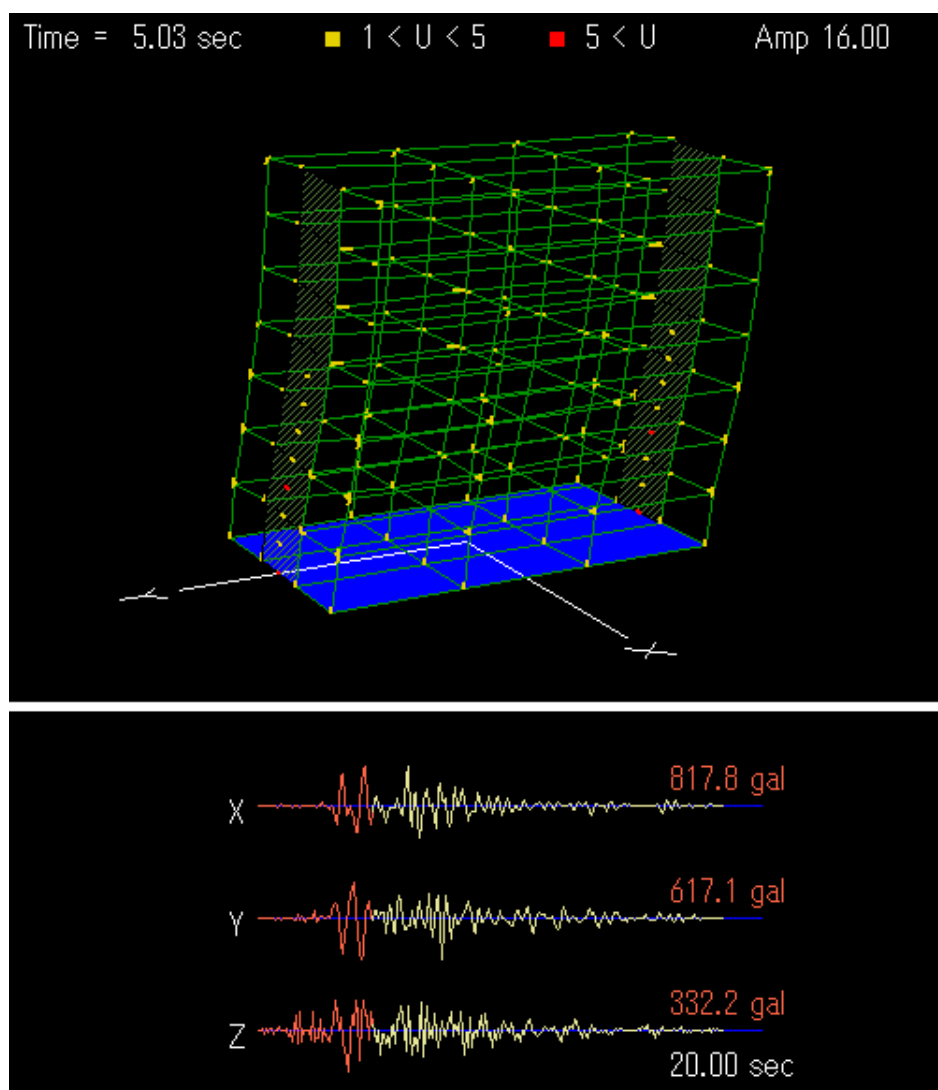


# *STERA 3D ver.11.6*

## *Structural Earthquake Response Analysis 3D*



Dr. / Prof. Taiki SAITO

TOYOHASHI UNIVERSITY OF TECHNOLOGY (TUT), JAPAN

## Preface

This software is developed for the following analyses of steel / reinforced concrete / steel reinforced concrete / seismic isolation / response control / buildings:

- 1) Linear modal analysis,
- 2) Nonlinear static push-over analysis,
- 3) Nonlinear cyclic analysis
- 4) Nonlinear earthquake response analysis.

This software is distributed for free for the use of research and educational purposes.

Since this software is still under development, the author cannot take any responsibility for the results of the software. It is greatly appreciated to have any opinion for future improvement.

1 March, 2015

Taiki SAITO  
E-mail: [tsaito@ace.tut.ac.jp](mailto:tsaito@ace.tut.ac.jp)  
Professor, Dr. of Engineering,  
Toyohashi University of Technology,  
Japan

## Update history

2016/11/12	STERA_3D Ver.8.5 is uploaded. <ul style="list-style-type: none"><li>• Minor change of input data for Wall (direct input)</li></ul>
2016/12/03	STERA_3D Ver.8.6 is uploaded. <ul style="list-style-type: none"><li>• Minor error in static analysis using Mode distribution is fixed.</li></ul>
2016/12/11	STERA_3D Ver.8.7 is uploaded. <ul style="list-style-type: none"><li>• Modification of stiffness degradation factor at the yield point of beam elements is adopted (see the detail in Technical Manual).</li></ul>
2016/12/25	STERA_3D Ver.8.8 is uploaded. <ul style="list-style-type: none"><li>• Bugs of masonry element and connection panel were fixed.</li></ul>
2017/01/18	STERA_3D Ver.8.9 is uploaded. <ul style="list-style-type: none"><li>• Effective modal mass is shown in Modal analysis</li><li>• You can set a different mass value in each node</li></ul>
2017/03/20	STERA_3D Ver.9.0 is uploaded. <ul style="list-style-type: none"><li>• You can set the parameters of band-pass filter to get the ground displacement.</li></ul>
2017/08/01	STERA_3D Ver.9.1 is uploaded. <ul style="list-style-type: none"><li>• Minor change of the format of output file</li></ul>
2017/09/11	STERA_3D Ver.9.2 is uploaded. <ul style="list-style-type: none"><li>• Hardening model is added for NRB of Seismic Isolator</li></ul>
2017/10/08	STERA_3D Ver.9.3 is uploaded. <ul style="list-style-type: none"><li>• Ground spring is added.</li></ul>
2017/10/24	STERA_3D Ver.9.4 is uploaded. <ul style="list-style-type: none"><li>• The default setting of “upper beam” of damper and masonry is changed to be “rigid” beam instead of “none”.</li></ul>
2017/11/27	STERA_3D Ver.9.6 is uploaded. <ul style="list-style-type: none"><li>• Fixed a mistake in mass setting of Ver. 9.4. Note that mass is set correctly in Ver. 9.3.</li></ul>
2019/02/03	STERA_3D Ver.10.0 is uploaded. <ul style="list-style-type: none"><li>• You can set a vibrator on a floor to shake the building.</li><li>• You can execute the program from command line.</li><li>• Automatic generator for lumped mass model is implemented.</li><li>• You can set an original load distribution for static analysis</li><li>• A new nonlinear spring is added for passive damper</li></ul>
2019/05/20	STERA_3D Ver.10.1 is uploaded. <ul style="list-style-type: none"><li>• You can consider radiation damping for ground spring.</li></ul>
2019/07/25	STERA_3D Ver.10.2 is uploaded. <ul style="list-style-type: none"><li>• You can apply dynamic wind forces to the building.</li></ul>
2019/10/08	STERA_3D Ver.10.3 is uploaded.

	<ul style="list-style-type: none"><li>• The buckling of steel members can be considered.</li><li>• You can conduct continuous analysis for dynamic inputs (earthquake and wind).</li></ul>
2020/03/16	STERA_3D Ver.10.4 is uploaded. <ul style="list-style-type: none"><li>• Pile can be considered for ground spring.</li><li>• Air spring has been added to the vertical spring.</li></ul>
2020/04/14	STERA_3D Ver.10.5 is uploaded. <ul style="list-style-type: none"><li>• You can set part of the floor to be rigid.</li></ul>
2020/06/11	STERA_3D Ver.10.6 is uploaded.
2020/08/04	STERA_3D Ver.10.7 is uploaded. <ul style="list-style-type: none"><li>• You can select rebar size from the table.</li></ul>
2020/09/24	STERA_3D Ver.10.8 is uploaded.
2021/10/10	STERA_3D Ver.11.0 is uploaded. <ul style="list-style-type: none"><li>• For RC column and RC wall, the nonlinear bending springs independent in x and y directions are introduced.</li><li>• For Steel beam, the nonlinear shear spring for hysteresis damper is introduced.</li><li>• Damage indices of members are introduced.</li></ul>
2022/08/22	STERA_3D Ver.11.1 is uploaded. <ul style="list-style-type: none"><li>• The input windows for columns and beams (direct input for parameters of hysteresis model) are changed.</li><li>• "Base Plate" and "Pendulum Spring" are added to External Spring.</li></ul>
2022/12/14	STERA_3D Ver.11.2 is uploaded. <ul style="list-style-type: none"><li>• FPB (Friction Pendulum Bearing) is added to the seismic isolation elements.</li></ul>
2023/03/10	STERA_3D Ver.11.3 is uploaded. <ul style="list-style-type: none"><li>• The formula of compression strength of Masonry element is changed.</li></ul>
2023/06/06	STERA_3D Ver.11.4 is uploaded. <ul style="list-style-type: none"><li>• Viscoelastic damper is added to the passive damper.</li><li>• You can set the restrained freedom of the partial rigid slab at the center of gravity.</li></ul>
2024/07/15	STERA_3D Ver.11.5 is uploaded. <ul style="list-style-type: none"><li>• Viscoelastic damper is added to the shear spring of direct beams.</li><li>• Some output files are in csv format.</li></ul>
2025/08/02	STERA_3D Ver.11.6 is uploaded. <ul style="list-style-type: none"><li>• Nonlinear model of the viscoelastic damper is modified.</li><li>• Vertical viscous damper is added to the external spring.</li><li>• The model of HDRB (high damping rubber bearing) is updated.</li></ul>

## Quick User Manual

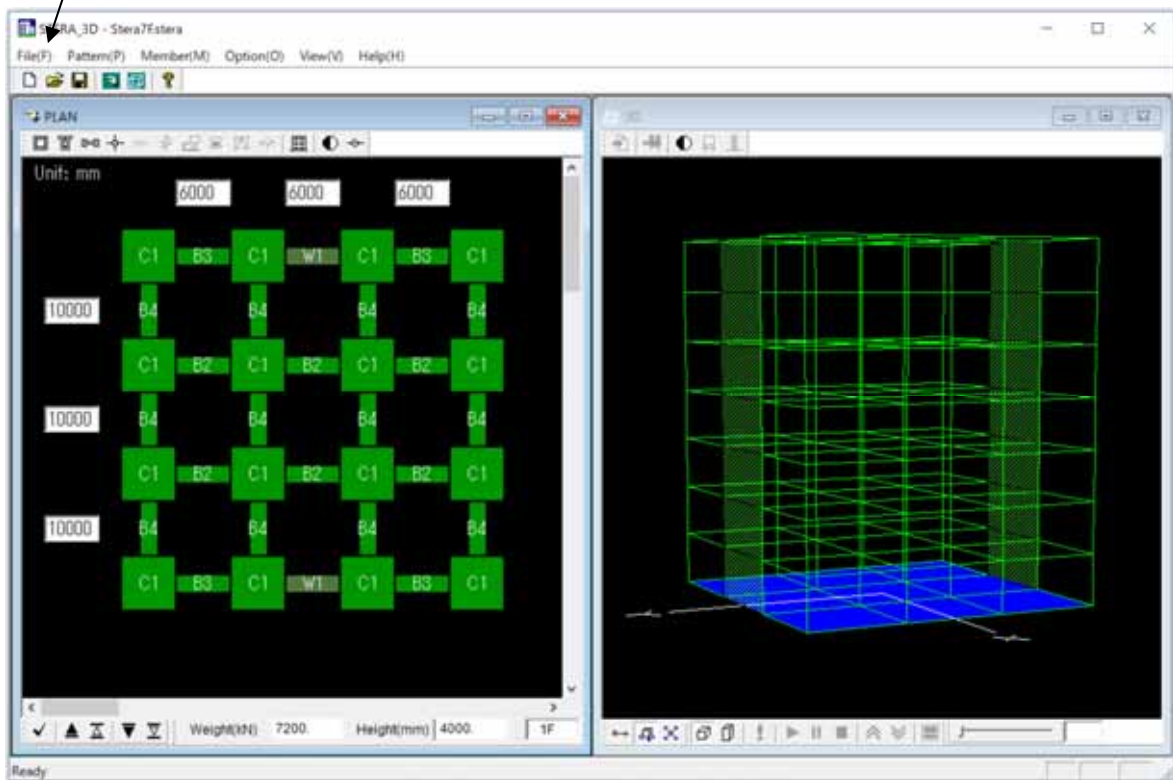
## READ BUILDING DATA

Double Click

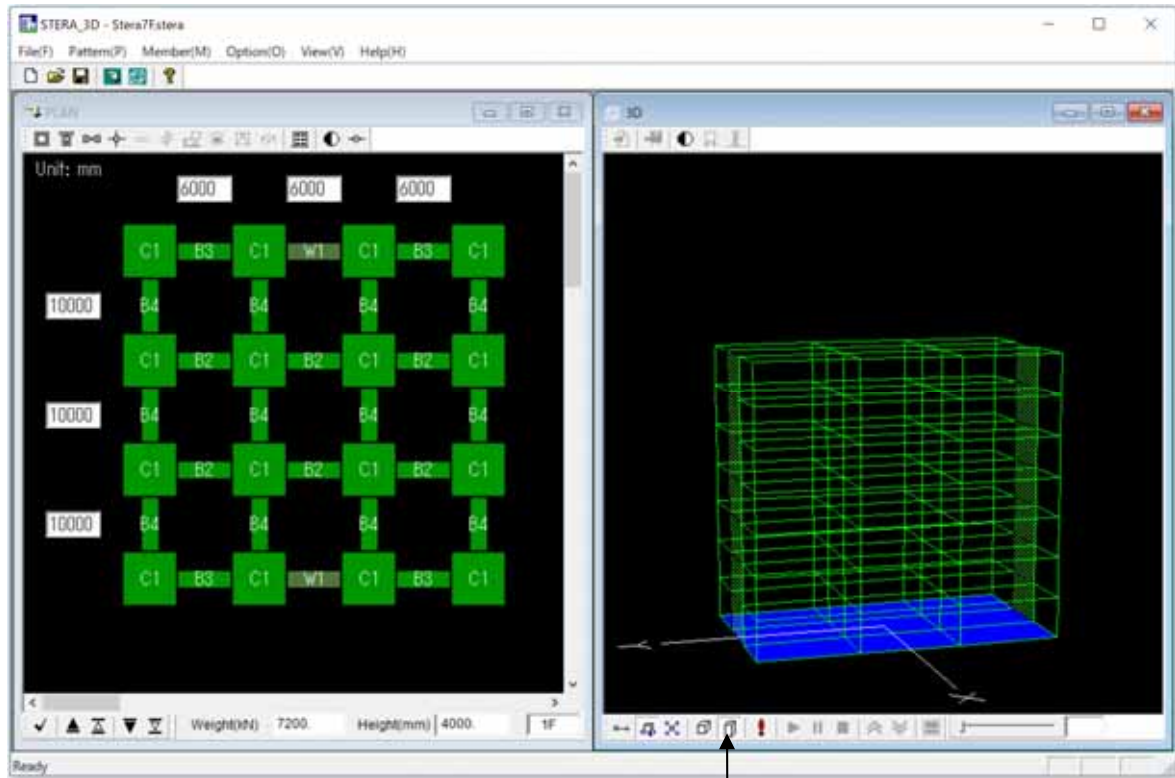



“File” → “Open”

Select an example building “Structure7F.stera”



## MOVE THE BUILDING

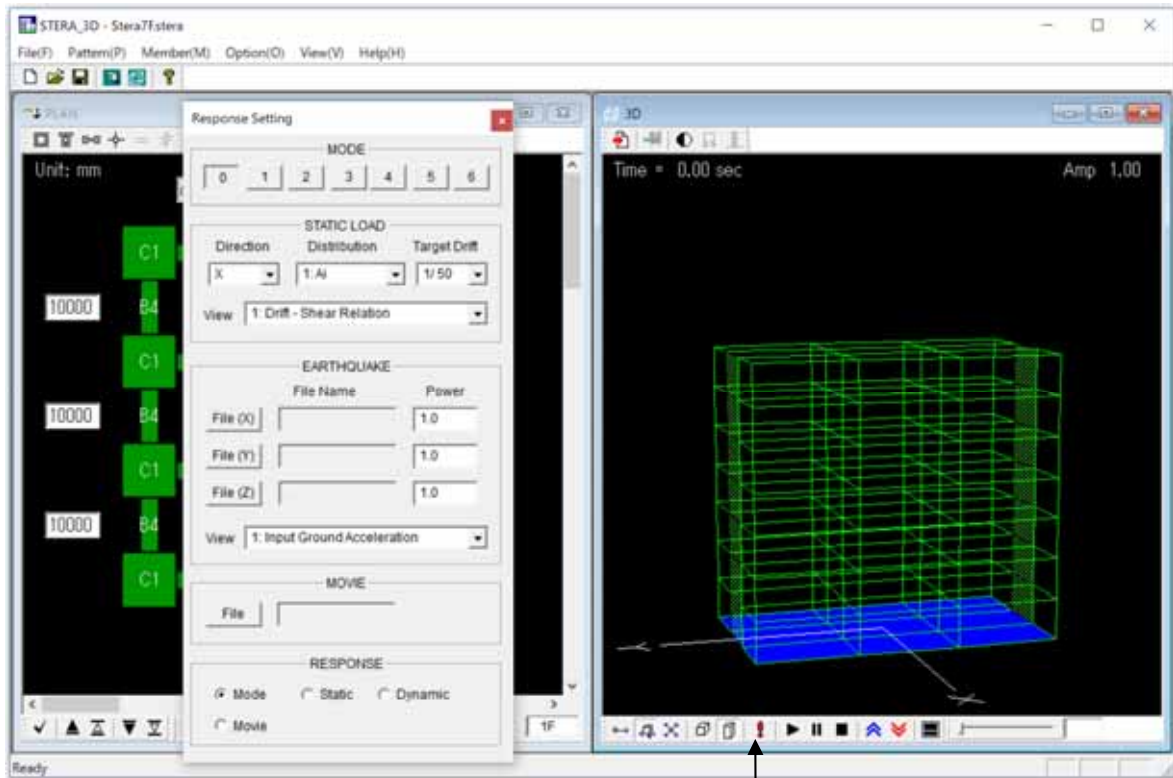


Click  to be actual size.

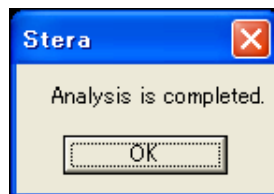
Drag the right mouse on the image to rotate the building.

Drag the left mouse on the image to enlarge and reduce.

# EARTHQUAKE RESPONSE



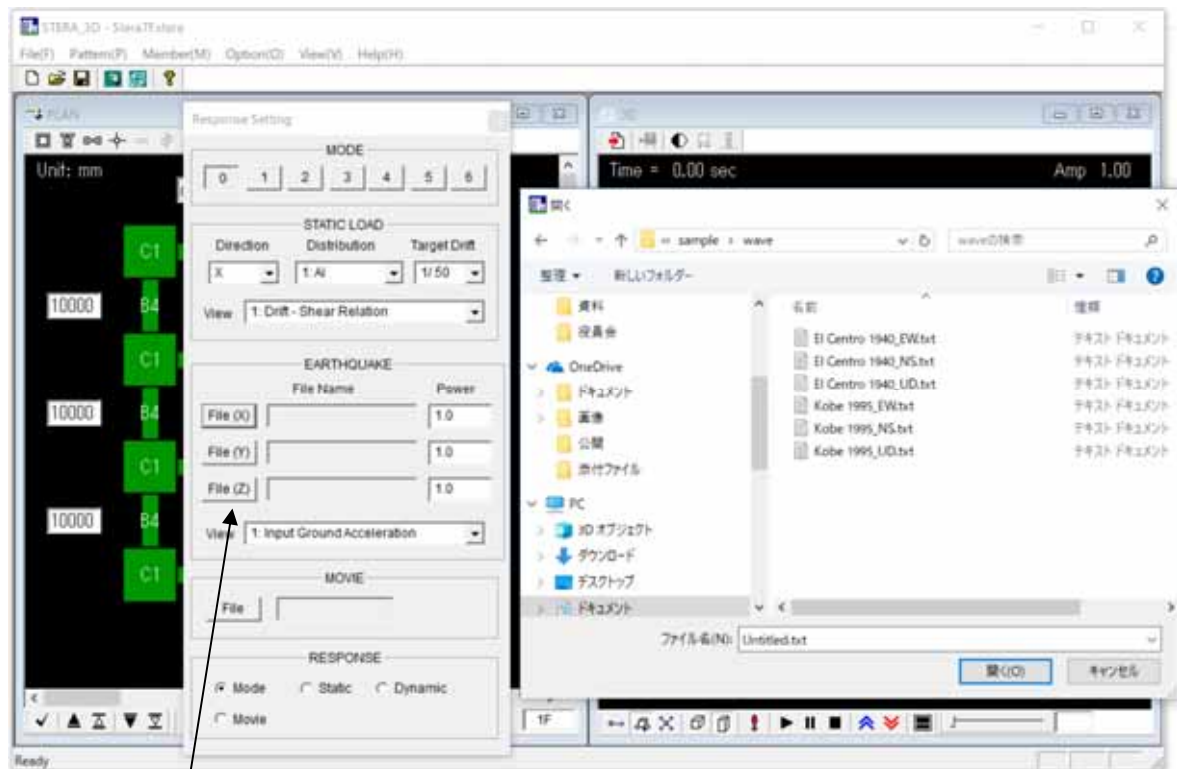
Click  to analyze the building.



After the message,

“Response Setting Dialog” will appear.





File (X)

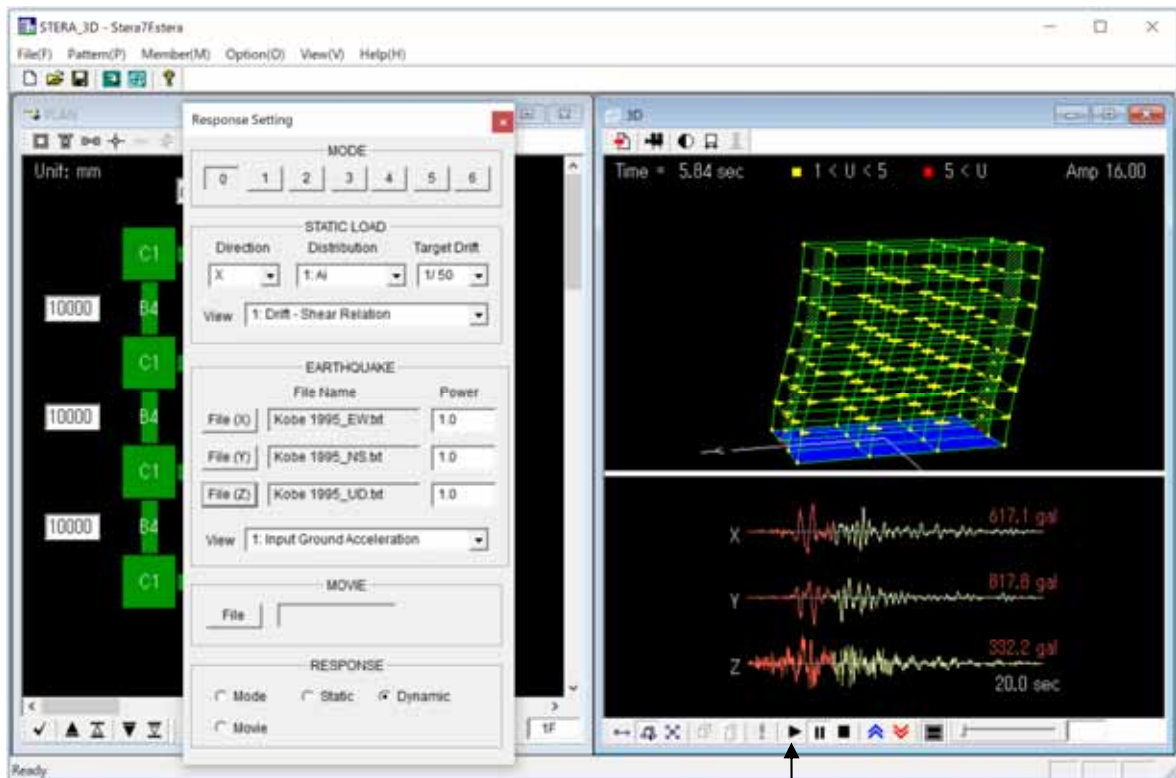
: Select X-direction earthquake data file.  
For example, "Kobe\_1995\_EW".

File (Y)

: Select Y-direction earthquake data file.  
For example, "Kobe\_1995\_NS".

File (Z)

: Select Z-direction earthquake data file.  
For example, "Kobe\_1995\_UD".



: Start the response



: Stop the response



: Amplify the response



: Reduce the response



:: Change the view from double screen to single screen

# User Manual

## Index

1	Basic Assumptions .....	14
2	File Arrangement .....	16
3	Initial View .....	17
4	Setting Member Pattern .....	18
5	Initial Setting of Building and Element .....	20
5.1	Element Menu .....	20
5.2	Activate Element .....	21
5.3	Change the number of stories, spans.....	25
6	Input Element Information .....	26
6.1	RC Column.....	26
6.2	RC Beam .....	29
6.3	RC Wall .....	31
6.4	Steel Column .....	33
6.5	Steel Beam .....	34
6.6	Steel Wall (Brace) .....	35
6.7	SRC (Steel Reinforced Concrete) Column .....	36
6.8	SRC (Steel Reinforced Concrete) Beam.....	37
6.9	SRC (Steel Reinforced Concrete) Wall.....	38
6.10	Column (Direct input for parameters of hysteresis model) .....	39
6.11	Beam (Direct input for parameters of hysteresis model).....	42
6.12	Wall (Direct input for parameters of hysteresis model).....	46
6.13	Column (Mixed mode) .....	49
6.14	Beam (Mixed mode) .....	49
6.15	Wall (Mixed mode) .....	50
6.16	Floor Slab (2D Rigid) .....	51
6.17	Floor Slab (3D Rigid) .....	51
6.18	Floor Slab (Flexible) .....	52
6.19	Floor Slab (Mix) .....	53
6.20	Connection Panel.....	54
6.21	External Spring .....	55
6.22	Seismic Isolator .....	61
6.23	Passive Damper .....	68
6.24	Masonry Wall.....	73
6.25	Ground Spring (Cone model).....	75
6.26	Ground Spring (Direct).....	77

7	Initial Setting of Analysis Condition.....	78
7.1	Restrained freedom, Rigid floor, P-Delta effect, Mass distribution.....	78
7.2	Condition of static analysis.....	81
7.3	Condition of dynamic analysis.....	84
8	3D View of Building and Response.....	86
8.1	3D View of building.....	86
8.2	Modal analysis.....	88
8.3	Nonlinear static push-over analysis.....	89
8.4	Nonlinear earthquake response analysis.....	92
8.5	Nonlinear vibrator response analysis.....	97
8.6	Wind response analysis.....	98
8.7	Response output member.....	99
8.8	Save Nonlinear Earthquake Response as a Movie File.....	100
8.9	Change Analysis.....	102
9	Input Earthquake Ground Motion.....	103
9.1	Format of input earthquake data file.....	103
10	Save and Open Files.....	104
10.1	Save building data.....	104
10.2	Save results of analysis in text files.....	105
10.3	Output text files.....	107
11	Continuous Analysis.....	124
12	Automatic generation of Lumped Mass Model (LMM).....	125
13	Command line execution.....	129

## 1 Basic Assumptions

- 1) In the default setting, a floor diaphragm is considered to be rigid for in-plane deformation and free for out-of-plane deformation. Elastic deformation of a floor diaphragm for in-plane deformation can be considered by selecting the FEM model in the option menu.
- 2) All structural elements are modeled by line-elements with nonlinear springs except the floor diaphragm which can be by a FEM model.
- 3) Beam element is represented by the model with nonlinear flexural springs at the both ends and a nonlinear shear spring in the middle of the element,
- 4) Column element is represented by the MS (multi spring) model with nonlinear axial springs in the sections of the both ends and two directional nonlinear shear springs in the middle of the element by default,
- 5) Wall element is represented by the MS (multi spring) model with nonlinear axial springs in the sections of the both ends and nonlinear shear springs in the middle of the wall panel as well as in the two side columns by default,
- 6) Steel brace is represented by the truss element,
- 7) Base-isolation element is represented by the MSS (multi shear spring) model with nonlinear shear springs in X-Y plane,
- 8) Energy dissipation dampers and masonry element are introduced as nonlinear shear springs in a frame,
- 9) Shear deformation of connection panel between beam and column is considered by selecting elastic connection in the option menu.
- 10) In the default setting, structural damping is proportional damping to initial stiffness. It can be changed to be other types of damping by the option menu.

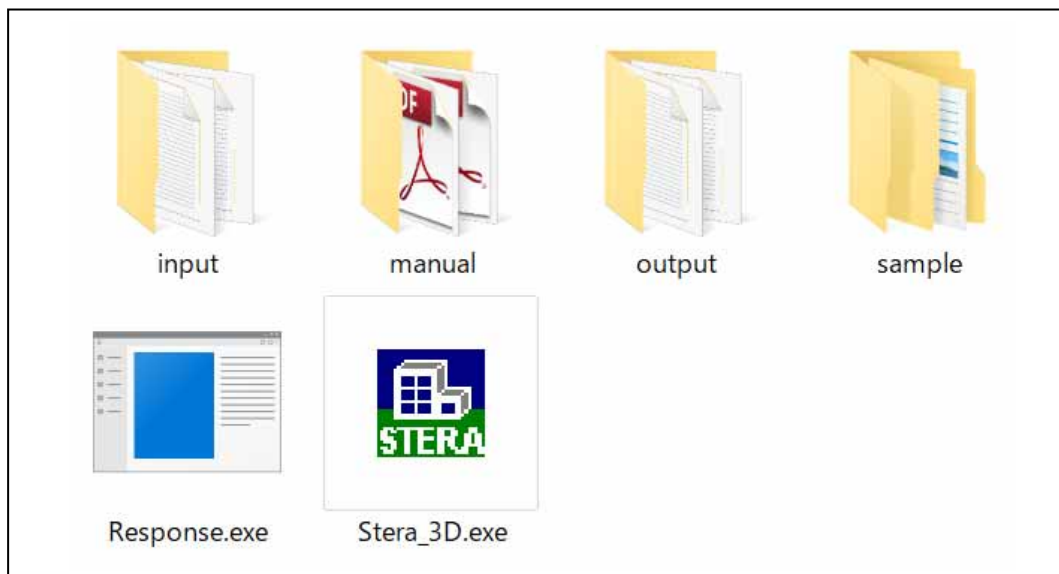
- 11) Rebar sizes follow U.S. and Japanese standards, but can be changed to Euro standards, etc. in the options menu.

Other assumptions and details are written in “Technical Manual”.

## 2 File Arrangement

Please check if you have the following files and folders in the folder “STERA 3D V\*.\*”:

Stera_3D_J.exe	... Main program	} Please keep them in the same folder.
Response.exe	... Sub-program for response output	
input /	... Folder for input (empty)	
output/	... Folder for output (empty)	
manual/	... Folder for manuals	
STERA_user_manual		
STERA_technical_manual		
sample/	... Folder for sample	
building/	... Folder for sample building for STERA	
wave/	... Folder for sample input waves	





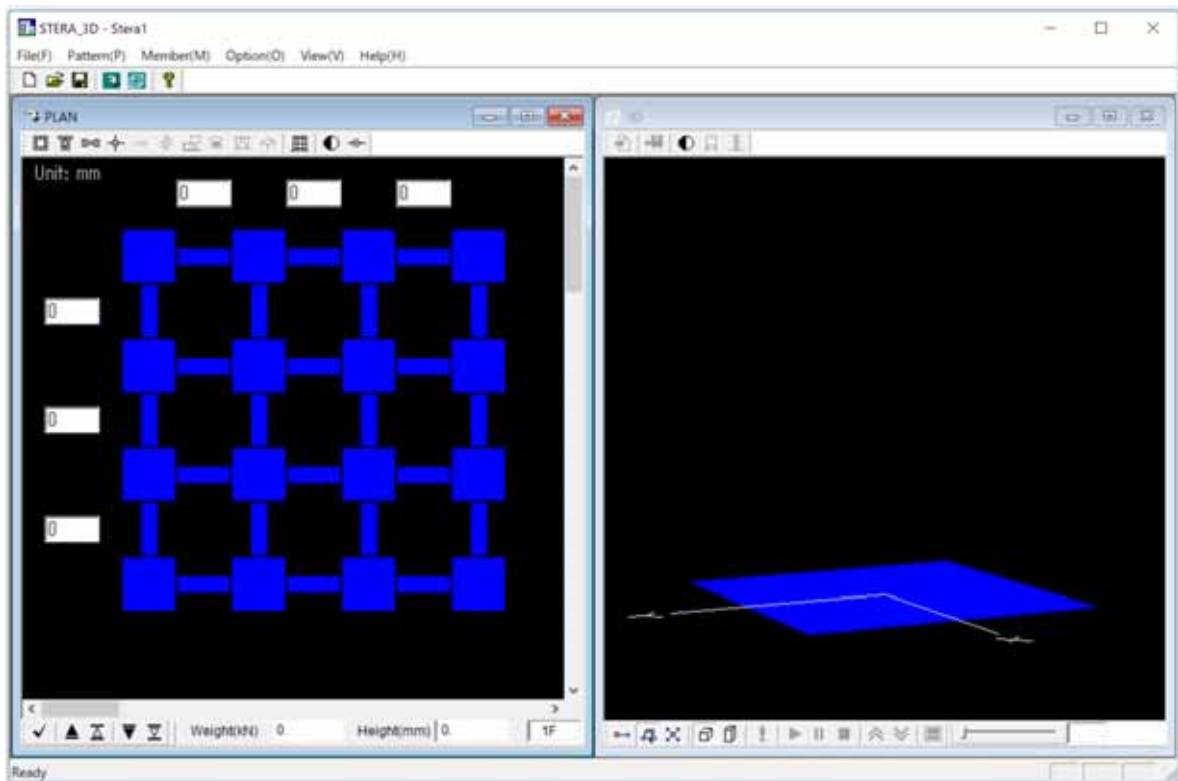
### 3 Initial View

Please double click “Stera 3D.exe”



The left view is “PLAN EDIT VIEW” where you input building plan data, and the right view is “3D VIEW” where you can see the building shape and its response after the analysis.

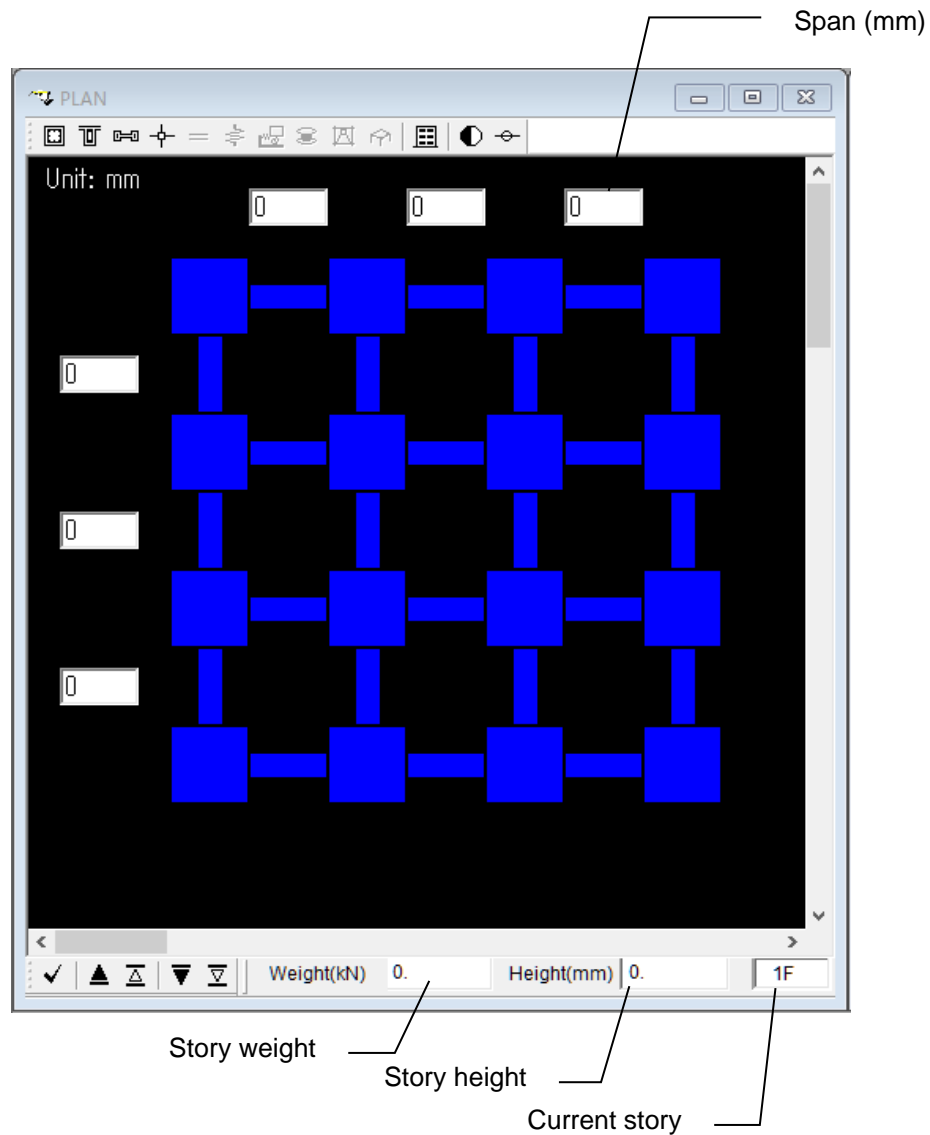
To open the building data already saved, [File]→ [Open], and select the file.



