

Software for Collapsing Analysis of Wooden Houses

wallstat (ver. 1.09)

User's Manual

Building Research Institute (independent administrative corporation)

What is *wallstat*?

In recent years, major earthquakes have caused serious damage to existing wooden houses, highlighting the need for wooden houses to be endowed with earthquake-proof performance. This field of research implements a great number of response analyses and large-scale tests using shaking tables, yielding an abundance of knowledge regarding the behavior of wooden houses in times of earthquakes. Prof. Takafumi Nakagawa have applied this knowledge to develop a collapse analysis program for assessments of the damage status and likelihood of collapse in whole buildings during times of seismic motion. The reproduction of wooden houses' collapse behavior has been difficult due to the need for consideration of the extremely non-linear properties of members breaking or being dispersed, but this has now become possible by means of an original analysis technique using the basic theory of the Distinct Element Method. *Wallstat* is software that has been improved so that researchers and technicians who specialize in wooden structures can make use of those research results. By using *wallstat*, numerical analysis models can be created on PCs and, by assigning arbitrary seismic motion to the basal level as per shaking table experiments, the size of analysis model transformations and indication of collapse can be assessed.

Targeted structures

This software is designed for timber structures that are built according to general frame wall principles. It can also be applied to large-scale glued laminated timber structures as well as to mid-rise and high-rise wooden buildings, provided that these are built according to general frame wall principles. Program improvements may also enable support for other types of structures (please consult the developer).

Anticipated users

This software is designed for use by researchers and technicians who specialize in wooden structures. There are requirements for digital data on test results from walls/joints, seismic waveforms inputted to the basal level of analysis models, etc. Fundamental knowledge concerning structural analysis is also required.

Usage method

The use of this program is restricted to research or education purposes. Analysis results may differ from reality depending on logical input values; however, the developer cannot be held responsible for analysis results. If you contact the developer if bugs are found in this program, support will be provided to the extent possible.

Other

If this software is used in research results published in theses or at academic meetings, etc., please insert a line of text saying “Building Research Institute's *wallstat* ver.*.* was used for this research.” If possible, please quote the following references.

References:

T. Nakagawa, M. Ohta, et. al. "Collapsing process simulations of timber structures under dynamic loading III: Numerical simulations of the real size wooden houses", *Journal of Wood Science*, Vol.56, No.4, p.284-292 (2010)

Please send all theses, etc. to the developer (address below).

Takafumi Nakagawa
Research Institute for Sustainable Humanosphere, Kyoto University
E-mail: tkfm0820@gmail.com

The developer is keen to incorporate all users' opinions in future improvements of *wallstat*. If you would like to voice any opinions or thoughts, please direct correspondence to the address provided above.

Table of contents

1. Getting started	1
2. Flow of calculations	2
3. Creating analysis models	3
4. Creating parameter files	10
5. Creating external force files	17
6. Modifying calculation condition files	22
7. Executing calculations	23
8. Analysis of calculation results	26

1. Getting started

1.1 About this manual

This manual describes rules of usage for *wallstat*, the software for analytical study of the collapse of wooden houses. Please contact the developer if you wish to reproduce or reprint this document.

1.2 Install/uninstall

Please decompress the compressed software file into the appropriate folder on the PC's hard disk before use. The registry will not be modified. Uninstall is completed by deleting the folder.

1.3 Organization of files

Decompressed files are organized as follows:

gui.exe	...Interface program
calc.exe	...Calculation program
calc_b.exe	...Calculation program (for use with older CPU designs)
manual.pdf	...This manual file
OpenTK.dll	...DLL file required for 3D display
default.ini	...Initialization file
sample/	...Folder containing sample files

1.4 System requirements

OS: Windows[®] XP、Windows[®] Vista、Windows[®] 7

Microsoft .NET Framework ver.2.0 and above (freely available)

Computer: PC or WorkStation suitable for running the Operating Systems above

OpenGL-compatible graphics card (recommended)

*Anti-virus software may block the execution of this software. If this should occur, please alter the settings on the anti-virus program to allow the software to work.

*Please contact the developer before redistributing parts of this software which are yet to be made publicly available.

*The developer will not be held responsible should the use of this software cause any damage to the user's PC, etc.

*This software uses "GLSharp" for 3D display. (See http://sky.geocities.jp/freakish_osprey)

2. Flow of calculations

As shown in the following illustrations, the calculation program (calc.exe) conducts calculations and outputs a results file using information based on the four input files.

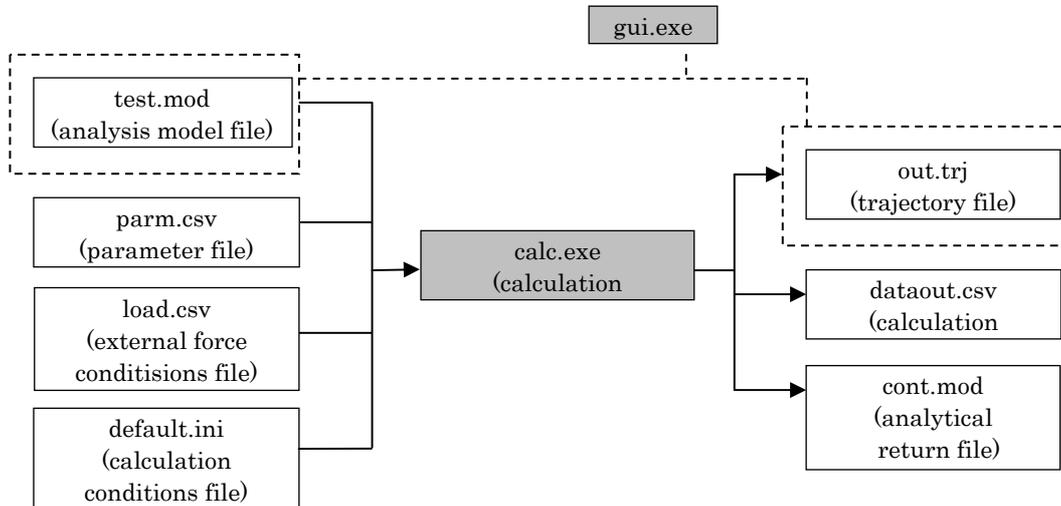


Fig. 2.1 Flow of calculations

【Input connections】

Analysis model file	A component of the analysis model, this file contains the information for the spring. Created using gui.exe. (test.mod)
Parameter file	File containing parameter information for each spring type. (parm.csv)
External force conditions file	File containing conditions of the external force of seismic wave inputs, scale factor inputs, pushover position, etc. (load.csv)
Calculation conditions file	File containing calculation conditions of the external force of calculation frequency, increments, points of view, etc. (default.ini)

【Output connections】

Trajectory file	Saves transient data of the coordinates of each component of the analysis model. This file is used for browsing in gui.exe. (out.trj)
Calculation results	Saves the story shearing on each floor of the analysis model, the designated point of absolute displacement on each floor, and the characteristic values of monitoring-designated components. (dataout.csv)
Analysis continuation file	File containing information concerning springs and components of the analysis model after calculations. As an input file, this file can input consecutive external forces by performing recalculations. (cont.mod)

The chapters below explain how to create input connection files.

3. Creating analysis models

This chapter explains the procedure for creating the analysis model. The analysis model file is created using gui.exe, but the following four CSV (Comma Separated Value) files must be prepared.

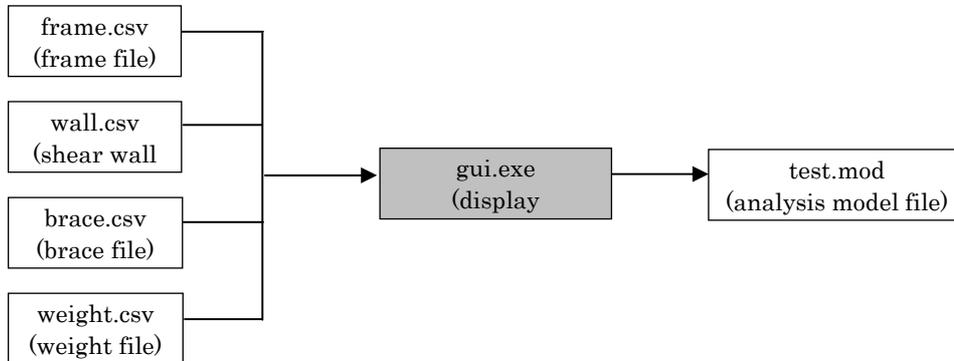


Fig. 3.1 Creation of analysis model file

3.1 Creating frame files

A “frame file” containing information relating to the analysis model frame will be created. The frame file format is as shown in fig. 3.2. Lines starting with “#” are ignored when brought up by comment lines in gui.exe. When creating in Excel and the like, please save as a CSV (Comma Separated Value) file.

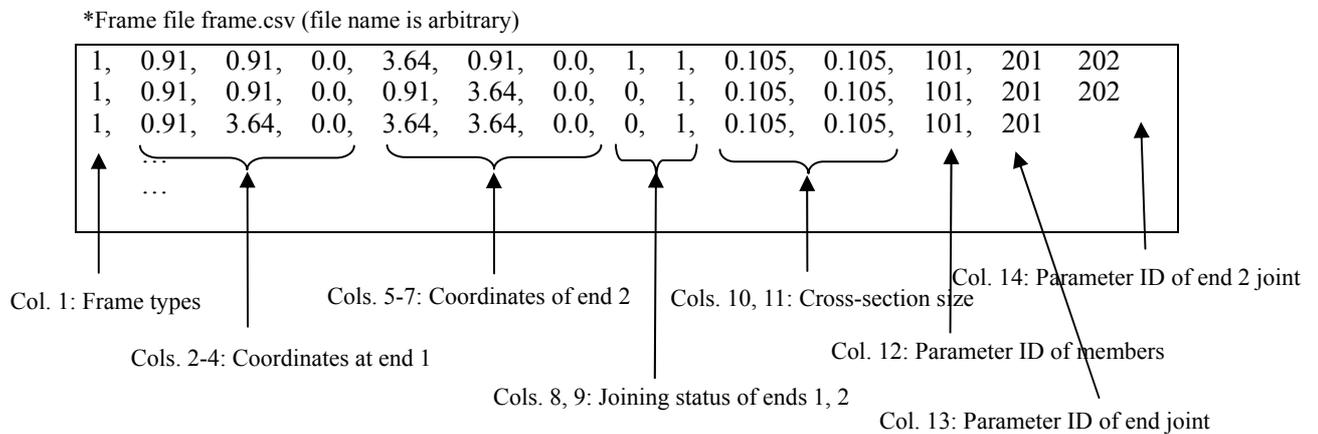


Fig. 3.2 Frame file formats

【Explanation of columns】

Column no.	Explanation
1	Frame types. Horizontal framing members are 1, columns are 2.
2~4	Absolute coordinates of member end 1. X, Y, Z order. (Units: m) Core-to-core specification.
5~7	Absolute coordinates of member end 2. X, Y, Z order. (Units: m) Core-to-core specification.
8	Success or failure of joint of member end 1. Success is 1, failure is 0. (Refer to fig. 3.3) No joint at end is indicated by 1.
9	Success or failure of joint of member end 2.
10	Width of cross-section of member. (Units: m)
11	Reach (height) of member's cross-section. (Units: m)
12	Parameter ID of member. Number designated by the parameter file created in chapter 4.
13	Parameter ID of both ends of member joint.
14	Parameter ID of joint of member end 2. (Omission is acceptable if types differ at each end.)

