Mesh," "Material Sorting," "Material Property," "Boundary Condition," and "Analysis."
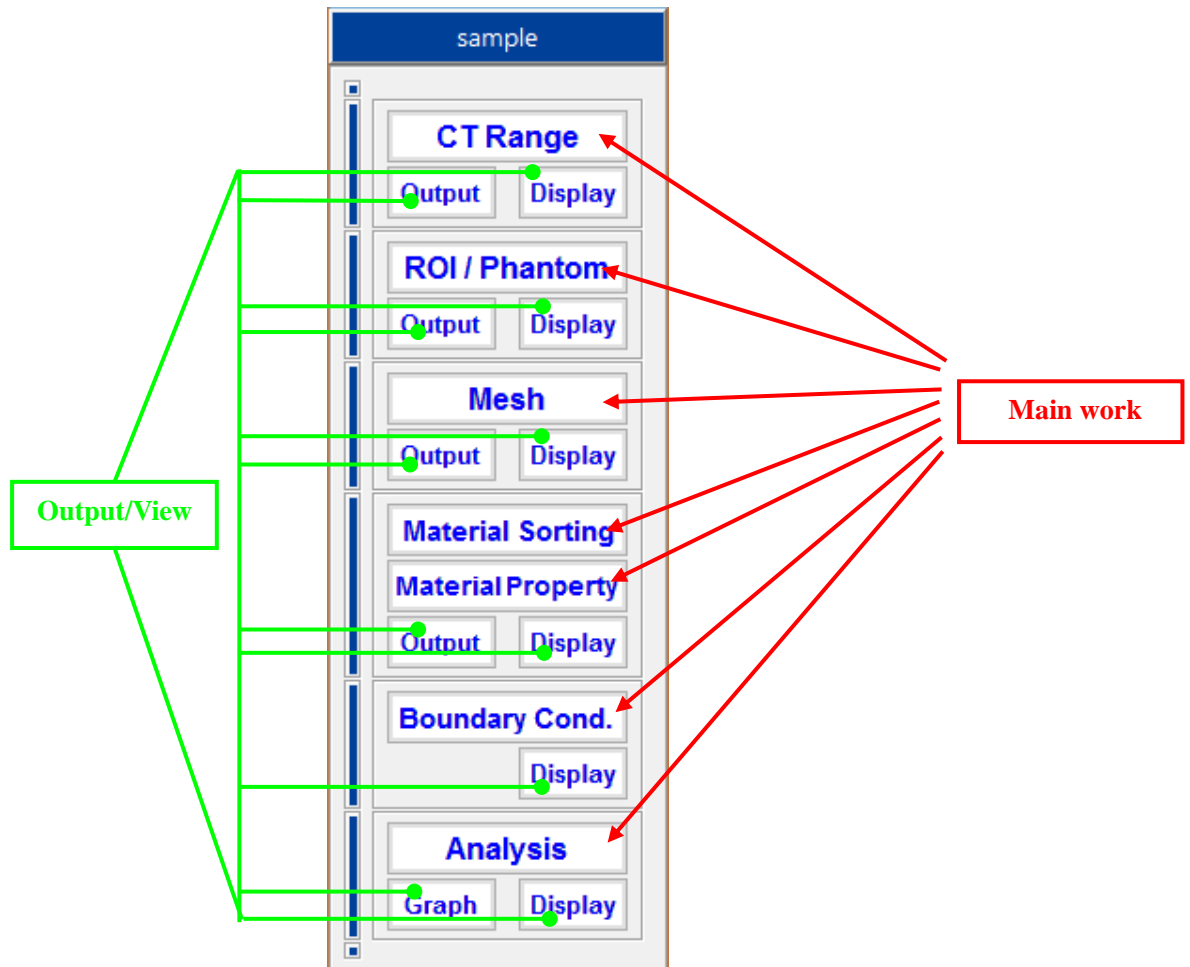


- The processing procedure is in order from top to bottom. To start each process, click the corresponding button. The actual work for analysis is done with the big buttons of "CT range" and "ROI · phantom," "Display," "Output," and "Graph" under each button are for result display, output function, and confirmation.

- The line to the left of the button indicates the progress of processing.

- The unprocessed state is represented by a black line, the processed state is indicated by a blue line, and the state being processed is indicated by a red line. You cannot click the processing button in the unprocessed state.

- In the top "project," the name of the working project is displayed. When it is a new project, it will be "project: None."

## 3.3 Progress Bar
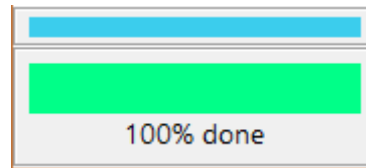


There are two kinds of progress bars.

The upper progress bar (light blue part) shows the process progress of the internal process and the lower progress bar (green part) shows the process progress of the external process.
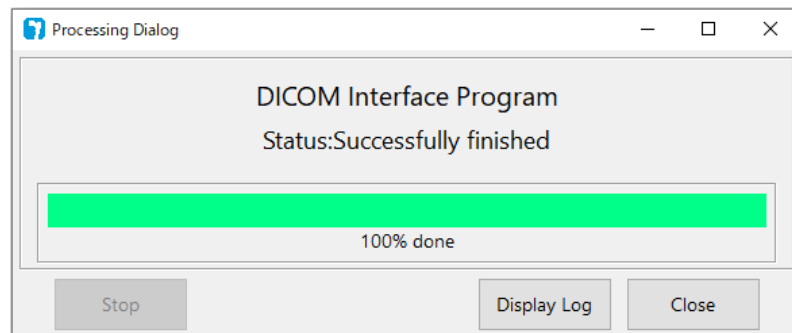
The external process bar is activated with following programs and works in conjunction with the dialog as shown below.

- DICOM interface program
- internal meshing program
- analysis processing program

# Chapter 4 CT Range

- In "CT range," you can specify the volume range (analysis target range) to be used for shape extraction processing in the DICOM data of the loaded series.
- For example, as shown in the figure below, the CT images with the left and right femurs is reduced to only the range of the left femur.
- By excluding out-of-range volumes, the response of each process improves and operational procedures can be reduced.
- In addition, the CT slice is displayed three-dimensionally, which is effective for checking DICOM data.
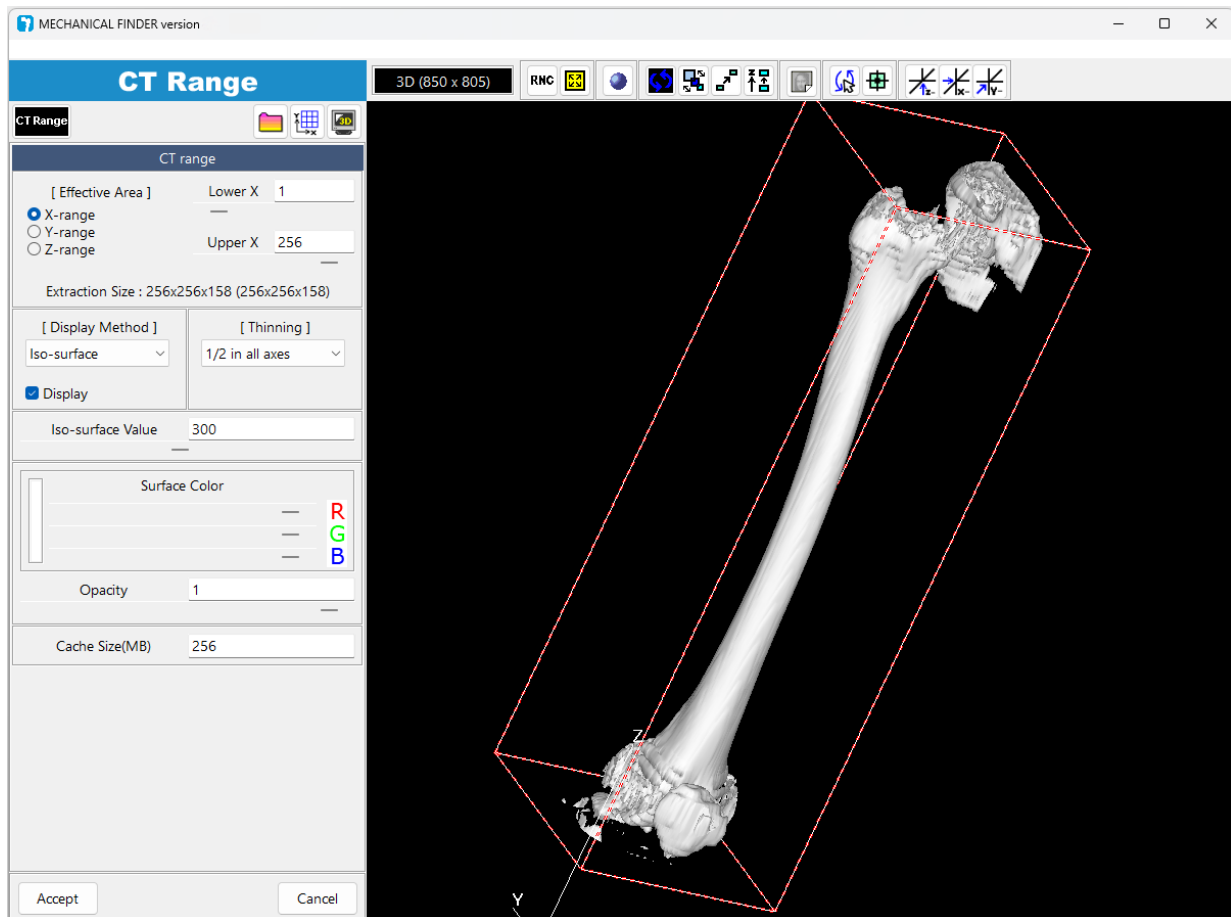


| Icon | Function |
| --- | --- |
| CT Range | CT Range |
| | Data Information |
| | Surface Display |
| | Viewer Settings |

# Chapter 5 ROI extraction/Phantom settings

- In "ROI extraction" (2D ROI Processing/3D ROI Processing/AI ROI Processing), the purpose is to extract the part to be analyzed (bone part) as binarized data from the data of the CT original image.
- Binarized data is data expressed by a part to be analyzed (true) or a part excluded from analysis (false).
- Since the "ROI extraction" function has a slightly complicated configuration, we recommend that you first read "5.1 2D ROI Processing/3D ROI Processing/AI ROI Processing."
- In the extended edition, you can set multiple ROI groups.
- In the case of a CT original image taken together with a bone density calibration phantom or by using "phantom function," it becomes possible to perform accurate calibration of bone density values.



| Icons | Functions | |
|---|---|---|
| **2D ROI** | 2D ROI Processing | 【 common work 】 see 5.1 for detail |
| **3D ROI** | 3D ROI Processing | |
| **AI ROI** | AI ROI Processing | |
| **Phantom** (Option) | Phantom Settings | |
| | Data Information | |
| | Viewer Settings | |

## 5.1 2D ROI Processing/3D ROI Processing/AI ROI Processing
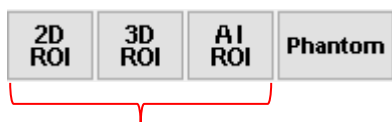


The three functions (2D ROI Processing/3D ROI Processing/AI ROI Processing) in the above figure are available to extract ROI.

Since these three functions share "CT original image" and "ROI data," any "ROI data" changed by one function is immediately reflected in the other two functions.



Each function has the following characteristics.

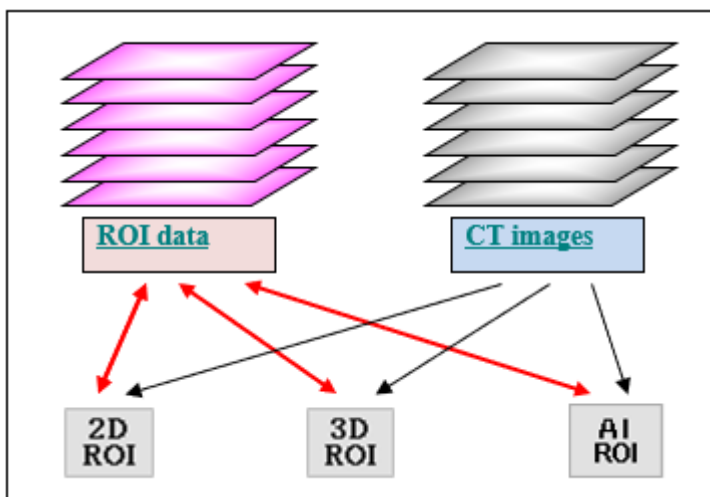| Functions | Characteristics |
|---|---|
| 2D ROI Processing | This is the basic function of ROI extraction. It displays 2-dimensional images in the axial/coronal/sagittal plane and extracts the bone. It is suitable for fine detail correction. |
| 3D ROI Processing | This confirms or executes the extraction or deletion of the part as one block in the 3-dimensional image. For example, when there are left and right parts, only one part can be extracted or deleted at once. A part means a connected chunk. There are also modes that can perform cutting and drilling, using tools. |
| AI ROI Processing | This aims to support to extract the shape of bone using deep learning.<br>It is possible to automatically extract the bone just with the pre-trained data, and currently supports "femur", "femur + pelvis", and "spine".<br>**Corresponding GPU made by NVIDIA** can do that more rapidly than CPU. The corresponding status of GPU will be updated, so don't hesitate to contact MF technical support if you have any questions.<br>**After the auto extraction, check the result in [2D ROI] and [3D ROI] definitely.**<br>About the used deep learning model, refer 'Q3. About the detail of "AI ROI" function' of "Appendix 5.2 About ROI/Phantom". |

The ROI extraction task is to extract the bone part to be analyzed from the data of the CT original image. In order to extract this bone part, it is most efficient to perform the following procedure.

I. For the entire area of the original CT image (all slices), apply the threshold parameter designation that can uniformly extract the bones to be analyzed.

II. If there is a part to be excluded from analysis, either delete the part not being analyzed, or extract the part to be analyzed.

III. In 2D ROI Processing, perform bone modification on all slices as needed, such as using mouse operations on the CT original image.

The usage of each function when performing the above procedure is described below.

| Process | 2D ROI Processing | 3D ROI Processing | AI ROI Processing |
|---------|-------------------|-------------------|-------------------|
| I | Initialization of all slices of the CT original image by specifying threshold. Initialization of all slices by threshold designation using 2D image processing. | | Execute processes I and II by the auto extraction using deep learning. |
| II | | Extract/delete parts on 3D display. | |
| III | Make fine modifications of the region of interest by operating the mouse on the CT original image. | | |

Any of the applicable functions (two-dimensional ROI processing, three-dimensional ROI processing, AI ROI processing) can be used in the above processes I, II, and III. Please select and use the function appropriate to the imaging condition, analysis target, and user's preference.

## 5.1.1 ROI

**(1) General method**

- The ROI extraction (2D ROI Processing/3D ROI Processing/AI ROI Processing) task is to extract the part to be analyzed (bone part) from the data of the CT original image.
- The bone part to be extracted at this time refers to the entire bone including [cortical bone + cancellous bone] in this software.
- Generally, in normal work, we extract the ROI area as shown on the left in the figure below.
- However, if you want to analyze across multiple bones, such as the vertebrae on the right of the figure below, it is necessary to extract discs between the bones in another ROI number or import some shape and connect the bones when generating the mesh.
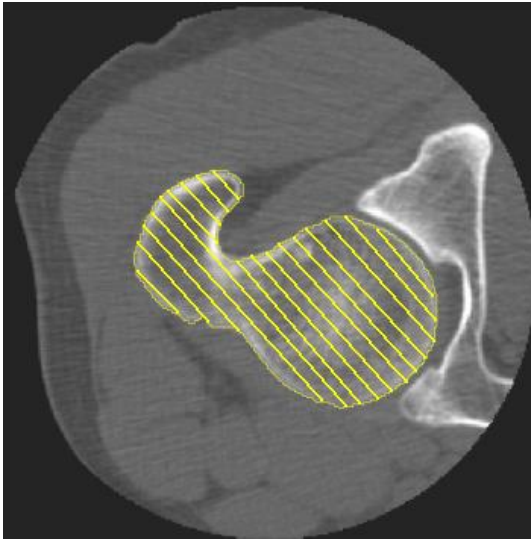


Analysis model with only femur.
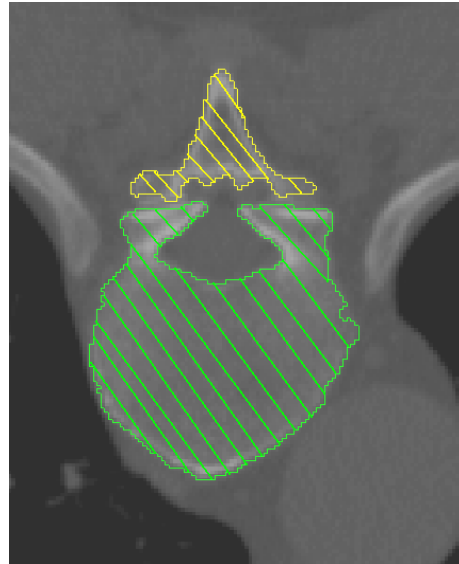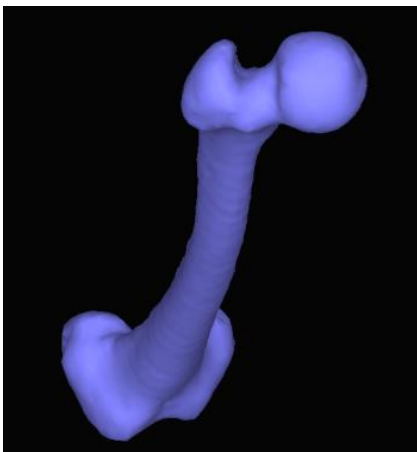


Analysis model including multiple vertebrae. (EE)
- ROI numbers are set separately for adjacent bones.
- The intervertebral disks also are in another ROI number.





In the next section, we show a slightly different method of ROI extraction.

**(2) Less general method**

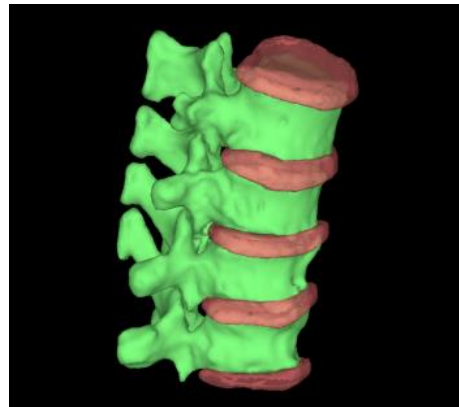- Generally, in this software, the bone part to be extracted as ROI refers to the entire bone including [cortical bone + cancellous bone], but it may be sorted to different ROI numbers depending on bone quality or other similar features.
- In the left figure below, as with the general method on the previous section, the lower jaw, including the teeth, is specified as the same ROI number. In this case, the mesh analysis is represented as one mesh of the entire lower jaw, including the teeth.
- On the other hand, alternatively, the right figure below shows a method in which the lower jaw and teeth are specified in different ROI number. In this case, mesh for analysis and material properties can be separately set for mandible and teeth. When making such designation, pay attention to setting of duplicate parts between ROI numbers. (EE)
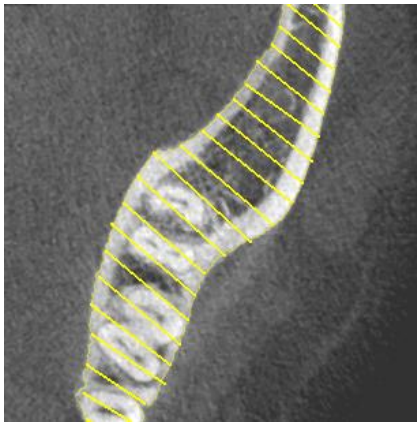


Designate lower jaw in one ROI number.
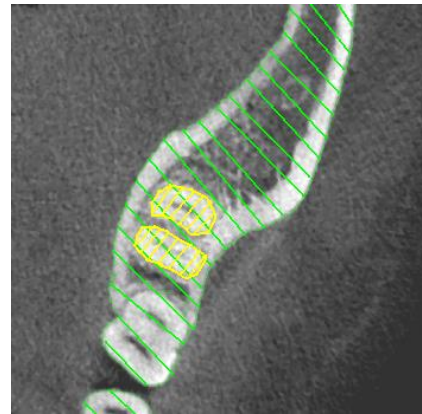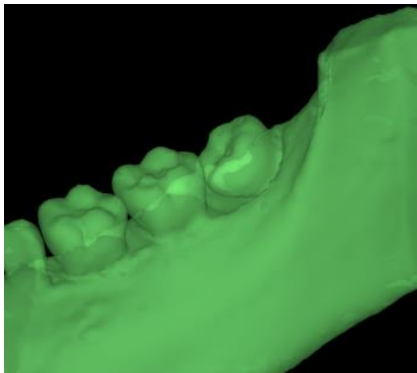


Designate lower jaw in multiple ROI numbers. (EE)
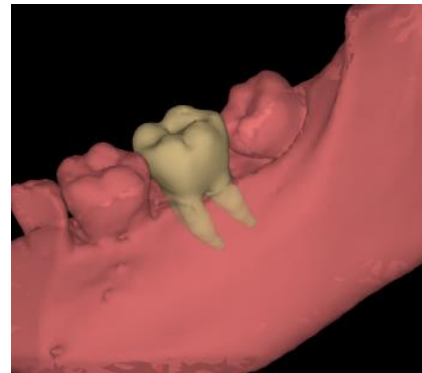


View in sagittal plane

ROI specification method
Two ROI numbers are designated in duplicate at the tooth embedding part (red part).

## 5.2 Phantom Settings

When all of the following conditions are satisfied, material properties are set more accurately by applying a phantom setting.
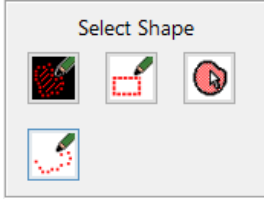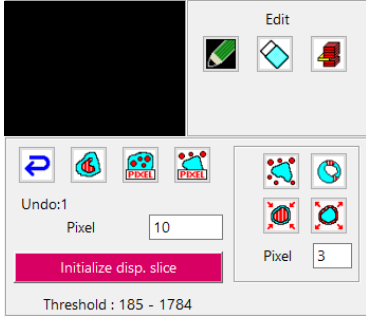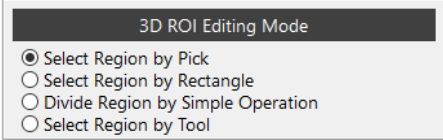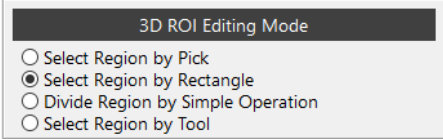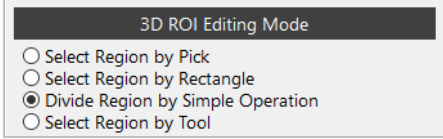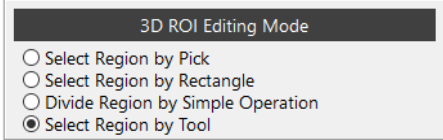
(1) CT imaging is scanned with a bone density calibration phantom.

(2) Each ROD density value (mg/cm$^3$) of the phantom is known.

(3) You are going to set material properties as inhomogeneous material.

*Caution

In the Phantom Setting, the phantom is not applied by default.

## 5.3 ROI Extraction Operation

This section provides references of the actual operation example corresponding to each icon function in ROI work in order.

| Processing Type | Operation Screen | Link to Detail |
|---|---|---|
| 2D ROI Processing |  | 5.3.1 2D ROI operation example 1 (shape specification) |
| 2D ROI Processing |  | 5.3.2 2D ROI operation example 2 (edit and special operation) |
| 3D ROI Processing |  | 5.3.3 3D ROI operation example (example of deletion by specifying pick) |
| 3D ROI Processing |  | 5.3.4 3D ROI operation example (example of rectangle specification) |
| 3D ROI Processing |  | 5.3.5 3D ROI operation example (example of Divide Region by Simple Operation) |
| 3D ROI Processing |  | 5.3.6 3D ROI operation example (example of deletion by tool) |

| AI ROI Processing |  | |

5.3.1 2D ROI operation example 1 (shape specification)



Indicates the operation of the  buttons at the time of ROI extraction.

In the following example, it is assumed that (  ) is selected in [  ].

**(1) Arbitrary Shape mode**

| |
|---|
| Press the button to enter the Arbitrary Shape mode. |
| <br>Specify the necessary part by right mouse drag as above. |
| <br>Then the interior of the shape is added. |
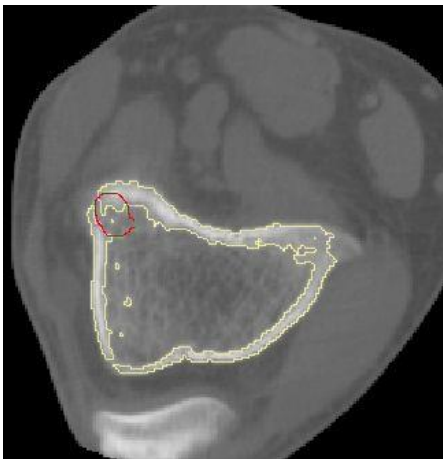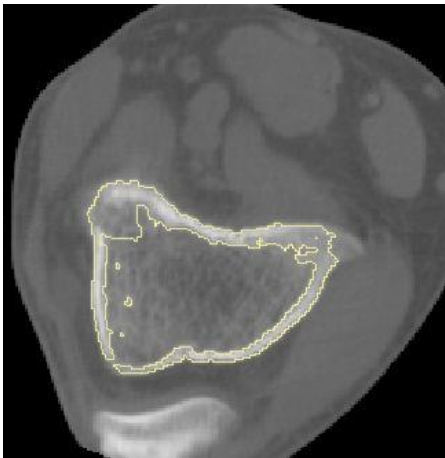
## (2) Rectangle mode

Press the button to enter the Rectangle mode.
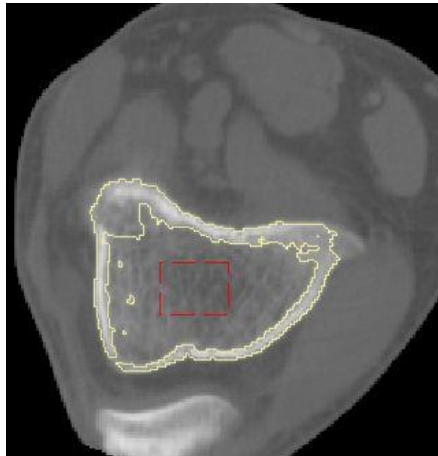


Specify the necessary part by right mouse drag as above.
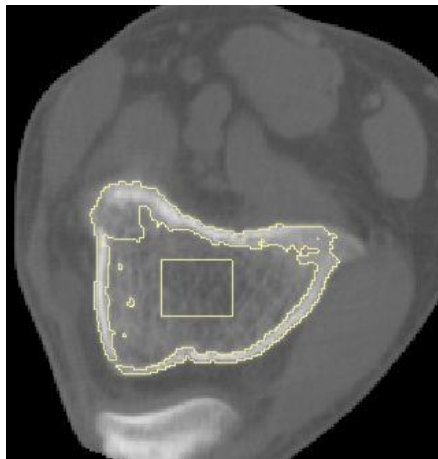


Then the interior of the shape is added.

## (3) Contour Segment mode

Press the button to enter the Contour Segment mode.



Specify the indicated shape (here, closed non-ROI area) by right pick.
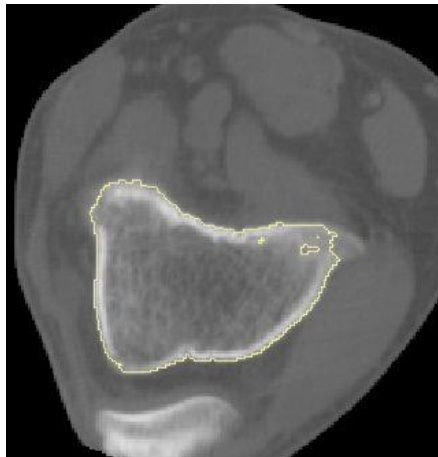
Then the inside of the closed non-ROI area is added.

## (4) Arbitrary Line mode

 Press the button to enter the Arbitrary Line mode.
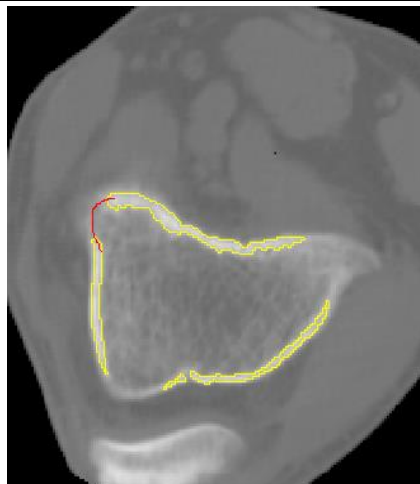


As above, draw a line by right-dragging with the mouse.
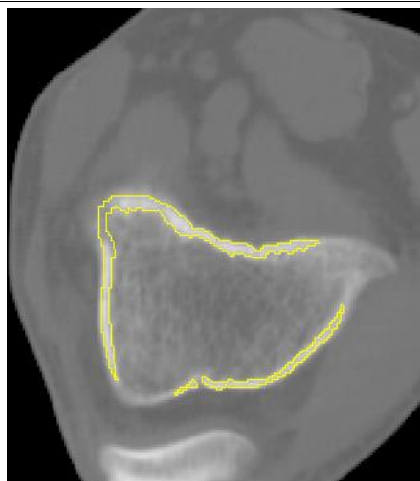


Then the place where the line was drawn is added to the ROI.

If you want to change the line width, change it with the  on the right of the button.

5.3.2 2D ROI operation example 2 (edit and special operation)
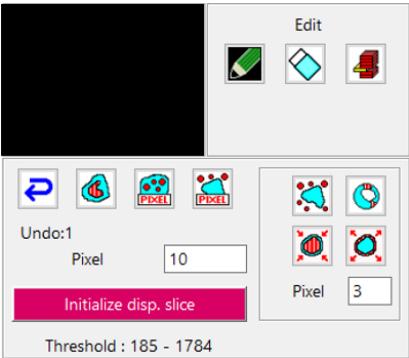


Below is a description of the operations of buttons in [Edit] [Special] item when extracting ROI.

In the following example, it is assumed that [CT Image ] is selected in [Select Data]; [Arbitrary Shape ] is selected in [Select Shape].

## (1) Binarization by specified threshold value
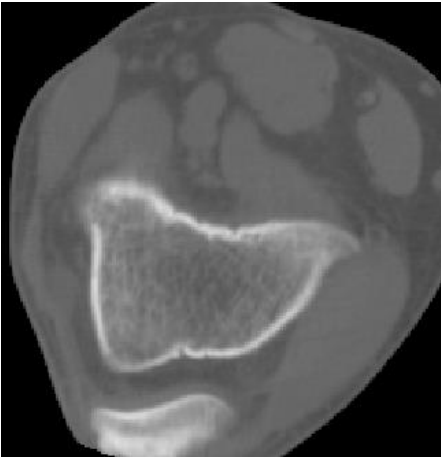


When you enter the threshold and initialize the original image as above, the ROI is extracted as shown below.
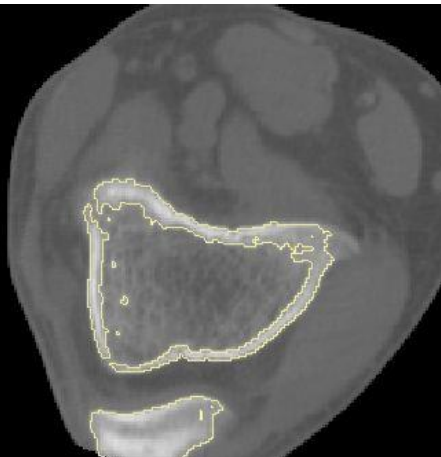


Please refer to the histogram displayed by pushing the [Graph Drawing] button and set the threshold value.

**(2) Delete mode**

Press the button to enter Delete mode.
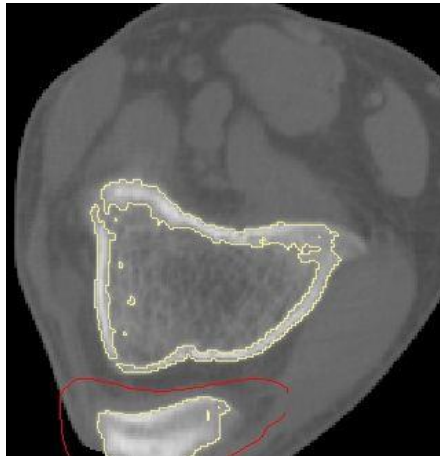


As shown above, surround the unnecessary part with a right mouse drag.
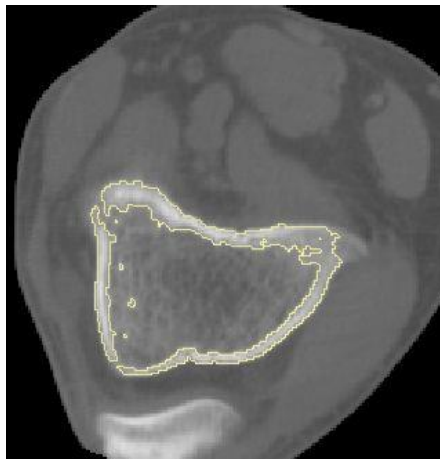


The enclosed part is deleted from the ROI.

**(3) Add mode**

Press the button to enter Add mode.
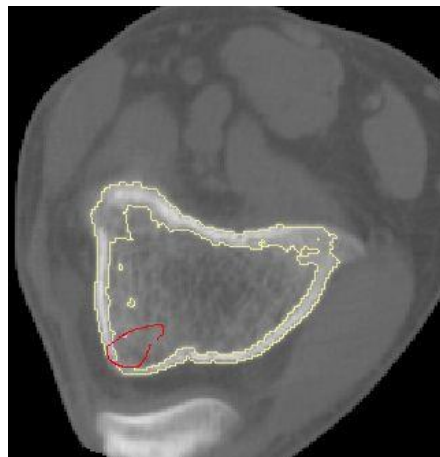


As shown above, surround the necessary part with the right mouse drag.
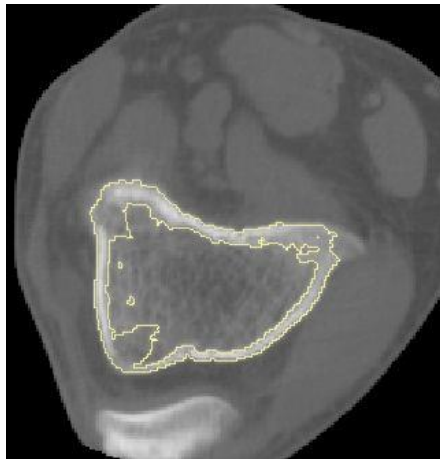
The enclosed part is added to the ROI.

## (4) Extraction mode

Press the button to enter Extraction mode.
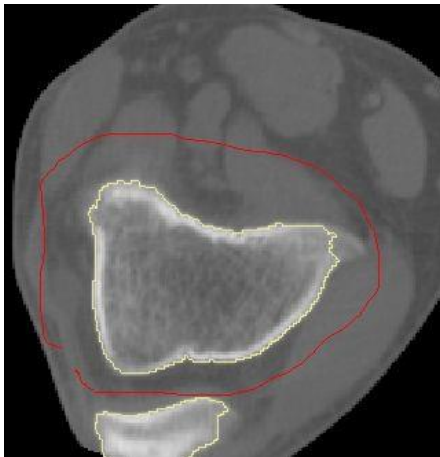


As shown above, surround the part you want to extract with the right mouse drag.
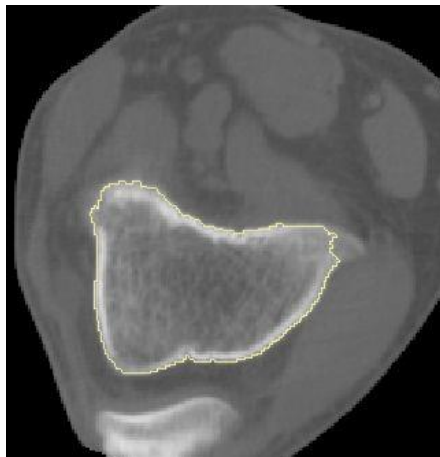


Then, only the enclosed part is extracted as the ROI.

**(5) Move mode**

 Select "Contour Segment" from "Select Shape," and select the "Move" button. Then, select the ROI group you want to convert to.
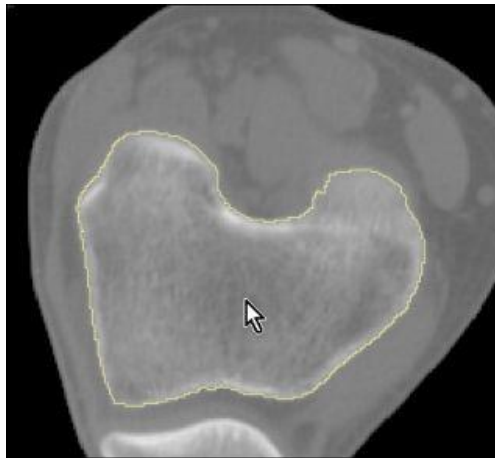


As above, specify the area you want to change in the ROI group with a right click of the mouse.
At that time, select the same ROI group as that to be specified.
(In the above case, ROI group 1, which is yellow, is selected)
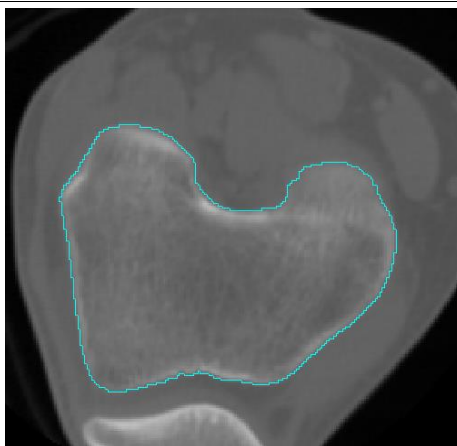




Then, the ROI group is converted from the designated ROI group 1 (yellow) to the ROI group 3 (blue) you want to convert to.

## (6) Filling / Pixel Filling
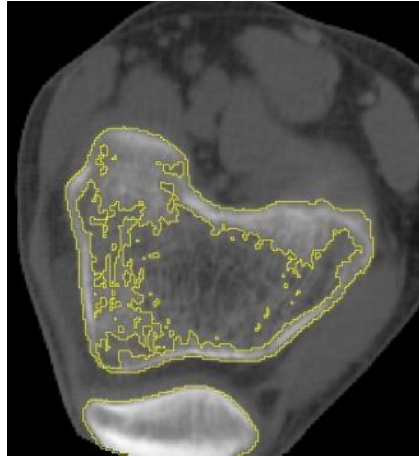

*Original state*

If you push this button in the original state as shown above, fill in the hole of the ROI as shown below.
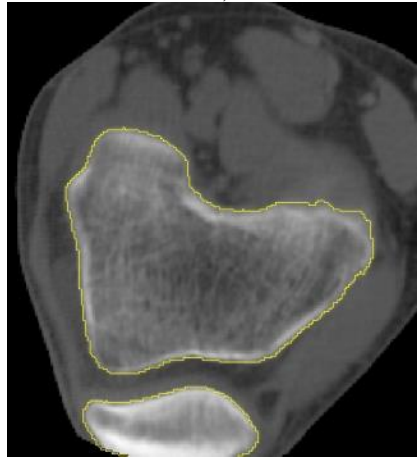


When this button is pushed in the original state (as shown above), fill the holes with the number of pixels less than the specified number in the ROI (as shown below).
The figure below is shown when the value of "Pixel" is 1000.
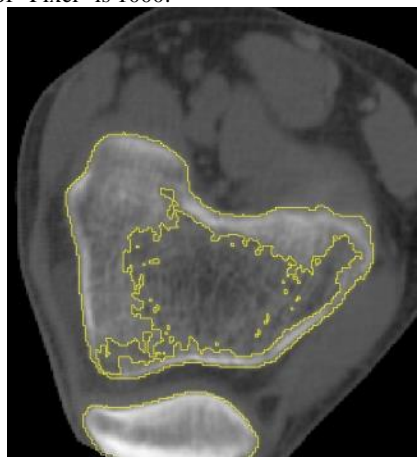
**(7) Denoising / Pixel Denoising**
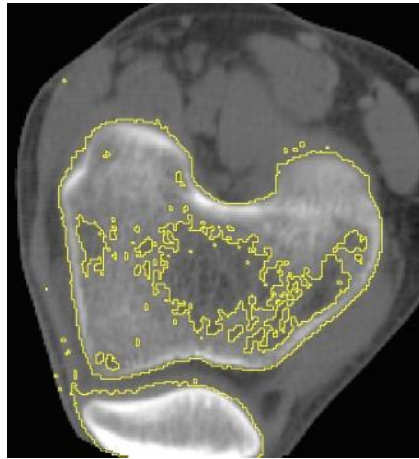

*Original state*

Pressing this button in the original state as shown above will remove the noise as shown below. The figure below shows the case where the value of "Pixel" is 2.
Note: Even in areas other than noise, the corners of the contour are rounded.
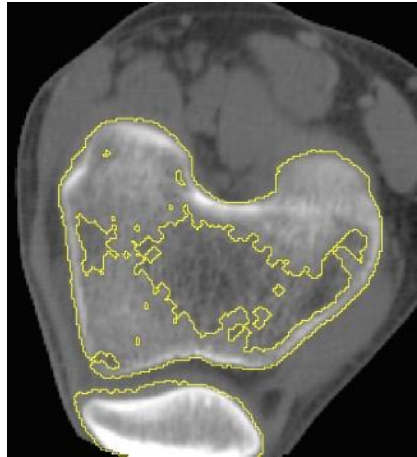


When this button is pressed in the original state as shown above, noise, which is less than the specified number of pixels within the ROI, is removed as shown below.
Note: The corners of the contour are not rounded.
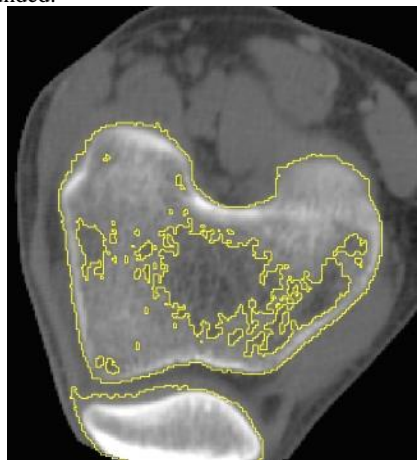
**(8) Closing**
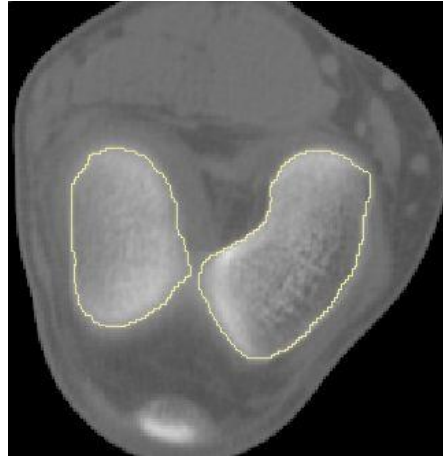


*Original state*

When this button is pushed in the original state (as shown in the above figure), connect the parts that are within a short distance (as shown in the figure below).
Even in areas other than noise, processing is done to round out the concave angle of the contour.



**(9) Shrink / Expand**
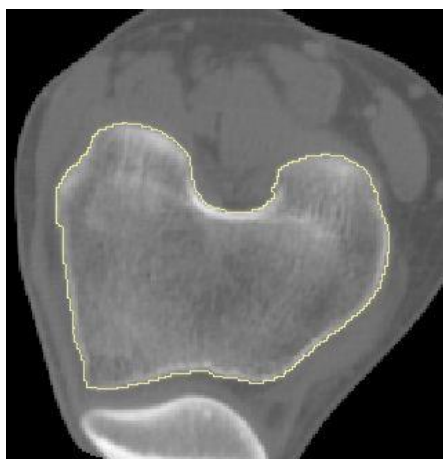


*Original state*

When this button is pushed in the original state as shown above, it shrinks ROI shown below.
Even in areas other than noise, processing is done to round out the concave angle of the contour.
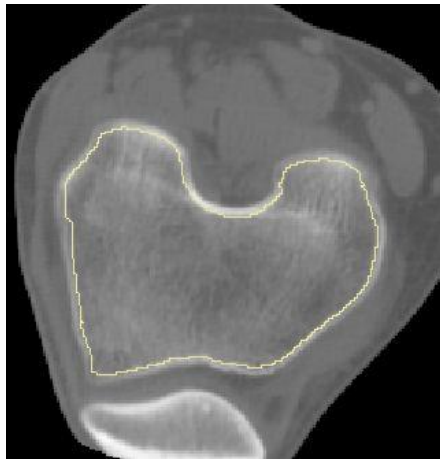
When this button is pushed in the original state as shown above, it expands ROI shown below.
Even in areas other than noise, the corners of the contour are rounded.
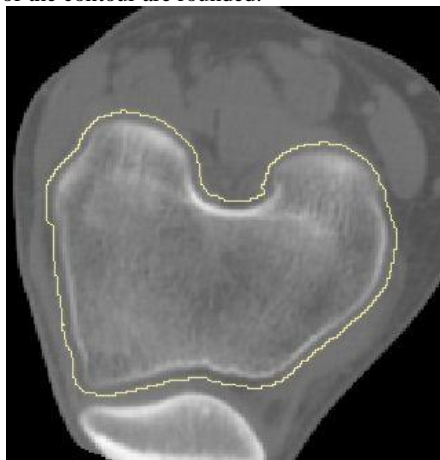
### 5.3.3 3D ROI operation example (example of deletion by specifying pick)

By "Select Region by Pick," it is possible to designate groups (parts) that are connected three-dimensionally, and to perform extraction and cutting.

In the following example, the patella is deleted after binarization by thresholding.



1. Select the "3D ROI" menu.
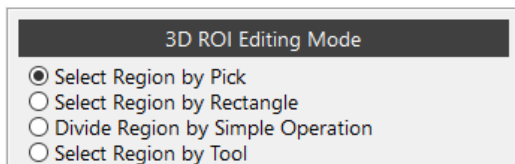


2. Select "Select Region by Pick" mode to delete the patella.
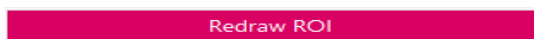


3. Display the area extracted as ROI by the "Redraw ROI" button.
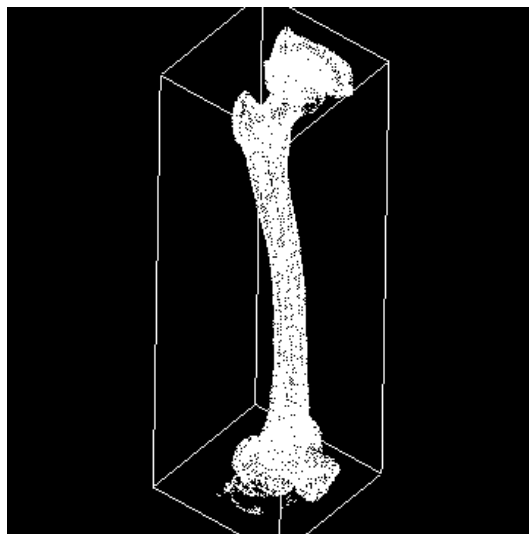


4. Pick the patella by right mouse click.
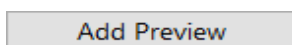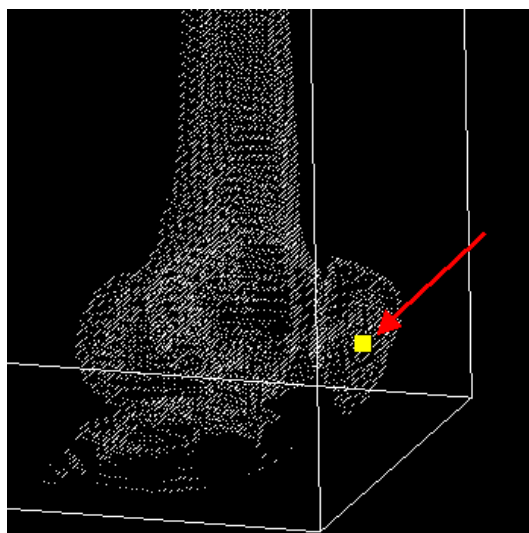




5. If the position of the right mouse pick is correct, press the "Add Preview" button.
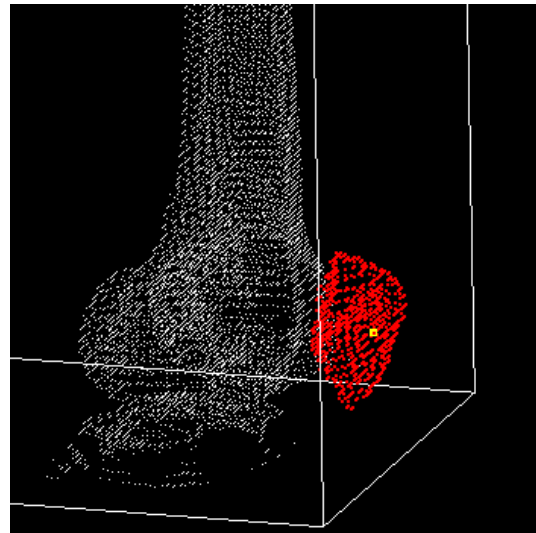
**[ Processing of Preview Region ]**

- ○ Extract from Selected ROI
- ● Delete from Selected ROI
- ○ Move from Selected ROI to Target ROI
- ○ Add from Selected ROI to Target ROI
- ○ Delete from Target ROI

6. If an area is specified, the mode selection will be active as shown on the left; select Delete mode to delete the area.

**Execute**

7. Press the button in the left figure to delete.



MECHANICAL FINDER

Delete preview region from selected ROI, OK?

Yes    No
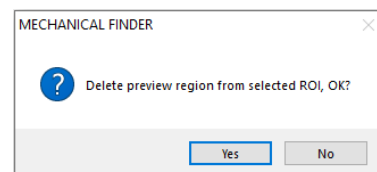
8. A dialog will be displayed; select "Yes" if everything is correct.
   When the preview area (shown in red) is deleted, it is immediately reflected in the data extracted as ROI.

**2D ROI**

9. When you confirm the deleted part with the "2D ROI" button, it is reflected as follows: "ROI area of patella has been deleted."



39
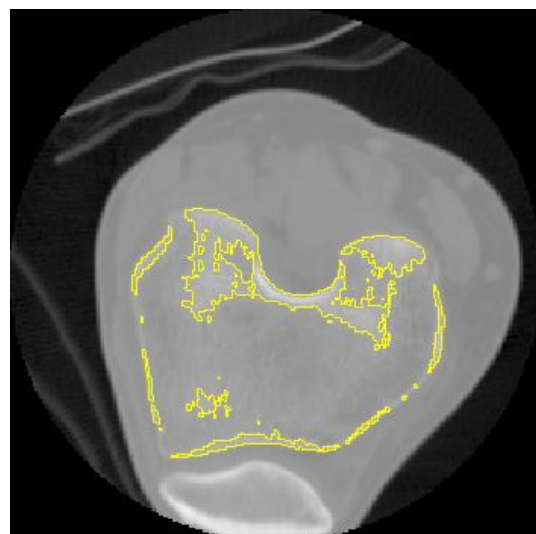
## 5.3.4 3D ROI operation example (example of rectangle specification)
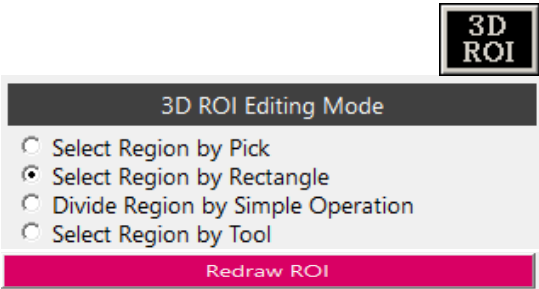


Select the "3D ROI" menu.

Select "Select Region by Rectangle" mode to delete the distal femur.



Display the area extracted as ROI by the "Redraw ROI" button.
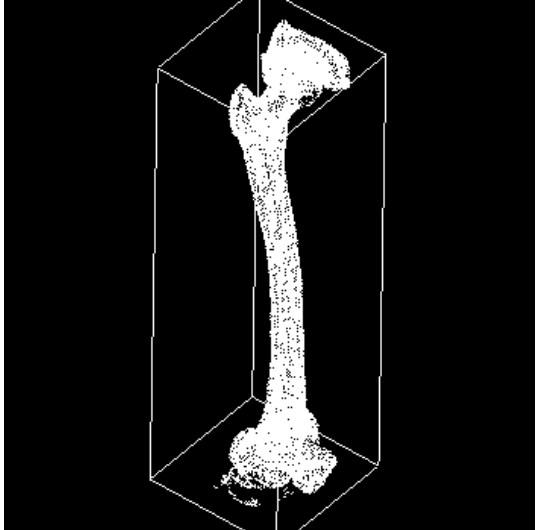


Enclose the distal end of the femur with a rectangle by right mouse drag.
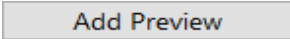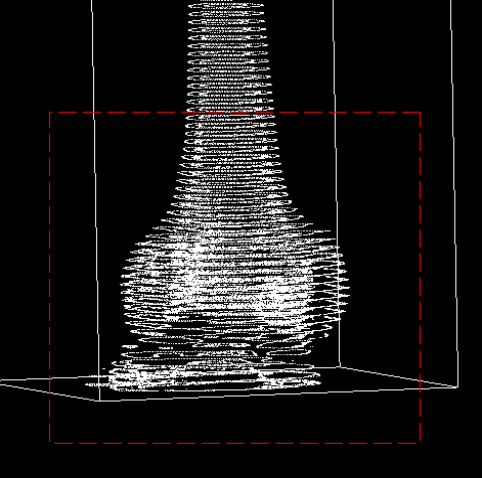


If the selection range is correct, press the "Add Preview" button.
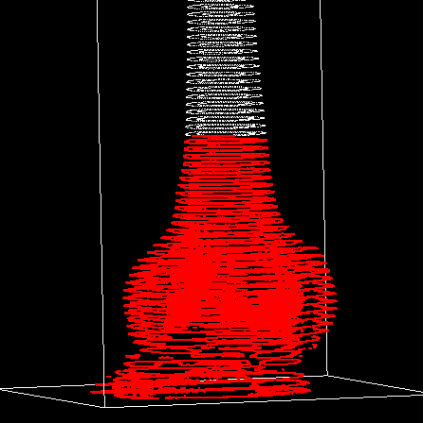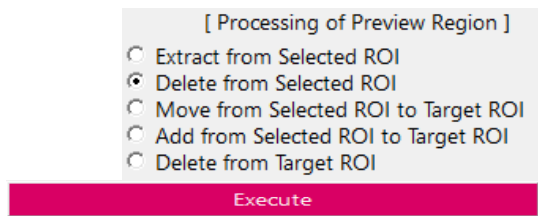
**[ Processing of Preview Region ]**

○ Extract from Selected ROI
● Delete from Selected ROI
○ Move from Selected ROI to Target ROI
○ Add from Selected ROI to Target ROI
○ Delete from Target ROI

Execute

If an area is specified, the mode selection will be active as shown on the left; select Delete mode to delete the area.