PLAN EDIT VIEW

3D VIEW

## 4 Setting Member Pattern



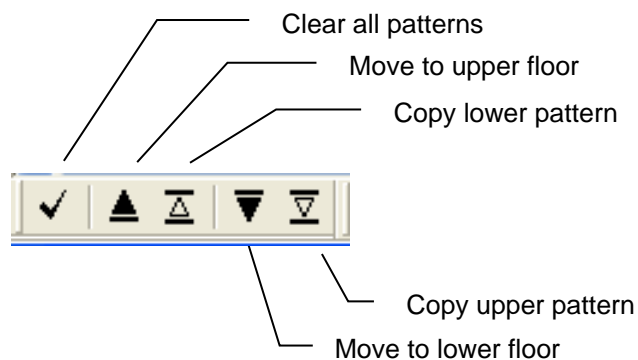
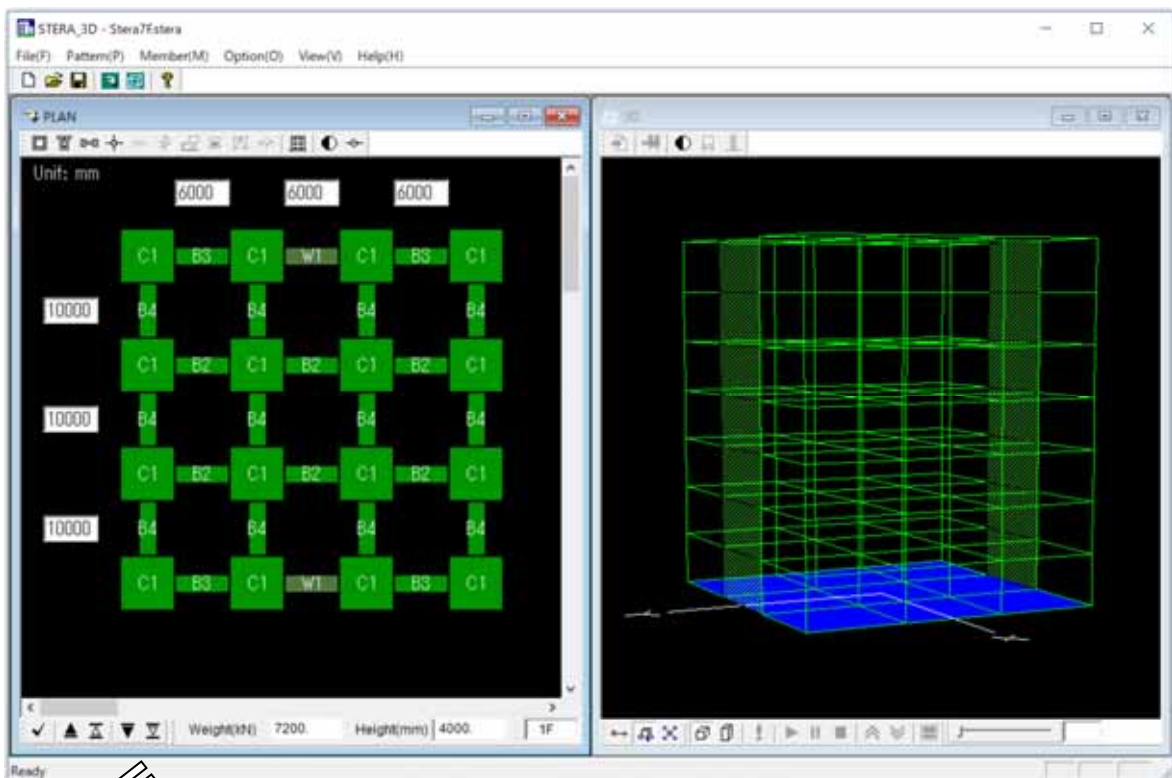
“PLAN EDIT VIEW” starts from 1<sup>st</sup> floor (1F) of a building.

- Please click the place you want to set.
  - Please click again to change the element. It will be changed in the following order:
    - ✧ Column (green) → Empty → Column(green)
    - ✧ Beam (green) → Wall (dark green) → Empty → Beam (green)
- But, in case of the basement floor (BF), the order is changed as:
- ✧ Base Spring (brown) → Empty → Base Spring (brown)

If you select Masonry element, Damper element, Isolator element, External Spring,

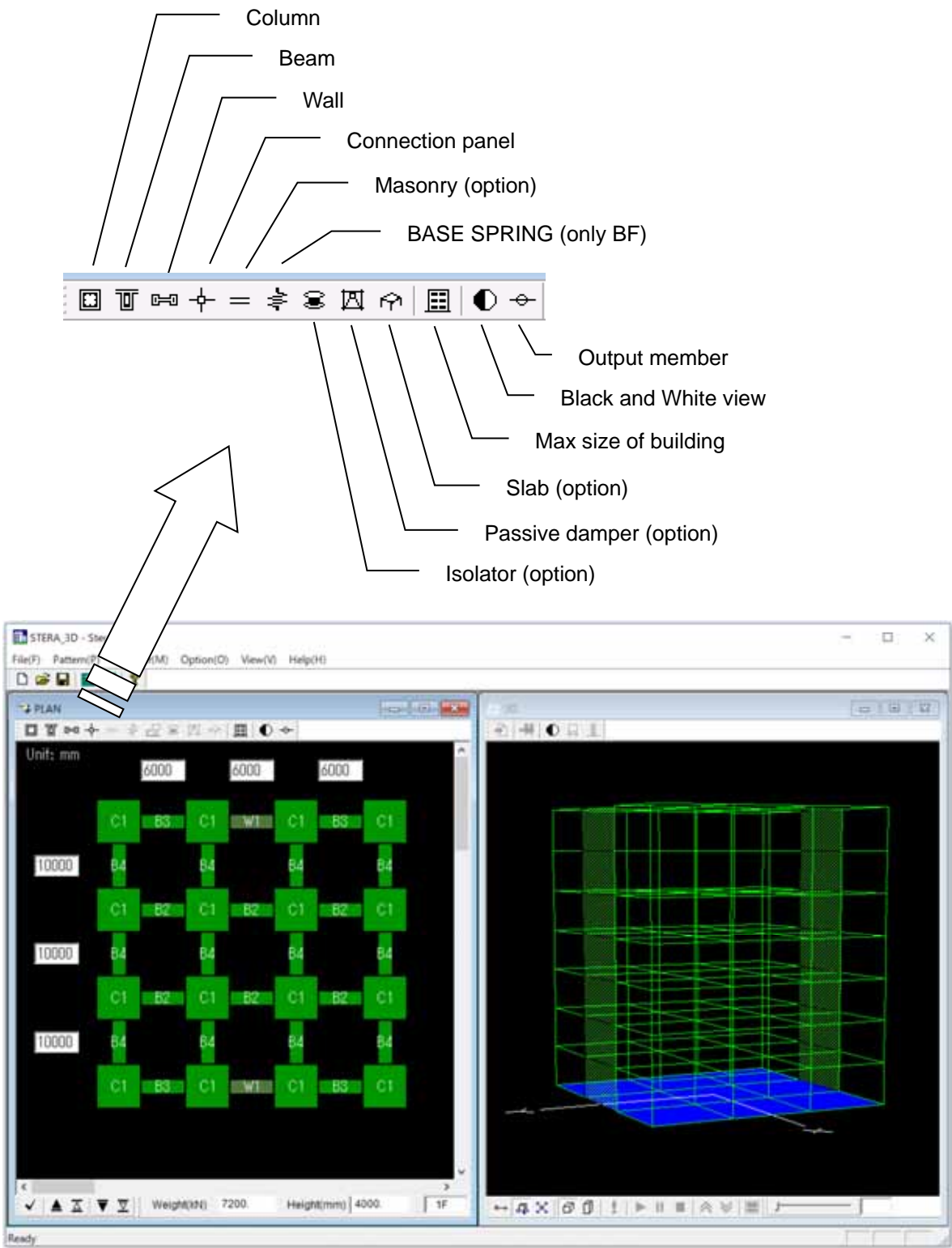
- ✧ Column (green) → Isolator (brown) → Empty → Column(green)
- ✧ Beam (green) → Wall (dark green) → Damper (brown) → Masonry (brown) → External Spring (brown) → Empty → Beam (green)
- ✧ If you click while holding down the control key (Ctrl), you can delete immediately.
- By dragging your mouse in a region, you can set all the elements in the region at once.
- By clicking the right button of your mouse, you can change the number of element type for column (C1-C100), for beam (B1-B100), and for wall (W1-W100) etc.
- To move to another floor and copy or clear the member patterns, you can use the following buttons arranged at the bottom of the PLAN EDIT VIEW:

You can check the arrangement of members on the “3D VIEW”.

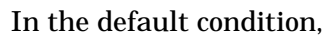


5 Initial Setting of Building and Element

5.1 Element Menu



## 5.2 Activate Element



- To change structural type and activate other elements, please select “Option” in the main menu and select “Member” from the pull down menu.



Option → Member

**[1] Column, [2] Beam, [3] Wall**

RC: Reinforced concrete,

S: Steel,

SRC: Steel Reinforced Concrete,

Direct: Direct input of force and displacement,

Mix: Mixed mode

If you select [RC], [S], [SRC], [Direct], all members will have the same structure. Therefore, the message will appear asking "Clear all member information?".

If you select [Mix], you can use different structure for each member.

#### [4] Floor Slab

2D Rigid: Rigid for in plane deformation and free for out-of-plane deformation

3D Rigid: Rigid for all directional deformation

Flexible: Elastic for in plane deformation and free for out-of-plane deformation

Mix: Different slab condition for each floor

#### [5] Ground Spring

Cone model: Calculate complex ground stiffness by cone model

Direct: Direct input of stiffness and damping of ground spring

#### [6] Isolator

#### [7] Passive Damper Device

#### [8] Masonry Wall

#### [9] External Spring above basement including air spring

#### [10] Nonlinear Shear Spring

If it is not considered, shear springs in beams, columns and walls are elastic.

#### [11] Nonlinear Flexural Spring

If it is not considered, flexural springs in beams, columns and walls are elastic.

#### [12] Young's Modulus of Steel

#### [13] Rebar Size Table

In the default setting, rebar size is based on the standard of Japan and U.S.

Rebar Size Table 1. Japan/U.S. Set

Bar Size Table

Reinforcing Bar Size and Area (mm<sup>2</sup>)

Standard		Original	
D 6(# 2)	31.67	D29(# 9)	642.4
D 8	49.51	D32(#10)	794.2
D10(# 3)	71.33	D35	956.6
D13(# 4)	126.7	D38	1140
D16(# 5)	198.6	D41	1340
D19(# 6)	286.5	D51	2027
D22(# 7)	387.1		
D25(# 8)	506.7		

S 1

S 2

S 3

S 4

S 5

S 6

S 7

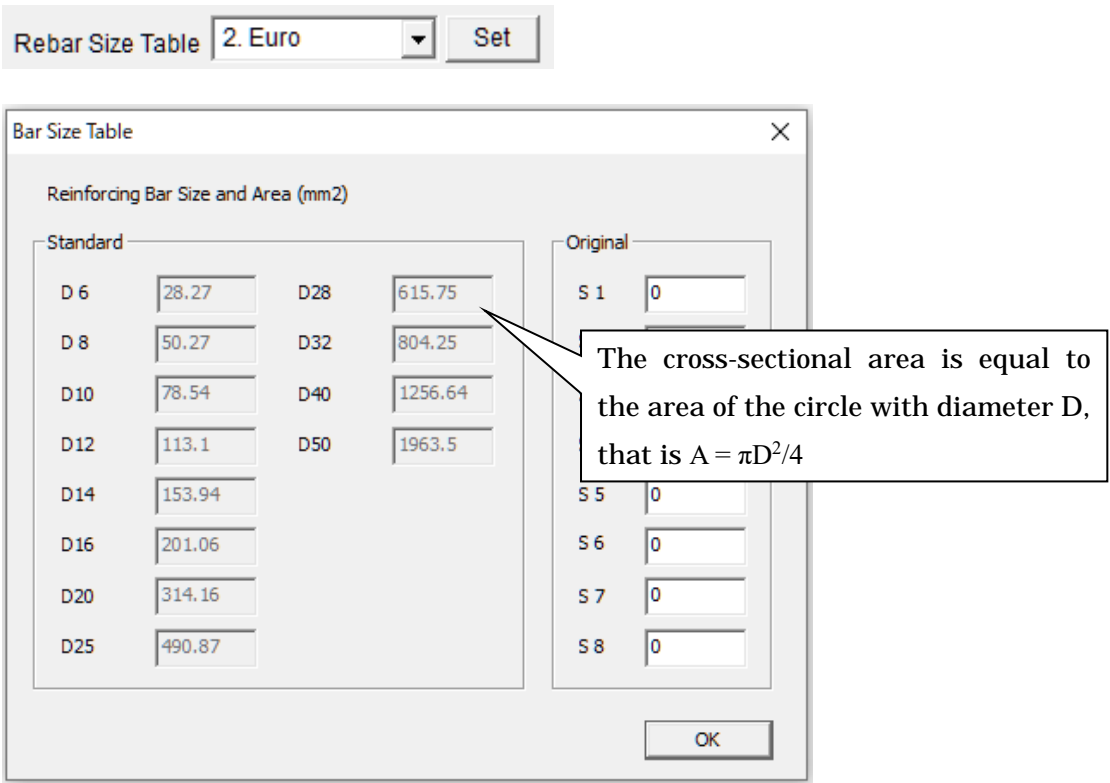
S 8

OK

Original rebar sizes (cross-sectional area) that are not in the standard can be defined by users.

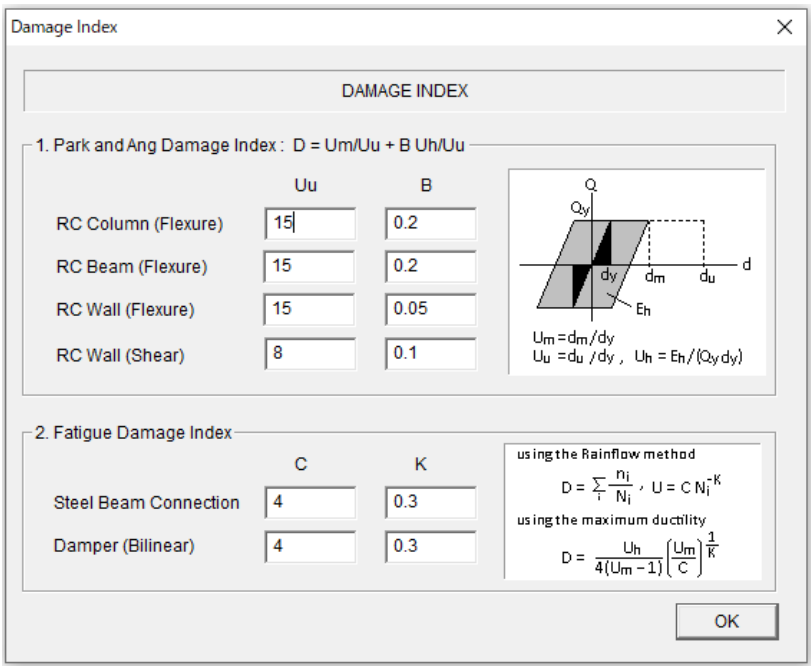
#2 ~ #10 are U.S. standard

You can select Euro (Eurocode) from the pull-down menu.



#### [14] Damage Index

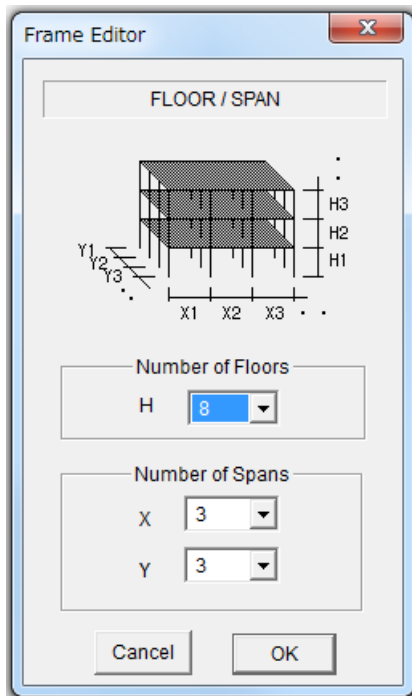
The parameters for the two types of damage index, Park and Ang damage index and Fatigue damage index, are given as follow by default. Please refer “Technical Manual” for the detail of each damage index.





### 5.3 Change the number of stories, spans

MAX. SIZE OF BUILDING (  )

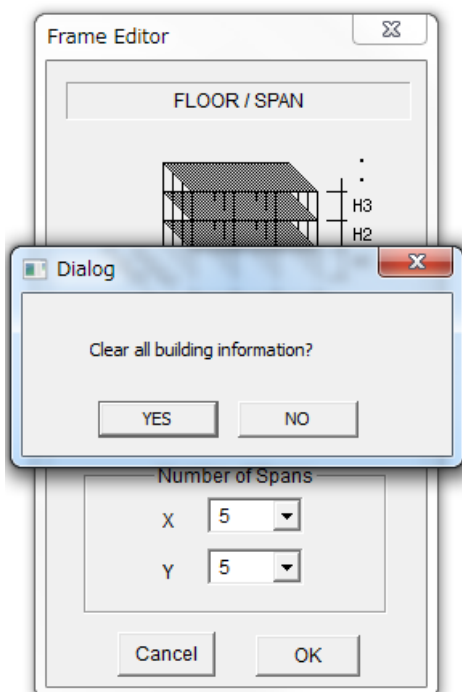


You can change the maximum number of spans and stories of the building. The default setting is

Story : up to 8  
Span : up to 3 in X-direction  
up to 3 in X-direction

The maximum numbers you can select are:

Story : up to 61  
Span : up to 30 in X-direction  
up to 20 in X-direction

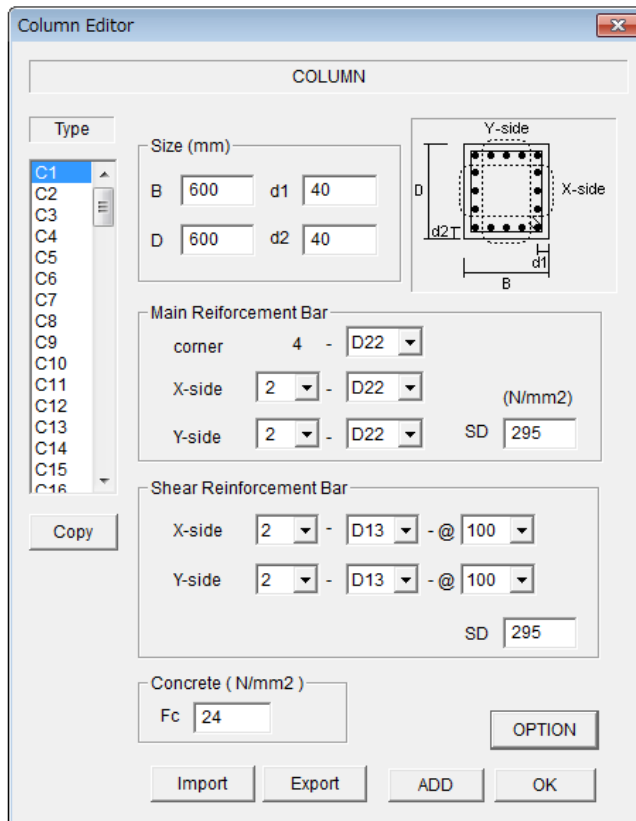


After you select new numbers, a dialog will appear asking if you clear all building information or not. If you select “NO”, you can keep the same building information.

## 6 Input Element Information

### 6.1 RC Column

COLUMN ()



Column Editor

COLUMN

Type

C1  
C2  
C3  
C4  
C5  
C6  
C7  
C8  
C9  
C10  
C11  
C12  
C13  
C14  
C15  
C16

Copy

Size (mm)

B 600 d1 40

D 600 d2 40

Y-side

X-side

Main Reinforcement Bar

corner 4 - D22

X-side 2 - D22 (N/mm2)

Y-side 2 - D22 SD 295

Shear Reinforcement Bar

X-side 2 - D13 - @ 100

Y-side 2 - D13 - @ 100 SD 295

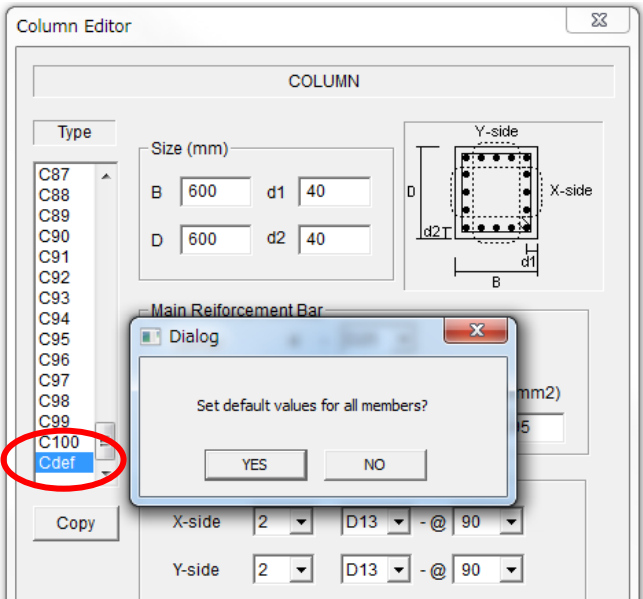
Concrete (N/mm2)

Fc 24

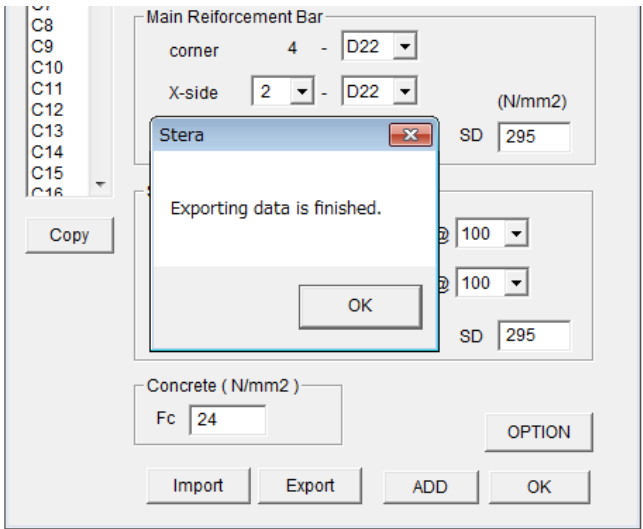
OPTION

Import Export ADD OK

- Please input the section size where d1 and d2 are the distances of X-rebars and Y-rebars respectively. If rebars are arranged in two layers, the distance is determined as the center of rebar area.
- For the number of reinforcing bars and their size, please select the values from the popup windows.
- For steel strength SD and concrete compressive strength Fc, you can input values by changing the default values.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.



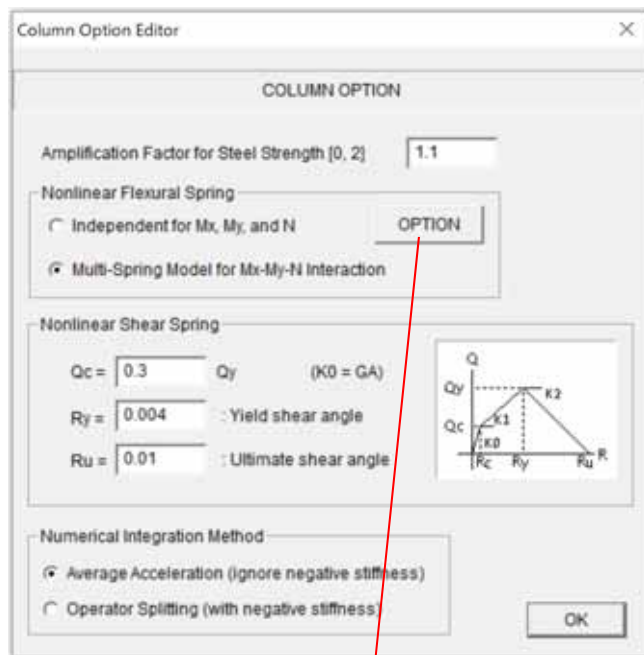
- You can set the default values for all members by selecting the last member type “Cdef”.



- You can export member data to a text file by [Export] button. Automatically, the text file, “Data\_column\_rc.txt” is created in the same folder of STERA\_3D.
- You can import member data from a text file by [Import] button. The format of the import file must be the same as the exported file.

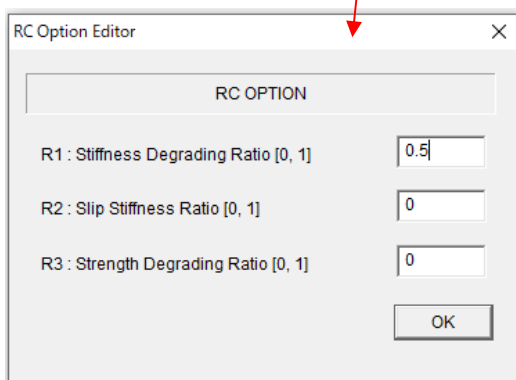
“Data\_column\_rc.txt” is a text file with member data separated by TAB.

n	Width(mm)	Height(mm)	d1	d2	vsize_C	vno_X
1	600	600	40	40	9	1
2	600	600	40	40	9	1
3	600	600	40	40	9	1
4	600	600	40	40	9	1
5	600	600	40	40	9	1
6	600	600	40	40	9	1
7	600	600	40	40	9	1
8	600	600	40	40	9	1
9	600	600	40	40	9	1
10	600	600	40	40	9	1
11	600	600	40	40	9	1
12	600	600	40	40	9	1



In [OPTION] menu,

- The default steel strength is assumed to be 1.1 times larger than the nominal strength.
- For nonlinear flexural spring, you can select from 1) the model with independent springs for Mx, My and N or 2) MS (multi-spring) model for Mx-My-N nonlinear interaction. The default setting is MS model.
- Shear crack strength  $Q_c$  is defined as the ratio of the yield strength  $Q_y$ . The default value is 0.3.
- The default values of yield and ultimate shear deformation angles,  $R_y$  and  $R_u$  are 0.004 (=1/250) and 0.01 (1/100), respectively.
- You can identify the numerical integration method in earthquake response analysis from Average Acceleration method or Operator Splitting method. The default is Average Acceleration method which replace the negative stiffness to be a small positive stiffness for the stability of calculation.



In [OPTION] menu, the parameters to control the shape of hysteresis model are defined as follows:

- R1: the default value of stiffness degrading ratio in the trilinear hysteresis is 0.5. (0: no degradation)
- R2: the default value of slip stiffness ratio in the trilinear hysteresis is 0.0 (0: no slip).
- R3: the default value of strength degrading ratio in the trilinear hysteresis is 0.0.

Please refer "Technical Manual" for the detail.

## 6.2 RC Beam

BEAM (  )

- Please input the section size where d1 and d2 are the distances of upper and bottom rebars. If rebars are arranged in two layers, the distance should be the center of rebar area.
- For the number of reinforcing bars and their size, please select the values from the popup windows.
- For the material strength, SD and Fc, you can input values by changing the default values.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Bdef".
- You can export member data to the text file "Data\_beam\_rc.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

BEAM OPTION	
1. Amplification Factor for Steel Strength [0,2]	1.1
2. Rs : Effective Slab Ratio [0,0.5]	0.1
3. R1 : Stiffness Degrading Ratio [0,1]	0.5
4. R2 : Slip Stiffness Ratio [0,1]	0
5. R3 : Strength Degrading Ratio [0,1]	0
6. Ru : Ultimate Rotation Angle [0,1]	0.02
7. Kp/Ky : Stiffness Ratio over Ry [0, 1]	0.001
8. Ku/Ky : Stiffness Ratio over Ru [-1, 1/1000]	0.001

OK

In [OPTION] menu,

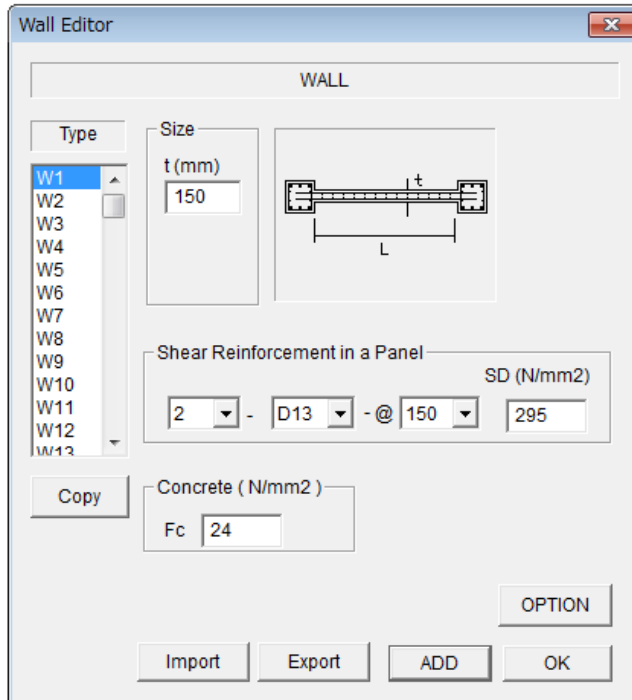
- The default steel strength is assumed to be 1.1 times larger than the nominal strength.
- Rs: the effective slab width to contribute flexural behavior of beam is assumed to be 0.1 times beam length.

The parameters to control the shape of hysteresis model are defined as follows:

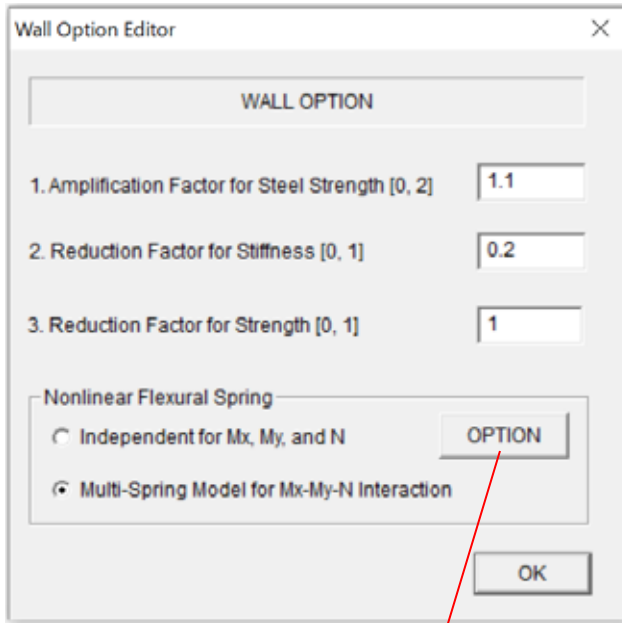
- R1: the default value of stiffness degrading ratio in the trilinear hysteresis is 0.5. (0: no degradation)
- R2: the default value of slip stiffness ratio in the trilinear hysteresis is 0.0 (0: no slip).
- R3: the default value of strength degrading ratio in the trilinear hysteresis is 0.0.
- Ru: the default value of Ultimate rotation angle Ru is 1/50 (=0.02)
- Kp/Ky: the default value of stiffness ratio over Ry is 0.001
- Ku/Ky: the default value of stiffness ratio over Ru is 0.001 (could be negative)

### 6.3 RC Wall

WALL ()

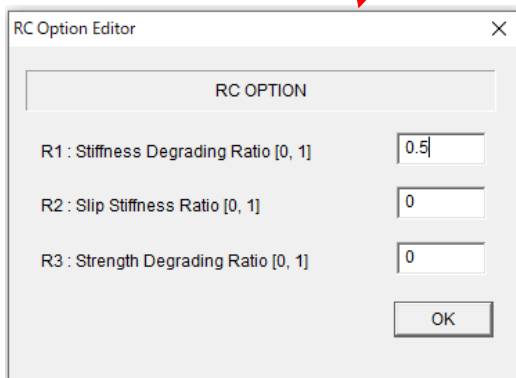


- Please input the section size.
- For the number of reinforcing bars and their size, please select the values from the popup windows.
- For the material strength, SD and Fc, you can input values by changing the default values.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Wdef".
- You can export member data to the text file "Data\_wall\_rc.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.



In [OPTION] menu,

- The default steel strength is assumed to be 1.1 times larger than the nominal strength. You can change the ratio in [OPTION] menu.
- Considering the early cracks in reinforced concrete walls, you can reduce the shear stiffness by multiplying a reduction factor. The default value is 0.2.
- Considering openings in a wall element, you can reduce the shear strength by multiplying a reduction factor. The default value is 1.0.
- For nonlinear flexural spring, you can select from 1) the model with independent springs for Mx, My, and N or 2) MS (multi-spring) model for Mx-My-N nonlinear interaction. The default setting is MS model.



In [OPTION] menu, the parameters to control the shape of hysteresis model are defined as follows:

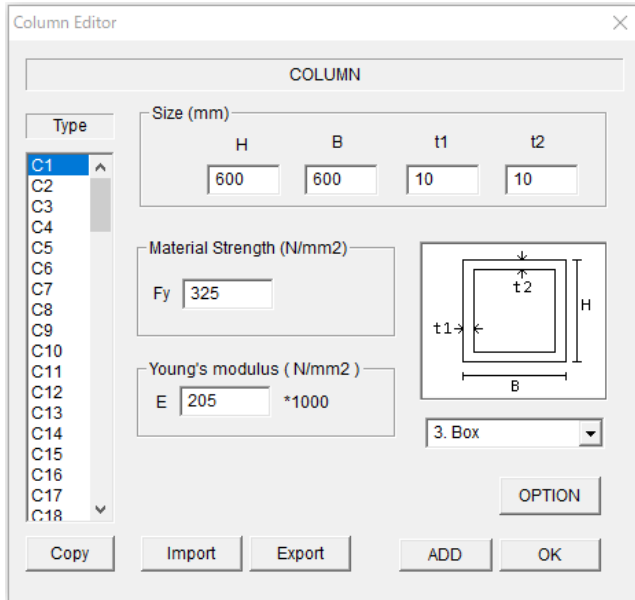
- R1: the default value of stiffness degrading ratio in the trilinear hysteresis is 0.5. (0: no degradation)
- R2: the default value of slip stiffness ratio in the trilinear hysteresis is 0.0 (0: no slip).
- R3: the default value of strength degrading ratio in the trilinear hysteresis is 0.0.

Please refer "Technical Manual" for the detail.

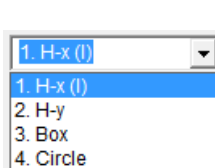
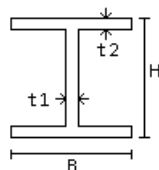


## 6.4 Steel Column

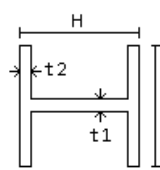
COLUMN 



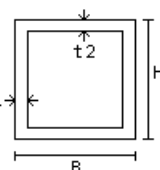
- Please select the shape of section from the pull-down menu.
- Please input the section size.
- For the material strength,  $F_y$ , and Young's modulus,  $E$ , you can change the default values.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Cdef".
- You can export member data to the text file "Data\_column\_steel.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

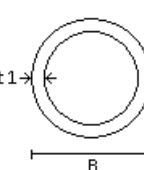
H-x(I)



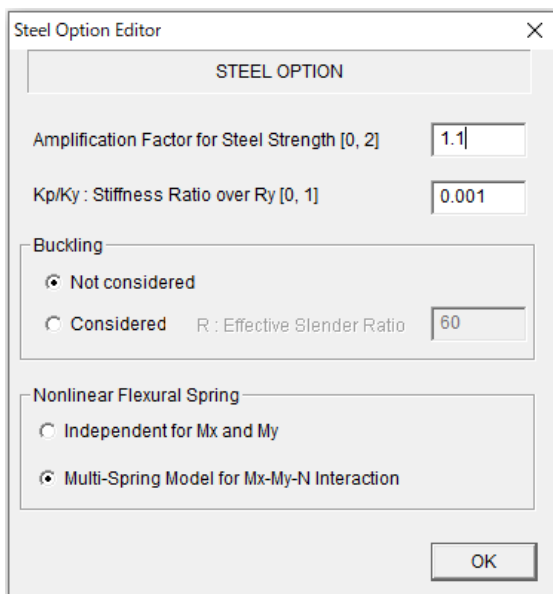
H-y



Box



Circle

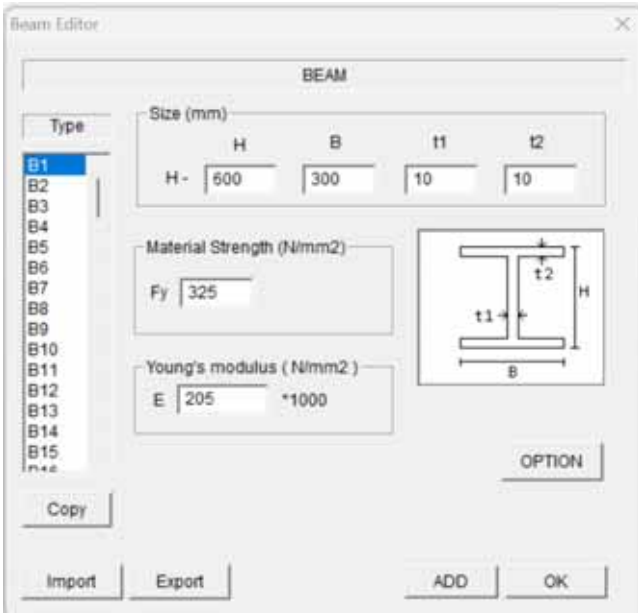


In [OPTION] menu,

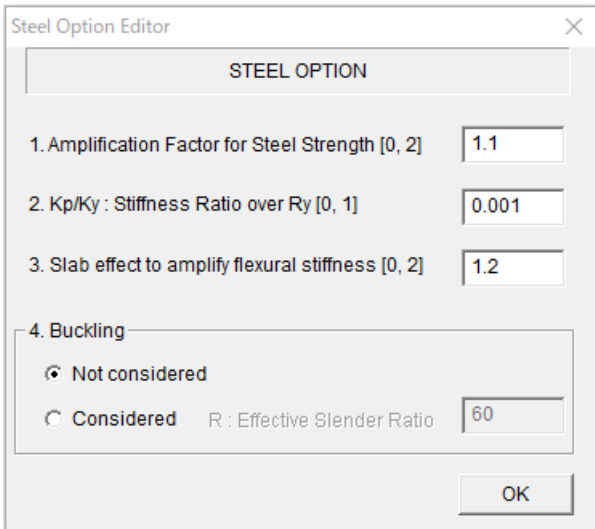
- The default steel strength is assumed to be 1.1 times larger than the nominal strength and the stiffness ratio after yielding is 0.001.
- Non-linear hysteresis due to buckling can be considered. The initial setting is "Not considered". To consider the buckling, you must enter an effective slender ratio.
- For nonlinear flexural spring, you can select from 1) the model with independent springs for  $M_x$  and  $M_y$  or 2) MS (multi-spring) model for  $M_x$ - $M_y$ - $N$  nonlinear interaction. The default setting is MS model.