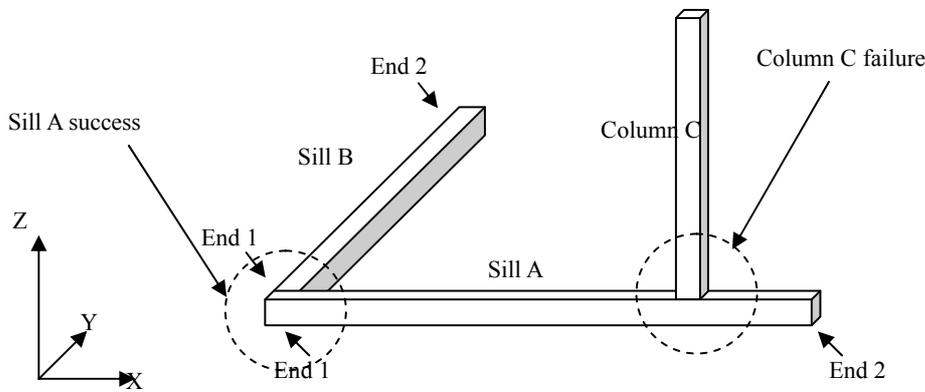


Fig. 3.3 Schematic diagram of frame

Each line in the frame file corresponds to one frame member of the analysis model. Please always set line 1 to the sill level of existing members (for stone-base building, this is the end of the stone base level of the column on floor 1). Coordinate input is 3D by default, but if you wish to conduct 2D analysis please use an X-Z coordinate system so that all Y coordinates are set to 0.

For companion members of identical coordinates, joints (parameter ID of columns 13 and 14) are automatically created for the ends. When this occurs, “failure” member automatically offsets the companion member's proportionate width.

If the example format of the member is the floor plan found in fig. 3.2, the member will be arranged as per fig. 3.4.

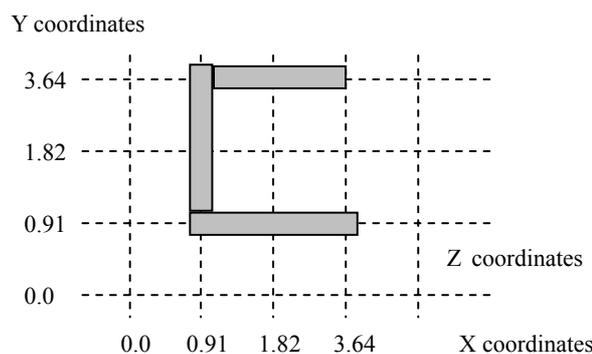


Fig. 3.4 Example of member arrangement in floor plan

3.2 Creating shear wall files

“Wall files” containing information concerning the shear walls of the analysis model will be created. This format is shown in fig. 3.5. This file is designated not only for the vertical wall but also for the horizontal diaphragm. The brace is not contained in this file but is rather designated in the “brace file” as described in the next paragraph.

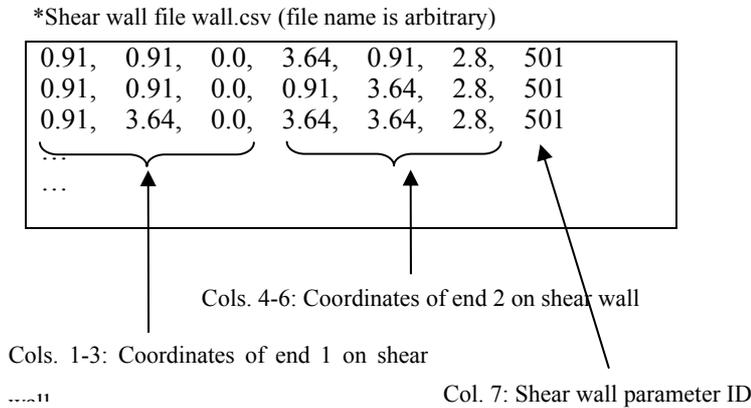


Fig. 3.5 Shear wall file formats

【Explanation of columns】

Column no.	Explanation
1~3	Absolute coordinates of end 1 of shear wall. Refer to fig. 3.6's X, Y, Z order. (Units: m) Core-to-core frame specification. End of frame is required for end.
4~5	Absolute coordinates of end 2 of shear wall. Refer to fig. 3.6's X, Y, Z order. (Units: m) Core-to-core frame specification.
6	Shear wall parameter ID. Number designated in the parameter file created in chapter 4.

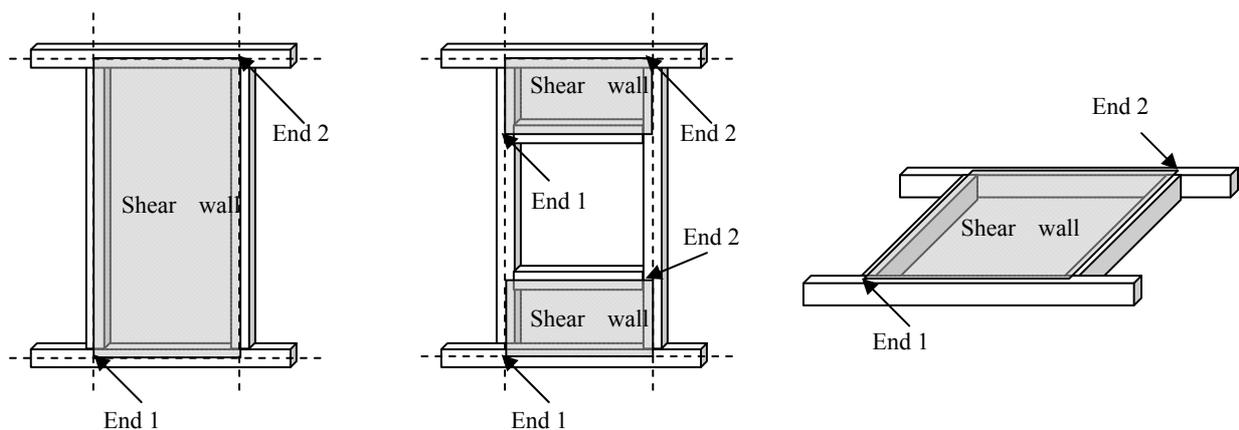


Fig. 3.6 Schematic diagram of shear walls

If there is an opening in the shear wall, please add the horizontal framing members (window seat, lintel beam) for the top and bottom parts of the opening to the frame file, and add the hanging wall and low partition wall to the wall file.

3.3 Creating brace files

“Brace files” containing information concerning the analysis model brace wall will be created. This format is as shown in fig. 3.7.

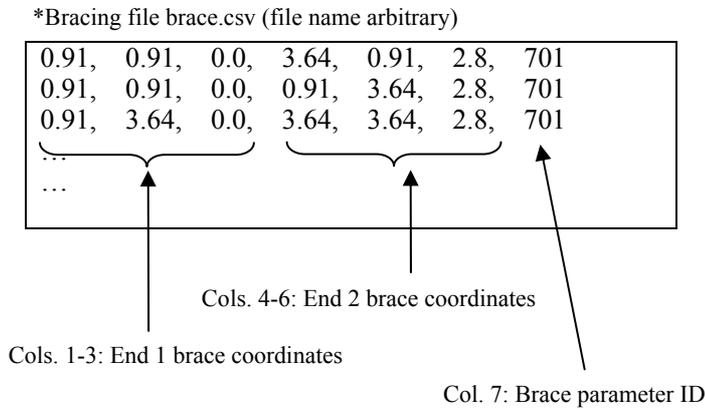


Fig. 3.7 Bracing file formats

【Explanation of columns】

Column no.	Explanation
1~3	Absolute coordinates of brace end 1. Refer to fig. 3.8's X, Y, Z order. (Units: m) Core-to-core frame specification. End of frame is required for end.
4~5	Absolute coordinates of brace end 2. Refer to fig. 3.8's X, Y, Z order. (Units: m) Core-to-core frame specification.
6	Brace parameter ID. Designated by the parameter file created in chapter 4.

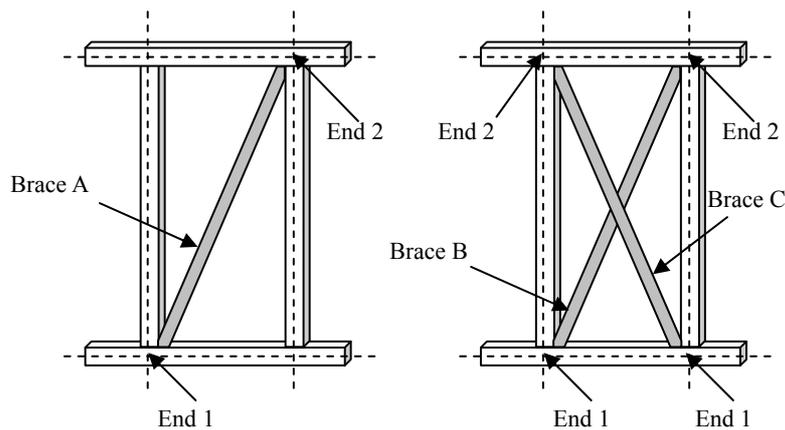


Fig. 3.8 Schematic diagram of shear wall

When two individual braces are braced in a crisscross fashion, please designate these braces in 2 lines.