Press the button in the left figure to delete.

MECHANICAL FINDER ×

❓ Delete preview region from selected ROI, OK?

Yes    No
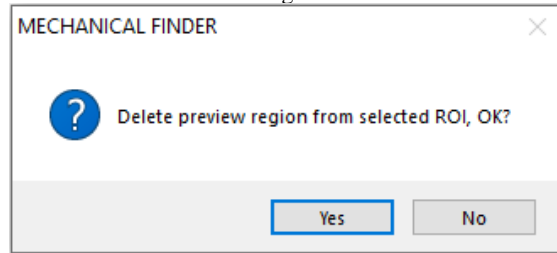
1. A dialog will be displayed; select "Yes" if everything is correct.
   When the preview area (shown in red) is deleted, it is immediately reflected in the data extracted as ROI.



41
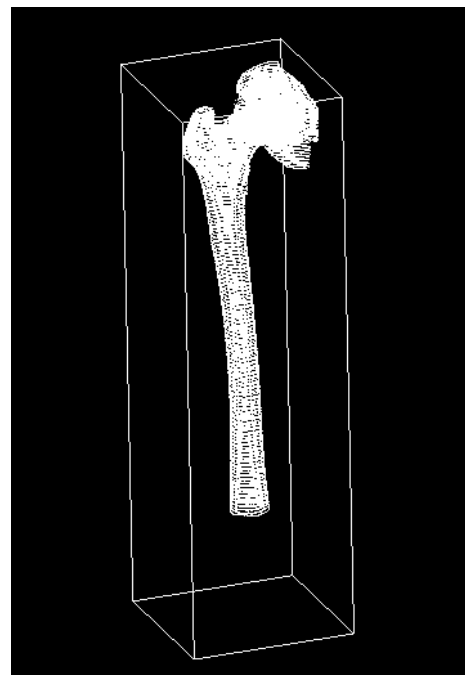
5.3.5 3D ROI operation example (example of Divide Region by Simple Operation)

1. Select the "3D ROI" menu.

**3D ROI Editing Mode**
- ○ Select Region by Pick
- ○ Select Region by Rectangle
- ● Divide Region by Simple Operation
- ○ Select Region by Tool
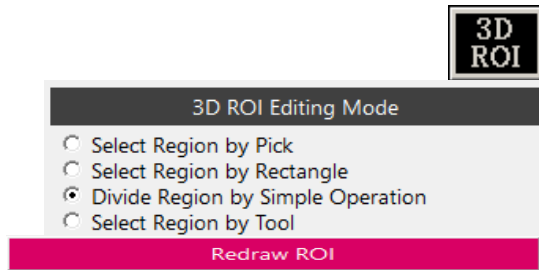
**Redraw ROI**

2. Select the mode "Divide Region by Simple Operation" to divide femur head from acetabulum.

3. Use the "Redraw ROI" button to display the ROI extraction area.
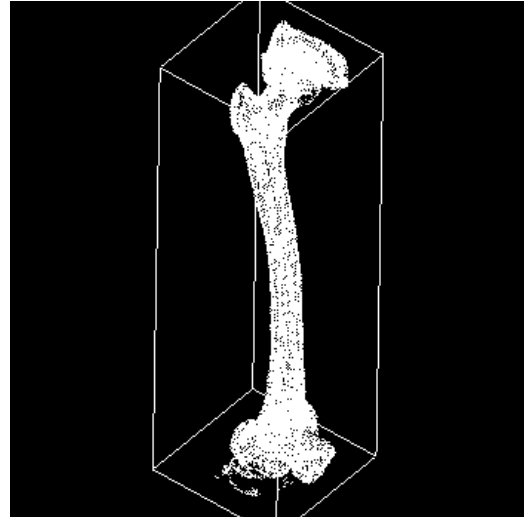


4. Right-drag the mouse to select a proximal portion of femur in a rectangular shape.
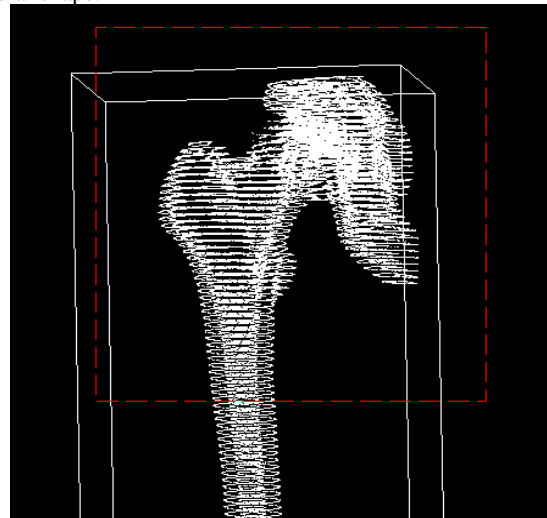


**Add Preview**

5. If the selection is OK, press the "Add Preview" button.
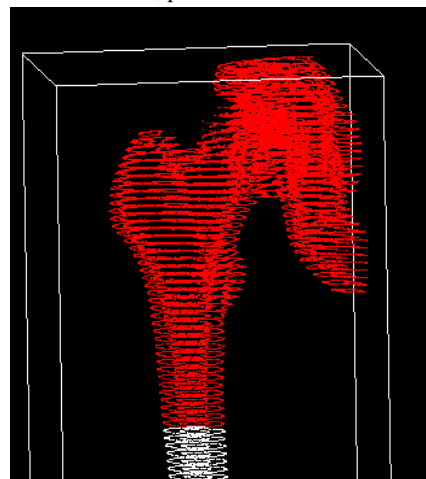
6. Right-click a divided portion to leave in ROI and a portion to delete from ROI.



Pick point 2

Pick point 1
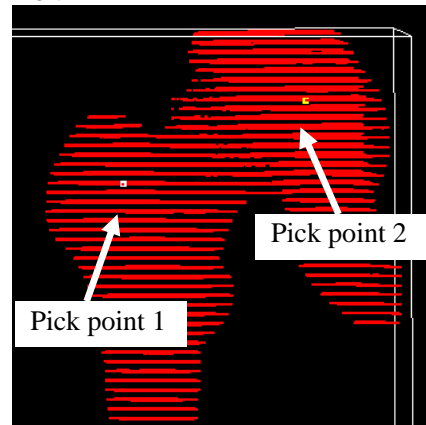
7. Press the "Separate Preview" button.
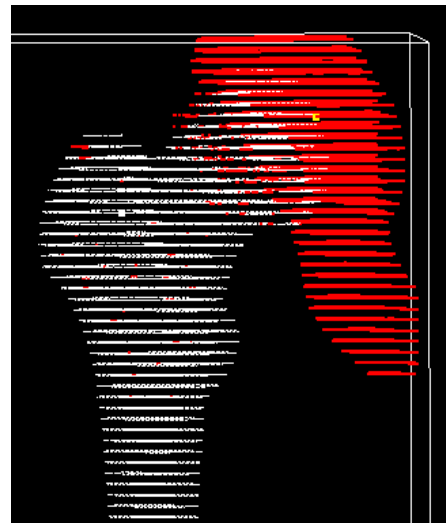
**Separate Preview**



8. After internal calculation, the portion to be divided remains in the preview. Select the mode "Delete from Selected ROI" to delete it.

[ Processing of Preview Region ]

○ Extract from Selected ROI
● Delete from Selected ROI
○ Move from Selected ROI to Target ROI
○ Add from Selected ROI to Target ROI
○ Delete from Target ROI

**Execute**

9. Press "Execute" button to delete the portion that you want to divide from ROI.



MECHANICAL FINDER ✕

? Delete preview region from selected ROI, OK?

Yes    No
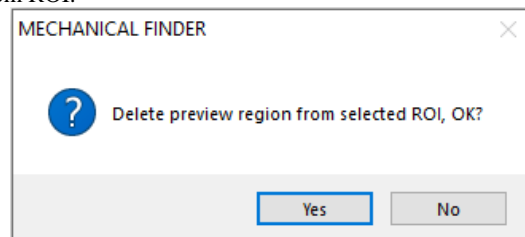
10. When the dialog selection is displayed, select "Yes" if it is OK.

**When the area is deleted, it is immediately reflected in the ROI extraction data.**

43
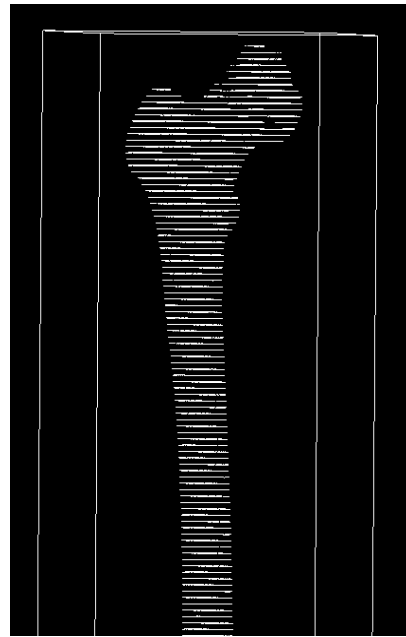
## 5.3.6 3D ROI operation example (example of deletion by tool)

In "ROI editing," proceed with the ROI extraction work to a certain point, then select "Select Region by Tool," and delete the part you want to delete.

In the following example, a pipe-shaped hole is made in the femoral head.
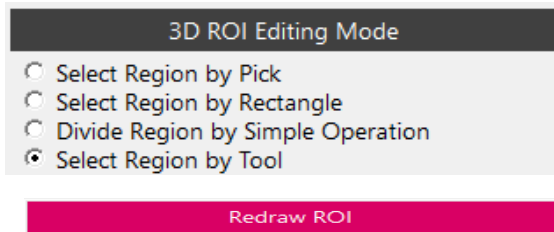
**1.** Select the "3D ROI" menu.

2. Because we will make a hole by tool, select the "Select Region by Tool" mode.

3. Display the area extracted as ROI by the "Redraw ROI" button.
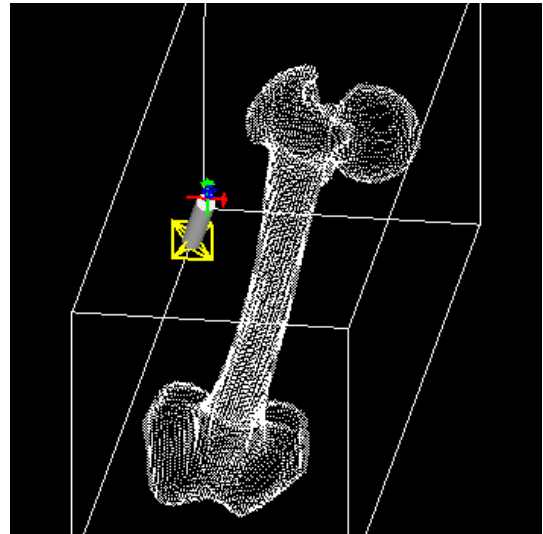


4. As the tool, we will use the original pipe shape.

5. Select the button on the left figure.

6. Select "two-point pick" button on the left figure. Then specify 2 points with the right mouse pick as shown below.



pick point 2

pick point 1
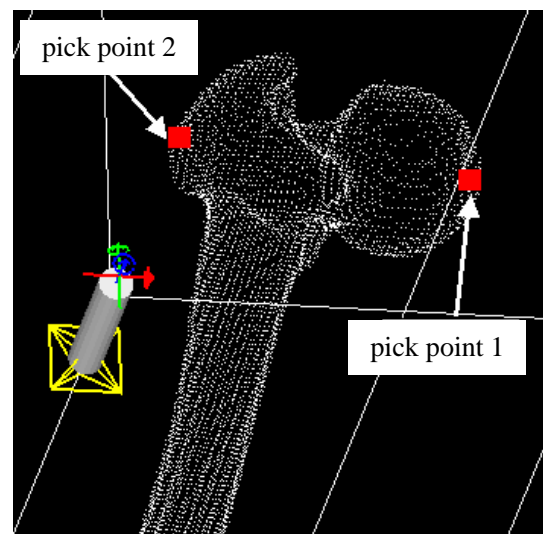
Then the tool will be placed as shown below.
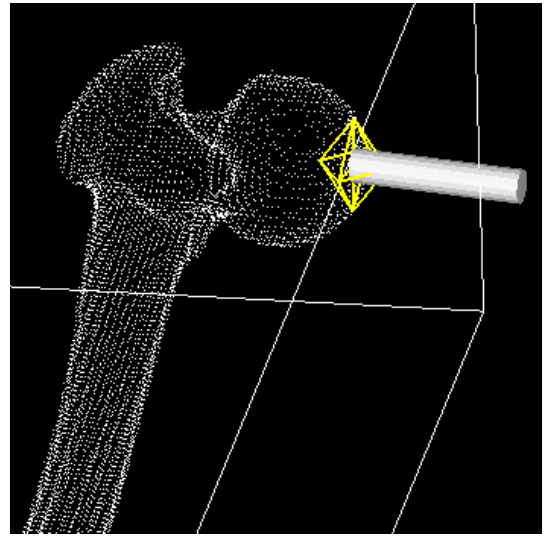


7. Change to move processing with the button on the left figure.

8. Select the button of move by local coordinates. Then, the axis, indicating the direction, is displayed by the tool as shown below.
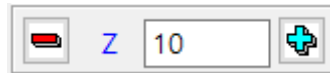
9. To delete about 30 mm in the blue arrow (Z) direction, press the minus button on the left figure three times to move the tool.
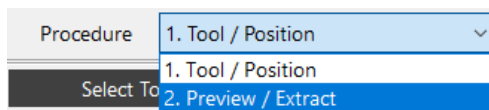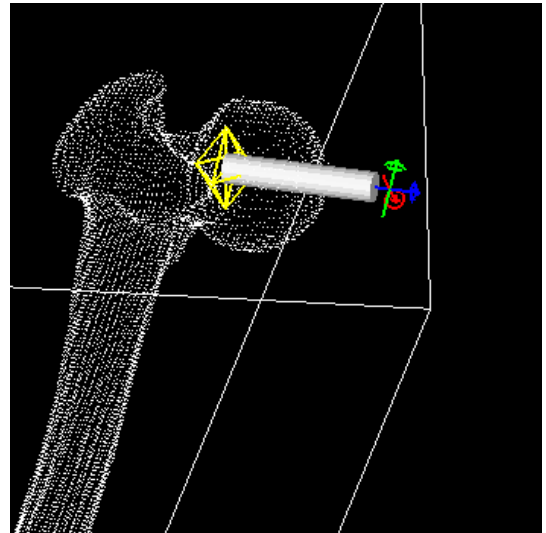


10. From "Procedure," select "2. Preview/Extraction."

**Add Preview**

11. If the placement of the tool is correct, press the "Add Preview" button.
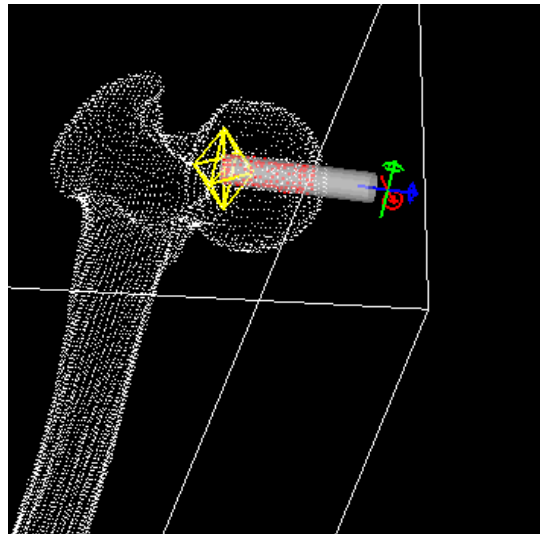


12. If there are additional places to select, repeat the same procedure.

**[ Processing of Preview Region ]**
- ○ Extract from Selected ROI
- ● Delete from Selected ROI
- ○ Move from Selected ROI to Target ROI
- ○ Add from Selected ROI to Target ROI
- ○ Delete from Target ROI

13. If an area is specified, the mode selection will be active as shown on the left; select the Delete mode to delete the area.

**Execute**

14. Press the button on the left figure to delete.



MECHANICAL FINDER ✕

? Delete preview region from selected ROI, OK?

Yes   No
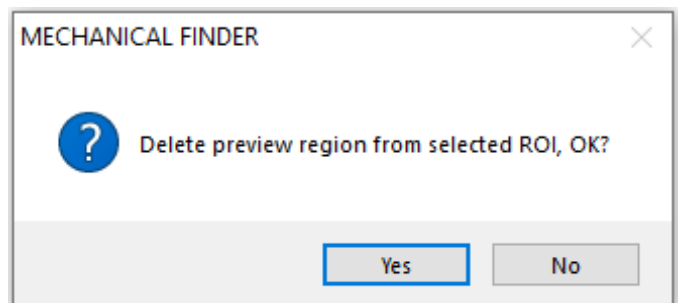
15. A dialog will be displayed; select "Yes" if everything is correct. When the preview area (shown in red) is deleted, it is immediately reflected in the data extracted as ROI.

2D
ROI

16. When you confirm the deleted part with the "2D ROI" button, it is
    reflected as in the following figure.
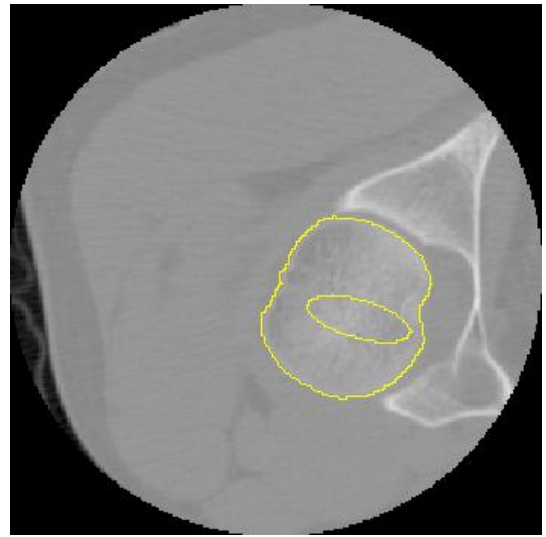
## 5.3.7 AI ROI Processing operation example

"AI ROI processing" is aimed at performing ROI extraction of the specific bone shape using deep learning. Now only femur extraction can be used. Refer 'Q3. About the detail of "AI ROI" function' of "Appendix 5.2 About ROI/Phantom" about the deep learning model.

In the following example, using the data of the leg imaging, the shape of the femur is extracted.
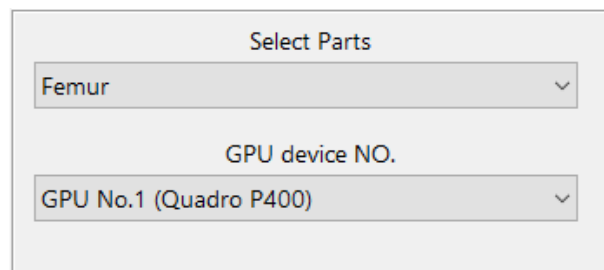
1. Select the "AI ROI Processing" menu.
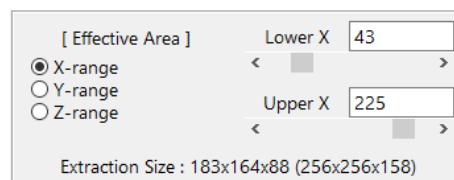
2. Confirm that "Step 1" is selected.

3. Select the part from "Select Parts" and GPU used for the process from "GPU device NO." (if no GPU machine, select CPU).



4. The range of AI processing is selected from "Effective Area". The selected range is displayed as the red line in 2D view and 3D model in the right side of UI.
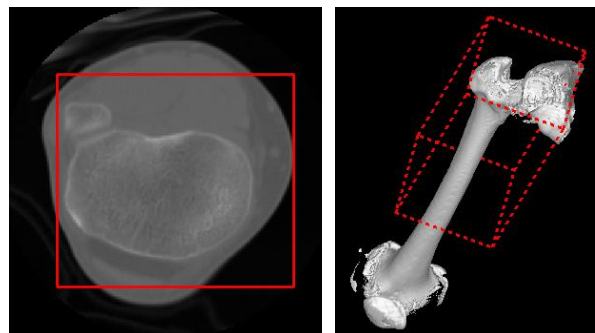




5. Click "AI Segmentation" button, and AI processing executes. Wait for 100% of the progress. Note that it takes long time if Effective Area is wide or CPU is selected.
When 100%, click "Close" button.

Processing Dialog

AI Segmentation Program
Status:Successfully finished

100% done

Stop      Log      Close
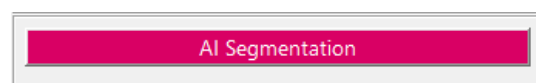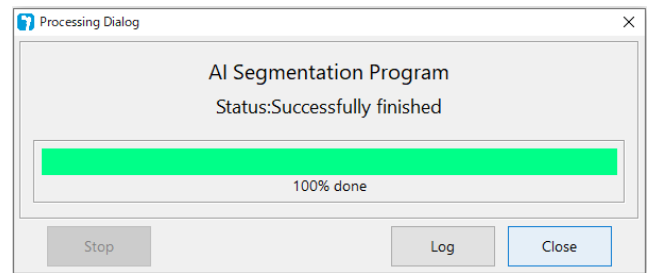
6. Select "Step2".
   In the list, ROI extracted by AI processing is displayed under "AI Segmentation Result", and ROI already set to ROI group is displayed under "Existing ROI".



Select items, and set ROI NO. using ROI Group

### AI Segmentation Result ###
NO.1 ---> ROI 1
### Existing ROI ###
ROI 1 ---> ROI 1
ROI 2 ---> ROI 2
ROI 3 (unused)
ROI 4 (unused)
ROI 5 (unused)
ROI 6 (unused)
ROI 7 (unused)

ROI Group  ☒ 1 2 3 4 5 6 7
3D View    ☑ Always View 3D of Sel. Item
Update ROI with Contents in List
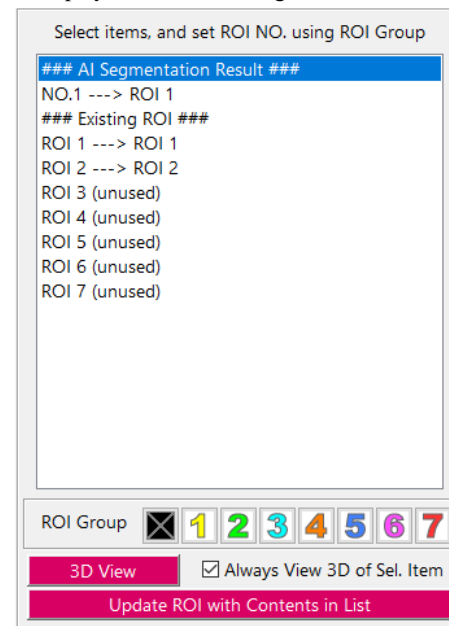
7. When "NO.1" of "AI Segmentation Result" is selected, the bone area extracted by AI is displayed in the right side of UI. If "3" from "ROI Group" under the list is selected, which the number is unused now, the extraction area is displayed in sky blue; the color of ROI group 3.



Select items, and set ROI NO. using ROI Group

### AI Segmentation Result ###
NO.1 ---> ROI 3
### Existing ROI ###
ROI 1 ---> ROI 1
ROI 2 ---> ROI 2
ROI 3 (unused)
ROI 4 (unused)
ROI 5 (unused)
ROI 6 (unused)
ROI 7 (unused)

ROI Group  ☒ 1 2 3 4 5 6 7
3D View    ☑ Always View 3D of Sel. Item
Update ROI with Contents in List
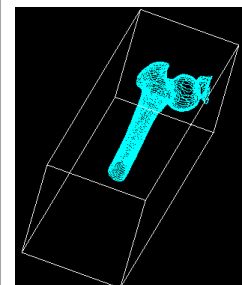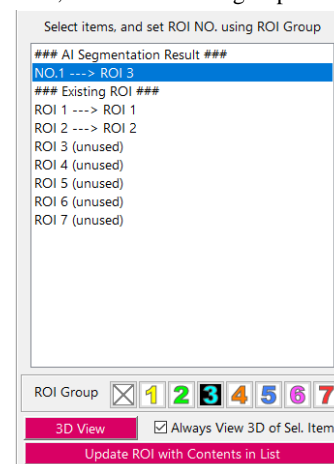
Update ROI with Contents in List

8. Click "Update ROI with Contents in List" button, and the setting in the list is reflected in the already extracted ROI group. Check the contents of "ROI 3" of "Extracted ROI" in the list.

50

```
### AI Segmentation Result ###
NO.1 ---> ROI 3
### Existing ROI ###
ROI 1 ---> ROI 1
ROI 2 ---> ROI 2
ROI 3 ---> ROI 3
ROI 4 (unused)
ROI 5 (unused)
ROI 6 (unused)
ROI 7 (unused)
```

9. The result of AI processing can be output to an external file. Reading the file, ROI group setting can be redone, not executing long time AI process again.
For this, select "Display Settings".

**Write**

10. With "Write" button, output the files of the result of extraction (the pair of .mhd file and .raw file).
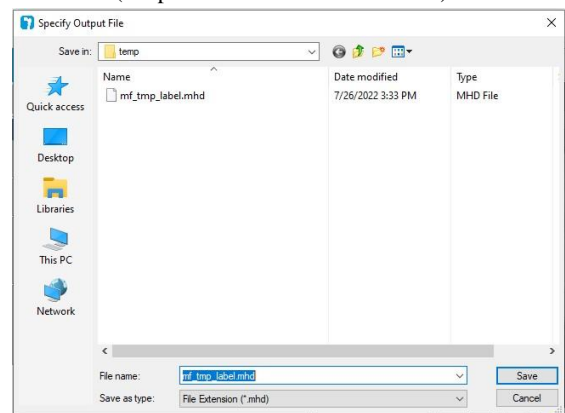


**Read**

11. With "Read" button, read the saved files (.mhd file and .raw file). The display is automatically switched to "Step 2".

12. After this, check and modify the extraction area with "2D ROI" and "3D ROI".

## 5.3.8 Keyboard shortcut

In "2D ROI processing" and "3D ROI processing," you can use keyboard shortcuts which change editor tools, move to other slice number, reset the camera direction, etc. The key bindings are "Ctrl key + designated key." You can see the detail by clicking "Keyboard Shortcut" button in the left-up of "ROI / Phantom" window.

**Keyboard Shortcut**

| | |
|---|---|
| Initialize All Camera Settings | Ctrl+R |
| Axial | Ctrl+Q |
| Coronal | Ctrl+W |
| Sagittal | Ctrl+E |
| 1 Slice Backward | Ctrl+B |
| | Ctrl+Left |
| 1 Slice Forward | Ctrl+F |
| | Ctrl+Right |
| Arbitrary Shape | Ctrl+1 |
| Rectangle | Ctrl+2 |
| Contour Segment | Ctrl+3 |
| Arbitrary Line | Ctrl+4 |
| Add | Ctrl+A |
| Delete | Ctrl+D |
| Undo | Ctrl+Z |
| Filling | Ctrl+X |

**Keyboard Shortcut**

| | |
|---|---|
| Initialize All Camera Settings | Ctrl+R |
| View from Z+ direction | Ctrl+5 |
| View from Z- direction | Ctrl+6 |
| View from X+ direction | Ctrl+7 |
| View from X- direction | Ctrl+8 |
| View from Y+ direction | Ctrl+9 |
| View from Y- direction | Ctrl+0 |

2D ROI

| Action | Key binding | Icon |
|---|---|---|
| Initialize All Camera Settings | Ctrl + R | |
| Axial | Ctrl + Q | |
| Coronal | Ctrl + W | |
| Sagittal | Ctrl + E | |
| 1 Slice Backward | Ctrl + B<br>Ctrl + <- | |
| 1 Slice Forward | Ctrl + F<br>Ctrl + -> | |
| Arbitrary Shape | Ctrl + 1 | |
| Rectangle | Ctrl + 2 | |
| Contour Segment | Ctrl + 3 | |
| Arbitrary Line | Ctrl + 4 | |
| Add | Ctrl + A | |
| Delete | Ctrl + D | |
| Undo | Ctrl + Z | |
| Filling | Ctrl + X | |

3D ROI

| Action | Key binding | Icon |
|---|---|---|
| Initialize All Camera Setting | Ctrl + R | RNC |
| View from Z+ Direction | Ctrl + 5 | |
| View from Z- Direction | Ctrl + 6 | |
| View from X+ Direction | Ctrl + 7 | |
| View from X- Direction | Ctrl + 8 | |
| View from Y+ Direction | Ctrl + 9 | |
| View from Y- Direction | Ctrl + 0 | |
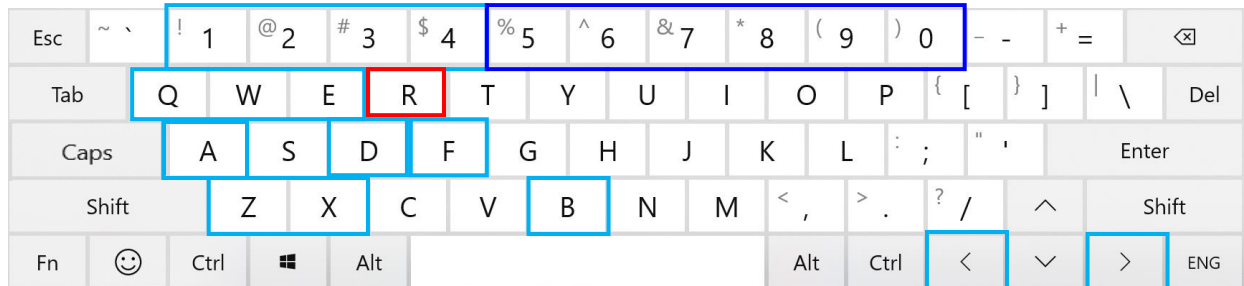


Red: Common    Sky blue: 2D ROI    Blue: 3D ROI

## 5.4 Image processing

Image processing includes image processing used in 2D ROI processing and image processing used in effect processing. They are similar, but there are differences in image processing procedures, and differences in whether they are done two-dimensionally or three-dimensionally.

**\*Effect Processing has been deleted in MECHANICAL FINDER 12.**

### (1) Image processing of "2D ROI Processing"

Image processing used in "2D ROI processing" processes the original CT image through the process as shown below. The processing calculation is done on a slice-by-slice basis, and it is possible to process multiple slices at the same time in a "Transparent Image."

Original Images

whether or not Clamp Processing

Clamp Processing

whether or not Image Processing

Noise Level

Edge Enhancement

Synthesize Processing

Threshold
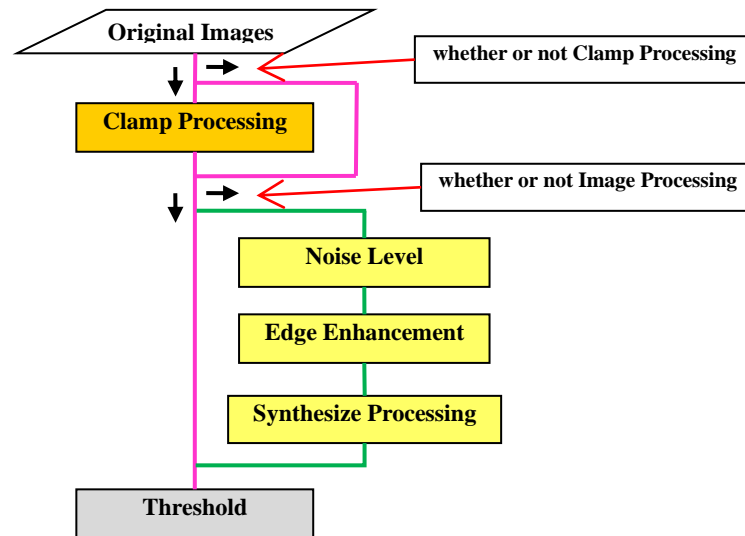
### (2) Image processing of "Effect Processing"

Image processing used in "Effect Processing" will process the original CT image through the process as shown below. The processing calculation is done with all slices (voxel) and is calculated three-dimensionally.

Original Images(All)

2D Viewer

whether or not Clamp Processing

Clamp Processing

whether or not Image Processing

Increment Data

3D Image Processing

Threshold

Morphology Processing

Extract Part

3D Viewer
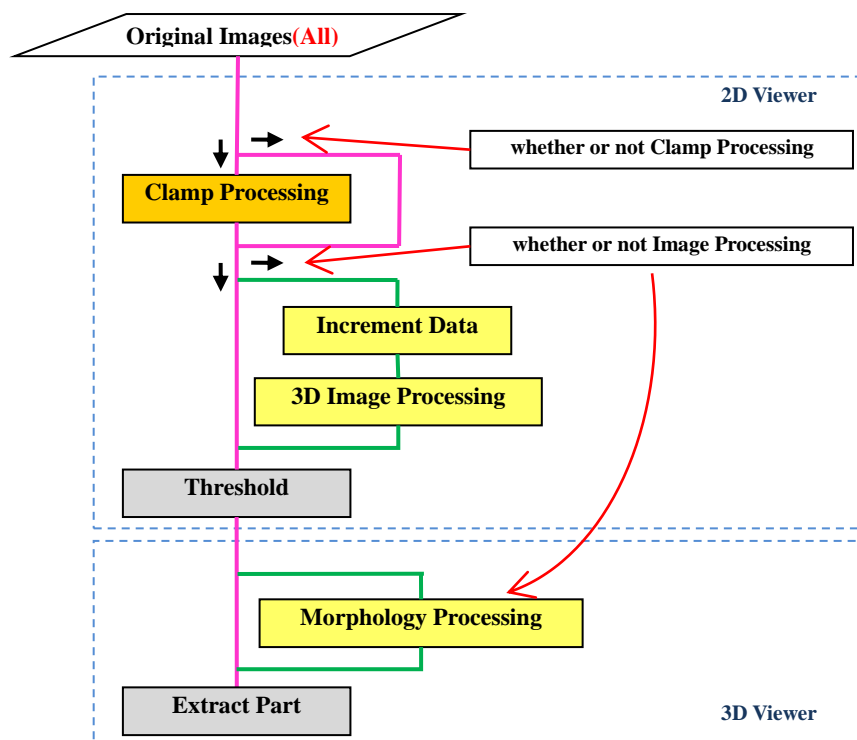
**(3) Clamp processing details**

Clamp processing is a function to limit the minimum and maximum values of a CT original image. By setting the threshold by the range (%), it is possible to process the image with only the ROI. All density values below the minimum value of the range are limited to the minimum value of the range, and all density values above the maximum value of the range are limited to the maximum value of the range. In the figure below, the CT original image is displayed with/without a clamp processing. (Contrast is disabled.)
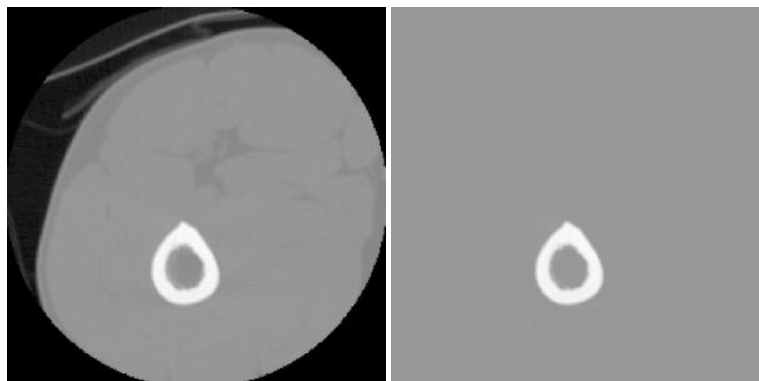


**(4) Noise Level**

Noise level processing is mainly used to suppress noise emphasis by edge reinforcement processing. The stronger the setting of edge reinforcement, the more noise is added and tends to be amplified, so it is necessary to select the optimum setting value together with the set value of edge reinforcement.

**(5) Edge Reinforcement (2D ROI Processing)**

The edge reinforcement processing emphasizes the part of the CT original image where the gradient is large (density value change is large). Since it is possible to emphasize areas, such as the boundary between soft tissues and bones, it is easier to separate by image processing based on threshold values. However, since the possibility of picking up noise also increases, it is necessary to find the value of the setting that reduces noise and makes bone extraction easier by combining with "noise level," "clamp processing," and other factors.

**(6) Magnification / Merge**

You can specify the magnification of the edge-reinforced image when compositing with the original image. Increasing the numerical value of the magnification during composition leads to a more reinforced edge.

You can also select the method of combining with the original image (ascending gradient, descending gradient, or ascending/descending gradient).

**(7) Interpolating**

When performing 3D image processing by Effect Processing, the large distance gap difference between the pixels and the slice of the original image is problem. The interpolating process is to increase the number of slices to reduce this distance gap difference in 3-dimensional image processing. By increasing the number of slices up to 4 times at maximum, 3D image processing results will be good.

Since this process requires a lot of storage capacity, it is designed to be processed in the disk area, so it will not increase memory usage unnecessarily.

**(8) Edge Reinforcement (Effect Processing)**

This is very similar to the above procedure (Noise Level/Edge Reinforcement/Magnification/Merge), and 3-dimensional image processing, which handles a CT as 3-dimensional data instead of 2-dimensional image processing and is applied in a series of steps.

**(9) Morphological Operation**

This is the process to be performed when extracting parts in the "Extraction in 3D" of Effect Processing. In the three-dimensional construction, when extracting by the region enlargement method, even if the parts are slightly in contact with each other, they are treated as the same part. For this reason, processing is performed to make it easier to separate parts by shrinkage and expansion by three-dimensional morphology processing.

**(10) Image Processing Threshold**

Specify the threshold after image processing. The threshold at this time is a value with respect to the numerical value after the image processing, and it is different from the CT value of the original image.

# Chapter 6 Mesh

In the mesh generation process in this software, mesh shapes are generated in the following order.

1. Generate surface mesh.

2. Insert and align import shape.

3. Generate internal mesh.

If you click "OK" after making the necessary settings, the internal mesh generation program will be executed.
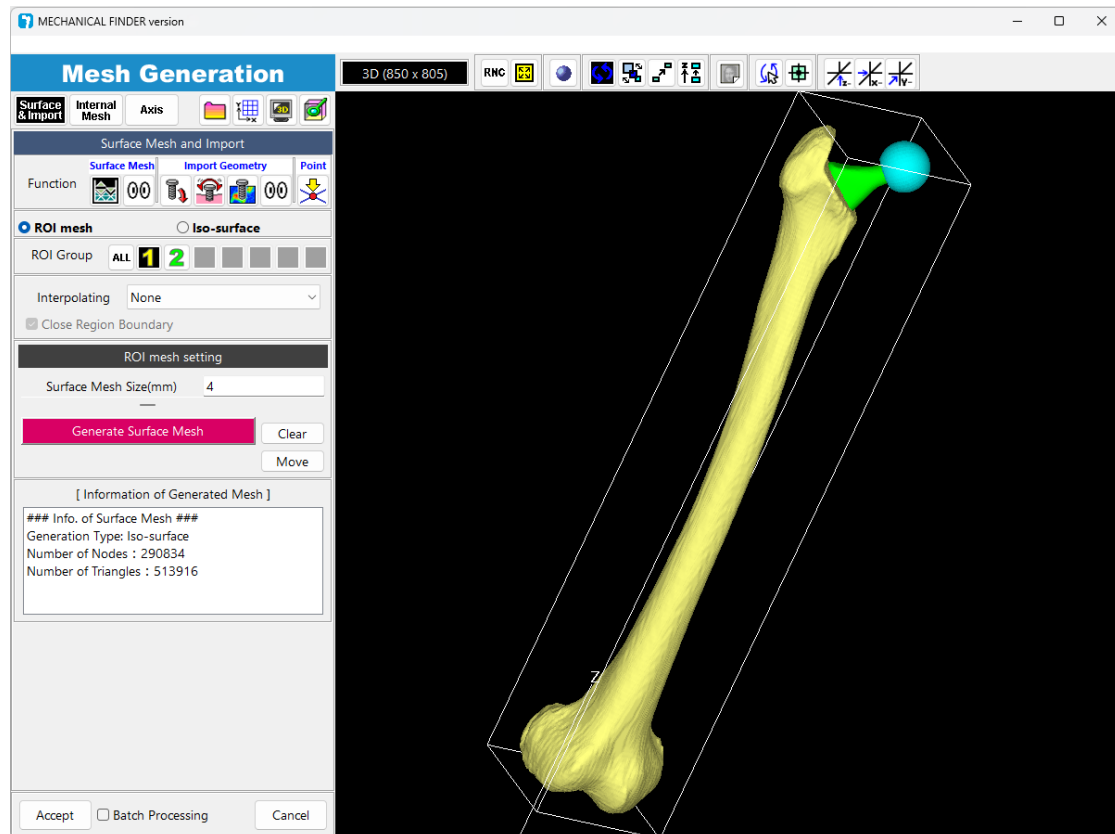


| Icon | Function |
|---|---|
| **Surface & Import** or **Surface Mesh** | Surface Mesh and Import |
| **Internal Mesh** | Setting of Internal Mesh (EE) |
| **Axis** | Setting of Axis |
| | Data Information |
| | Surface Display |
| | Viewer Settings |
| | CT Display |

## 6.1 Mesh Shapes / Import Shapes

Mesh shape and import shape are described below.

| Title | Detail |
|---|---|
| 6.2 Mesh Elements | Describes the type and nature of analysis elements of this software. |
| 6.3 Interpolating | Describes the algorithm of the interpolation process at surface mesh generation. |
| 6.4 Import process operation example (EE) | Steps through the operation of actually inserting the stem (implant) into the femoral bone. |
| 6.5 Generate shape for import (EE) | Describes how to use import shapes and how to create import shapes. |
| 6.6 Mesh / Import Basics | Describes the basic conventions for the import method. Please be sure to read it when importing. |

## 6.2 Mesh Elements

In "Chapter 6 Mesh," the following is used as input data, and a 3D mesh is automatically generated for it.

1. Data obtained by subjecting the CT original image to ROI extraction and binarization.

2. Geometry data, such as implant shape.

Please refer "Q2" in "Appendix 5.3 About Mesh Generation" for the algorithms used in the mesh generation.

### (1) Generated elements

The basic elements generated by this software are two types, the solid element and the shell element. (However, it is also possible to define gap elements, contact elements, and others. For details, refer to "Chapter 7 Material Sorting")

Whether or not to use the shell element is determined in "7.1 Material Sorting"
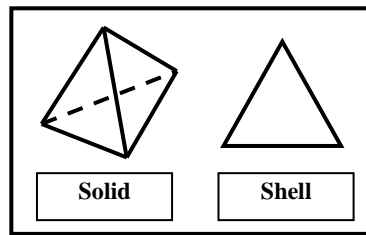


| Solid | Tetrahedral Element | For details, see "Appendix 4.4 Used Element" |
|-------|---------------------|-----------------------------------------------|
| Shell | Triangular element with virtual thickness to be generated on the surface. | |

### (2) Points to remember

(a) Shapes which meshes can be generated for:

> For hollow shape data, when the distance between the inner surface and the outer surface is extremely short (wall thickness is thin), satisfactory mesh shape cannot be generated.

(b) Algorithm characteristics:

> Since the mesh generation function of this software is an algorithm suitable for curved surface construction, it is not recommended for target planar construction.

(c) Relationship between mesh size and slice width and pixel width:

> If the binarized image has irregularities, depending on the fineness of the slice width and pixel width, the mesh to be generated also reflects the unevenness. This tendency becomes high, especially when the slice pitch of CT data is large.

(d) Increase in the number of meshes:

> The increase in mesh number is greatly affected by the following factors. Please consider the following before setting.
>
> i. The more complicated the shape, the higher the mesh number tends to be.
>
> ii. The smaller the "Minimum Size" in "Setting of Internal Mesh," the more the number of meshes tends to increase.

## 6.3 Interpolating

The interpolation processing type in "Surface Mesh and Import" can be selected from the following three types.

- None

- Slice x2

- Slice x3

By doing this, it is possible to eliminate jaggies on the surface mesh that occur when the slice interval of the DICOM data is large or when the "Basic Mesh Size" is set to a small value.

### (1) Example of effect

Following figures show surface meshes of CT data with large slice depth generated with 4mm mesh.



| None | Slice x3 |
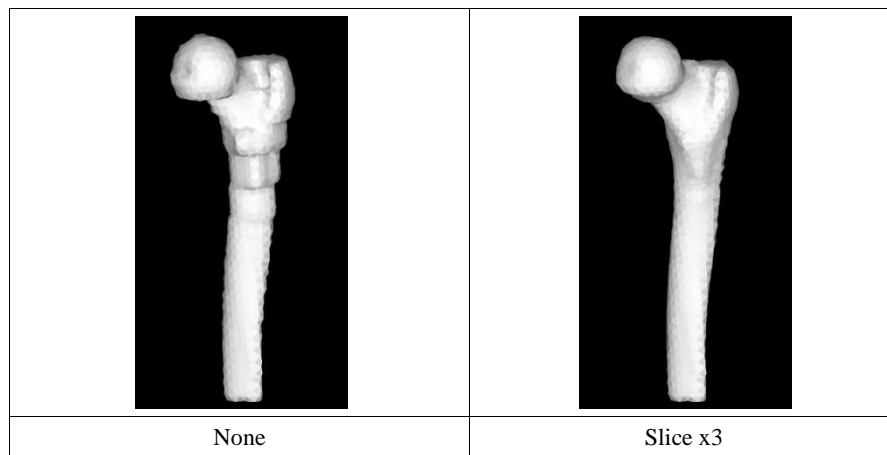
### (2) Internal processing routine

The routine used for interpolation processing generates new ROI data based on reference ROI data. This is done in order to increase the amount of information of new data without changing the original data, so the value of the original data will not be changed unnecessarily.

As an example, we will explain using two consecutive images along a slice axis (Z axis). (This process is applied for all slices.)



| (I) Image a | (II) Image b |
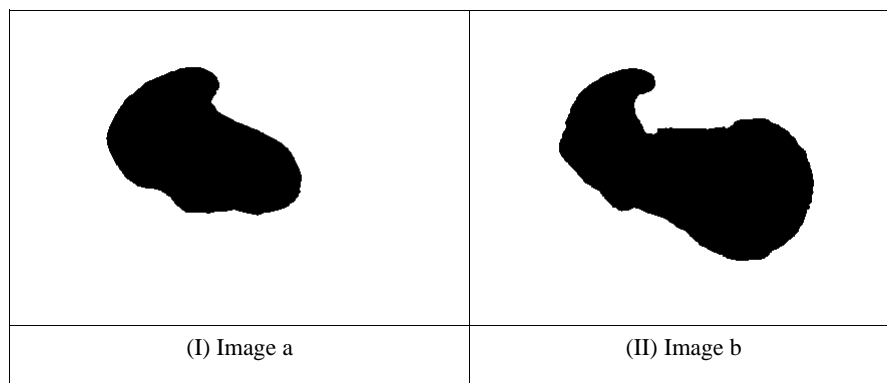
Assume that Images a and b in the above figure are present in consecutive slices.

(1) None

In this process, the original Image a and Image b are processed as they are.

(2) Slice x2

In this process, Image n as shown below is obtained from Image a and Image b, which are original data, and they are processed as new data in order of Image a/Image n/Image b.

(III) Image n
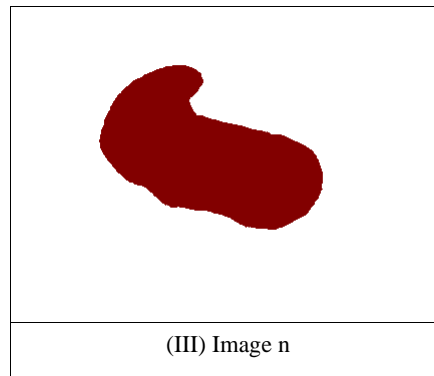
(3) Slice x3

In this process, Images n1 and n2, as shown below, are obtained from the original Images, a and b, by calculation, and processed as new data in the order of Image a/Image n1/Image n2/Image b.



| (IV) Image n1 | (V) Image n2 |
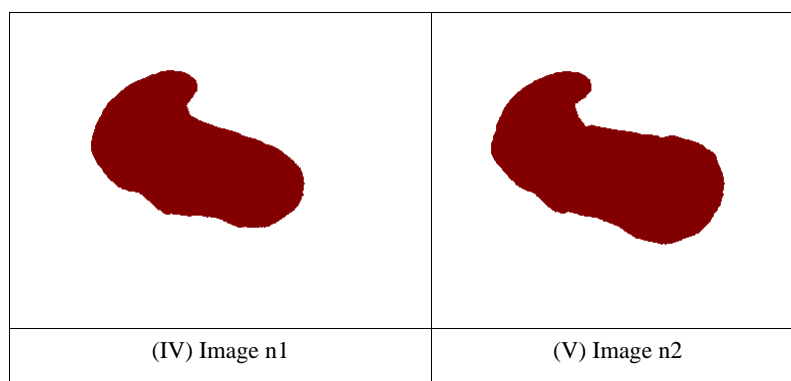
* Caution

Although this process can improve the shape of the surface mesh, this effect does not mean that you can increase the slice interval during imaging. This is because increasing the slice interval decreases the numerical reliability of material properties when material properties are determined from CT images.

## 6.4 Import process operation (EE)

After completing the surface mesh setting in "Surface Mesh," perform "Import Geometry." In the following example, we will insert the stem at the location of the femoral head. (The femoral head has been deleted simply in ROI extraction.)
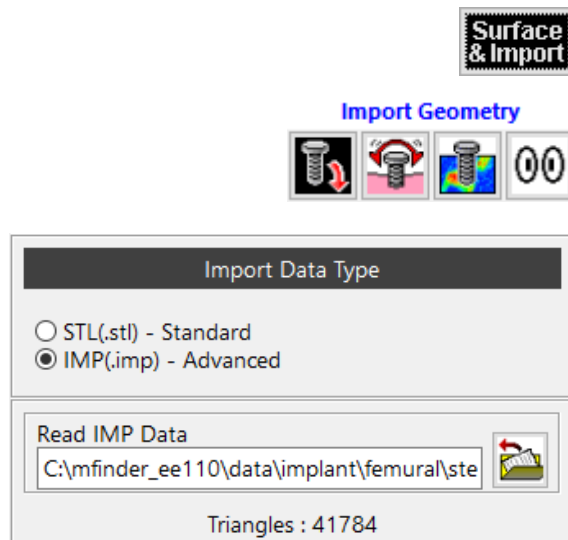
1. Select the "Surface Mesh and Import" menu.

2. Select the icon on the far left of "Import Geometry" (Read Geometry).

3. In this example, we read the stem of IMP format, which is an extension function.

- Select IMP format.

- Specify the file to read.