## 6.5 Steel Beam

BEAM (  )



- Please input the section size.
- For the material strength,  $F_y$ , and Young' s modulus,  $E$ , you can change the default values.
- In [OPTION], you can enter the material properties of the steel beam, slab effects, and buckling considerations.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Bdef".
- You can export member data to the text file "Data\_beam\_steel.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

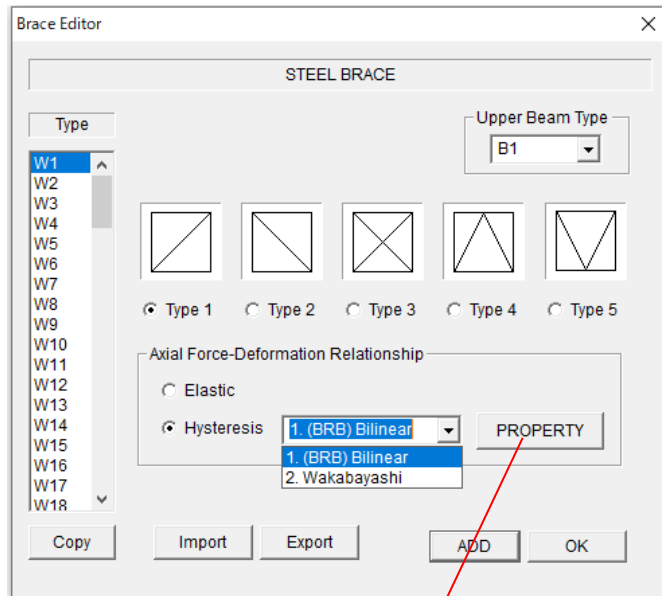


In [OPTION] menu,

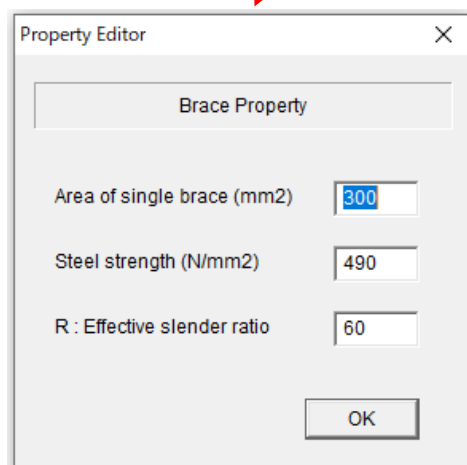
- The default steel strength is assumed to be 1.1 times larger than the nominal strength and the stiffness ratio after yielding is 0.001.
- The amplification factor for flexural stiffness because of slab effect is 1.2 for the default value. If the slab is attached both sides of a beam, the value becomes square.
- Non-linear hysteresis due to buckling can be considered. The initial setting is "Not considered". To consider the buckling, you must enter an effective slender ratio.

## 6.6 Steel Wall (Brace)

WALL ()



\*BRB: buckling restrained brace



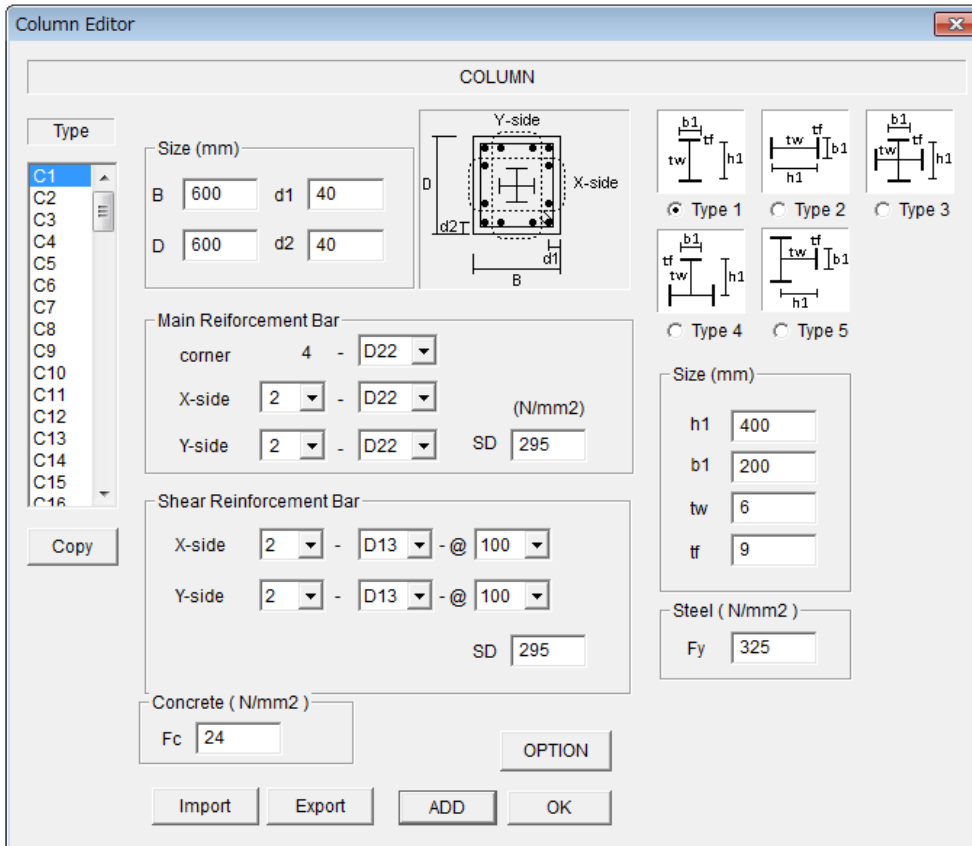
- If there is a beam upper of Brace element, please select the beam type number from the pop-up menu.
- Please select brace types from the pictures (Type1, 2, 3, 4, 5).
- You can select Axial force deformation relationship from "Elastic" and "Hysteresis". Non-linear hysteresis due to buckling can be selected from "1.(BRB) Bilinear" and "Wakabayashi".
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Wdef".
- You can export member data to the text file "Data\_brace.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

In [PROPERTY] menu,

- Please input "Area of single brace", "Steel strength", and "Effective slender ratio".

## 6.7 SRC (Steel Reinforced Concrete) Column

COLUMN (  )



**Column Editor**

**COLUMN**

Type: C1, C2, C3, C4, C5, C6, C7, C8, C9, C10, C11, C12, C13, C14, C15, C16

Size (mm): B 600, d1 40, D 600, d2 40

Main Reinforcement Bar: corner 4 - D22, X-side 2 - D22, Y-side 2 - D22 (N/mm2) SD 295

Shear Reinforcement Bar: X-side 2 - D13 - @ 100, Y-side 2 - D13 - @ 100 (N/mm2) SD 295

Concrete (N/mm2): Fc 24

Size (mm): h1 400, b1 200, tw 6, tf 9

Steel (N/mm2): Fy 325

OPTION

Import Export ADD OK

- RC part is the same as RC Column.
- [OPTION] menu is the same as RC Column.
- Please input Steel size (h1, b1, tw, tf).
- Please input Steel strength (Fy).
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Cdef".
- You can export member data to the text file "Data\_column\_src.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

## 6.8 SRC (Steel Reinforced Concrete) Beam

BEAM ()

**Beam Editor**

**BEAM**

Type: B1, B2, B3, B4, B5, B6, B7, B8, B9, B10, B11, B12, B13, B14, B15, B16, B17, R18

Size (mm): B 300, D 600, S 150, d1 40, d2 40

Main Reinforcement Bar: TOP 2 - D22 (N/mm2), BOTTOM 2 - D22 SD 295

Shear Reinforcement Bar: 2 - D13 - @ 150 SD 295

Slab Reinforcement: 1 - D13 - @ 200 SD 295

Concrete (N/mm2): Fc 24

Steel (N/mm2): Fy 325

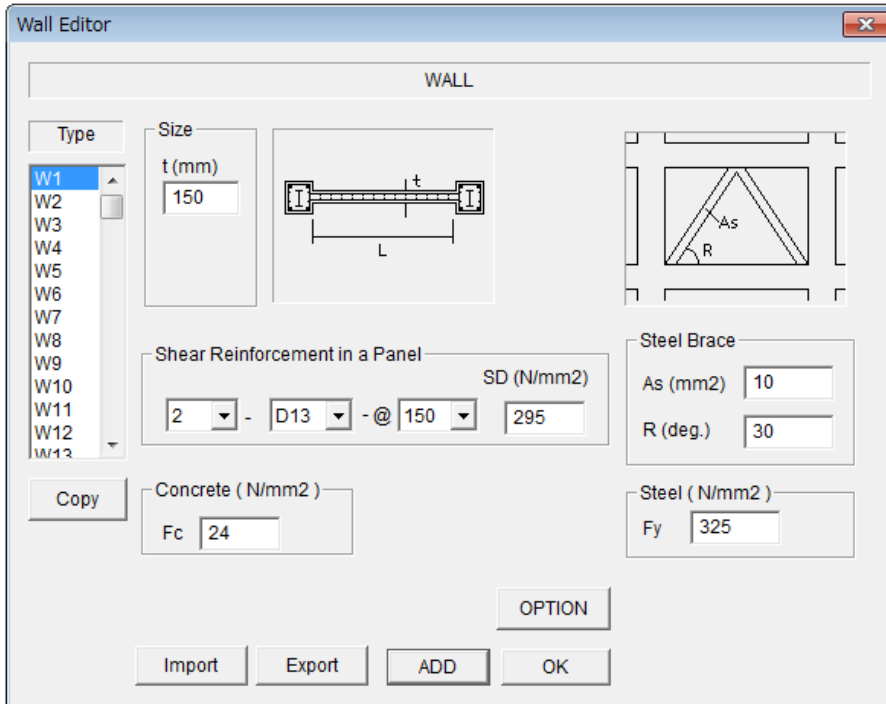
OPTION

Import Export ADD OK

- RC part is the same as RC Beam.
- [OPTION] menu is the same as RC Beam.
- Please input Steel size (h1, b1, tw, tf).
- Please input Steel strength (Fy).
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Bdef"
- You can export member data to the text file "Data\_beam\_src.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

## 6.9 SRC (Steel Reinforced Concrete) Wall

WALL ()



Wall Editor

WALL

Type

- W1
- W2
- W3
- W4
- W5
- W6
- W7
- W8
- W9
- W10
- W11
- W12
- W13

Size

t (mm)

150

Diagram: Wall section with thickness  $t$  and length  $L$ .

Shear Reinforcement in a Panel

SD (N/mm2)

2 - D13 - @ 150 295

Steel Brace

As (mm2) 10

R (deg.) 30

Concrete (N/mm2)

Fc 24

Steel (N/mm2)

Fy 325

Copy

OPTION

Import Export ADD OK

- RC part is the same as RC Wall.
- [OPTION] menu is the same as RC Wall.
- Please input area ( $A_s$ ) and angle ( $R$ ) of the steel brace.
- Please input Steel strength ( $F_y$ ).
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Wdef".
- You can export member data to the text file "Data\_wall\_src.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

## 6.10 Column (Direct input for parameters of hysteresis model)

COLUMN ()

Column Editor

**COLUMN**

Type: C1, C2, C3, C4, C5, C6, C7, C8, C9, C10, C11, C12, C13, C14, C15, C16, C17, C18

Section (m)

A (area): 0 m<sup>2</sup>

I<sub>y</sub> (moment of inertia): 0 m<sup>4</sup>

I<sub>x</sub> (moment of inertia): 0 m<sup>4</sup>

Material

E (Young's modulus): 22000 N/mm<sup>2</sup>

Poisson ratio: 0.16666

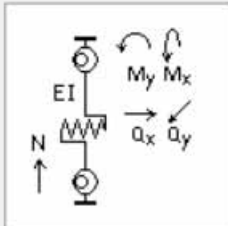
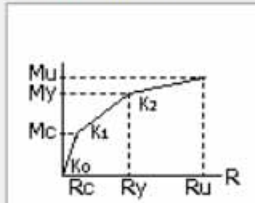
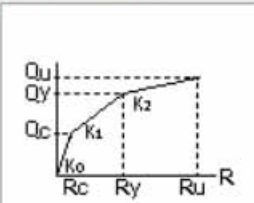
M-R relationship

K<sub>0</sub> = 6EI / L

Q-R relationship

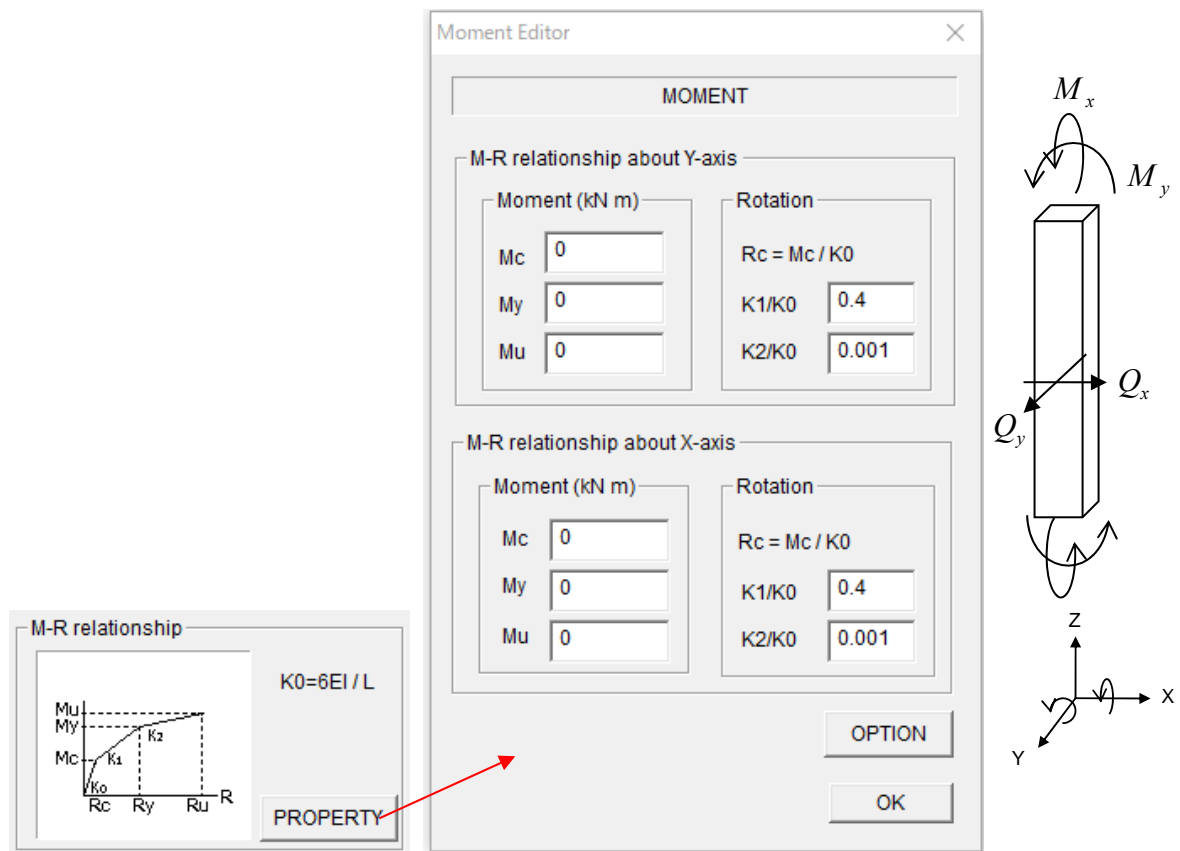
K<sub>0</sub> = GA

Copy, Import, Export, ADD, OK

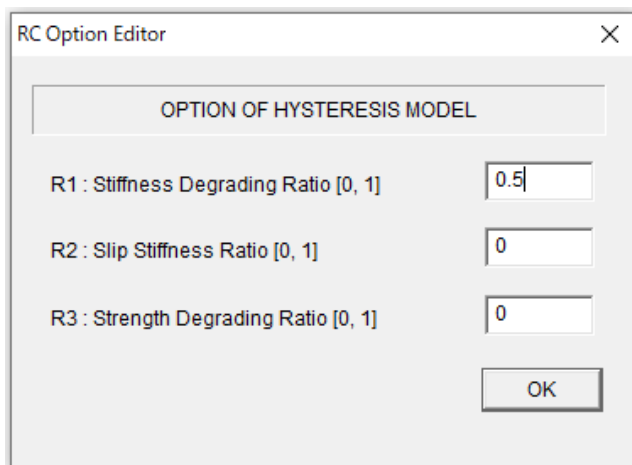




- Please input the area of section (A), the moment of inertia around x and y directions (I<sub>y</sub> and I<sub>x</sub>), Young's modulus (E), and Poisson's ratio (ν) of material that is used for
  - Axial stiffness of the section EA
  - Initial flexural stiffness  $K_0 = 6EI/L$  (L is the member length)
  - Initial shear stiffness  $K_0 = GA/L$  (G is the shear modulus =  $0.5E/(1+\nu)$ )
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type "Cdef".
- You can export member data to the text file "Data\_column\_direct.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

- Please input the parameters of M-R (moment-rotation) relationship by [PROPERTY] bottom.

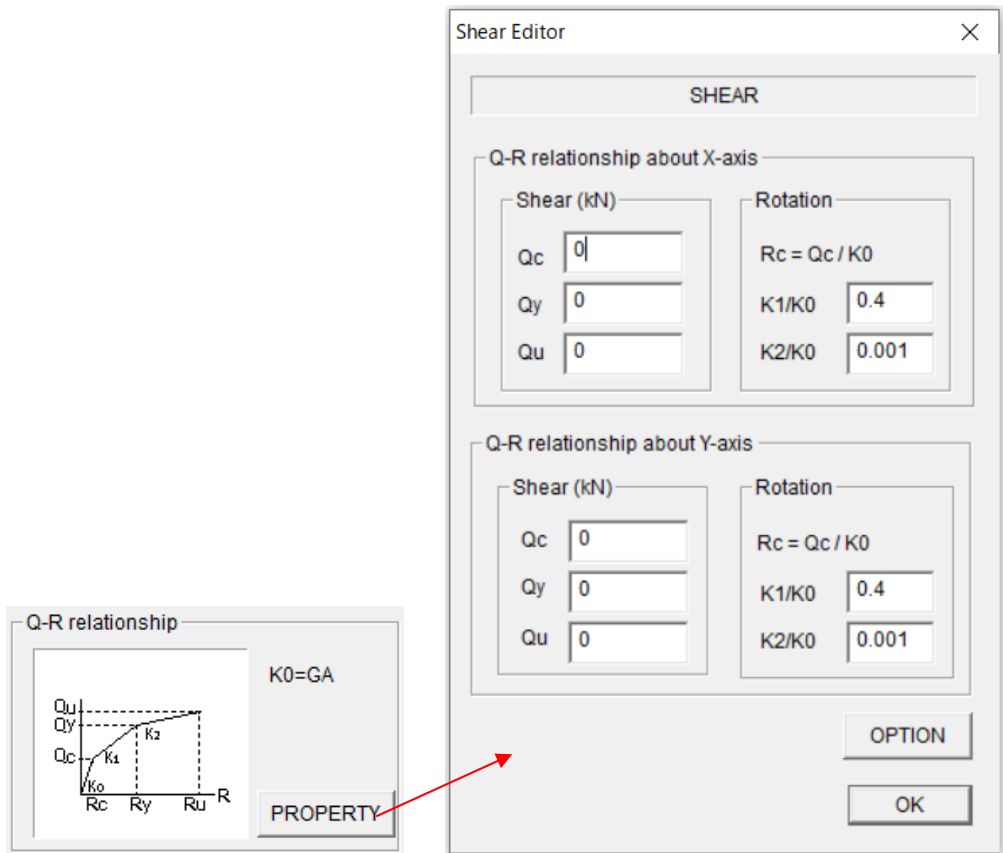


- Please input parameters for moment ( $M_y$ )-rotation relationship about Y axis.
- Please input parameters for moment ( $M_x$ )-rotation relationship about X axis.
- [OPTION] button is activated for RC structural type. From [OPTION] dialog, you can set the parameters for hysteresis model as follows:

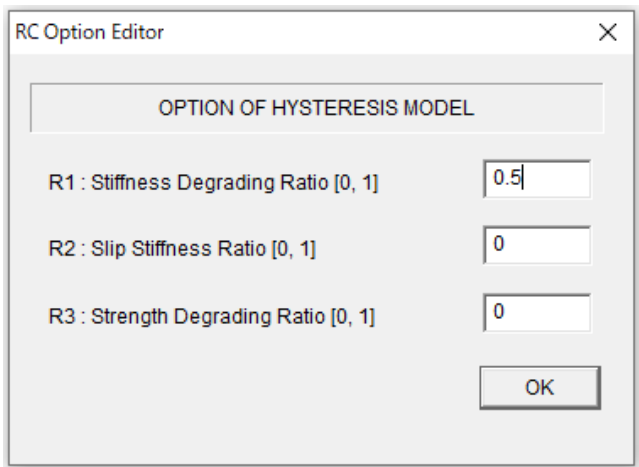




- Please input the parameters of Q-R (shear-rotation) relationship by [PROPERTY] bottom.

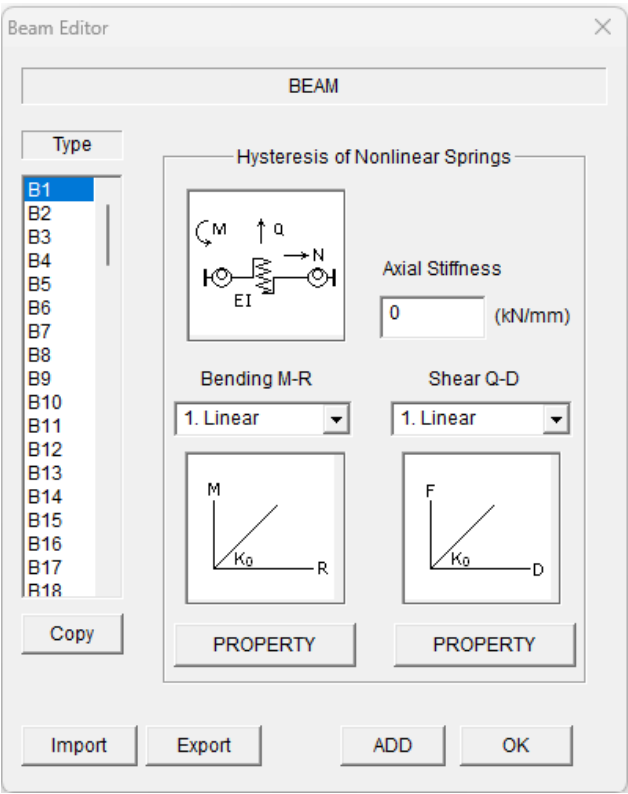


- Please input parameters for shear ( $Q_x$ )-rotation relationship in X axis.
- Please input parameters for shear ( $Q_y$ )-rotation relationship in Y axis.
- [OPTION] button is activated for RC structural type. From [OPTION] dialog, you can set the parameters for hysteresis model as follows:



6.11 Beam (Direct input for parameters of hysteresis model)

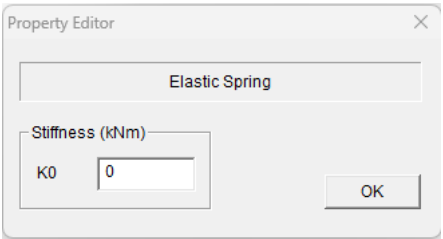
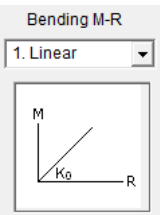
BEAM (  )



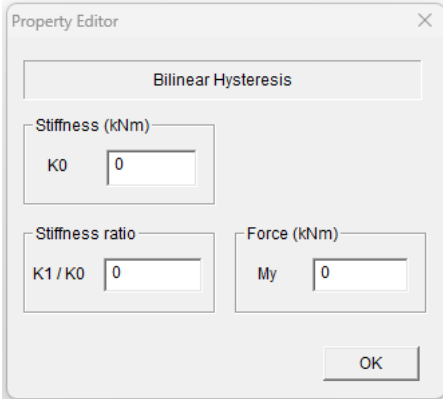
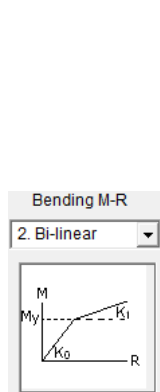
- It consists of bending springs, shear spring, and axial spring.
- Please input the axial stiffness directly.
- Please select hysteresis types for shear and bending springs from the pull-down menu.
- You can input the parameters of the hysteresis in [PROPERTY] view.
- You can set default values for all members by selecting the last member type “Bdef”.
- You can export member data to the text file “Data\_beam\_direct.txt” by [Export] button.
- You can import member data from a text file by [Import] button.

[1] Bending M-R

Linear



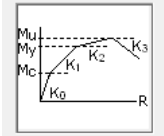
Bi-linear



D-Trilinear

Bending M-R

3. D-Trilinear



Property Editor

Degrading Trilinear Hysteresis

Stiffness (kNm)

K0 0

Stiffness ratio

K1 / K0 0

K2 / K0 0

K3 / K0 0

Force (kNm)

Mc 0

My 0

Mu 0

Hysteresis control parameters

Stiffness Degrading Ratio [0, 1] 0.5

Slip Stiffness Ratio [0, 1] 0

Strength Degrading Ratio [0, 1] 0

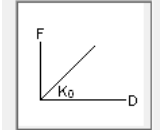
OK

[2] Shear Q-D

Linear

Shear Q-D

1. Linear



Property Editor

Elastic Spring

Stiffness (kN/mm)

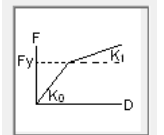
K0 0

OK

Bi-linear

Shear Q-D

2. Bi-linear



Property Editor

Bilinear Hysteresis

Stiffness (kN/mm)

K0 0

Stiffness ratio

K1 / K0 0

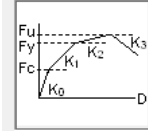
Force (kN)

Fy 0

OK

D-Trilinear

Shear Q-D  
3. D-Trilinear



Property Editor

Degrading Trilinear Hysteresis

Stiffness (kN/mm)  
K0 0

Stiffness ratio  
K1 / K0 0  
K2 / K0 0  
K3 / K0 0

Force (kN)  
Fc 0  
Fy 0  
Fu 0

Hysteresis control parameters  
Stiffness Degrading Ratio [0, 1] 0.5  
Slip Stiffness Ratio [0, 1] 0  
Strength Degrading Ratio [0, 1] 0

OK

Viscoelastic (Voigt model)

Shear Q-D  
4. VE (Voigt)



Property Editor

Viscoelastic Damper

Stiffness (kN/mm)  
K1 0  
K2 0

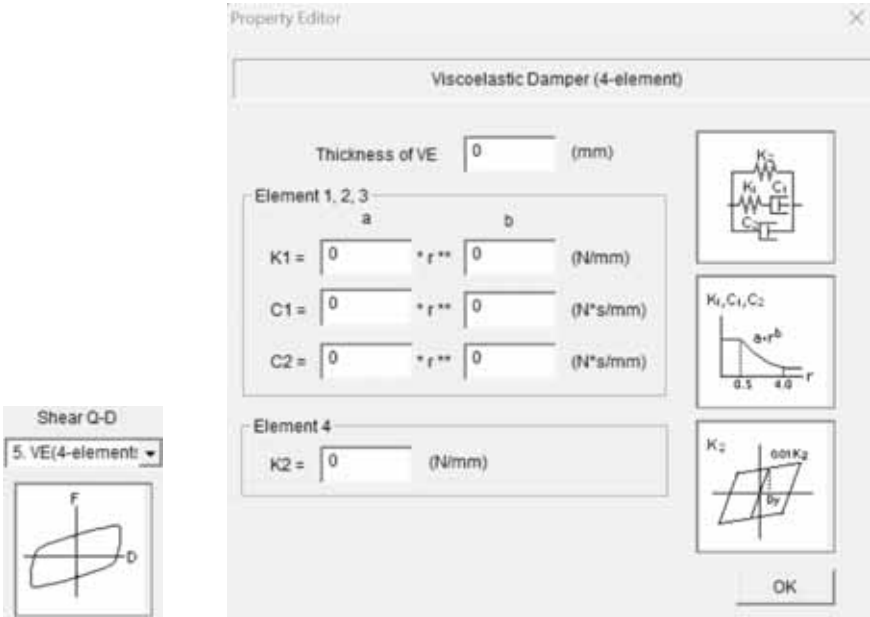
Damping (kN\*s/mm)  
C 0

Damping ratio  
C2 / C1 0

Relief velocity (mm/s)  
V0 0

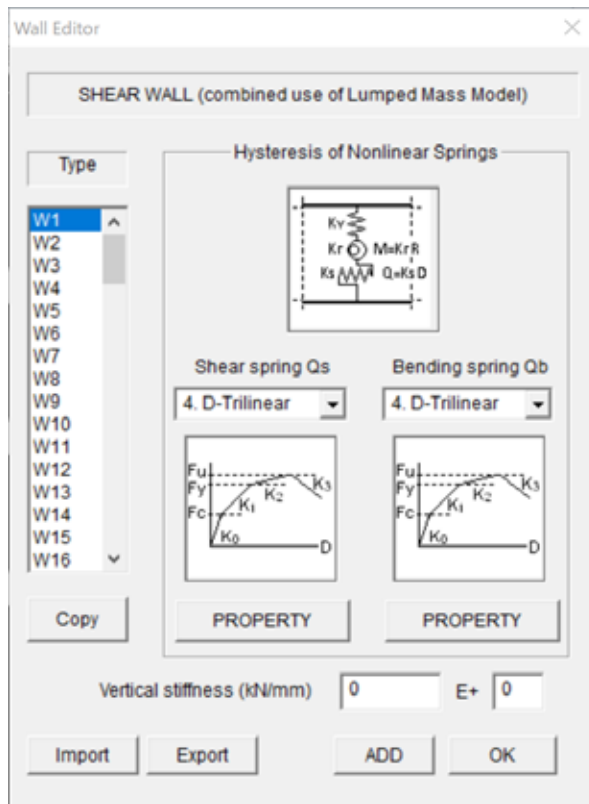
OK

Viscoelastic (4-element model)



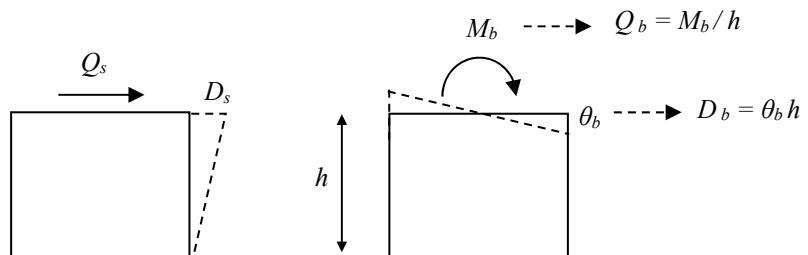
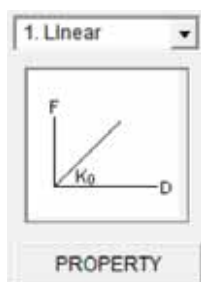
## 6.12 Wall (Direct input for parameters of hysteresis model)

WALL ()

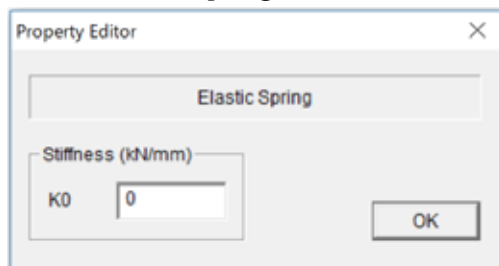


- It consists of shear spring, bending spring and axial spring.
- Please select hysteresis types for shear and bending springs from the pull-down menu.
- You can input the parameters of the hysteresis in [PROPERTY] view.
- Please input vertical stiffness directly.
- You can set default values for all members by selecting the last member type "Ddef".
- You can export member data to the text file "Data\_wall\_direct.txt" by [Export] button.
- You can import member data from a text file by [Import] button.

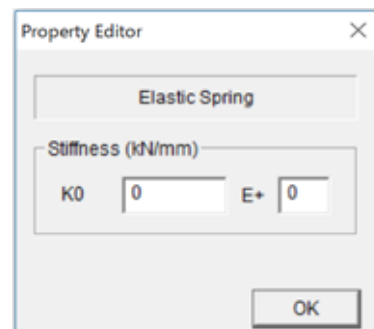
[1] Linear model



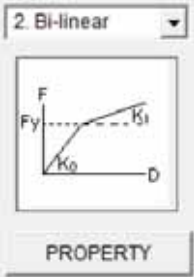
Shear spring ( $Q_s - D_s$ )



Bending spring ( $Q_b - D_b$ )



[2] Bi-linear model



Shear spring ( $Q_s$  - $D_s$ )

Bending spring ( $Q_b$  - $D_b$ )

Property Editor

Bilinear Hysteresis

Stiffness (kN/mm)

K0

Stiffness ratio

K1 / K0

Force (kN)

Fy

OK

Property Editor

Bilinear Hysteresis

Stiffness (kN/mm)

K0  E+

Stiffness ratio

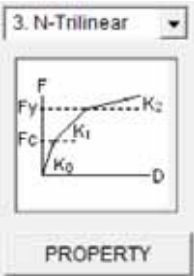
K1 / K0

Force (kN)

Qby  E+

OK

[3] Normal-Trilinear model



Shear spring ( $Q_s$  - $D_s$ )

Bending spring ( $Q_b$  - $D_b$ )

Property Editor

Trilinear Hysteresis

Stiffness (kN/mm)

K0

Stiffness ratio

K1 / K0

K2 / K0

Force (kN)

Fc

Fy

OK

Property Editor

Trilinear Hysteresis

Stiffness (kN/mm)

K0  E+

Stiffness ratio

K1 / K0

K2 / K0

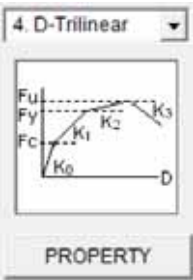
Force (kN)

Qbc  E+

Qby  E+

OK

[4] Degrading Trilinear model



Shear spring ( $Q_s - D_s$ )

Property Editor

Degrading Trilinear Hysteresis

Stiffness (kN/mm)

K0

Stiffness ratio

K1 / K0

K2 / K0

K3 / K0

Force (kN)

Fc

Fy

Fu

Hysteresis control parameters

Stiffness Degrading Ratio [0, 1]

Slip Stiffness Ratio [0, 1]

Strength Degrading Ratio [0, 1]

OK

Bending spring ( $Q_b - D_b$ )

Property Editor

Degrading Trilinear Hysteresis

Stiffness (kN/mm)

K0  E+

Stiffness ratio

K1 / K0

K2 / K0

K3 / K0

Force (kN)

Qbc  E+

Qby  E+

Qbu  E+

Hysteresis control parameters

Stiffness Degrading Ratio [0, 1]

Slip Stiffness Ratio [0, 1]

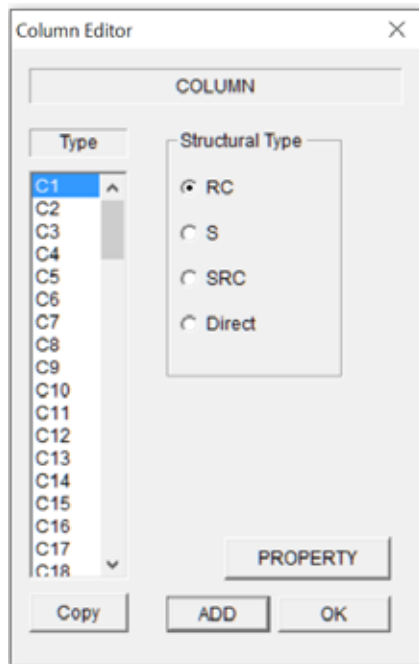
Strength Degrading Ratio [0, 1]

OK



### 6.13 Column (Mixed mode)

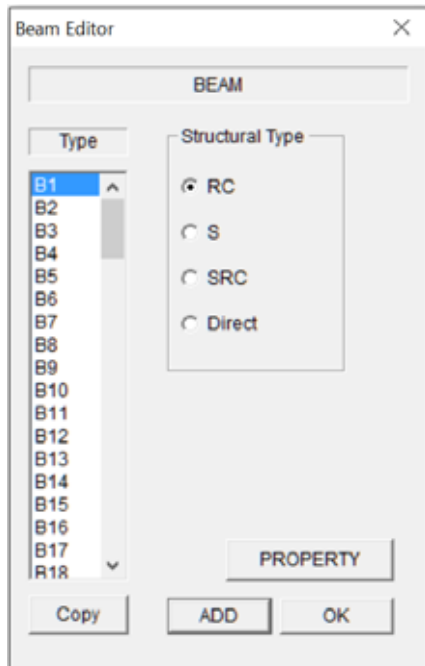
COLUMN ()



- You can select different “Structural Type” for each member type (C1, C2, ...).
- You can input member properties by [PROPERTY] button.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type “Cdef”.

### 6.14 Beam (Mixed mode)

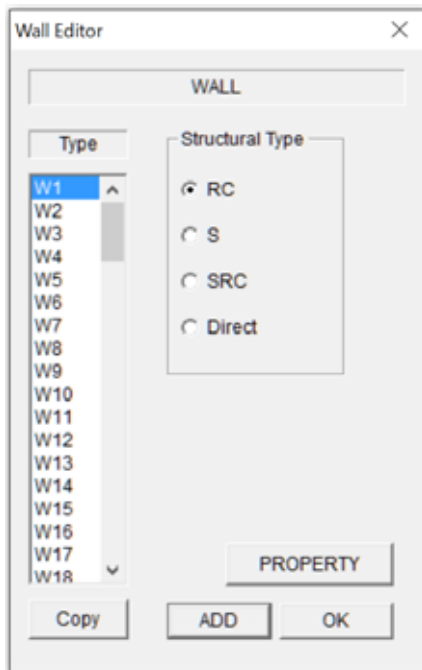
BEAM ()



- You can select different “Structural Type” for each member type (B1, B2, ...).
- You can input member properties by [PROPERTY] button.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type “Bdef”.