3.3 Creating weight files

“Weight files” containing information concerning the analysis model weight will be created. This format is as shown in fig. 3.9.

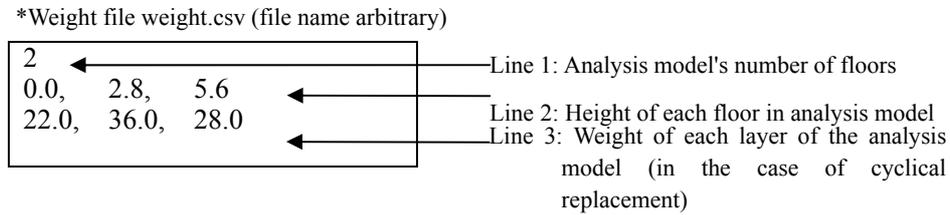


Fig. 3.9 Weight file formats

【Explanation of lines】

Line no.	Explanation
1	Number of floors in analysis model (1-4).
2	Designation of the height of each floor of the analysis model. (Units: m) The first column is GL. Subsequent columns are entered as story 2 floor level, story 3 floor level, etc. The last is the roof beam level. Fig. 3.10 is in the order of h0, h1, h2.
3	The equivalent mass for each story's floor in the analysis model primarily after cyclical replacement. Refer to fig. 3.10. (Units: kN) The first column is half the weight of the floor 1.

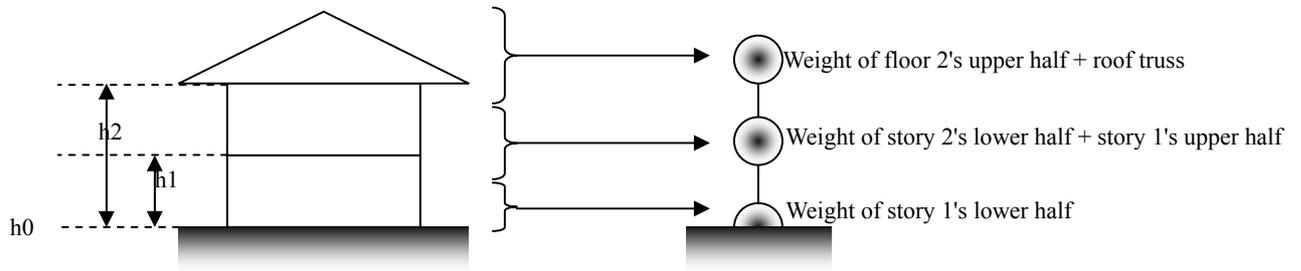


Fig. 3.10 Designated weight of analysis model

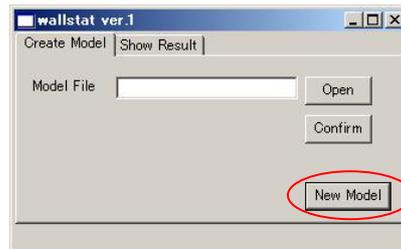
The designated weight of each layer is divided from the number of components existing at the corresponding height and is distributed evenly.

3.4 Creating analysis model files

The following procedure uses the four files created in the preceding paragraphs to create an analysis model.

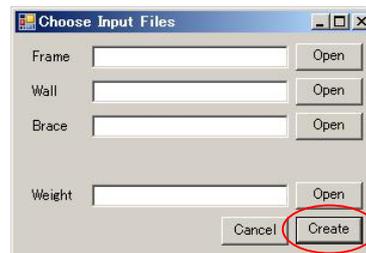
① Launching gui.exe

When gui.exe is launched, the following type of window will appear. Click the “New Model” button in the “Create Model” tab.



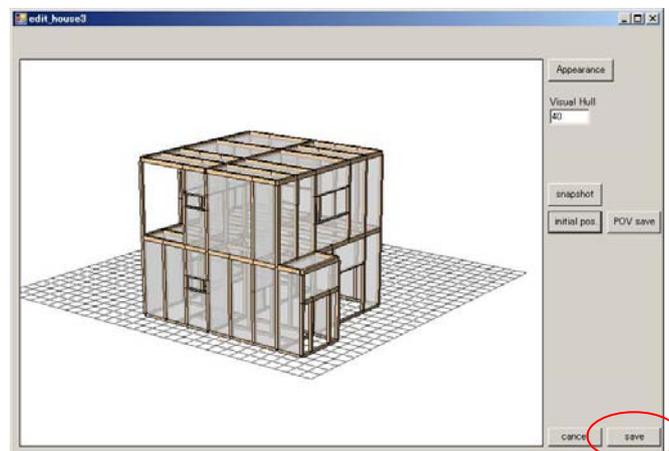
② File selection

Click the “Open” button (found on the same line as Frame, Wall, Brace, Weight) to choose the CSV files created in the preceding paragraphs. Once your selection is complete, click “Create”.



③ Saving analysis model files

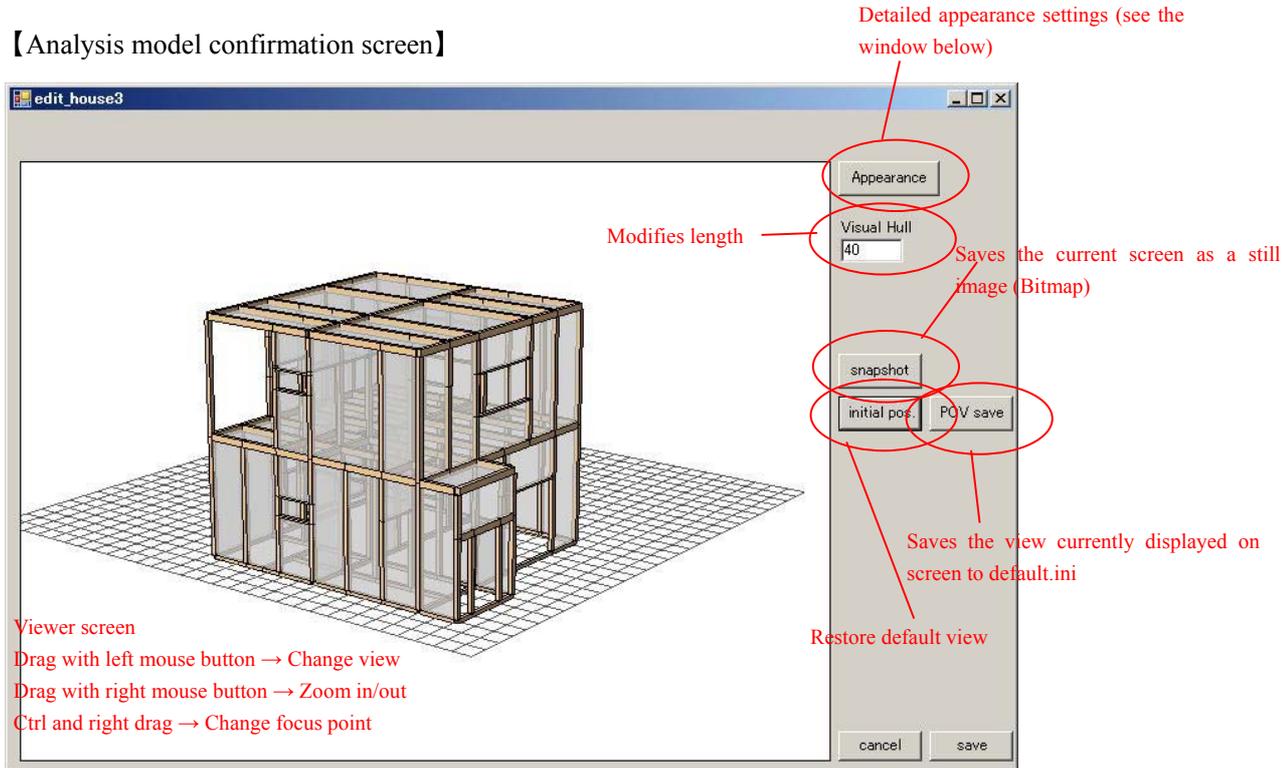
If there are no errors, the following screens will be displayed. Confirm the analysis model in the central window, click “Save” if there are no problems, designate a file name, and an analysis model will be created.



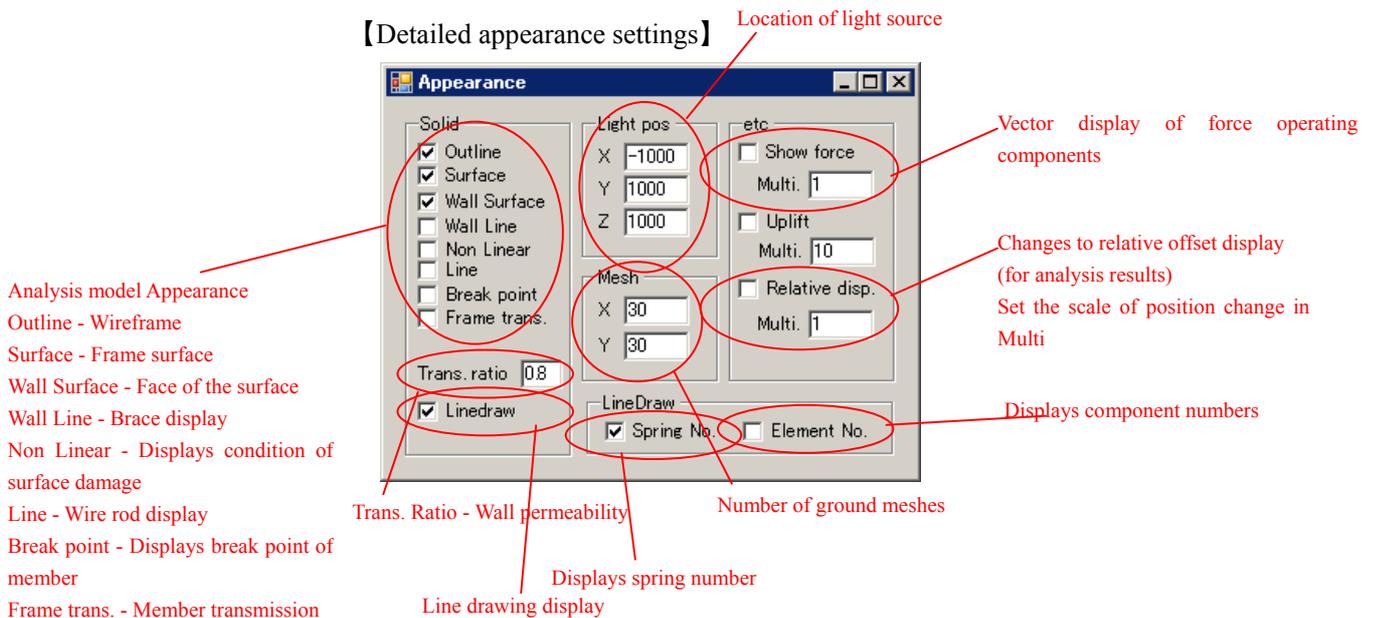
*Analysis model confirmation screen control

In the confirmation screen for the analysis model, you can visually confirm the analysis model using the mouse cursor, Ctrl button, and on-screen buttons. If the screen flickers when using mouse control, or the appearance of the analysis model is not accurate, please check the “Linedraw” box under in the “Appearance” window (this enables line drawing display).