**Import Data Type**

○ STL(.stl) - Standard
◉ IMP(.imp) - Advanced

**Read IMP Data**
C:\mfinder_ee110\data\implant\femural\ste

Triangles : 41784

4. Then the stem appears as shown below.
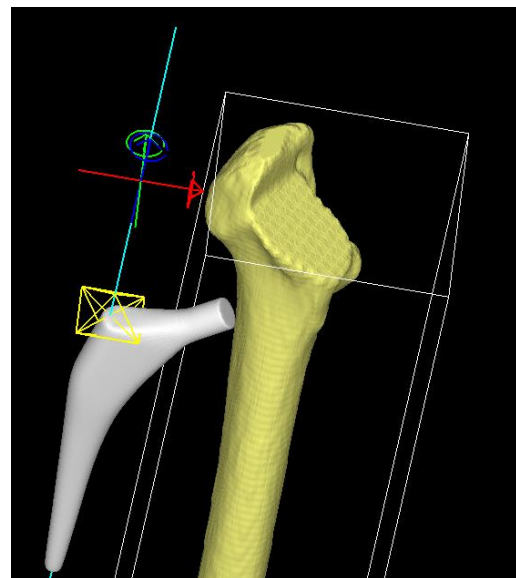


5. Change the menu to "Position Settings of Read Geometry."
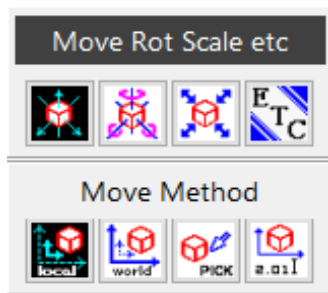
6. Select "Transfer"/"Move to Picked Point" and then move it by right clicking on the part shown in the figure below with the mouse.
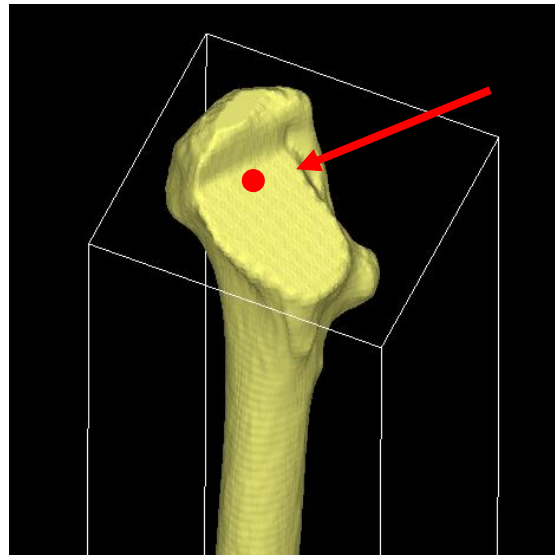


7. Then next, the implant is placed as shown in the figure below.
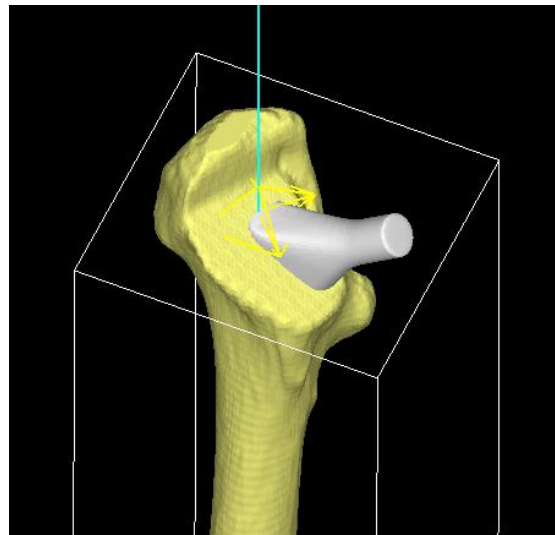


8. Select "Local Cord." Then, three axes indicating the direction of the implant are displayed as shown in the figure below.
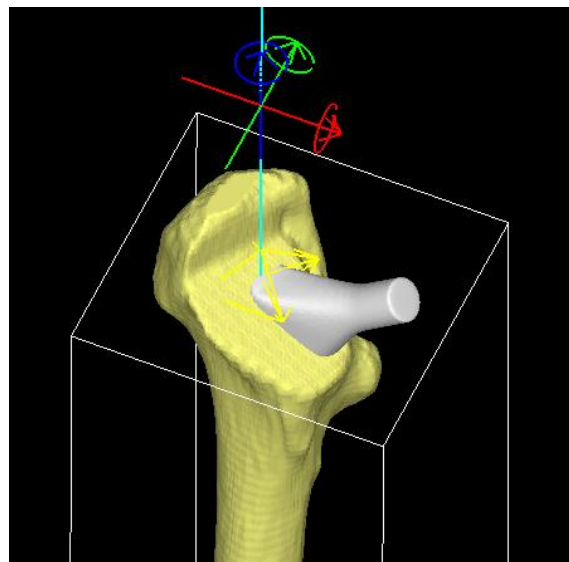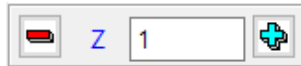
9. To move the implant in the blue arrow (Z) direction, press the minus button on the left figure several times (unit is mm).
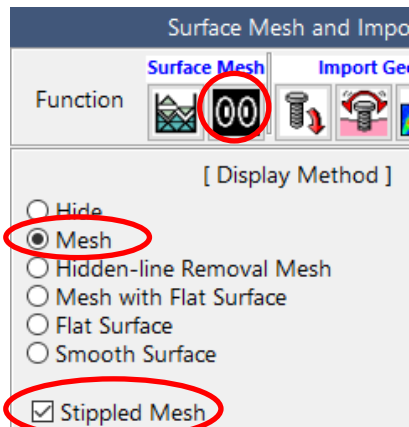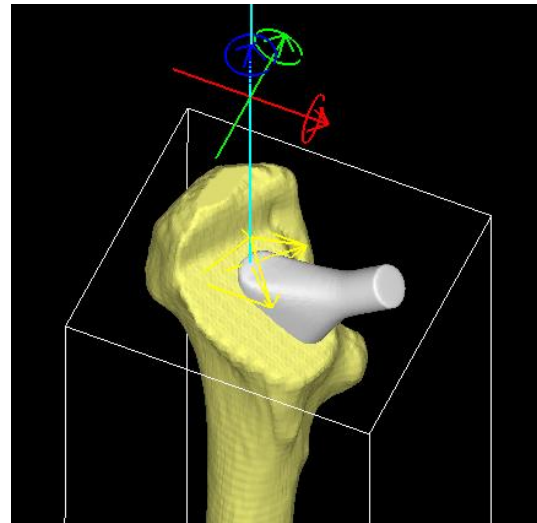




10. When the surface mesh is opaque and you want to know the inside, display the surface mesh in "Mesh" style or "Stippled Mesh" style.
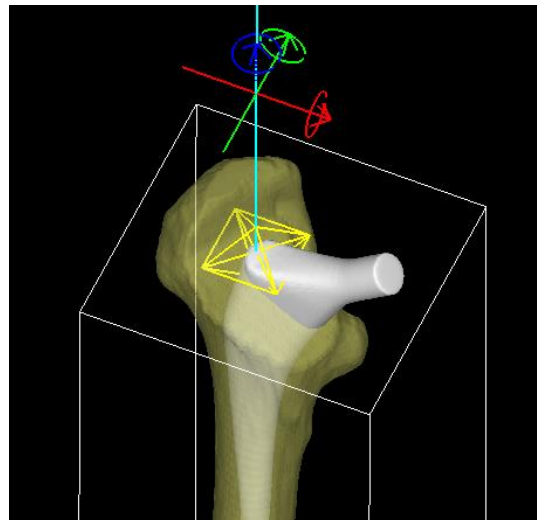


11. Then make fine adjustments by "Transfer," "Rotate," and "Scale," and place them in the proper position.
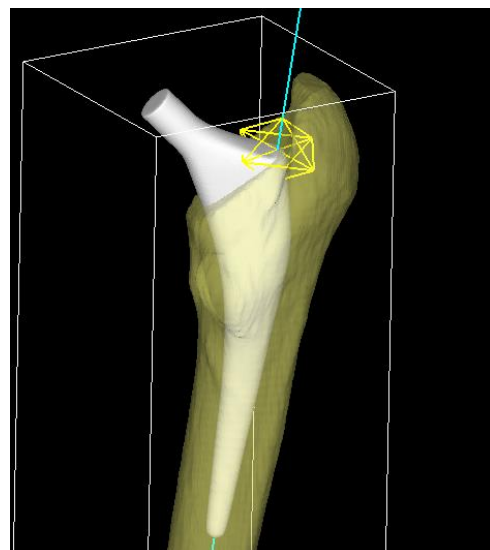
12. Although it is possible to perform positioning in this 3D mode, here, we introduce the method of positioning while displaying the bone density of the original CT image in 2D mode.

13. If you want to align in 2D mode, switch to the 2D position setting of the import shape with the icon on the left.
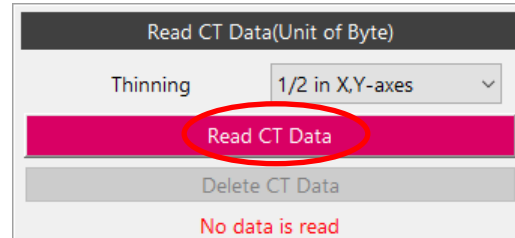


Then, pressing the "Read CT Data" button will display the 2D view as below and the slice display of the original CT image along the import axis, axial plane, coronal plane, sagittal plane and the cross section of the import shape will be displayed.
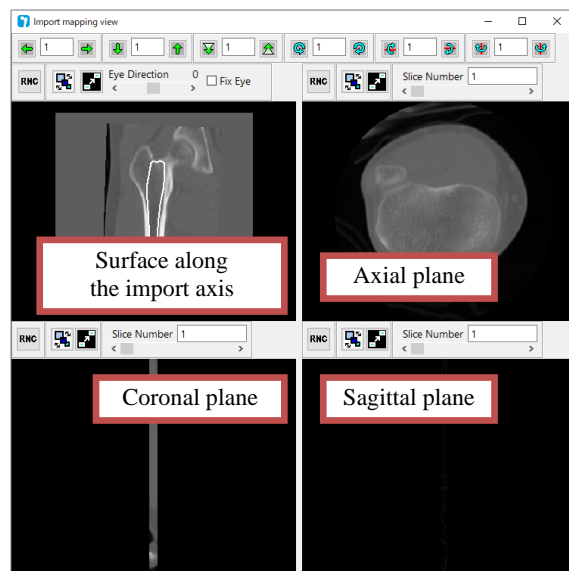


14. You can arrange the import shape in this 2D view. It is designed to align with the menu in the figure below.
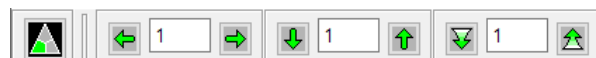


- The pairs of arrows in the upper figure are "move left and right," "move up and down," "move forward and backward," respectively.
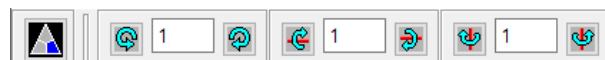


- The pairs of arrows in the upper figure are "rotate left and right," "rotate back and forth," "rotate around the basic axis," respectively. Use these to align.

15. Using these methods, you can place the import shape in the desired position/rotational state.

16. To confirm the import, proceed as follows.

65

- Select an import number from the list.

- Set the "Processing" to "Add."

- Press the "Add" button.
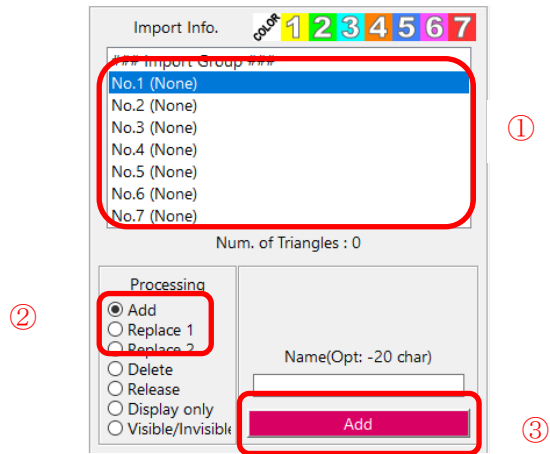


Up to 20 subgroups can be registered in one group of import numbers. (Total 7 x 20 = 140 shape)

If you press the "Add" button twice in succession, two identical shapes will be added. Please be careful.

17. If you want to add the same shape to another position, you do not need to reread the import shape again.
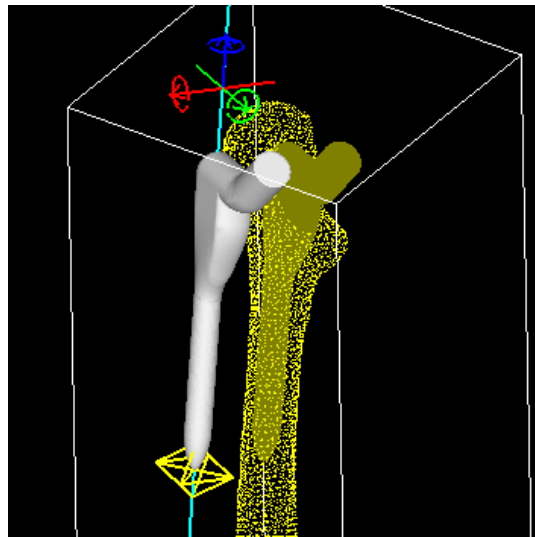


18. Please move, rotate, and scale from the current position and continue processing.

## 6.5 Generate shape for import (EE)

### (1)　Import shape in this Software

This software has shapes for import (STL format and IMP format) in the categories "femoral," "for jaw implant (jawbone)," and "others" in the following locations. The contents can be confirmed by opening "ReadMe.bmp" in each directory. However, these data are just sample geometries not practical ones.
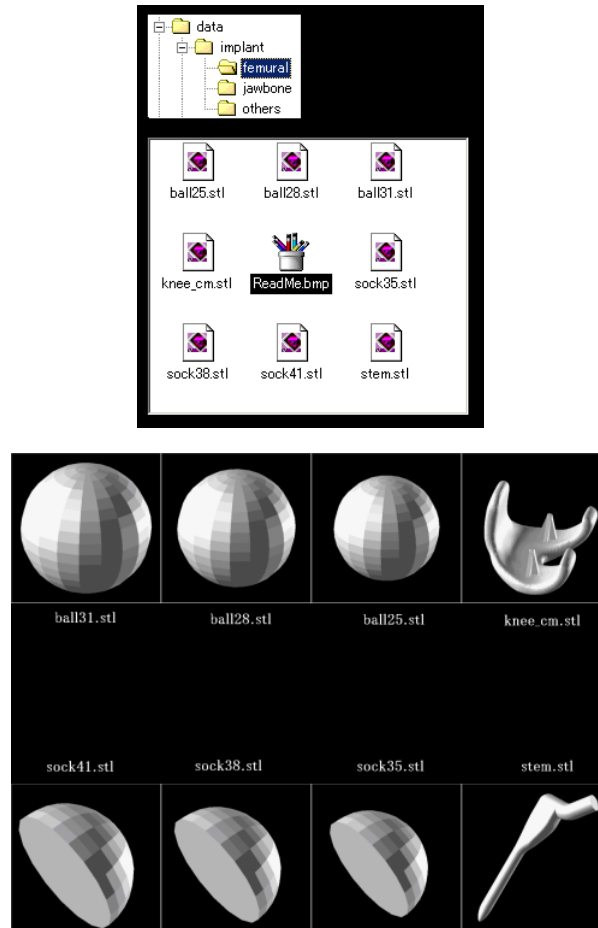




### (2)　How to create an import shape

The following methods are options for shape creation.
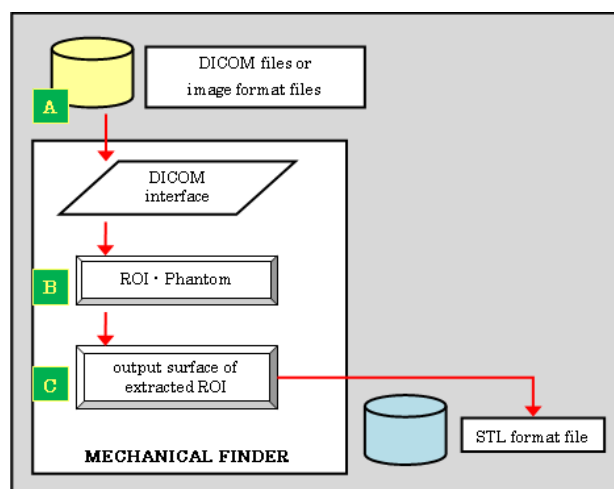


I. The procedure for creation on this software is to follow the operational steps below, which will create a shape that can be imported.

i.       Prepare the DICOM data on which the import shape is imaged (BMP, JPEG, TIFF are also available).

ii.       Extract ROI on this software and extract its shape.

iii.       Select "Chapter 12 "Output" Function" (for the above example, "12.2 Output Surface (ROI)"), generate the shape, and save the file as STL data.
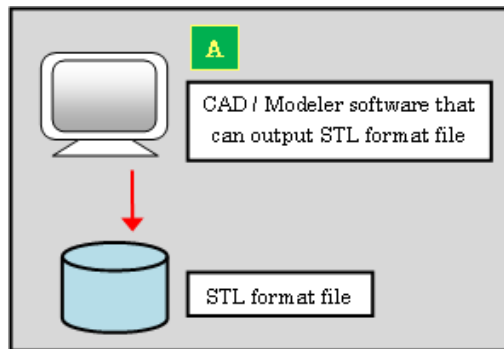


II.  The procedure for creation on CAD or modeler.

Create shapes on the CAD software or modeler software that can output STL.

## 6.6 Mesh / Import Basics

When generating meshes with groups, such as multiple ROI and multiple imports, please follow the following basic conditions. If these conditions are not satisfied, the operation load in the "Material Sorting" may increase, or a part of the mesh may be lost.

**(1)    How to set up groups in insertion work**
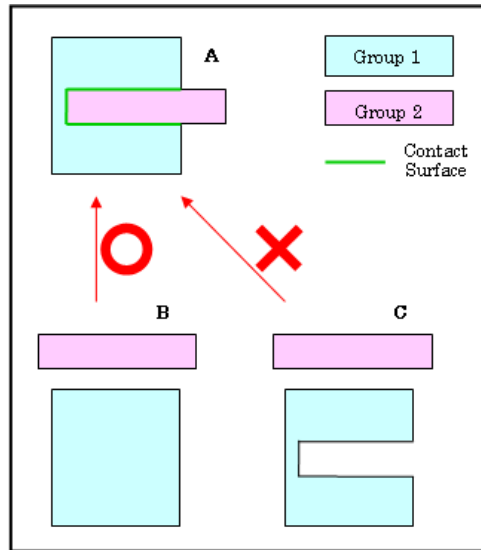


When mesh generation is performed, assuming insertion as shown in the above figure (A), please do not cut "Group 1" inserted as shown in the above figure (B). If cutting is performed as shown in the above figure (C), the definition of boundary becomes ambiguous, and internal mesh generation may not be performed normally.

**(2)    Between adjacent group faces**
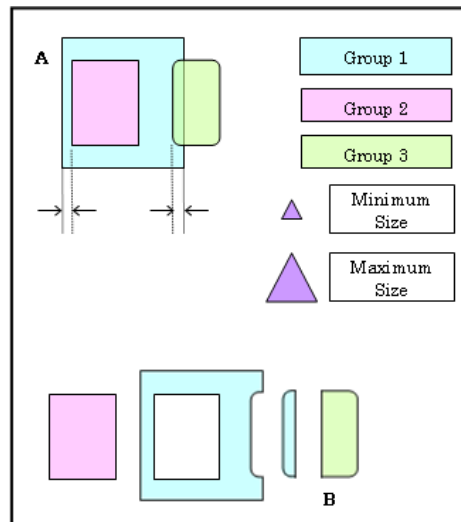


If there are faces that are very close to each other as shown in the above figure (A), mesh generation cannot be performed normally in internal mesh generation.

It seems to be easier to understand if we assume that the mesh generation is divided into groups as shown in the upper figure (B) in the case of the upper figure (A), with the characteristic of the internal mesh generation. (Nodes on a boundary surface are shared by both group).

Assuming mesh generation in the above figure (B), there is a part smaller than the "minimum size" of the "internal mesh setting," so there is a possibility that mesh generation cannot be performed normally.

When these problems occur, the following solutions are possible, but using the second option will lead to an increase in the number of meshes, so caution is necessary.

1) Change the composition of the surface mesh.

2) Reduce the minimum size.

**(3)    The existence of a disconnected group**
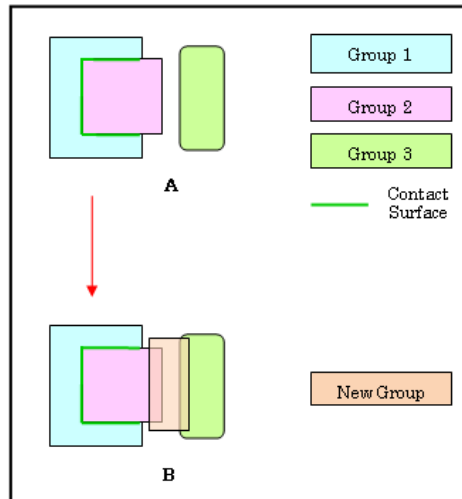


Aside from exceptions, do not make group settings that are completely separated as "Group 3" in the above figure (A). This is because even if internal meshes can be generated, such a model configuration is not allowed as an object to be analyzed. If "Group 3" is necessary, it is necessary to connect by ROI or import processing as shown in the above figure (B). (For example, even if only the femur and tibia are extracted, it will be as shown in the above figure (A), so it is necessary to set a connecting material simulating cartilage)

When performing mesh generation of extremely fine shapes and complicated shapes, completely separated groups may be generated when setting the material type. In such a case, it is necessary to make the discrete group an unused material.

However, in the following cases, the above figure (A) is permitted:

•   In the contact element of special material, the separated faces are defined as (primary/secondary face). Then, change the load type from "Load" to "Forced Displacement."

•   Give the initial velocity in the separated and constraint setting. For example, as shown in the figure on the right, in the case of analysis ware the ball strikes the ribs, the ball and the ribs are separated from each other, and calculation is performed by giving the initial velocity to the ball.
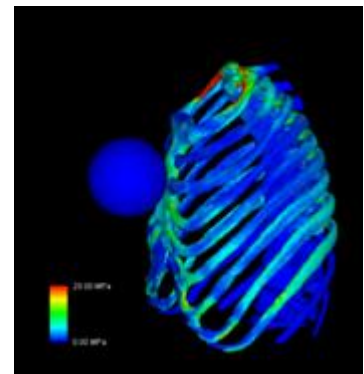
# Chapter 7 Material Sorting

In this chapter, we set various mesh conditions for analysis. Set the following items.

• Material Type (Solid/Shell/Special)

• Setting of Axis

• Coordinate Transformation
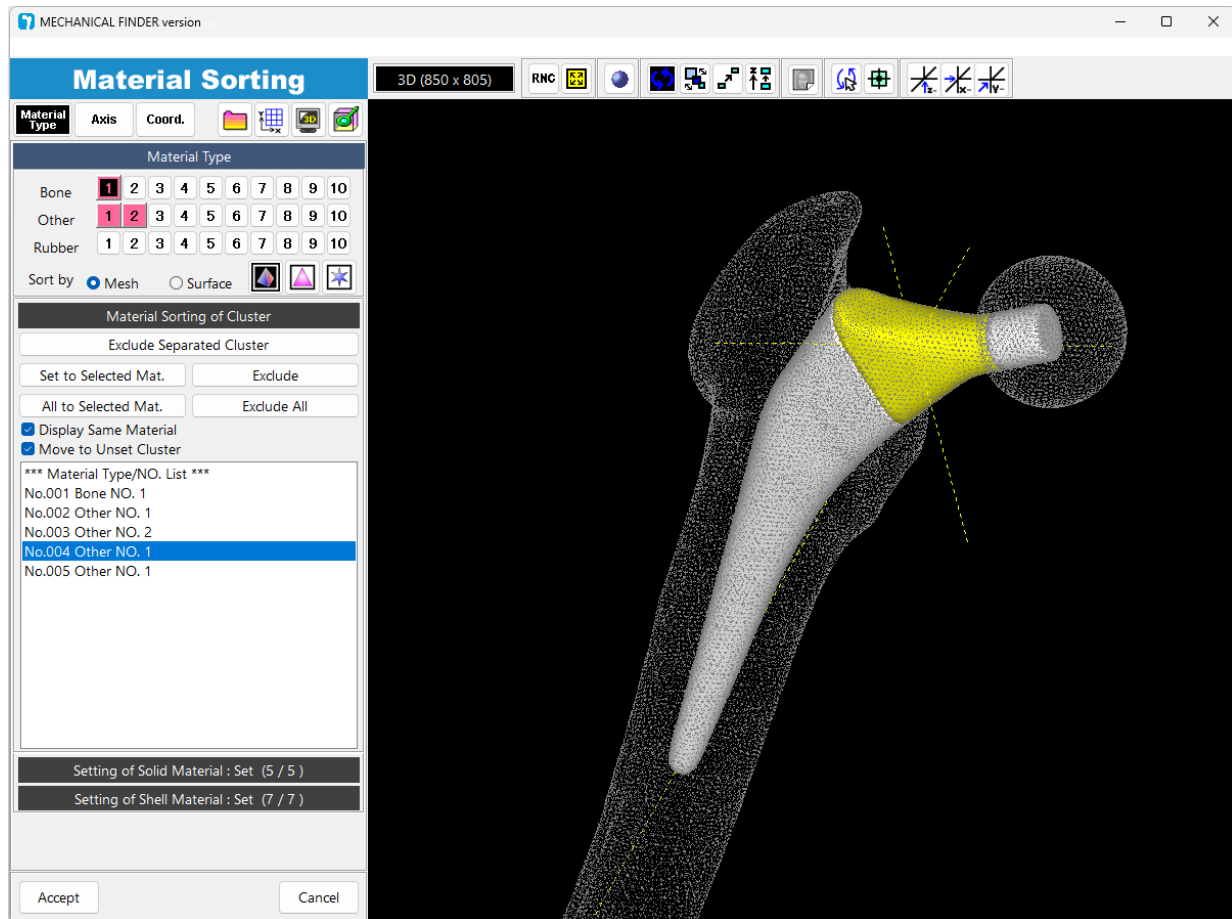


| Icon | Function |
|---|---|
| Material Type | Material Type |
| Axis | Setting of Axis |
| Coord. | Coordinate Transformation |
| (folder icon) | Data Information |
| (grid icon) | Surface Display |
| 3D | Viewer Settings |
| (CT icon) | CT Display |

## 7.1 Material Sorting

Material sorting by "Material Type" is the process of classifying and specifying which material is assigned to a mesh, divided into multiple parts after mesh generation is completed. At that time, it is possible to set whether to use a shell element based on the assigned material or to set special material (node separation/gap element/contact element). It is described below with an example.
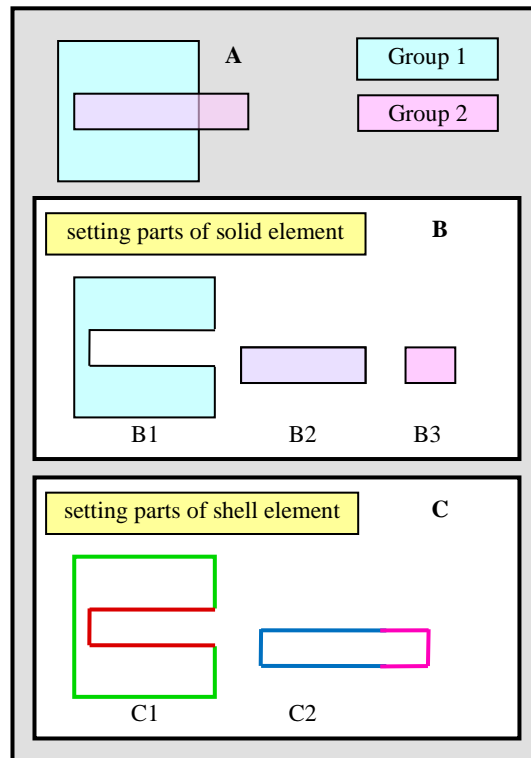


**[Explanation]**

- Assume that mesh generation is complete with "Group 2" inserted in "Group 1" as shown in the above figure (A).

- In "Material Sorting of Cluster" in "Material Sorting," it is necessary to set which material is to be set for each of the three groups as shown in the above figure (B). (In the case where the contact surface is close, or there are surfaces that overlap each other complicatedly, a lot of material designation may be necessary.)

- If you set "Material Sorting of Cluster" as two different materials of (B1) and (B2, B3) in the above figure, items in "Setting of Use of Shell" are as follows.

o C1: It is necessary to set the presence/absence of shell elements on [green line not touching anywhere] and [red line touching C2].

o C2: It is necessary to set the presence/absence of shell elements on [purple line not touching anywhere] and [blue line touching C1]. This varies depending on how the grouping of materials (bone materials 1 to 10, other materials 1 to 10), performed in "Material Sorting of Cluster," was set.

- Special material can be set to the area of the shell element above.

> * In "Material Sorting of Cluster," you can also set not to use it as the analysis mesh, as well as to which material it is assigned.

> * Shell element cannot be set for rubber material.

## 7.2 Special Material

The special material in this software states the following and is mainly the condition setting between materials (bone material/other material).

- Separate Node

- Gap Element

- Contact

- Truss Element

As an example, there is a mesh shape as shown below, and it is divided into a gray material and a green material. (Material type is already set.) We will explain these special materials by enlarging a part of the interface between the materials.

*Sample Shape*



(1)、(2)、(3)、(4)

(5)
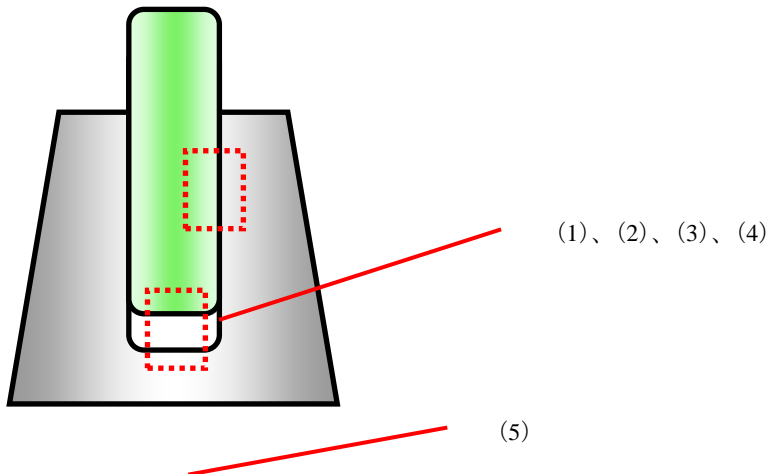
| (1) | No special material | When special materials are not used, nodes are shared between the materials, so the materials are bonded and will not separate. |
| --- | --- | --- |

| | |
|---|---|
| (2)　　　Separate Node | Node separation refers to separating shared nodes in a specified area. (Distance between nodes is 0.0 mm.) In this case, no force propagates to another separated node. It is possible to confirm that there is no force propagation either way by displaying the "Deformation Display" in the result.<br><br><span style="color:red">* If there are discrete elements and nodes other than when using contact elements described below, it may not be possible to analyze normally. Please be careful not to create such elements and nodes that are completely discrete when using node separation.</span> |
| (3)　　　Gap Element | The gap element is a spring element that can propagate force in the compression direction. In addition, by setting the sliding resistance, there is resistance force in the sliding direction, and friction force can be simulated. (Distance between nodes is 0.0 mm.) For details, see "Appendix 4.4 Used Element."<br><br>The gap element in this software was introduced to replace the contact element which is very time consuming. It is a complement to the contact element. |
| (4)　　　Setting of Contact | Contact elements are those that can analyze the contact condition in a specified area. Since it is a structure as shown on the left, force can propagate only in the compression direction. Specify the following two areas as one contact element group.<br><br>• Primary side area (red): Touched side<br><br>• Secondary side area (green): Touching side<br><br>For stable analysis, we recommend setting softer material to secondary side and harder material to primary side.<br><br>In this software, when nodes of the primary side and the secondary side are shared while setting the contact element, node separation is performed and the node is automatically set as <span style="color:red">single point constraint</span>. Single point constraint is having a pair of nodes connected (constrained) to each other only during the initial loop at analysis.<br><br><span style="color:red">* Analysis using contact elements can be very time consuming. Especially when destruction starts, it takes a long time and it can reach up to 20 times.</span> |
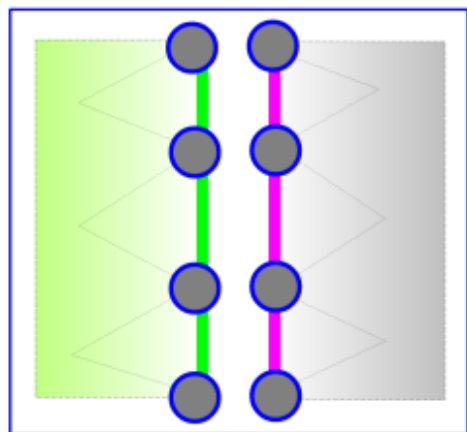
(5)        Setting of Truss Element
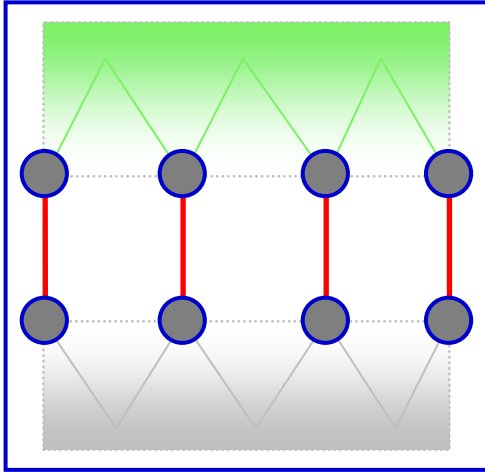


A truss element is a line element connecting distant nodes. Since it is possible to define the relationship between strain and tension in the truss element, it can generate resistance (tension) only in the tensile direction without resistance in the compression direction. The node has a pin structure and rotates freely. No bending resistance is generated.

\* The relationship between strain and tension can be defined nonlinearly. If the slope of strain-tension changes during analysis, there is a possibility that strong stress may be generated in the model due to sudden change in resistance. In order to solve the analysis stably, please consider an analysis condition that will suppress the occurrence of sudden stress, such as reducing the load step.

## 7.3 Operating method for special material area (EE)

In order to specify the area of special material, it is necessary to specify the boundary between materials inside the analysis element, following the procedure below.

1. Select the reference material type and number.

2. Select the boundary you want to set in the material number (the same selection method as for shell).

3. Set an arbitrary area by selection or mouse operation.

Here, we will explain how to designate the area of special material using sample data (project name: sample). (Operation on the area of special material is the same operation in any case of "Separate Node"/"Gap Element"/"Contact," <primary side/secondary side>.)

### (1) Select the reference material type and number.

First, choose which materials to set as special material on the surface. Selection is made by selecting the number of the bone material/other material. (The figure below assumes that bone material number 1 is selected.)
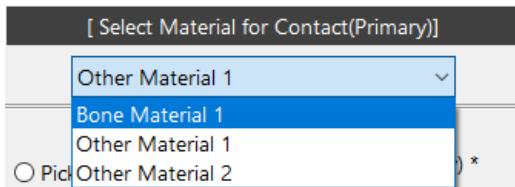


The point to note here is that there are two selection methods for the boundary surface where different material numbers are in contact. For example, to select the boundary of the yellow area in the figure below, there are the following selection methods.

- Bone material number 1 side of the interface between bone material number 1 [femur] and other material number 1 [stem]. (Bone material number 1 selected.)

- Other material number 1 side of the interface between bone material number 1 [femur] and other material number 1 [stem]. (Other material number 1 selected.)

In node separation and gap elements, since only one side (the side where nodes are separated) is specified, there is no difference in specifying either. However, since the primary side and the secondary side are need to be specified at the contact element, it is necessary to know which interface is specified and to which material number the specified boundary surface belongs.
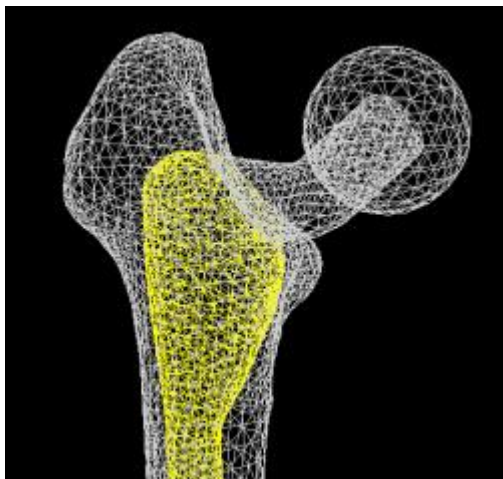


### (2) Select the boundary you want to set in the material number. (It is assumed that bone material number 1 is selected.)

For the selected material number, you will select the special material area, but if the area is the boundary surface inside the material, it will be difficult to specify. Therefore, here, the faces within the specified material number are grouped so that you can select the area that corresponds to the list.
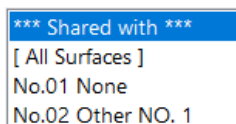
An example is grouping for sample data. By picking in the list, it changes to the following display.

| | |
|---|---|
| _([All Surfaces])_ | _(No.01 None)_ |
| _(No.02 Other No.1)_ | |
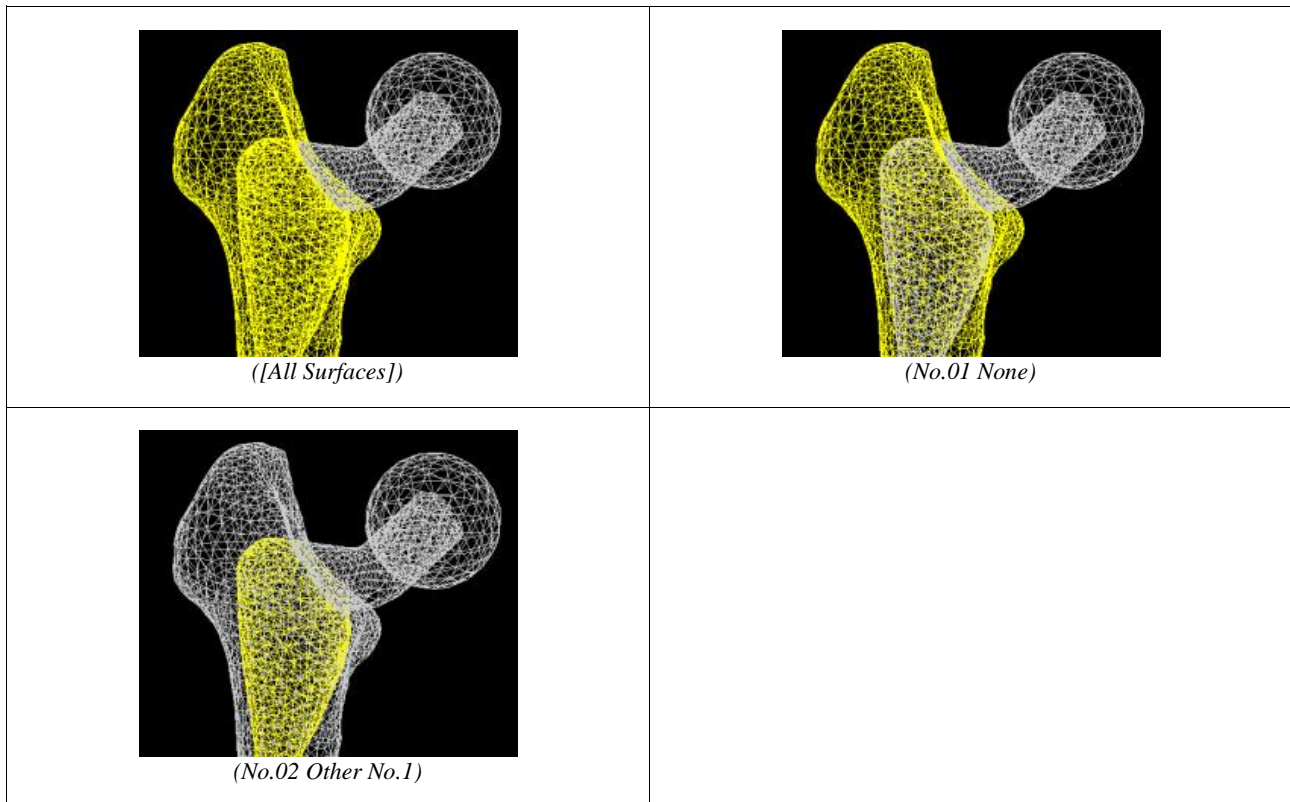
This yellow area is the area that corresponds to the designation, and other gray nodes and elements are not selected. By using this, you can easily specify the red area as below.
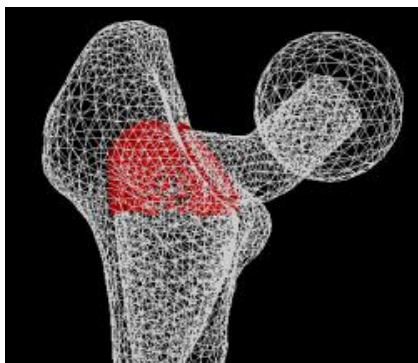


How to specify using a mouse and other methods are explained in the next section. In this case, please select "No.02 Display Part" from the list.

**(3) Set an arbitrary area by selection or mouse operation.**

The following menu explains how to select the desired area.
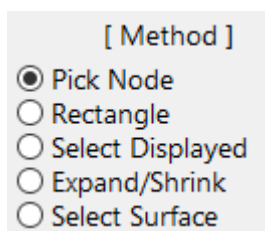


- **Pick Node**

Every time you right click on the displayed shape directly, add or delete the area.
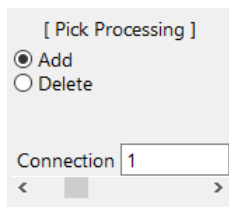
By changing the value of "Connection," you can change the influence range from the clicked node. At that time, only the area selected and displayed in yellow in 2. is added or deleted. Therefore, if the area displayed in yellow is only the inner surface, you can add or delete on the inner surface, ignoring the outside of the shape with this pick operation. "Connection" is fixed at 0 when setting truss elements.
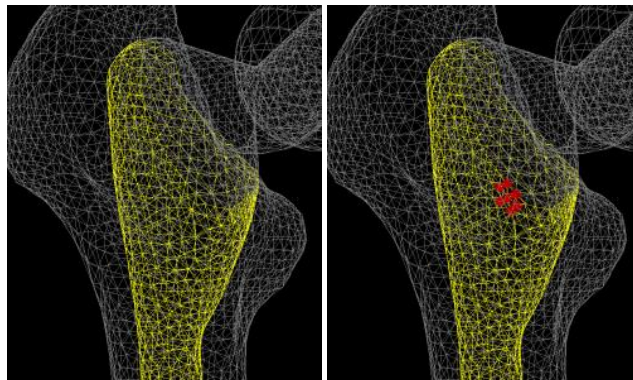


*(Right-picked place is selected)*

- **Rectangle**

This is a method by which you can add, replace, or delete an area by specifying the range by right-dragging with the mouse on the displayed shape, then pressing the "Add"/"Replace"/"Delete" button. From the inside of the square, from the front to the back, is the selection target, but only the area selected and displayed in yellow in 2. is selected.
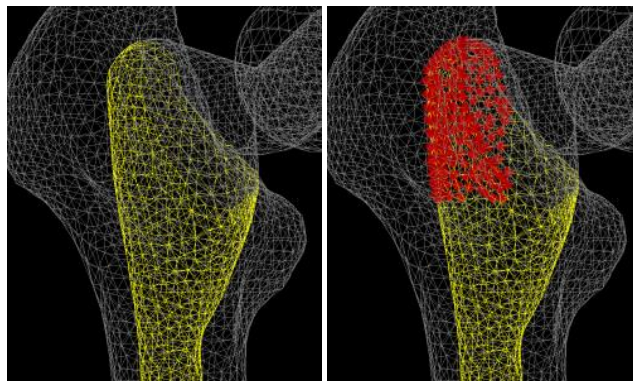


*(Rectangle range right-dragged with mouse is selected)*

- **Select Displayed**

By pressing the "Add"/"Replace"/"Delete" button, you can add, replace, or delete all the areas selected in 2. and displayed in yellow.
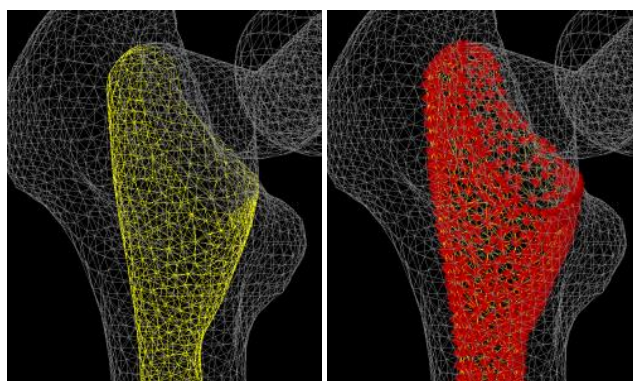
*(All yellow regions are selected)*

- **Expand/Shrink**

  By pressing the "Expand"/"Shrink" button, you can increase or decrease the currently set area by one element each time the button is pressed. This is a convenient method when you want to expand or shrink the contact surface slightly. (Expansion/shrinkage is effective only within the area of yellow.)
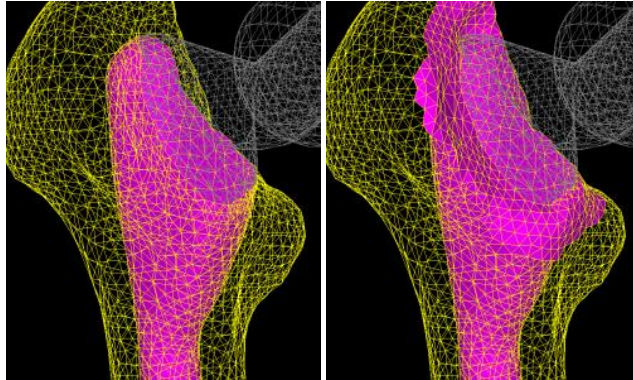


*(Selected area when "Expand" is executed multiple times)*

- **Select Surface**

  This function is available only when setting the contact element's secondary face. You can set the same secondary or primary surface by selecting the contact element already defined from the list of "Select Defined Surface." This allows you to define contact conditions such that the three or more surfaces are in contact with each other.

  "No.xx" of list means the number of Special Material, "primary/secondary" means that defined contact surface is either primary surface or secondary surface, the number means number of triangles.



Select Defined Surface

**Select Surface**
No.01 primary=7332
No.01 secondary=7156
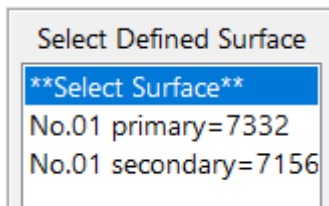
# Chapter 8 Material Property

In this step we give material properties to mesh elements. We also set the shell configuration (material properties and thickness).
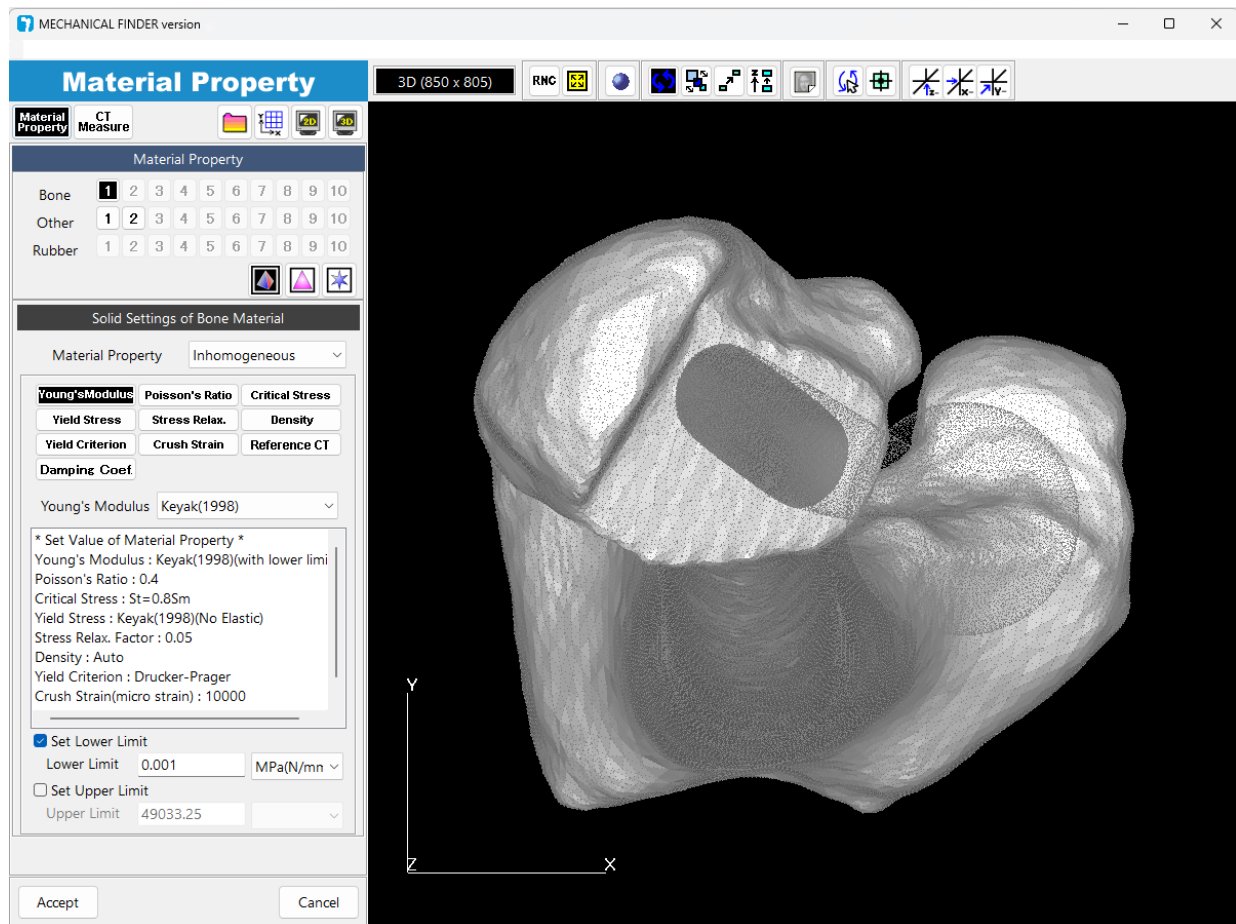


| Icons | Functions |
|---|---|
| Material Property | Material Property |
| CT Measure | Image Measurement |
| (folder icon) | Data Information |
| (surface display icon) | Surface Display |
| 2D  3D | Viewer Settings |

## 8.1 Material Property

Please refer to "Appendix 3.1 About Material Properties of Inhomogeneous Material" for predefined setting items of each material property.

- **Young's Modulus**

  Select the setting method of Young's modulus [kgf/mm$^2$] from the pulldown.



| User-established | Apply the user-established conversion formula to each element. |
|---|---|
| Keyak (1998) | Apply the formula proposed by Keyak for each element. |
| Carter (1977) | Apply the formula proposed by Carter for each element. |
| Keller (1994) Vertebrae | Apply the formula for vertebrae proposed by Keller for each element. |
| Keller (1994) Combined | Apply the formula for compound bone proposed by Keller for each element. |
| Suzuki (2020) Callus | Apply the formula for callus proposed by Keller for each element. |



  In the Extended Edition, you can set the lower limit and upper limit in converting to Young's modulus.

- **Poisson's Ratio**

  Select the method of setting Poisson's ratio from the pulldown.



| User-established | Apply the user-established conversion formula to each element. |
|---|---|
| 0.49–0.0 | Apply the selected numerical value to each element. |
| Minamisawa (1981) | Apply the formula proposed by Minamisawa for each element. |

- **Critical Stress**

  Select the formula for the critical stress [kgf/mm$^2$] from the pulldown.



  Apply the selected formula for each element.

- **Yield Stress**

Select the method of setting the yield stress [kgf/mm$^2$] from the pulldown.



| User-established | Apply the user-established conversion formula to each element. |
|---|---|
| Keyak (1998) | Apply the formula proposed by Keyak for each element. |
| Carter (1977) | Apply the formula proposed by Carter for each element. |
| Keller (1994) Vertebrae | Apply the formula for vertebrae proposed by Keller for each element. |
| Keller (1994) Combined | Apply the formula for compound bone proposed by Keller for each element. |
| Suzuki (2020) Callus | Apply the formula for callus proposed by Keller for each element. |



For the yield stress, there is also a toggle in the above figure (in OFF state at the beginning); you can choose whether to treat elements with a density value of 200 mg/cm$^3$ or less as an elastic element.

- **Stress Relaxation Factor**

Select the setting value of stress relaxation factor from pull down.



| User-established | Apply the user-established conversion formula to each element. |
|---|---|
| 0.1 | Apply 0.1. (See "Appendix 3 Material Property") |
| 0.05 | Apply 0.05. (See "Appendix 3 Material Property") |
| Perfect Elastoplasticity | Apply a value close to zero ($1.0 \times 10^{-20}$). |

- **Density**

Select conversion formula for density from pull down.



| | |
|---|---|
| Auto | Density is automatically calculated from CT value for each element.<br>• When phantom is not set: Apply Standard Conversion Equation.<br>• When setting phantom: Apply the phantom calibrated conversion formula. |
| Arbitrary | Apply the user-established conversion formula. When selected, the following setting items are displayed.<br><br>Specify the conversion formula with "constant a" and "constant b." |

In converting to density, it is possible to set the lower limit and upper limit of the density value. (Both upper limit and lower limit are OFF in the initial state.) The upper limit value is mainly used to suppress a density increase influenced by an artifact.

- **Reference CT**

  Specify whether to set the reference position of the CT image to calculate the density to the current coordinate position of the bone material or to the original position of the imported shape that was rotated and moved. If the position of the bone material is rotated and moved from the position in the CT image, select the import position here.



| Initial Coordinates | Refer to the CT image with the current coordinates of bone material. |
|---|---|
| Imported Coordinates | If the current coordinates of the bone material have been changed from their original position, select the imported shape that was moved/rotated from the following list to calculate the original position.  |

- **Damping Coefficient**



  Enter the value of the structural damping coefficient to be used when performing dynamic analysis.

## 8.2 Operation method in Image Measurement

In the shell material setting, you can set the "CT Value for Shell" and "Shell Thickness (mm)" by observing the plotted graph. For the graph drawing method, three kinds of chart types shown below can be selected.
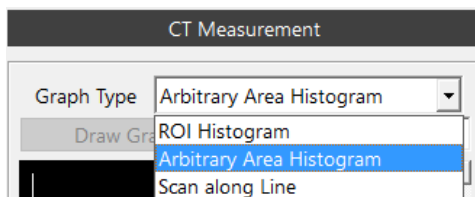


The operation method for the three types of graph drawing is shown.

**(1)  ROI histogram**

This graph shows the CT value of the part extracted in the ROI extraction work as a histogram. The frequency of CT values in all the slices from which the ROI was extracted is displayed as a graph.