## 6.15 Wall (Mixed mode)

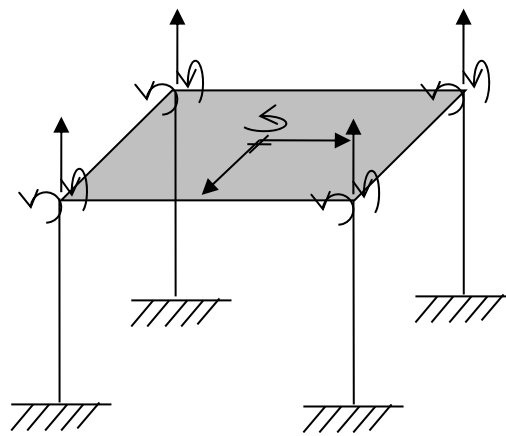
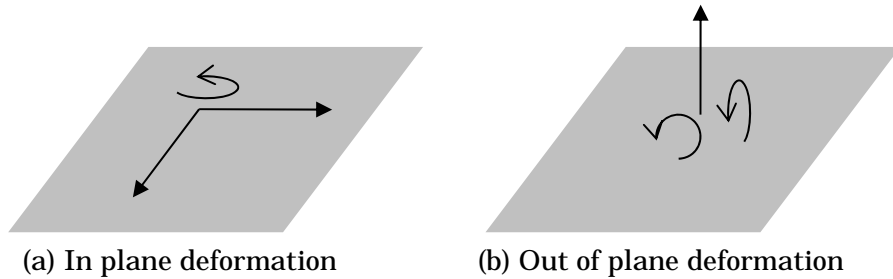
WALL ()



- You can select different “Structural Type” for each member type (W1, W2, ...).
- You can input member properties by [PROPERTY] button.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type “Wdef”.

### 6.16 Floor Slab (2D Rigid)

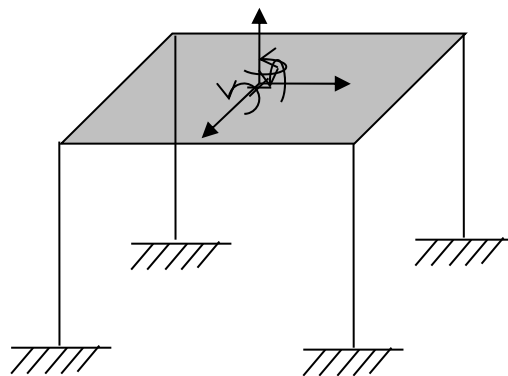
- Rigid for in plane deformation and free for out-of-plane deformation. In plane deformations at nodes are dependent to those of center of gravity.



(c) Independent freedom for 2D Rigid


### 6.17 Floor Slab (3D Rigid)

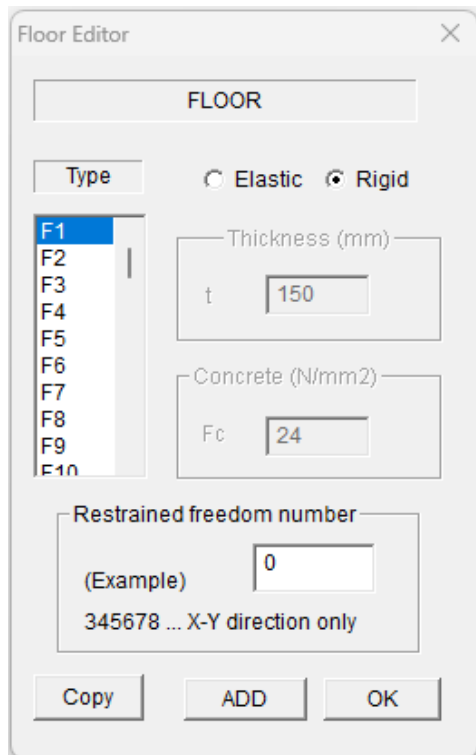
- Rigid for all directional deformation. Deformations at nodes are dependent to those of center of gravity.



(d) Independent freedom for 3D Rigid

## 6.18 Floor Slab (Flexible)

SLAB (  )

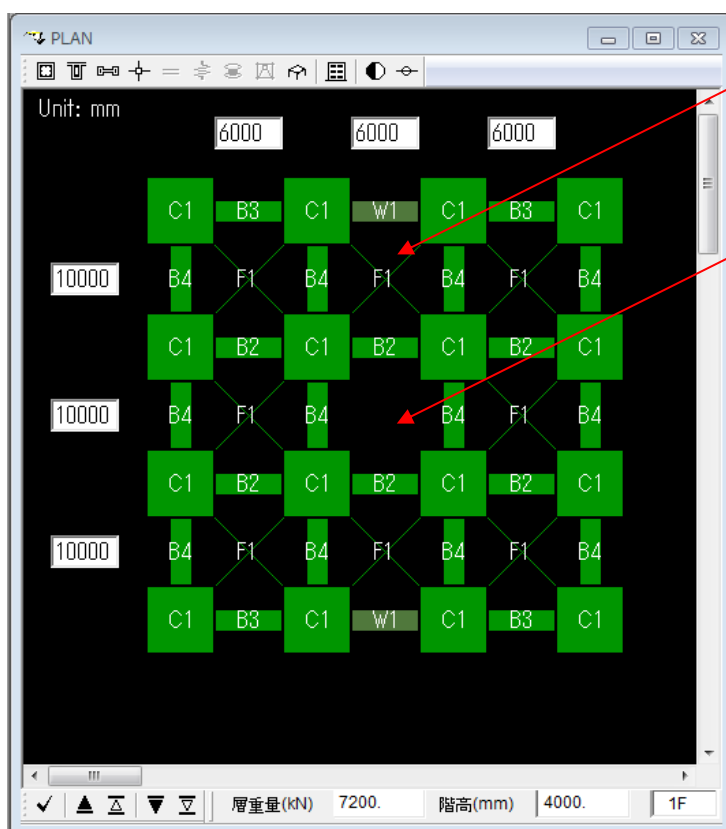


The **Floor Editor** dialog box is used to configure floor slab properties. It includes a list of member types (F1 to F10), a selection of **Elastic** or **Rigid** behavior, input fields for **Thickness (mm)** (t) and **Concrete (N/mm<sup>2</sup>)** (Fc), and a **Restrained freedom number** field with an example showing '0' and a note '345678 ... X-Y direction only'. Buttons for **Copy**, **ADD**, and **OK** are at the bottom.

- **Elastic:** elastic for in plane deformation and free for out-of-plane deformation.

Please input thickness and concrete strength.


- **Rigid:** the selected slab will be rigid both in plane and out-of-plane. You can set the restrained freedom of the partial rigid slab at the center of gravity.
- You can set default values for all members by selecting the last member type "Fdef".

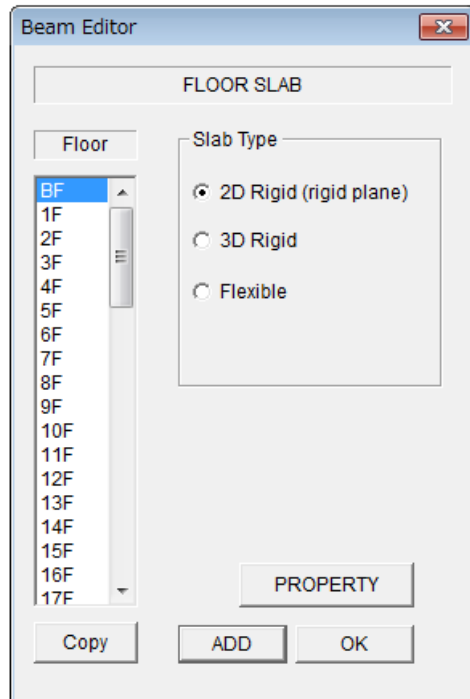


You can set member number (F1 ~ F100).

By clicking the slab, you can remove it. If you click it again, the slab will appear.

## 6.19 Floor Slab (Mix)

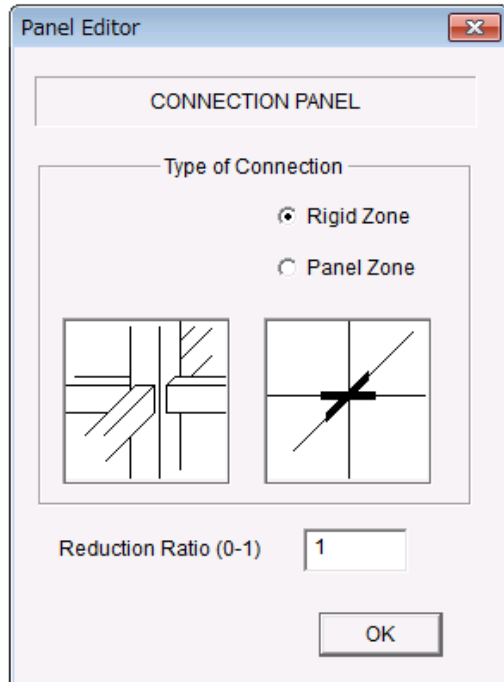
SLAB (  )



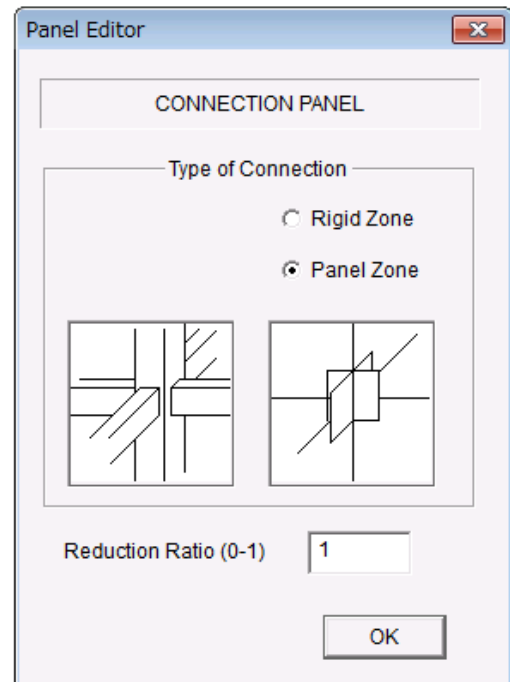
- You can select different “Slab Type” for each floor (BF, 1F, 2F, ...).
- In case of “Flexible”, you can input member properties by [PROPERTY] button.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.

## 6.20 Connection Panel

CONNECTION PANEL (  )




Rigid Zone

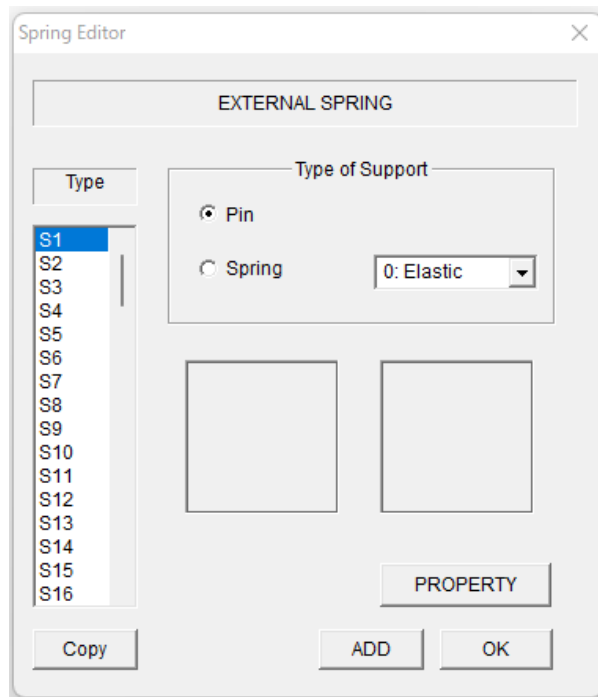


Panel Zone

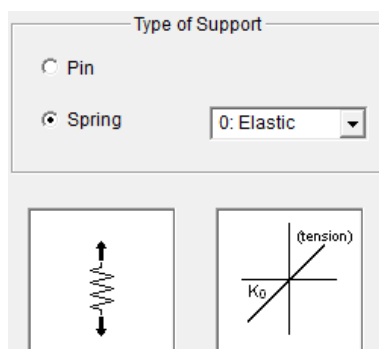
You can set the ratio of the length of rigid zone or panel zone inside connection area. The default value is 1.0 (to the member face).

## 6.21 External Spring

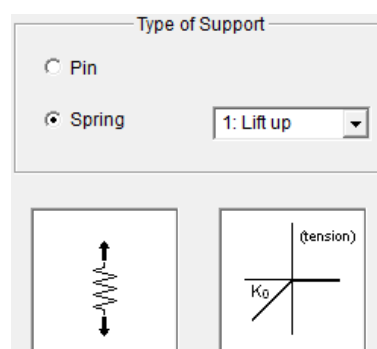
EXTERNAL SPRING (  ) (NOTE: only available at the Basement Floor or you select External Spring in Option menu. Default is PIN)



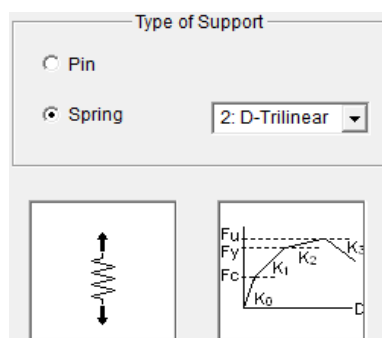
- You can select “Pin” support or “Spring” support. The default setting is “Pin”.
- As for “Spring”, you can select from “0. Elastic, 1. Lift up, 2: D-Trilinear, 3: Air Spring, 4: Base Plate, 5: Pendulum”.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by selecting the last member type “Sdef”.



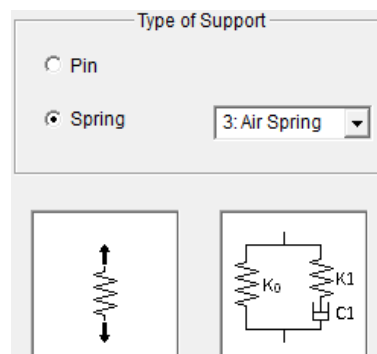
Elastic



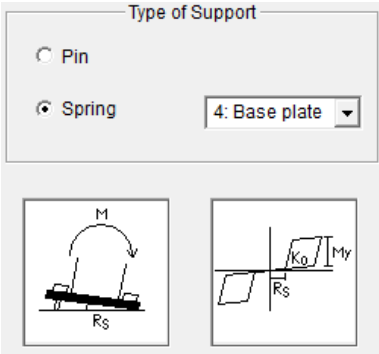
Lift up



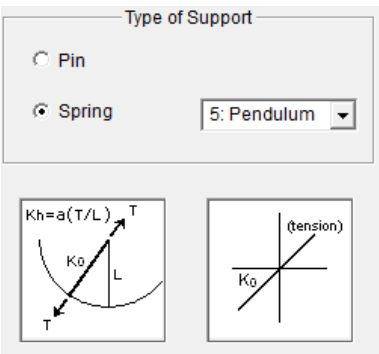
D-Trilinear



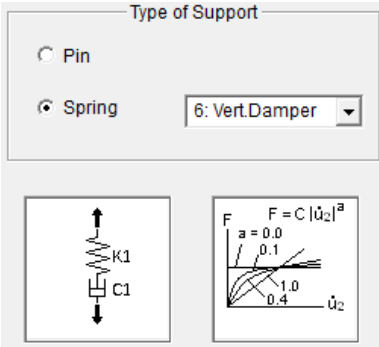
Air Spring



Base plate



Pendulum

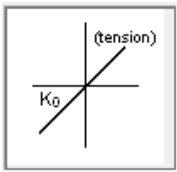


Vertical Viscous Damper

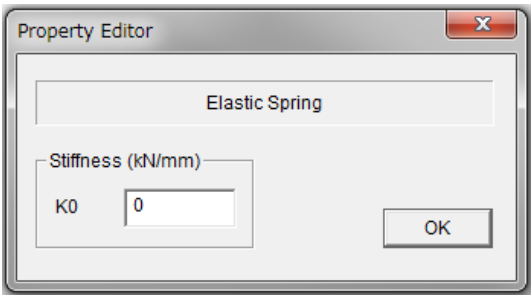
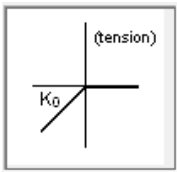


- [PROPERTY] menu

[0] In case of “Elastic”

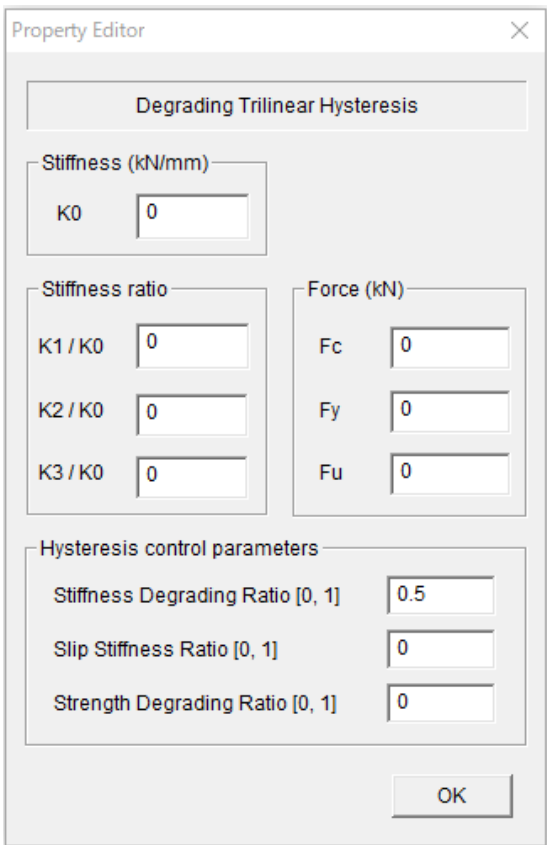
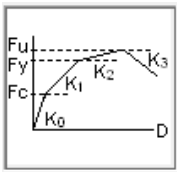


and [1] “Lift Up”

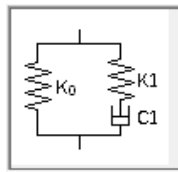


- Please input spring stiffness from the [PROPERTY] menu.

[2] In case of “D-Trilinear”



- Please input spring properties from the [PROPERTY] menu.



[3] In case of “Air Spring”

Property Editor

Air Damper

Stiffness 1 (kN/mm)  
K0 0

Stiffness 2 (kN/mm)  
K1 0

Damping (kN\*s/mm)  
C1 0

Nonlinear Parameter  
B 0

damping force =  $C1 \dot{V}^B$

OK

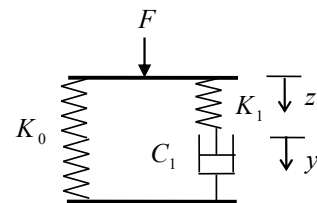
- Please input spring properties from the [PROPERTY] menu.

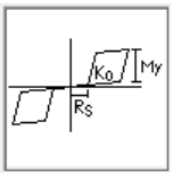
The air spring force is expressed as

$$F = K_1(z - y) + K_0 z$$

$$K_1(z - y) = C_1 \cdot \dot{y}^B$$

Please refer “Technical Manual” for the detail.





[4] In case of “Base Plate”

Property Editor

Slip Model

Stiffness (kNm)

$K_0$

Moment (kNm)

$M_y$

Stiffness ratio

$K_1 / K_0$

Initial Slip Angle

$R_x (+)$

$R_x (-)$

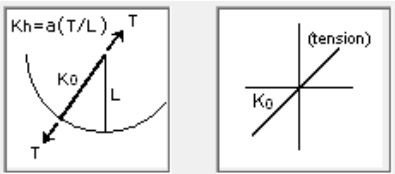
$R_y (+)$

$R_y (-)$

OK

A schematic diagram of a base plate spring model. It shows a horizontal line representing the base plate, with a vertical line representing the spring. The spring is labeled with  $K_0$  and  $M_y$ . A coordinate system is shown with  $R_s$  on the horizontal axis and  $M_y$  on the vertical axis.

- Please input spring properties from the [PROPERTY] menu.



[5] In case of “Pendulum”

Property Editor

Pendulum Spring

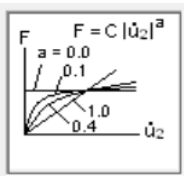
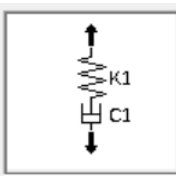
1.  $K_0$  : Axial Stiffness (kN/mm)

2.  $a$  : Horizontal Stiffness Ratio [0, 2]   
( Horizontal Stiffness  $K_h = a (T/L)$  )

OK

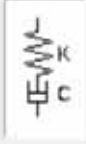
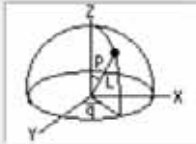
- Please input spring properties from the [PROPERTY] menu.

[6] In case of “Vertical Spring”



Property Editor

Vertical Viscous Damper



K=0 (no spring)

Length (mm)  
L 0

Stiffness (kN/mm)  
K 0

Damping (kN\*s/mm)  
C 0


Angle (degree)  
p 0  
q 0

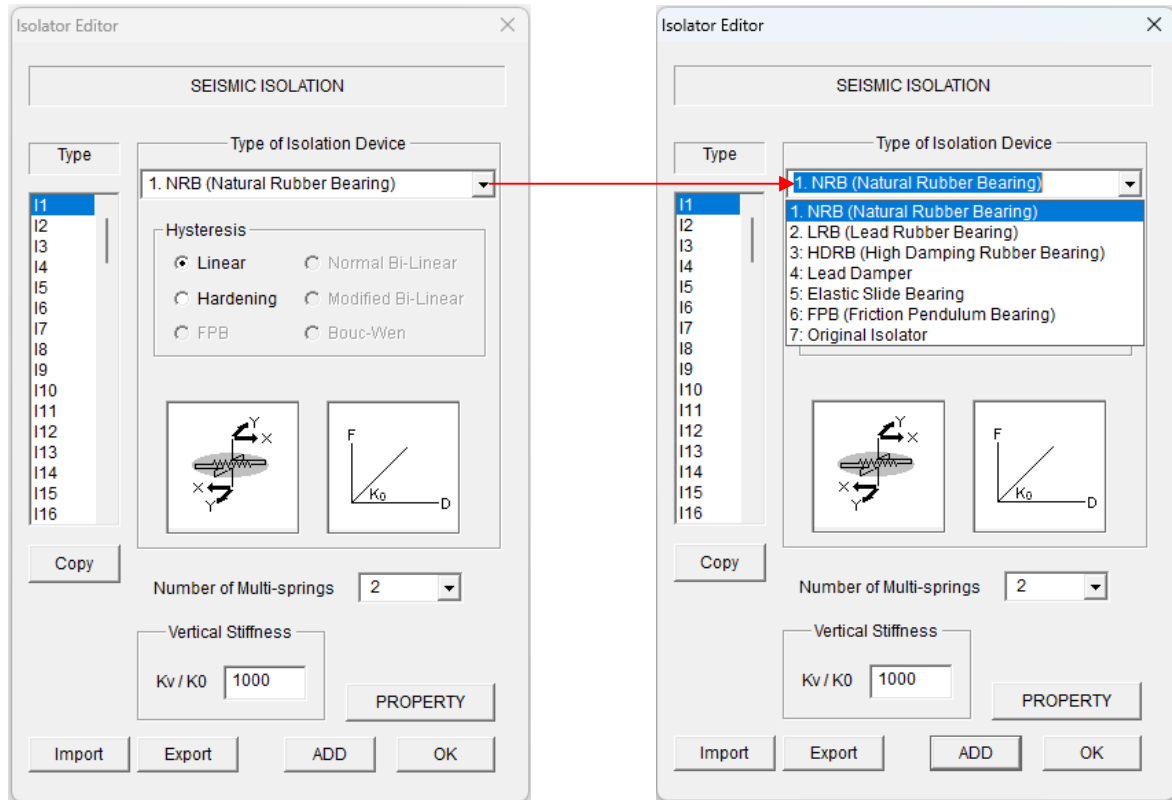
Exponent  
a 0

OK

- Please input spring properties from the [PROPERTY] menu.

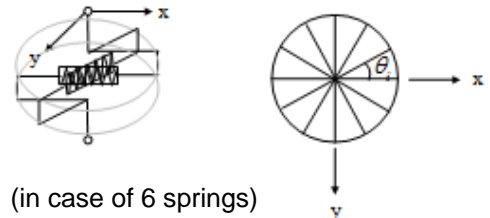
## 6.22 Seismic Isolator

ISOLATOR (  ) (NOTE: only available when you select Isolator in Option menu)



You can select Isolator device from the pull-down menu:

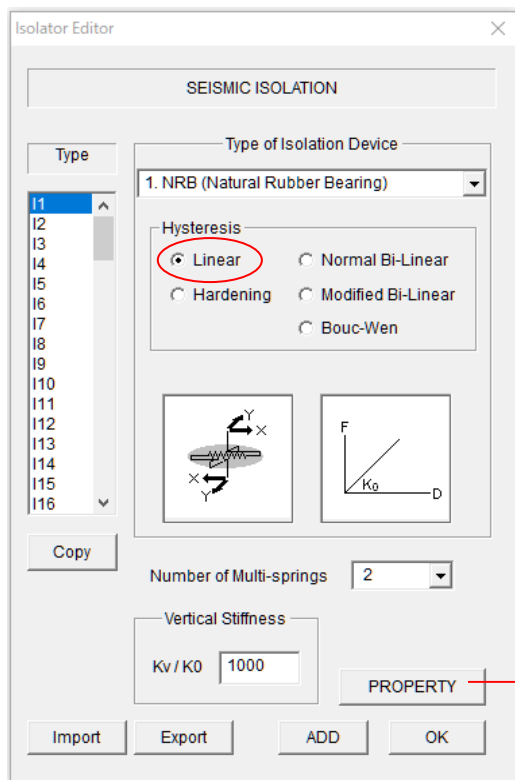
1. NRB (Natural Rubber Bearing)
2. LRB (Lead Rubber Bearing)
3. HDRB (High Damping Rubber Bearing)
4. Lead Damper
5. Elastic Slide Bearing
6. FPB (Friction Pendulum Bearing)
7. Original Isolator



Please read “STERA3D\_technical\_manual” about the detail of each device.

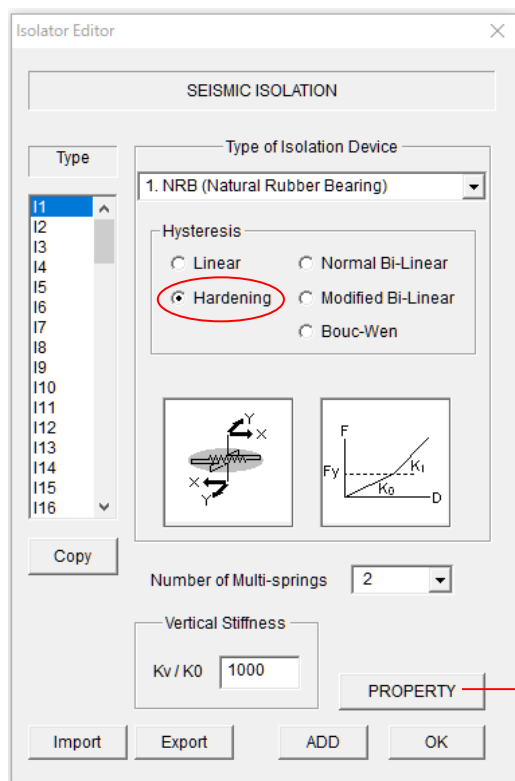
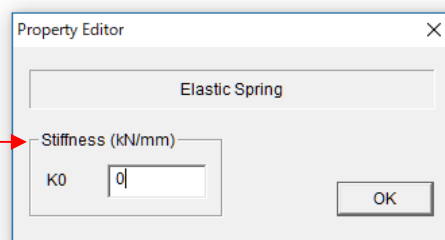
- The default value of the ratio between vertical stiffness,  $K_v$ , and the horizontal stiffness,  $K_0$ , is 1000.
- You can select the number of multi-springs from the pull-down menu (2, 4, 6, 8, 10).
- You can set default values for all members by selecting the last member type “Idef”.
- You can export member data to the text file “Data\_isolator.txt” by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

[1] NRB Isolator

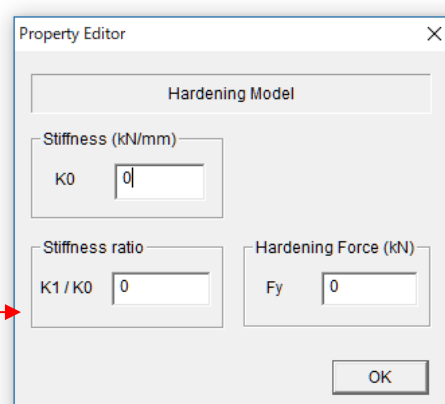


- For NRB isolator, you can select “Linear” or “Hardening” hysteresis.
- Please input spring property by [PROPERTY] button.

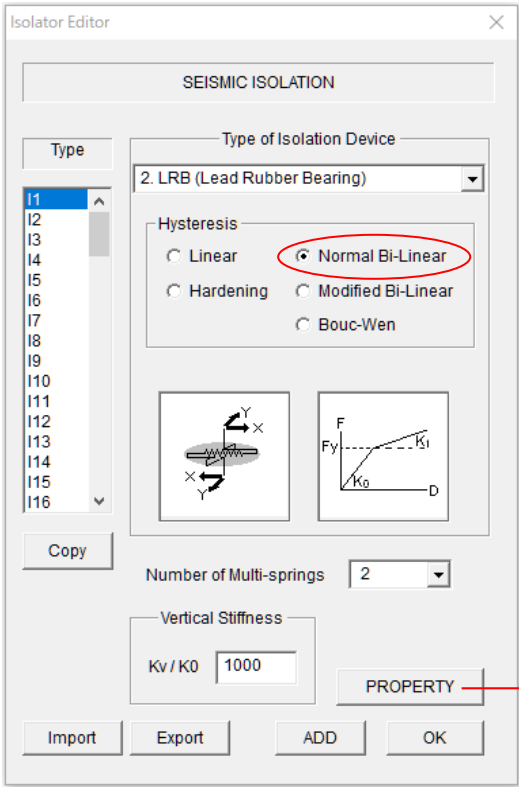
In case of “Linear”



In case of “Hardening”

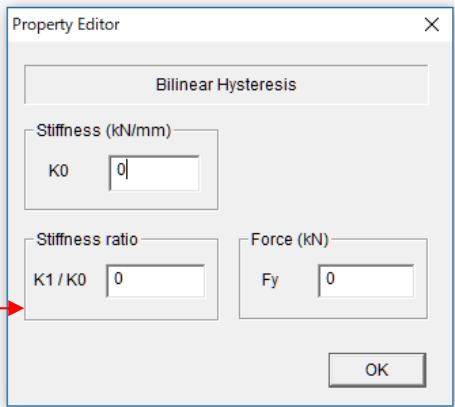


[2] LRB Isolator



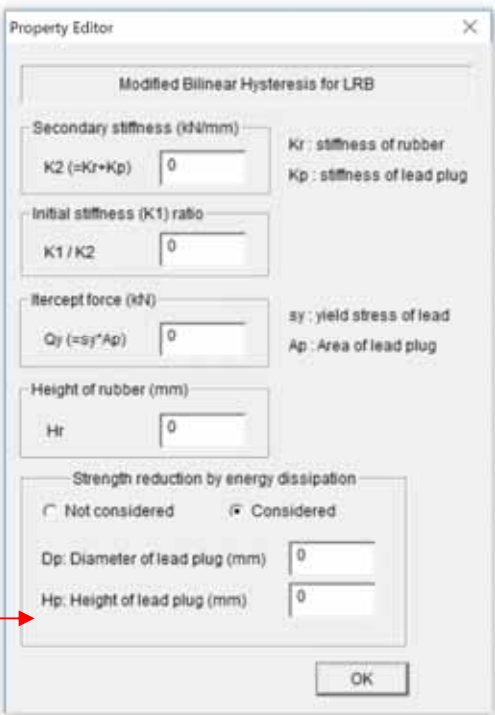
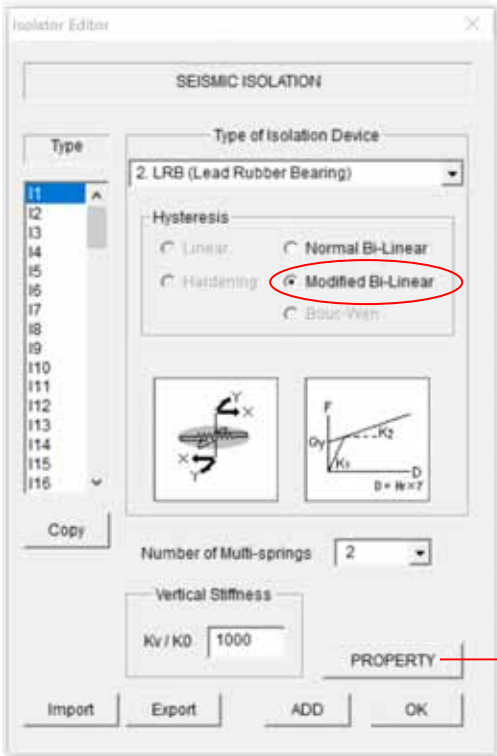
- For LRB isolator, you can select “Normal Bi-Linear” or “Modified Bi-Linear” hysteresis.
- Please input spring property by [PROPERTY] button.

In case of “Normal Bi-Linear”

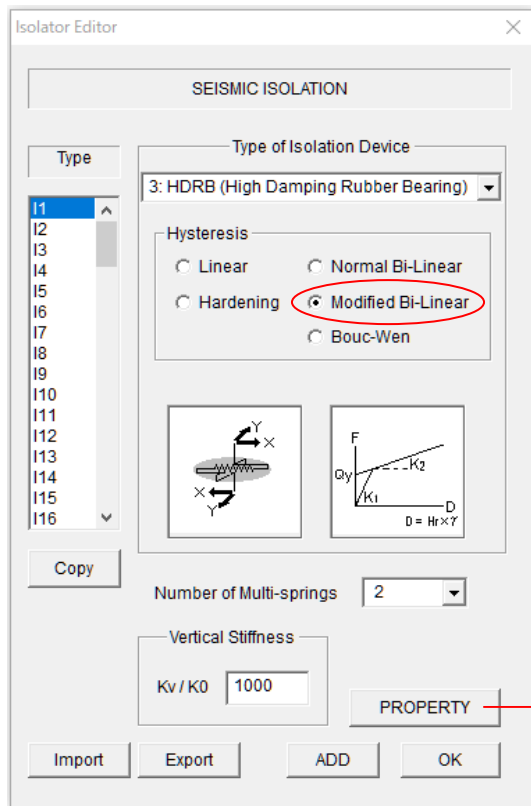


In case of “Modified Bi-Linear”

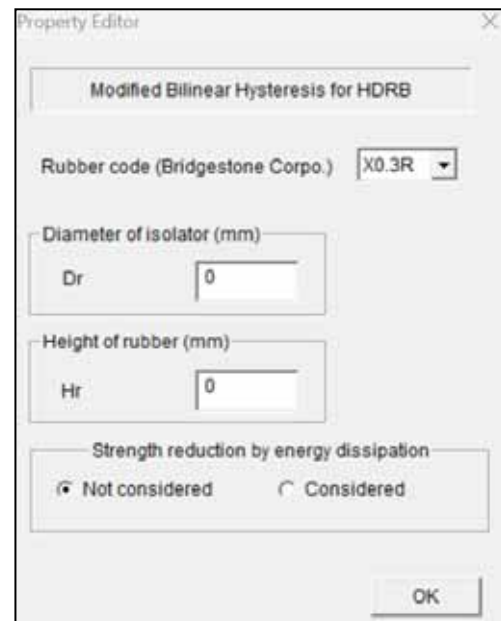
- In case of “Modified Bi-Linear” hysteresis, you can consider strength reduction by energy dissipation.



[3] HDRB (High Damping Rubber Bearing) Isolator

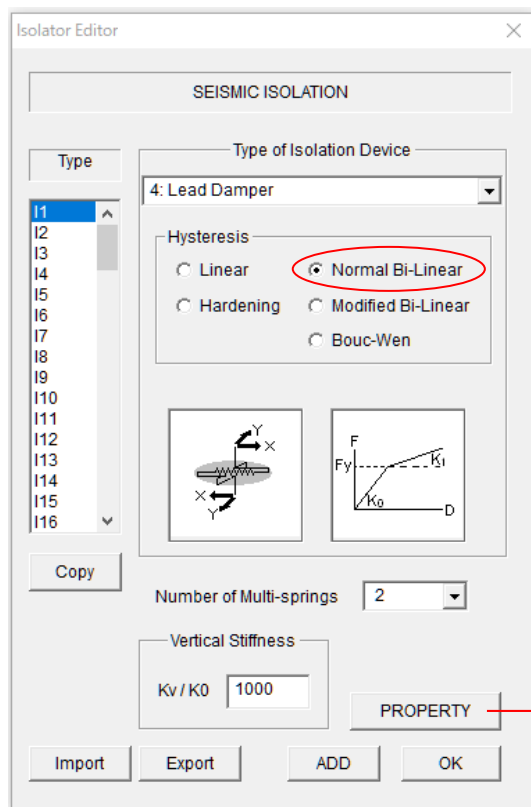


- For HDRB isolator, you can select “Modified Bi-Linear” hysteresis only.
- Please input spring property by [PROPERTY] button.
- You can consider strength reduction by energy dissipation.