【Analysis model confirmation screen】



【Detailed appearance settings】



4. Creating parameter files

This chapter explains parameter files. The format of parameter files is as found in fig. 4.1.

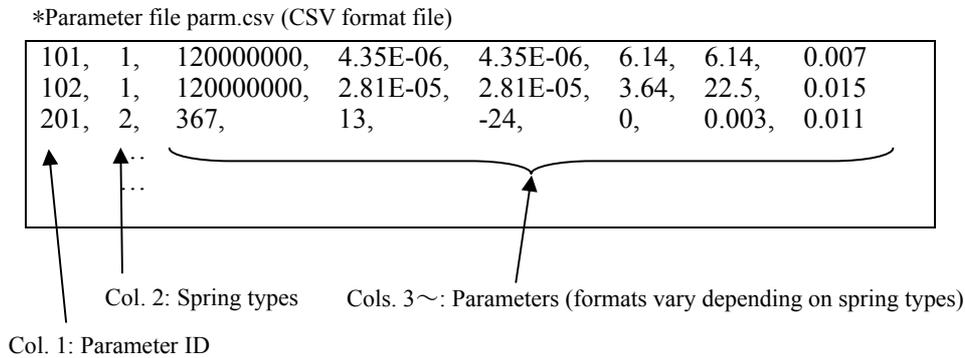


Fig. 4.1 Parameter file formats

【Spring types】

Spring types	Explanation
1	Frame spring. Sets Young's modulus for spring of beam component, bending strength, etc.
2	Joint's tensile spring. Sets the non-linear spring skeleton curve for joint tension.
3	Joint's tensile spring. Sets the non-linear spring skeleton curve in resistance to the joint moment.
5	Shear wall spring. Sets the skeleton curve and damping for the non-linear shear wall spring.
6	Tensile bracing spring. Sets the skeleton curve and damping for the non-linear tensile brace spring.
7	Compression bracing spring. Sets the skeleton curve and damping for the non-linear compression brace spring. Must be part of a set with 6's tensile brace.

Each line corresponds to the parameter of each spring type. For example, when shear walls of varying specifications are mixed, parameters can be set so that IDs are allocated for each specification. Line formats vary in “Col. 2: Spring types.” The tensile brace and compression brace are paired together and modeled as one brace. The parameter ID must be set so that the tensile brace ID (e.g., 601) + 100 equates with the compression brace ID (e.g., 701). Settings methods for each spring type are explained below.

4.1 Frame spring settings

The frame is modeled on the plastic rotational spring (plastic hinge) + elastic component, as indicated on the left-hand side of fig. 4.1. It is defined as being M-θ-related in accordance with the skeleton curve indicated on the right-hand side of fig. 4.1, and the moment starts to fall once the maximum bending moment has been exceeded, with transformation into a pinned state occurring at the point in time when θ is reached, at which point the member is adjudged to have been broken.

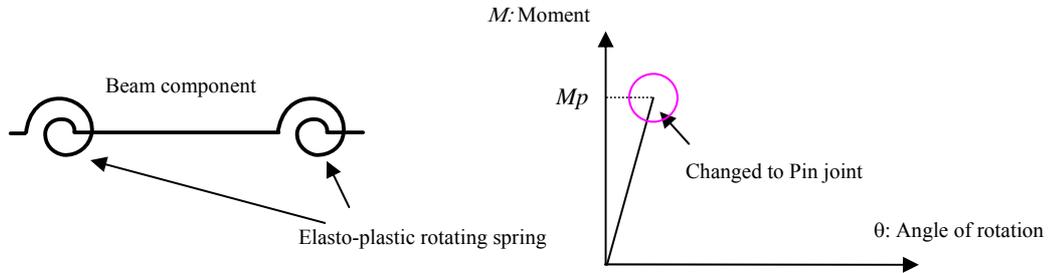


Fig. 4.2 Schematic diagram and skeleton curve of frame spring

The parameter file format for the frame spring is as per Fig. 4.3. If the frame has the same cross-section and members, the same parameter ID will be established as for line 1.

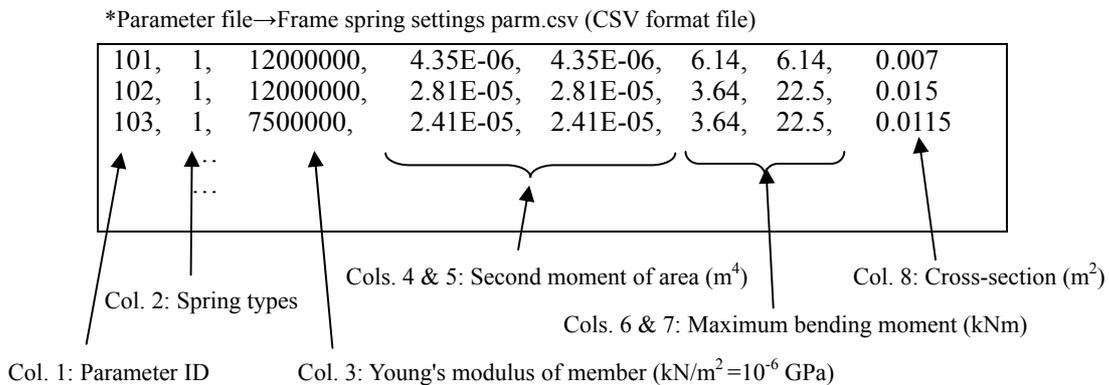


Fig. 4.3 Parameter file formats

【Explanation of columns】

Column no.	Explanation
1	Parameter ID. Arbitrary integers.
2	Spring types. Designated as 1 due to being a frame spring.
3	Young's modulus of member. (Units: kN/m ² =10 ⁻⁶ GPa)
4~5	Second moment of area. (Units: m ⁴) Set in the order of width, reach.
6~7	Maximum bending moment of member (refer to fig. 4.2). (Units: kNm) Set in the order of width, reach.
8	Cross-section of members. (Units: m ²)

4.2 Joint spring settings

As indicated in fig. 4.4, joints are modeled using the rotational spring + elasto-plastic spring (for strong shearing). Log characteristics of the compression/tensile elasto-plastic spring are set as per the one side elastic + one side slip-type indicated in fig. 4.4 (b), and the log characteristics of the rotational spring are set as per the slip-type indicated in fig. 4.4 (c). Rotational springs function independently in each strong axis/weak axis direction. When either the tensile spring or the rotational spring of the joint exceed the maximum structural strength/moment and 0 is reached, it will be adjudged to have been broken and the spring be annihilated. Experimental data for metal fastener joints are compiled in the Sample folder's parameter file.

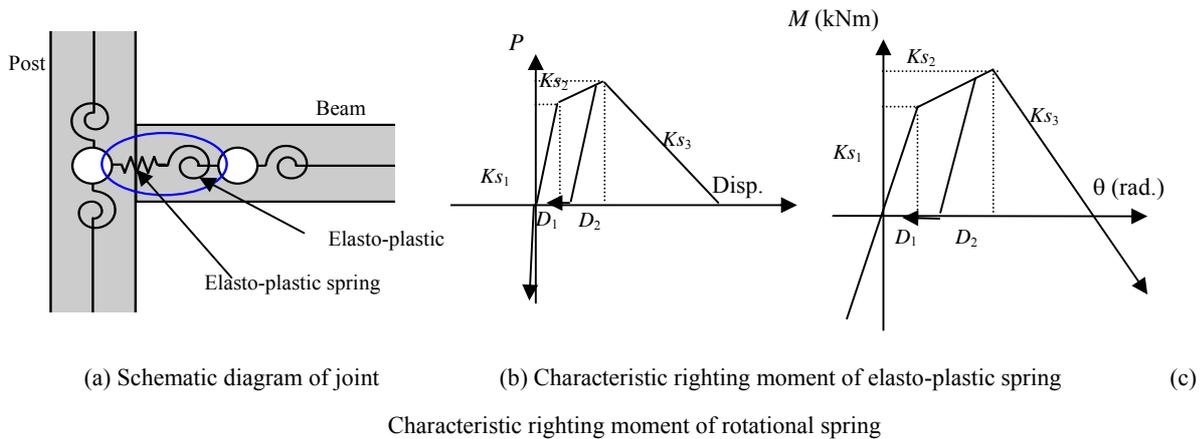


Fig. 4.4 Outline of joint modeling

① Parameters of joint's tensile spring

The parameter file format for the joint's tensile spring is as per fig. 4.5. If the joint has the same specification, the same parameter ID will be established as for line 1.