A.    Select "ROI Histogram" in "Graph Type."

B.    Press the "Draw Graph" button.

C.    Statistics will be displayed on the graph.
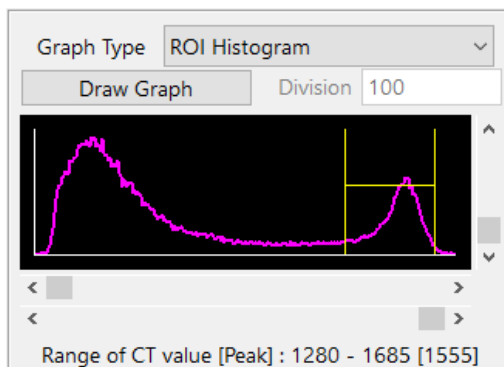


D.    By left-dragging on the graph, the minimum and maximum values of the CT value within the drag range and the peak CT value within the range are numerically displayed.

E.    Press the "Stock Value" button to stock the value.

* If you select this type of graph, the display in the viewer switches and the original CT image within the area from which the ROI was extracted is displayed. If there is no part extracted as a ROI in the displayed slice, nothing is displayed in the display viewer, so please change the slice number. You can confirm that only the original CT image in the region from which the ROI was extracted is displayed.
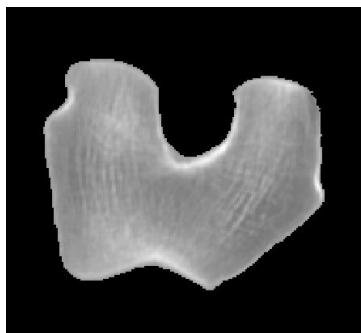


**(2)  Arbitrary Area Histogram**

This graph displays the histogram of the area by specifying the area in the CT original image slice displayed in the viewer. The frequency of the CT value in the area specified on the displayed slice is displayed as a graph.

A.    Select "Arbitrary Area Histogram" in "Graph Type."

B.  Perform right-dragging on the viewer and specify the area.



C.  Statistics will be displayed in the graph.
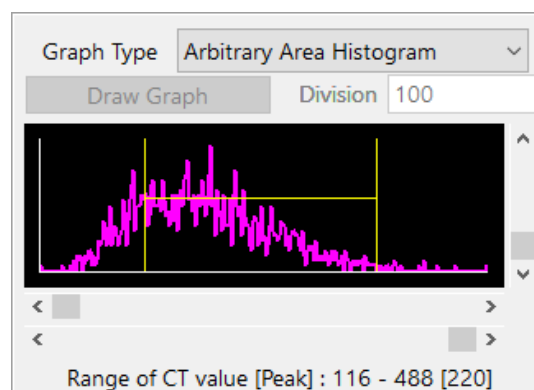


D.  By left-dragging on the graph, the minimum and maximum values of the CT value within the drag range and the peak CT value within the range are numerically displayed.

E.  Press the "Stock Value" button to stock the value.

**(3)  Scan along Line**

In this graph, by specifying a straight line within the slice of the CT original image displayed in the viewer, the density value along the line is displayed.

A.  Select "Scan along Line" in "Graph Type."

B.  Perform right-dragging on the viewer and specify a straight line.
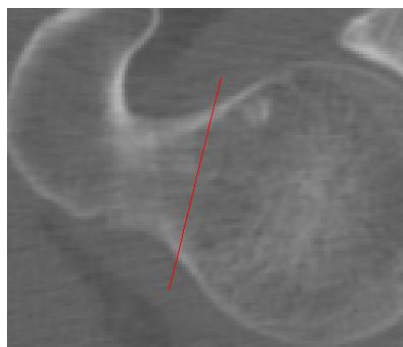


C.  The density value along the line is displayed as a graph.

Length(mm) : 3.4099
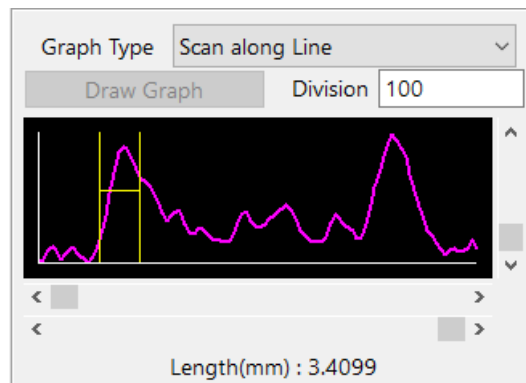
D. By left-dragging on the graph, the length of the drag range is displayed numerically.

E. Press the "Stock Value" button to stock the value.

The frequency of sampling density values on a straight line can be subdivided by increasing "division number."

## 8.3 Method for setting material properties of special materials

The "Special Material" button  switches the screen to set the material properties of the special material. Here you can set the material properties of gap elements, contact elements, and truss elements.



**(1) Gap Element**

Gap element parameters are set automatically.



| | When [solver V2] used |
|---|---|
| Method of setting parameter of gap element | • Spring Stiffness: Automatically determined internally.<br><br>• Shear Spring Coefficient: Fixed to 0. |

**(2) Contact Element**

By selecting a contact element from the list, you can set the "Friction Coefficient" and "Relaxation Coefficient."



| | |
|---|---|
| Method of setting parameter of contact element | It has the following parameters.<br><br>• Friction Coefficient: Enter a numerical value between 0 and 1 as the dynamic/static friction coefficient.<br><br>• Relaxation Coefficient: It is a parameter for stability of calculation. If the calculation is difficult to converge by introducing friction, enter a value greater than 1.<br><br>For details, refer to the explanation of the friction model in "Appendix 4.7 Contact Treatment" |

**(3) Truss Element**

When you select a truss element from the list, you can set the "Truss Element Tension."



| Method of setting parameter of truss element | It has the following parameters.<br><br>Truss Element Group: Select the predefined truss element group.<br><br>Truss Element Tension: Select the predefined "strain-tension table" from the list. For the definition method of "strain-tension table," please refer to "Appendix 3.2 Definition Method of Material Properties of Truss Element."<br><br>Magnification of Tension: Set the multiplier of truss element tension.<br><br>* Assign the definition of one "strain-tension table" to one truss element group. Tension multiplied [Magnification of Tension / the total number of truss elements in the group] will be assigned to each truss. Also, see "Appendix 3.3 Allocation of Material Property to the Truss Element." |
| --- | --- |

# Chapter 9 Boundary Condition

In this chapter, set the load condition, constraint condition, initial velocity setting (optional).
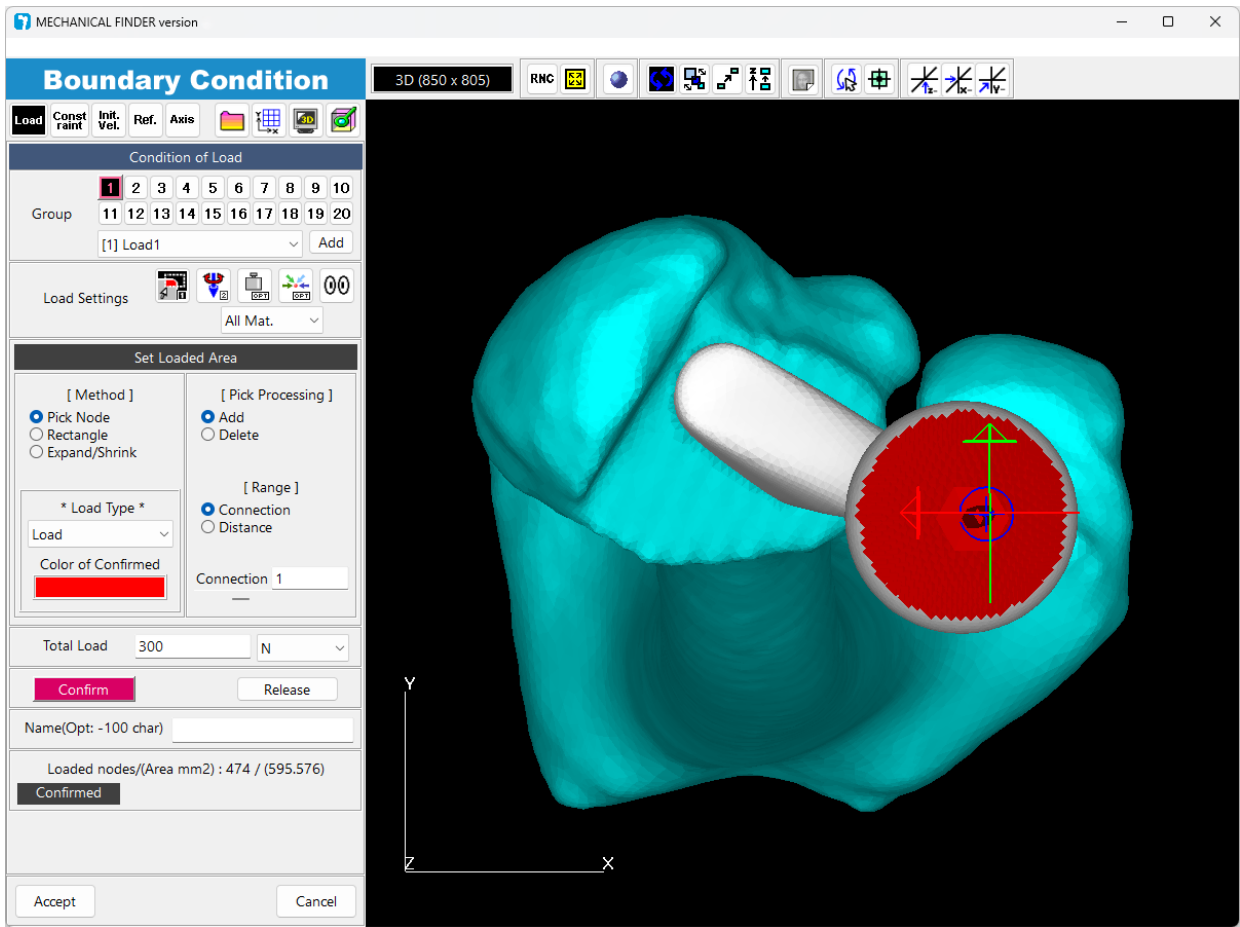


| Icons | Functions |
|---|---|
| Load | Condition of Load |
| Const raint | Condition of Constraint |
| Init. Vel. | Setting of Initial Velocity |
| Ref. | Setting by Reference |
| Axis | Setting of Axis |
| (folder icon) | Data Information |
| (grid icon) | Surface Display |
| (3D icon) | Viewer Settings |
| (CT icon) | CT Display |

## 9.1 Setting by Reference

The "Setting by Reference" is a function to automatically set the load/constraint condition, to read the conditions of other projects, and then to reflect the conditions on the current project. It is mainly aimed at "Reproduce load/constraint condition" and "Simplify load/constraint setting work."

The following three methods are prepared for reference setting. We will describe applicable conditions and application examples for each.

**Pattern calculation (only femur)**

**Condition**

- The part is a femur.

- The axis is set by "Femoral Method."

**Application Example**

- When you want to perform processing by giving the same load/constraint condition to multiple subject data.

- When you want to reproduce a complicated condition such as falling.

**Refer Project**

**Condition**

- There is a project with approximate load/constraints set.

**Application Example**

- When you want to compare the states (of the same person) during the healing process.

- When you want to perform processing by giving the same load/constraint condition to multiple subject data.

**Refer Coordinate of Analyzed Project**

**Condition**

- A model that has the same mesh geometry as that of a working project and has been analyzed must exist.

**Application Example**

- When you want to inherit the previous result (only displacement) and perform analysis.

## 9.2 Load Type/Constraint Direction/Initial Velocity

We explain the difference between the four types of load ("Load," "Load + Moment," "Forced Displacement," "Pressure"), the types of constraint direction (X, Y, Z, θx, θy, θz) and the initial velocity. Please select the optimum setting as the desired analysis condition.

**(1) Load Type**

- **Load**
  "Load" is the load in the linear direction (arrow direction).
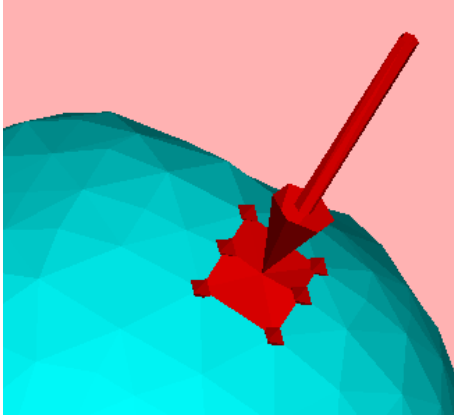


*(Shown in red when load is fixed)*

The area is determined by selecting the nodes in the area where the translational load is to be applied. (7 points are selected in the above figure.) For example, if you set "Total Load" to 70 kgf for the area in the above figure, if there is no weight setting, it will be set to apply a translational load of 10 kgf per node.

- **Load + Moment**
  "Load + Moment" is the moment given to a point on a shell element.
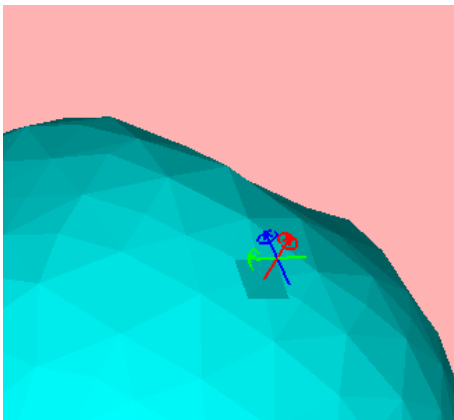


*(Shown in light blue when load is fixed)*

The position is determined by specifying only one node in the area to apply moment. (See example in the above figure.) For example, in the state shown above, set the moment value (N·mm or kgf·mm) of θx (red arrow) · θy (green arrow) · θz (blue arrow). When setting the moment, you can also specify the translational load on the node.
Pay attention that the moment around an axis perpendicular to shell element is invalid.

- **Forced Displacement**

"Forced Displacement" is the displacement (mm) in the linear direction (arrow direction).
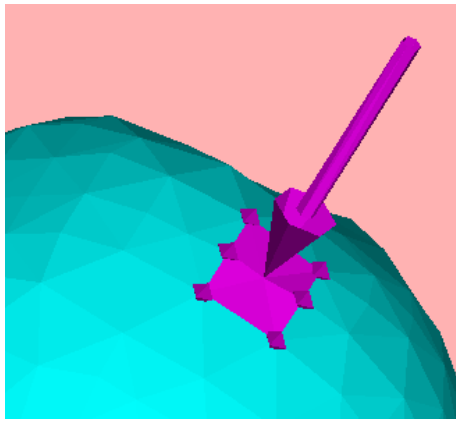

*(Shown in purple when load is fixed)*

The area is decided by selecting the nodes in the area where the forced displacement is to be given. (7 points are selected in the above figure.) For example, if 3 mm is set as "Forced Displacement (mm)" in the state shown above, analysis is performed under the condition that all 7 nodes are equivalently displaced by 3 mm.

- **Pressure**

"Pressure" is the load applied in the direction perpendicular to the surface.
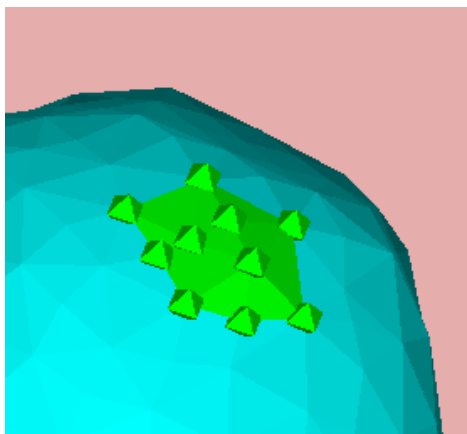

*(Shown in green when load is fixed)*

The area is determined by selecting the nodes in the area where you want to apply pressure. (Example in the above figure.) For example, when the "Pressure" is set to 10 MPa in the state shown in the figure, according to the area covered by each node, the force applied to the node is calculated in order to be equal pressure, and analysis is performed under the load condition in the direction perpendicular to the surface.
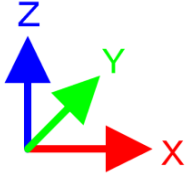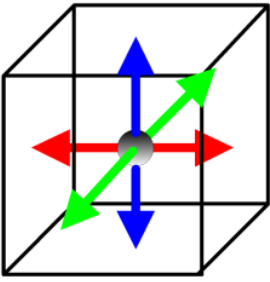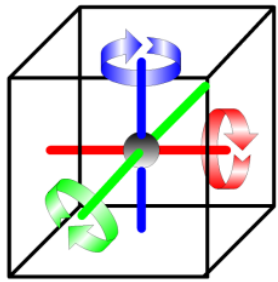
**(2) Constraint Direction**

For the constraint direction, select the direction you want to constrain from the 6 directions of X, Y, Z, θx, θy, and θz. The direction of constraint is as follows and the checked direction is constrained.

X, Y, Z: Set constraint for movement in each axis direction.
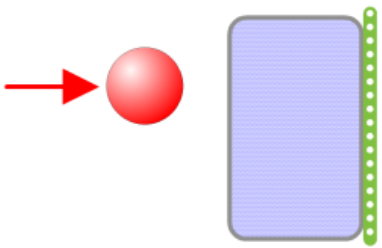
θx, θy, θz: Set constraint on the movement of rotation around each axis.

| In coordinates (physical coordinates) as shown below. | Constraint direction X, Y, Z | Constraint Direction θx, θy, θz |
|---|---|---|
|  |  |  |

**(3)    Initial Velocity**

Initial velocity can be given in units of clusters (groups) shared by nodes. You can set it by giving the initial velocity direction (X, Y, Z) and magnitude to any cluster. If you set the initial speed, you can analyze without setting "Condition of Load" and "Condition of Constraint." (When initial velocity is not set, "Condition of Load" and "Condition of Constraint" are mandatory.)



| | |
|---|---|
|  | • Reference 1 [Example of giving initial speed to the cluster and striking another fixed cluster]<br>The "load condition" is not set under the condition of the left figure. However, it is necessary to set "contact element" in the area to be hit. |
|  | • Reference 2 [Example of hitting clusters by giving initial speed]<br>In the condition on the left, "load condition" and "constraint condition" are not set. However, it is necessary to set "contact element" in the area to be hit. |

## 9.3 Load direction

This section describes the load direction.
The setting method for load direction is selected from XYZ designation, Polar Coordinate (Selected Axis) designation, Polar Coordinate (3-axis) designation, and Two-nodes direction designation.

- **XYZ designation**
  The direction is set by designating components of X, Y, and Z axes for unit vector.

- **Polar coordinate (Selected axis) designation**
  The set axis is selected, and the angle to the axis is designated to set the direction.

- **Polar coordinate (3-axis) designation**
  The direction is set by designating negative/positive angle to X, Y, and Z axes.

- **Two-nodes direction designation**
  Two nodes are designated to set the direction connecting them as a load direction. The load direction keeps parallel to the direction connecting two nodes under analysis.

  | Load Direction 1 | Load Direction 2 |
  |---|---|
  | [Set Direction] | Start Node 17870 |
  | ○ XYZ | |
  | ○ Selected Axis | End Node 16458 |
  | ○ Polar Coordinate | |
  | ◉ Two Nodes | |
  | (Direction changes with movement of the Nodes) | Set two nodes |
  | | Clear mode |
  | ☐ Disp. Tensile Load | |

# Chapter 10 Analysis

Analyze with mesh geometry, material property, boundary condition set. After setting the calculation condition, pressing "OK" will execute analysis processing.
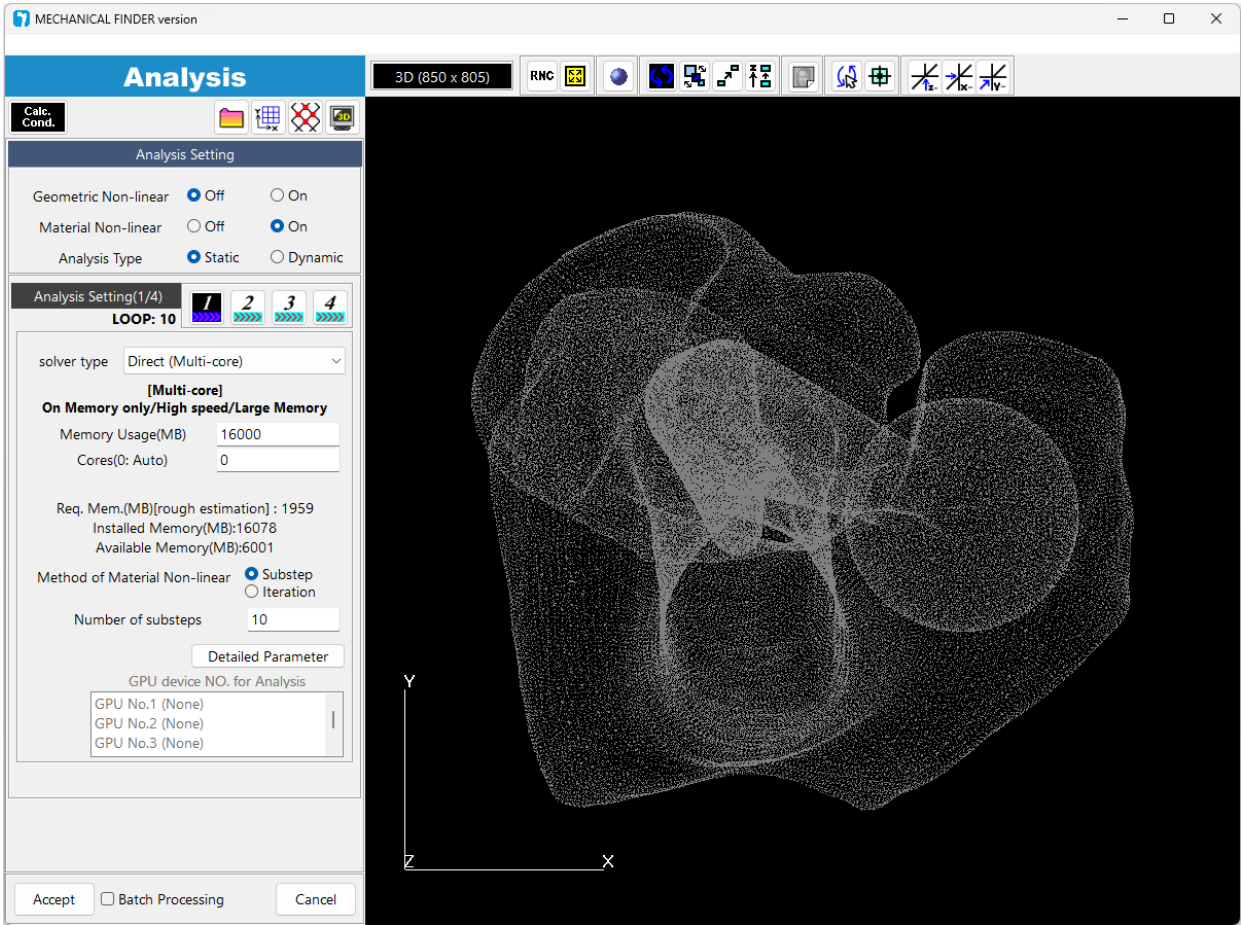


| Icons | Functions |
|---|---|
| Calc. Cond. | Analysis Setting |
| | Data Information |
| | Surface Display |
| | Constraint Display |
| | Viewer Settings |

## 10.1 Analysis Options

Three kinds of analysis options are available for solver installed in this software.



Since each analysis option has an ON/OFF switch, we can make various analysis by combining these.

- **Geometric Non-linear**
  o OFF: Analyze based on infinitesimal deformation theory.
  o ON: Analyze considering large deformation.

- **Material Non-linear**
  o OFF: Compression failure (yield/crush) and tensile failure are not considered.
  o ON: Consider compression failure and tensile failure.

- **Analysis Type**
  o Static: Perform static analysis.
  o Dynamic: Perform dynamic analysis.

Note: since MECHANICAL FINDER Version13, Solver V1 of the previous version has not been able to be used.

## 10.2 Analysis Setting

This section explains how to set the calculation conditions.



| Items | Detail |
|---|---|
| Solver type | Select the solver type to use for analysis. There are almost no differences in results in either method. Please refer to "Appendix 5.5 About Analysis" for the difference in processing method.<br><br>• Direct (Multi-core)<br><br>• Direct (Single-core)<br><br>• Iterative (CG-method)<br><br>• Iterative (CG by GPU) |
| Calculation Parameters | • Memory Usage (MB): The maximum memory that you allow solver to use for analysis. The unit is megabyte. Please refer to "Recommended amount" or "Required amount" below.<br><br>• Cores: Specify the number of parallel calculations in "Direct (Multi-core)."<br><br>• Method of Material Non-linear: Select calculation method after an element fails.<br><br>• Number of substeps: In case select "Substep", enter the divided number of load steps after an element fails.<br><br>• Iteration of Failure Judge: In case select "Iteration", enter the number of iterative calculations for convergence of the unbalance force after an element fails. |
| Pre-conditioning | Select pre-conditioning method of CG method. Pre-conditioning affects the computational performance of the CG method. If it takes time to analyze or cannot be analyzed, there is a possibility of improvement by changing the pre-conditioning. |

| | |
|---|---|
| α Value of Drucker-Prager Yield Criterion | The Drucker-Prager yield criterion, which takes hydrostatic effect into account, is a factor related to material curing under multiaxial compression. The higher the value, the less likely the material to yield under multiaxial compression. When it is 0, it is equal to Von-Mises yield criterion without considering the hydrostatic pressure effect. |
| Tolerance | • Enter the tolerance in convergence judgment of the translation component and rotation component of the unbalance force. |
| Max Iteration (Unbalanced Force) | • It becomes active when "Auto Iter Limit for Convergence of Unbalanced Force" is OFF. Enter the upper limit of the number of iterations to converge the unbalanced force in one load step. We recommend that you set a large value for this setting (computation time will increase) because the unbalanced force will hardly converge when analyzing the model defining the contact surface and the gap element. <br><br> • When "Auto Iter Limit for Convergence of Unbalanced Force" is ON, this option is set as follows. This option is set to 1 if "Prediction of Plastic Failure" is performed and contact surface and gap elements are not defined. Otherwise, the default value is assigned. |
| Max Iteration (Contact) | Enter the upper limit of the number of iterations to calculate until the contact state does not change in one load step. As the contact surface is wider (more nodes are included), the contact state is less likely to converge, so in that case we recommend setting a large value for this option (calculation time will increase). |
| Deceleration Coefficient of Friction Force | The ratio of contact force in the previous step included in the contact force for friction force calculation gets bigger, if this value < 1. The contact condition is converged easier because the change in friction force as steps is suppressed. |
| Batch Processing | When it is OFF, analysis processing is performed immediately. When it is ON, you can perform analysis processing collectively even after the completion of this project using the following. <br><br> "14.5 Batch Program (Mesh /Analysis)" <br><br> "14.6 Remote Batch Program" <br><br> In that case, there is no need to run this software. |

## 10.3 Step Setting

Here, we explain how to set the "Step Setting."

**When static analysis is selected**

**Analysis Setting**

| | | |
|---|---|---|
| Geometric Non-linear | ● Off | ○ On |
| Material Non-linear | ○ Off | ● On |
| Analysis Type | ● Static | ○ Dynamic |

Step Setting(2/4)
LOOP: 10    1  2  3  4

Total Analysis time - (fixed 0.0-1.0s)

Steps    10

Apply Steps

No.01 t=0.100 div=1
No.02 t=0.200 div=1
No.03 t=0.300 div=1
No.04 t=0.400 div=1
No.05 t=0.500 div=1
No.06 t=0.600 div=1
No.07 t=0.700 div=1
No.08 t=0.800 div=1

Time    0.100    LOOP:10
Sub Steps    1    Apply to All

Apply Sub Steps

Apply Step Setting

Output Interval 2    step    OUT:5

**When dynamic analysis is selected**

Total Analysis time(s)    0.05000    s

Steps    10

Apply Analysis time and Steps

No.01 t=0.00500 div=1
No.02 t=0.01000 div=1
No.03 t=0.01500 div=1
No.04 t=0.02000 div=1
No.05 t=0.02500 div=1
No.06 t=0.03000 div=1
No.07 t=0.03500 div=1
No.08 t=0.04000 div=1

Time    0.00500    LOOP:10
Sub Steps    1    Apply to All

Apply Sub Steps

Apply Step Setting

Output Interval 1    step    OUT:10

| Items | Detail |
|---|---|
| Time condition setting | Here we set the nonlinear time condition.<br><br>• Total Analysis Time<br><br>  Enter the total time for dynamic analysis. You can input only when dynamic analysis is selected.<br><br>• Steps<br><br>  Set the number of points to change the load within the calculation time. The number of points is 1 to 10, which is common for all load groups.<br><br>• Time / Sub Steps<br><br>  When selecting an item from the list, the numerical value is reflected in the input column. You can change the length of time of the selected step and the number of divisions within the selected step, and it will be reflected in the list with the "Apply Sub Steps" button.<br><br>  When you press the "Apply to All" button, the number of divisions in the step entered will be reflected for all steps.<br><br>• Apply Step Setting<br><br>  Set the set time condition list to load groups.<br><br>• Output Interval<br><br>  Set the interval of outputting the analysis result file to step/second. The step and second are switched by selecting the unit list. |

* The number of steps is displayed as follows.

TOTAL: Total number of steps calculated from the number of steps and the numbers of sub steps.

OUT: The number of steps in which the result is actually output

* It is reflected by pressing "Apply Step Setting" button.

  The load value is set to linearly increase for each step.

  If it is reflected, the OUT item is displayed in black, and if it is not reflected, the OUT item is displayed in orange.

10.3.1 Output interval

The step for which the analysis result is output is set to the interval of output steps or the interval of output time.
The outline is described as follows.

- **When specifying the interval of output steps:**
    When "step" is selected for the output interval, the output is performed per interval of the step specified. If the interval of output steps is "1," the result is output at each step, and if it is "2," the result is output at an interval of one step. In either case, however, the result is always output at the final step.

**Interval of output steps = 1**

Load (N)

Time (sec)

Result is output at each step

**Interval of output steps = 2**

Load (N)

Time (sec)

Result is output at an interval of one step

(It is output at the final step)

●: Result is output
○: Result is not output
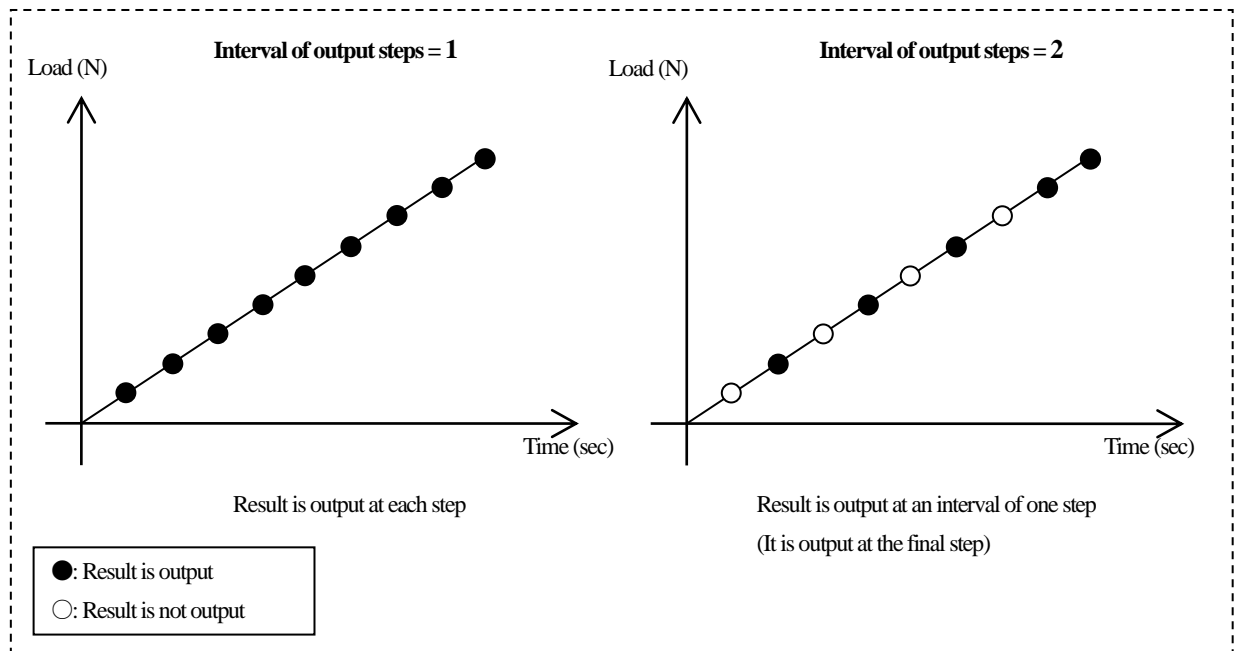
- **Specifying the interval of output time to output results at an equal interval**
    When multiple steps are specified in multiple intervals, the analysis steps may be at unequal intervals. In this case, selecting "sec" for the output interval and specifying the interval of output time may allow the output to be at an equal interval.

Interval of output steps = 2

Load (N)

Time (sec)

Output time is at unequal interval

Interval of output time = 0.2 sec

Load (N)

0.2 sec

Time (sec)

Output time is at equal interval

●: Result is output
○: Result is not output
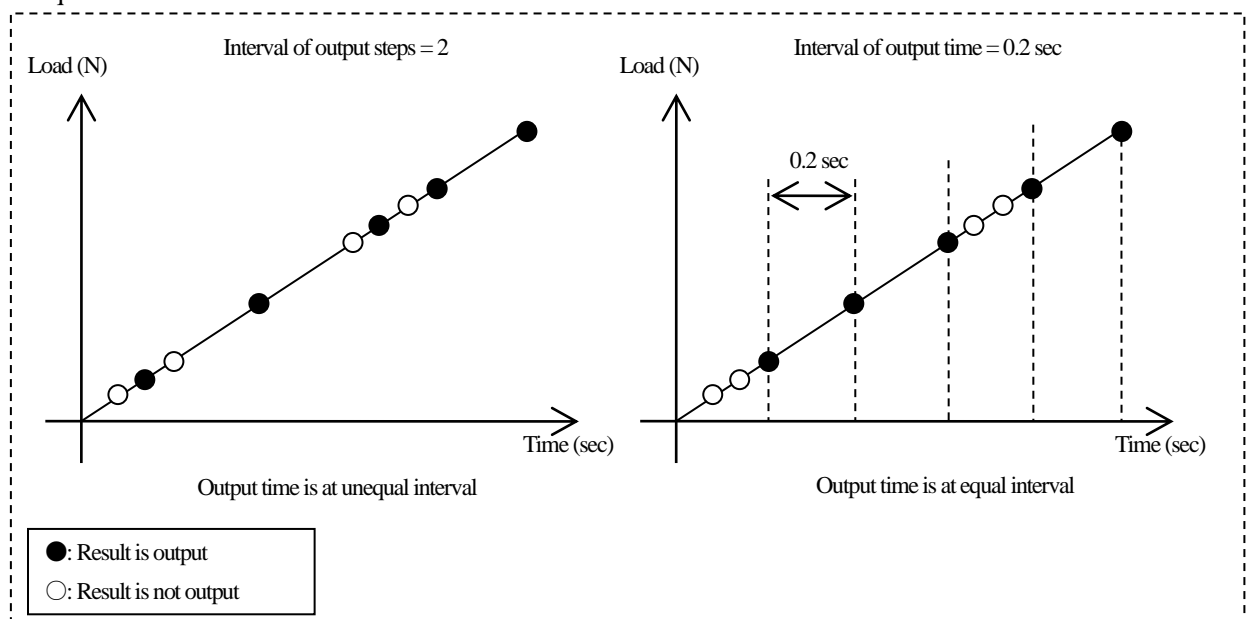
- **Output when the interval of output time is not equal to the time of analysis step**

  When the output time does not correspond to the analysis step, the output is performed for the analysis step that exceeds each output time for the first time. In this case, it is not assured that the output time is at an equal interval.



Interval of output time = 0.2 sec

●: Result is output
○: Result is not output

Analysis steps exceeding output time are output
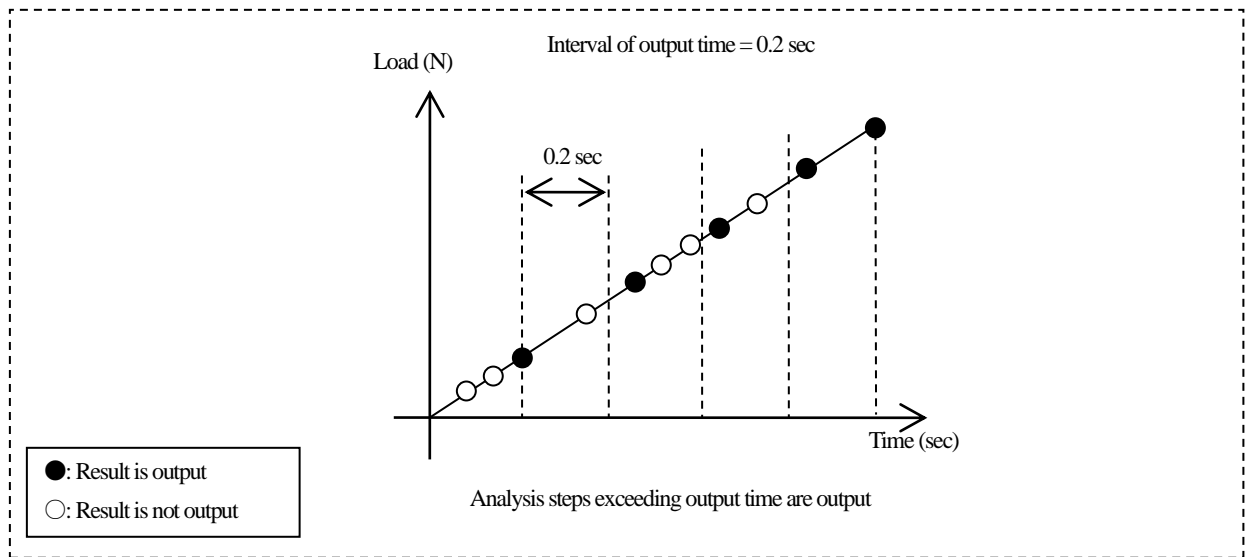
## 10.4 Load Settings

This section explains how to set the load per step. If you want to load nonlinearly for each step rather than equally dividing the load value by the number of steps, please set the nonlinear load here.
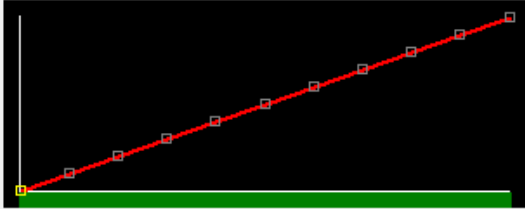


| Items | Detail |
|---|---|
| Group | Select the load group to change the load. When multiple load groups are set, please set for each load group. |
| Step Load Setting | This setting is linearly loaded by default. You can select a step on the displayed graph and change the load value by left-dragging with the mouse. In addition, you can change the step picked on the displayed graph by entering a numerical value.<br><br>*1<br><br>For the load value, "Load" and "Moment" can be set respectively. In the case of a "Load," you can change the maximum translational load value of |

105

| | |
|---|---|
| | the designated load group by changing the value of "Maximum Load" and pressing the "Change" button. "Load in Step" is also processed in the same way. To apply the changed load values to the specified load group, press the "Apply load in each step of this group" button. |
| Load No. | If you want to change the load direction in the selected step, select it here. However, the "Add Load Direction" setting must be performed when load condition is set. Additional numbers set in the "Add Load Direction" setting are displayed here, so please select a desired load direction. |

* Please be careful that the load value in a step is not reflected unless "Apply load in each step of this group" button is pushed. Also, if multiple load groups are set, please do the same for other groups as well.

*1. From version 6.1 it is possible to give smaller loads than the previous steps.
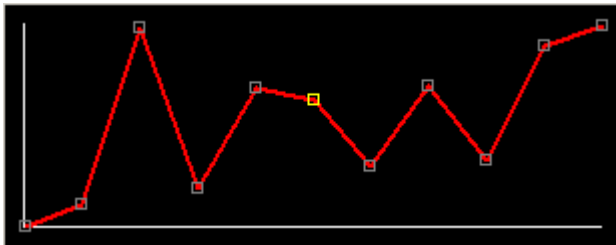
## 10.5 In-Out Setting

This section explains how to set the "In-Out Setting." During "Prediction of Plastic Failure," the number of failed elements is added as a condition to end analysis. In addition, we can inherit the stress state from the previous analysis result.



| Items | Detail |
|---|---|
| Inheriting analysis result | Starting with version 6.2, it is now possible to inherit the stress state from previous analysis results. Analysis can be divided into the following two stages.<br>1. First, check "Save as Inheritable Project" and analyze it.<br>2. Next, after changing the boundary condition and analysis setting, check "Inherit Analyzed Project" and analyze. With the stress state as the result of the analysis at 1 for the initial state of the model, the following analysis is performed. |
| Criteria for Termination | When performing "Prediction of Plastic Failure," you can give the condition "End analysis if destruction progresses to this state." For each of shell elements and solid elements, enter the number of elements in the "Cracked"/"Plastic"/"Crushed" state as a condition. (When 0 is entered, it is set as no condition,) If any of the conditions are satisfied during analysis, the analysis ends at that point. |
| No output until the element is broken | When turned on, regardless of the setting of the output step interval, the result will not be output to file until the element yield or tensile failure occurs in the element in material nonlinear analysis. |

## 10.6 Message at the end of analysis

When the analysis ends, the "Status" part of the progress bar window changes to "Successfully finished" or "Abnormally finished."
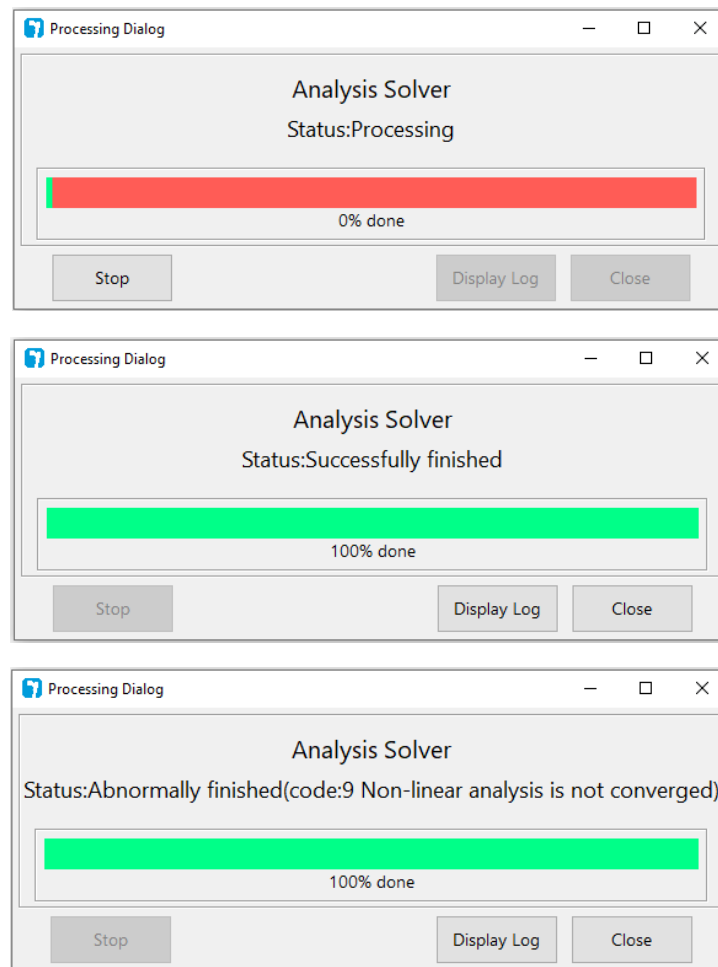


If the analysis terminates abnormally, the codes and messages that indicate possible causes are displayed; refer to the following code table and change the analysis conditions. In addition, even if analysis is completed normally, the following codes may be displayed.

| Code | Message<br>(**Abnormally finished**/<br>**Successfully finished**) | Description |
|---|---|---|
| code: 2 | code: 2: Input data error | This is displayed when the data format of the input file for the solver program is invalid, or when the constraint nodes is constrained as the secondary nodes of the contact element by the primary surface. Please check the condition of the contact element. |
| code: 3 | code: 3: Element is unsuitable for analysis | This is displayed when the element shape is bad and cannot be analyzed. Please remesh with different mesh condition.(size , etc.) |
| code: 4 | code: 4: Inadequate Constraint | This is displayed when simultaneous linear equations cannot be solved in the first iterative calculation of the initial step. There is a possibility that the model is not completely constrained or there is an independent floating element. |
| | code: 4 | When using the sparse matrix method (direct method), this will be displayed if the |

| | | calculation cannot be performed during analysis. It may be displayed when the rigidity locally approaches 0 due to element failure. Analysis results up to that point can be evaluated correctly. |
|---|---|---|
| code: 5 | code: 5: Insufficient memory | This is displayed when the memory necessary for analysis is insufficient. There is a possibility that the setting value of "Memory Usage (MB)" is small. Alternatively, if the sparse matrix method is selected, there is a possibility that this setting value is set too high for the memory installed in the PC. If this error occurs due to reasons other than the above, you may need to create the model again with larger mesh size to reduce the number of elements of the analysis model, or you may need to increase the memory of your computer. |
| | code: 5 Insufficient memory | When contact analysis is performed, this is displayed when the memory is insufficient during analysis. Analysis results up to that point can be evaluated correctly, but you can continue analyzing further by increasing "Memory Usage (MB)" slightly. |
| code: 6 | code: 6: Initially broken | This is displayed when failure occurred in the initial step and calculation cannot be continued any further. Reduce the total load or adjust the number of steps and the number of sub steps so that the load value per step decreases. |
| code: 7 | code: 7: Cannot solve at initial step | This is displayed when failure does not occur and cannot be calculated in the initial step. This error may occur when special materials, such as contact elements and gap elements, are used. It is possible that this error can be avoided by setting the condition so that the spring stiffness of the gap element is reduced or the load value per step is small. |
| code: 8 | code: 8: Cannot solve by CG-method | Unused. |
| | code: 8 | When the CG method (iterative method) is used, it is displayed when the calculation cannot be performed during analysis. It may be displayed when the rigidity locally approaches 0 due to element failure. Analysis results up to that point can be evaluated correctly. |
| code: 9 | code: 9: Non-linear analysis is not converged | This is displayed when the unbalance force cannot be converged within the iteration limit at initial step with geometric non-linear or material non-linear analysis of solver V2. |
| | code: 9 | This is displayed when the unbalance force cannot be converged within the iteration limit during analysis with geometric non-linear or material non-linear analysis of solver V2. |
| code: 10 | code: 10: Aborted by stop bottom | This is displayed when an analysis is aborted by "stop" button without result. |
| | code: 10: Result until aborted | This is displayed when an analysis is aborted by "stop" button with any result. |

# Chapter 11 Display Function

This chapter explains the display functions in the main menus. There are two ways to start the desired display function.

(1) Using the desired "Display" icon in the main menu (displaying the contents of the working project only).

(2) Selecting from the option menu (displaying the saved or working projects).

**(1) Using the desired "Display" icon in the main menu**

        This is a method by which you start the desired display function using one of the [Display] buttons in the following menu.
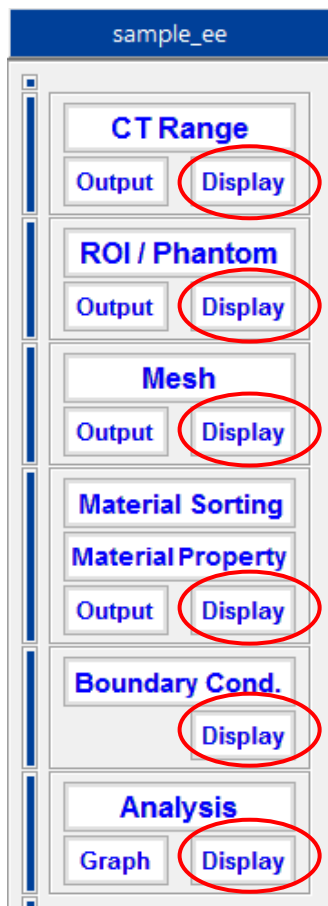


**[Characteristics]**

- You can display the working project being opened at the moment.

- You cannot use the "Display" function for an unexecuted process.

   (For example, for a project in which settings up to "Material Property" have been completed, "Display" of "Boundary Condition" cannot be displayed.

- When "Display" is selected, you cannot carry out other work for the working project.

   (Close the display screen, and you can carry out work normally.)

**(2) Selecting from the option menu**

This is a method by which you start the desired display function from the option menu.
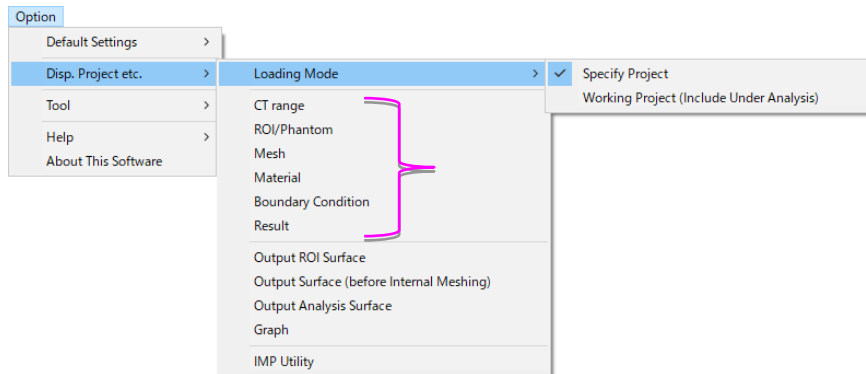


**[Characteristics]**

- You can display not only an already saved project but also the working project.

- To display a saved project, you do not need to open it.

- When merely using the project display function, unlike the method in (1), the process of opening a project and copying it to the work directory is not required.

　So, you can view the project quickly.

- You cannot use the "Display" function for an unexecuted process.

　However, by selecting "Working project (Include Under Analysis)" for the read mode, if one or more steps have been output for analysis results, you can display the progress of analysis.

- You can start multiple display screens to display multiple projects or display the same project in different manners.

**[Read Mode]**

- Specify Project <<Initial setting mode>>

　--- When this display function is selected, a file browser opens. Then, specify any project.

- Working project (Include Under Analysis)

　--- When this display function is selected, the working project is automatically selected.