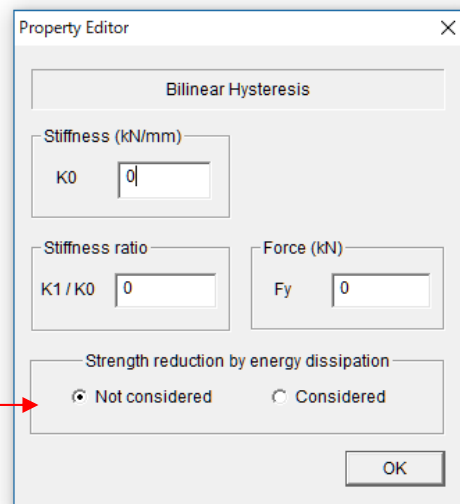




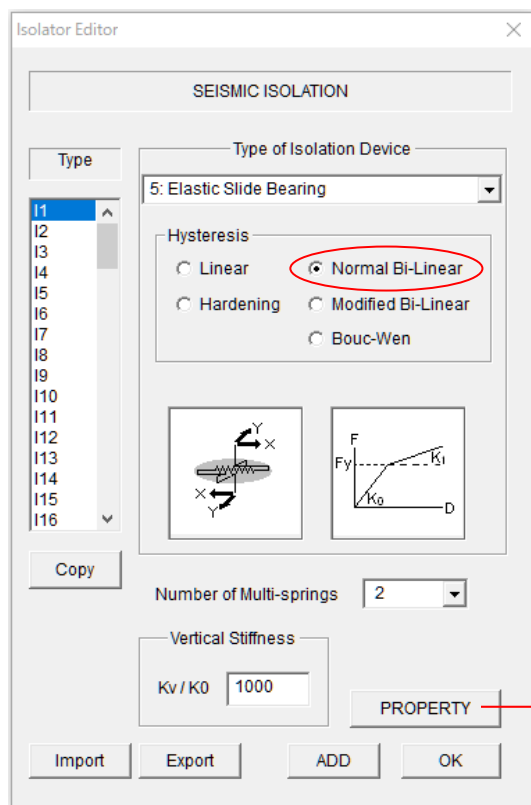
[4] Lead Damper



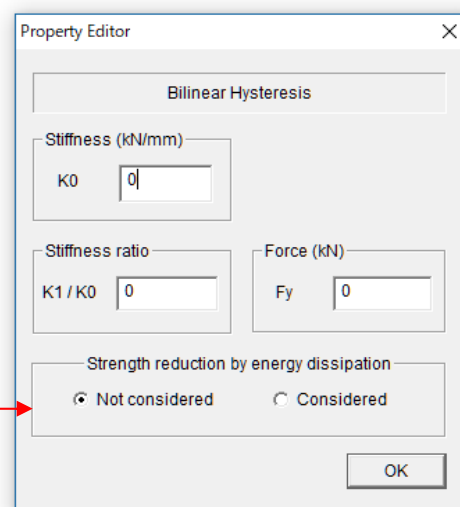
- For Lead Damper, you can select “Normal Bi-Linear” hysteresis only.
- Please input spring property by [PROPERTY] button.
- You can consider strength reduction by energy dissipation.



[5] Elastic Slide Bearing

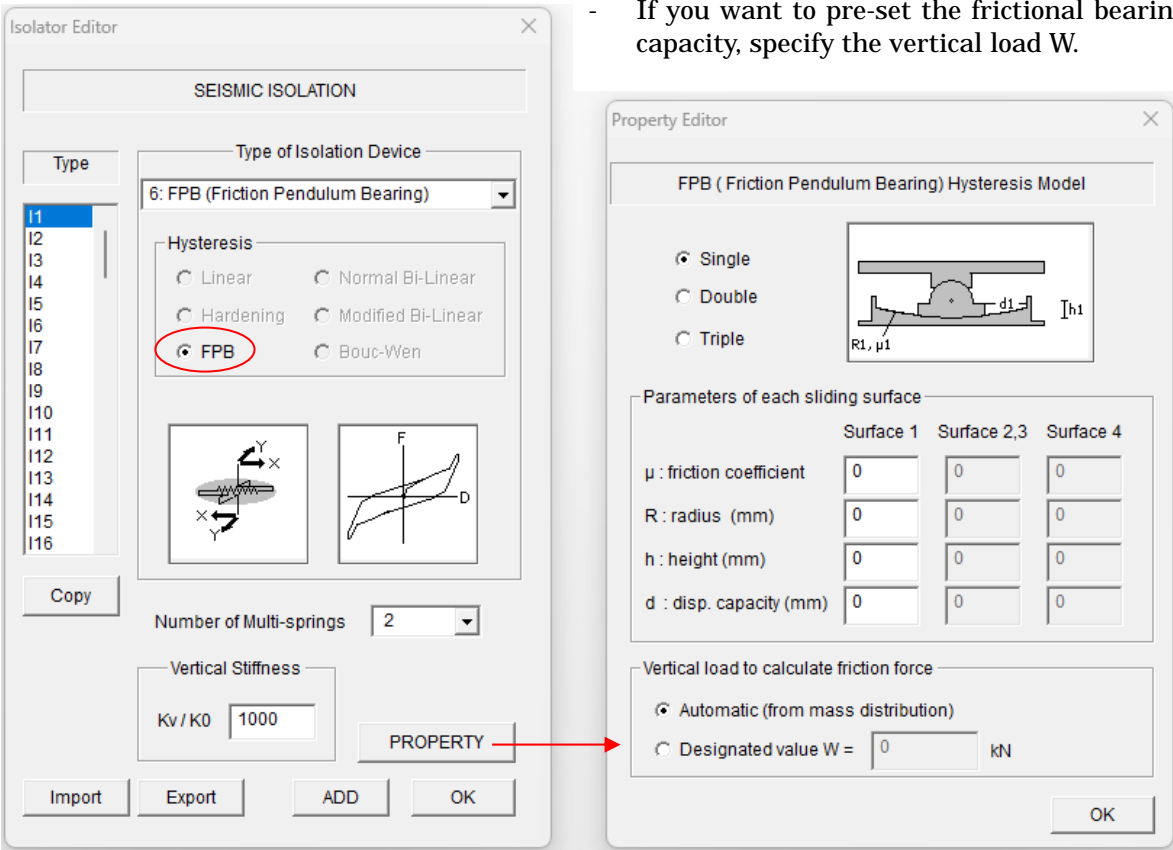


- For Elastic Slide Bearing, you can select “Normal Bi-Linear” hysteresis only.
- Please input spring property by [PROPERTY] button.
- You can consider strength reduction by energy dissipation.

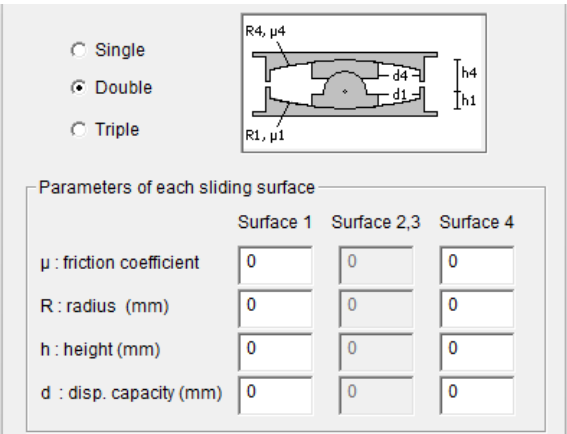


[6] FPB (Friction Pendulum Bearing)

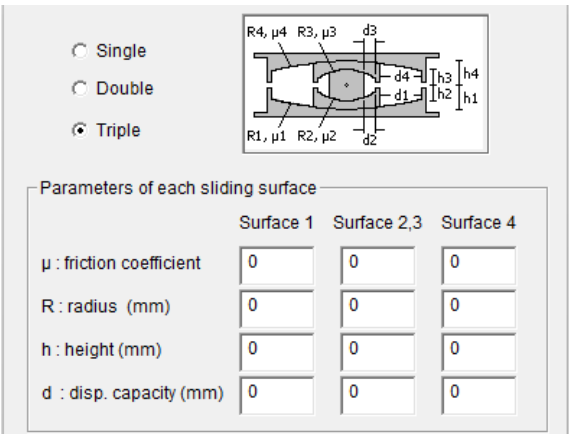
- Please input spring property by [PROPERTY] button.
- If you want to pre-set the frictional bearing capacity, specify the vertical load W.



In case of Double,

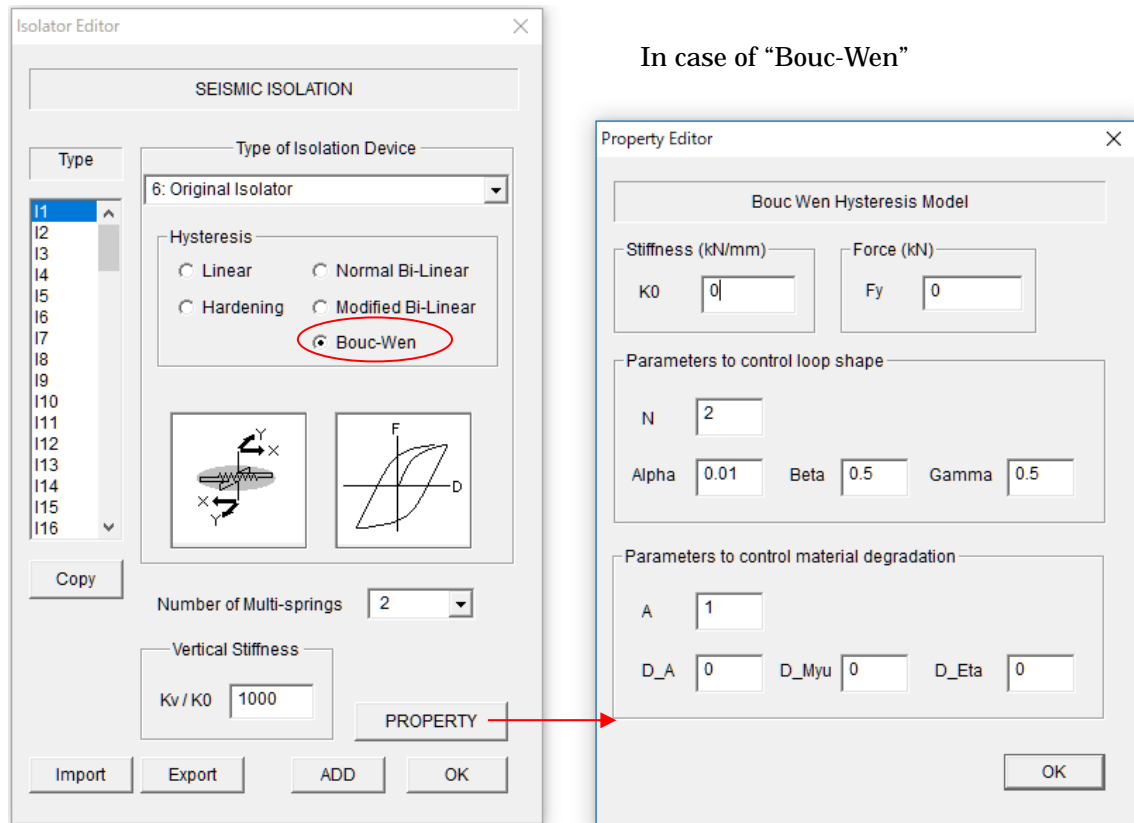


In case of Triple



[7] Original Isolator

- If you want to use your original isolator device, please select “Original Isolator”. You can select the hysteresis from “Linear”, “Normal Bi-Linear”, and “Bouc-Wen”.
- Please input spring property by [PROPERTY] button.



Definition of Bouc-Wen Model

(Please refer “Technical Manual” for the detail.)

$$f = \alpha k_0 x + (1 - \alpha) k_0 z$$

$$\dot{z} = \frac{A\dot{x} - \left\{ \beta |\dot{x}| |z|^{N-1} z + \gamma \dot{x} |z|^N \right\} v}{\eta}$$


$$A = A_0 - \delta_A e, \quad v = 1 + \delta_v e, \quad \eta = 1 + \delta_\eta e$$

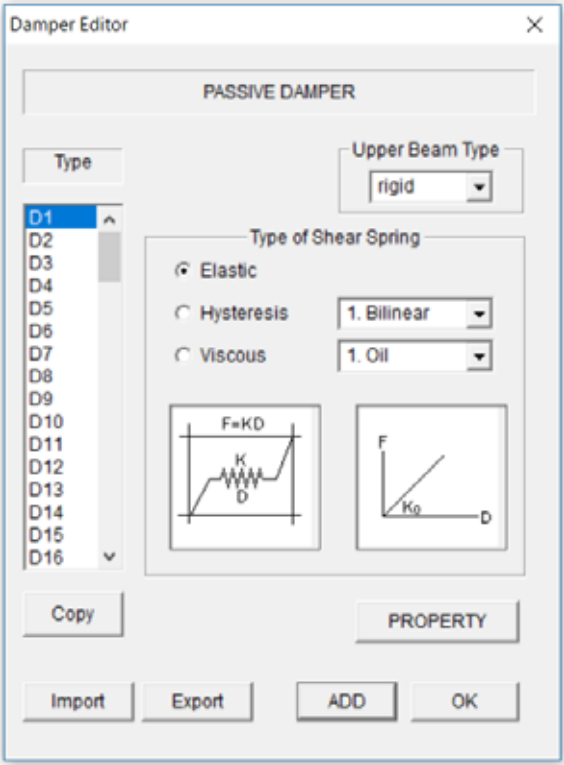
$$\text{Alpha} = \alpha, \text{Beta} = \beta, \text{Gamma} = \gamma$$

$$A = A_0$$

$$D\_A = \delta_A, D\_Myu = \delta_v, D\_Eta = \delta_\eta,$$

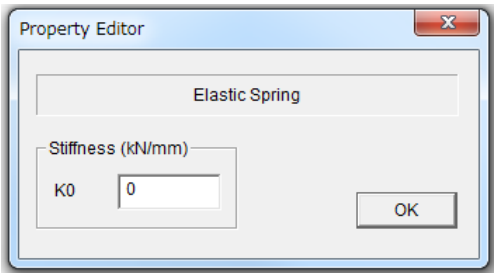
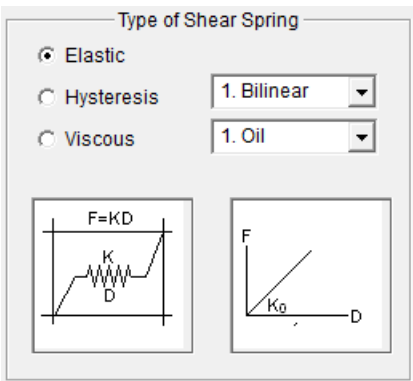
6.23 Passive Damper

PASSIVE DAMPER (  ) (NOTE: only available when you select Passive Damper in Option menu)



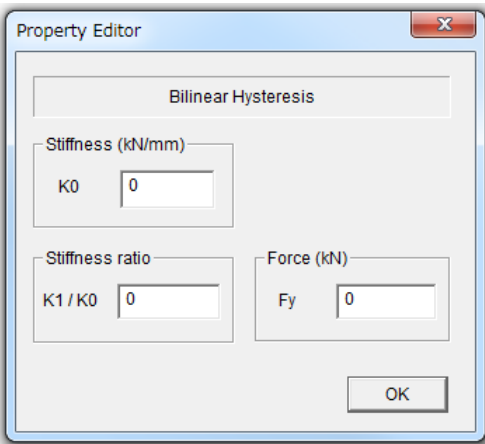
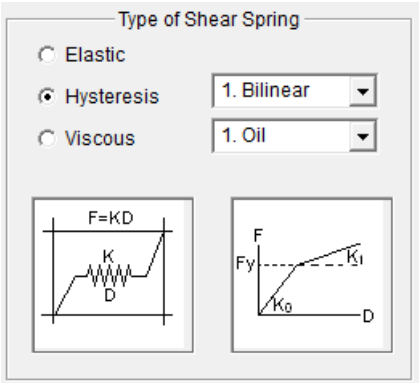
- Please select damper type from Elastic, Hysteresis and Viscous and its detail characteristics from the pull down menu.
- If there is a reinforcement concrete beam upper of Damper, please select the upper beam type number from the pop-up menu. The default is “rigid beam”.
- You can set default values for all members by selecting the last member type “Ddef”.
- You can export member data to the text file “Data\_damper.txt” by [Export] button.
- You can import member data from a text file by [Import] button.
- You can input the detail characteristic of the Damper in [PROPERTY] view.
- Please click [OK] to finish.

[1] Elastic spring

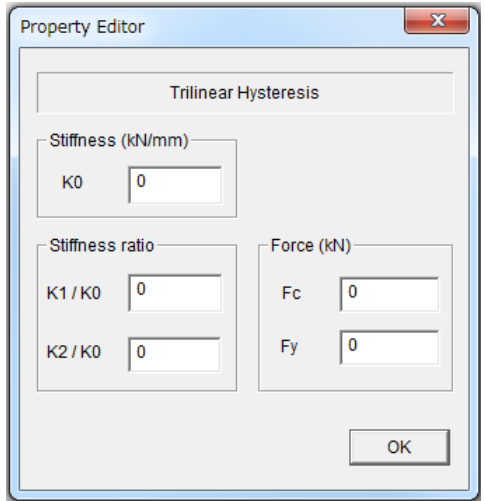
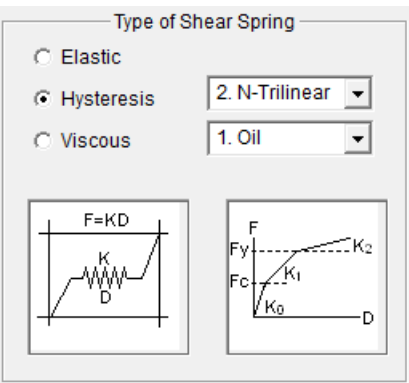


[2] Hysteresis Dampers

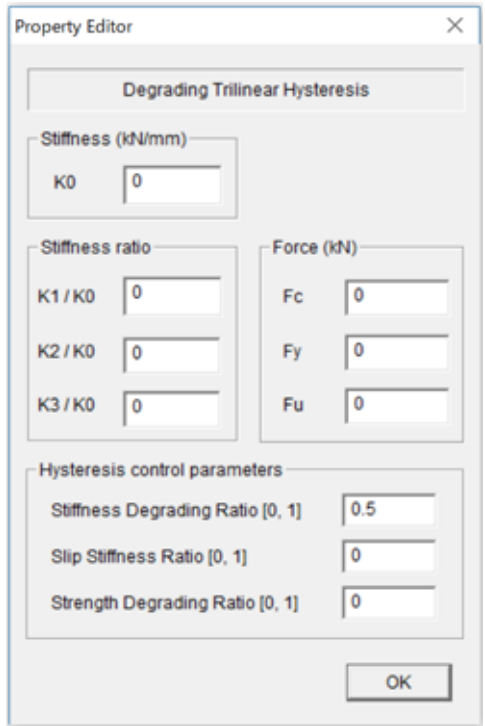
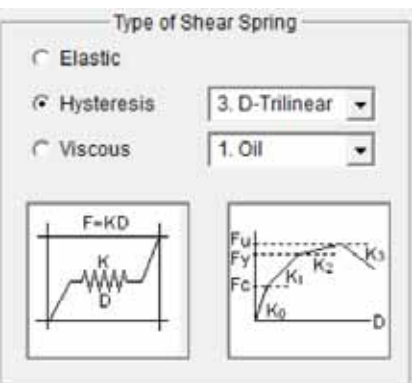
Bilinear



Normal Trilinear



Degrading Trilinear



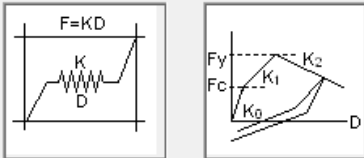
Poly-linear Slip

Type of Shear Spring

☐ Elastic

☒ Hysteresis 4. Poly-Slip

☐ Viscous 1. Oil



Property Editor

Poly-linear Slip Hysteresis

Stiffness (kN/mm)

K0 0

Stiffness ratio

K1 / K0 0

K2 / K0 0

Force (kN)

Fc 0

Fy 0

Numerical Integration Method

☒ Average Acceleration (ignore negative stiffness)

☐ Operator Splitting (with negative stiffness)

OK

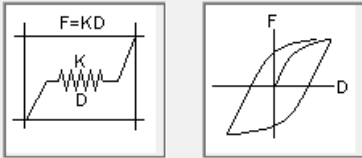
Bouc-Wen

Type of Shear Spring

☐ Elastic

☒ Hysteresis 5. Bouc-Wen

☐ Viscous 1. Oil



Property Editor

Bouc Wen Hysteresis Model

Stiffness (kN/mm)

K0 0

Force (kN)

Fy 0

Parameters to control loop shape

N 2

Alpha 0.01 Beta 0.5 Gamma 0.5

Parameters to control material degradation

A 1

D\_A 0 D\_Myu 0 D\_Eta 0

OK

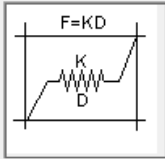
Nonlinear Spring (without hysteresis)

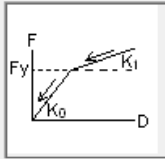
Type of Shear Spring

☐ Elastic

☒ Hysteresis 6. N-Spring

☐ Viscous 1. Oil





Property Editor

Bilinear Hysteresis

Stiffness (kN/mm)

K0 0

Stiffness ratio

K1 / K0 0

Force (kN)

Fy 0

OK

[3] Viscous Dampers

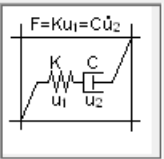
Oil damper

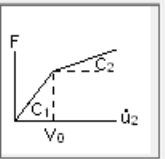
Type of Shear Spring

☐ Elastic

☐ Hysteresis 4. M-Bilinear

☒ Viscous 1. Oil





Property Editor

Oil Damper

Stiffness (kN/mm)

K 0



K=0 (no spring)

Damping (kN\*s/mm)

C1 0

Relief velocity (mm/s)

V0 0

Damping ratio

C2 / C1 0

OK

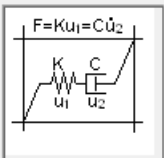
Viscous damper

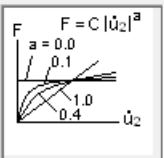
Type of Shear Spring

☐ Elastic

☐ Hysteresis 4. M-Bilinear

☒ Viscous 2. Viscous



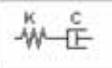


Property Editor

Viscous Damper

Stiffness (kN/mm)

K 0



K=0 (no spring)

Damping (kN\*s/mm)

C 0

Exponent

a 0

OK

Viscoelastic damper (Voigt model)

Type of Shear Spring

☐ Elastic

☐ Hysteresis 1. Bilinear

☒ Viscous 3. VE(Voigt)

$F = K_1 u_1 + K_2 u_2 + C \dot{u}_2$

$F$

$\dot{u}_2$

Property Editor

Viscoelastic Damper

Stiffness (kN/mm)

K1 0

K2 0

K1=0 (no spring)

☐ with Friction Device

Damping (kN\*s/mm)

C 0

Friction Force (kN)

Fy 0

Damping ratio

C2 / C1 0

Relief velocity (mm/s)

V0 0

OK

Viscoelastic damper (4-elements model)

Type of Shear Spring

☐ Elastic

☐ Hysteresis 1. Bilinear

☒ Viscous 4. VE(4-element)

$F$

$D$

Property Editor

Viscoelastic Damper (4-element)

Thickness of VE 0 (mm)

Element 1, 2, 3

a b

K1 = 0 \* r \*\* 0 (N/mm)

C1 = 0 \* r \*\* 0 (N\*s/mm)

C2 = 0 \* r \*\* 0 (N\*s/mm)

Element 4

K2 = 0 (N/mm)

☐ with Friction Device

Friction Force (N)

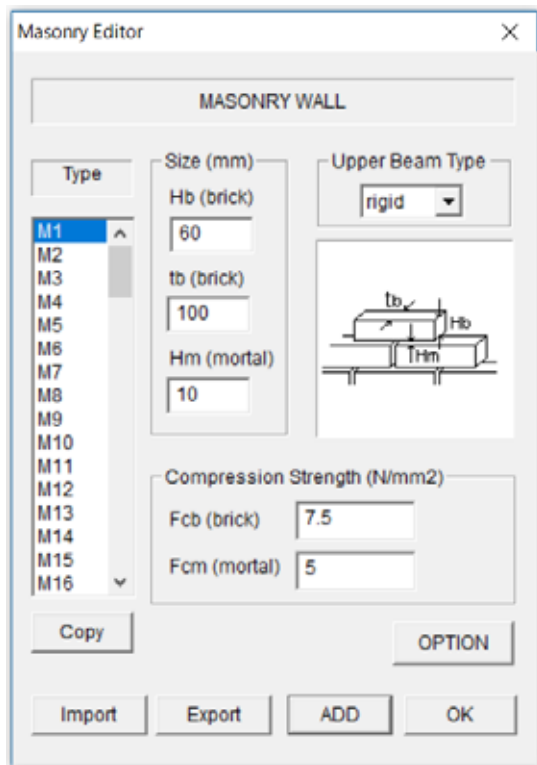
Fy = 0

OK

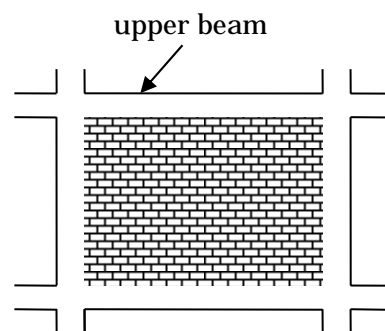


## 6.24 Masonry Wall

MASONRY ( = ) (NOTE: only available when you select Masonry in Option menu)



- Please input the size of brick unit and thickness of mortal and compression strength of these materials.
- If there is a reinforcement concrete beam upper of Masonry Wall as shown below, please select the upper beam type number from the pop-up menu. The default is “rigid beam”.



- You can set default values for all members by selecting the last member type “Mdef”.
- You can export member data to the text file “Data\_masonry.txt” by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

Masonry Option Editor

MASONRY WALL OPTION

1. Stiffness Modification Factor

2. Strength Modification Factor

Compressive Strength of Masonry Prism

☒ Pauley and Priestley 1992

☐ Eurocode 6 :  $f_m = k * F_{cb}^a * F_{cm}^b$

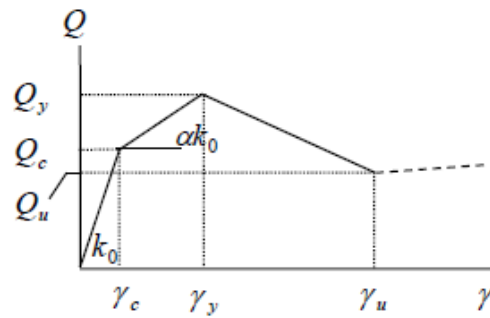
k  a  b

Numerical Integration Method

☒ Average Acceleration (ignore negative stiffness)

☐ Operator Splitting (with negative stiffness)

OK



- You can modify the stiffness and shear strength by multiplying modification factors in [OPTION] menu. The default values are 1.0.

- You can select the formula of compression strength of masonry prism from:

- Pauley and Priestley 1992
- Eurocode 6

See the technical manual for details.

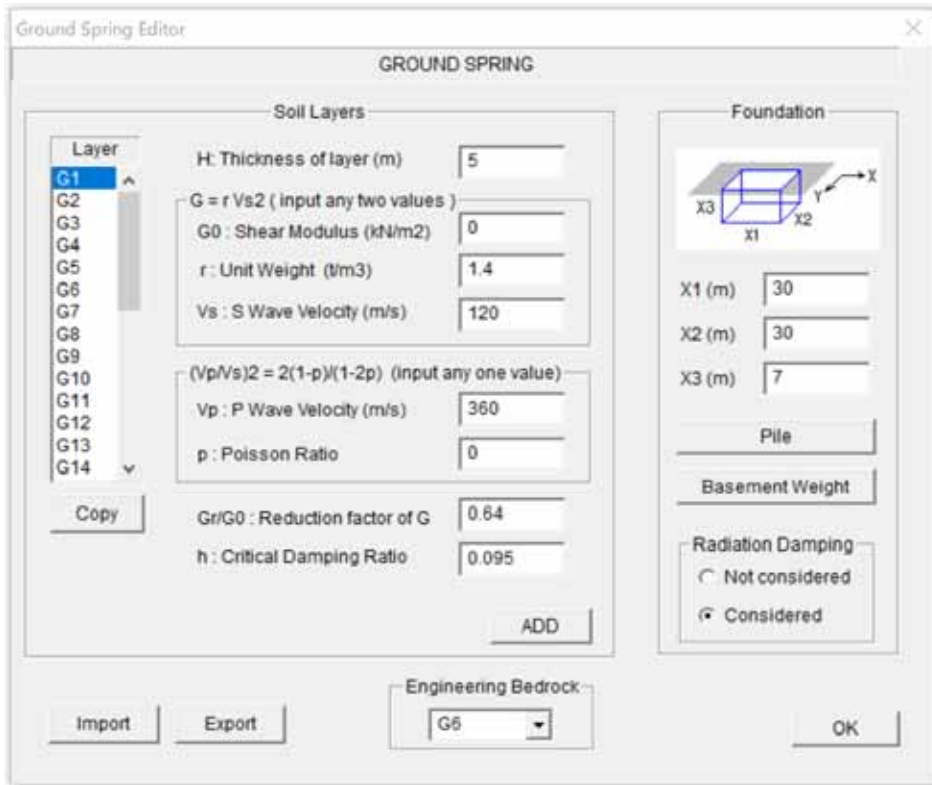
- Since the backbone curve of the shear spring has the negative stiffness after yielding, you can identify the numerical integration method in earthquake response analysis from:

- Average Acceleration method
- Operator Splitting method.

## 6.25 Ground Spring (Cone model)

GROUND SPRING ()

(NOTE: only available when you select Cone model in Option menu)

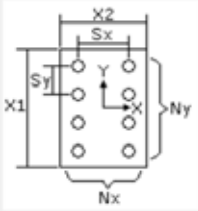


- Please input thickness and soil properties in each layer.
- Since the relationship  $G_0 = rV_s^2$  holds, if any two variables are input, the remaining variable is automatically calculated.
- Also, since the relationship  $\frac{V_p^2}{V_s^2} = \frac{2(1-p)}{1-2p}$  holds, if any one variable except  $V_s$  is input, the remaining variable is automatically calculated.
- Please input the size of foundation.
- Please input the number of the layer which is regarded as an engineering bedrock.
- Please input the properties of piles by [Pile] button.
- Please input the weight the basement by [Basement Weight] button.
- Please select "Radiation Damping" to consider or not.
- To move to the next layer, please click [ADD] button.
- You can copy the previous layer by [COPY] button.
- You can set default values for all layers by selecting the last layer type "Gdef".
- You can export layer data to the text file "Data\_ground\_cone.txt" by [Export] button.
- You can import layer data from a text file by [Import] button.
- Please click [OK] to finish.

When you click [Pile] button, the input window of pile location and property appears.

Pile Editor

Pile



Size of Basement (m)

X1 30 X2 30

Pile Space (m)

Sx 0 Sy 0

Number of Piles

Nx 0 Ny 0

Average Property of Pile

Diameter (m2) 0

Length (m) 0

Area (m2) 0

Moment of Inertia (m4) 0

E (N/mm2) 0 \*1000

OK

When you click [Basement Weight] Button, the input window of basement weight appears.

Property Editor

Basement Weight

Weight (kN)

0

Rotational Inertia Weight (kNm2)

around

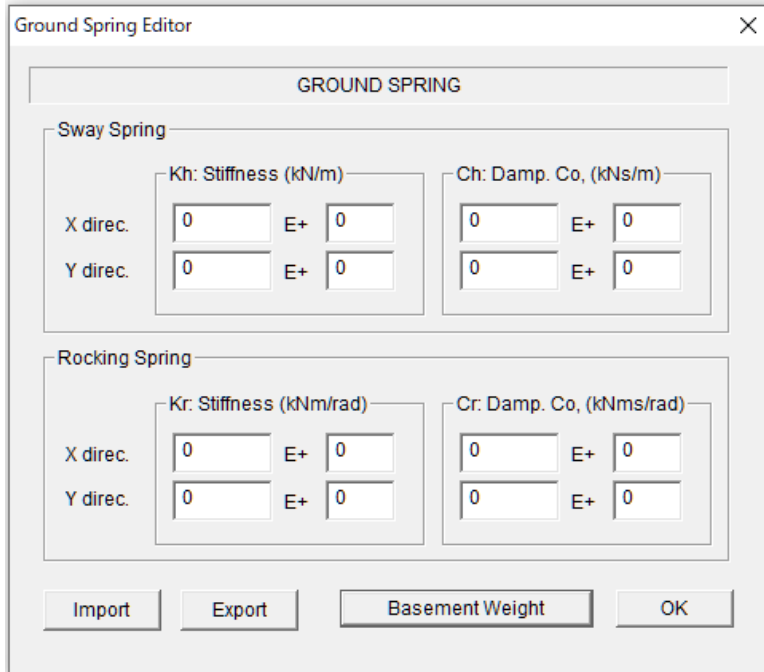
Y axis 0 E+ 0

X axis 0 E+ 0

OK

## 6.26 Ground Spring (Direct)

GROUND SPRING (  ) (NOTE: only available when you select Direct in Option menu)



The Ground Spring Editor dialog box is titled "Ground Spring Editor" and contains a "GROUND SPRING" section. It is divided into two main parts: "Sway Spring" and "Rocking Spring".

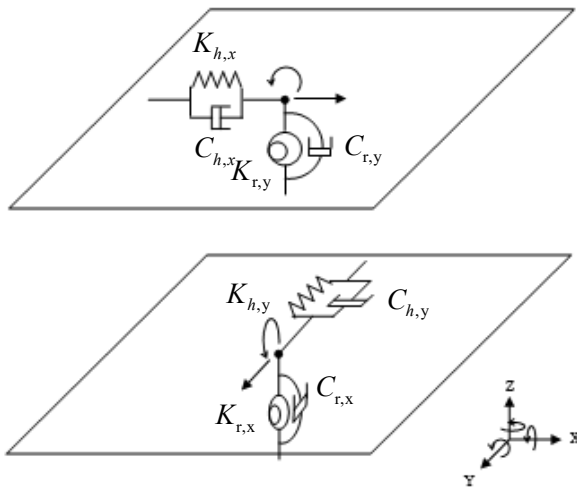
**Sway Spring:**

- Kh: Stiffness (kN/m):**
  - X direc.: 0 E+ 0
  - Y direc.: 0 E+ 0
- Ch: Damp. Co. (kNs/m):**
  - X direc.: 0 E+ 0
  - Y direc.: 0 E+ 0

**Rocking Spring:**

- Kr: Stiffness (kNm/rad):**
  - X direc.: 0 E+ 0
  - Y direc.: 0 E+ 0
- Cr: Damp. Co. (kNms/rad):**
  - X direc.: 0 E+ 0
  - Y direc.: 0 E+ 0

At the bottom, there are four buttons: "Import", "Export", "Basement Weight", and "OK".



- Please input Sway spring properties  $K_h, C_h$  and Rocking spring properties  $K_r, C_r$  in each direction.
- By [Basement Weight] button, please input the weight and rotational inertia weight of the basement.
- You can export member data to the text file "Data\_ground\_direct.txt" by [Export] button.
- You can import member data from a text file by [Import] button.
- Please click [OK] to finish.

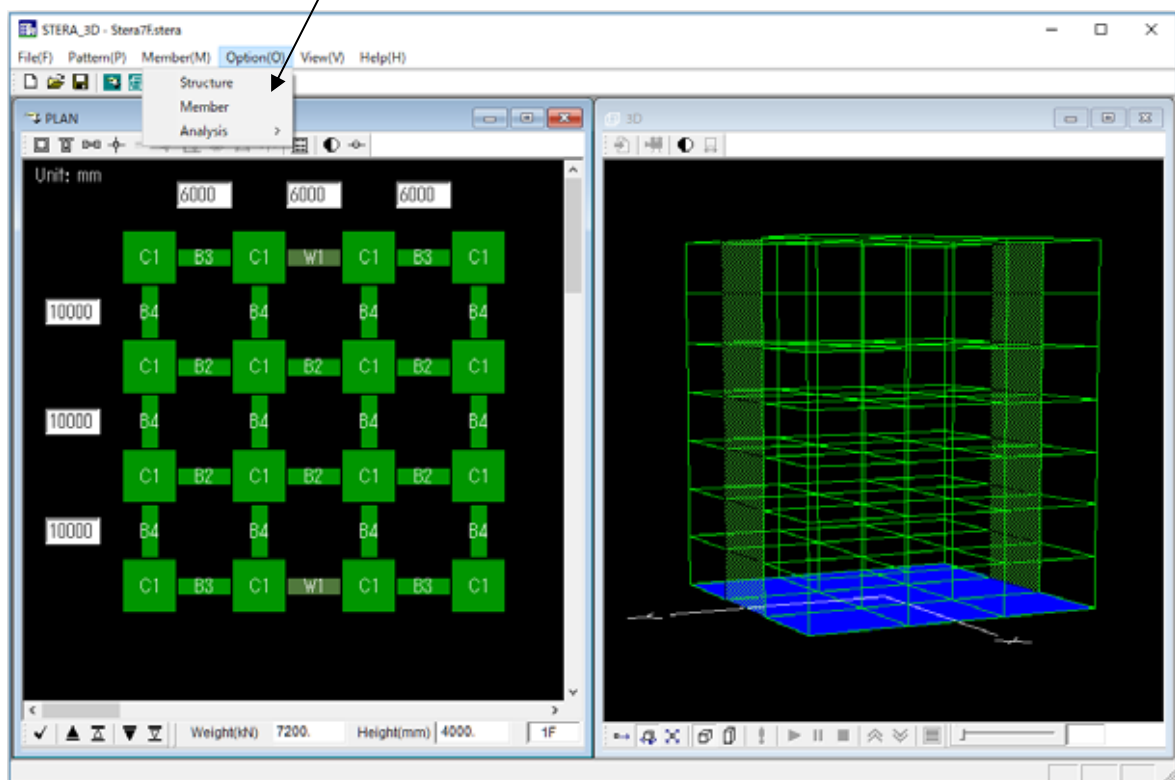
## 7 Initial Setting of Analysis Condition

### 7.1 Restrained freedom, Rigid floor, P-Delta effect, Mass distribution

In the default condition,

- The number of freedom at each node is eight including three horizontal deformations, three rotational deformations and two shear deformations.
- The P-Delta effect for column and wall elements is not considered.
- The mass of floor is distributed in proportion to influence area

To change the default condition, please select “Option” in the main menu and select “Structure” from the pull down menu.



Option → Structure

**[1] Freedom**

Restrained freedom number:

Example:

- 2467 ... X-direction only
- 1568 ... Y-direction only
- 45678 ... no rotation freedom
- 78 ... rigid connection

1(Ux), 2(Uy), 3(Uz) : lateral freedom  
 4(Rx), 5(Ry), 6(Rz) : rotation freedom  
 7(Gx), 8(Gy) : shear rotation freedom

**[2] P-Delta Effect**

☒ Not considered ☐ Considered

**[3] Mass Distribution**

☐ Same at all nodes  
☒ In proportion to influence area  
☐ Independent at each node

Import

OK

**[1] Restrained freedom number**

Please indicate the freedom numbers to be restrained.

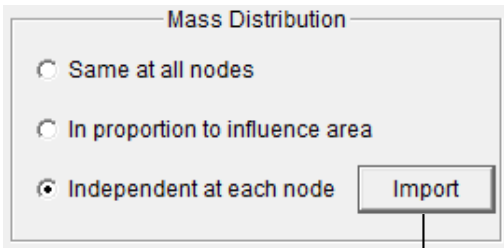
**[2] P-Delta Effect**

Considered → P-Delta effect is considered in element stiffness matrix of column and wall.