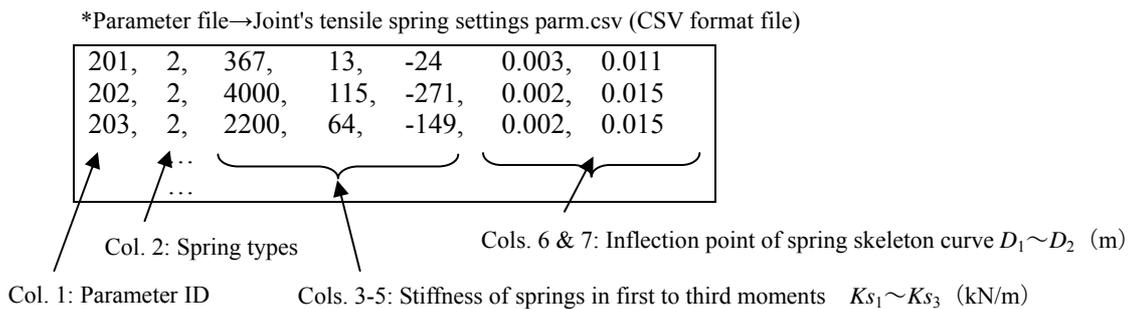


Fig. 4.5 Parameter file formats

【Explanation of columns】

Column no.	Explanation
1	Parameter ID. Arbitrary integers.
2	Spring types. Designated as 2 due to being a joint's tensile spring.
3~5	Stiffness of slip spring in first, second, third moments. (Units: kN/m) (Refer to fig. 4.4(a))
6, 7	Inflection point of slip spring skeleton curve $D_1 \sim D_2$. (Units: m) (Refer to fig. 4.4(b))

*Please modify settings so that second moment stiffness becomes true.

②Parameters of joint's rotational spring

The parameter file format for the joint's rotational spring is as per fig. 4.6. If the joint has the same specification, the same parameter ID will be established as for line 1.

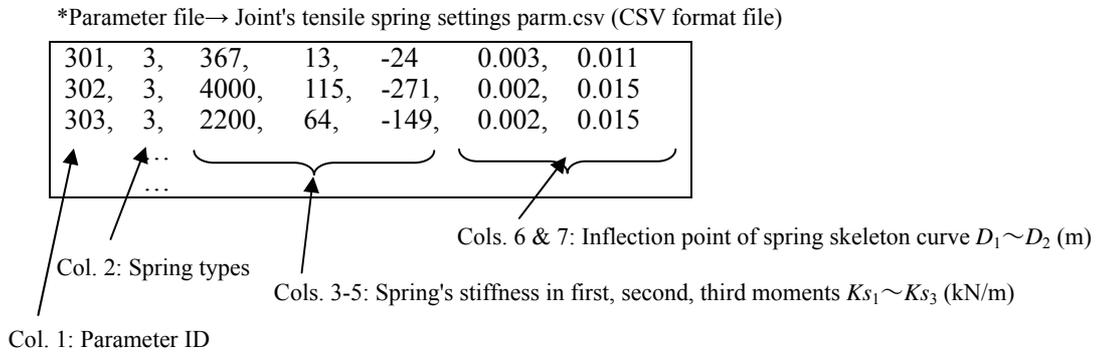


Fig. 4.6 Parameter file formats

【Explanation of columns】

Column no.	Explanation
1	Parameter ID. Arbitrary integers. Designates equal tensile spring as having parameter ID + 100.
2	Spring types. Designated as 3 due to being a joint's rotational spring.
3~5	Stiffness of slip spring in first, second, third moments. (Units: kN m/rad) (Refer to fig. 4.4(b))
6, 7	Inflection point of slip spring skeleton curve $D_1 \sim D_2$. (Units: rad) (Refer to fig. 4.4(b))

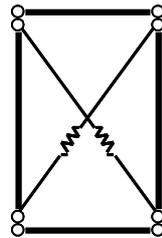
*Please modify settings so that second moment stiffness becomes true.

*The joint's rotational spring and tensile spring are form a pair. Please modify settings so that the tensile spring parameter ID + 100 equates with the rotational spring parameter ID in the same joint. (E.g., tensile spring parameter ID = 201 → rotational spring parameter ID = 301)

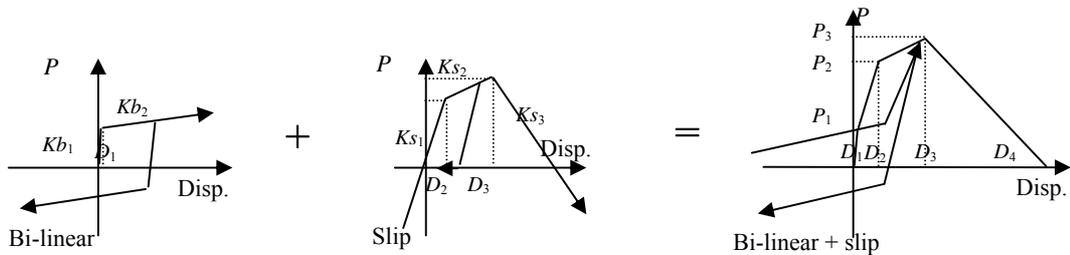
*If the tensile spring parameter ID + 100 is not found in the parameter file, the parameter ID is selected by default as being 301.

4.3 Shear wall spring settings

Vertical shear walls are modeled by the replacement of braces in truss elements as indicated in fig. 4.7 (a). Log characteristics are expressed according to the bi-linear + slip-type log models indicated in fig. 4.7 (b). Parameters designate bi-linear and slip skeleton curves. Parameter values automatically measure the 2 brace replacement springs and calculations are made through angularity correction, so please set as parameters the load-deformation connection (experimental results, etc.) for shear wall measurements 1P (0.91m) × 3P (2.73m).



(a) Shear wall springs (brace replacement)



(b) History characteristics of shear wall

Fig. 4.7 Outline of vertical shear wall

*Parameter file→Shear wall spring settings parm.csv (CSV format file)

501,	5,	0.5,	2.35,	2.45	0,	0.001	0.01	0.075	0.15	0.05
502,	5,	1.0,	4.7,	4.9,	0,	0.001	0.01	0.075	0.15	0.05
503,	5,	1.0,	4.0,	5.0,	0,	0.005	0.1	0.2	0.55	0.05

Col. 1: Parameter ID Col. 2: Spring types Cols. 3-6: Breaking point of spring load $P_1 \sim P_4$ (kN) Cols. 7-10: Breaking point of spring displacement $D_1 \sim D_4$ (m) Col. 11: Viscous damping of spring

Fig. 4.8 Parameter file formats

【Explanation of columns】

Column no.	Explanation
1	Parameter ID. Arbitrary integers.
2	Spring types. Designated as 5 due to being a shear wall spring.
3~6	Breaking point of spring load $P_1 \sim P_4$. (Units: kN) (Refer to fig. 4.7) P_1 is the bi-linear breaking point (location of pinching). P3 is the maximum load. $P_1 < P_2 < P_3$
7~10	Breaking point of spring displacement $D_1 \sim D_4$. (Units: m) (Refer to fig. 4.7) D_1 is the bi-linear breaking point.
11	Viscous damping of spring.

4.4 Bracing spring settings

Bracing shear walls are modeled by the positioning of two compression and tensile truss elements for each brace, as indicated in fig. 4.9. As with shear wall springs, bracing spring log characteristics are expressed according to the bi-linear + slip-type log models. Parameter values automatically measure the two brace replacement springs and calculations are made through angularity correction, so please set as parameters the load-deformation connection (experimental results, etc.) for bracing shear wall measurements 1P (0.91m) × 3P (2.73m). The tensile brace and compression brace are paired together and modeled as one brace. The parameter ID must be set so that the tensile brace ID (e.g., 601) + 100 equates with the compression brace ID (e.g., 701).

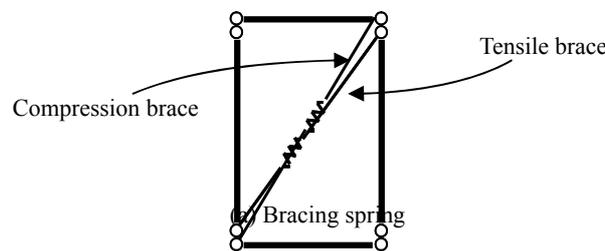


Fig. 4.9 Outline of bracing shear wall

*Parameter file→Shear wall spring settings parm.csv (CSV format file)

601,	6,	0.5,	2.35,	2.45	0,	0.001	0.01	0.075	0.15	0.05
602,	6,	1.0,	4.7,	4.9,	0,	0.001	0.01	0.075	0.15	0.05
701,	7,	1.0,	4.0,	5.0,	0,	0.005	0.1	0.2	0.55	0.05

Col. 1: Parameter ID Col. 2: Spring types Cols. 3-6: Breaking point of spring load $P_1 \sim P_4$ (kN) Cols. 7-10: Breaking point of spring displacement $D_1 \sim D_4$ (m) Col. 11: Viscous damping of spring

Fig. 4.8 Parameter file formats

【Explanation of columns】

Column no.	Explanation
1	Parameter ID. Arbitrary integers. Tensile brace ID (e.g., 601) + 100 must equate to the compression brace ID (e.g., 701).
2	Spring types. For the tensile brace, designate 6. For the compression brace, designate 7.
3~6	Breaking point of spring load $P_1 \sim P_4$. (Units: kN) (Refer to fig. 4.7) P_1 is a bi-linear breaking point (location of pinching). P_3 is the maximum load. $P_1 < P_2 < P_3$
7~10	Breaking point of spring displacement $D_1 \sim D_4$. (Units: m) (Refer to fig. 4.7) D_1 is a bi-linear breaking point.
11	Viscous damping of spring.

The parameter file in the Sample folder compiles shear wall-related data for reference, including the following documents.

- 1) Ministry of Land, Infrastructure, Transport and Tourism Housing Office Construction Guidance “Earthquake Resistance Diagnosis and Reinforcement Techniques for Wooden Housing (1st edition, vol. 7)” The Japan Building Disaster Prevention Association
- 2) Takafumi Nakagawa, Naohito Kawai, Takahiro Tsuchimoto, Minoru Okabe “Verification for Seismic Performance of Existing Wood Houses by Shaking Table Tests : Part 15 Structural Performance of Wooden frame Walls in the Building by Shaking Table Tests” Summaries of Technical Papers from Annual Meeting of Architectural Institute of Japan (Kanto), C-1, p.395-396, 2006

5. Creating external force files

This chapter explains about external force file. There are 5 external force file formats: “seismic wave input (displacement),” “pushover analysis 1,” “pushover analysis 2,” “seismic wave input (acceleration),” and “repeated loading.” Seismic wave input conducts enforced noise input for all base level components, as indicated in fig. 5.1(a). Pushover analysis 1 is for a fixed base level as indicated in fig. 5.1(b), with enforced displacement in a horizontal direction for all components at an arbitrary height within the analysis model. Pushover analysis 2 is for a fixed base level as indicated in fig. 5.1(c), taking the form of a method for gravitational acceleration of the entire analysis model in a horizontal direction. Repeated loading is for fixed components at the base level and conducts repeated enforced displacement in a horizontal direction for all components at an arbitrary height within the analysis model. These file types variously correspond with the formats described in figs. 5.2 through 5.6.