　(The project needs to be being opened. )

**[Status that can be displayed]**

| Project processing stage | Read mode | |
|---|---|---|
| | Specify Project | Working project (Include Under Analysis) |
| 1. "CT Range" completed | "CT Range" can be displayed. | "CT Range" can be displayed. |
| 2. "ROI・phantom" completed | "ROI・phantom" can be displayed. | "ROI・phantom" can be displayed. |
| 3. "Mesh Generation" completed | "Mesh" can be displayed. | "Mesh" can be displayed. |
| 4. "Material Sorting" "Material Property" completed | "Analysis Material" can be displayed. | "Analysis Material" can be displayed. |
| 5. "Boundary Condition" completed | "Boundary Condition" can be displayed. | "Boundary Condition" can be displayed. |
| 6. "Analysis" completed | "Analysis Results" can be displayed. | "Analysis Results" can be displayed. |
| 7. For "Analysis," if analysis has been done for one or more steps | Nothing can be displayed. | "Analysis Results" can be displayed. |

## 11.1 CT Range Display

This function displays an original CT image read through the DICOM interface.
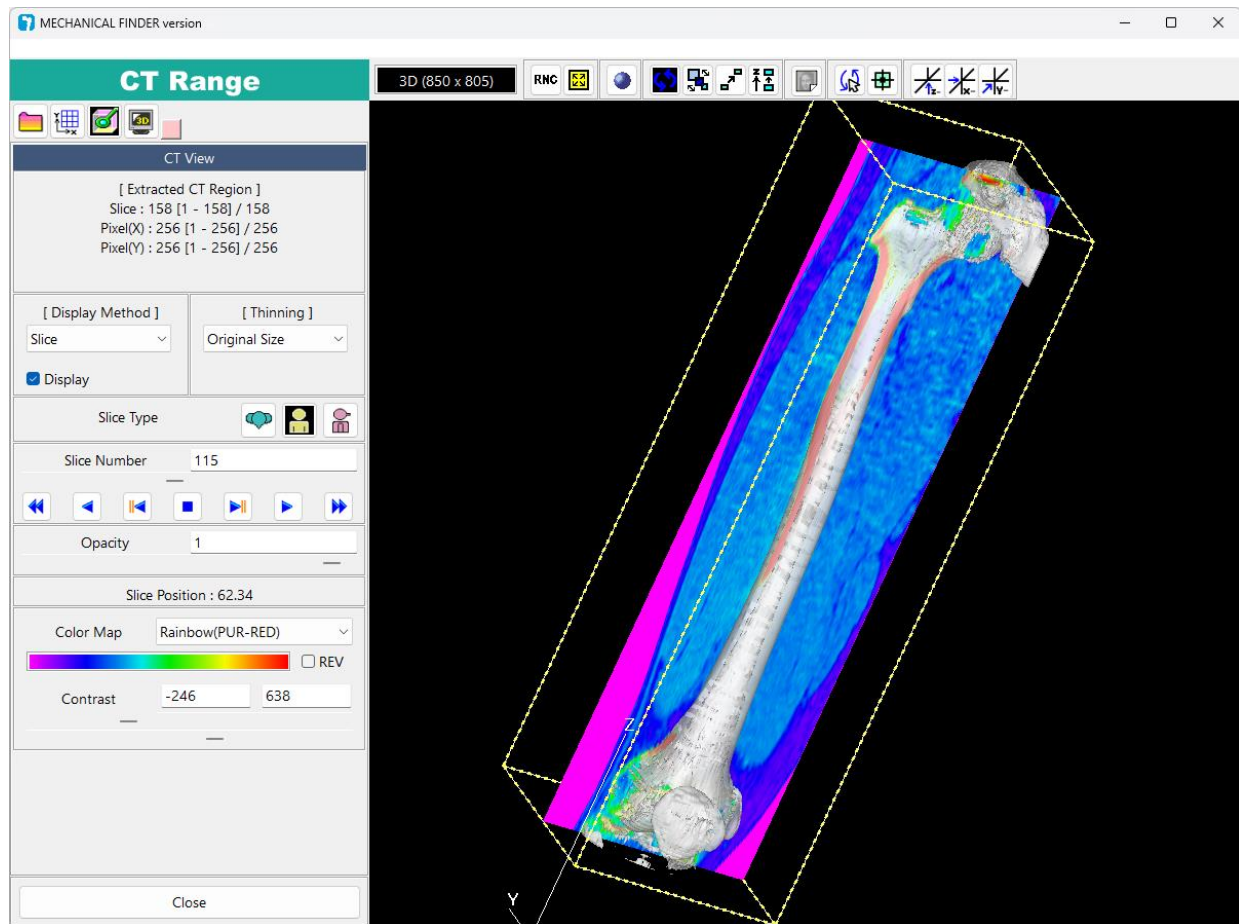


| Icon | Function |
|------|----------|
|  | Data Information |
|  | Surface Display |
|  | CT Display |
|  | Viewer Settings |
|  | Post Control Function |

## 11.2 ROI/Phantom Display

This function displays an image binarized through "ROI Extraction" processing.
A binarized image can also be displayed by being merged with its original CT image.
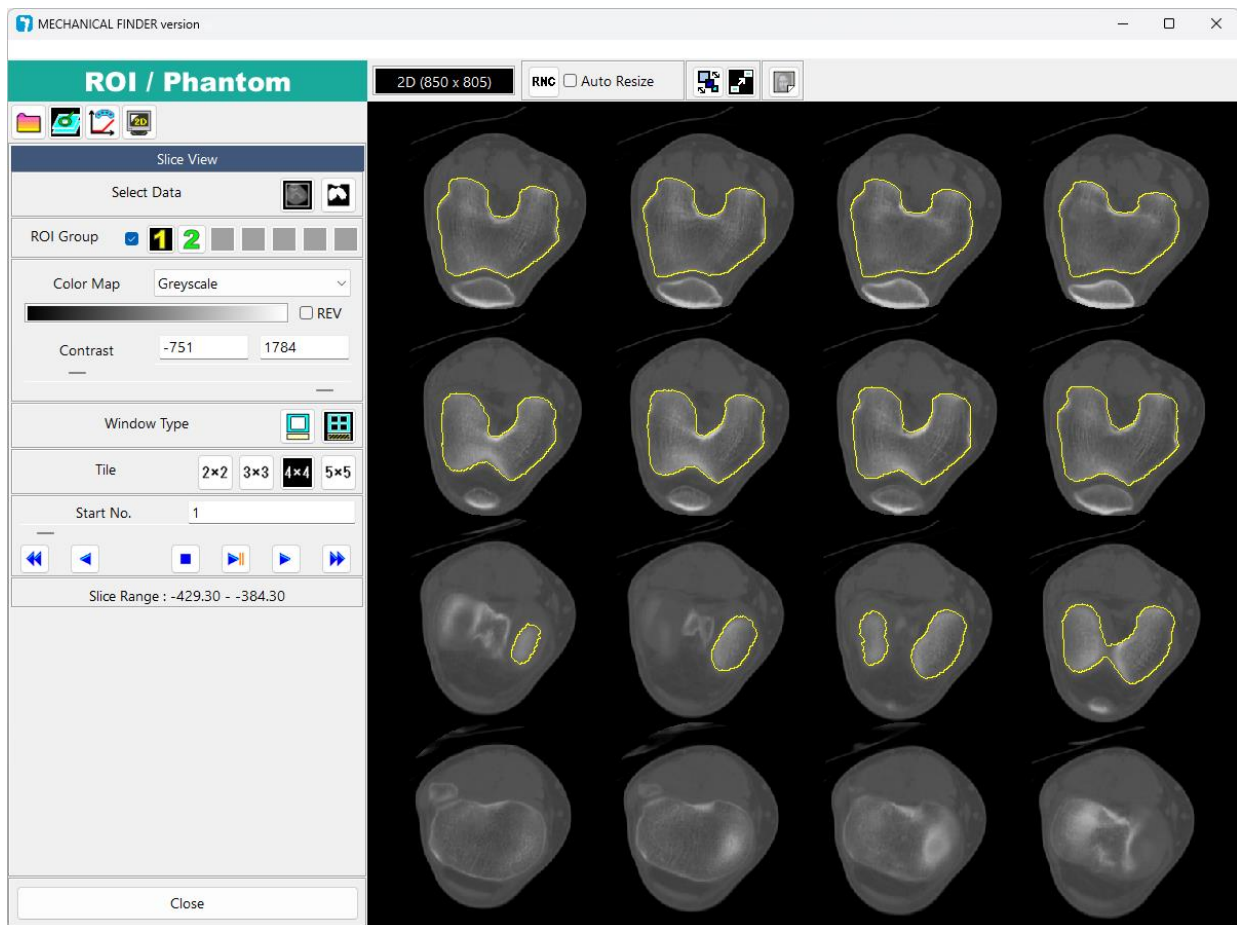


| Icon | Function |
|------|----------|
|  | Data Information |
|  | Slice Display |
|  | Phantom Display |
|  | Viewer Settings |

## 11.3 Mesh Display

This function displays a mesh geometry generated through "Mesh Generation" processing.
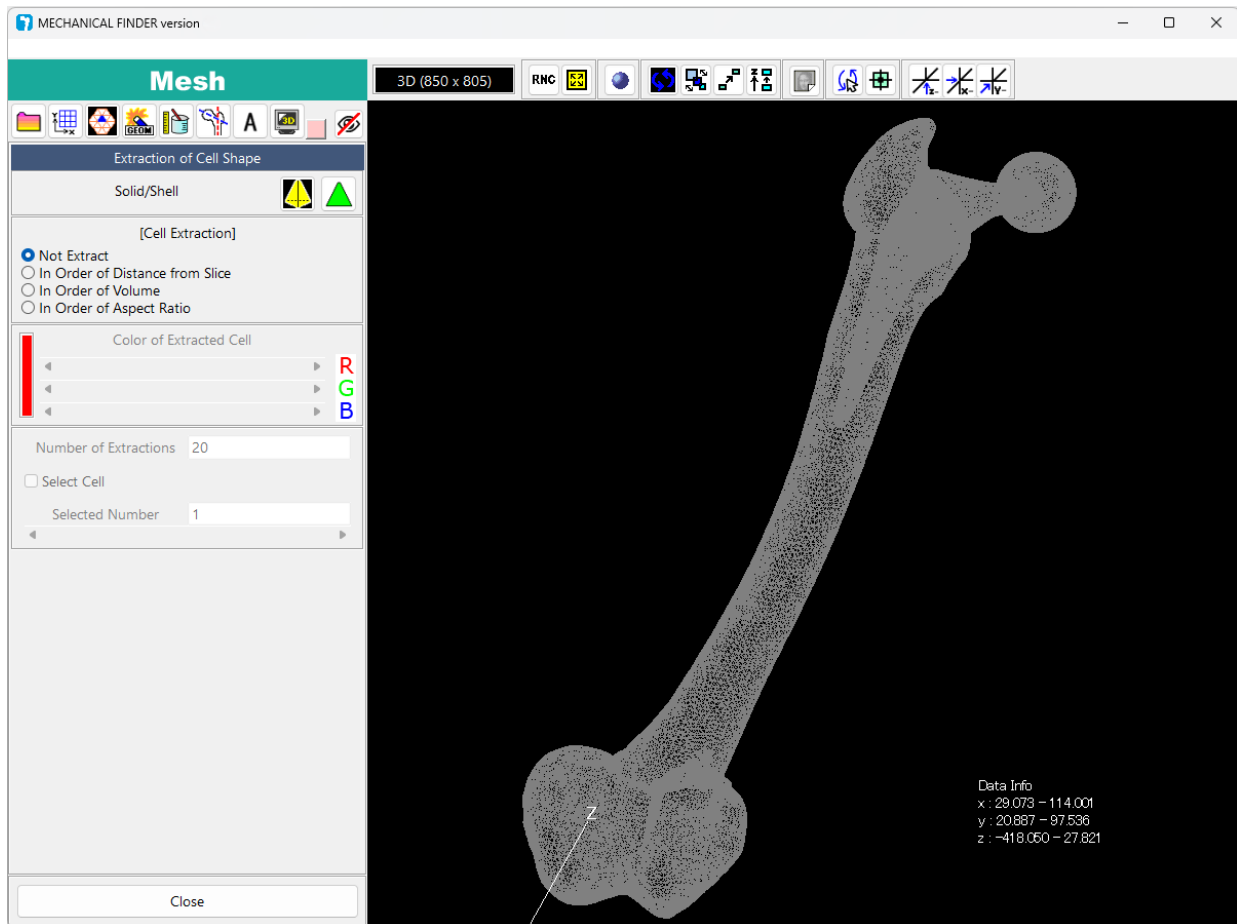


| Icon | Function |
|------|----------|
|  | Data Information |
|  | Surface Display |
|  | Cell Shape Extraction |
|  | Data Extraction |
|  | Measurement |
|  | Axis View |
|  | Label Display |
|  | Viewer Settings |
|  | Post Control Function |

## 11.4 Analysis Material Display

This function displays the material properties set through "Material Sorting" and "Material Property" processing.
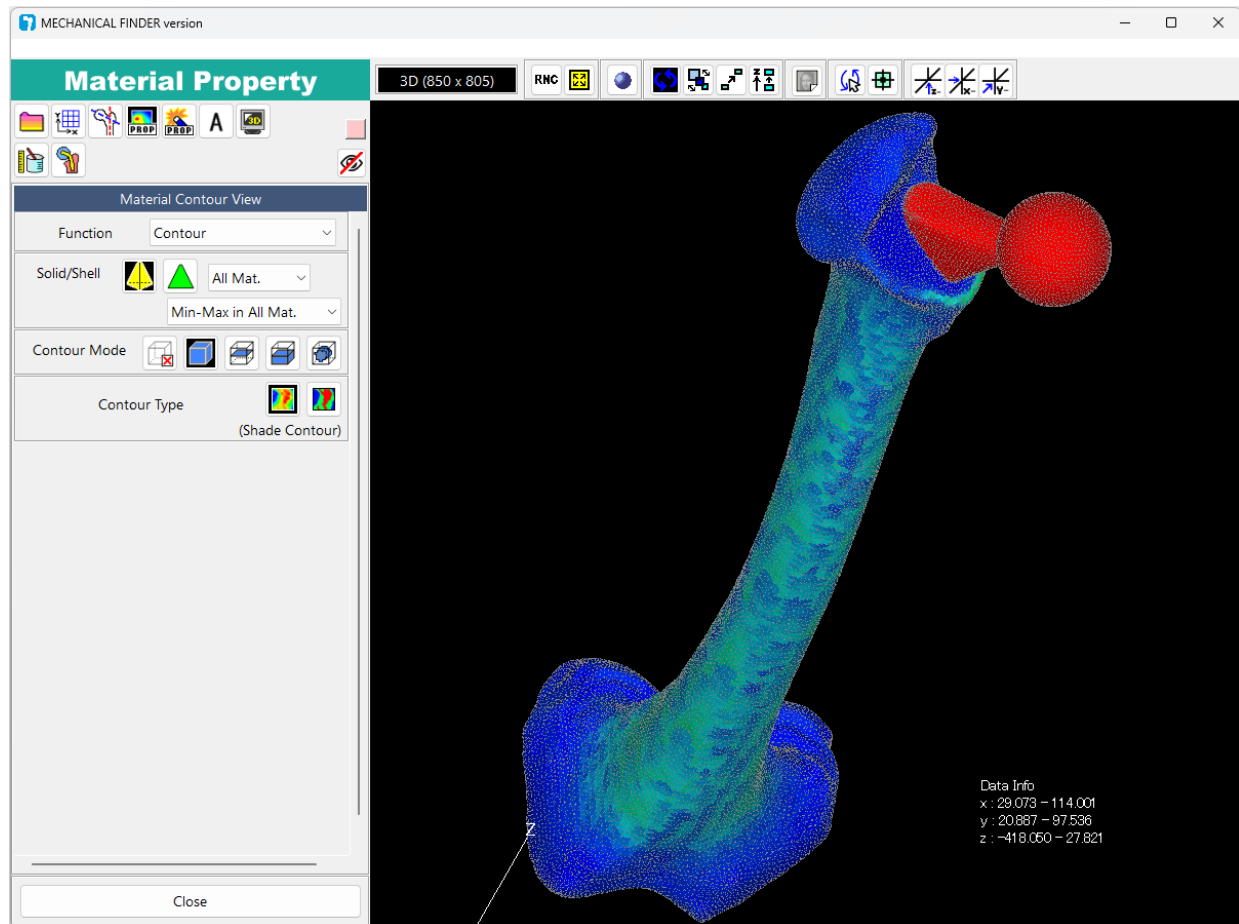


| Icon | Function | Icon | Function |
|------|----------|------|----------|
| | Data Information | | Material-Based Model Display |
| | Surface Display | | Data Extraction |
| | Axis View | | Label Display |
| | Material Property Contour Display | | Viewer Settings |
| | Measurement | | Post Control Function |

## 11.5 Boundary Condition Display

This function displays the analysis conditions set through "Boundary Condition" processing.
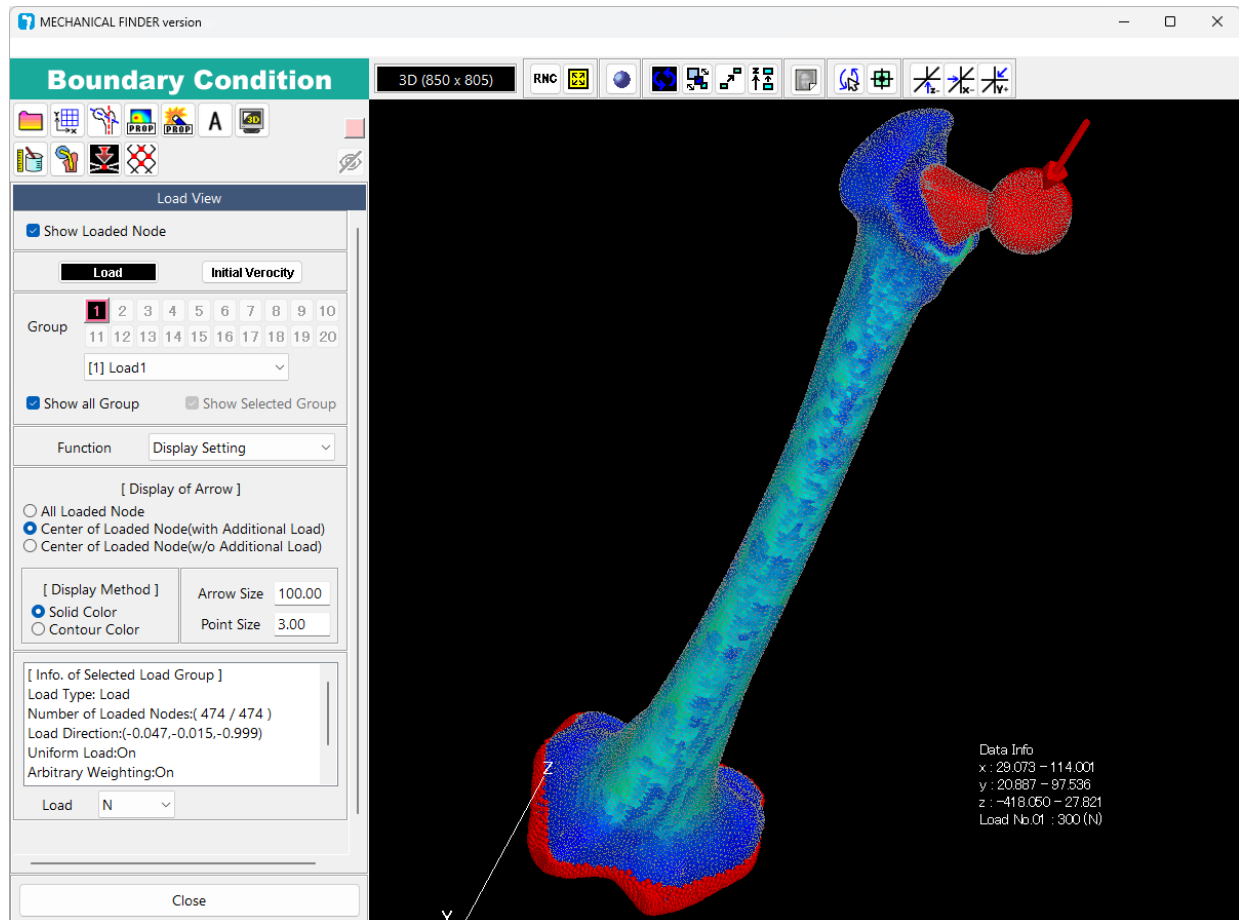


| Icon | Function | Icon | Function |
|------|----------|------|----------|
| | Data Information | | Material-Based Model Display |
| | Surface Display | | Label Display |
| | Axis View | | Viewer Settings |
| | Material Property Contour Display | | Node Load Display |
| | Data Extraction | | Constraint Display |
| | Measurement | | Post Control Function |

## 11.6 Result Display

This function displays the results calculated through "Analysis" processing.

This function can also display the analysis conditions set through "Material Property" and "Boundary Condition" processing.
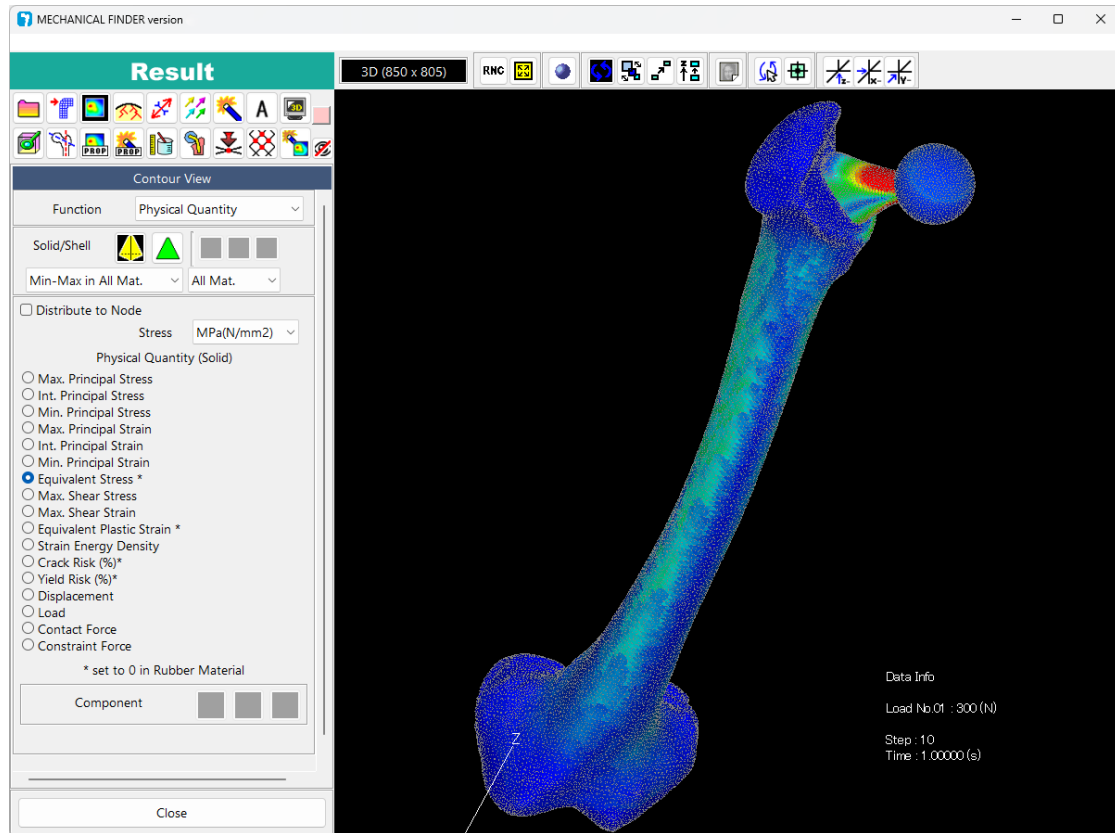


| Icon | Function | Icon | Function | Icon | Function |
|---|---|---|---|---|---|
| | Data Information | | CT Display | | Post Control Function |
| | Deformation Display | | Axis View | | |
| | Contour Display | | Material Property Contour Display | | |
| | Plastic Failure Display | | Data Extraction | | |
| | Tensor Display | | Measurement | | |
| | Vector Display | | Material-Based Model Display | | |
| | Data Extraction | | Node Load Display | | |
| | Label Display | | Constraint Display | | |
| | Viewer Settings | | Data Processing and Visualize | | |

# Chapter 12 "Output" Function

This chapter explains a function for outputting a surface geometry into a file.

A generated surface geometry can be output in STL or DXF file format.


An output geometry differs depending on the output process among the following.


1. "12.1 Output surface (for Special Purpose)" (EE)

    You can select this after completing "CT Range" work.

    This process is different from other surface geometry output processes and is provided mainly for analyzing a fine trabecular bone structure as a target.

    A geometry extracted here is used for being imported at the time of "Mesh Generation" processing.


2. "12.2 Output Surface (ROI)"

    You can select this after completing "ROI Extraction" work. An external geometry is created from ROI volume data.


3. "12.3 Output Surface"

    You can select this after completing "Mesh Generation" work. An external geometry is generated by a routine for creating triangle with lower aspect ratios. For the import section, the read geometry is used as is.


4. "12.4 Output Surface (Analysis mesh)"

    You can select this after completing "Material Sorting" work. Data after conditioned mesh generation can be output on the basis of an individual group or multiple groups.

## 12.1 Output Surface (for special purpose) (EE)

In the Extended Edition, after completing CT range work, you can perform special-purpose surface geometry output processing.

This process is provided mainly for analyzing a fine trabecular bone structure as a target, allowing you to create not only a trabecular bone section geometry, but also a compressor to set load/constraints.

To express a fine and complex trabecular bone structure, the number of elements becomes large. So, you must extract a smaller analysis region in the "CT Range" settings. Refer to the next page for operation.
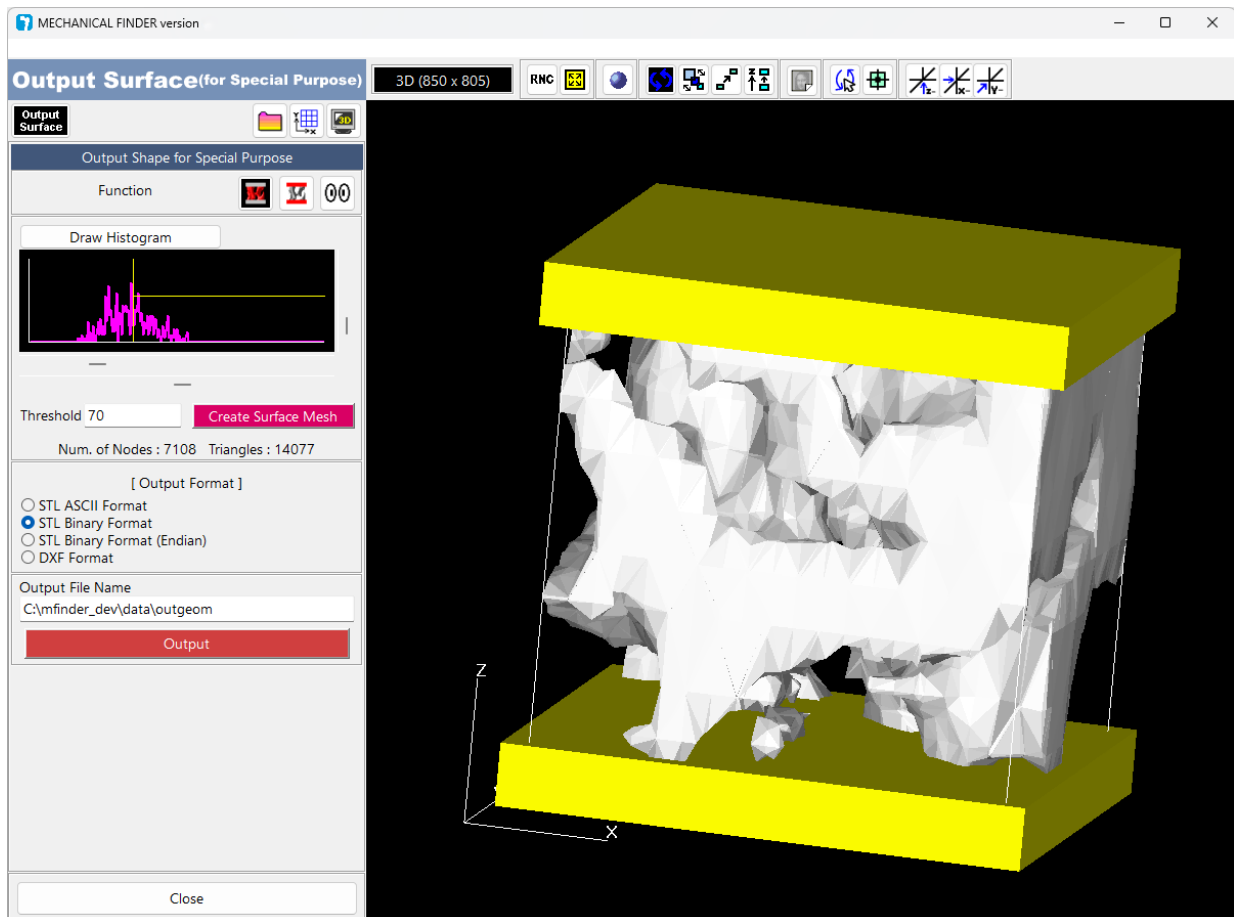
**Operation procedure when analyzing a trabecular bone structure**

If you want to analyze a fine and complex trabecular bone structure, we recommend analyzing it in a way slightly different from the way you usually use this software.

The following is a summary of the operation procedure.

| |
|---|
| 1. In "CT Range," set the range for the trabecular bone area and determine it. |
|     For a trabecular bone whose structure is complex, if the range is set too large, the number of elements may eventually become too large for the analysis. Be careful. |
|  |
| 2. In "Special-Purpose Surface Geometry v Output," create a trabecular bone structure geometry and output it in STL format. |
|     At that point, create a compressor together, if necessary, and output it in STL format as well. |
|  |
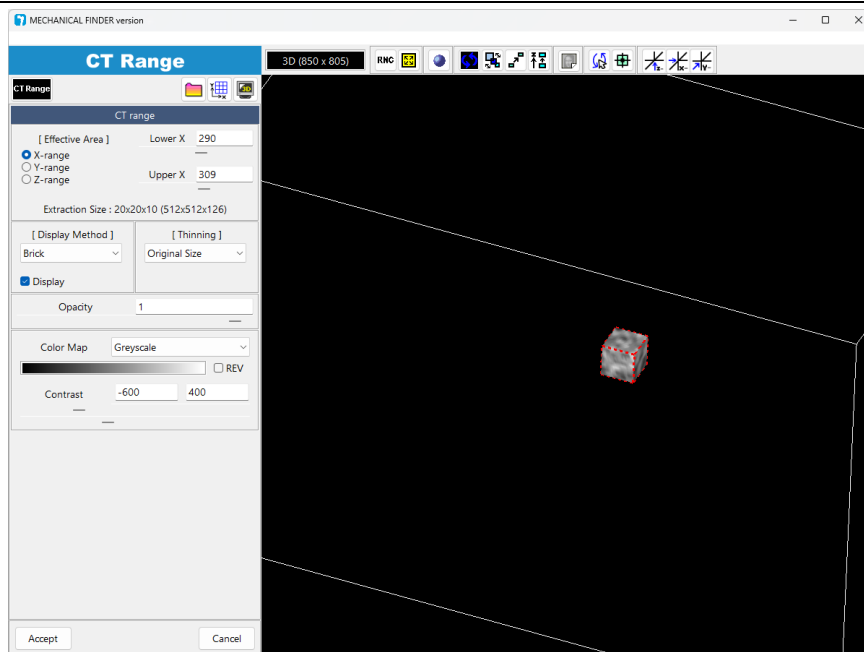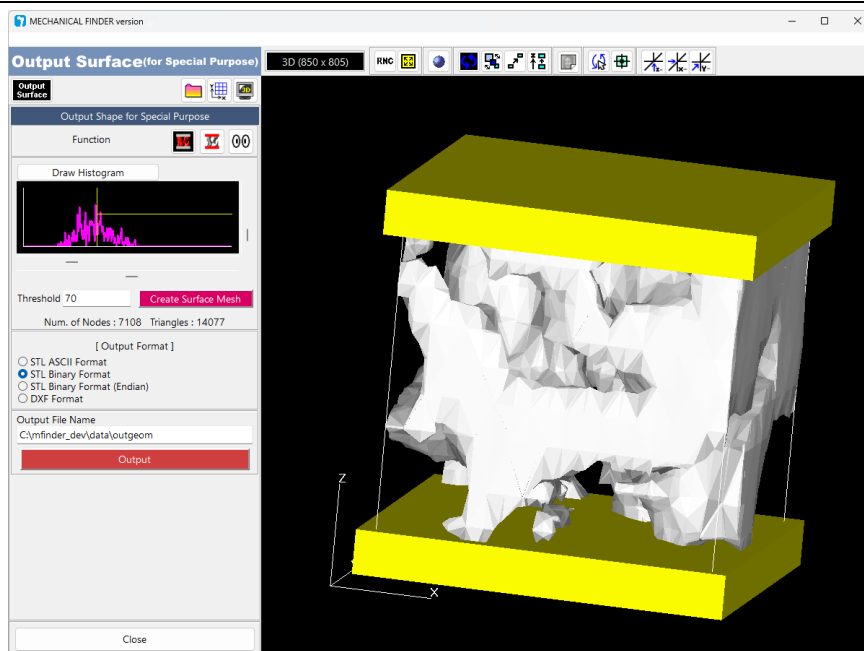
3. There is no need to perform the ROI-related work in "Chapter 5 ROI extraction/Phantom settings." If you need to configure phantom settings, configure phantom settings only and press the [OK] button to complete the process.
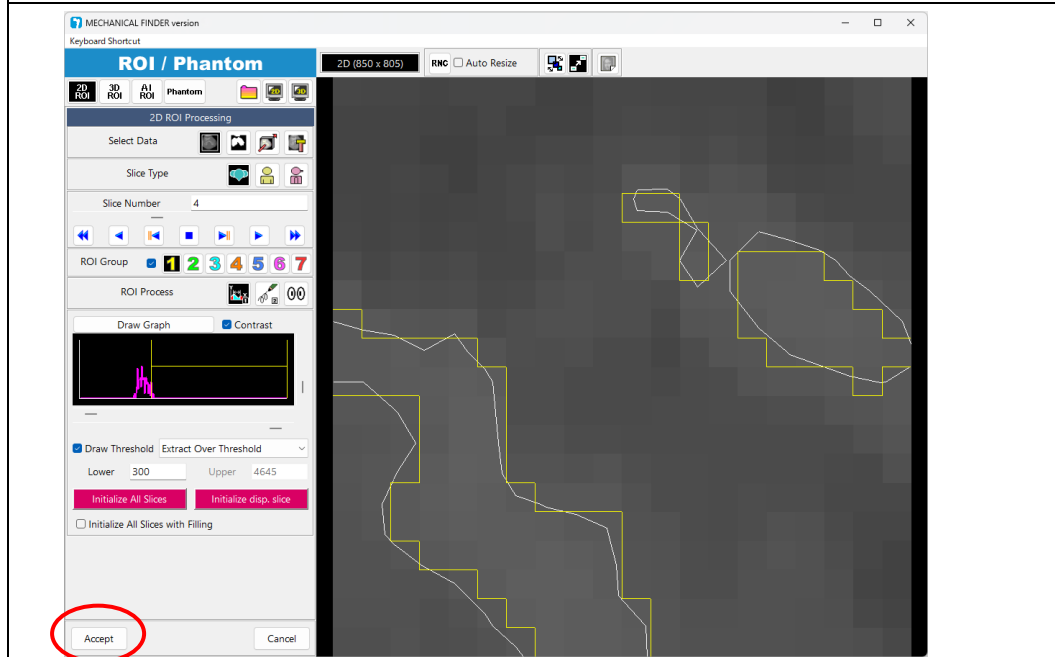


4. In "Chapter 6 Mesh Generation," without using a surface mesh, read a trabecular bone geometry and compressor geometry through import processing.
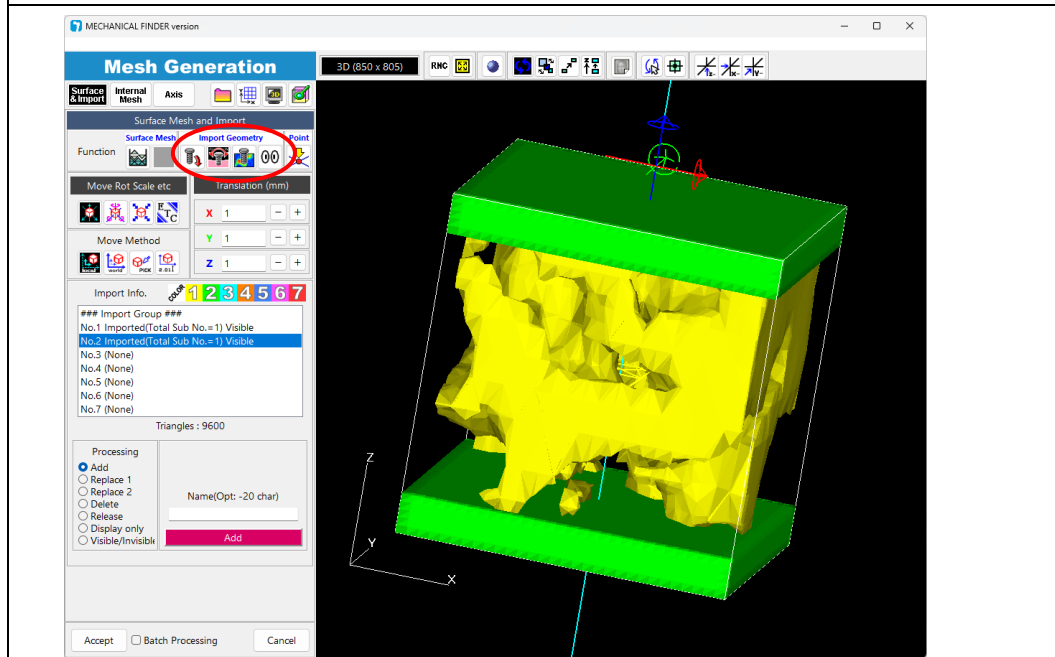


5. For the remaining processing, you can follow the ordinary procedures for this software.

**\*For a complex trabecular bone geometry, mesh generation and analysis processing require a longer time than those for a geometry whose number of elements is the same.**

## 12.2 Output Surface (ROI)

You can output an external geometry after ROI extraction work.

A surface geometry is generated from ROI volume data.
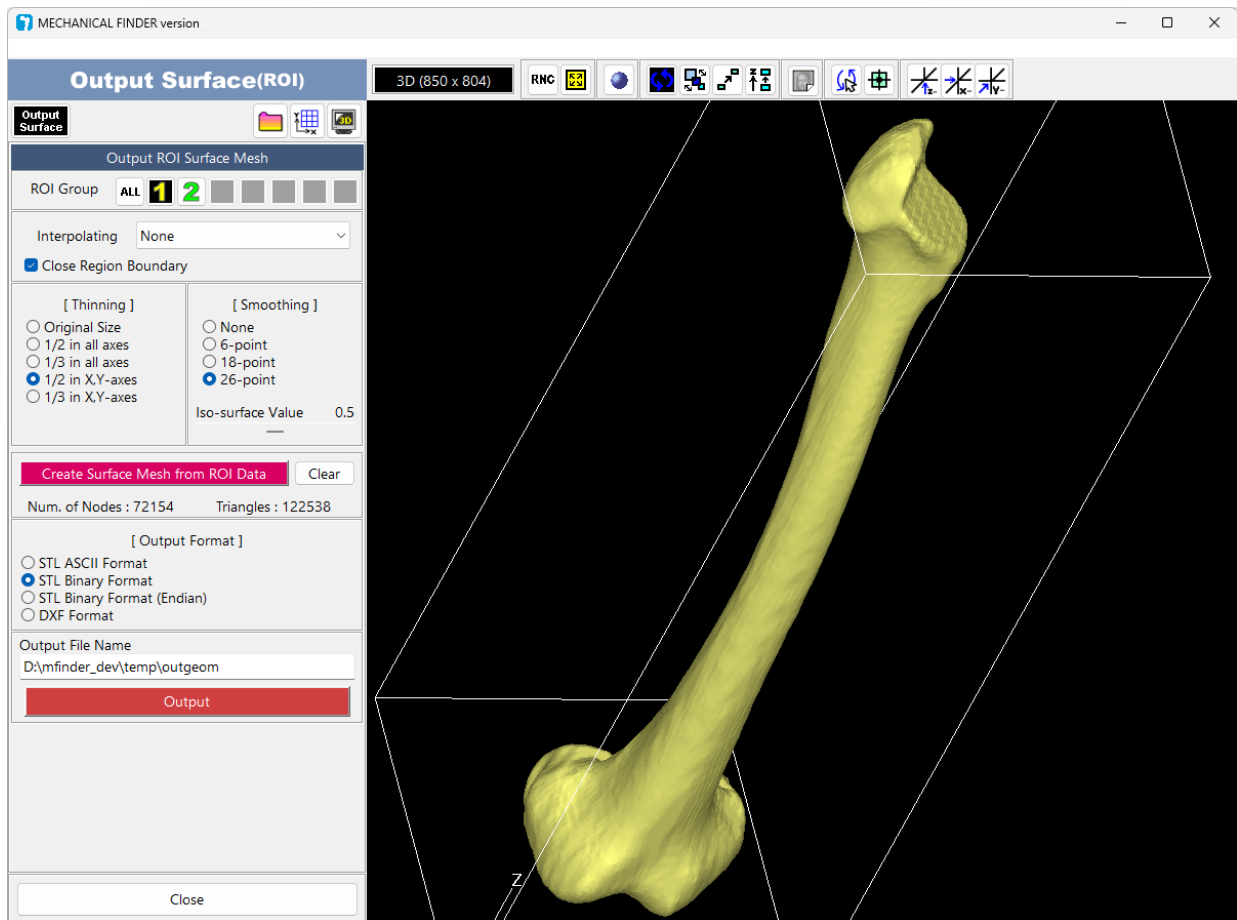
## Smoothing processingf

For "Output Surface (ROI)," smoothing processing, which makes data smooth, is available.

If isosurface processing is performed by using ROI extraction data as is, parts analyzed as "true" and "false" on a per-pixel basis appear as "jaggies." These jaggies can be eliminated through smoothing processing.

The following types of smoothing processing are provided.

- No smoothing

- Six neighborhoods smoothing

- Eighteen neighborhoods smoothing

- Twenty-six neighborhoods smoothing

Except "No smoothing," ROI extraction data values are changed for processing within the framework of this processing. Therefore, a generated geometry may be slightly different from the original data.

(*Since data are changed within the process "Output Surface (ROI)" only, the project file or other menu processes will not be affected.)

- Smoothing internal processing
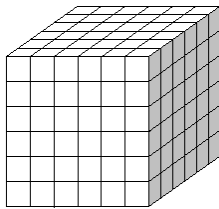
As shown in the left figure, ROI data are retained as voxel data in X, Y, and Z directions. Each element is given "true" (1) data or "false" (0) data.

1) No smoothing

When No smoothing is selected, data are treated as is, and isosurface processing is performed on it.

2) Six neighborhoods smoothing
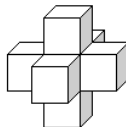
A numerical value for each element is assigned the mean value of its six neighbor elements. An assigned value is a floating-point number in the range between 0.0 and 1.0. This processing is performed on all elements for isosurface processing.

3) Eighteen neighborhoods smoothing
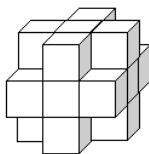
A numerical value for some element is assigned the mean value of its eighteen neighbor elements. An assigned value is a floating-point number in the range between 0.0 and 1.0. This processing is performed on all elements for isosurface processing.

4) Twenty-six neighborhoods smoothing
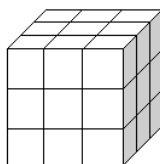
A numerical value for some element is assigned the mean value of its twenty-six neighbor elements. An assigned value is a floating-point number in the range between 0.0 and 1.0. This processing is performed on all elements for isosurface processing.

## 12.3 Output Surface

You can output a surface geometry after the surface mesh generation.

A generated surface mesh and an import geometry are output into a file as is.

The surface geometry output here becomes the same geometry as the one before the internal mesh generation.

To output a surface geometry after the internal mesh generation, follow the procedure described in "12.4 Output Surface (Analysis mesh)."
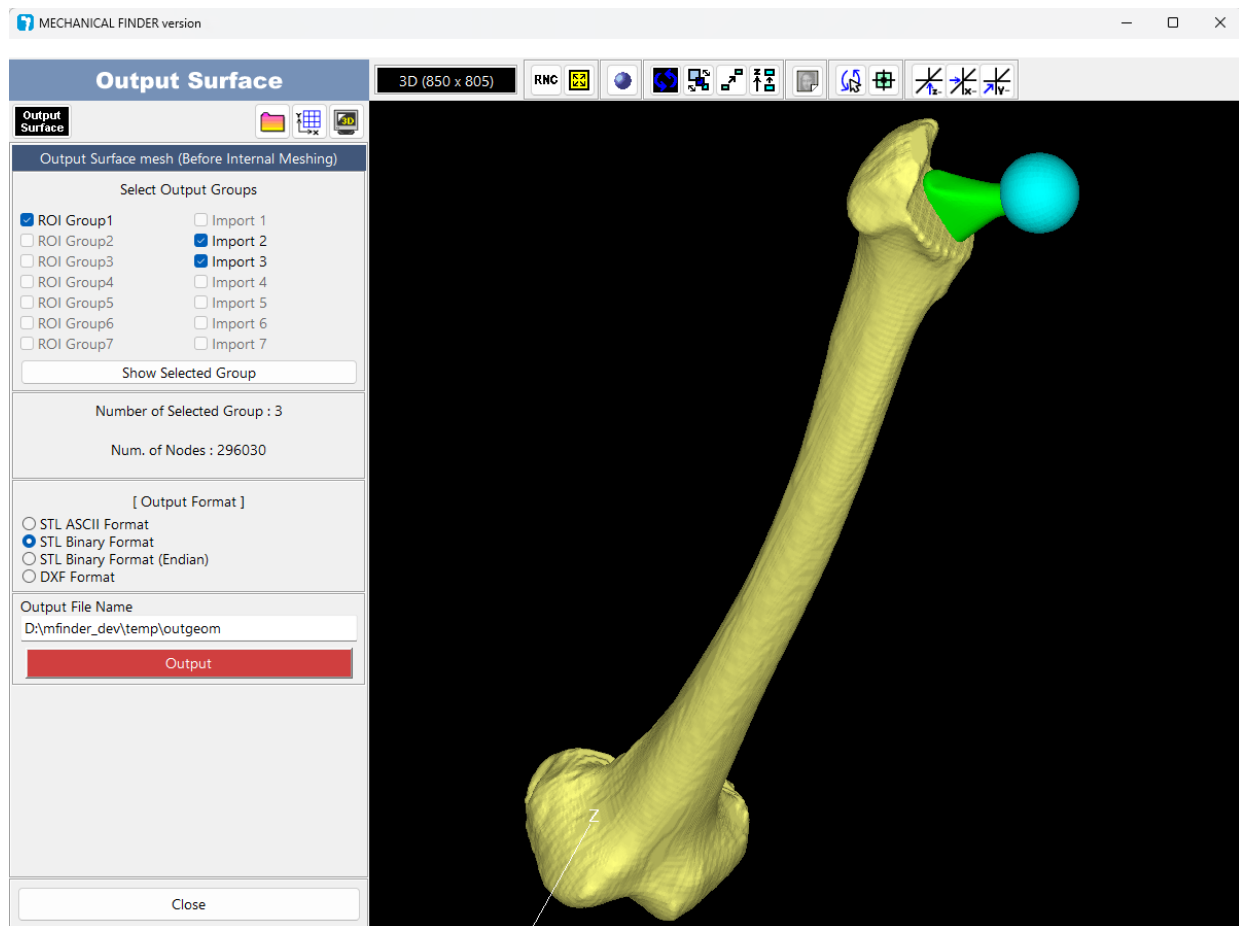
## 12.4 Output Surface (Analysis mesh)

You can output a surface geometry after the internal mesh generation.

A generated surface geometry is output into a file.
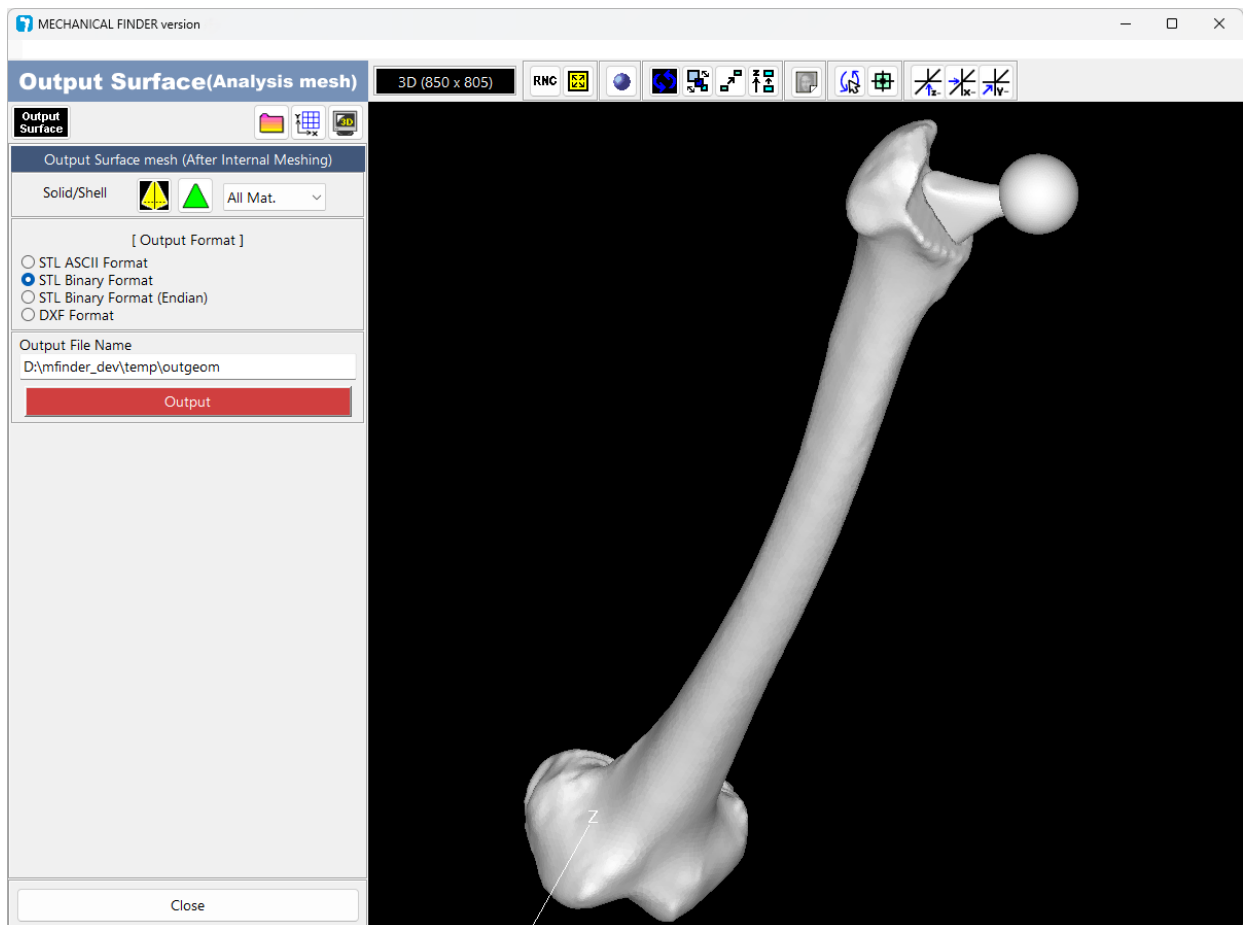
## 12.5 Differences in Output Surface

The following types of surface geometry output are available:

1) Output Surface (for Special Purpose)

2) Output Surface (ROI)

3) Output Surface

4) Output Surface (Analysis mesh)

Each type aims to output data into an STL/DXF file in the same way, but the amount of data, generated geometry, and the method are very different depending on the type.
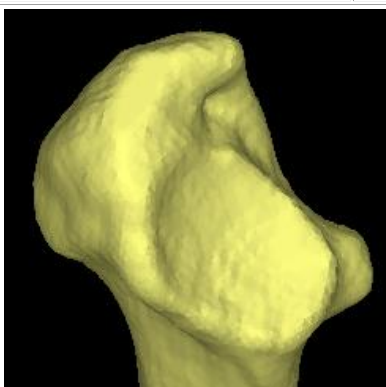
The differences and samples are shown below.

Note that descriptions, such as a sample, of "Output Surface (for Special Purpose)" are omitted because that output is used for a special purpose. However, its basic method is very similar to the method for "Output Surface (ROI)," and its generation method is isosurface processing.
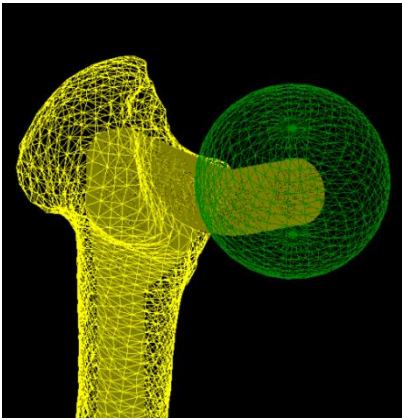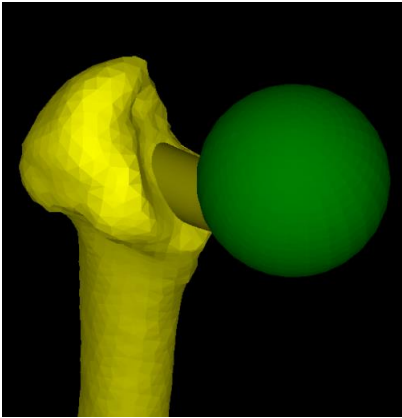
|  | Output Surface (ROI) | Output Surface | Output Surface (Analysis mesh) |
|---|---|---|---|
| Generated triangle | - Very many<br>   The number of triangles becomes large because operation is based on a voxel grid of ROI extraction data. | - Not many<br>   The number of triangles does not become large because generation is performed based on the Basic Mesh Size.<br>   However, for the import section, the read STL data geometry is used as is. | - Not many<br>   The number of triangles does not become large because generation is performed based on the Basic Mesh Size. |
| Generation method | - Isosurface processing<br>   Isosurface processing by the marching cube method is performed. | - Meshing processing applicable to numerical analysis<br>   Mesh division assuming that numerical analysis is performed is executed. | - Meshing processing applicable to numerical analysis<br>   Mesh division assuming that numerical analysis is performed is executed. |
| Generated triangle's shape | - Non-uniform shape<br>   Generated triangles are not uniform in terms of size and aspect ratio. A triangle similar to a dot or line may be generated. | - Uniform shape<br>   Triangles with little irregularities in size and with good aspect ratios are generated.<br>   However, for the import section, the read STL data geometry is used as is. | - Uniform shape<br>   Triangles with little irregularities in size and with good aspect ratios are generated. |
| Output with imported geometry | - Import geometry not to be output<br>   This is a process prior to import processing, so the import geometry is not output. | - For the ROI section and the import section, their shapes are different from each other.<br>   They may be overlapping each other on some surface, since this is a process before adjustment. | - The ROI section and the import section are well adjusted.<br>   There is no surface on which these sections overlap each other, since the analysis mesh is used as a reference. |

Example: Each type of processing is performed on sample data
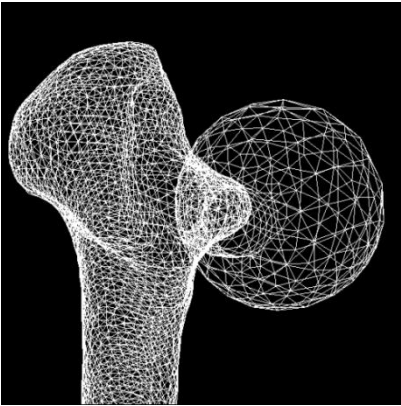
"Output Surface (ROI)"

| | |
|---|---|
|  | |
| - No interpolation processing<br><br>- Size 1/2<br><br>- No smoothing | No. of nodes 50,064<br>No. of triangles 100,124 |
|  | |
| - No interpolation processing<br><br>- Size 1/2<br><br>- Twenty-six neighborhoods smoothing | No. of nodes 48,320<br>No. of triangles 83,002 |
|  | |
| - No interpolation processing (jaggy resolution effect is large)<br>- Size 1/2<br>- Twenty-six neighborhoods smoothing | No. of nodes 95,068<br>No. of triangles 164,186 |

"Output Surface"



| With 3 mm mesh reference size in "Mesh Generation" | No. of nodes 26,283 |
| --- | --- |
| | No. of triangles 53,472 |

"Output Surface (Analysis mesh)"



| With mesh size: minimum 4 mm and maximum 12 mm in "Internal Mesh Generation." | No. of nodes 19,133<br>No. of triangles 12,674 |
| --- | --- |

# Chapter 13 "Graph"

Graphs using material properties and results can be drawn.
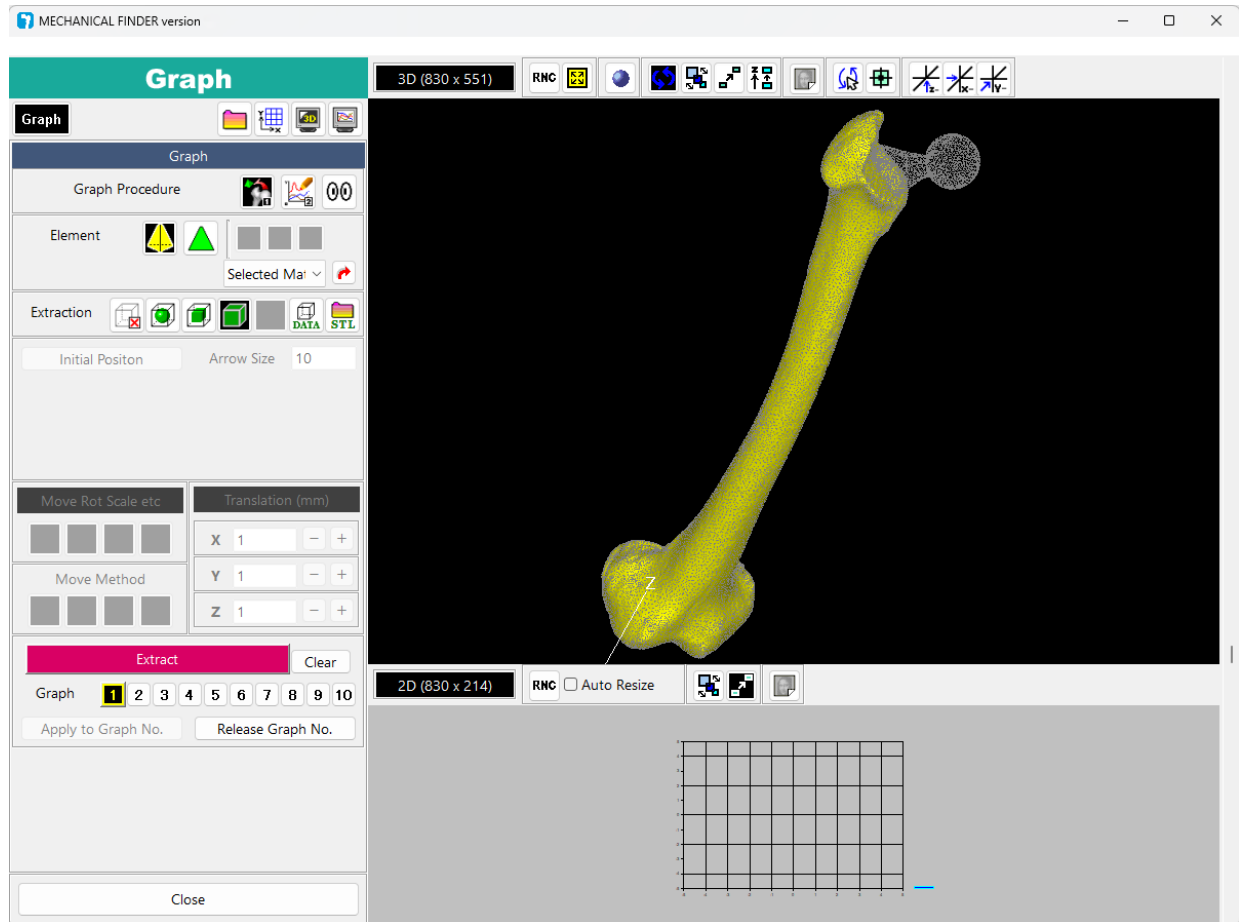
## 13.1 Viewer Window

The display of the viewer area in Graph is vertically split into two parts.