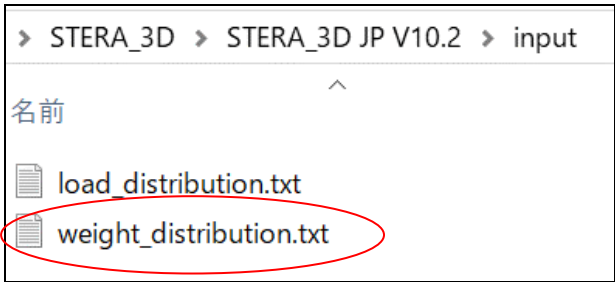
**[3] Mass distribution at nodes in a floor**

Please select from:

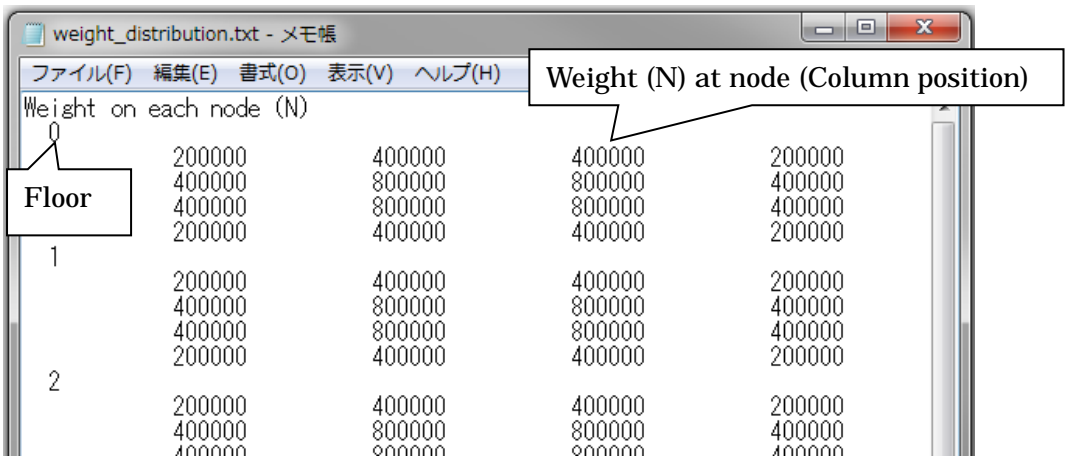
- Same at all nodes
- In proportion to influence area (default)
- Independent at each node



- Independent at each node  
By clicking “Import” button, please select the file of the weight distribution.



- After the initial analysis (see Chapter 8.1), the file “weight\_distribution.txt” is automatically created in the “input” folder. When you want to set different weight at each node, please modify this file and rename it.



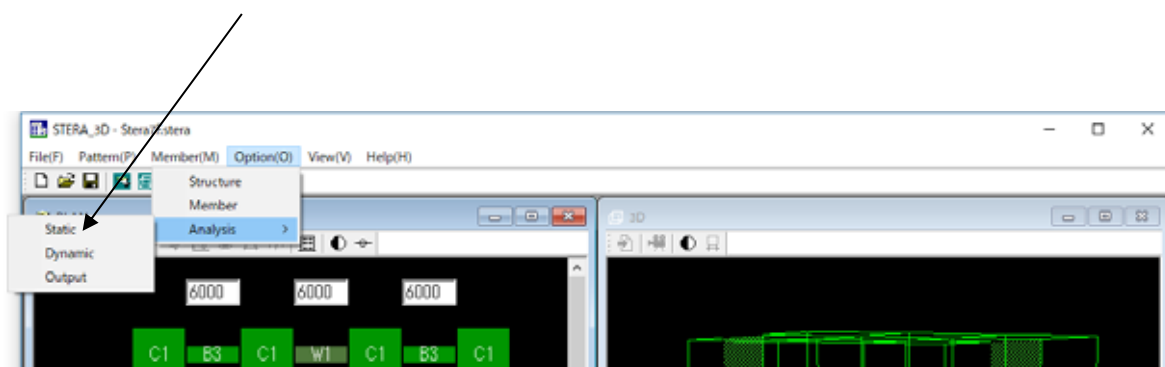


## 7.2 Condition of static analysis

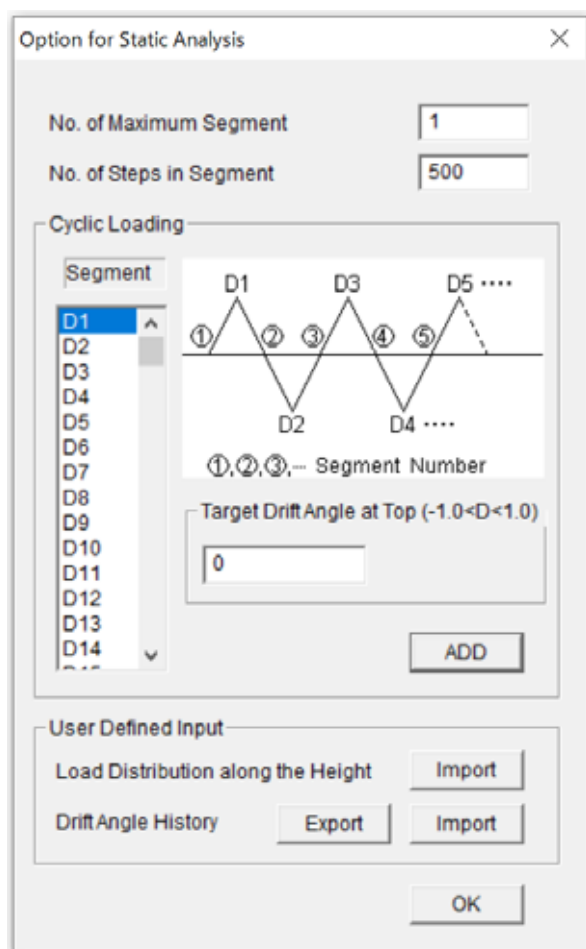
In the default condition,

- Static push over analysis will be done with 500 steps until the target drift (or force).

To change the default condition, please select “Option” in the main menu and select “Analysis” and “Static” from the pull down menu.



Option → Analysis → Static



- Cyclic loading is possible by setting the target drift angle at the top of a building in each loading segment.

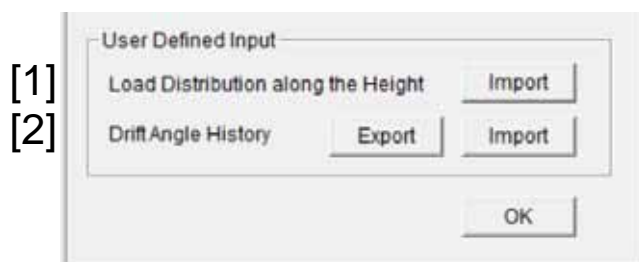
No. of Maximum Segment

The total number of segments in cyclic loading,

No. of Steps in Segment

Number of calculation steps in each segment for static analysis to increase the accuracy of nonlinear analysis,

- Loading program is defined by the target drift angle, D1, D2 ... D150, at the top of a building in each loading segment. If a negative value is entered, the force is applied in the opposite direction.
- To move to the next drift angle, please click [ADD] button.



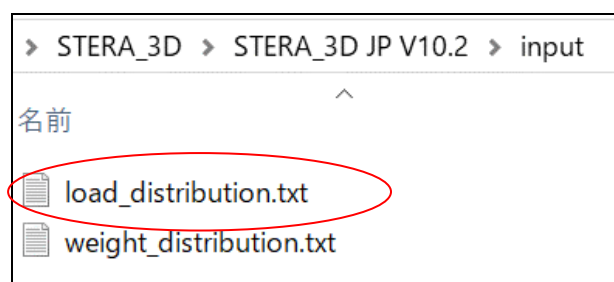
### [1] Load Distribution along the Height

In “8.3 Nonlinear static push-over analysis”, you can select the load distribution along the height from

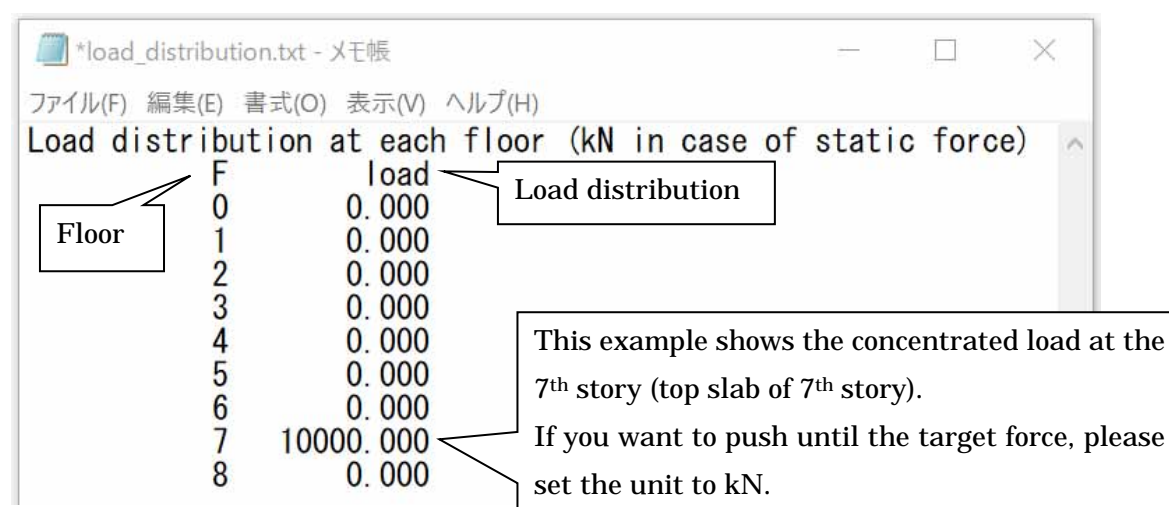
1. Ai 2. Triangular 3. Uniform 4. UBC 5. ASCE 6. Mode 7. User defined

If you want to use [7. User defined] distribution, please create the load distribution file as follows.

Firstly, in the initial analysis (see 8.1), a file of the horizontal load distribution is automatically created as "load\_distribution.txt" in the “input” folder.



Please modify this file and specify the load distribution (ratio) for each story and rename the file.



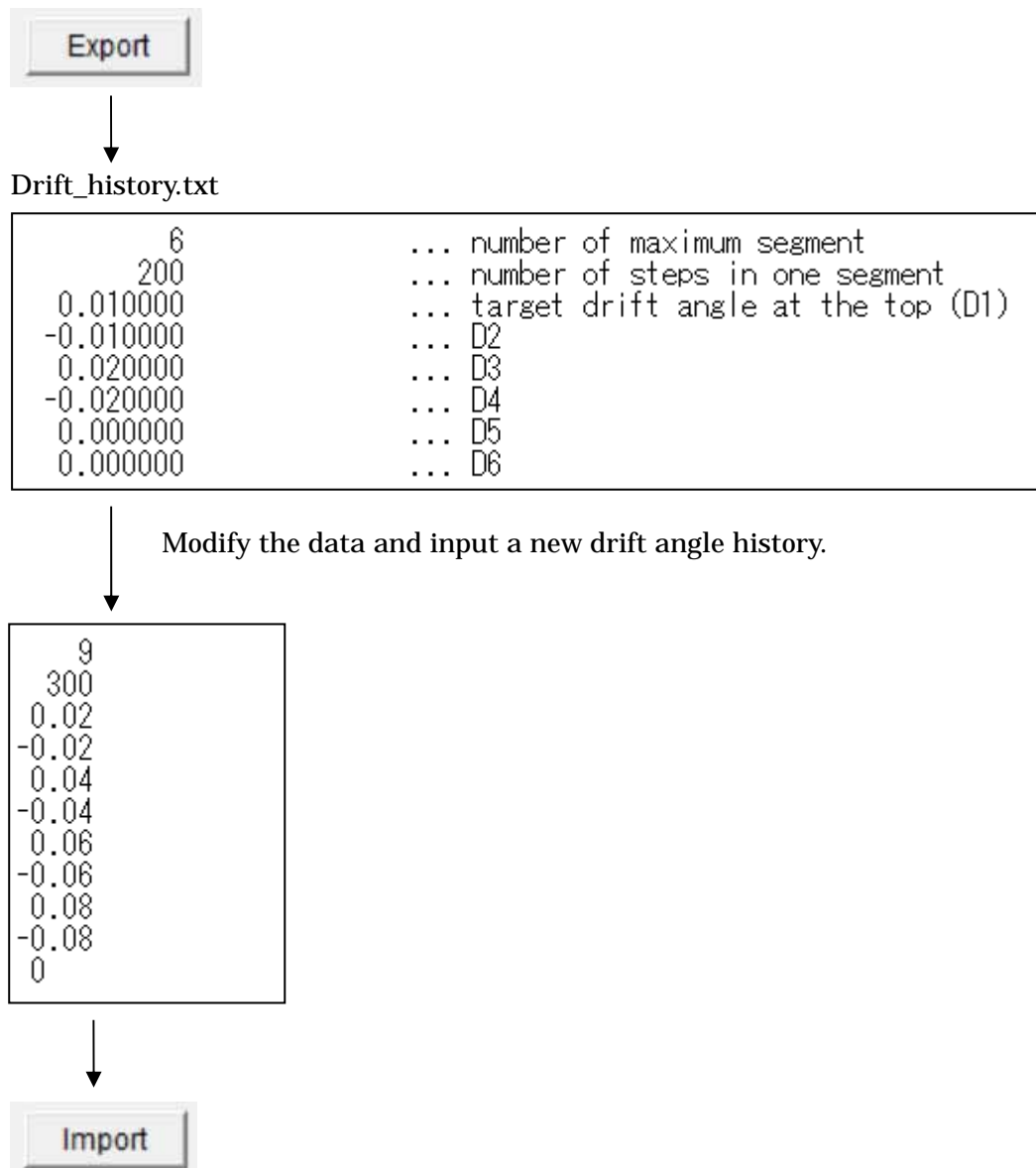
Then, by clicking  select this file.

## [2] Drift Angle History

You can export the drift angle history to the text file “Drift\_history.txt” by [Export] button.

You can import the drift angle history from a text file by [Import] button.

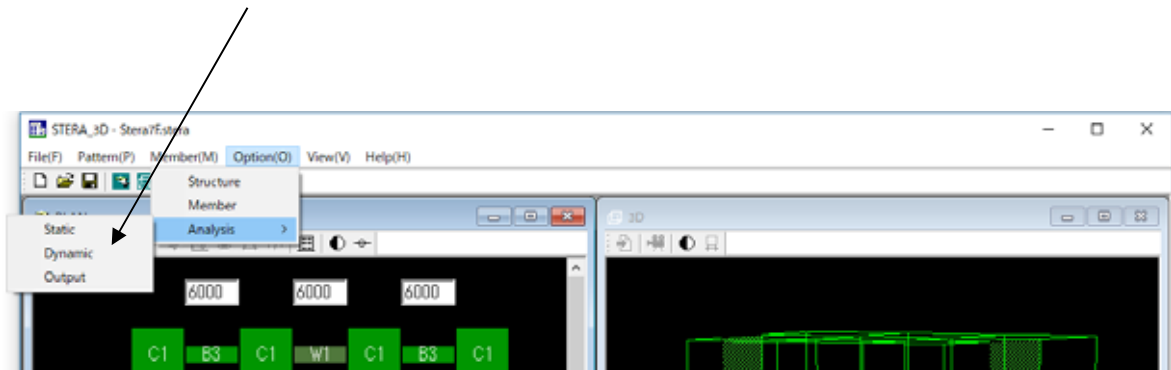
To make the original drift angle history, first, please export an arbitrary history into “Drift\_history.txt”. Then, please modify this file and input the original drift angle history in the same format.



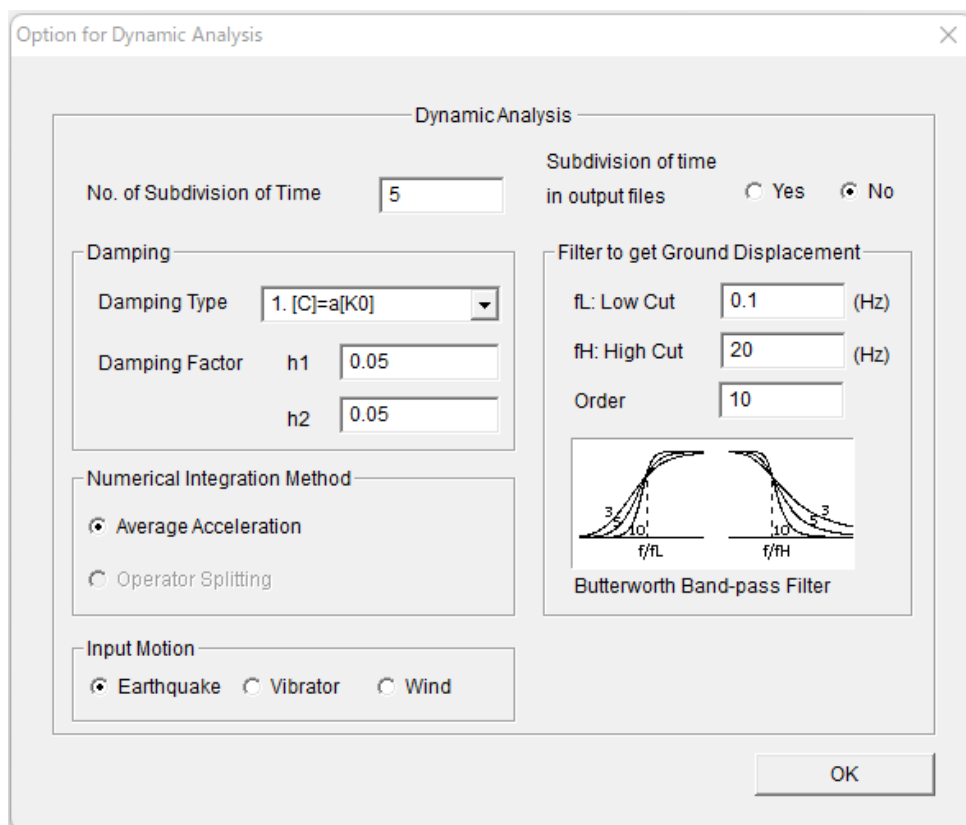
Import the modified file.

### 7.3 Condition of dynamic analysis

To change the default condition, please select “Option” in the main menu and select “Analysis” and “Dynamic” from the pull down menu.



Option → Analysis → Dynamic



- No. of Subdivision of Time

Separating the original time interval of input earthquake into a smaller time interval will increase accuracy and stability in numerical integration, however, it also increases calculation time.

- The maximum data size input earthquake is 60,000.

- Subdivision of time in output files

No: the time increments of the response output are the same as the time interval of the seismic wave data and are not subdivided.

Yes: the time increments of the response output are also subdivided. If the number of subdivisions is large, the output file size will be large.

- Damping

Three types of damping matrix are available:

$[C] = a[K_0]$  : proportional to  $[K_0]$

$[C] = a[K_p]$  : proportional to  $[K_p]$

$[C] = a[K_0] + b[M]$  : Rayleigh damping

The first mode damping factor,  $h_1$ , is used for type 1) and 2). The second mode damping factor,  $h_2$ , is used for type 3).

- Numerical Integration Method

You can select the method from the “Average Acceleration Method” and the “Operator Splitting Method”.

- Input Motion

- You can select input motion from “Earthquake” ground acceleration, “Vibrator” on a floor and “Wind” pressure.

- Filter to get Ground Displacement

You can set the parameters of Butterworth band-pass filter to get the ground displacement. The default values are:

Low cut filter frequency: 0.1Hz

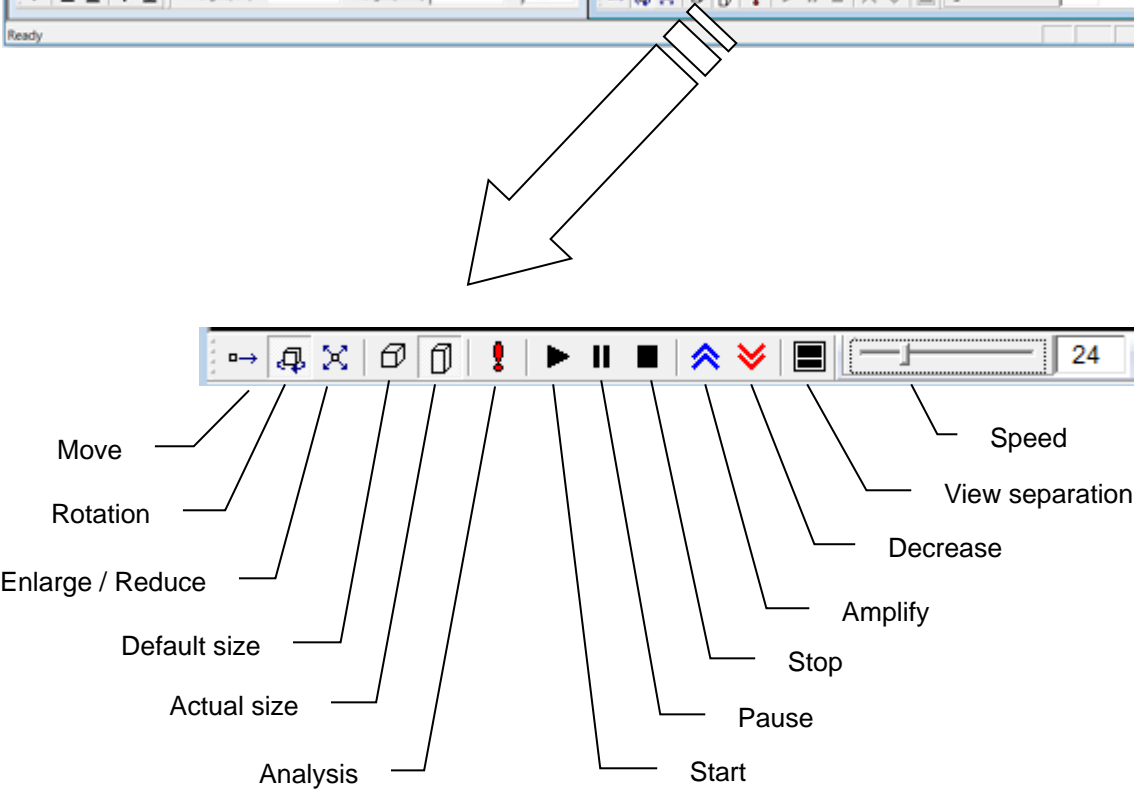
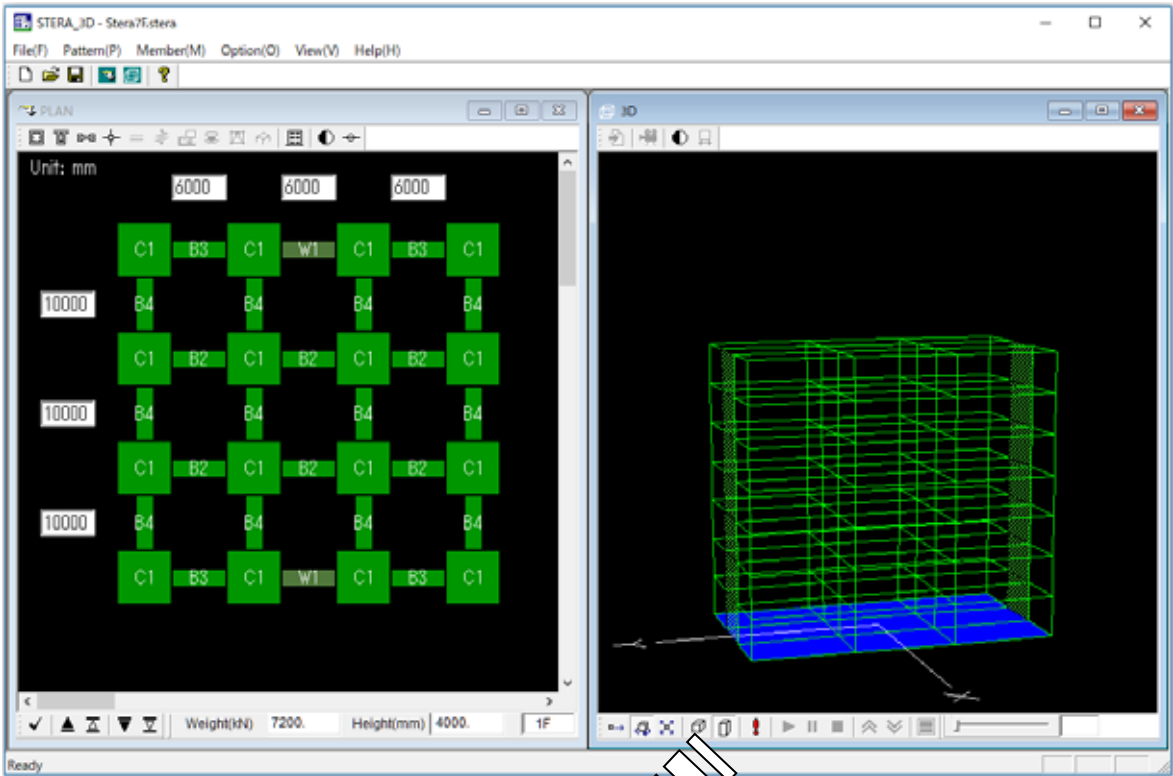
High cut filter frequency: 20Hz




Order of filtering: 10

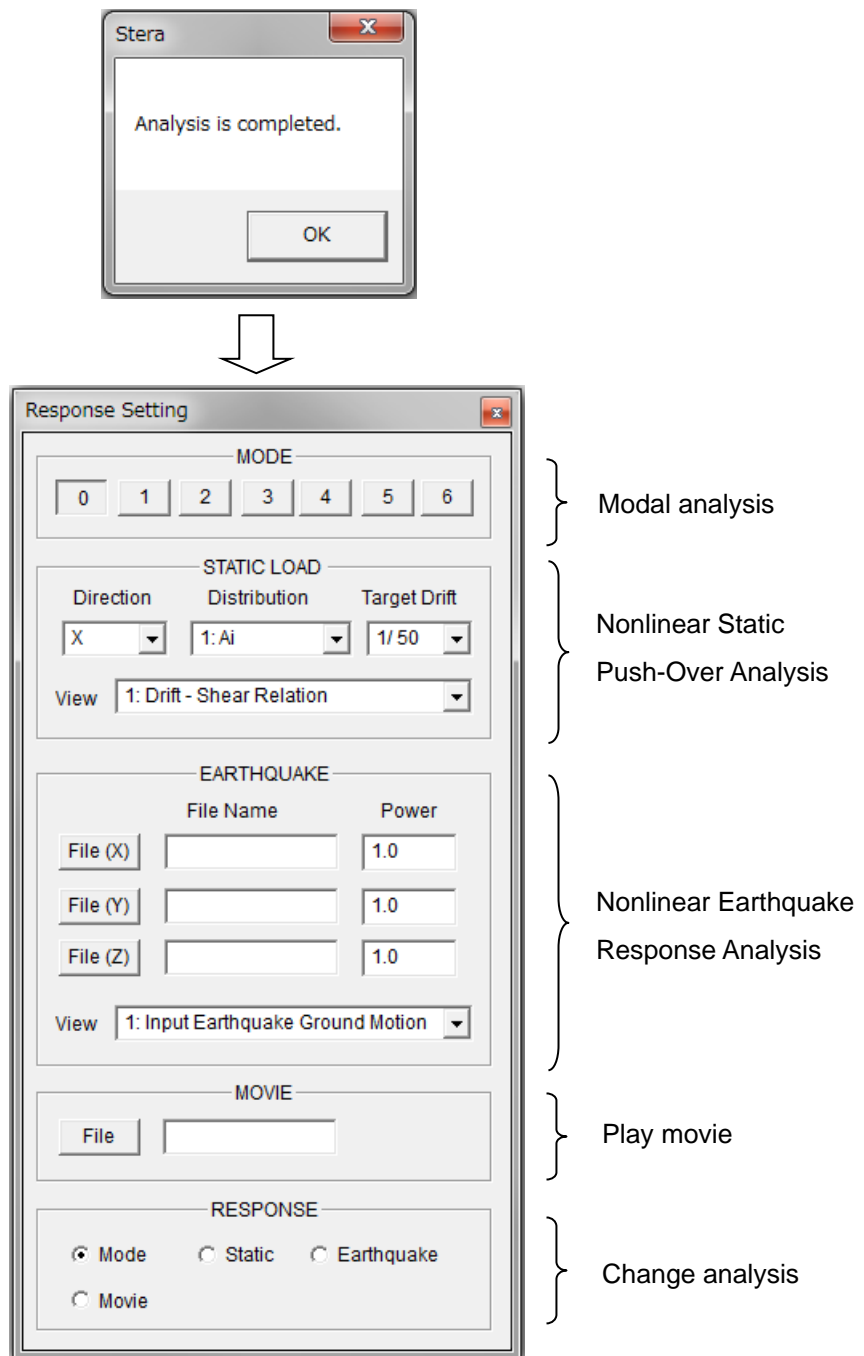
Please check “Technical Manual” for the detail.

8 3D View of Building and Response

8.1 3D View of building

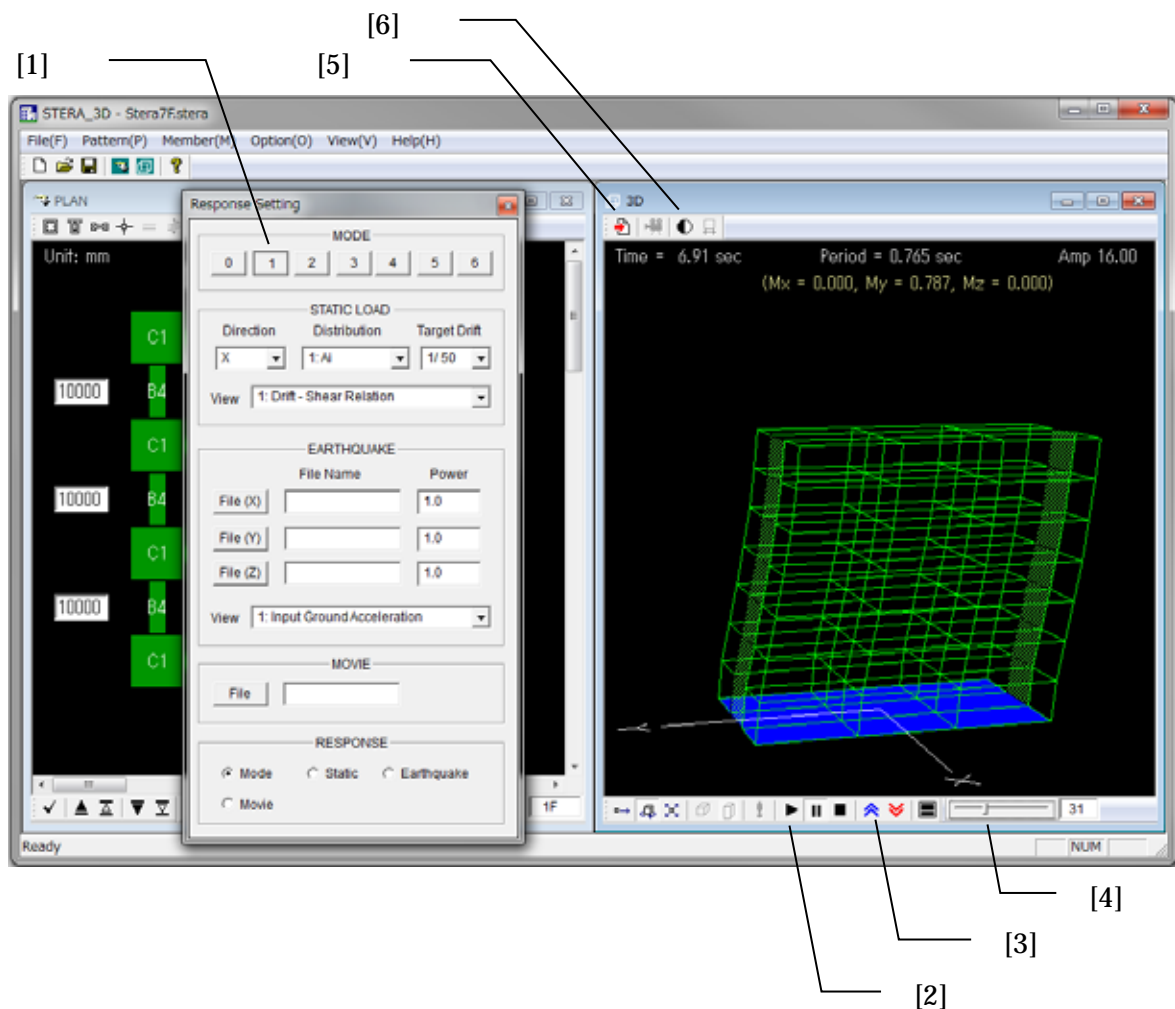


- [1] [Default] () set the default size. [Actual] () use the actual size based on input data.
- [2] If the [Analyze] () is activated, by clicking the button, you can make an initial analysis for getting natural periods and mode shapes.
- If the analysis is successfully done, the following message will appear on the screen.
- By clicking [OK] button, RESPONSE SETTING DIALOG will also appear.



## 8.2 Modal analysis

- [1] On the RESPONSE SETTING DIALOG, please click the MODE number from [0] to [6] to see the view of mode shape and the value of natural period (Period) and effective modal mass ratio ( $M_x$ ,  $M_y$ ,  $M_z$ ).
- [2] On the 3D VIEW, (▶) starts the vibration of each mode, (■) stops the vibration and (⏸) pauses the vibration.
- [3] (⬆) amplifies the response (⬇) reduces the response.
- [4] Slider (33) changes the speed of vibration.
- [5] (📄) will save the results into text files.
- [6] (◐) changes the color of the view to be black and white.