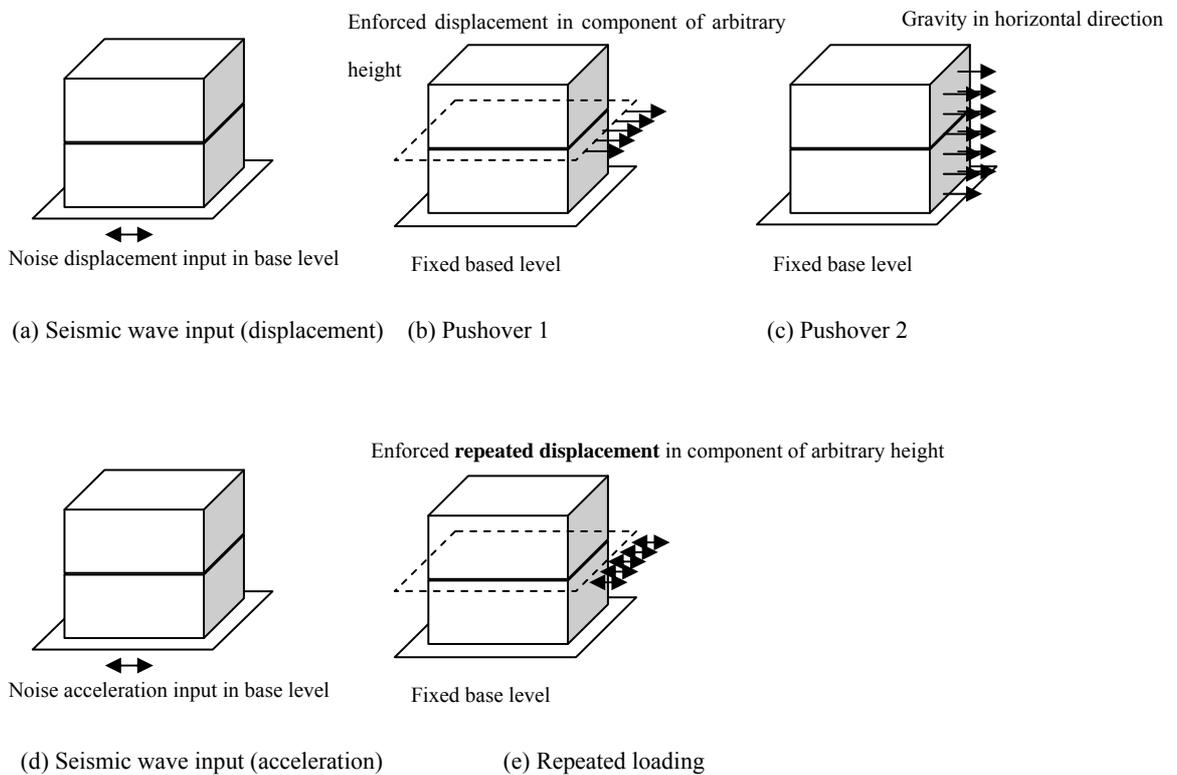


Fig. 5.1 External force input variations

5.1 Seismic wave input (displacement input)

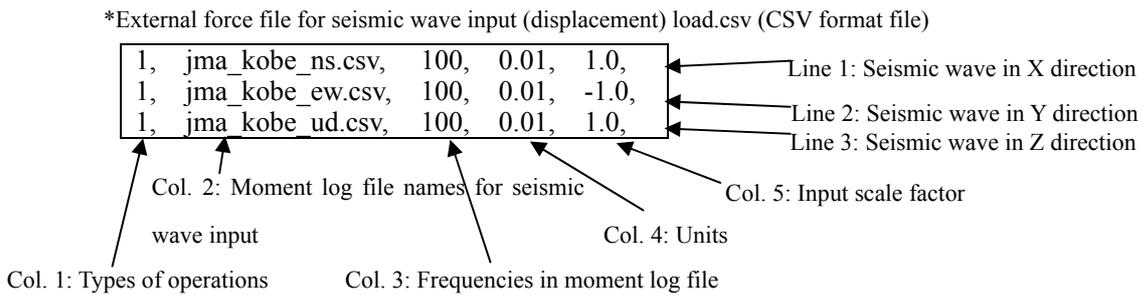


Fig. 5.2 External force file formats when inputting seismic wave displacement

【Explanation of columns】

Column no.	Explanation
1	Operation types for each direction. 0: fixed, 1: seismic wave input (displacement), 2: pushover 1, 3: pushover 2, 4: seismic wave input (acceleration), 5: repeated loading
2	Moment log file names for seismic wave input. The moment log of the displacement waveform is the file named in column 1.
3	Seismic wave file frequencies. (Units: Hz)
4	Units of values in seismic wave file. Scale factor against m. 0.01 when waveform file units are cm. For mm, 0.001. For m, 1.0
5	Input scale factor. Scale factor when amplifying seismic waves. Designating minus will turn the noise input in the opposite plus/minus direction (opposite phase).

When inputting seismic waves, a designated seismic wave waveform file must also be located in the same folder as the external force file. The waveform file must have one string of digital data containing character data. Old seismic waveform digital data can be downloaded from websites such as the address provided below. Seismic motion acceleration input is inputted in the format described in section 5.4 of this document.

3) National Research Institute for Earth Science and Disaster Prevention Kyoshin Network (K-NET)

<http://www.k-net.bosai.go.jp/k-net/>

4) PEER Strong Motion Database

<http://peer.berkeley.edu/smcat/>

The Sample folder contains displacement waveforms from “JMA Kobe” observed at the time of the Great Hanshin earthquake; this data was kindly provided by Chikahiro Minowa⁵⁾.

5) C. Minowa, "Development of a New Method of Baseline Correction on Earthquake Strong Motions and Its Application to Long Period Sloshing Responses of Liquid Storage Tanks During Strong Earthquakes" *Seismic Engineering*, ASME PVP-Vol.466, pp.203-210 (2003)

5.2 Pushover analysis 1

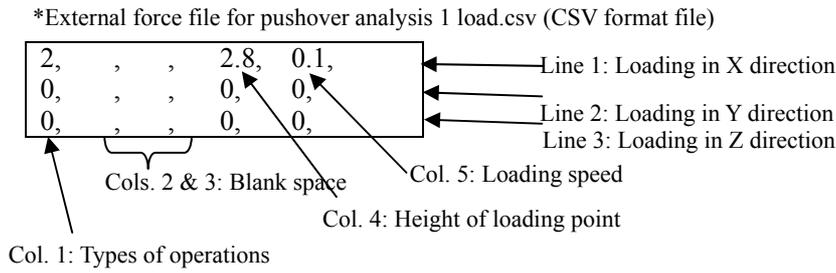


Fig. 5.3 External force file formats for pushover analysis 1

【Explanation of columns】

Column no.	Explanation
1	Operation types for each direction. 0: fixed, 1: seismic wave input (displacement), 2: pushover 1, 3: pushover 2, 4: seismic wave input (acceleration), 5: repeated loading
2 & 3	Blank space. Does not change even if values are entered.
4	Height of loading point. (Units: m)
5	Loading rate. Speed of enforced displacement. (Units: m/sec) Designating minus will turn loading in the opposite plus/minus direction.

5.3 Pushover analysis 2

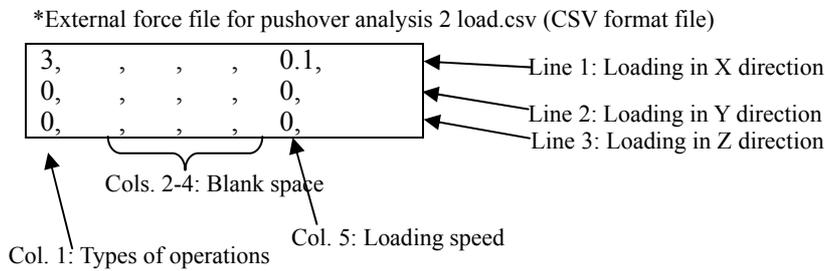


Fig. 5.4 External force file formats for pushover analysis 2

【Explanation of columns】

Column no.	Explanation
1	Operation types for each direction. 0: fixed, 1: seismic wave input (displacement), 2: pushover 1, 3: pushover 2, 4: seismic wave input (acceleration), 5: repeated loading
2~4	Blank space. Does not change even if values are entered.
5	Loading size. Scale factor of gravity inputted in a horizontal direction. The gradual increase of gravity from 0G is designated as the gravity in a horizontal direction reached in 1 second. (Units: G) Designating minus will turn loading in the opposite plus/minus direction.

5.4 Seismic wave input (acceleration input)

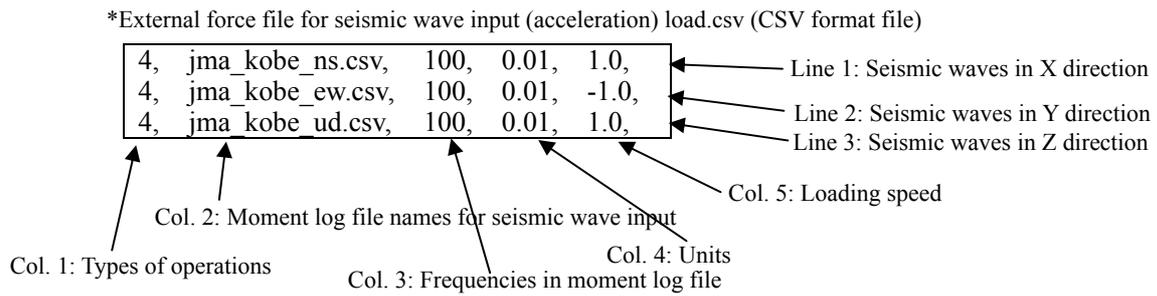


Fig. 5.5 External force file formats when inputting seismic wave acceleration

【Explanation of columns】

Column no.	Explanation
1	Operation types for each direction. 0: fixed, 1: seismic wave input (displacement), 2: pushover 1, 3: pushover 2, 4: seismic wave input (acceleration), 5: repeated loading
2	Moment log file names for seismic wave input. The moment log of the acceleration waveform is the file named in column 1.
3	Seismic wave file frequencies. (Units: Hz)
4	Units of values in seismic wave file. Scale factor against m/s^2 . 0.01 when waveform file units are gal. For G, 9.8. For m/s^2 , 1.0.
5	Input scale factor. Scale factor when amplifying seismic waves. Designating minus will turn the noise input in the opposite plus/minus direction (opposite phase).