As shown in the figure below, move the right slider up and down to change the ratio between two split parts.
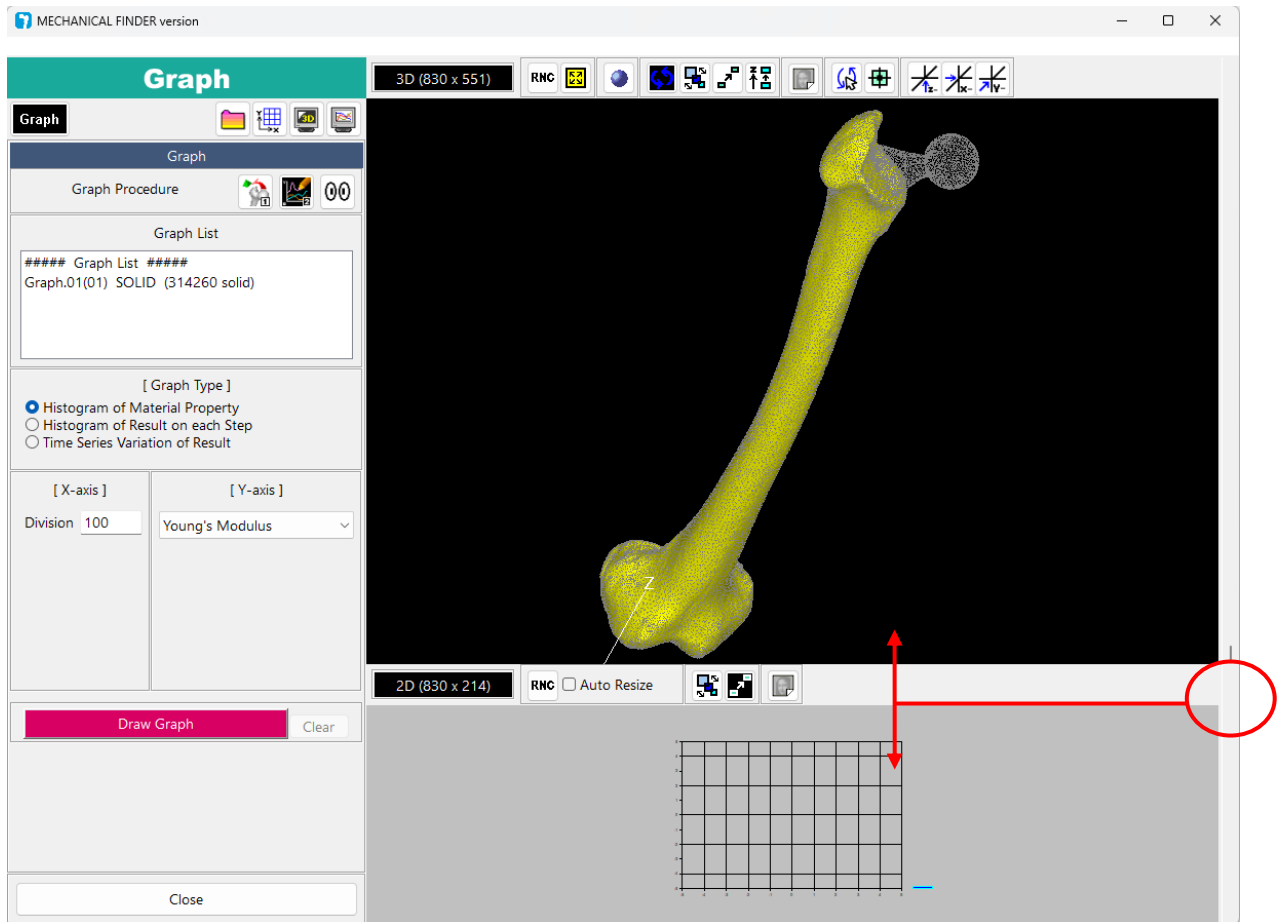
# Chapter 14 Tool

This SOFTWARE includes the following tools.

These tools can be started from Start Menu [STRAT] ->[PROGRAM」 ->[MECHANICAL FINDER」 ->[TOOL]

| Icon | Function |
|---|---|
| 14.1 Construct Project by Image | A project file of this software is generated from an image file such as BMP and JPEG and output |
| 14.2 Output Project Data in Text | A content of the project file of this software is output as ASCII (Text)-format file. |
| 14.3 Inhomogeneous Material Editor | A conversion curve of Young's modulus, yield stress value, critical stress value, etc. against density value can be additionally edited. |
| 14.4 Material Database | Young's modulus, yield stress value, critical stress value, etc. of uniform materials can be added and edited for each material. |
| 14.5 Batch Program (Mesh /Analysis) | This is a function that performs mesh generation, and analysis, of multiple projects by batch process.<br> This includes a simple scheduling function. |
| 14.6 Remote Batch Program | This is a remote process type of the above Batch Program.<br>However, no mesh generation can be performed.<br>Other project process can be performed while analyzing remotely, thus this is effective in efficient works. |
| 14.7 Project Manager | Renaming, copying, and moving of the project; deletion and editing of the patient information can be performed.<br>One project includes the maximum 14 project files which can be processed in a lump. |
| 14.8 Optional License Information | Registration and displaying information of an optional license of this software are carried out |

## 14.1 Construct Project by Image

This tool generates a project file of this software from image files having multiple sheets. Corresponding image formats are as follows:

     1) BMP format (Color is allowed. It becomes 8-bit gray scale after conversion. )

     2) JPEG format (Color is allowed. It becomes 8-bit gray scale after conversion. )

     3) TIFF format (Output by 16 bits. It becomes 16-bit gray scale after conversion. )


The project file generated using the tool has the following limitations.

     - The number of pixels in X and Y directions after conversion shall match to those of the image file with the minimum size.

     - The slice position between the image files shall be calculated as the equal distance slice by the specified "distance between images."

     - The density value after conversion shall be 256 gradation gray scale of 0 to 255. (Except TIFF)


[Usage Method]
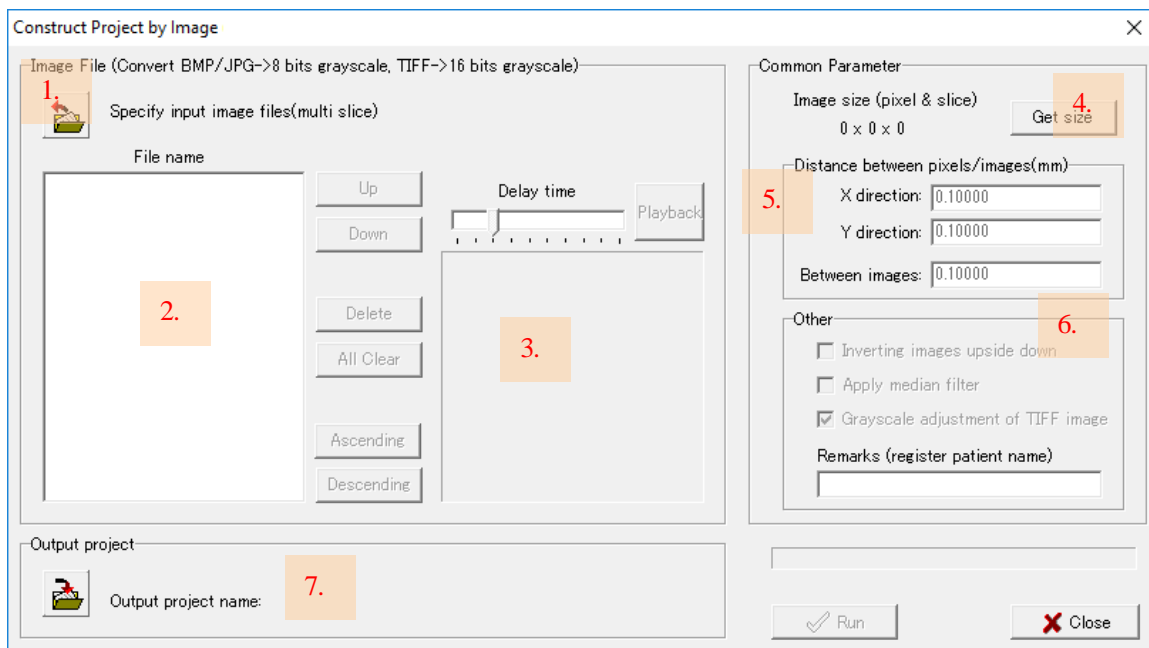


    I. Pushing [Specify Input Image File] button of 1 is followed by opening of file selection window in the figure below.
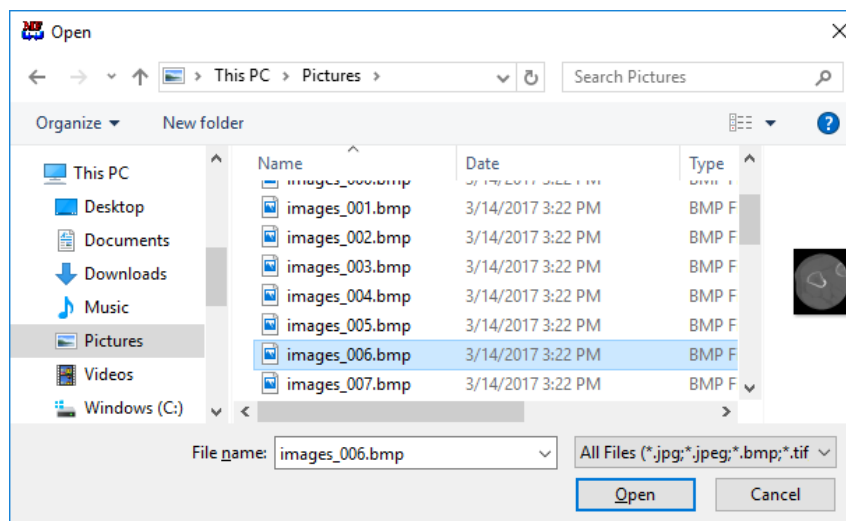


    Specify the directory including the image and then specify the image file in the directory.

    Using the shift key together, multiple files can be specified at the same time.


   II. The specified file is added to the file list area of 2
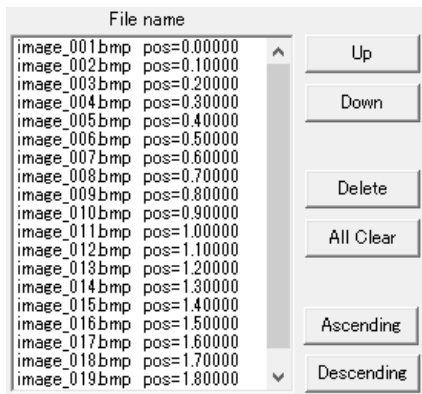
If the images exist in multiple directories, repeat the operation of (I) multiple times.

The multiple files are randomly in line when they are selected.

For three-dimensional construction, a consecutive row of files is needed. Here, an operation that arranges the image files in the file list into the correct row is carried out.

If the file names are in line in ascending or descending order, change the file row by [Ascending] button or [Descending] button. Otherwise, click inside the file list, then push [Up] button or [Down] button to change the file row. If there is an unnecessary file, click inside the file list, then push [Delete] button. If all files are needed to be deleted, push [All Clear] button.

*Normally, when the names of the imaged slice images are in line from bottom to top in ascending order, changing the file row with [Ascending Order Sort] button enables correct three-dimensional construction.

III. The area of 3 is a preview area to confirm the image file clicked in the file list and the consecutive image files by [Playback] button.



IV. With [Get size] button of 4, the volume size after three-dimensional construction can be confirmed.



Normally, each image file may have the same longitudinal and lateral size, and if the size of each image file is different, the size is adjusted to the size of the minimum image. In the above example, the final volume size is $500 \times 500 \times 19$.

V. At 5, an actual dimension of the volume is specified. This value determines the size at three-dimensional construction. The setting parameters at the area are as follows.



"X direction" and "Y direction" are found from the dimension of the image, and "Between Images" sets the dimension between the images.

For example, if an image is in the following case:



Y direction - 500 pixels
- Imaged range 60 mm

X direction - 500 pixels
- Imaged range 60 mm
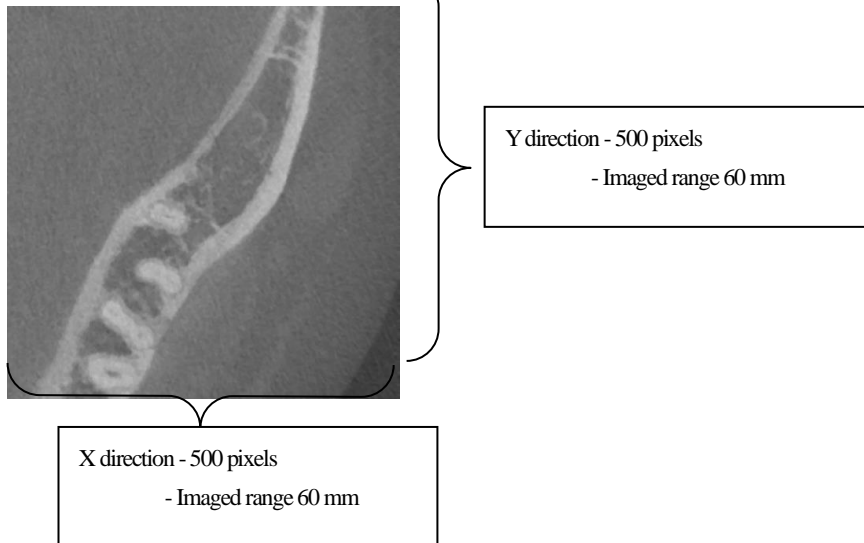
In this case, a distance between pixels that is a parameter of X and Y directions can be found by the following formula.

Distance between pixels in X and Y directions = Range length (mm) / (Number of pixels - 1)

Therefore, the setting value of the example is 0.12024...

On the other hand, enter a distance between adjacent images to the distance between images.

This tool only handles image files imaged at even intervals

(Image files imaged at uneven intervals do not convert correctly.)
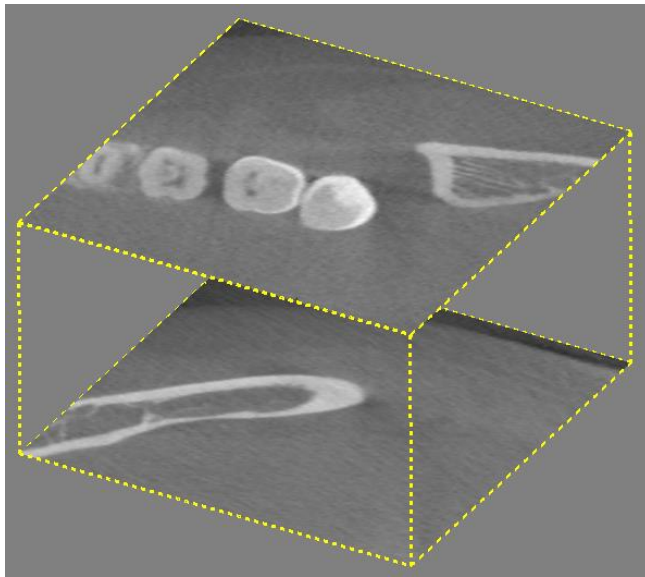
- Number of all images 19 sheets
- Distance from the lowest layer image to the highest layer image 36 mm

In this case, the distance between images is input, so it can be calculated by using the following formula.

Distance between images ＝ Range length of all images (mm) / （Number of images－1）

Therefore, the setting value of the example is 2.0.



(In the image example in the previous page, enter the above value as a parameter.)

VI. At 6, setting such as filtering can be performed.



If the image is an inverted image in the case of three-dimensional construction, turn on "Construct Image Upside down." Further, for an image with much noise, noise can be reduced by turning on "Perform Median Filter."

*There is a possibility that an image becomes inverted in right and left or in top and bottom when three-dimensional is constructed by this tool.

Make sure to perform a three-dimensional display by "CT Range" in the main menu to confirm that the image is not inverted.

VII. At the above settings, specify a file name of the file to be saved as a project file by pushing the button of 7.
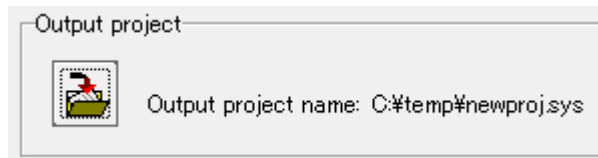


By pushing [Run] button, the file name is converted to the specified project file name.

(It may take much time depending on the number of images.)



VIII. Ended. End the tool by [Close] button.

## 14.2 Output Project Data in Text

This tool can output a content of the project file as ASCII (Text)-format file. The tool is required when secondary use or confirmation of the analysis condition are performed.
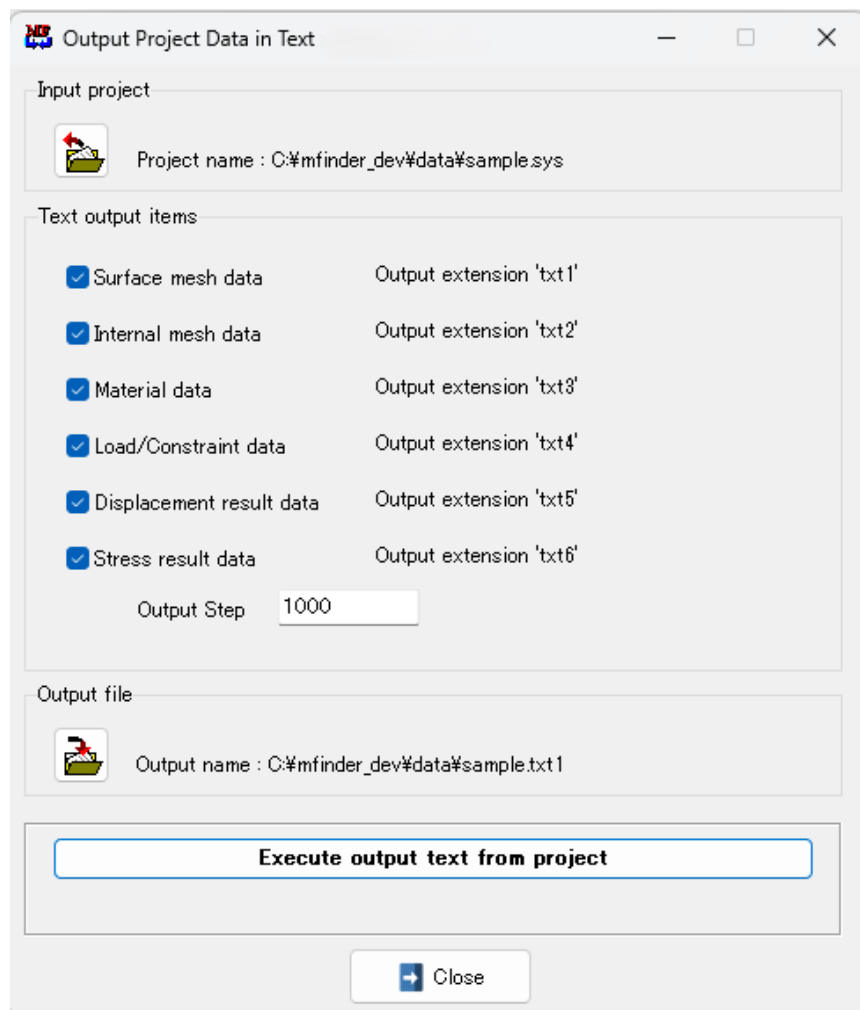


| Icon | Function |
|---|---|
| Input Project | Specify the project file.<br>Specification enables operation of items that is capable to be output. |
| Text Output Item | Checking an item to be output makes the item subject to output.<br>An extension is automatically attached. |
| Output Step | Specify the step number of the analysis. Just the result of the specified step is output. If the bigger number than the maximum step number of the analysis is specified, the result of the final step is output. |
| Output File | Specify the file name to be output. |
| Run | Pushing [Execute output text] button makes the file output. |
| Close | Close the tool. |

## 14.3 Inhomogeneous Material Editor

This tool can generate conversion formulas of "Density -Material" for inhomogeneous material is selected at material properties.



| Icon | Function |
|---|---|
| Series Name of Conversion Formulas | Combined conversion formulas of material properties (Young's modulus, yield stress value, critical stress value, Poisson's ratio, stress relaxation factor) are named as the conversion formulas series. <br> Definition of all material properties is not necessary. <br> Some series names of conversion formulas have been already registered and they cannot be modified. <br> - New series registration <br>   Set a name to be newly registered as a series name of conversion formulas. Pushing the button, the figure below appears. <br>  <br>   Enter the series name and file name, then register them. <br> - Deletion of selected series <br>   The selected conversion formulas series is deleted. |
| Graph, etc. | The conversion formulas of the material components specified by "Select Item" of the selected series name appear. <br>                 Auto range   ...   The maximum and minimum of the conversion formula is automatically |

| | | |
|---|---|---|
| | | adjusted to the graph area. |
| | X-axis, Y-axis range.  ... | When "Auto Range" is OFF, the graph range can be manually entered. |
| | Exclude non-fracture p.  ... | The non-fracture area of yield stress and critical stress do not appear in the graph area. |
| | X-axis, Y-axis unit  ... | A basic unit of the graph display can be selected. The value has the same unit of the value used in "Edit Work Area." |
| Selected Item | Select the material properties to be added or edited and displayed. The defined material properties appear in red. The item at work in progress and not reflected is indicated by black background and red character | |
| Edit Work Area | This is an area where the material properties specified by "Select Item" in the selected series name are added, modified, and deleted. The procedure is different between "Input Values" and "Input Equation". In "Input Values", conversion formula is set as discrete values by entering the pair of density value and selected material property value. An outline of this project is as follows. 1) By entering the density value and the value of the selected material properties and pushing [Add to List Items] button, the values are added to the list. 2) If deletion of an item of the conversion list is necessary, after the specification of the item in the conversion list, push [Delete from List] button. 3) If modification of an item of the conversion list is necessary, after the specification of the item in the conversion list, modify the value, then push [Replace item] button. 4) If referring to conversion formula of other series followed by reflecting it to the conversion list is necessary, push [Reference] button to select the desired series name. 5) If setting yield stress and critical stress to non-fracture area (1.0e20 MPa) is necessary, push [non-fracture] button. 6) To reflect the conversion formulas generated by the conversion list, push [Update list] button. At the time, if no item exists in the conversion list, registration of the conversion formula of the material component is canceled. In "Input Equation", the parameter of conversion formula is set expressed in the form F(D)=a*D^b+C (Dmin < D) An outline of this project is as follows. 1) Enter the parameter Dmin, a, b, c and inequality sign, and press "Add to list" button. Then, the contents are added to Parameter List. 2) If you would like to delete the item in Parameter List, select the item in the list and press "Del from list" button. 3) If you would like to modify the item in Parameter List, after selecting the item in the list, modify the parameter and press "Replace item" button. 4) To reflect the conversion formula added in Parameter List, press "Update list" button. | |
| Close | Close the tool. | |

## 14.4 Material Database Tool

This tool can add and modify the definition (Young's modulus, yield stress, etc.) of material for homogeneous material in "Material Properties."



| Icon | Function |
|---|---|
| Registered Uniform Material Database | Materials already registered appear in this list.<br>Since it is controlled for each "Material data name," the same "Material data name" cannot be added<br>Further, already registered "Basic material data" cannot be modified. |
| Operation Method | New registration method<br>- After entering from "Material name" to "Crushed strain," push [Add] button for new registration.<br>   The same material data name cannot be registered.<br>Method of deletion<br>- Select the material to be deleted from "Registered material database" and push [Delete] button to delete.<br>   "Basic material data" cannot be deleted.<br>Method of modification<br>- Select the material to be modified from "Registered material database," and after the modification of the registered content, push [Change] button to change the registered content.<br>   "Basic material data" cannot be modified. |
| Close | Close the tool. |

14.5 Batch Program (Mesh /Analysis)

This tool can perform process the project files in which "Batch Process" is turned on at the time of mesh generation and analysis in the background.

Multiple project files processed in a lump can be specified, and any order of processing can be specified.

The projects of both mesh generation process and analysis process can be mixed in the list.

Further, while processing, a new project can be submitted.

In order to process time-consuming mesh generation and analysis process in a lump, project can proceed efficiently by utilizing while PC is idling.

**\*When this tool is used, starring of the main menu of "MECHANICAL FINDER" is not necessary.**



| Icon | Function |
|---|---|
| Setting of Input Project | An outline of this procedure is as follows. <br> 1) Push [Input Project] button, followed by specifying a project file using the file browser. <br>   If the project can be analyzed, [Add] button becomes available. <br> 2) Push [Add] button to add the file to "Run Projects." <br> 3) Repeat the procedure above to add the project file to be batch processed to the list. <br> 4) Delete unnecessary project file by selecting the file from the list by clicking and then pushing [Delete] button. <br> 5) If changing the order of the projects to be analyzed is necessary, select it from the list by clicking and then push [Up], [Down] buttons. |
| Process Run | After setting the project to which the batch process is performed is completed following the procedures above, run the process by [Run] button. |

142

|  | The progress of the running appears in "Run Result Log." |
|---|---|
| Stop | When stopping the batch process, push [Stop] button.<br>Once "Stop" is carried out, processes for the project running currently and the subsequent project are canceled. |
| Close | Close the tool.<br>If a running process exists, whether the process of the project will be canceled is asked.<br>*If a continuous processing of the project running or waiting for processing is necessary, ensure that this tool must be started. |

*This tool repeats the following procedures.

    1) Read the project file and perform mesh generation process or analysis process.

    2) When normally ended, the original project file is rewritten.

    3) When abnormally ended, the original project file is not rewritten.

    4) If a subsequent project exists, return to 1) to perform processing.

14.6 Remote Batch Program

This tool can perform analysis process in a remote terminal of the project files in which "Batch Process" is turned on when analyzing.
Multiple project files processed in a lump can be specified, and any order of processing can be specified.

The analysis process is performed in the remote PC side while work of "MECHANICAL FINDER" is being performed in the main PC, thereby efficiently generating a project file.
Also, the advantage includes usage of the remote PC with faster CPU speed.

**\*When this tool is used, starring of the main menu of "MECHANICAL FINDER" is not necessary.**

[Running image figure]
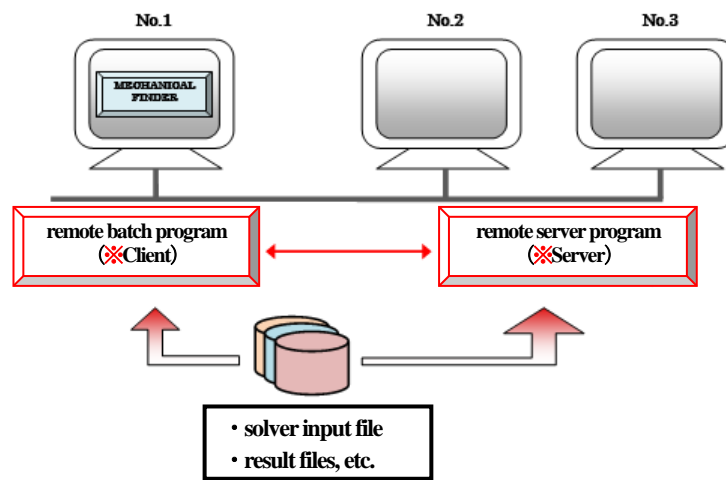


[Explanation of the image figure]

- PC with a license of MECHANICAL FINDER --------- NO. 1
- PC on the same network (no license necessary) --------- NO. 2, NO.3

[Operation method]

1) At NO. 2 and NO. 3, start "Remote Server" (Sever).

2) At NO. 1, start "Remote Batch Program" (Client)

3) At NO. 1, register and run the project group to be analyzed.

4) [Automatic] The project group of NO. 1 is transmitted to NO. 2.

5) [Automatic] The received project group is analyzed in order on NO. 2.

6) [Automatic] When normally ended on NO. 2, the project on NO. 1 is replaced.

[Case of executing analysis on NO. 3]

7) At NO.1, disconnect from NO. 2 and connect to NO. 3.

8) At NO.1, register and run the project group to be analyzed.

9) [Automatic] The project group of NO. 1 is transmitted to NO. 2.

10) [Automatic] The received project group is analyzed in order on NO. 3.

11) [Automatic] When normally ended on NO. 3, the project on NO. 1 is replaced.

12) After the completion of all analysis, the tools on the client side and server side are ended.

## 14.6.1 Client Side Program

This tool is a client side program of the "Remote Batch Program."
The tool runs on PC with a license of MECHANICAL FINDER.

The tool performs analysis process in a remote terminal in a lump of the project files in which "Batch Process" is turned on when analyzing.
Multiple project files processed in a lump can be specified, and any order of processing can be specified.

When this tool is used, the program on the server side (remote) shall be started.
For installation and usage method of the server side, refer to "14.6.2 Server Side Program" and "14.6.3 Server Side Installation."

*When this tool is used, starring of the main menu of "MECHANICAL FINDER" is not necessary.
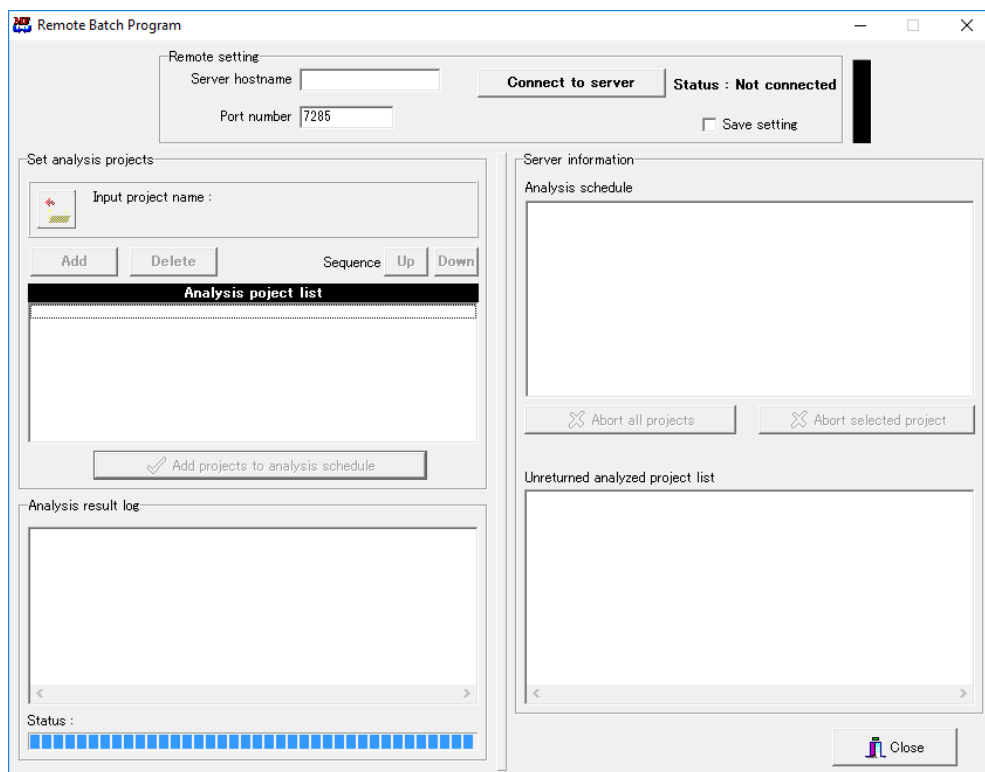


| Icon | Function |
|---|---|
| Remote Setting | Specify settings of the server side.<br><br>Server host name   ...    Specify an IP address or host name.<br><br>Port number   ...    Specify the port number (default 7285) specified by server side.<br><br>Connect to server   ...    Connect to the server side using the above settings.<br><br>Disconnect from server   ...    With connecting to the server "Connect to server" button changes to "Disconnect from server" button, and clicking it disconnects from the connected server.<br><br>Save setting   ...    Ending the program with check makes the above settings the initial |

145

| | |
|---|---|
| | value on the next start-up. |
| | *To perform analysis process, at first, connection to the server side using the above settings is necessary. |
| Setting of Input Project | Procedure |
| | 1) Push [Input Project] button, followed by specifying a project file using the file browser. |
| |     If the project can be analyzed, [Add] button becomes available. |
| | 2) Push [Add] button to add the file to "Analysis Project List." |
| | 3) Repeat the procedure above to add the project file to be batch processed to the list. |
| | 4) Delete unnecessary project file by selecting the file from the list by clicking and then pushing [Delete] button. |
| | 5) If changing the order of the projects to be analyzed is necessary, select it from the list by clicking and then push [Up], [Down] buttons. |
| | 6) After the completion of setting using the procedures above, run the program by [Add project to Analysis Schedule] button. |
| | The registered information appears in order on "Server Information." |
| Analysis Run Result Log | An analyzing state and a state after analysis results appear. |
| Server Information | On "Analysis Schedule," schedule information on analyzing or waiting for analysis appears. |
| | "Abort All projects" forcibly ends the analysis schedule sent by the client and analyzing project. |
| | On "Analyzed Project list (unreturned)," an unreturned project after analysis ended appears. |
| Close | Close the tool. |
| | All projects on analyzing or added to the analysis schedule sent by the client are forcibly ended. |

14.6.2 Server Side Program

This tool is a server side program of "Remote Batch Program."

No license of MECHANICAL FINDER is necessary; however, the license is necessary on the client side.
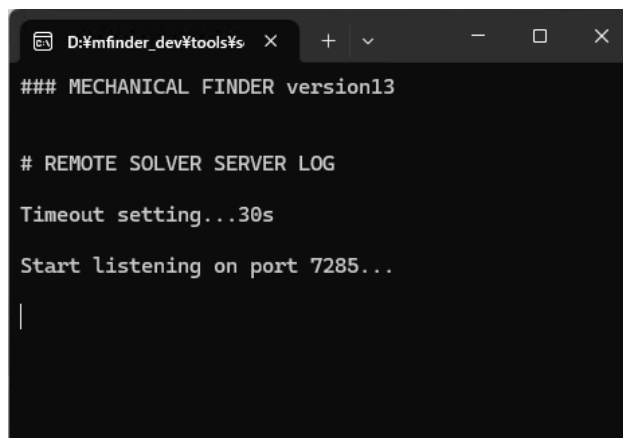
Batch analysis from a maximum of five clients (PC) can be accepted.

However, analysis process is performed one project at a time.

[Start-up and procedure]

| | |
|---|---|
| 1. | Select "MECHANICAL FINDER V13 Remote Server"→"Remote Batch Program (Server)" from the start menu. |
| 2. | Then, the following screen appears. |
| |  |
| 3. | After that, analysis process is automatically performed by commands from the client side. When ending, close the screen by [×] button on upper right. |

If changing the port number at start-up is necessary, select "Setting File" from the menu.

After changing the value on the editor and saving the file, the port number is changed at restart.

## 14.6.3 Server Side Install

This tool is installed by the procedures below.

**(1) Insert the provided CD-ROM into a drive, followed by opening "Server program" in the CD-ROM drive.**
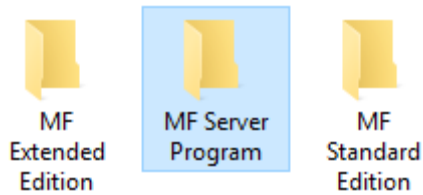


**(2) Run [setup.exe] among them.**



**(3) Proceed in accordance with the procedure shown by the installer.**
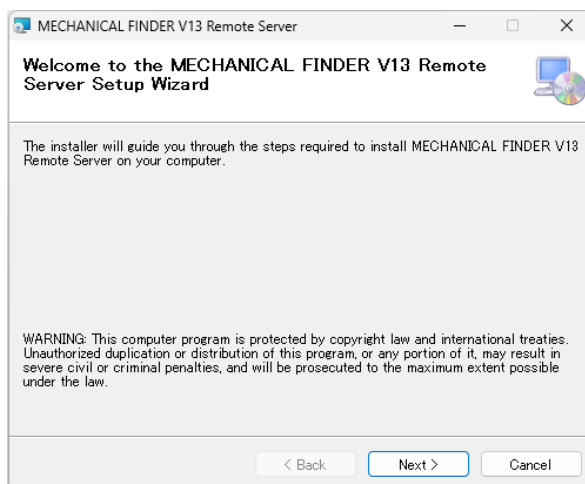


The installation work is started.
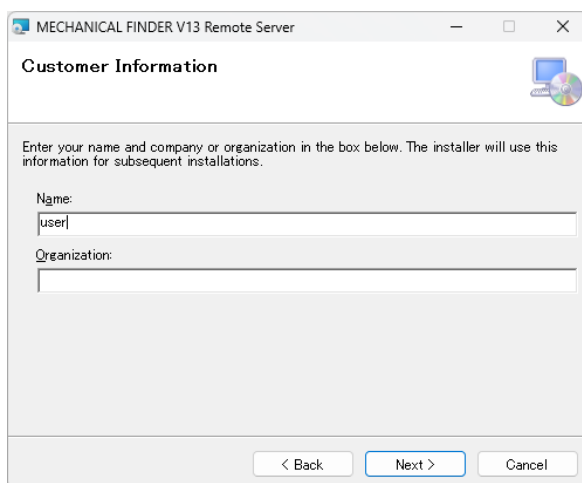


Enter "User Name" and "Organization."

148
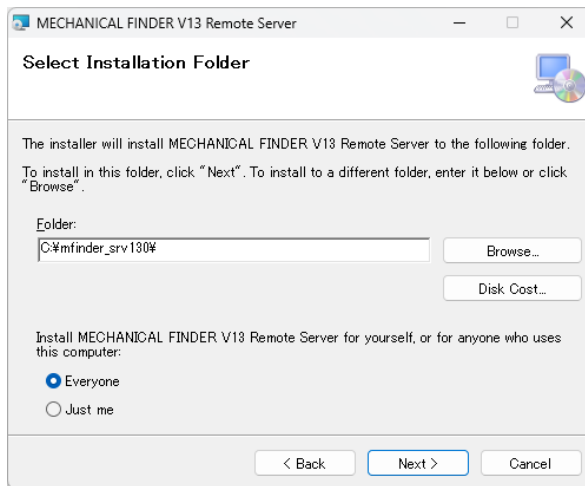
Select an install directory.
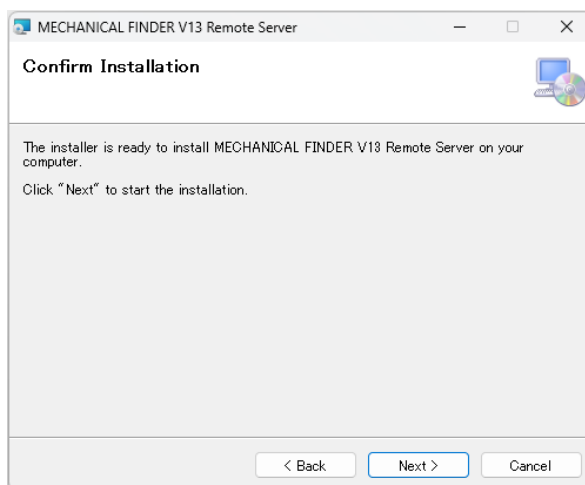


Confirm the information.

**It is copied to the disk.**
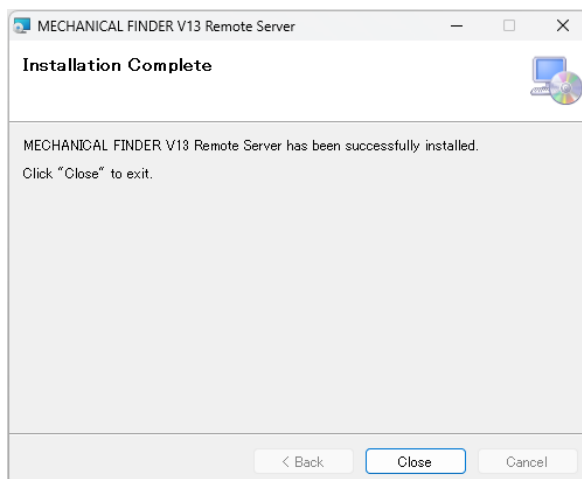


Ended.

14.7 Project Manager

This tool can perform the following operations to project file (Maximum of 14 files/1 project) of MECHANICAL FINDER:

- Change of the project name
- Copy and move of the project
- Deletion of the project (to Recycle Bin)
- Deletion of patient information of the project (for protection of personal data).

**\*The "Deletion of patient information of the project" function of this tool deletes "patient name," "date of birth," and "identification number," which are patient information of the project.**

   **Once deleted, the above information cannot be obtained from the project.**
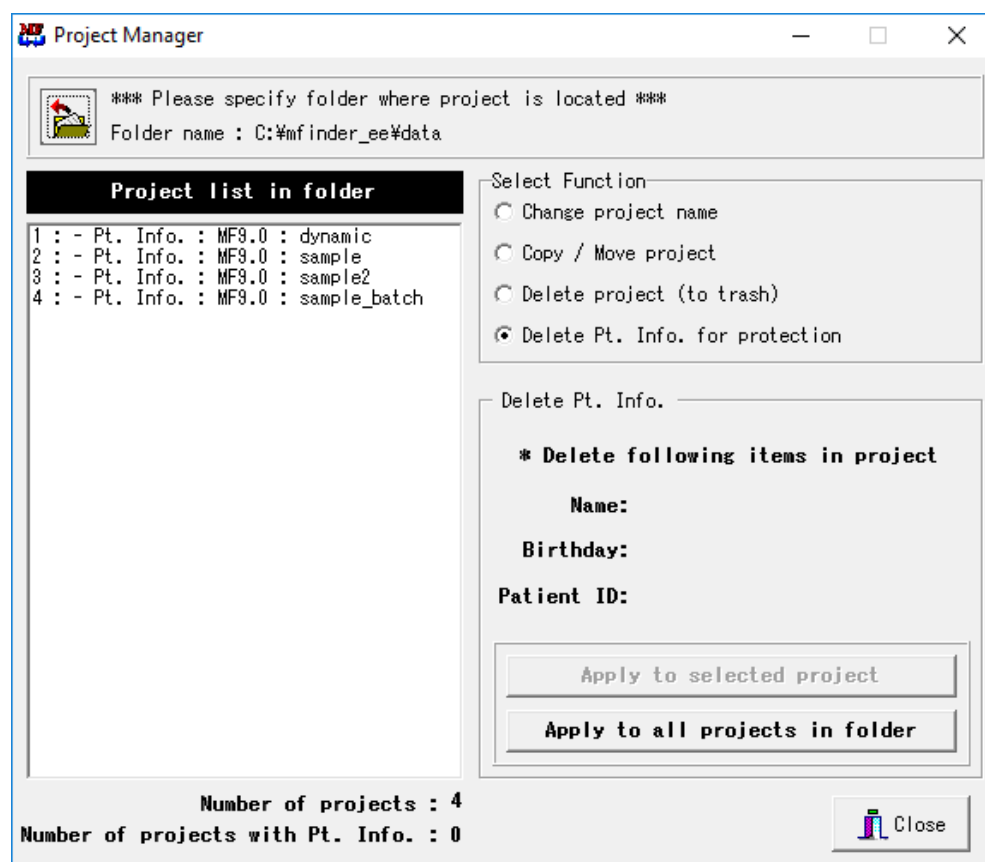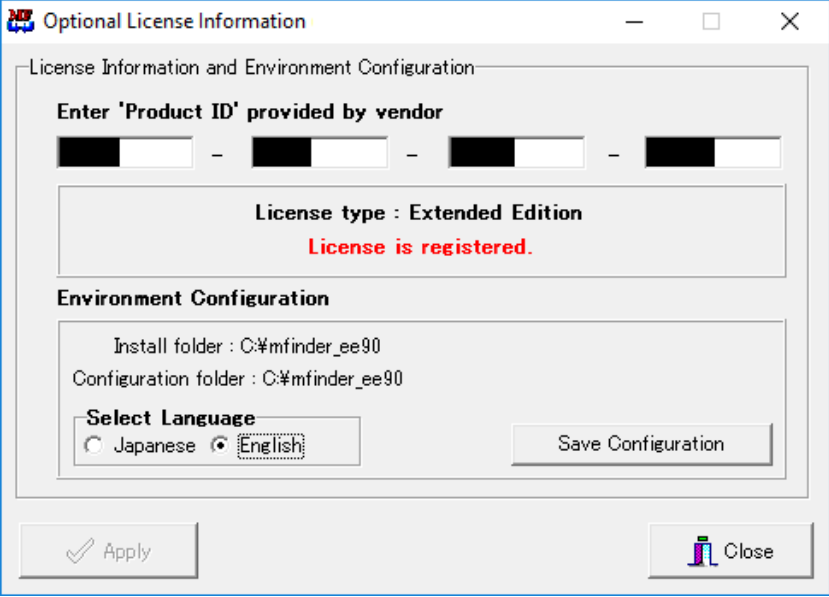


[Operation procedure]
１．Select the directory with projects.
２．On the list of projects, projects inside the directory appear.
３．Select a function to be performed (modification, copy, deletion, deletion of patient information, etc.)
４．Select the subjected project from the list of projects.
５．Perform operation using item input and apply button to run the process.

## 14.8 Optional License Information

This tool performs registration of an optional license of MECHANICAL FINDER.

*Normally, this tool is not used except installing.

# Chapter 15 IMP Utility (EE)

In this chapter, a utility generating IMP format file, which is an extended function, is explained.

At the extended edition, an IMP format file generated by the IMP utility can be inserted as an imported geometry at "Chapter 6 Mesh."
The IMP format file is semi-compatible with the STL format, which is a generic format, and its function can be expanded so that the file is easier to use than the STL format file at setting work of the import position.

**- Problem in import process**

Usage of geometry of the STL format in the process of "Chapter 6 Mesh" includes the following problems:

Importing a geometry that does not have axial symmetry (rotators) such as the stem shape of (α) in the figure below followed by positioning requires much labor to locate the shape inside the bone since the center of rotations does not match the stem axis.

Accordingly, a unique IMP format that can specify any rotation axes and indicators, etc. keeping the semi-compatibility with the STL format is prepared. Using this, when such asymmetrical shape is imported, it becomes much easier to locate the shape.

IMP format has the information about the center of rotation and location of indicator in addition to the STL geometry as shown in figure (β) below.

Further, the format has semi-compatibility, and conversion from the IMP format to the STL format only needs a modification of the extension; thus, almost all software can handle the format. (However, only the STL binary format has semi-compatibility.)



| (α) STL import state | (β) IMP import state |
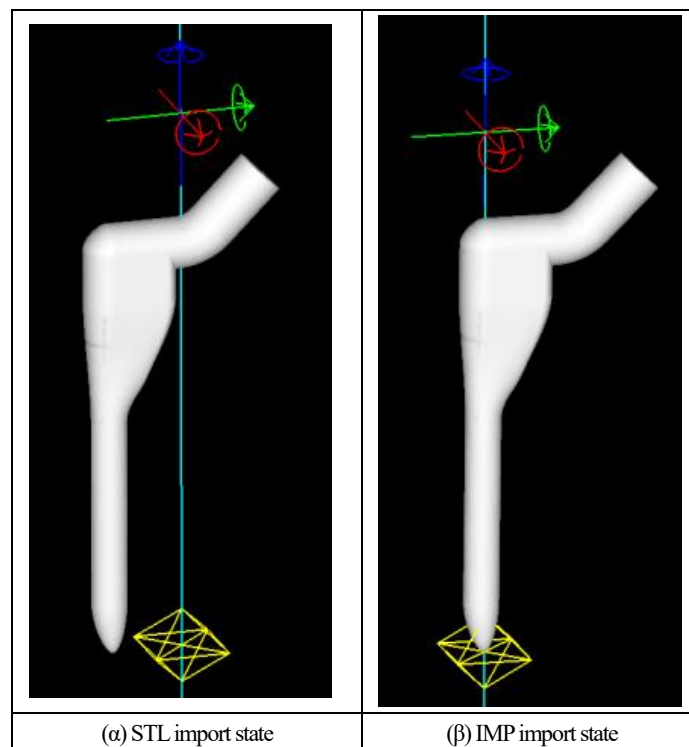
## 15.1 Start-up of IMP Utility

If MECHANICAL FINDER has already started, the IMP utility can always be started from the option menu as shown below.
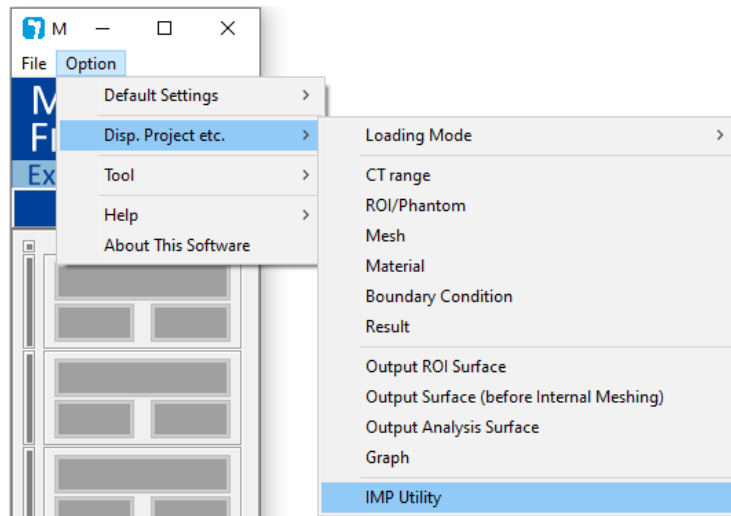


After the start-up, the screen shown in the figure below appears.