### 8.3 Nonlinear static push-over analysis

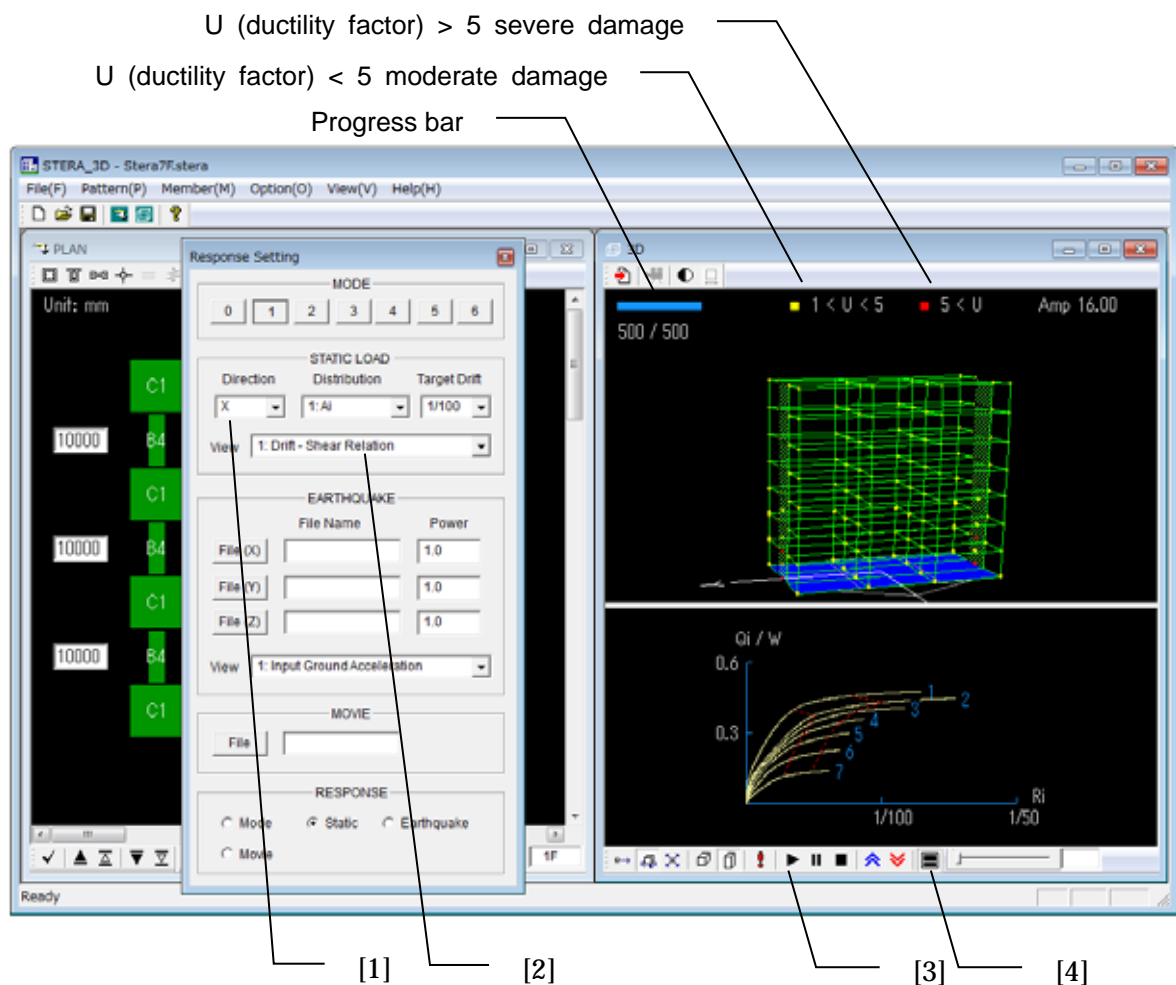
[1] Please set loading conditions for the STATIC LOAD:

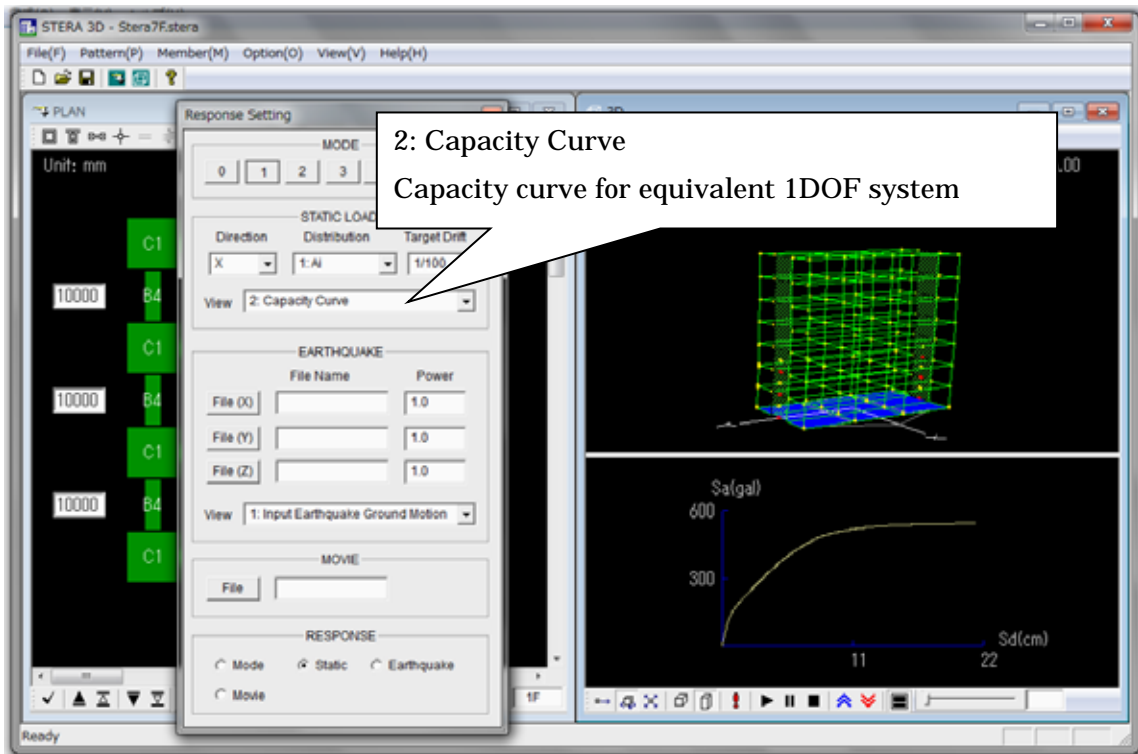
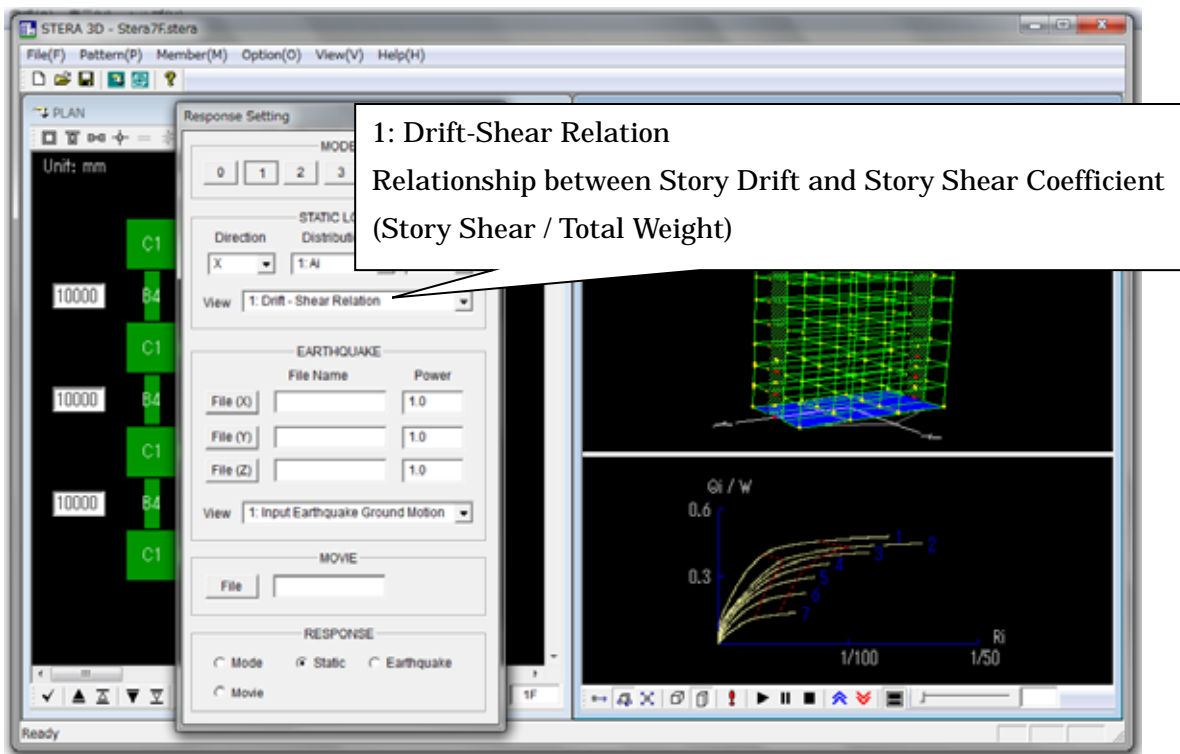
- “Direction”: please select loading direction from the menu.  
 1. X 2. -X (opposite to X) 3. Y 4. -Y (opposite to Y)
- “Distribution”: please select a loading distribution along the height of the building. The load is applied at the center of gravity in each floor.  
 1. Ai 2. Triangular 3. Uniform 4. UBC 5. ASCE  
 6. Mode 7. User defined
- “Target Drift”: please set the target drift ratio which is defined by the ratio between the top displacement and the height of the building.  
 “Cyclic” is the cyclic loading and “Force” is the target force as described in 7.2.  
 1. 1/50 2. 1/100 3. 1/200 4. Cyclic 5. Force

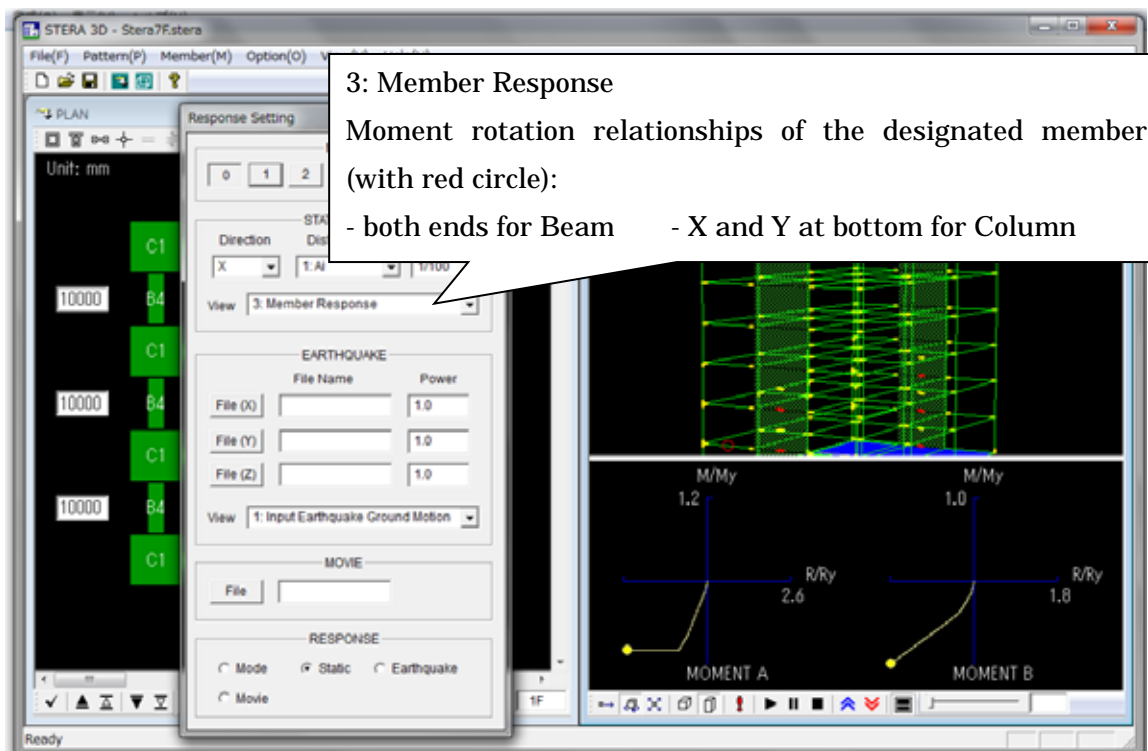
[2] Please select the response for the lower view window.

[3] On the 3D VIEW, (  ) starts, (  ) pauses and (  ) stops the response.

[4] (  ) will change the view from 2-screens to 1-screen and vice versa.







## 8.4 Nonlinear earthquake response analysis

(in case “Earthquake” is selected in the Option menu of dynamic analysis)


[1] On the RESPONSE SETTING DIALOG, please set earthquake data:


- “File(X)”: Please select earthquake input file for X-direction.
  - “File(Y)”: Please select earthquake input file for Y-direction.
  - “File(Z)”: Please select earthquake input file for Z-direction (up-down).
  - “Power”: Set the value to amplify the original earthquake
- The format of the input file is described in Section 9.1.


[2] Please select the response for the lower view window.

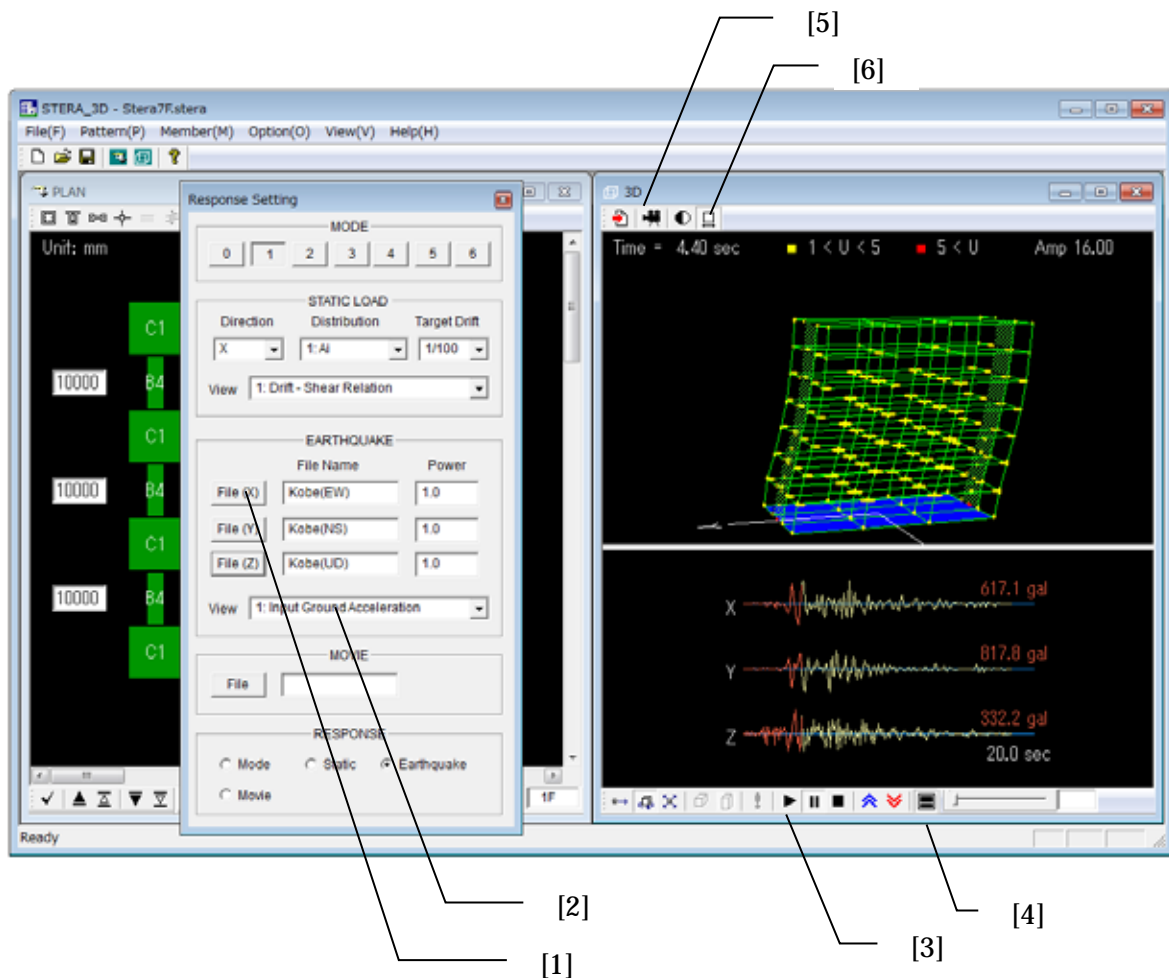
[3] On the 3D VIEW, (  ) starts, (  ) pauses and (  ) stops the response.

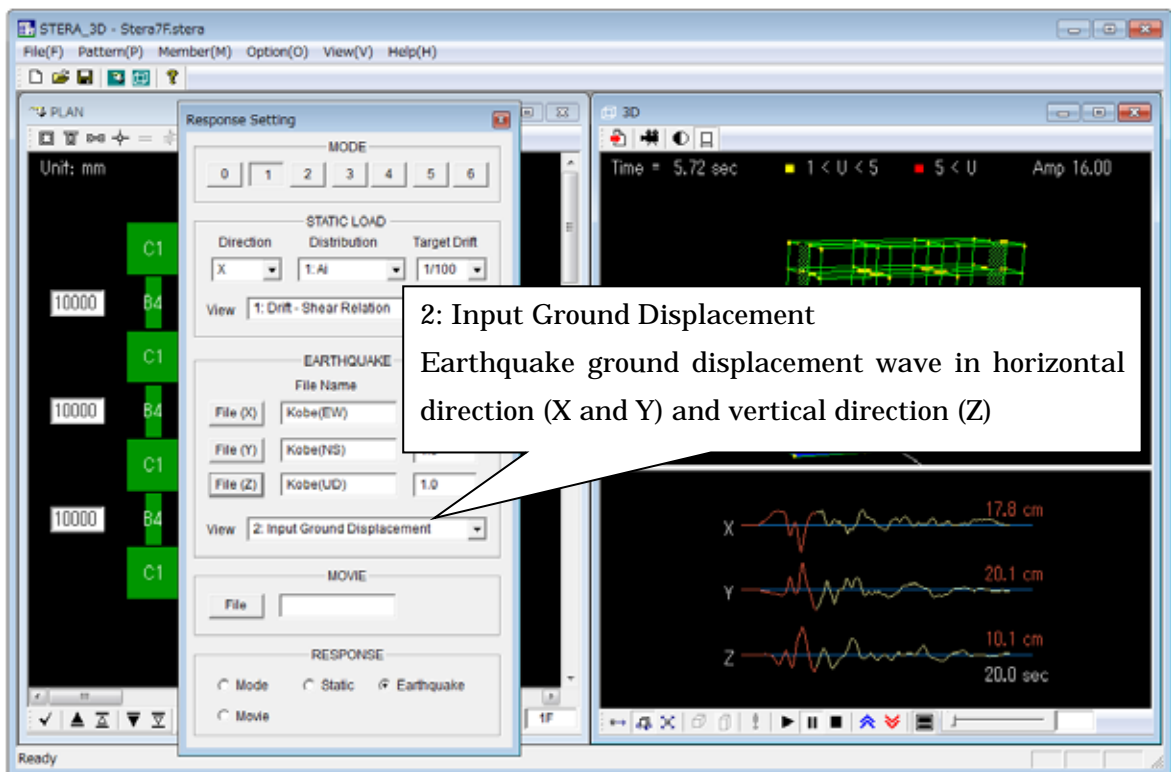
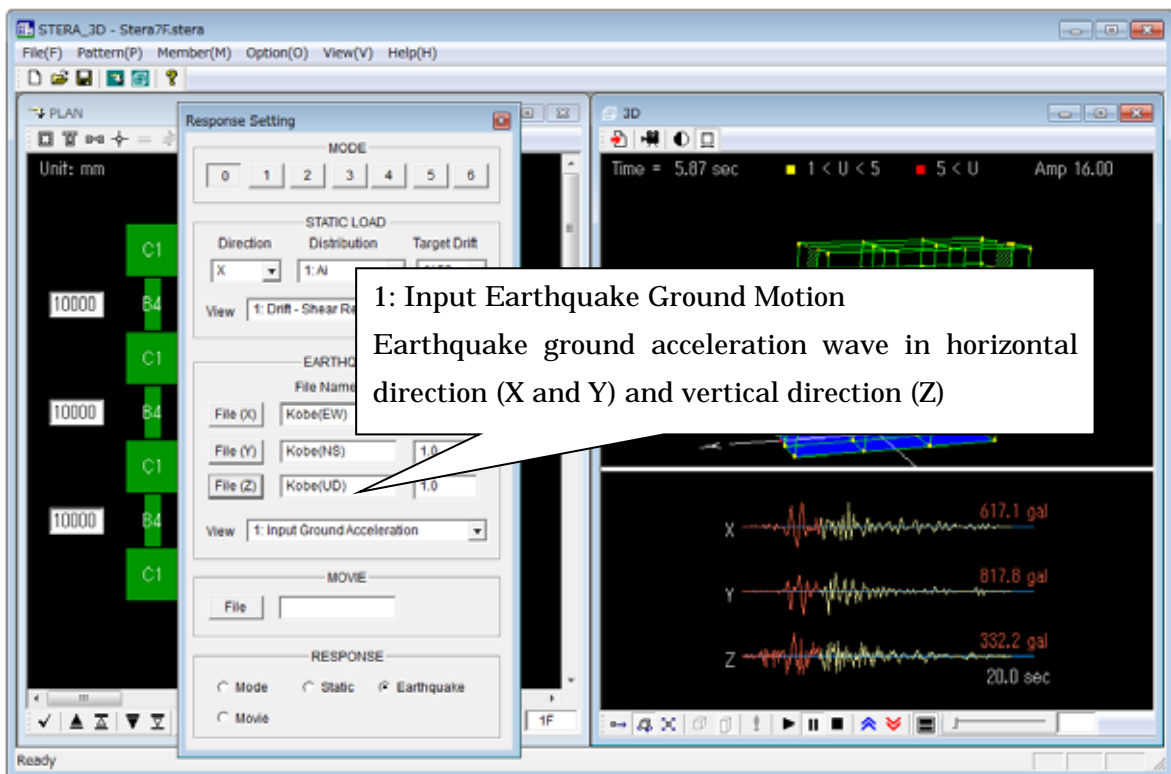
In the lower view, you can see the input earthquake wave and present status.

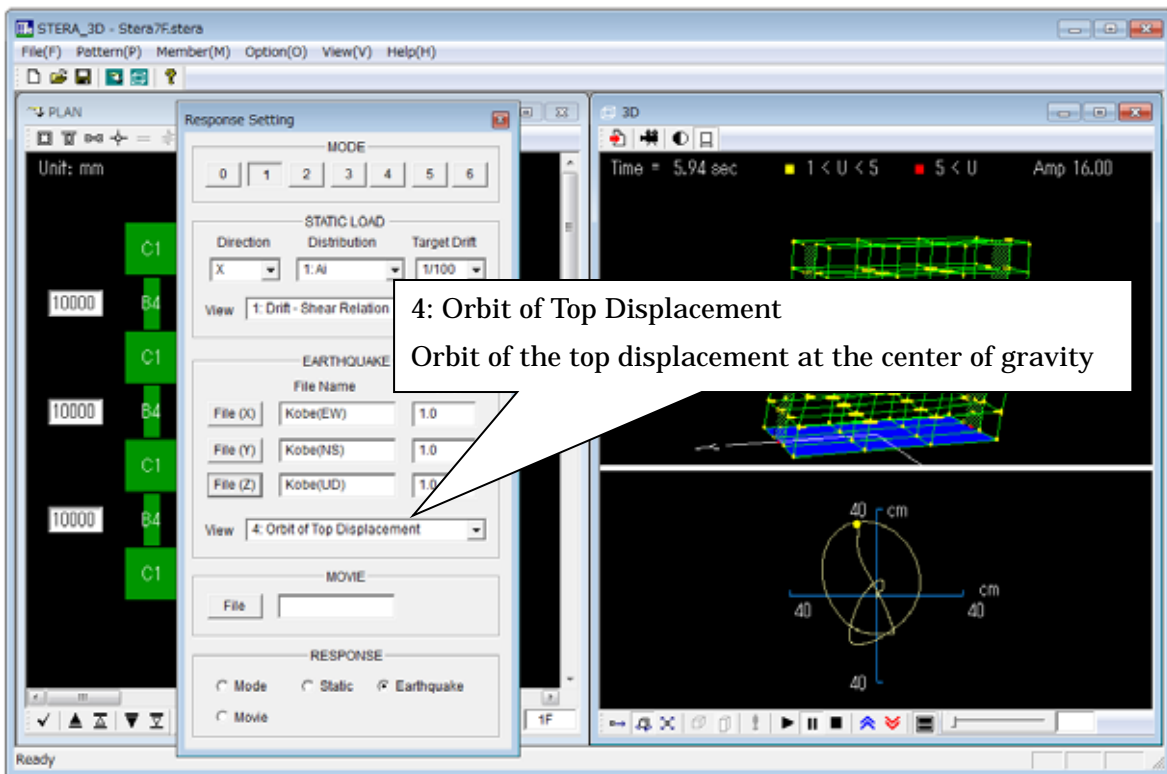
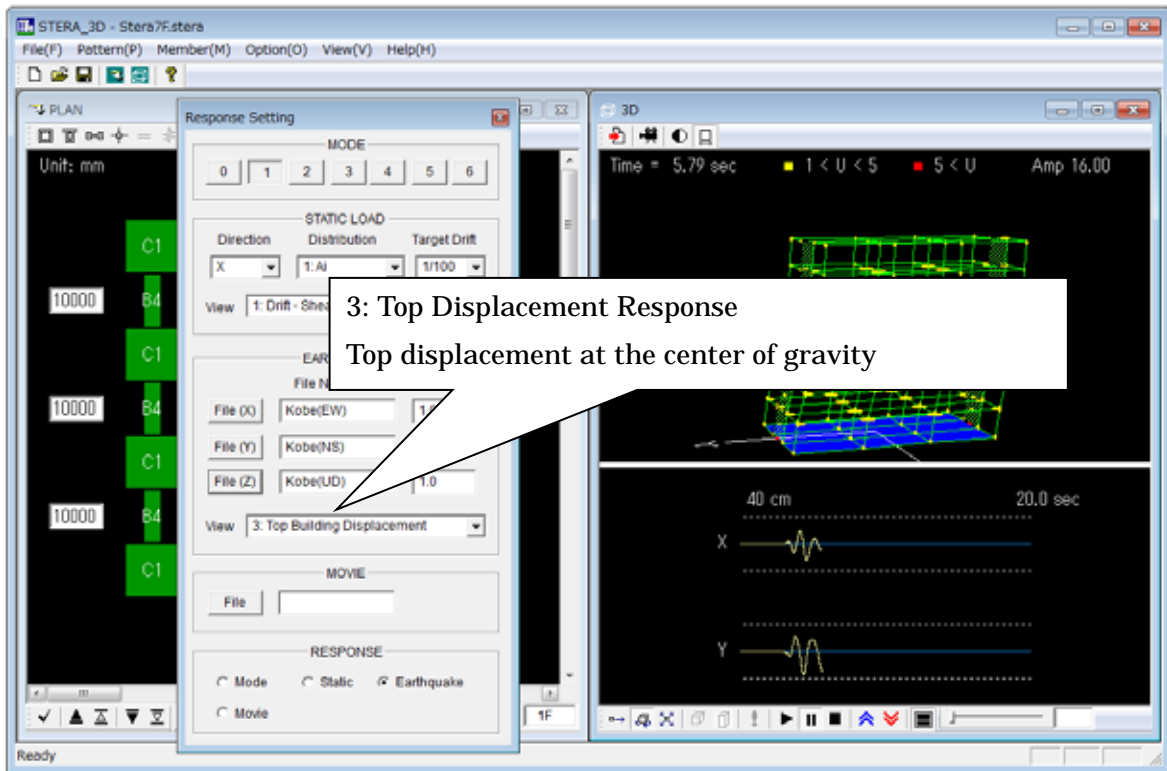
[4] (  ) will change the view from 2-screen to 1-screen and vice versa.

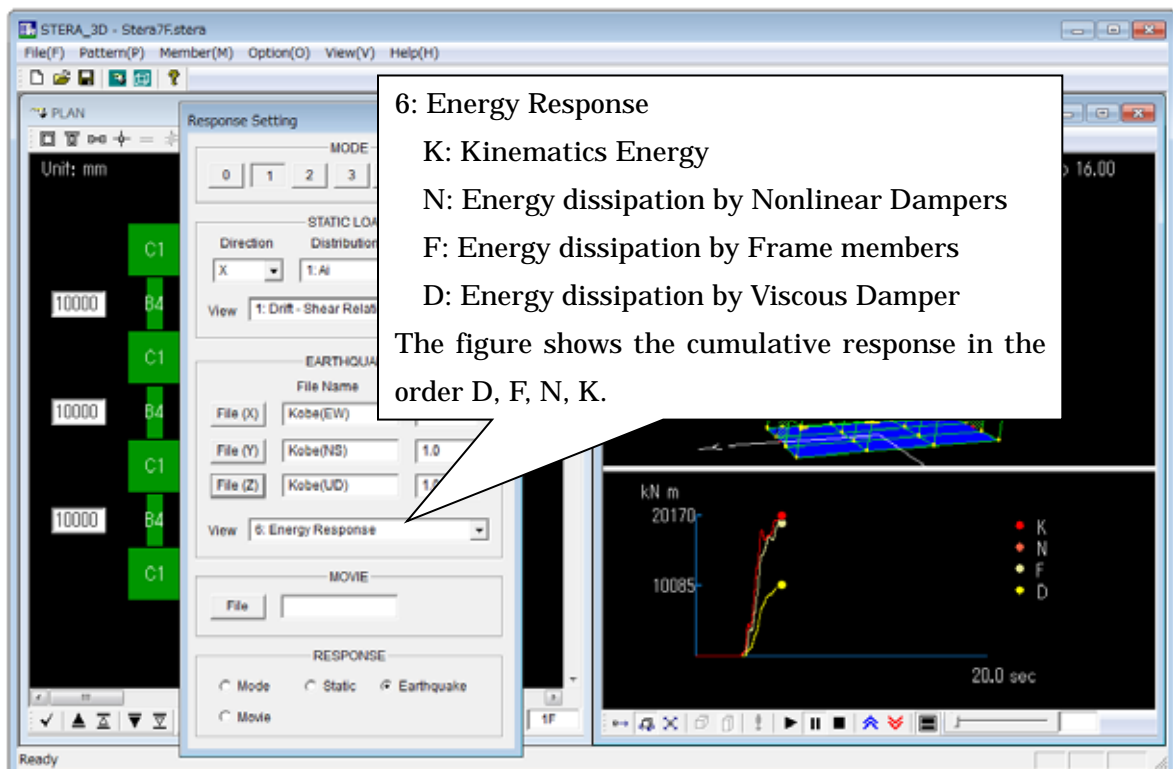
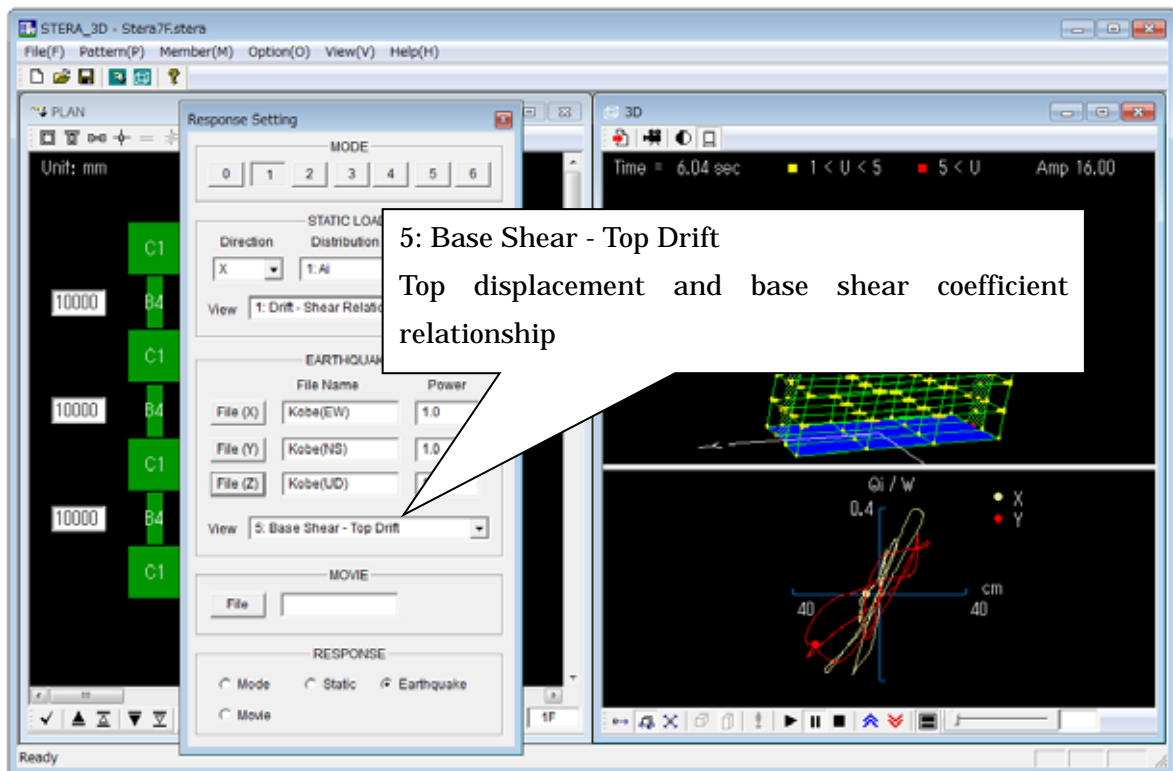
[5] (  ) will save the response animation as a movie file (see 5-5).

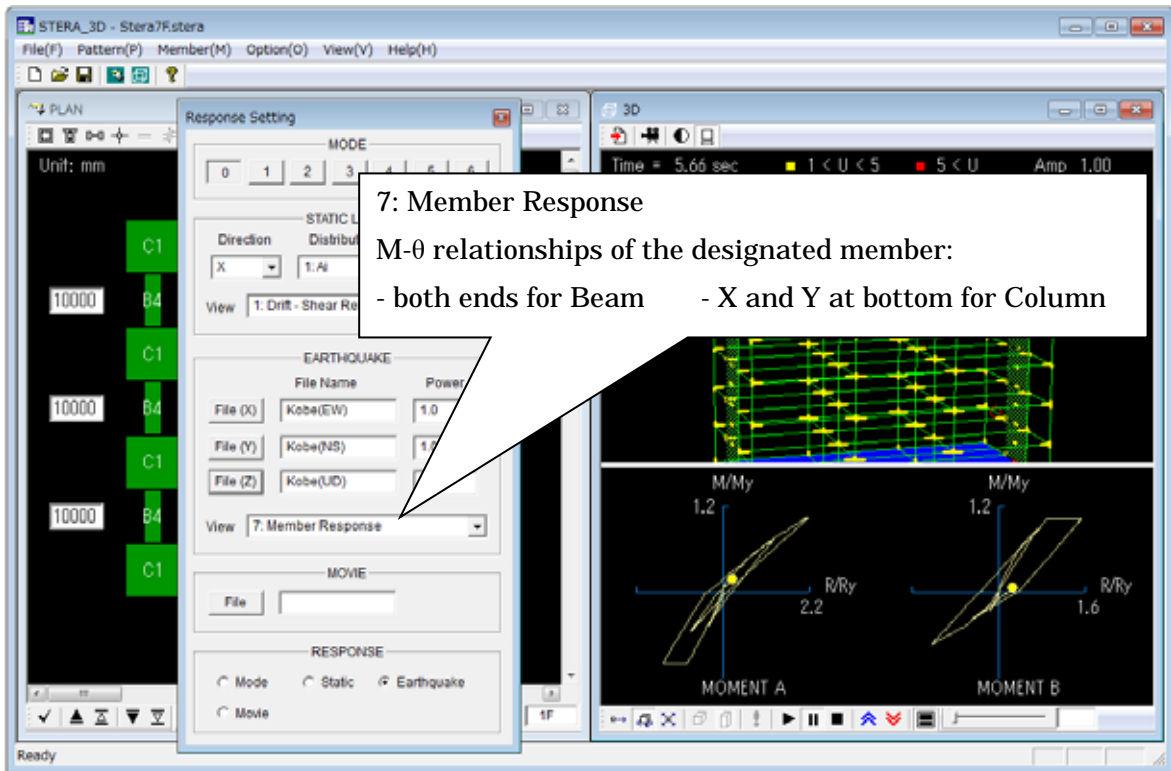
[6] (  ) will switch on and off to include ground movement.













## 8.5 Nonlinear vibrator response analysis

(in case “Vibrator on the Floor” is selected in the Option menu of dynamic analysis)

Response Setting

MODE

0 1 2 3 4 5 6

STATIC LOAD

Direction: X Distribution: 1: AI Target Drift: 1/50

View: 1: Drift - Shear Relation

VIBRATOR

Floor No.: 1F Direction: X

Amp (kN): 10.00 T(sec): 1.000

☐ Sine ☐ Random

File

View: 1: Input Vibrator Force

MOVIE

File

- Floor number and the direction of movement (X or Y) to set the vibrator.
- Loading force (kN) can be selected from “Sine” wave and “Random” wave.
- In case of “Sine” wave, you input amplitude and natural period.
- In case of “Random” wave. You select an input file from the dialog window.

VIBRATOR

Floor No.: 1F Direction: X

Amp (kN): 10.00 T(sec): 1.000

☐ Sine ☐ Random

File

View: 1: Input Vibrator Force

1: Input Vibrator Force

2: Top Building Acceleration

3: Top Building Displacement

4: Orbit of Top Displacement

5: Base Shear - Top Drift

6: Energy Response

7: Member Response

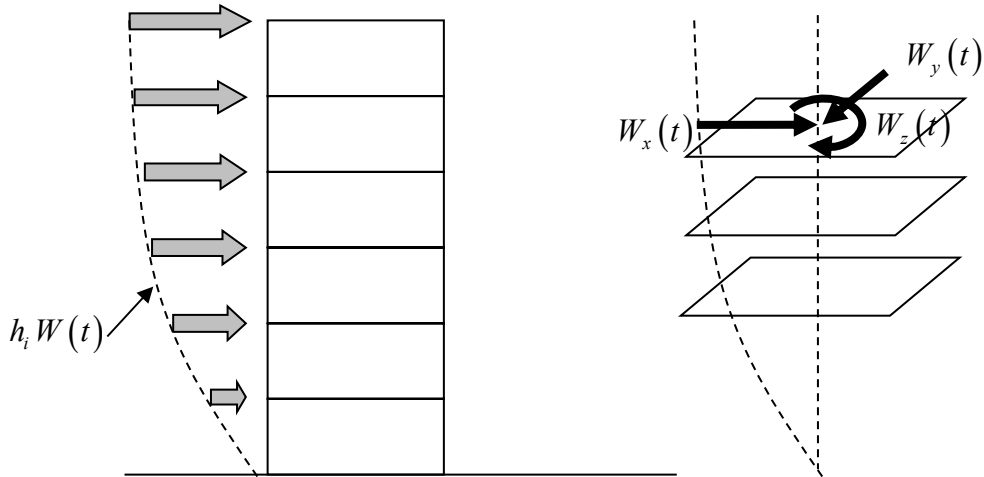
1. Input Vibrator Force
2. Top Building Acceleration  
Top acceleration at the center of gravity
3. Top Building Acceleration  
Top displacement at the center of gravity

The following menu is the same as Earthquake analysis.

## 8.6 Wind response analysis

(in case “Wind” is selected in the Option menu of dynamic analysis)

The dynamic wind force is assumed to be applied at the center of gravity at each floor with the constant distribution along the height of the building.



Response Setting

MODE

0 1 2 3 4 5 6

STATIC LOAD

Direction Distribution Target Drift

X 1: A1 1/50

View 1: Drift - Shear Relation

WIND

Wind force	Distribution	Power
Wx (kN)	Dist x	1.0
Wy (kN)	Dist y	1.0
Wz (kNm)	Dist z	1.0

View 1: Input Wind Force

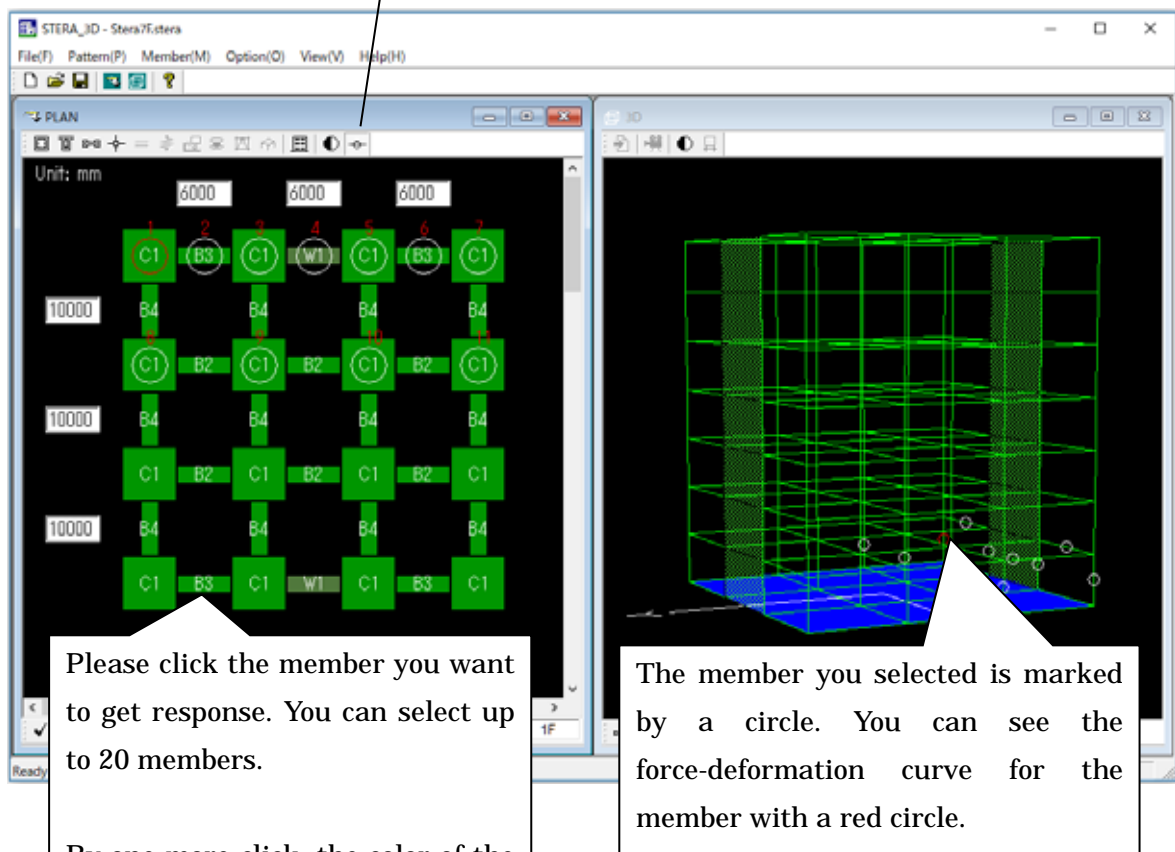
- Wx (kN): Please select an input file for wind lateral force in x-direction.
- Wy (kN): Please select an input file for wind lateral force in y-direction.
- Wz(kNm): Please select an input file for wind torque force in z-direction.
- The format of the input files of wind forces Wx, Wy and Wz are the same as the input earthquake acceleration data as described in Section 9.1
- Dist x, Dist y and Dist z: Please select input files for lateral distribution of the wind loads along the height.
- The format of the lateral distribution of the wind loads Dist x, Dist y and Dist z are the same as the user defined horizontal loads in static analysis as described in Section 7.2.
- Power: Set the value to amplify the original wind loads.