When confirming calculation results of accelerated input of seismic waves in gui.exe, and the displacement waveform integrated in the waveform acceleration used in the calculation by means of the average integral has not reached 0, the apparent absolute displacement of the analysis model may appear to have shifted. If this occurs, please check the “Relative disp.” box in “Appearance” settings and conduct confirmation.

5.5 Repeated loading

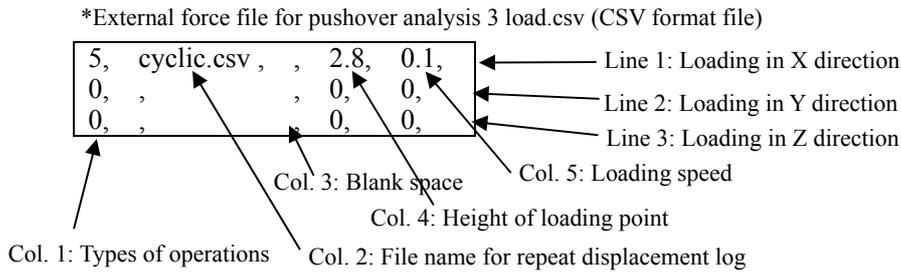
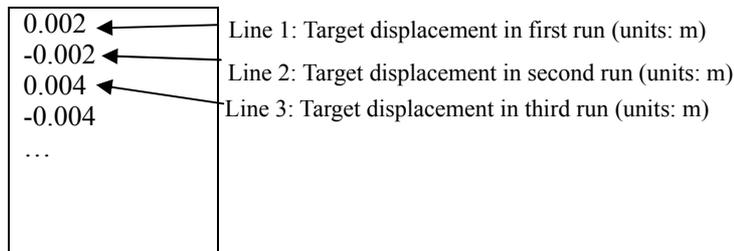


Fig. 5.6 External force file formats for pushover analysis 1

【Explanation of columns】

Column no.	Explanation
1	Operation types for each direction. 0: fixed, 1: seismic wave input (displacement), 2: pushover 1, 3: pushover 2, 4: seismic wave input (acceleration), 5: repeated loading
2	Name of file containing target displacement log for repeated loading.
3	Blank space. Does not change even if values are entered.
4	Height of loading point. (Units: m)
5	Loading rate. Speed of enforced displacement. (Units: m/sec)

When carrying out repeated loading, a designated file containing a target displacement log must also be located in the same folder as the external force file. The log file must have one string of digital data containing character data, as per fig. 5.7. A sample external force file and target displacement file can be found in 2p_wall inside the Sample folder.



6. Modifying calculation condition files

This chapter explains about calculation condition files. The calculation condition file “default.ini” is enclosed in *wallstat* at the time of distribution, but its contents must be modified when using the file for calculations. Calculation condition file formats are as follows. Please edit using Notepad, Editor, or similar programs. Line 2 is not to be edited as it contains visual point information.

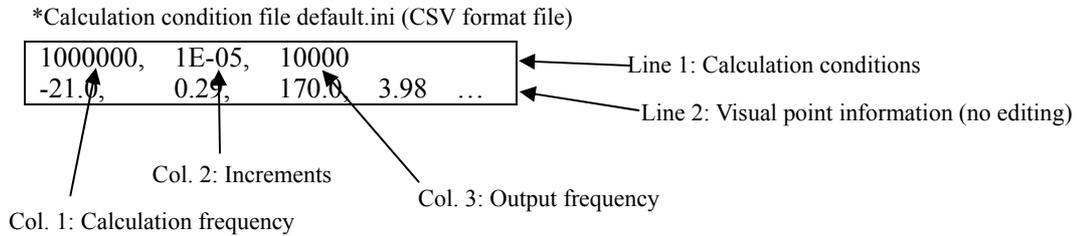


Fig. 6.1 Calculation condition file formats

【Explanation of columns】

Column no.	Explanation
1	Calculation frequency. Calculation frequency x increment equals the points in the analysis.
2	Calculation clock tick. Increments. Δt. *Please do not modify this.
3	Output frequency. Designates the output frequency of results when making calculations. For example, if it is set to 10000, then an image snapshot 1 time in 10000 times will be saved to out.trj, and from this analytical load/deformation information 1 time in 1000 times (1/10 of the previous rate) will be saved to dataout.csv. The number of image snapshots is limited to 1000, so please do not set the calculation frequency/output to exceed that.

7. Executing calculations

This chapter explains the method of executing calculations. Calculations will be executable if all necessary input files from the previous chapter are present. For calculation execution, put the four input files in fig. 6.1 inside the same folder as calc.exe and double-click on calc.exe to begin the process.

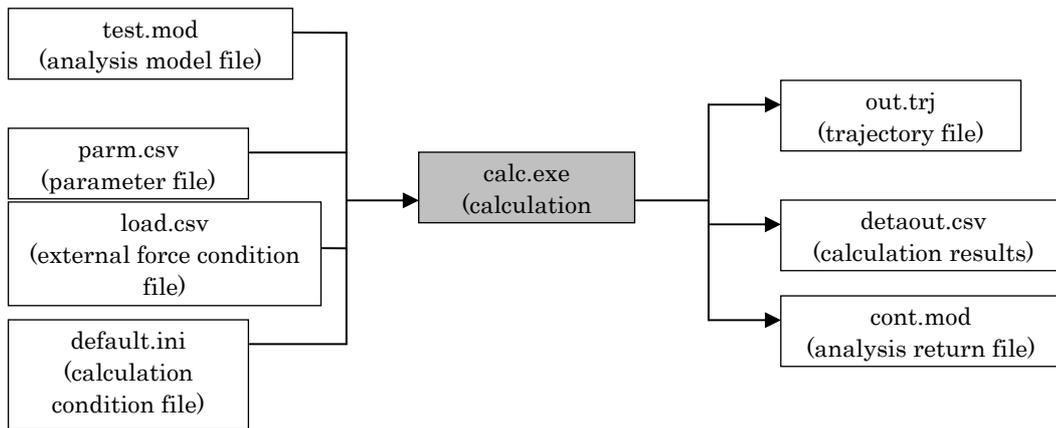


Fig. 6.1 Execution of calculations

7.1 How to begin calculations

A DOS window will appear with the message, “Please select a calculation method.” Enter “1” to retrieve the four test files in “test.mod” to begin calculations.

Select “2” to retrieve “cont.mod” and continue the previous calculation. **It is essential to use the names provided in fig. 6.1 for the four input files.** When entering data for seismic waves, waveform files are necessary in addition to the above mentioned input files.

Progress during calculations is shown as a percentage. 3 output files will be created when calculations have finished, and the DOS window will be closed.

Rough estimates of calculation durations are as follows.

Calculation environment: CPU: Core 2 Duo 2.8GHz

RAM: 2GHz Windows Vista PC

Calculation contents: Calculation time for a 20-second seismic wave input on the 2-storey wooden building in the sample: approximately 20-30 minutes.

If an error means that calc.exe does not execute calculations correctly, please try running calc_b.exe instead.

*Antivirus software may block the execution of this software.

If this occurs, please set this software as an exception in your antivirus software.

7.2 Monitoring of physical quantity of each component and spring

When performing calculations, the physical quantity (absolute displacement, weight, moment etc.) of each component and spring can be outputted by placing the “data.csv” file in the same folder as the “calc.exe” file.

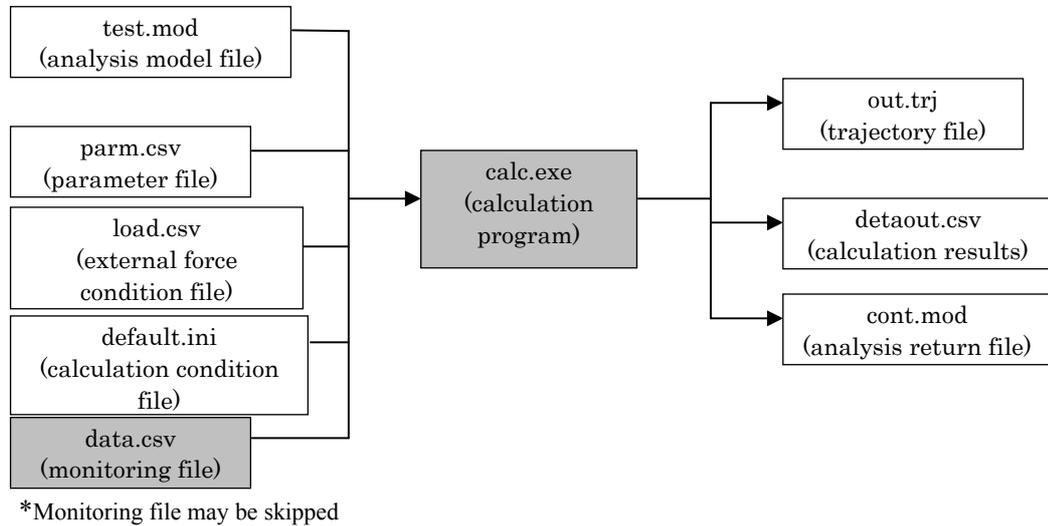
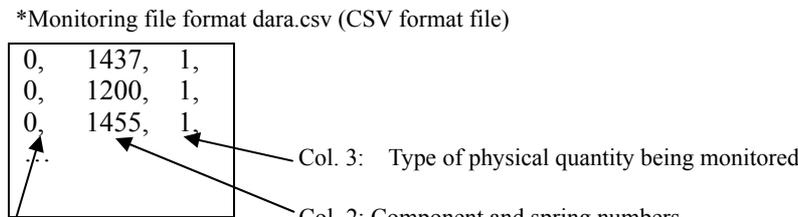


Fig. 6.2 Execution of calculations

The format of monitoring file data.csv is as per fig. 6.3. Components and spring numbers to be monitored are designated on each line. The confirmation method for components and spring numbers is provided on the next page.