Refer to the operation method in the following pages.
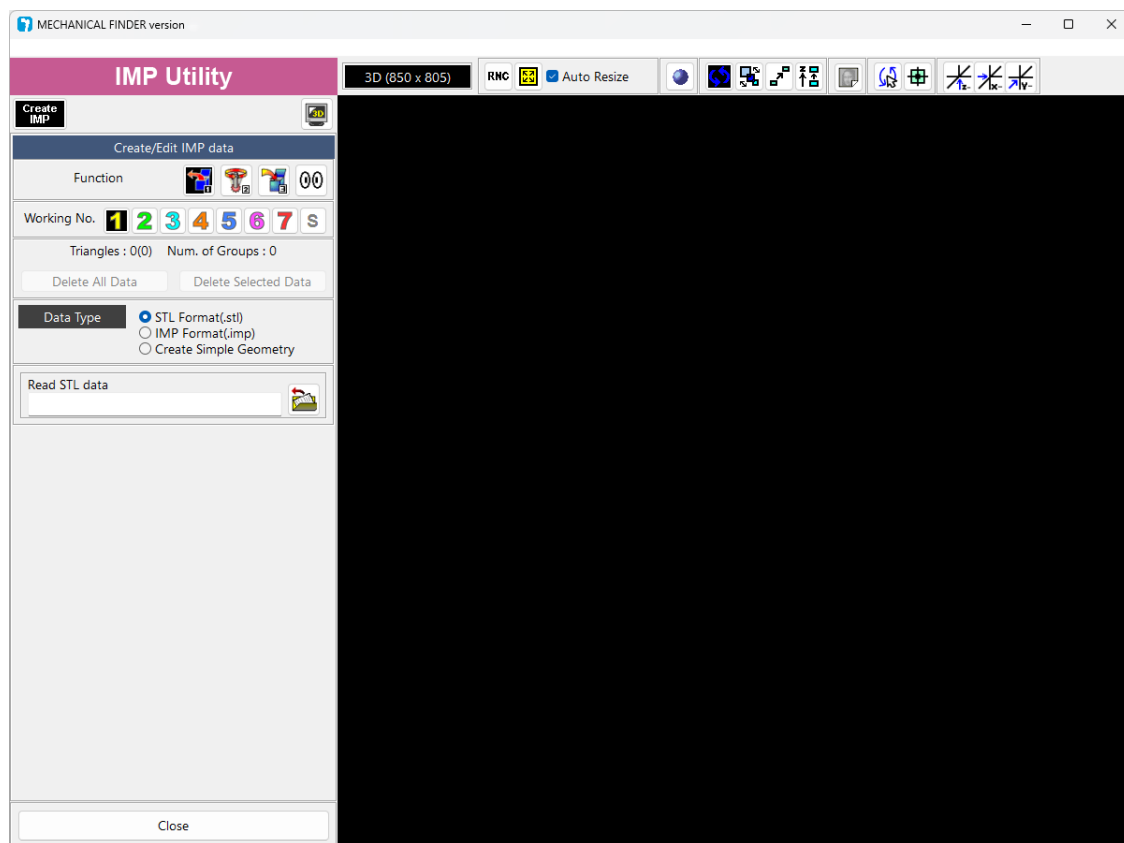


| Icon | Function |
|------|----------|
| Create IMP | For details, refer to 15.2. |
| 3D | Viewer Settings |

## 15.2 Conversion to STL Format File

The generated IMP format file has semi-compatibility with the STL format file.

Conversion for this IMP format file to the STL format file only needs modification of the extension.

| File name before changing | File name after changing |
|---|---|
| abcdefg.imp | abcdefg.stl |

Modification of the file name by changing the extension from "imp" to "stl" using the file explorer in Windows in a same manner above allows almost all application to handle the file.

However, in this case, the STL format file is a binary format; thus, an application that can handle only ASCII format cannot read the file. Ensure that the application used can handle the STL file with a binary format.

## 15.3 How to Use Expansion Process in IMP Utility

An expansion process exists in the IMP utility.

This is a function that expands or shrinks a shape with a constant thickness.

By using this, you can easily generate thin shape such as below.
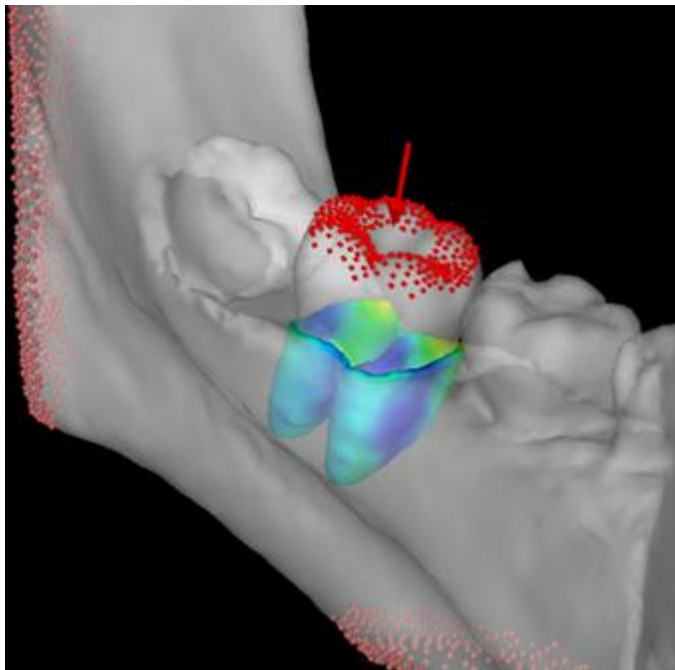

(Periodontal membrane with 0.5 mm thickness)

This is a process that the IMP utility, shown in the figure below, outputs two shapes, one is an original shape and the other is an expanded shape, as the IMP format files, then they are imported at the mesh generation process of "Surface Mesh and Import Settings."

Since the expanded shape is larger than the original size, in the material sorting of "7.1 Material Setting," it is necessary that an unnecessary mesh area portion appearing on the surface (among expanded areas, a portion other than Periodontal membrane) is made to be unused material.
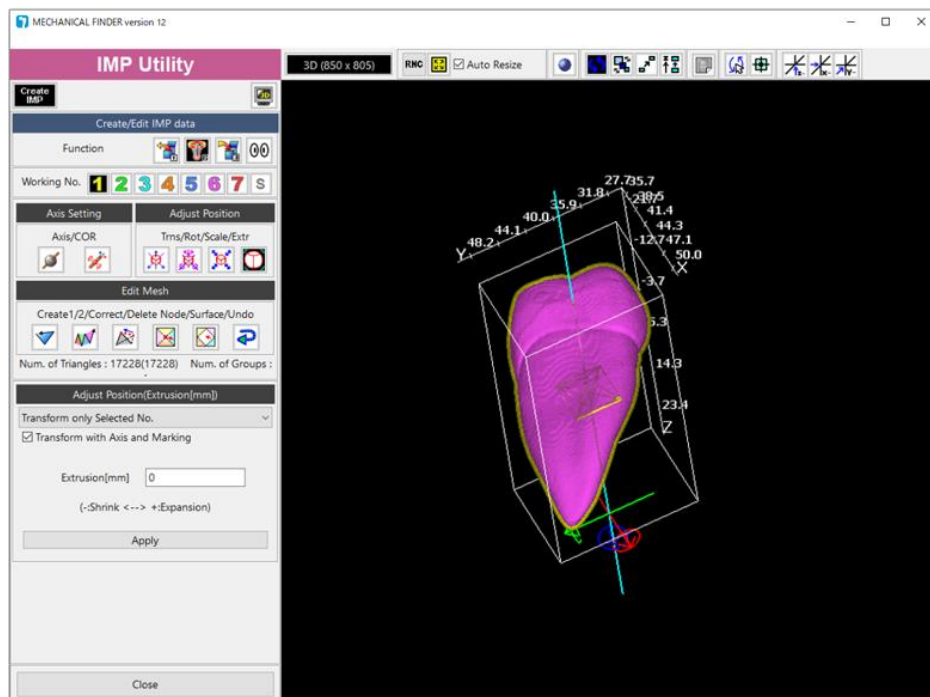
## 15.4 Auto Resize of IMP Utility

In "IMP Utility," camera is auto resized when you replace STL data, move the object in some direction, etc.

If you avoid that, please turn "Auto Resize" off.
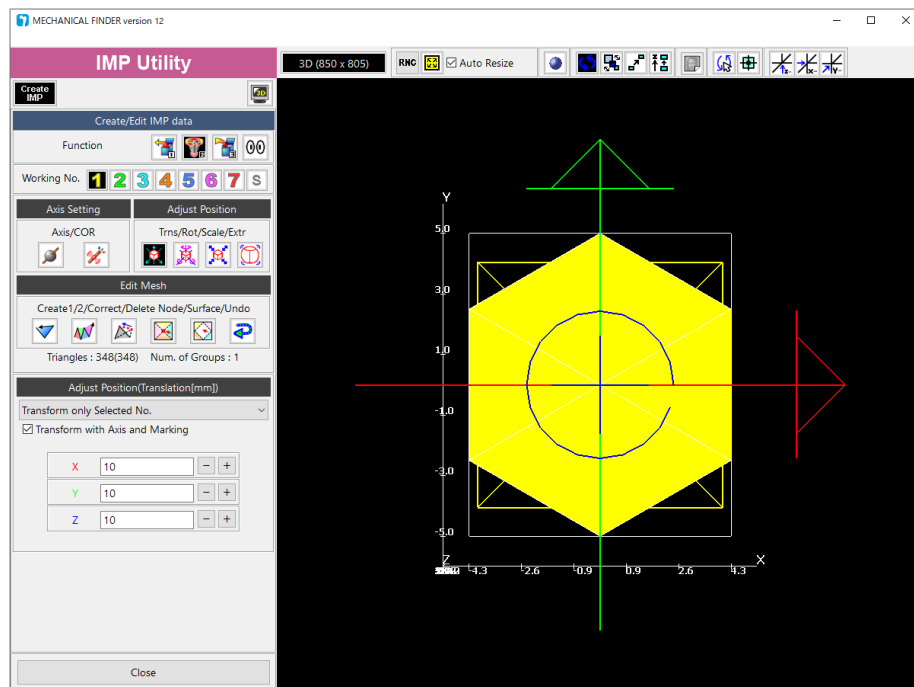




Move to X+ Direction

Auto Resize: ON
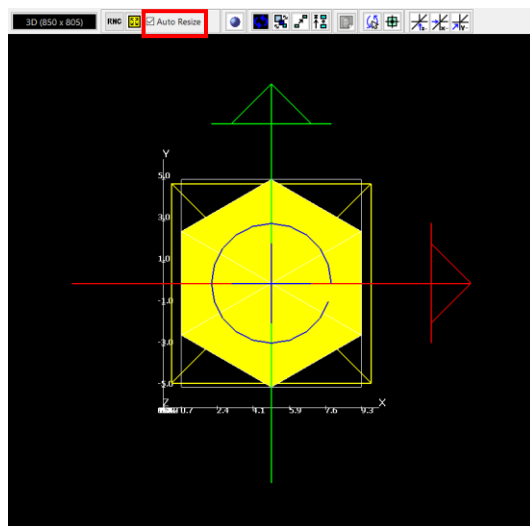
Auto Resize: OFF
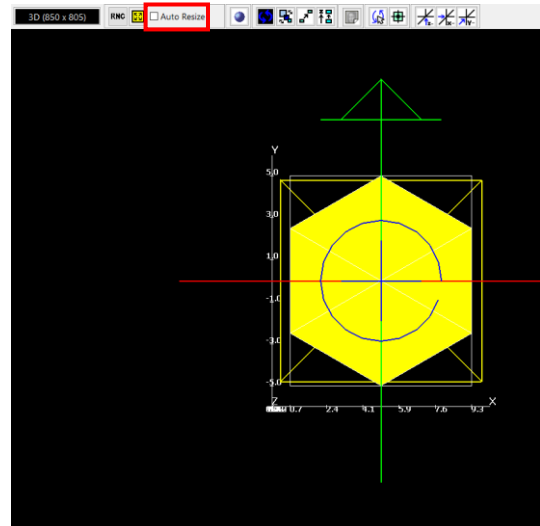
# Appendix 1 Important Notice

An important notice when using this software is explained.

Also, refer to "Appendix 5 FAQ"

### - Project

In this software, the result of the analysis in the past is managed by project.

1. About the project file group

A project file comprises a maximum of 15 files.

When moving to other directory or backup, operate these files as a whole.

"14.7 Project Management Tool" is also available.

2. Save project

After the modification of an analysis model or condition, if you need to keep the project before the modification, save the modified project as other project.

### - Analysis

1. In material non-linear analysis, decrease of the load value against the time axis can be set; however, once crush or a crack occurs, the element will not be recovered. But plastic element can return to elastic condition by decreasing the load value. Fatigue failure is not considered.

2. If the solver type is the direct method, comparing with the CG method, usage of disk and memory becomes temporarily larger.

Further, <direct method (Multi core)> in the direct method can process faster and <iteration method (CG method/GPU)> in the iteration method can process faster; however, both two will be out of operation when the memory is insufficient. In such a case, try <direct method (Single core)> <iteration method (CG method)>.

For details, refer to Q2 in "Appendix 5.5 About Analysis."

### - Display

1. "Movie Output," which is a function of the display viewer, saves images in "Memory" or "Disk" every time when display screen is refreshed after the recording has been started.

The process will continue unless the recording is stopped; thus, finally the system may be stopped. Make sure not to forget to stop the recording.

Appendix 1.1 About GUI Operation

(1) Input area



**To reflect the input area (value, character) in the user interface of this software, press to the return key after the input, or move the focus.**

(2) Mouse operation of display viewer
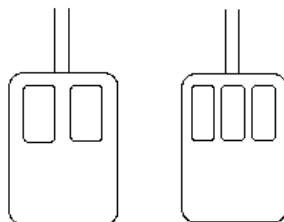


Two-button mouse  Three-button mouse

| Mouse Button | Operation |
|---|---|
| Left Button | Operation of the specifying mode(The initial value is rotation)<br>Refer to the left mouse operation in "Viewer Settings."<br>- +Alt ・・・ Rotation<br>- +Ctrl ・・・ Move<br>- +SHIFT ・・・ Scale up/Scale down |
| Middle Button<br>(Three-button mouse compatible) | - Rotation<br>- +Ctrl ・・・ Move<br>- +SHIFT ・・・ Scale up/Scale down |
| Right Button | It is used when specifying an object locally.<br>Binarization process of "ROI extraction."<br>When specifying the ROD position of "Phantom Setting."<br>When drawing various kinds of graphs for the shell element of "Material properties."<br>When specifying ranges of load and constraint of "Boundary Condition" or setting of the axis.<br>When specifying the slice position of the arbitrary surface of "Contour display."<br>When specifying nodes and cell elements of "Data extraction." |

Appendix 1.2 About Window Buttons

On each setting window, functions used frequently are disposed on the upper part of the viewer.
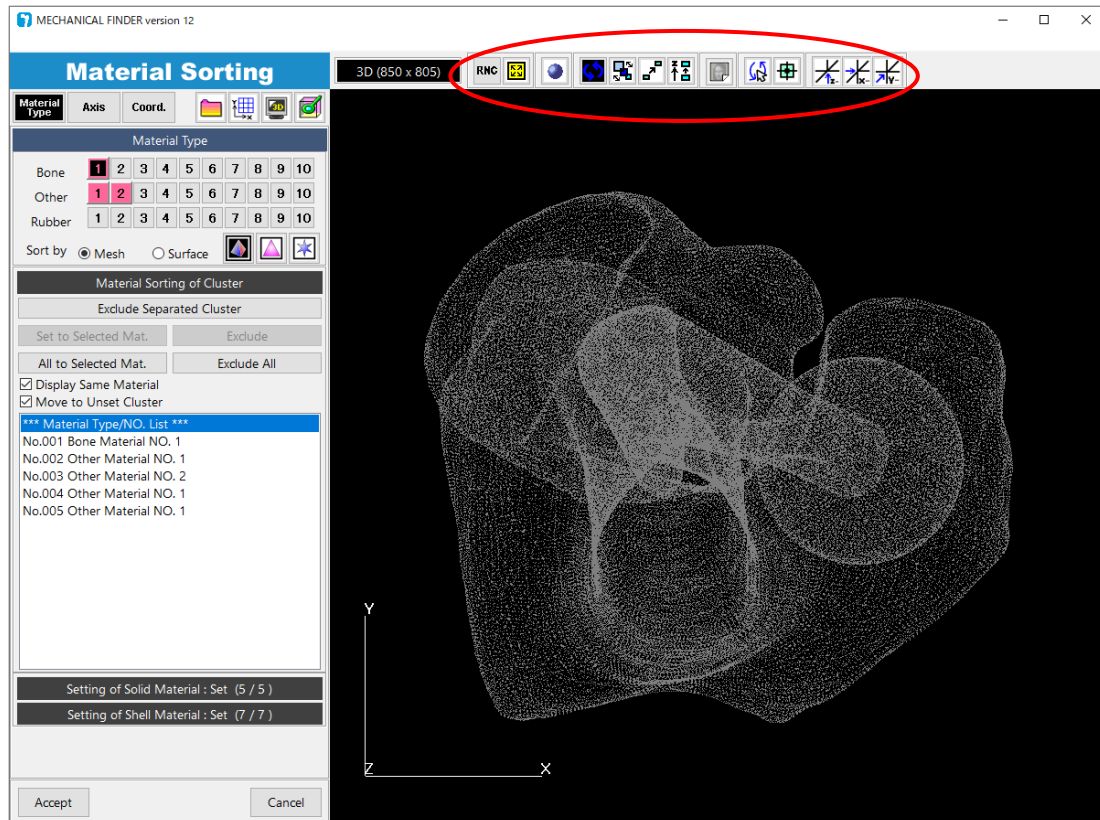


"Viewer Settings"  in each window has a function similar to the button.

As shown below, the menu item for the two-dimensional display is slightly different from that of three-dimensional display.

**- For the two-dimensional display**
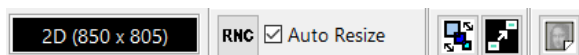


**- For the three-dimensional display**



In the three-dimensional display the button  switches the display of the bounding box instead of the geometry. Use the box when operation of rotating, scaling up and down, moving is heavy due to a large model.
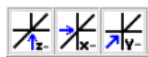
 changes the camera direction, and the direction +/- is switched by click. 

 is the function that outputs an image continuously every time when it is pushed.

"Output Image" in "Viewer Settings" determines a file name output continuously by the parameter.

 is the function that set and reset the center of rotation. Select the left-side button, and the center of rotation is set to the point of right click. Select the right-side button, and the center of rotation is reset to the center of the model.

Appendix 1.3 About Cache Size

The cache size represents memory size used for rendering a display.

The initial value of the cache size can be set by "Application" and is set to 256 MB at the installation.

If the large object whose size exceeds the cache size setting value is displayed with rendering method, operation response such as rotating, scaling up and down, etc. is influenced with extensively slower display speed, since the cache is not used.

If the size of the object exceeds the cache size, a warning window as shown in the figure below appears.
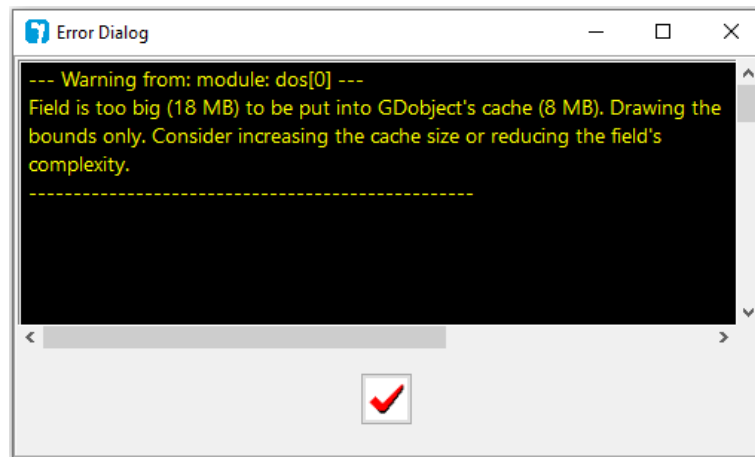


The warning in the figure above indicates that the cache size larger than 18 MB is required to display the object.

However, required cache size varies depending on the display means (mesh display, flat surface display), etc.

**\*Countermeasure**

**- If it occurs frequently...**

At "Application," setting cache size to be larger is necessary.

- If it occurs temporarily...

A cache size setting item that can apply only to the display function is prepared in the window parameter.

(Refer to the figure below)

Changing the value of the parameter item largely allows to display within the cache size.

## Appendix 1.4 Warning Dialog if closing window

The warning dialog shown below appears when "Cancel" button of main work window is clicked.

If you select "OK," the main work window is closed. Please be careful that the edited items are lost.

If you select "Cancel," the main work window is not closed.
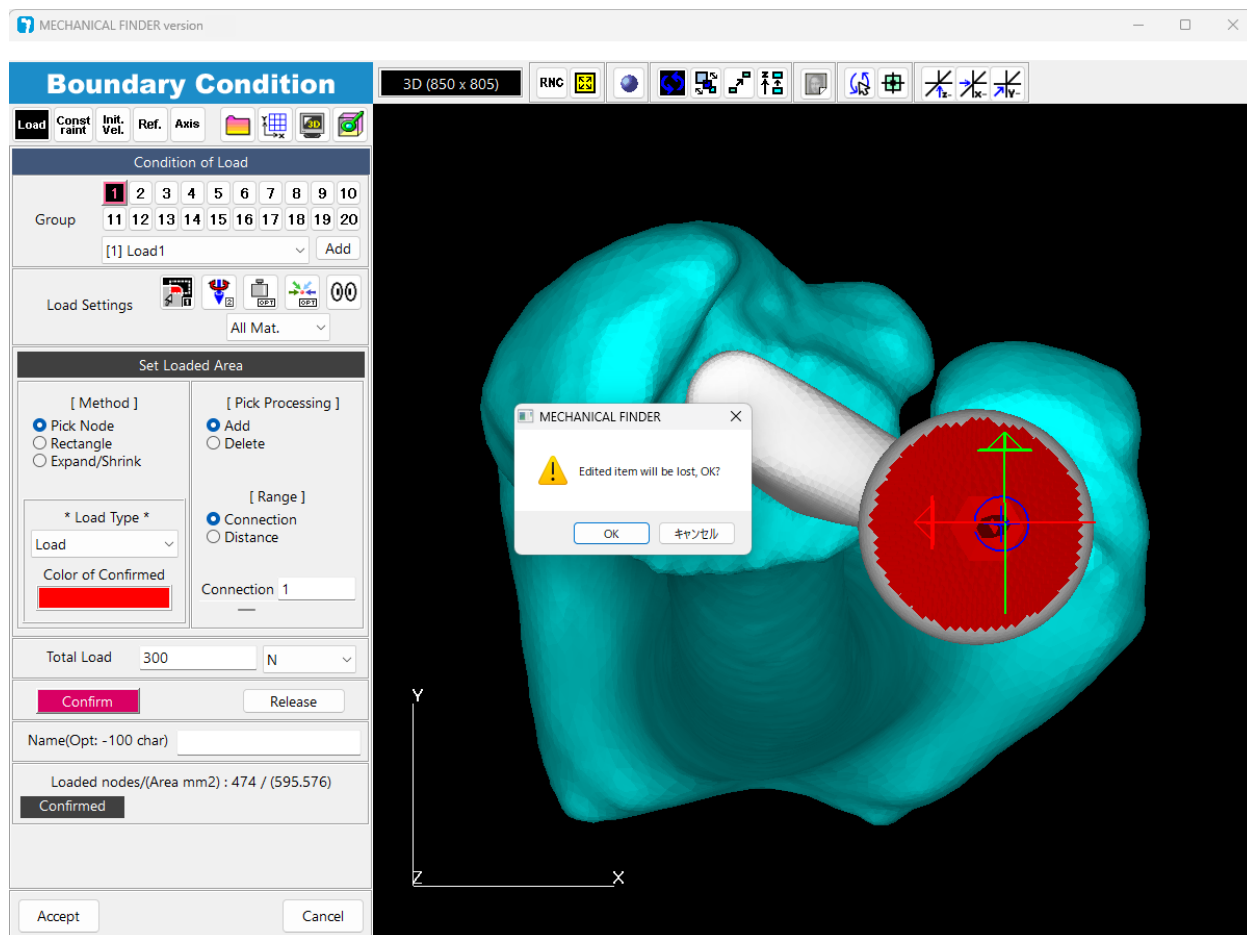
# Appendix 2 Process Flow

Here, the process flow of this software is explained.

### - General process flow

The general process flow of this software is shown below.
1) Open a new project, followed by reading the DICOM file. (Refer to "3.1 'New Project'.")
2) ROI extraction and phantom setting are carried out. (Refer to "Chapter 5 ROI Extraction/Phantom Setting.")
3) Mesh generation is carried out. (Refer to "Chapter 6 Mesh Generation.")
4) Setting material sorting is carried out (Refer to "Chapter 7 Material Sorting.")
5) Setting material properties is carried out. (Refer to "Chapter 8 Material Properties.")
6) Setting boundary condition is carried out. (Refer to "Chapter 9 Boundary Condition.")
7) Analysis is carried out. (Refer to "Chapter 10 Analysis.")
8) Display and investigation of the results are carried out. (Refer to "Chapter 11 Display Function.")
*The series of processes in the above (1) to (8) shall not be performed at the same time. The project can be saved at the step where a process in the middle of the series has been completed, and the sequel can be performed in the next opportunity.
(Refer to "Process flow when restarting the project uncompleted from the middle" in this chapter.)

### - Process flow when restarting the project uncompleted from the middle.

Process flow when restarting the project whose process is not completed to the end is shown below.
1) Open the existing project to be restarted from the middle. (Refer to "3.1 'Open Project'.")
2) Restart from uncompleted process. (Refer to this chapter "Process Flow for General Case.")
3) When the process is progressed, the project should be saved. (Refer to "3.1 'Save Project'.")

### - Process flow with modified model and conditions.

Process flow when the existing project is processed with modified model and analysis conditions is shown below.
1) Open the project whose model and condition are to be modified. (Refer to "3.1 'Open Project'.")
2) If you need to keep the project before the modification, save the project as a new project.
(Refer to "3.1 'Save Project As'.")
3) Restart from the process of the step to be modified. (Refer to this chapter "Process Flow for General Case.")
4) When the process is progressed, the project should be saved. (Refer to "3.1 'Save Project'.")

# Appendix 3 Material Property

Appendix 3.1 About Material Properties of Inhomogeneous Material

In this software, material properties for inhomogeneous material are calculated from density volumes of bone. Here, methods of density calculation and settings of each physical value are explained.

- Density

The density is obtained from the CT value in the ROI area by the conversion formulas

Density conversion formulas are determined by whether the density is "User defined" or "Automatic" or Phantom setting(*).

It is determined by the flow as shown below.
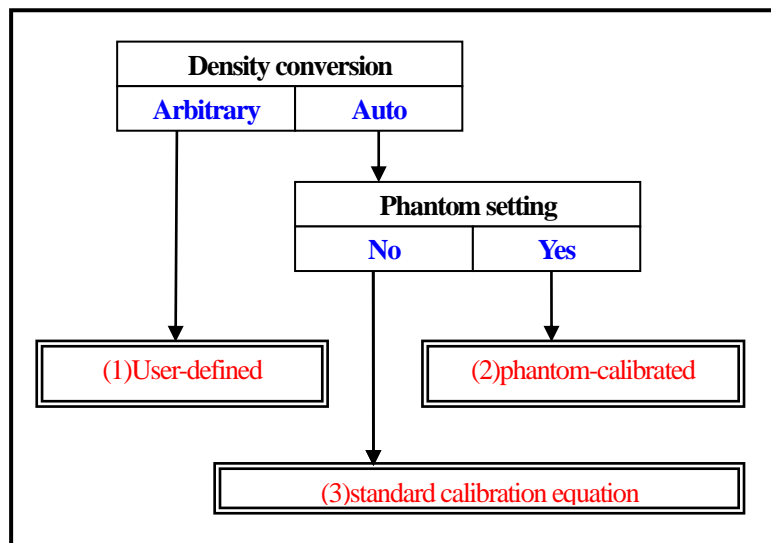


**(1) User-defined conversion formulas**

If "User defined" is selected, the following conversion formulas are applied.

Conversion formula

Density 〔mg/cm³〕 ＝ CT value 〔H.U.〕 * a ＋ b

"a" and " b" are parameter values arbitrarily input by User.

If Density 〔mg/cm³〕 is 0.0 or lower, make Density 〔mg/cm³〕 0.0.

**(2) Conversion formulas with Phantom setting**

If "Phantom setting" is selected when "Automatic" is set, the following conversion formulas are applied.

Conversion formula

Density 〔mg/cm³〕 ＝ CT value 〔H.U.〕 * a ＋ b

If Density 〔mg/cm³〕 is 0.0 or lower, make Density 〔mg/cm³〕 0.0.

(a and b are values obtained by "Phantom Settings")

**(3) Conversion formulas without Phantom (Standard calibration equation)**

If "Phantom setting" is not selected when "Automatic" is set, the following conversion formulas are applied.

Conversion formula

Density [g/cm$^3$] = (CT value [H.U.] + 1.4246) × 0.001 / 1.0580: (CT value > -1)

Density [g/cm$^3$] = 0.0 (CT value ≤ -1)
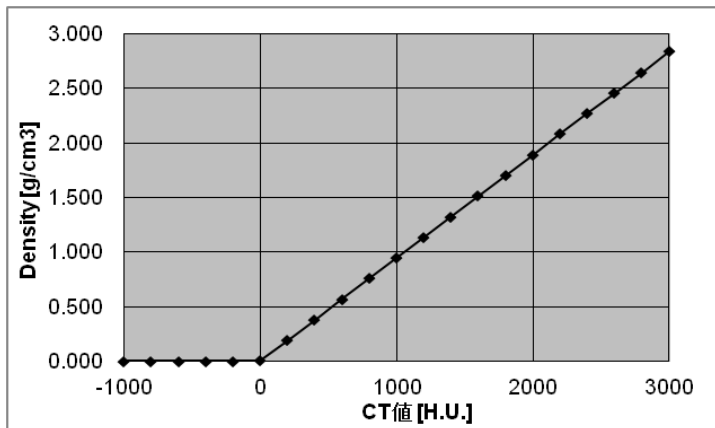


Figure 1: Calibration Curve for Density Conversion from CT Value

Note 1) It is supposed that the CT value is Hounsfield Unit.

Note 2) It is assumed that the X-ray tube voltage of the CT scanner is 125 [kV$_P$]

**\*About the settings of material properties other than density**

Various conversion formulas can be used for Poisson's ratio, Young's modulus, yield stress value, tensile direction critical stress value, and stress relaxation factor.

The provided conversion formulas (refer to below) and also conversion formulas defined by User can be used as these formulas.

The conversion formulas defined by user can be generated by the tool described by "14.3 Inhomogeneous Material Editor."

- Poisson's ratio

In addition to User-established formulas, a value can be selected from a fixed value ranging from 0.49 to 0.0 or set by the method based on Mr. Minamizawa.

Table 1: Settings by Minamizawa

| Density[g/cm$^3$] | Poisson's ratio |
|---|---|
| 1.8 ≦ ρ | 0.22 |
| 0.2 < ρ < 1.8 | 0.15 |
| ρ ≦ 0.2 | 0.49 |

- Young's modulus

In addition to User-established formulas, formulas can be selected from each conversion formula based on papers of Keyak, Carter, Keller, and Suzuki.

Table 1: Settings Based on Keyak

| Density[g/cm$^3$] | Young's modulus E[MPa] |
|---|---|
| $\rho = 0$ | $E = 0.001$ |
| $0 < \rho \leqq 0.27$ | $E = 33900 \rho^{2.20}$ |
| $0.27 < \rho < 0.6$ | $E = 5{,}307 \rho + 469$ |
| $0.6 \leqq \rho$ | $E = 10.200 \rho^{2.01}$ |

Table 2: Settings Based on Carter

| Density[g/cm$^3$] | Young's modulus E[MPa] |
|---|---|
| $\rho = 0$ | $E = 0.001$ |
| $0 < \rho$ | $E = 3790 \varepsilon^{0.06} \rho^{3}$ |

where $\varepsilon = 0.01$

Table 3: Settings Based on Keller (Vertebrae)

| Density[g/cm$^3$] | Young's modulus E[MPa] |
|---|---|
| $\rho = 0$ | $E = 0.001$ |
| $0 < \rho$ | $E = 1890 \rho^{1.92}$ |

Table 4: Settings Based on Keller (All)

| Density[g/cm$^3$] | Young's modulus E[MPa] |
|---|---|
| $\rho = 0$ | $E = 0.001$ |
| $0 < \rho$ | $E = 10500 \rho^{2.57}$ |

Table 5: Settings Based on Suzuki (Callus)

| Density[g/cm$^3$] | Young's modulus E[MPa] |
|---|---|
| $\rho = 0$ | $E = 0.001$ |
| $0 < \rho$ | $E = 0.2391 e^{8.00 \rho}$ |

## - Yield stress calculation

In addition to User-established formulas, formulas can be selected from each conversion formula based on papers of Keyak, Carter, Keller, and Suzuki.

Table 1: Settings Based on Keyak

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho \leqq 0.2$ | $\sigma_y = 1.0 \times 10^{20}$ |
| $0.2 < \rho < 0.317$ | $\sigma_y = 137 \rho^{1.88}$ |
| $0.317 \leqq \rho$ | $\sigma_y = 114 \rho^{1.72}$ |

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho < 0.317$ | $\sigma_y = 137 \rho^{1.88}$ |
| $0.317 \leqq \rho$ | $\sigma_y = 114 \rho^{1.72}$ |

Without elastic element

Table 2: Settings Based on Carter

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho \leqq 0.2$ | $\sigma_y = 1.0 \times 10^{20}$ |
| $0.2 < \rho$ | $\sigma_y = 68\, \varepsilon^{0.06}\, \rho^2$ |

where $\varepsilon = 0.01$

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho$ | $\sigma_y = 68\, \varepsilon^{0.06}\, \rho^2$ |

where $\varepsilon = 0.01$ and without elastic element

Table 3: Settings Based on Keller (Vertebrae)

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho \leqq 0.2$ | $\sigma_y = 1.0 \times 10^{20}$ |
| $0.2 < \rho$ | $\sigma_y = 284\, \rho^{2.27}$ |

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho$ | $\sigma_y = 284\, \rho^{2.27}$ |

Without elastic element

Table 4: Settings Based on Keller (All)

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho \leqq 0.2$ | $\sigma_y = 1.0 \times 10^{20}$ |
| $0.2 < \rho$ | $\sigma_y = 117\, \rho^{1.93}$ |

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho$ | $\sigma_y = 117\, \rho^{1.93}$ |

Without elastic element

Table 5: Settings Based on Suzuki (Callus)

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho \leqq 0.2$ | $\sigma_y = 1.0 \times 10^{20}$ |
| $0.2 < \rho$ | $\sigma_y = 30.49\, \rho^{2.41}$ |

| Density[g/cm$^3$] | Yield stress[MPa] |
|---|---|
| $\rho$ | $\sigma_y = 30.49\, \rho^{2.41}$ |

Without elastic element

- Critical stress

In addition to User-established formulas, formulas can be selected from the formulas below that calculate the critical stress from the yield stress value.

| Critical stress[MPa] |
| --- |
| $\sigma_I = 1.0\,\sigma_y$ |
| $\sigma_I = 0.9\,\sigma_y$ |
| $\sigma_I = 0.8\,\sigma_y$ |
| $\sigma_I = 0.7\,\sigma_y$ |
| $\sigma_I = 0.6\,\sigma_y$ |
| $\sigma_I = 0.5\,\sigma_y$ |
| $\sigma_I = 0.4\,\sigma_y$ |
| $\sigma_I = 0.3\,\sigma_y$ |
| $\sigma_I = 0.2\,\sigma_y$ |
| $\sigma_I = 0.1\,\sigma_y$ |

- Stress relaxation factor

In addition to User-established formulas, a constant parameter can be selected from the followings

| Stress relaxation factor |
| --- |
| 0.1 |
| 0.05 |
| $1 \times 10^{-20}$ （perfect elasto-plasticity） |

- References

［1］ Joyce H.Keyak, Stephen A.Rossi, Kimberly A.Jones, Harry B.Skinner,"Prediction of femoral fracture load using automated finite element modeling,"Journal of Biomechanics 31, pp.125-133,1998.

［2］ Dennis R.Carter, Wilson C.Hayes,"The Compressive Behavior of Bone as a Two-Phase Porous Structure,"the journal of bone and joint surgery, 59A No.7, October, pp.954-962, 1977.

［3］ Keller TS. Predicting the compressive mechanical behavior of bone. J Bimech. 1994 Sep;27 （9） : 1159-68.

［4］ T. Suzuki, "Biomechanics of callus in the bone healing process, determined by specimen-specific finite element analysis." Bone 132 (2020) 115212.

## Appendix 3.2 Definition Method of Material Properties of Truss Element

The material properties of the truss element are defined by giving a relationship between "Strain" and "Tension." The definition file is prepared in "data\props\truss" folder below the installation folder of this software.

- "pgroup.prp" file

When installing this software, the "Strain-Tension Table" as material properties of seven kinds of ligament is defined. This can be confirmed by "pgroup.prp" file. The contents are as follows.

```
#
# Mechanical property curve of truss element tension
#
DATABASE-GROUP        1
#
# GROUP 1 Spinal Ligaments
#
GROUP-NAME Spinal Ligaments
GROUP-PROPERTIES          7
#
# (Tension name)
# (File name)
ALL(HJ Kim,2009)
ALL(HJ Kim,2009).dat
PLL(HJ Kim,2009)
PLL(HJ Kim,2009).dat
LF(HJ Kim,2009)
LF(HJ Kim,2009).dat
TL(HJ Kim,2009)
TL(HJ Kim,2009).dat
CL(HJ Kim,2009)
CL(HJ Kim,2009).dat
ISL(HJ Kim,2009)
ISL(HJ Kim,2009).dat
SSL(HJ Kim,2009)
SSL(HJ Kim,2009).dat
#
```

A line started from "#" ...this is a comment line. It can be described arbitrarily, but ignored.

A line started from "DATABASE-GROUP" …Total number of defined "Strain-Tension Table" groups

A line started from "GROUP-NAME" …Name of "Strain-Tension Table" group defined under this

A line started from "GROUP-PROPERTIES" …Number of "Strain-Tension Table" defined under this

Lines lower than "GROUP-PROPERTIES" … "Name of Strain-Tension Table" and "Name of definition file" are described alternatively.
The number of pair is equal to the number defined in "GROUP-PROPERTIES".
Always use the blank or comment line after the last "Name of definition file".

The contents in the line from "GROUP-NAME" to the pair of "Name of Strain-Tension Table" and "Name of definition file" are repeatedly written as many times as the total number of groups.

**- "Strain-Tension Table" definition file**

The content of respective "Strain-Tension Table" is written in the definition file whose name is described in "pgroup.prp."

The content of initially defined "ALL (HJ Kim, 2009)" is shown below.

```
# strain(non-dimensional) tension(N)
TENSION-VALUE 4
-1 0
0 0
0.12 59.6
1 1180.7
```

A line started from "#" ...This is a comment row. It can be described arbitrarily, but ignored.

A line started from "TENSION-VALUE" …Number of the relationship between "Strain" and "Tension"

Lines lower than "TENSION-VALUE" …The relationship between "Strain" and "Tension" are described in a manner of being separated by space. The unit of "Strain" is non-dimension, and the unit of "Tension" is Newton (N).

**- The method of addition of definition to "Strain-Tension Table"**

**In the case of addition to already existing group**

1. Increase the final number of "pgroup.prp" file, "GROUP-PROPERTIES" line by one.

```
                          :
#
PROPERTIES-DATABASE        8
#
                          :
```

2. Add a line of "Name of Strain-Tension Table" and a line of "Name of definition file" to the last line of "pgroup.prp" file. Here, for example, the file is generated with a name of "ADDED TABLE," name it with a simple name.

```
                          :
SSL(HJ Kim,2009).dat
ADDED TABLE
ADDED TABLE.dat
```

3. At the same place with "pgroup.prp" file, make a file having the same name with "Name of definition file" added at 2. In the case of writing the path including the folder in "Name of definition file", if a folder with the same name is made, the definition file can be put in the folder.

```
ADDED TABLE.dat
ALL(HJ Kim,2009).dat
CL(HJ Kim,2009).dat
ISL(HJ Kim,2009).dat
LF(HJ Kim,2009).dat
pgroup.prp
PLL(HJ Kim,2009).dat
SSL(HJ Kim,2009).dat
TL(HJ Kim,2009).dat
```
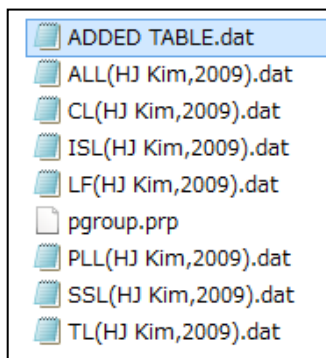
4. Edit "Name of definition file" to define the relationship between "Strain" and "Tension." Refer to the existing definition file for contents. Always, the last number of the line of "TENSION-VALUE" shall match to the number of the defined relationship between "Strain" and "Tension."

```
# strain(non-dimensional) tension(N)
TENSION-VALUE 4
-1 0
0 0
0.1 30.0
1 1000.0
```

That is all.

To reflect the addition of definition, restart this software, or click "Reread Defined Truss" button in Material Property display.

**In the case of addition to new group**

1. Increase the final number of "DATABASE-GROUP" by one in the group that would like to be added, in "pgroup.prp" file.

```
                        :
#
DATABASE-GROUP          2
#
                        :
```

2. Add "GROUP-NAME" and "GROUP-PROPERTIES" to the last line in "pgroup.prp". The group is arbitrarily named, but it is recommended to name the group that what kind of data is defined is easily understood. In "GROUP-PROPERTYS" line, write the number of "Strain-Tension Table" added later.

```
                        :
#
GROUP-NAME ADD-GROUP-NAME
GROUP-PROPERTIES    1
#
                        :
```

3. In the next line of the one added in 2., add "NAME" and "Name of definition file" of the Strain-Tension Table that would to be newly added. Here, for example, it is named "ADDED TABLE". Write the easily understood name.

```
                        :
ADDED TABLE
ADDED TABLE.dat
```

4. After this, it is same as 3. and 4. of "**In the case of addition to already existing group**" in the previous page.

## Appendix 3.3 Allocation of Material Property to the Truss Element

The material properties defined by "Strain-Tension Table" are allocated to the truss element. It needs to take care if one truss element group includes multiple truss elements.

It is explained below how to allocate the material properties to the truss element.

Assume that the "Strain-Tension Table" in the following graph is defined.
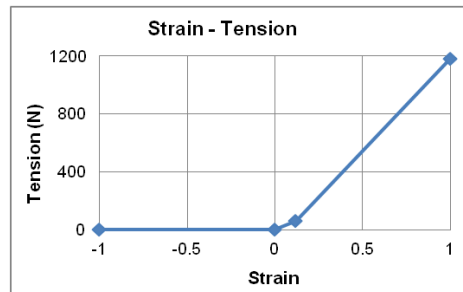


In one case that this table is set to group A in which one truss element is defined and group B in which four truss elements are defined, the actual "Strain-Tension" relationships allocated to each truss element are as follows.

A) Group in which one truss element is defined.          B) Group in which four truss elements are defined.
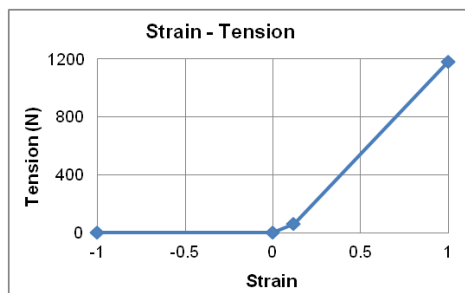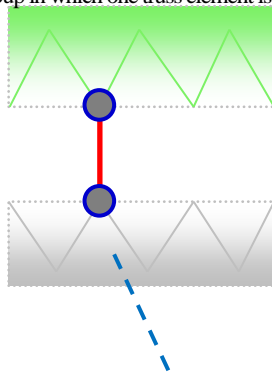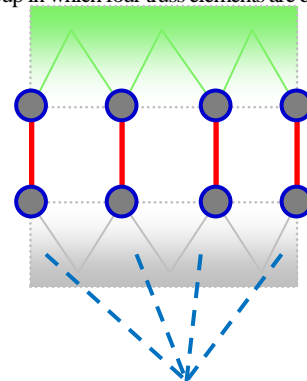


Since there are four truss elements, the tension is divided by four.
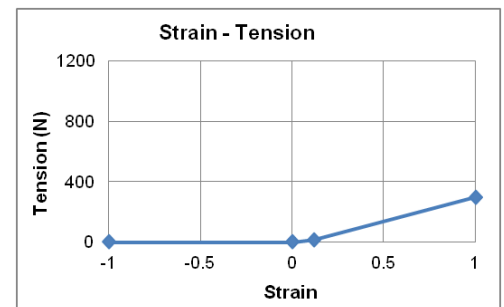
The property is allocated so that the whole group has "Strain-Tension Table" properties. Therefore, truss elements for different ligaments shall not be included in the same group.

If truss tension defined by "Strain-Tension Table" would like to be set to multiple truss elements in the same group, enter the same value as the number of the truss elements in "Magnification of Tension". The original tension value multiplied by "Magnification of Tension" is allocated to each truss element.

# Appendix 4 Solver Theory

The solver theory employed in this software is explained.

| | |
|---|---|
| Appendix 4.1 Analysis Type | Difference in failure process depending on analysis types (Elastic analysis, Elastic fracture line prediction analysis, non-linear fracture line prediction analysis) and the flow of each process are described. |
| Appendix 4.2 Formulation of Stress Analysis by Finite Element Method | The procedure of finite element method is described. |
| Appendix 4.3 Method of Solution of Equilibrium Equation | The method of solution of equilibrium equation is briefly described. |
| Appendix 4.4 Used Element | Element types and their configuration handled by this software are described. |
| Appendix 4.5 Crack Process | The internal process in tensile failure is described by element type. |
| Appendix 4.6 Non-linear Method of Solution | The internal process in non-linear fracture line prediction analysis is described. |
| Appendix 4.7 Contact Treatment | The process with a contact element, which is a special material, is described. |

## Appendix 4.1 Analysis Type

The solver of this software has analysis options as follows:

Since each analysis option has an ON/OFF switch, we can make various analysis by combining these.

- Geometric Non-linear

    OFF: Analyze based on infinitesimal deformation theory

    ON: Analyze considering large deformation.

- Material Non-linear

    OFF Compression failure (yield/crush) and tensile failure are not considered.

    ON: Consider compression failure and tensile failure

- Analysis Type

    Static: Perform static analysis.

    Dynamic: Perform dynamic analysis.

Appendix 4.1.1 Elastic Analysis

```
                          ┌─────────────────┐
                          │     start       │
                          └─────────────────┘
                                  │
                    ╱─────────────────────────────╲
                    ╲      loop: load step         ╱
                    
                          ┌─────────────────────────┐
                          │ set incremental load dF │
                          └─────────────────────────┘
                                  │
                    ╱─────────────────────────────╲
                    ╲      loop: convergence of     ╱
                         unbaranced force
```

calcurate elemental stiffness matrix **Ke**, and add all **Ke** to all stiffness matrix **K**

$\quad$ **Ke**= $\int$ **B$^T$DB**dv $\qquad$ **B** : strain - displacement matrix

$\qquad\qquad\qquad\qquad\qquad$ **D** : stress - strain matrix

$\quad$ **K**=$\Sigma$**Ke**

calculate incremental displacement **dU** from equation **dF** = **KdU**

$\quad$ **dU** = **K$^{-1}$dF**

calculate stress and strain of earch element from incremental displacement **dU**

$\quad$ **d**$\varepsilon$ = **BdU** $\quad$ $\varepsilon^n$ = $\varepsilon^{n-1}$+**d**$\varepsilon$

$\quad$ **d**$\sigma$ = **Dd**$\varepsilon$ $\quad$ $\sigma^n$ = $\sigma^{n-1}$+**d**$\sigma$

calculate inner force and unbaranced force over the whole model from elemental stress

$\quad\quad$ inner force : **F$_{in}$** = $\Sigma$ $\int$ **B$^T$**$\sigma^n$dv

$\quad$ unbaranced force : **R** = $\Sigma$**dF** - **F$_{in}$**

jadge convergence of unbaranced force

unconvergence $\qquad\qquad$ convergence

end loop: convergence of unbaranced force

output results

end loop: load step

end

Appendix 4.1.2 Non-linear Analysis

○Material non-linear analysis (calculation by iterations)

```
                          ┌─────────────┐
                          │    start    │
                          └─────────────┘

                     ⬡  loop: load step  ⬡

                     ┌──────────────────────────┐
                     │ set incremental load dF  │
                     └──────────────────────────┘

                     ⬡  loop: convergence of
                            unbaranced force  ⬡
```

calculate elemental stiffness matrix **Ke**, and add all **Ke** to all stiffness matrix **K**

$$\mathbf{Ke} = \int \mathbf{B^T DB}\,dv \qquad\qquad \mathbf{B} : \text{strain - displacement matrix}$$

$$\mathbf{D} : \text{stress - strain matrix}$$

$$\mathbf{K} = \Sigma\mathbf{Ke}$$

calculate incremental displacement **dU** from equation **dF = KdU**

$$\mathbf{dU} = \mathbf{K^{-1}dF}$$

calculate stress and strain of earch element from incremental displacement **dU**

$$\mathbf{d}\varepsilon = \mathbf{BdU} \qquad \varepsilon^n = \varepsilon^{n-1} + \mathbf{d}\varepsilon$$

$$\mathbf{d}\sigma = \mathbf{Dd}\varepsilon \qquad \sigma^n = \sigma^{n-1} + \mathbf{d}\sigma$$

judge whether yield/crashed/cracked elements exist

exist                    not exist

modify stiffness matrix **Ke** and recalculate stress

calculate inner force and unbalanced force over the whole model from elemental stress

$$\text{inner force}: \quad \mathbf{F_{in}} = \Sigma \int \mathbf{B^T}\sigma^n dv$$

$$\text{unbalanced force}: \quad \mathbf{R} = \Sigma\mathbf{dF} - \mathbf{F_{in}}$$

judge convergence of unbalanced force

unconvergence            convergence

end loop: convergence of unbalanced force

output results

end loop: load step

end

○Material non-linear analysis (calculation by substeps)

```
                          ┌─────────────────┐
                          │      start       │
                          └─────────────────┘
                                   │
              ┌─────────────────────────────────────────┐
              │            loop: load step               │
              └─────────────────────────────────────────┘
                                   │
                          ┌─────────────────────────┐
                          │  set incremental load dF │
                          └─────────────────────────┘
```

**set incremental load dF**

calculate elemental stiffness matrix **Ke**, and add all **Ke** to all stiffness matrix **K**

$$\mathbf{Ke} = \int \mathbf{B^{T}DB} dv$$     **B** : strain - displacement matrix

**D** : stress - strain matrix

$$\mathbf{K} = \Sigma \mathbf{Ke}$$

calculate incremental displacement **dU** from equation **dF** = **KdU**

$$\mathbf{dU} = \mathbf{K^{-1}dF}$$

calculate stress and strain of earch element from incremental displacement **dU**

$$\mathbf{d}\varepsilon = \mathbf{BdU} \qquad \varepsilon^{n} = \varepsilon^{n-1} + \mathbf{d}\varepsilon$$

$$\mathbf{d}\sigma = \mathbf{Dd}\varepsilon \qquad \sigma^{n} = \sigma^{n-1} + \mathbf{d}\sigma$$

judge whether broken elements exist

exist          not exist

modify stiffness matrix **Ke** and recalculate stress.
only when a element breaks first, subsequent incremental load becomes one tenth.

calculate inner force and unbalanced force over the whole model from elemental stress

inner force :     $\mathbf{F_{in}} = \Sigma \int \mathbf{B^{T}}\sigma^{n} dv$

unbalanced force :     $\mathbf{R} = \Sigma \mathbf{dF} - \mathbf{F_{in}}$

output results

end loop: load step

end

○Geometrical non-linear analysis + contact analysis

```
                          ┌──────────────────────┐
                          │        start         │
                          └──────────────────────┘
                                     │
                        ⬡ loop: load step ⬡
                                     │
                          ┌──────────────────────┐
                          │ set incremental load dF │
                          └──────────────────────┘
                                     │
                        ⬡ loop: convergence of  contact
                                    states ⬡
                                     │
                        ⬡ loop: convergence of
                              unbalanced force ⬡
                                     │
```

calculate elemental stiffness matrix **Ke**, and add all **Ke** to all stiffness matrix **K**

$$\mathbf{Ke} = \int \mathbf{B^T DB}\,dv \qquad \mathbf{B} : \text{strain - displacement matrix}$$

$$\mathbf{D} : \text{stress - strain matrix}$$

$$\mathbf{K} = \Sigma\,\mathbf{Ke}$$

calculate incremental displacement **dU** from equation $\mathbf{dF} = \mathbf{KdU}$

$$\mathbf{dU} = \mathbf{K^{-1}dF}$$

calculate stress and strain of earch element from incremental displacement **dU**

$$\mathbf{d}\varepsilon = \mathbf{BdU} \qquad \varepsilon^{n} = \varepsilon^{n-1} + \mathbf{d}\varepsilon$$

$$\mathbf{d}\sigma = \mathbf{Dd}\varepsilon \qquad \sigma^{n} = \sigma^{n-1} + \mathbf{d}\sigma$$

calculate inner force and unbalanced force over the whole model from elemental stress

$$\text{inner force} : \quad \mathbf{F_{in}} = \Sigma \int \mathbf{B^T}\sigma^{n}\,dv$$

$$\text{unbalanced force} : \quad \mathbf{R} = \Sigma\,\mathbf{dF} - \mathbf{F_{in}}$$

judge convergence of
unbalanced force

unconvergence          convergence

end loop: convergence of
unbalanced force

judge convergence of
**contact states**

unconvergence          convergence

end loop: convergence of
contact states

```
                          ┌──────────────────────┐
                          │    output results    │
                          └──────────────────────┘
```

end loop: load step

```
                          ┌──────────────────────┐
                          │         end          │
                          └──────────────────────┘
```

# Appendix 4.2 Formulation of Stress Analysis by Finite Element Method

The outline of formulation based on the principle of virtual work is indicated.

Fundamental equations

(1) Principle of virtual work

(2) Displacement–Strain relational expression

(3) Stress–Strain relational expression

The principle of virtual work can be expressed as follows.

$$\int_v \{\varepsilon\}^T \{\sigma\}\, dv \;-\int_v \delta\{u\}^T \{P_v\} dv \;-\; \int_s \delta\{u\}^T \{P_t\} ds \;=\; 0 \qquad (2.1)$$

where $\{\sigma\}$ is stress, $\{\varepsilon\}$ is strain, $\{u\}$ is displacement, $\{P_v\}$ is body force, $\{P_t\}$ is surface force vector.

The relation between displacement and strain is expressed by,

$$\{\varepsilon\} \;=\; [A]\,\{u\} \qquad (2.2)$$

and, the relation between stress and strain expressed by,

$$\{\sigma\} \;=\; [D]\{\varepsilon_e\} \qquad (2.3)$$

where $\{\varepsilon\}$, $\{\varepsilon_e\}$ are total strain and elastic strain, respectively, and $[D]$ is elastic matrix.

Assume that displacement of any points in an element can be interpolated using a displacement of a node, if the interpolation function is $[N]$ and displacement of node is $\{d\}$, the displacement can be expressed by the following formula.

$$\{u\} \;=\; [N]\,\{d\} \qquad (2.4)$$

Strain inside the element is expressed by displacement of the node

$$\{\varepsilon\} \;=\; [A]\,[N]\,\{d\} \;=\; [B]\,\{d\} \qquad (2.5)$$

Here, $[B]$ is referred to Strain–Displacement Matrix.

Substitute formulas (2.3) to (2.5) into formula (2.1), and the following formula is obtained after sorting out.

$$\int_{ve} [B]T\,[D]\,[B]\, dv\,\{d\} \;=\; \int_{se} [N]^T \{Ps\}\, ds \;+\int_{ve} [N]^T \{Pv\}\, dv \qquad (2.6)$$

Formula (2.6) is a balance equation of force of elements and can be rewritten as follows:

$$[K]\{d\} \;=\; \{fs\} \;+\; \{fv\} \qquad (2.7)$$

, where

$$[K] \;=\int_{ve} [B]^T [D]\,[B]\, dv \qquad (2.8)$$
$$\{fs\} \;=\int_{se} [N]^T \{Ps\}\, ds \qquad (2.9)$$
$$\{fv\} \;=\int_{ve} [N]^T \{Pv\}\, dv \qquad (2.10)$$

Assembling formula (2.7) in respect to all elements generates the equilibrium equations for the whole system.

Flowchart of Finite Element Analysis

```
                          ┌──────────────┐
                          │    start     │
                          └──────────────┘
                     ╱── loop: load step ──╲
                          │
                    │ set incremental load dF │
                     ╱── loop: convergence of  unbaranced force ──╲
```

calculate elemental stiffness matrix **Ke**, and add all **Ke** to all stiffness matrix **K**

$$\mathbf{Ke} = \int \mathbf{B^T DB} dv \qquad \mathbf{B} : \text{strain - displacement matrix}$$
$$\mathbf{D} : \text{stress - strain matrix}$$
$$\mathbf{K} = \Sigma \mathbf{Ke}$$

calculate incremental displacement **dU** from equation **dF = KdU**

$$\mathbf{dU} = \mathbf{K^{-1} dF}$$

loop: element

kind of element

solid                                                            shell

loop: integration point

loop: layer

**solid branch:**

calculate incremental strain **d ε**

$$\mathbf{d\varepsilon} = \mathbf{BdU}$$
$$\mathbf{d\varepsilon} = \{ \mathbf{d\varepsilon_x, d\varepsilon_y, d\varepsilon_z,} \mathbf{d\gamma_{xy}, d\gamma_{yz}, d\gamma_{zx}} \}$$

calculate incremental stress **d σ**

$$\mathbf{d\sigma} = \mathbf{Dd\varepsilon}$$
$$\mathbf{d\sigma} = \{ \mathbf{d\sigma_x, d\sigma_y, d\sigma_z,} \mathbf{d\tau_{xy}, d\tau_{yz}, d\tau_{zx}} \}$$

calculate total strain and total stress

$$\varepsilon^n = \varepsilon^{n-1} + \mathbf{d\varepsilon}$$
$$\sigma^n = \sigma^{n-1} + \mathbf{d\sigma}$$

**shell branch:**

calculate incremental strain **d ε**

$$\mathbf{d\varepsilon} = \mathbf{BdU}$$
$$\mathbf{d\varepsilon} = \{\mathbf{d\varepsilon_x, \ d\varepsilon_y, \ d\gamma_{xy}}\}$$

calculate incremental stress **d σ**

$$\mathbf{d\sigma} = \mathbf{Dd\varepsilon}$$
$$\mathbf{d\sigma} = \{\mathbf{d\sigma_x, \ d\sigma_y, \ d\tau_{xy}}\}$$

calculate total strain and total stress

$$\varepsilon^n = \varepsilon^{n-1} + \mathbf{d\varepsilon}$$
$$\sigma^n = \sigma^{n-1} + \mathbf{d\sigma}$$

end loop: layer

end loop: integration point

end loop: element

calculate inner force and unbalanced force over the whole model from elemental stress

$$\text{inner force}: \mathbf{F_{in}} = \Sigma \int \mathbf{B^T} \sigma^n dv$$
$$\text{unbalanced force}: \mathbf{R} = \Sigma \mathbf{dF} - \mathbf{F_{in}}$$

judge convergence of
unbalanced force

unconvergence                    convergence

end loop: convergence of
unbalanced force

output results

end loop: load step

┌──────────────┐
│     end      │
└──────────────┘

## Appendix 4.3 Method of Solution of Equilibrium Equation

The equilibrium equations for the whole system can be separated into two parts: in the one, the displacement is known and in the other unknown. In the suffix, f corresponds to unknown and s corresponds to the known degree of freedom in formula (3.1).

$$\begin{bmatrix} K_{ff} & K_{fs} \\ K_{sf} & K_{ss} \end{bmatrix} \begin{Bmatrix} d_f \\ d_s \end{Bmatrix} = \begin{Bmatrix} P_f \\ P_s \end{Bmatrix}$$

(3.1)

Pf: Given load

Ps: Unknown reaction force

df: Unknown displacement

ds: Known displacement

The calculation procedure is as follows:

$$K_{ff}\, d_f + K_{fs}\, d_s = P_f \qquad (3.2)$$

$$\mathbf{d}_f = K_{ff}^{-1}\, (P_f - K_{fs}\mathbf{d}_s) \qquad (3.3)$$

From formula (3.3), the unknown displacement $d_f$ is obtained. However, solving an inverse matrix of the matrix $K_{ff}$ is less efficient in calculation, using a character of symmetric matrix, the LU decomposition, which is one of the fastest solving methods, is used.

Appendix 4.4 Used Element

[4 nodes solid element], [3 nodes shell element], and [Gap element] are described.

## - 4 nodes solid element
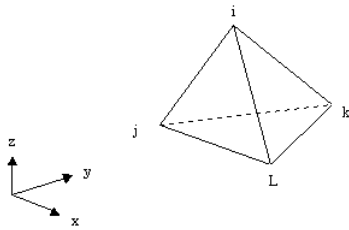


Displacement degree of freedom: Ux, Uy, Uz

Stress component: σx, σy, σz, τxy, τyz, τzx

Strain component: εx, εy, εz, γxy, γyz, γzx

Stress and strain are expressed by a global coordinate system of six components. Strain is constant in the element.

## - 3 nodes shell element
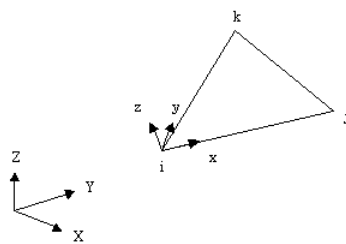


Displacement degree of freedom: Ux, Uy, Uz, θx, θy, θz

Stress component: σx, σy, τxy

Strain component: εx, εy, γxy

(1) Layered shell

Considering crack and yield, the stress distribution in the direction of the thickness will not be a linear distribution. Accordingly, the direction of the thickness is divided into layers and then stress and strain are calculated by each layer.

(2) Off-plane direction axis peripheral stiffness

The node displacement degree of freedom of the shell element has, considered by element coordinate system, five degrees of freedom of Ux, Uy, Uz, θx, θy.

Degree of freedom θz around the vector normal to the shell surface of each element is not considered since torsional stiffness is extremely larger than bending stiffness. However, if the shell elements construct a three-dimensional structure, displacement degree of freedom in the global coordinate system has six components.

Thus,when a node is connected to multiple shell elements in the same plane, the node is given virtual stiffness in the degree of freedom around the vector normal to the plane.
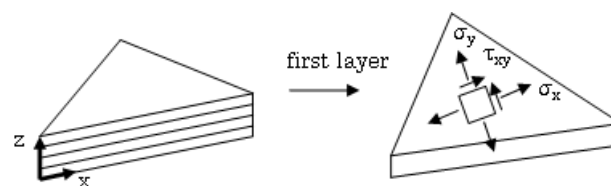
## - Gap element

The gap element is a one-dimensional element with a spring property defined between nodes.

If both nodes displace so as to compress the gap element, reaction force corresponding to a compression length and spring stiffness are generated to each node. By this, contact analysis between nodes can be performed artificially.

No reaction force occurs against the displacement of the gap element in the tensile direction.

The spring stiffness coefficient is calculated from neighbor elements of constructing nodes of the gap element. The calculation method is as follows:

   Assume that nodes n and n' are the construct the gap element. Each node is one of the nodes constructing elements $e_1$ to $e_5$ and $e_1$' to $e_6$'as constructing nodes.

   Make area of a contact surface of each element $A_1$ to $A_5$ and $A_1$' to $A_6$', respectively, and make Young's modulus $E_1$ to $E_5$ and $E_1$' to $E_6$', respectively, then the spring stiffness $E_G$ of the gap element can be obtained from the following formula.

   $E_G = 0.5 \times (\Sigma EA/3 + \Sigma E'A'/3)/L$
   (A value obtained from the average of the control area of the node multiplied by the sum of the element stiffness)
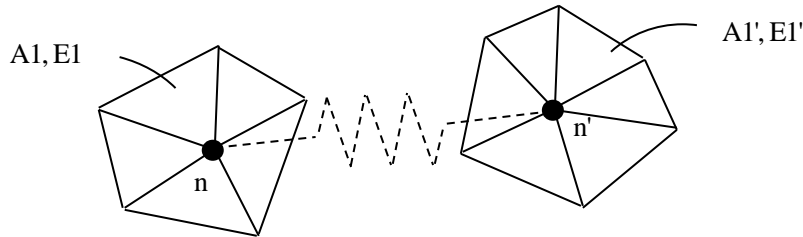


Here, a parameter related to the element length is entered to L. Increased spring stiffness of a gap element makes the convergence of unbalanced force at analysis worse; however, the amount of inload between the nodes decreases. The magnitude of spring stiffness shall be adjusted by L so as to converge the unbalanced force and decrease the amount of inroad.

## Appendix 4.5 Crack Process

The crack process in the elastic fracture line prediction analysis is explained.

### - Solid element

(1) Case without crack

| D matrix | It shall be isotropic. |
| --- | --- |
| Crack judging method | Comparing σ1 and ft, when σ1 > ft, a one-directional crack occurs. |

σ1 First principal stress

ft Tensile strength

(2) One-directional crack

| D matrix | It shall be orthogonal anisotropic with stiffness 0 in direction one. |
| --- | --- |

$$[D]=[Q]^T[D_{123}][Q]$$

$$[D_{123}]=\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ \square & X & Y & 0 & 0 & 0 \\ \square & \square & X & 0 & 0 & 0 \\ \square & \square & \square & G' & 0 & 0 \\ \square & SYM & \square & \square & G & 0 \\ \square & \square & \square & \square & \square & G' \end{bmatrix}$$

$$X=\frac{E}{\left(1-\nu^2\right)}$$

$$Y=\frac{\nu E}{\left(1-\nu^2\right)}$$

$$G=\frac{E}{2\left(1+\nu\right)} \qquad G'=0.25G$$

$$[Q]^T=\begin{bmatrix} L_1{}^2 & L_2{}^2 & L_3{}^2 & 2L_1L_2 & 2L_2L_3 & 2L_3L_1 \\ M_1{}^2 & M_2{}^2 & M_3{}^2 & 2M_1M_2 & 2M_2M_3 & 2M_3M_1 \\ N_1{}^2 & N_2{}^2 & N_3{}^2 & 2N_1N_2 & 2N_2N_3 & 2N_3N_1 \\ L_1M_1 & L_2M_2 & L_3M_3 & L_1M_2+L_2M_1 & L_2M_3+L_3M_2 & L_3M_1+L_1M_3 \\ M_1N_1 & M_2N_2 & M_3N_3 & M_1N_2+M_2N_1 & M_2N_3+M_3N_2 & M_3N_1+M_1N_3 \\ N_1L_1 & N_2L_2 & N_3L_3 & N_1L_2+N_2L_1 & N_2L_3+N_3L_2 & N_3L_1+N_1L_3 \end{bmatrix}$$

$L_1, M_1, N_1$: direction cosine of X1 axis
$L_2, M_2, N_2$: direction cosine of X2 axis
$L_3, M_3, N_3$: direction cosine of X3 axis

direction of X1 and σ1

cracked surface

direction of X1 and σ1

direction of X1 and σ1
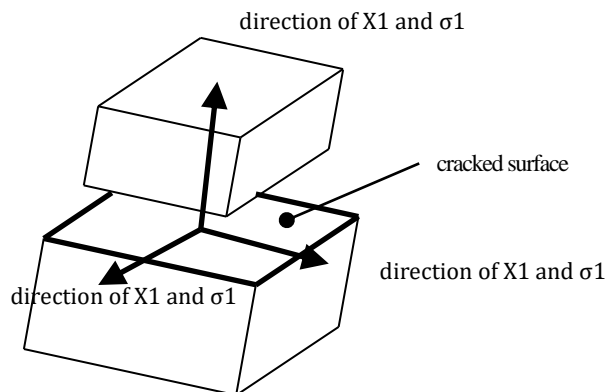
(3) Two- or Three-directional crack

Two- or Three-directional crack does not occur.

### - Shell element

(1) Case without crack

| D matrix | It shall be isotropic. |
|---|---|
| Crack judging method | Comparing σ1 and ft, when σ1 > ft, a one-directional crack occurs. |

σ1 First principal stress

ft Tensile strength

(2) One-directional crack

| D matrix | It shall be orthogonal anisotropic with stiffness 0 in direction one. |
|---|---|

$[D]=[T\sigma]^T[D_{12}][T\varepsilon]$

$$[D_{12}]=\begin{bmatrix} 0 & 0 & 0 \\ \square & E & 0 \\ SYM & \square & G' \end{bmatrix}$$

E: Young's modulus

$$G=\frac{E}{2(1+\nu)}$$

$$G'=0.25G$$

$$[T\sigma]=\begin{bmatrix} L^2 & M^2 & 2LM \\ M^2 & L^2 & -2LM \\ -LM & LM & (L^2-M^2) \end{bmatrix} \qquad [T\varepsilon]=\begin{bmatrix} L^2 & M^2 & LM \\ M^2 & L^2 & -LM \\ -2LM & 2LM & (L^2-M^2) \end{bmatrix}$$

direction of X2 and σ2    direction of X1 and σ1

cracked line

L=cosθ, m=sinθ

θ: the angle from the X axis of element coordinate system to the X1 axis
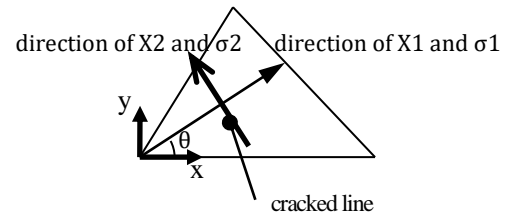
(3) Two-directional crack

Two-directional crack does not occur.

184

## Appendix 4.6 Non-linear Method of Solution

As mechanical behavior of bone, brittle fracture of tensile area (crack), and ductile fracture of compressed area (crushed yield, crush) are considered.

Load incremental method that divide loads and then adds increment to the divided load is a methods handling such material non-linearity.

It can be considered that it is elastic calculation inside an incremental interval.

Indication of the incremental method is obtained as

$$[K]\{\Delta d\} = \{\Delta fs\} + \{\Delta fv\} \cdot \cdot \cdot (3.1)$$

, where

$$[K] = \int ve\,[B]^T\,[D]\,[B]\,dv$$
$$\{\Delta fs\} = \int se\,[N]^T\,\{\Delta Ps\}\,ds$$
$$\{\Delta fv\} = \int ve\,[N]^T\,\{\Delta Pv\}\,dv$$
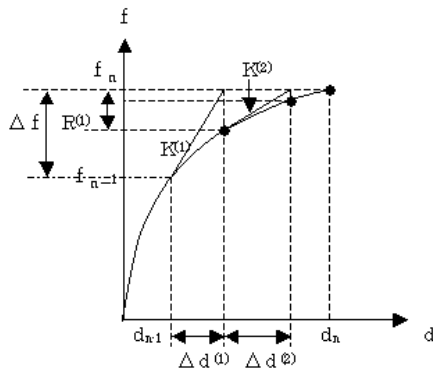


Assume that the equilibrium condition is satisfied in a state of n-1 which is one step before.

The first approximation is obtained from the stiffness K (1) calculated at the start point of incrementation against the increment load of $\Delta f$.

$$d(1)n+1 = dn + \Delta d(1)$$
$$\varepsilon(1)n+1 = \varepsilon n + \Delta\varepsilon(1)$$
$$\sigma(1)n+1 = \sigma n + \Delta\sigma(1)$$

Here, unbalanced force R （1） is obtained by the following formula.

$$R(1) = \sum_{\square}^{n+1} \Delta f - \sum \int_{elem}^{\square} BT\sigma(1)_{n+1} dv$$

By applying the unbalanced force to the load in the next iterative step, calculation is repeated until the unbalanced force becomes sufficiently small.

The convergence judgment is as follows.

（1）　solid

$$\frac{\|R(i)\|}{\|\Sigma_{\square}^{n+1}\Delta f\|} < \varepsilon \quad ,$$

where ‖ ‖ means Euclid norm.

(2) solid+shell mixture

Translation component

$$\frac{\|R(i)f\|}{\|\Sigma_\square^{n+1}\Delta ff\|} < \varepsilon \quad ,$$

where R(i)f: is Translation component of unbalanced force and Δff:is Translation component of increment load.

Rotational component

$$\frac{\|R(i)m\|}{\|\Sigma_\square^{n+1}\Delta fm\|} < \varepsilon \quad ,$$

where R(i)m is Rotational component of unbalanced force and Δfm is Rotational component of increment load.

$$\frac{\|R(i)f\|}{\|\Sigma_\square^{n+1}\Delta ff\|} < \varepsilon \quad ,$$

$$\frac{\|R(i)m\|}{\|\Sigma_\square^{n+1}\Delta fm\|} < \varepsilon \quad ,$$

## Appendix 4.7 Contact Treatment

If two element surfaces are defined as contact surfaces (primary/secondary), judgment of contact between the secondary node and the primary surface is done. At the time the secondary node takes the following four states.

1) Non-contact state
2) Point contact state
3) Edge contact state
4) Face contact state

In the states 2) to 4), some degrees of freedom of the secondary nodes are bounded by the primary nodes.

### (1) Non-contact

If the secondary node does not pass through the primary surface or constraint force on the secondary node is in the direction away from the primary surface, it is judged to non-contact. The secondary node does not interfere with the primary node and vice versa.
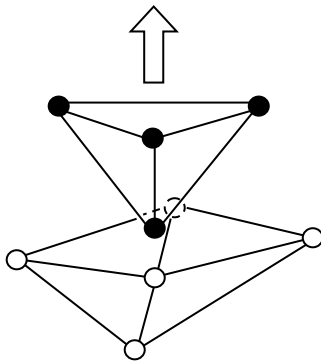


### (2) Point contact

If the secondary node is pressed to the primary node, it is judged to point contact.

At the time, each degree of freedom of the secondary node is bounded to the same degree of freedom of the primary node (single point constraint)
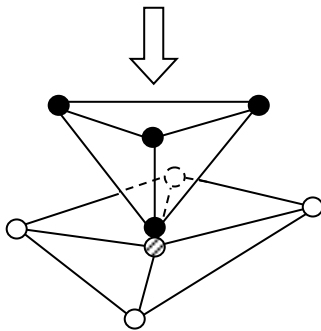
(3) Edge contact

If the secondary node slides on the line connecting the primary nodes, it is determined as edge contact.

At the time, two degrees of freedom of the secondary node (directions normal to the line) are bounded to the other one degree of freedom of the secondary node and the degrees of freedom of the primary nodes at the both ends of the line.
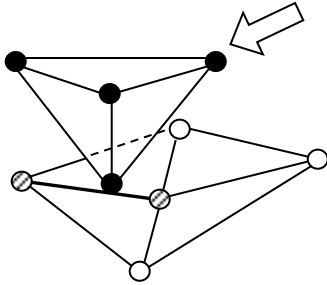


(4) Face contact

If the secondary node slides on the primary surface, it is judged to face contact.

At the time, one degree of freedom of the secondary node (directions normal to the surface) is bounded to the other two degrees of freedom of the secondary node and the degrees of freedom of the primary nodes constructing the surface.
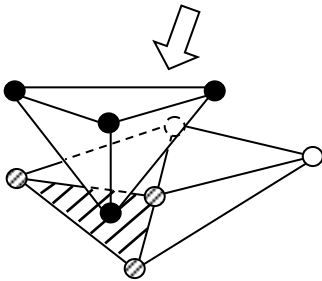
In analysis of the model in which the contact surface is defined, a contact loop is entered in addition to the equilibrium repetition loop. After the judgment of convergence of the unbalanced force, contact judgment is performed according to the positional relationship of the contact surface.

```
                          ┌─────────────┐
                          │    start    │
                          └─────────────┘
                                 │
                      ⬡ loop: load step ⬡
                                 │
                      ┌──────────────────────┐
                      │ set incremental load dF │
                      └──────────────────────┘
                                 │
                 ⬡ loop: convergence of   contact
                                states ⬡
                                 │
                      ⬡ loop: convergence of
                          unbalanced force ⬡
                                 │
      ┌───────────────────────────────────────────────────┐
      │ correct of displacement by change of contact state, and calculate elemental stiffness │
      │ matrix Ke                                          │
      └───────────────────────────────────────────────────┘
                                 │
      ┌───────────────────────────────────────────────────┐
      │ calculate displacement U, stress σ, strain ε        │
      └───────────────────────────────────────────────────┘
                                 │
                  ◇ judge convergence of
                       unbalanced force ◇
                                 │ unconference          convergence
                  ⬡ end loop: convergence of
                       unbalanced force ⬡
                                 │
                  ◇ judge convergence of
                       contact states ◇
                                 │ unconference          convergence
                  ⬡ end loop: convergence of
                       contact states ⬡
                                 │
                      ┌─────────────────┐
                      │  output results │
                      └─────────────────┘
                                 │
                  ⬡ end loop: load step ⬡
                                 │
                          ┌─────────────┐
                          │     end     │
                          └─────────────┘
```

From version 6.2, Solver V2 can consider friction on the contact surface.

Friction is expressed by the following model.



**Fn: External force acting object (perpendicular to floor surface)**

**F: External force acting object (parallel to floor surface)**

**ur: Relative displacement between the object and the floor surface.**

**Ft: Friction force on the object**

**μ: Friction coefficient**
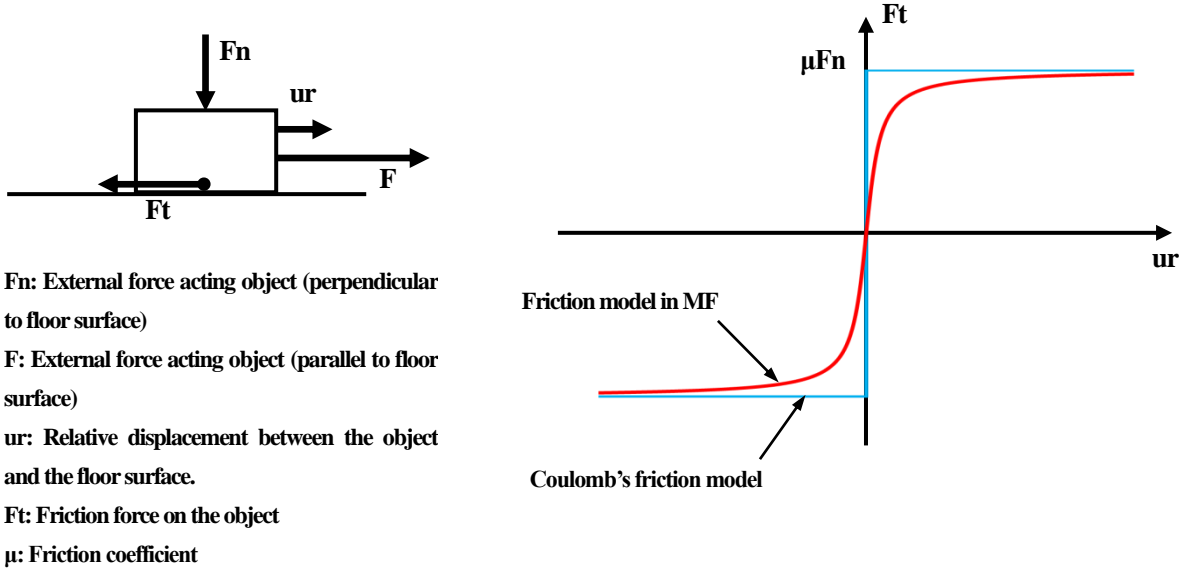
The relationship between $u_r$ and $F_t$ in case that the external force acts like right upper graph is shown in the right upper figure.

(1) Coulomb's friction model

While $F \leq \mu F_n$, the object does not move (namely, $u_r = 0$), the friction force $F_t$ as much as F acts. When $F > \mu F_n$, the object moves, the friction force acting on the object at the time is constant, $F_t = \mu F_n$ regardless of $u_r$.

(2) Friction model in MF

Since it is difficult to obtain a convergence solution with the Coulomb's friction model, MF adopts the model shown by the red line in the right upper figure. This curve is expressed by the following functions.

$$F_t = \mu F_n \frac{2}{\pi} arctan \frac{u_r}{u_{r0}}$$

The shape of the curve is determined by the magnitude of $u_{r0}$. For smaller $u_{r0}$, the friction model in MF is close to the Coulomb's friction model nearer. $U_{r0}$ is automatically set to the value by which convergence is possible in the program. If the convergence of the calculation is worse, making $u_{r0}$ larger allows to improve the convergence by multiplying the default $u_{r0}$ by a relaxation factor α.

Appendix 4.8 About the Specification of Damping in Dynamic Analysis Model

In the dynamic analysis, a structural damping coefficient is used in the calculation. Here, the specification method of the structural damping coefficient is explained.

(1) Consider one degree-of-freedom system consisting of mass m, spring constant k, and viscosity damping coefficient b. The equation of motion of the system is as follows.

$$m\ddot{u} + b\dot{u} + ku = f$$

The damping property of this system is determined by the following damping ratio (ratio of the damping coefficient to the critical damping coefficient).

$$\zeta = \frac{b}{b_{cr}} = \frac{b}{2\sqrt{mk}}$$

$\zeta$ : damping ratio, $b_{cr}$ : critical damping coefficient

(2) The equation of motion of the multi-degrees-of-freedom system is as follows.

$$[M]\{\ddot{u}\} + [B]\{\dot{u}\} + [K]\{u\} = \{F\} \; - \; ①$$

If no {F}, it is free vibration. For simplifying, let, consider free oscillation, $\{F\} = \{0\}$. The solution is determined by the initial condition and is superposition of multiple vibration modes.

(3) The formula 1 is mode decomposed. Substitute followings to the formula 1.

$$\{u\} = \sum_{i=1}^{N} \xi_i\{\phi_i\} = [\phi]\{\xi\} \; - ②$$

$\{\phi_i\}$ : mode vector

$[\phi] : [\{\phi_1\} \quad \{\phi_2\} \quad \cdots \quad \{\phi_N\}]$

$$\{\xi\} : \begin{Bmatrix} \xi_1 \\ \xi_2 \\ \vdots \\ \xi_N \end{Bmatrix}$$

And then multiply $[\phi]^T$ from left. The following equation is obtained.

$$[\phi]^T[M][\phi]\{\ddot{\xi}\} + [\phi]^T[B][\phi]\{\dot{\xi}\} + [\phi]^T[K][\phi]\{\xi\} = \{0\}$$

Here $[\phi]^T[M][\phi]$ is diagonalized as (same in $[\phi]^T[B][\phi]$, $[\phi]^T[K][\phi]$)

$$[\phi]^T[M][\phi] = \begin{bmatrix} m_1 & \square & \square & \square \\ \square & m_2 & \square & \square \\ \square & \square & \ddots & \square \\ \square & \square & \square & m_N \end{bmatrix}$$

thus, the equation of motion is mode decomposed to

$$m_i\ddot{\xi}_i + b_i\dot{\xi}_i + k_i\xi_i = 0.$$

Solve it and then substitute $\xi_i$ to 2, a free vibration solution with superimposing of vibration mode is obtained.

(4) In the damping model of MECHANICAL FINDER, $[B] = C_K[K]$, thus the damping ratio of each mode is

$$\zeta_i = \frac{b_i}{2\sqrt{m_ik_i}} = \frac{C_Kk_i}{2\sqrt{m_ik_i}} = \frac{C_K}{2}\sqrt{\frac{k_i}{m_i}} = \frac{C_K}{2}\omega_i.$$

The damping ratio is proportional to the natural angular frequency of each vibration mode. Namely for $\{u\}$ obtained by superimposing of each mode vector, the mode with lager natural angular frequency is damped faster.

(5) Specification method of damping

Determine $C_K$ so that in the dominant vibration mode (mode with lager $\xi_i$, normally i = 1), a desired damping ratio is obtained. Exactly,

$$\zeta_i = \frac{C_K}{2}\omega_i$$

From that

$$C_K = \frac{2\zeta_i}{\omega_i}$$

$\zeta_i$ : expected damping ratio, $\omega_i$ : natural angular frequency of dominant oscillation mode

(6) When no damping (the structural damping coefficient =0) is specified, the minimum damping acts as default in order to suppress vibration of higher harmonics generated by the contact process, etc. Exactly,

$$\omega_{i,max} = \left(\frac{\pi}{l}\sqrt{\frac{E}{\rho}}\right)_{max}$$

$\omega_{i,max}$ : maximum natural angular frequency of element

$l$ : element size

$E$ : young's modulus of element

$\rho$ : dencity of element

Thus, the default $C_K$ is determined as follows so that $\zeta_i = 1$ in this mode.

$$C_K = \frac{2 \cdot \zeta_i}{\omega_i} = \frac{2 \cdot 1}{\omega_{i,max}}$$

# Appendix 5 FAQ

In this chapter, FAQs for each process of MECHANICAL FINDER are described.

Appendix 5.1 About DICOM Data, Image File

A1. The following DICOM formats are suitable for input images:

   1) An image from X-ray CT system(Available from MRI system)

   2) An image output as a gray scale(Negative image is not suitable)

   3) An image saved as a non-compressed image

   4) An image where one slice is saved as one file

   5) An image output as Hounsfield Unit

   However, if the CT value is not referred when setting material properties, data from MRI system, or negative gray scale, data not in Hounsfield can be used.

   This interface automatically collects files with the same series instance UID (tag = 0x0020, 0x000e) in the same directory, then read them in the order of scanning positions.
   Therefore, in case that different imaged data is stored into the same directory as the same series instance UID at the time of CT imaging, normal processing cannot be performed.
   In that case, separate the saved directory or save the data as different series instance UID.

A2. It is adjusted as follows.

   (When the data is BMP, JPEG, or TIFF with 8-bit gradation)

   - A color image is converted to a grayscale image.

   - Images with different sizes (pixel numbers in height and width) are adjusted to the minimum image size.

   - Images are stacked with a regular interval in the direction of stacking (Z-axis direction).

   - The value range is converted to 256 gradations from 0 to 255.

   (When the data are TIFF with 16-bit gradation)

   - Images with different sizes (pixel numbers in height and width) are adjusted to the minimum image size.

   - Images are stacked with a regular interval in the direction of stacking (Z-axis direction).

   - The value range is converted keeping the 16-bit gradation.

A3. The harmful effects below are considered.

   - Number of operations during ROI extracting becomes larger, and performing shape extraction becomes harder.

   - If inhomogeneous material is selected at material properties setting, the artifact portion is set to a hard material. About the countermeasure, refer to "Appendix 5.4 About Material Sorting, Material Properties."

A4. In this software, three-dimensional construction considering the gantry tilting angle is not performed.

   Thus, when the model is constructed from images with gantry tilting, the shape is deformed as much as the gantry tilting angle.

   Use a DICOM data without a gantry tilting angle.

   Further, for a DICOM data with a gantry tilting angle, information appears at the time of DICOM conversion.

A5. The large difference between medical CT and dental CT is in general whether output CT value is output in Hounsfield Unit or not.

Almost all output CT values of dental CT are not in Hounsfield Unit; thus, even though inhomogeneous material properties are calculated by referring the CT value, the correct material properties cannot be obtained.

If the analysis of inhomogeneous material based on data imaged by dental CT is performed, image an indicator such as a bone quantity phantom at the time of imaging, followed by performing calibration setting with "ROI/Phantom Menu."

CT scanning should be performed with a bone density calibration phantom to calibrate bone density at "ROI/Phantom."

## Appendix 5.2 About ROI/Phantom

<span style="color:blue">Q1. Which portion of the bone should be extracted?</span>

A1. Region to be analyzed including cortical bone and cancellous bone shall be extracted.

   Refer to "5.1.1 About ROI Extraction Area."

   Further, to perform analysis, all elements in a model must have been connected each other.

   Therefore, final discrete shape data as an analysis mesh are not allowed.

   However, this shall not apply to the case where an analysis with forcible displacement or a contact analysis with initial velocity is performed.

<span style="color:blue">Q2. Is the phantom setting necessary?</span>

A2. As referred in "Appendix 3 Material Properties," if the phantom setting is not carried out, the density calculation is performed assuming that the X-ray tube voltage of the imaging CT scanning is 125 [kVp]

   Therefore, if the value of the X-ray tube voltage of the imaging CT scanning is different, the density calculation value is also different.

   And, in the case other than X-ray tube voltage;

   1) When modification (rearrangement) of data is carried out when converting the imaged raw image to a DICOM file;

   2) Degradation of the CT imaging system is recognized;

   3) When a metal such as a bolt, etc. is imaged simultaneously;

   4) When an image is scanned by dental CT that does not output the image in Hounsfield Unit;

   CT value is influenced.

   Therefore, it is recommended to perform CT scanning with phantom for correct bone density.

   In addition, even in the condition where phantom has not been imaged, if density volue needs to be modified, usage of the phantom setting is possible. (For example, determine that cortical bone is XXX mg/cm$^3$ and cancellous bone is XXX mg/cm$^3$, for calibration.)

   [What is a phantom?]

   It is a device referred to bone density calibration phantom. Multiple RODs which are made of substance equivalent to hydroxyapatite are packed in the device, and the density of each ROD is different.

   Calculation of bone density based on CT image is accurately performed due to the following reasons: It is made of a material equivalent to hydroxyapatite whose X-ray absorption amount is close to bone; the density is known; and multiple RODs with different density value are prepared.

<span style="color:blue">Q3. About the detail of "AI ROI" function.</span>

A3. Auto segmentation of bones with deep learning is executed in "AI ROI" function. The deep learning model of femur/vertebrae segmentation whose link is noted in the following paper is implemented. Refer this if used in your paper.

   *Femur Segmentation
   Uemura, K., Otake, Y., Takao, M. et al. Development of an open-source measurement system to assess the areal bone mineral density of the proximal femur from clinical CT images. Arch Osteoporos.(2022).
   https://doi.org/10.1007/s11657-022-01063-3

   *Vertebral Segmentation
   Payer, C. et al. Coarse to Fine Vertebrae Localization and Segmentation with SpatialConfiguration-Net and U-Net. Proceedings of the 15th International Joint Conference on Computer Vision, Imaging and Computer Graphics Theory and Applications; 2020 Feb 27-29. Valletta, Malta. VISAPP; 2020. Volume 5:124-133.
   https://doi.org/10.5220/0008975201240133

Uemura, K. et al. Development of a system to assess the two- and three-dimensional bone mineral density of the lumbar vertebrae from clinical quantitative CT images. Arch Osteoporos. (2023).
https://doi.org/10.1007/s11657-023-01216-y

Cheng, Z. et al. Fully Automatic Vertebra Segmentation for Spine and Pelvis Alignment Analysis in a Large-scale CT Database. Proceedings of Biological Imaging and Medical AI; 2020 May 10. JSMBE; 2020. Abstract no.7.
http://biomedimg.umin.jp/2ndMAI_proceedings_v2.pdf

Appendix 5.3 About Mesh Generation

A1. Several reasons can be considered.

●Factor on surface mesh

1. If the value of "Surface Mesh Size" in "ROI mesh" is too large, a mesh with a good shape is not generated.

The surface mesh is used as a reference shape before internal mesh generation, and generated internal mesh depends on the "Global Base Size (fTetWild)" or "Minimum Size (ANSYS ICEM CFD)" of "Setting of Internal Mesh." Therefore, keep in mind to enter values to the value of "Surface Mesh Size" so that a correct shape is obtained.

2. The algorism in "ROI mesh" is subject to curve construction, thus the algorism is not suitable to a planer shape. There is a possibility that making the setting of "Surface Mesh Size" smaller will not bring a good result.

At the time, constructing a surface mesh by "Iso-surface" sometimes brings a good geometry, use it as a reference once.

3. If the distance between slices imaged at DICOM is large, when a mesh is generated by "Surface Mesh Import Setting," a step between the slices (jaggy) appears, thus a good mesh shape may not be obtained. In such case, apply the interpolation process in "Surface Mesh Import Setting." The step in the slices may be canceled.

●Factor on internal mesh

1. There is a possibility that the value of the parameter "Global Base Size (fTetWild)" or "Minimum Size (ANSYS ICEM CFD)" in "Setting of Internal Mesh" is too large.

Decrease the value of "Minimum Size" bit by bit to adjust it to have appropriate number of the internal mesh and a shape.

2. If the imported shape has an edge, it is necessary to turn on the checkbox "Keep Edge of Shape" of "Setting of Internal Mesh." If it is OFF (ANSYS ICEM CFD), the geometry after the internal mesh generation may be rounded at the edge.

A2. The following means is used.

(1) Standard Edition

Generates solid elements using the Advancing Front method for a surface mesh. The surface mesh is preserved in tetra mesh.

(2) Extended Edition

A) fTetWild*

Generates tetra elements using the Delaunay method. The surface mesh is re-meshed(not preserved).

B) ANSYS ICEM CFD

Generates tetra elements using the Octree method. The surface mesh is re-meshed(not preserved).

\* Yixin Hu, Teseo Schneider, Bolun Wang, Denis Zorin, and Daniele Panozzo. 2020. Fast tetrahedral meshing in the wild. ACM Trans. Graph. 39, 4, Article 117 (July 2020), 18 pages.

A3. The fact occurs when each surface approaches closer to each other in the surface mesh and the import group.

The following measures are considered as a countermeasure.

- Make "Minimum Size" of "Setting of Internal Mesh" smaller.

- Assume an implant is inserted to a bone, do not cut the region of bone geometry where the implant is inserted.

- Make the imported shape simple.

Make "6.6 Mesh / Import Basics" as a reference.

Appendix 5.4 About Material Sorting, Material Properties

A1. The thickness (mm) of the shell element means the thickness from the top layer to the bottom layer. For example, if the setting value is 0.5 mm, the thickness from top layer to the middle layer is 0.25 mm and the thickness from middle layer to the bottom layer is 0.25 mm; thus, the thickness from the top layer to the bottom layer is 0.5 mm.

Q2. What is the purpose in attachment of the shell element to the surface of the solid element?

A2. Mainly two reasons exist.
   1. To simulate cortical bone

   For example, in the case of the bone with a very thin cortical bone such as a femoral head, to represent the cortical bone by a solid shape, massive number of small mesh element is needed, thereby making the analysis very difficult. On the other hand, if the analysis is performed without considering the material properties of the cortical bone, the analysis is performed with lower strength than an actual strength of the femur; thus, there is a problem. Accordingly, by applying a shell element to the surface of the solid and giving the cortical bone material to the shell element the cortical bone can be simulated so as to keep the strength.
   2. In order to easily grasp the stress distribution of surface of solid and direction of fracture.

   For example, attachment of a very thin (0.1 mm, etc.) shell element to a surface of a solid element and allocation of the same material of neighboring solid element to the material of the shell element allows to generate analysis model almost unchanged in strength. Analyzing and displaying the result allows to very clearly display the stress direction (tensor) of the surface of the elements and the direction of the fracture line and its progress. This result displays the value of the shell element as it is. Thus, compared to the case where the result value at the surface is obtained by the interpolation process from the solid elements, it is an accurate and less rounding result.

Q3. What is the Young's modulus?

A3. The gradient (proportion constant) of a strain-stress curve is defined as Young's modulus. Larger Young's modulus (the gradient is steep) means harder material and smaller Young's modulus (the gradient is gentle) means softer material. (E in the formula $\sigma = E\varepsilon$ is defined as Young's modulus.)

Q4. What is the Poisson's ratio?

A4. It means a ratio of transverse strain to longitudinal strain in the event of tensile load or compressive load.
   For example, when a load is applied in x direction $\nu_{xy}$, $\nu_{xz}$ of the formulas
   $\nu_{xy} = -(\varepsilon_y/\varepsilon_x)$
   $\nu_{xz} = -(\varepsilon_z/\varepsilon_x)$
   are defined as Poisson's ratio. If it is isotropic, $\nu_{xy} = \nu_{xz}$.

Q5. What is the yield stress?

A5. It is a stress value that reaches at elastic limit in the compressive direction stress.
   (About the tensile direction, at the initial value, the critical stress is set so that a crack occurs before a stress reaches at the yield stress (described later) Therefore, this software does not consider the yield stress in the tensile direction.) When applying a load to the compressive direction, if the equivalent stress value exceeds this yield stress value, the element becomes plastic state.

A6. It is a stress value that reaches at crack limit in the tensile direction stress.

   When applying a load in the tensile direction, a crack occurs if the principal stress exceeds the critical stress.

   This software defines the occurrence of crack if the maximum principal stress > the critical stress is fulfilled, and the direction of the crack is perpendicular to the direction of the principal stress.

A7. It is used for obtaining the value referred to "Work hardening rate"

   When drawing a strain-stress curve, Young's modulus is expressed by a line with a constant gradient (linear equation), if the element becomes plastic state by a load in the compressive direction, it is not referred to Young's modulus but referred to as "Work hardening rate" and is calculated by the following formula.

   "Work hardening rate (HO')" = Young's modulus*Stress relaxation coefficient.

A8. It means a criterion defining the yield state of material under multiple axes stress and has the following features.

   - Von-Mises (Mises model) (*The initial setting of other material)

     It is generally used for a model using ductile material.

   - Drucker-Prager (Drucker-Prager-model) (*The initial setting of bone material)

     It is used for a model using brittle material.

A9. If the minimum principal strain of an element in plastic state exceeds the crush strain (in other words, specified strain occurs in the compressive direction), the element compressively breaks down (crush)
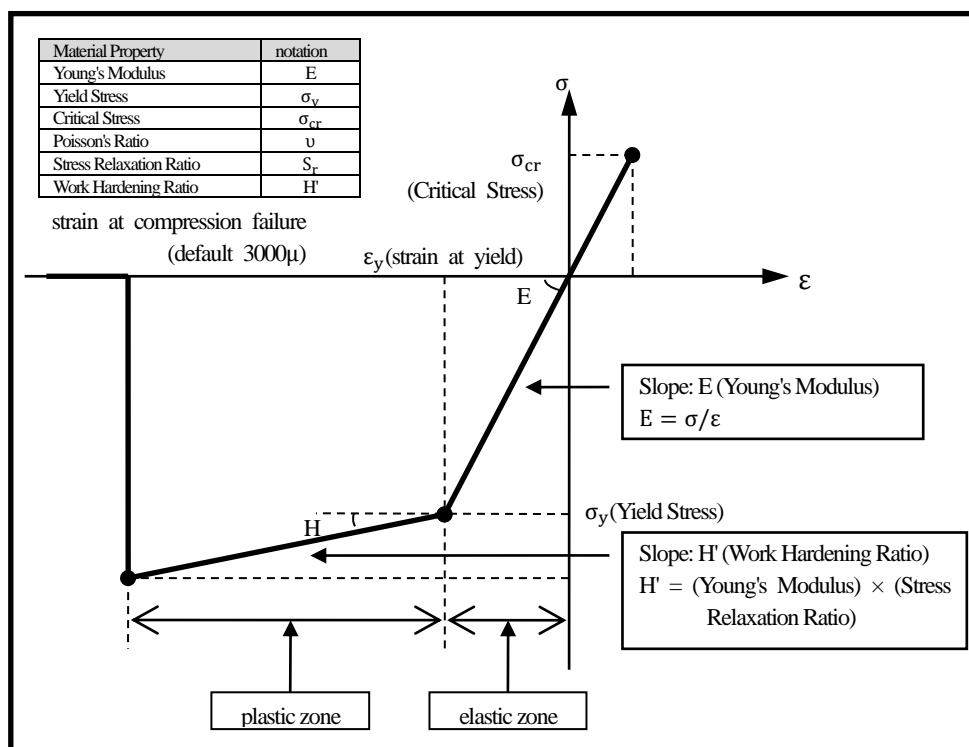
A10. An artifact region has a very high CT value. If the density conversion formulas are applied to inhomogeneous material as it is, the density value and Young's modulus largely different from the original values are allocated. As a countermeasure, turning on "Set Upper Limit of Density" of "8.1 Material Property" and entering the density value to be the upper limit of the bone density can mitigate a harmful effect such as that the density value is set to very high.

# Appendix 5.5 About Analysis

A1. The displacement method is used for the Finite Element Method.

(Almost all commercially available Finite Element Method programs use the displacement method)

In this software, the following three analysis types are available.

| | | |
|---|---|---|
| Consideration of material non-linear analysis | : | You can select whether crack and plasticity are to be considered. |
| Consideration of large deformation analysis | : | You can select whether geometrical non-linearity is to be considered. If the model deforms largely or rotates, considering this allows to obtain a result closer to reality. |
| Consideration of dynamic analysis | : | You can select whether inertia is to be considered. If an instantaneous load is simulated, considering this allows to obtain a result closer to reality. |

Q2. What is the difference of the solver type?

A2. It is the difference that whether the direct method or the iterative method (CG method) is used for the method of solving equations at analysis. In addition, in the direct method, selection from PARDISO or Solver for which the out-core memory is available is possible, and in iterative method, selection from Solver calculating in CPU or Solver using GPU is also possible. The obtained result is the same with sufficient accuracy; however, the direction method has good convergence, difference may occur in a judgment step when completely fractured.

The comparison of performance to a model is as follows.

| | | |
|---|---|---|
| Direct (Multi-core) | : | Usage of memory is large. Use of PARDISO It processes faster than <Direct method (Single-core)> |
| Direct (Single-core) | : | Usage of memory is large. Use this if necessary memory is not ensured in <Direct method (multi-core)> |
| Iterative (CG-method) | : | Usage of memory is small. Use this when a model is relatively large. |
| Iterative (CG by GPU) | : | Usage of memory is small. If the GPU manufactured by NVIDIA is installed, using this makes processing faster than <Iteration method (CG method>*. |
| | | *It depends on the performance of the installed GPU and convergence of the analysis. |
| | | *The calculable size of the model depends on the memory of the GPU. |

As a reference, comparison of calculation times when analyzing sample.sys, sample2.sys, dynamic.sys is shown.

Calculation Requirements: Windows XP 32 bit

Intel Core 2 Quad Q9650 3.0 GHz

NVIDIA GeForce GTX 580

| Analysis Type | sample.sys | sample2.sys | dynamic |
|---|---|---|---|
| Direct method (multi-core) 1 core | 317.7s | 17.4s | 79.6s |
| Direct method (multi-core) 4 cores | 228.2s | 12.0s | 54.8s |
| Direct (Single-core) | 661.4s | 24.0s | 100.9s |
| Iterative (CG-method) | 1392.5s | 124.1s | 207.1s |
| Iterative (CG by GPU) | 269.2s | 12.6s | 60.1s |

**Q3. Does "equivalent stress" in "Contour map" and "Physical quantity" mean the MISES equivalent stress?**

A3. The equivalent stress of this software is determined by the yield criteria when setting material properties.

- If "Von-Mises" is selected as yield criterion for material, it is the equivalent stress for the MISES model.
- If "Drucker Pragger" is selected for as yield criterion material, it is the equivalent stress for "Drucker Pragger" model.

**Q4. I want to know about the failure in detail.**

A4. The failure classification of the shell element or solid element includes "tensile failure" and "compressive failure," and the state can be displayed by "11.6 Result Display." About tensile failure and compressive failure, refer to the following answer.

**Q5. I want to know about the tensile failure in detail.**

A5. It appears in "Fracture figure display" as a crack. It appears on the shell element as a line with a direction, and on the solid element as a plane with directions.

(About crack, refer to "Appendix 5.4 Material properties, about Phantom setting Q4 What is the critical stress?")

**Q6. I want to know about the compressive failure in detail.**

A6. In "Fracture Image Display," an element frame appears as a line drawing.

It appears as a triangle screen for the shell element and as a trigonal pyramid for the solid elements. It appears in yellow for plastic state and in red when crushed. The compressive failure occurs when the minimum principal strain of plastic element exceeds the crush strain value of the material property (the initial value is 10000) (in other words, when a strain lager than the crush strain occurs toward compression direction).

**Q7. I want to know "tensile stress strength ratio (%)" and "compressive stress strength ratio" in "Contour map" and "Physicul quantity."**

A7. When a crack occurs in the element, "crack risk (%)" is 100%, otherwise is calculated by maximum principal stress × 100/critical stress value, showing a ratio (%) until the element reaches at tensile failure.

When plasticity or crush occurs in the element, "yield risk (%)" is 100%, otherwise is calculated by equivalent stress × 100/yield stress value, showing a ratio (%) until the element reaches at plasticity.

In this software, the analysis of inhomogeneous material (bone) takes a main part; thus, a stress value reaching at failure is different by element. In such a case, these "stress strength ratios" represent a criterion reaching at failure independent on the element.

\* In material non-linear analysis, an element can have only one deformative status. Thus, for example, in case yield risk gets 100% or over after tensile failure, the value has no meaning.

However, in elastic analysis (elements are not broken), the crack / yield risk over 100% is effective. It is useful to recalculate the load value by which the risks are controlled under 100%

**Q8. Sometime analysis stops in the midway. Does it mean the bone fracture?**

A8. Stoppage of analysis in the middle of the load step occurs when the geometry of the model and the material properties become inappropriate; thus, it is unrelated to a "fracture,"

This software does not define the state of "bone fracture" clearly.

By analysis, the information of tensile failure, compressive failure (number, location state, etc.) for each element is obtained. Use the information as a standard when judging a state of "bone fracture." (On display, make "Fracture image" as a reference.)

## Appendix 6 Comparison between Editions

This software has the following two editions in accordance to the purpose.

- Standard edition
   This is an edition aiming the Finite Element Method analysis for a single bone.

- Extended edition
   This is an edition aiming the Finite Element Method analysis for a case where an implant is virtually inserted.

About the difference between the editions, refer to the following table.

# MECHANICAL FINDER

version 13    Function List by each Edition

| | | Standard | Extende |
|---|---|---|---|
| Input File | DICOM | ✓ | ✓ |
| | BMP/JPEG/TIFF | ✓ | ✓ |
| ROI and Phantom | 2D/3D ROI Extraction/Effect processing | ✓(only one group) | ✓ (up to 7 groups) |
| | Bone density calibration phantom setting | ✓ | ✓ |
| Mesh generation | Mesh generation | ✓(only for single bone) | ✓(possible on overlapped models) |
| | Mesh generation for multiple parts | | ✓ |
| | Changing mesh size by part | | ✓ |
| | Implant(stl data) installation | | ✓ |
| | Multi-core meshing | | ✓ |
| Materials | Inhomogeneous material properties for bone | ✓ | ✓ |
| | Translation and rotation of bone | | ✓ |
| | Homogeneous materials | ✓(depending on mesh) | ✓ |
| | Rubber materials(homogeneous) | ✓(depending on mesh) | ✓ |
| | Truss elements defined by strain-stress | | ✓ |
| Boundary condition | Load/Forced displacement condition | ✓ | ✓ |
| | Initial velocity setting | | ✓ |
| Analysis (Finite Element Analysis) | Elasticity analysis/Material nonlinear analysis | ✓ | ✓ |
| | Contact analysis | | ✓ |
| | Dynamic analysis | ✓(no initial velocity setting) | ✓ |
| | Geometrical nonlinear analysis | ✓(depending on mesh) | ✓ |
| Output surface mesh | STL/DXF format | | ✓ |
| Display function | Result display (contour/deformation/etc.) | ✓ | ✓ |
| | Data extraction function (material/result/etc.) | ✓ | ✓ |
| | Image output(BMP/JPEG/TIFF/PNG format) | ✓ | ✓ |
| | Video output(AVI/MPEG format) | ✓ | ✓ |
| | 3D animation output(GFA format) | ✓ | ✓ |
| Graph/ Measurement | Graph drawing function | ✓ | ✓ |
| | Measurement function | ✓ | ✓ |
| Tools | Input from image file(BMP/JPEG/TIFF) | ✓ | ✓ |
| | Text output of project information | ✓ | ✓ |
| | Batch(meshing & analysis) processing program | ✓ | ✓ |
| | Remote batch processing program | ✓ | ✓ |

**Appendix 7 Support**

# <u>Inquiry for MECHANICAL FINDER</u>

Research Center of Computational Mechanics, Inc.

| | |
|---|---|
| About product outline and maintenance, etc. | support@mechanical-finder.com |
| About technical issues | tech@mechanical-finder.com |