Col. 1: Designation of component or spring

Fig. 5.6 External force file formats for pushover analysis 1

【Explanation of columns】

Column no.	Explanation
1	Designation of spring or component as monitoring subject. 0: component, 1: spring.
2	Component and spring numbers (confirmation method on next page).
3	Type of physical quantity being monitored (numbers designation in tables 5.1-5.2).

The type of physical quantity being monitored is designated using the following numbers.

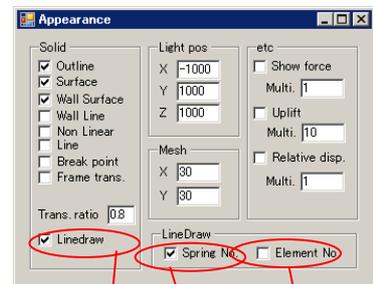
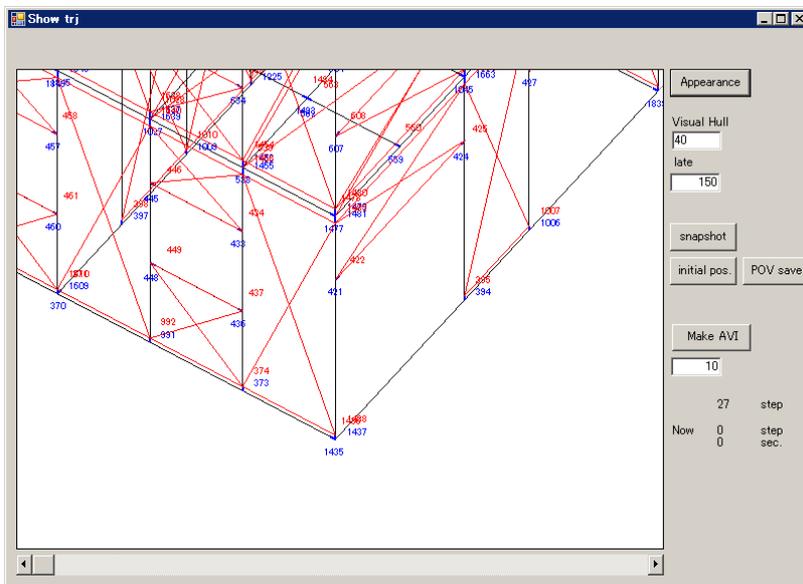
Table 5.1 Physical quantities that can be monitored in components

Physical quantity no.	Physical quantity
1	Displacement x
2	Displacement y
3	Displacement z
4	Force f_x
5	Force f_y
6	Force f_z

Table 5.2 Physical quantities that can be monitored in springs

Physical quantity no.	Physical quantity
1	Displacement d
2	Load P
3	Moment of end 1 M_{y1}
4	Moment of end 1 M_{z1}
5	Moment of end 2 M_{y2}
6	Moment of end 2 M_{z2}
7	Angle of rotation of end 1 θ_{y1}
8	Angle of rotation of end 1 θ_{z1}
9	Angle of rotation of end 2 θ_{y2}
10	Angle of rotation of end 2 θ_{z2}
11	Axial load P_x

Component and spring numbers are displayed by checking “LineDraw” in “Appearance” in gui.exe and selecting either “Spring No.” or “Element No.” Blue text is for joint springs, red text is for rotational springs. If the numbers displayed are overlapping, please rotate the viewpoint.



Displays line

Displays spring numbers

Displays component numbers

8. Analysis of calculation results

This chapter explains the analysis of calculation results.

8.1 Analyzing calculation results

All kinds of calculation results are outputted to “dataout.csv,” the file created after calculations. The order of results is as follows: foundation counter-force, horizontal force of each floor, absolute ground displacement, absolute displacement of each floor. Moment logs are kept, with the period covered being the reciprocal number of “frequency x increment” designated in default.ini.

The “absolute displacement of each floor” is recorded in a moment log of absolute displacement at the 4 corners of each floor in the analysis model. The value of each physical property is counted by the following calculations.

Story 1 shearing power (base shear) = foundation counter-force

Story i shearing power = sum of the horizontal force of floors i and above

Relative displacement of floor i = (i + absolute displacement of floor 1) – (absolute displacement of floor i)

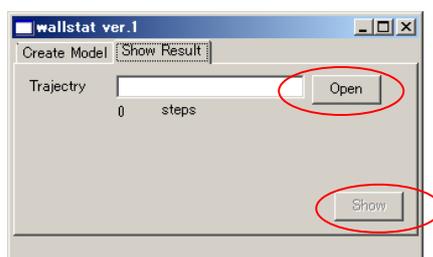
As for absolute displacement in dataout.csv, for example the formula “2F_x+y+_x” shows absolute displacement of the highest FL level height in direction X as well as direction X of the highest corner in direction Y at 2F, which is designated in the weight file. “3F_x-y+_y” shows absolute displacement of the lowest point of FL level height in direction as well as direction Y of the highest corner in direction Y at 3F.

8.2 Confirming calculation results in animations

Snapshots (trajectories) of analysis models during calculation are saved in “out.trj,” which is created after the calculation. gui.exe is used to confirm trajectories. All processes are displayed below:

① Launching gui.exe

When gui.exe is launched, the following type of window will appear. Click on the “Open” button in the “Show Result” tab and select from the dialog box the “*.trj” file to be displayed. This may take a few minutes to retrieve.

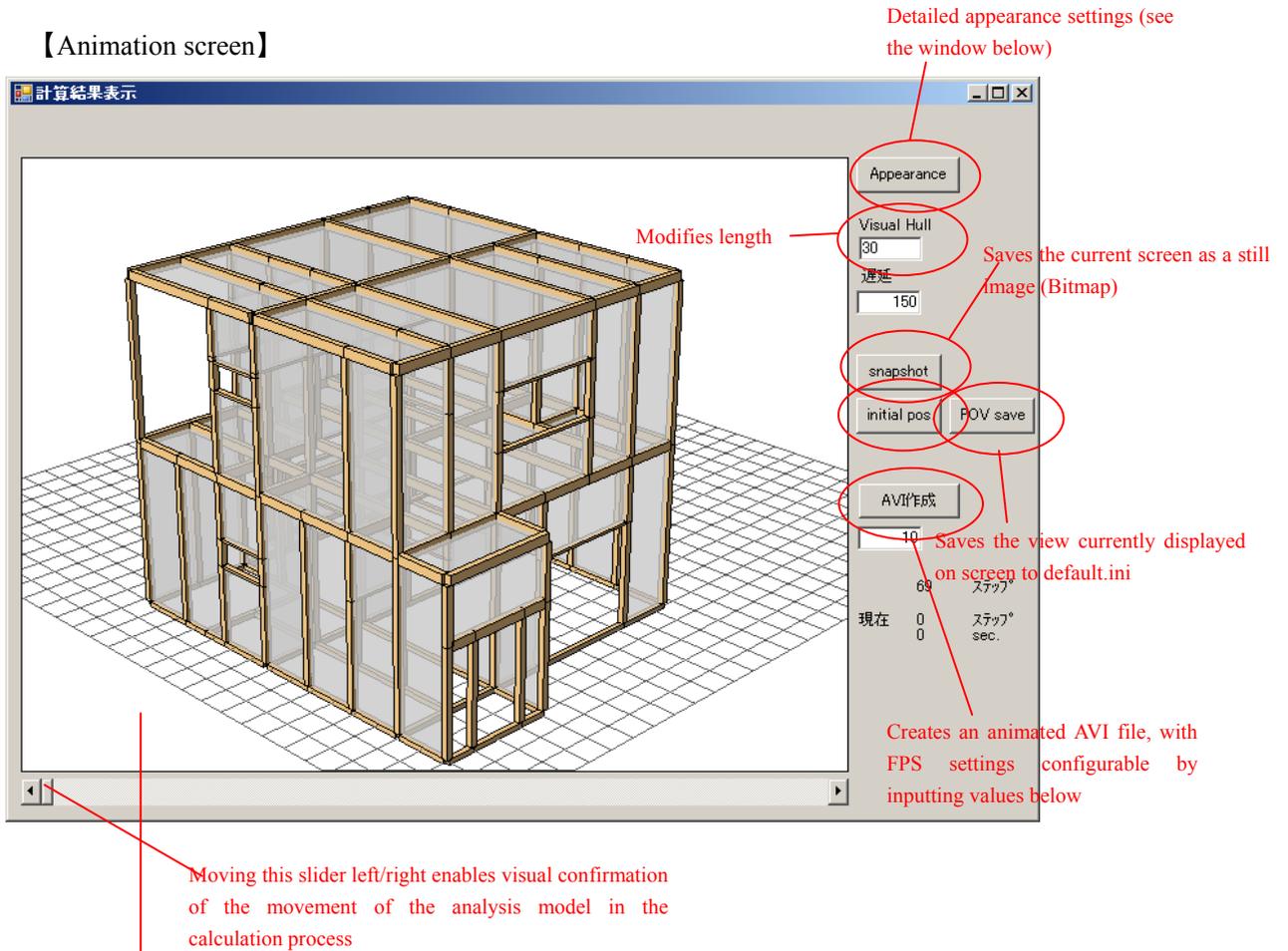


Once the file has been retrieved the “Show” button will become active; click on the “Show” button to bring up the animation screen. Operation of the animation screen is largely similar to that of the analysis model confirmation screen described on page 9.

*Operation of animation screen

In the animation screen, you can visually confirm calculation results using the mouse cursor, Ctrl button, and on-screen buttons.

【Animation screen】



- Viewer screen
- Drag with left mouse button → Change view
- Drag with right mouse button → Zoom in/out
- Ctrl and left drag → Change focus point