## 8.7 Response output member

You can obtain the response time history of the designated member.

OUTPUT MEMBER (  )

If you click this button, you can designate the output member. By one more click, you can cancel it.



By one more click, the color of the circle will change to be red and its force-deformation curve will be displayed in 3D view.


By the right click, you can cancel the selection.

## 8.8 Save Nonlinear Earthquake Response as a Movie File

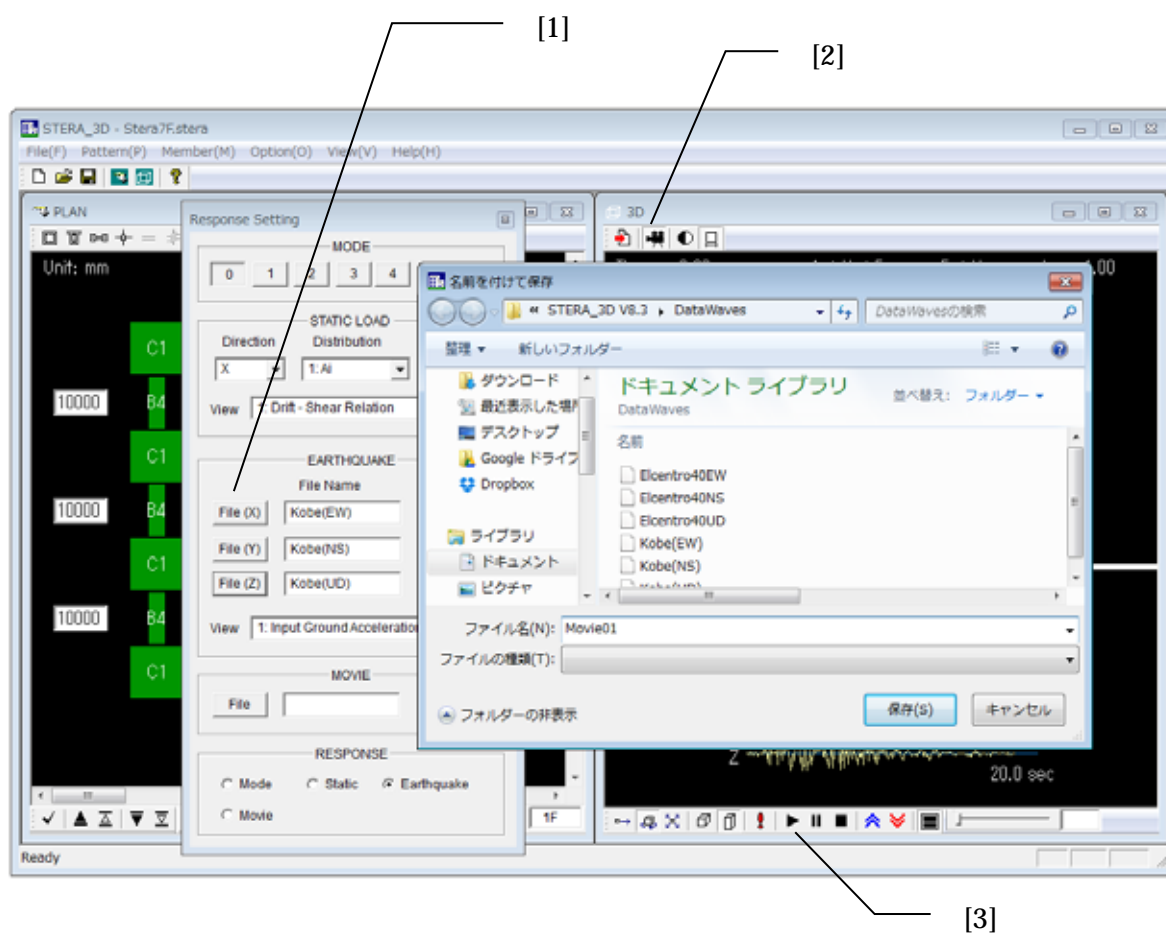
Generally, it takes long time to calculate earthquake response of a building. In this case, you can save the response of the building in a movie file and later you can play the movie to see the response quickly.

### 1) Record movie


[1] On the RESPONSE SETTING DIALOG, please select earthquake input files in the menu “EARTHQUAKE”.




[2] Please push the movie button (  ) and write the file name such as “Movie.txt”.

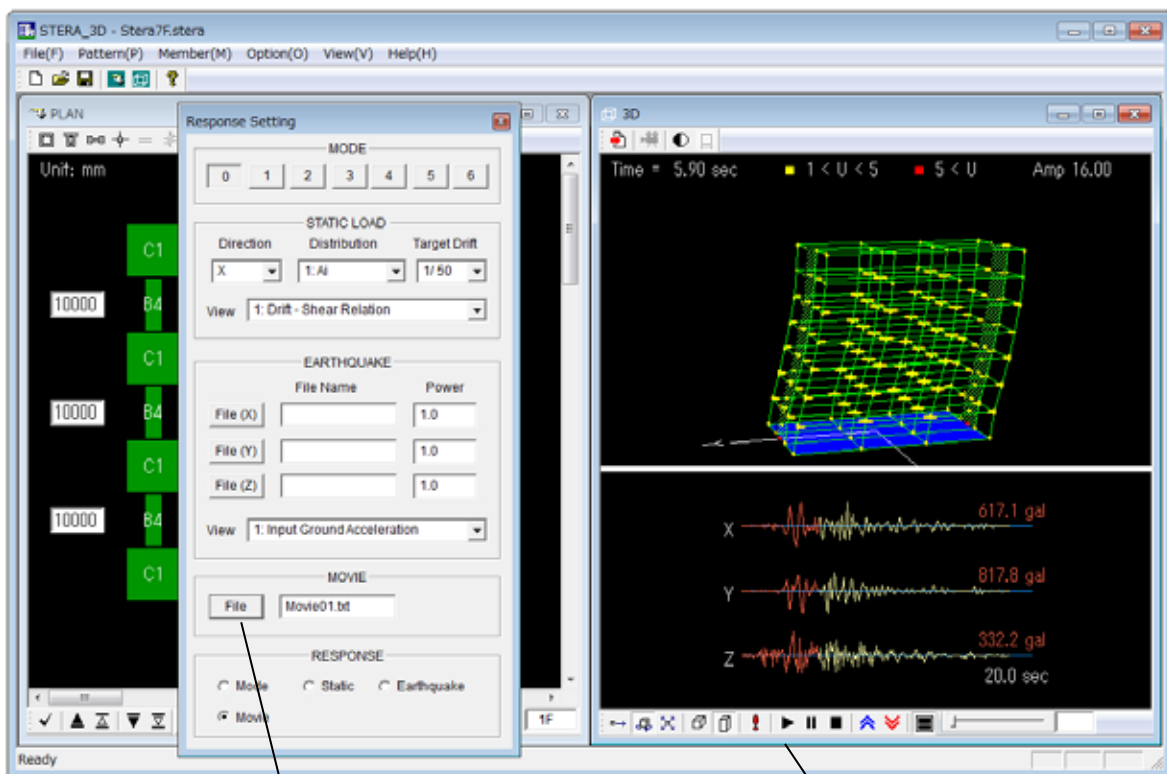
[3] Automatically the recording will start.



## 2) Play movie

[1] On the RESPONSE SETTING DIALOG, please push  in the “MOVIE” menu to select a movie file.

[2] () starts, () pauses and () stops the response.



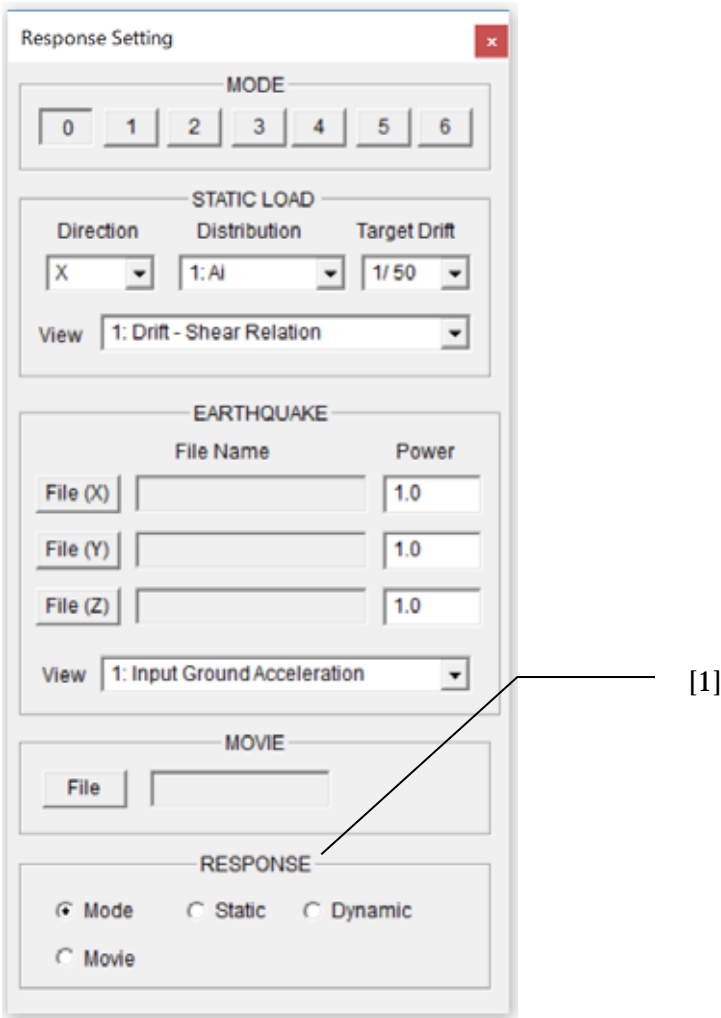
[1]

[2]

8.9 Change Analysis

[1] On the RESPONSE SETTING DIALOG, you can change the analysis:

- Mode: Modal Analysis
- Static: Nonlinear Static Push-Over Analysis
- Dynamic: Earthquake / Vibrator / Wind Response Analysis
- Movie: Movie for Nonlinear Earthquake Response Analysis



## 9 Input Earthquake Ground Motion

### 9.1 Format of input earthquake data file

When you prepare an input earthquake file by yourself, please arrange the data format as follows:

Order	Type	Information	Comments
1 <sup>st</sup> data (NDATA)	INT	Number of data	The number of data for acceleration
2 <sup>nd</sup> data (DT)	REAL	Time interval	(sec)
3 <sup>rd</sup> data and later	REAL	Acceleration (cm/sec <sup>2</sup> )	Please arrange NDATA data separated by commas or spaces.

The maximum data size of input earthquake (NDATA) is 60,000. ( NDATA < 60,000 )

The ground moves according to the ground displacement are automatically calculated from acceleration data.

Example)

This is the earthquake data “Kobe 1995\_NS.txt” in the “./sample/wave/” folder.

```

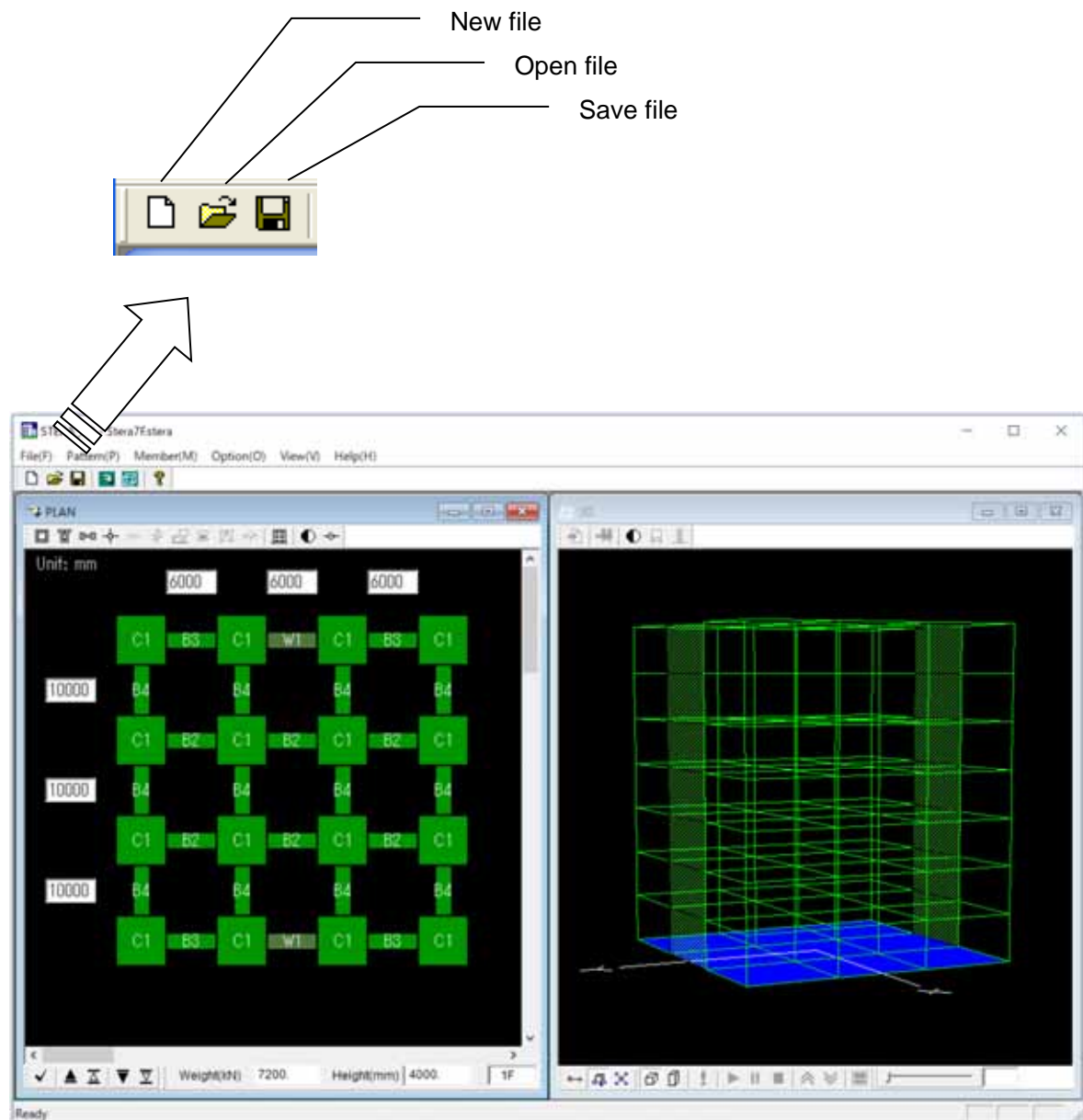
1000      ...NDATA
0.0200    ...DT (time interval, sec)
0.70      0.70      -0.30      -2.00      -2.80      -1.70      -0.30      -0.90      -0.40      3.30
3.50      -2.00      -6.30      -5.70      -3.60      -4.10      -2.50      0.20      -0.50      -4.50
-9.30      -5.70      2.50      4.70      4.50      9.20      13.70      8.20      6.60      4.00
-6.50      -11.00      0.40      14.90      2.20      -8.00      4.40      15.90      24.40      36.60
38.30      20.10      3.60      -1.80      0.00      14.80      3.40      -40.00      -49.60      -36.00
-21.90      -9.60      -0.90      0.40      -20.60      -31.30      -24.80      -14.00      3.70      11.00
-2.10      -16.70      -16.30      -12.70      0.30      -14.80      7.30      30.40      13.40      -12.00
-20.60      24.10      65.30      44.70      0.30      -14.80      7.30      30.40      13.40      -12.00
-24.00      -28.40      -14.00      -10.60      -5.40      13.50      18.30      27.90      33.00      31.50
40.00      8.60      -23.40      -38.80      -26.10      26.90      21.00      9.30      15.40      13.70
25.30      7.30      -17.30      -23.60      -20.80      -12.60      -28.50      -28.50      -15.60      -15.00

```

## 10 Save and Open Files

### 10.1 Save building data

You can save the building data in a file and open it later. The file has an extension “.stera”.






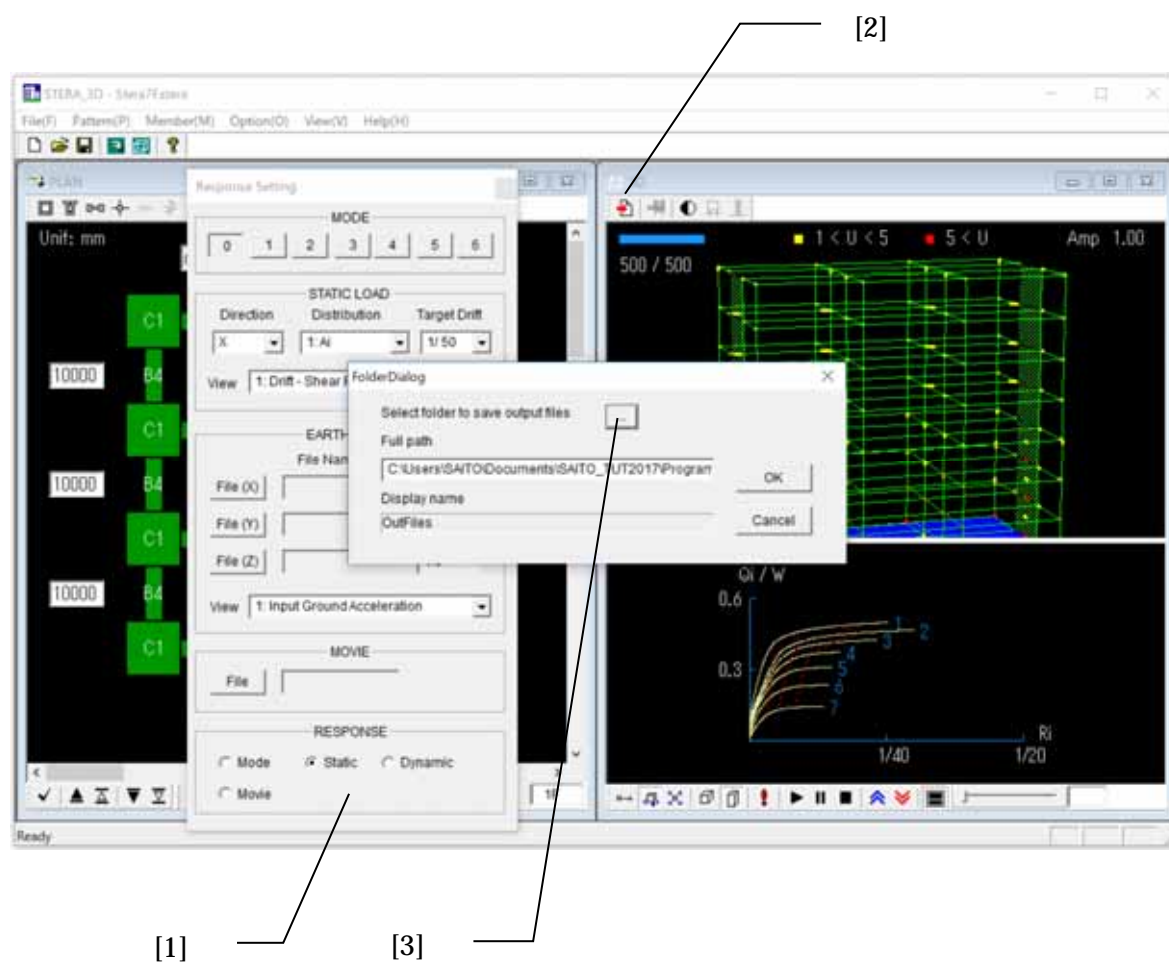
## 10.2 Save results of analysis in text files

You can save the results of modal analysis, nonlinear push-over analysis and nonlinear earthquake response analysis in the text files..

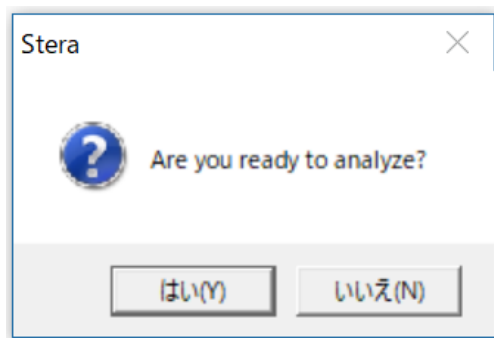
[1] On the RESPONSE SETTING DIALOG, please set the condition of analysis.

[2] Please push the “Save Data” button (  ).

[3] Please select folder to save output text files.



[4] When you push “OK”, a message window appears to start calculation and save output data to the designated folder.



If you select “Yes”, the analysis starts automatically.

```
>>>> Start initial analysis
>>>> Start elastic modal analysis
>>>> Start nonlinear dynamic analysis
      1 % finished
      2 % finished
      3 % finished
      4 % finished
      5 % finished
      6 % finished
      7 % finished
      8 % finished
```

```
     90 % finished
     91 % finished
     92 % finished
     93 % finished
     94 % finished
     95 % finished
     96 % finished
     97 % finished
     98 % finished
     99 % finished
    100 % finished
```

10.3 Output text files

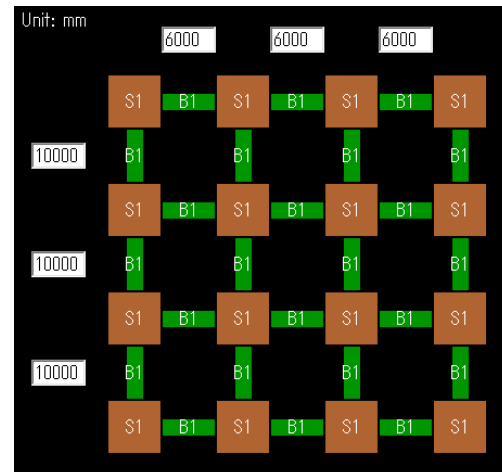
In the designated folder, the following files are automatically created:

<div><div><div></div><div>data_beam.txt</div></div><div><div></div><div>data_bi.txt</div></div><div><div></div><div>data_column.txt</div></div><div><div></div><div>data_damper.txt</div></div><div><div></div><div>data_floor.txt</div></div><div><div></div><div>data_ground.txt</div></div><div><div></div><div>data_panel.txt</div></div><div><div></div><div>data_pulley.txt</div></div><div><div></div><div>data_spring.txt</div></div><div><div></div><div>data_structure.txt</div></div><div><div></div><div>data_wall.txt</div></div></div>	<div>data_***</div> <div>Input data of members and building</div> <div>beam : Beam</div> <div>bi : Base Isolator</div> <div>column : Column</div> <div>damper : Damper and Nonstructural wall</div> <div>floor : Floor slab</div> <div>ground : Ground spring</div> <div>node : node</div> <div>panel : connection panel</div> <div>spring : External spring</div> <div>structure : Building</div> <div>wall : Wall</div>
<div><div><div></div><div>max_beam.csv</div></div><div><div></div><div>max_bi.csv</div></div><div><div></div><div>max_column.csv</div></div><div><div></div><div>max_damper.csv</div></div><div><div></div><div>max_floor.csv</div></div><div><div></div><div>max_ground.csv</div></div><div><div></div><div>max_member01.csv</div></div><div><div></div><div>max_member02.csv</div></div><div><div></div><div>max_member03.csv</div></div><div><div></div><div>max_node.csv</div></div><div><div></div><div>max_panel.csv</div></div><div><div></div><div>max_pulley.csv</div></div><div><div></div><div>max_spring.csv</div></div><div><div></div><div>max_structure.csv</div></div><div><div></div><div>max_wall.csv</div></div><div><div></div><div>mode_node.csv</div></div></div>	<div>max_***</div> <div>Maximum response of members and building</div> <div></div> <div>mode_node</div> <div>Mode participation vector of each node</div>
<div><div><div></div><div>response_eigen.csv</div></div><div><div></div><div>response_energy.csv</div></div><div><div></div><div>response_member01.csv</div></div><div><div></div><div>response_member02.csv</div></div><div><div></div><div>response_member03.csv</div></div><div><div></div><div>response_structure.csv</div></div></div>	<div>response_eigen</div> <div>Natural period and mode</div> <div>response_energy</div> <div>Energy response</div> <div>response_floor01, 02, ...</div> <div>Response of 3D rigid floor (6 components)</div> <div>response_member01, 02, ...</div> <div>Response of designated members</div> <div>response_structure</div> <div>Response of floors (horizontal components)</div>

[1] "data\_beam.txt"

Member number for Beam (total = 178)

0F	0	1	0	2	0	3	0
	4	0	5	0	6	0	7
	0	8	0	9	0	10	0
	11	0	12	0	13	0	14
	0	15	0	16	0	17	0
	18	0	19	0	20	0	21
	0	22	0	23	0	24	0
1F	0	25	0	0	0	26	0
	27	0	28	0	29	0	30
	0	31	0	32	0	33	0
	34	0	35	0	36	0	37



```

--- member properties (cm, kN) member = 1 --- ( type = 1 )
b : 60.000 d : 150.000 slab : 15.000
Ec : 0.230E+04
area : 11360.820
Iy : 0.284E+08
steel reinforcement
(up) 10- at = 11.400
(down) 10- at = 11.400
slab reinforcement
1- at = 0.713 @ 20.000
shear reinforcement
2- at = 5.067 @ 6.000
material strength
Fc = 2.50 Sy = 42.90 Sy(shear) = 42.90
moment from bottom rebars
Mc = 0.102E+06 My = 0.643E+06 Mu = 0.838E+06 Qm = 0.322E+04
Rc = 0.135E-03 Ry = 0.495E-02 Ry2 = 0.200E-01
moment from top rebars
Mc = 0.111E+06 My = 0.661E+06 Mu = 0.855E+06 Qm = 0.329E+04
Rc = 0.147E-03 Ry = 0.509E-02 Ry2 = 0.200E-01
parameters of damage index for flexural failure
Um = 15.0 beta = 0.200E+00
shear
Qc = 0.112E+04 Qy = 0.335E+04 Qu = 0.867E+04
Dc = 0.656E-01 Dy = 0.208E+01 Du = 0.520E+01

```

b: width d: height slab: thickness

Ec: Young's Modulus

area: Area

Iy: Moment of inertia

Fc: concrete strength Sy: steel strength Sy(shear): steel strength of hoop

Mc: crack moment My: yield moment Mu: ultimate moment Qm: shear force from My

Rc: crack rotation Ry: yield rotation of nonlinear spring Ry2: yield rotation

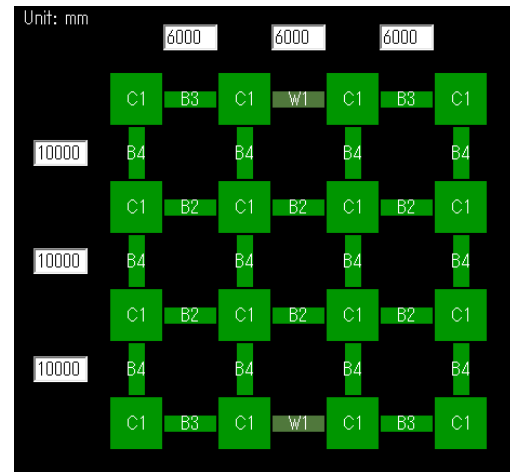
Qc: crack shear force Qy: yield shear force Qu: ultimate shear force

[2] "data\_column.txt"

Member number for Column (total = 112)

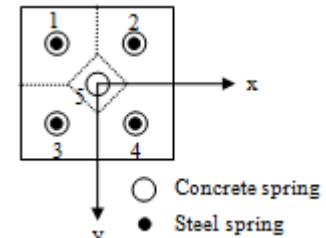
```

OF
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
1F
  1  0  2  0  3  0  4
  0  0  0  0  0  0  0
  5  0  6  0  7  0  8
  0  0  0  0  0  0  0
  9  0 10  0 11  0 12
  0  0  0  0  0  0  0
 13  0 14  0 15  0 16
  
```



```

--- member properties (cm, kN) member = 1 --- ( type = 1 )
b : 80.000 d : 80.000
Ec : 0.230E+04
area : 7318.336
Iy : 0.421E+07
Ix : 0.421E+07
steel reinforcement
  (corner) 4- at = 9.586
  (X-side) 4- at = 9.586
  (Y-side) 4- at = 9.586
shear reinforcement
  (X-side) 2- at = 5.067 @ 6.000
  (Y-side) 2- at = 5.067 @ 6.000
material strength
  Fc = 2.50 Sy = 42.90 Sy(shear) = 42.90
bending-spring: lelmc = 1
  axial force = 0.140E+04
moment
  Mc_y = 0.481E+05 My_y = 0.158E+06 Qm_y = 0.112E+04
  Rpc_y = 0.139E-03 Rpy_y = 0.369E-02 Ry_y = 0.421E-02
  Mc_x = 0.481E+05 My_x = 0.158E+06 Qm_x = 0.112E+04
  Rpc_x = 0.139E-03 Rpy_x = 0.369E-02 Ry_x = 0.421E-02
multi-spring No. 1
  x = -0.247E+02 y = -0.247E+02
  (concrete)
    Fc = -0.107E+04 Fy = -0.320E+04 Dc = -0.863E-02 Dy = -0.127E+00
  (steel)
    Fc = 0.410E+03 Fy = 0.123E+04 Dc = 0.191E-01 Dy = 0.127E+00
  }
multi-spring No. 5
  x = 0.000E+00 y = 0.000E+00
  (concrete)
    Fc = -0.347E+03 Fy = -0.104E+04 Dc = -0.112E-01 Dy = -0.127E+00
  (steel)
    Fc = 0.000E+00 Fy = 0.000E+00 Dc = 0.000E+00 Dy = 0.000E+00
parameters of damage index for flexural failure
  Um = 15.0 beta = 0.200E+00
shear
  Qc_x = 0.563E+03 Qy_x = 0.188E+04 Qu_x = 0.188E+02
  Dc_x = 0.300E-01 Dy_x = 0.112E+01 Du_x = 0.280E+01
  Qc_y = 0.563E+03 Qy_y = 0.188E+04 Qu_y = 0.188E+03
  Dc_y = 0.300E-01 Dy_y = 0.112E+01 Du_y = 0.280E+01
  
```



Multi-springs

multi-spring No. 1

x = -0.247E+02 y = -0.247E+02 location of spring

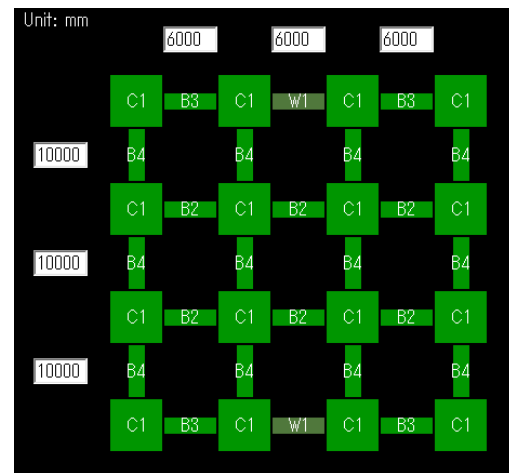
Fc: 1<sup>st</sup> force point Fy: yield force Dc: 1<sup>st</sup> deformation Dy: yield deformation

[3] "data\_wall.txt"

Member number for Wall (total = 14)

```

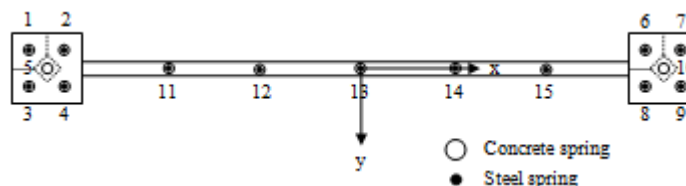
0F
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
1F
  0  0  0  1  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  0  0  0  0
  0  0  0  2  0  0  0
  
```



```

--- inelastic properties (cm, kN) member = 1 --- ( type = 1 )
thick: 0.300E+02
Ec : 0.230E+04
ac1 : 0.732E+04 ac2 : 0.732E+04 aw : 0.314E+05 ash : 0.237E+05
ic1 : 0.421E+07 ic2 : 0.421E+07 iw : 0.168E+10
bending-spring: lelmw = 1
axial force = 0.560E+04
moment
  Mc_y = 0.217E+07 My_y = 0.652E+07
  Rpc_y = 0.158E-04 Rpy_y = 0.102E-02 Ry_y = 0.118E-02
multi-spring No. 1
  x = -0.326E+03 y = -0.256E+02
(concrete)
  Fc = -0.107E+04 Fy = -0.320E+04 Dc = -0.863E-02 Dy = -0.139E+00
(steel)
  Fc = 0.410E+03 Fy = 0.123E+04 Dc = 0.191E-01 Dy = 0.139E+00
  }
multi-spring No.15
  x = 0.173E+03 y = 0.000E+00
(concrete)
  Fc = -0.221E+04 Fy = -0.663E+04 Dc = -0.863E-02 Dy = -0.139E+00
(steel)
  Fc = 0.419E+03 Fy = 0.126E+04 Dc = 0.191E-01 Dy = 0.139E+00
parameters of damage index for flexural failure
  Um = 15.0 beta = 0.500E-01
shear-spring
  Qc = 0.186E+04 Qy = 0.558E+04 Qu = 0.572E+04
  Dc = 0.224E-01 Dy = 0.112E+01 Du = 0.280E+01
parameters of damage index for shear failure
  Um = 8.0 beta = 0.100E+00
  
```

ac1: area of column 1 ac2: area of column 2 aw: area of wall ash: area for shear  
ic1: I(moment of inertia) of column 1 ic2: I of column 2 iw: I of wall



軸ばね(multi-spring)

[4] "data\_ground.txt"

\*\*\*\* GROUND SPRING \*\*\*\*

<Foundation>				<Pile>			
Sway							
F_RKhx	F_IKhx	F_RKhy	F_IKhy	P_RKhx	P_IKhx	P_RKhy	P_IKhy
(kN/cm)	(kN/cm)	(kN/cm)	(kN/cm)	(kN/cm)	(kN/cm)	(kN/cm)	(kN/cm)
0.1343E+05	0.2551E+04	0.1343E+05	0.2551E+04	0.1439E+05	0.2735E+04	0.3916E+05	0.7438E+04
Rocking							
F_RKry	F_IKry	F_RKrx	F_IKrx	P_RKry	P_IKry	P_RKrx	P_IKrx
(kNcm/rad)	(kNcm/rad)	(kNcm/rad)	(kNcm/rad)	(kNcm/rad)	(kNcm/rad)	(kNcm/rad)	(kNcm/rad)
0.4514E+11	0.8577E+10	0.4514E+11	0.8577E+10	0.5902E+11	0.1121E+11	0.1377E+12	0.2616E+11
Radiation							
F_Chx	F_Chx	F_Cry	F_Crx	P_Chx	P_Chx	P_Cry	P_Crx
(kNs/cm)	(kNs/cm)	(kNs/cm)	(kNs/cm)	(kNs/cm)	(kNs/cm)	(kNs/cm)	(kNs/cm)
0.1512E+04	0.1512E+04	0.2153E+10	0.2153E+10	0.1586E+04	0.1134E+04	0.0000E+00	0.0000E+00
mass							
	Ix	Iy					
(kNs <sup>2</sup> /cm)	(kNcms <sup>2</sup> )	(kNcms <sup>2</sup> )					
0.0000E+00	0.0000E+00	0.0000E+00					
1							
Tx	Tswx	Trkx	Ty	Tswy	Trky		
0.575	0.270	0.197	0.838	0.196	0.261		
h							
0.030							
hx(2)	hx(3)	hswx	hrky	r_hswx	r_hrky		
0.202	0.109	0.095	0.095	0.609	0.113		
hy(2)	hy(3)	hswy	hrkx	r_hswy	r_hrkx		
0.066	0.040	0.095	0.095	0.189	0.044		

F(foundation), P(pile), R(real), I(imaginary), K(stiffness), C(damping), h(sway), r(rocking),  
x(X-axis), y(Y-axis)

For example

F\_RKhx: Foundation Sway Stiffness Spring, Real part, in X-axis

P\_IKry: Pile Rocking Stiffness Spring, Imaginary part, around Y-axis

F\_Chx: Foundation Sway Damping coefficient, in X-direction

Tx : Building Period in X-axis, Tswx : Sway Period in X-axis, Trky : Rocking Period around Y-axis

Ty : Building Period in Y-axis, Tswy : Sway Period in Y-axis, Trkx : Rocking Period around X-axis

h : Building Damping factor

$hx(2) = h + (Tswx/Tx)^2 (hswx + r_hswx) + (Trky/Tx)^2 (hrky + r_hrky)$

$hx(3) = h + (Tswx/Tx)^3 (hswx + r_hswx) + (Trky/Tx)^3 (hrky + r_hrky)$

hswx : Sway Damping factor in X-axis, hswy : Sway Damping factor in Y-axis

hrky : Rocking Damping factor around Y-axis, hrkx : Rocking Damping factor around X-axis

r\_hswx : Radiation Sway Damping factor in X-axis,

r\_hswy : Radiation Sway Damping factor in Y-axis

r\_hrky : Radiation Rocking Damping factor around Y-axis,

r\_hrkx : Radiation Rocking Damping factor around X-axis,

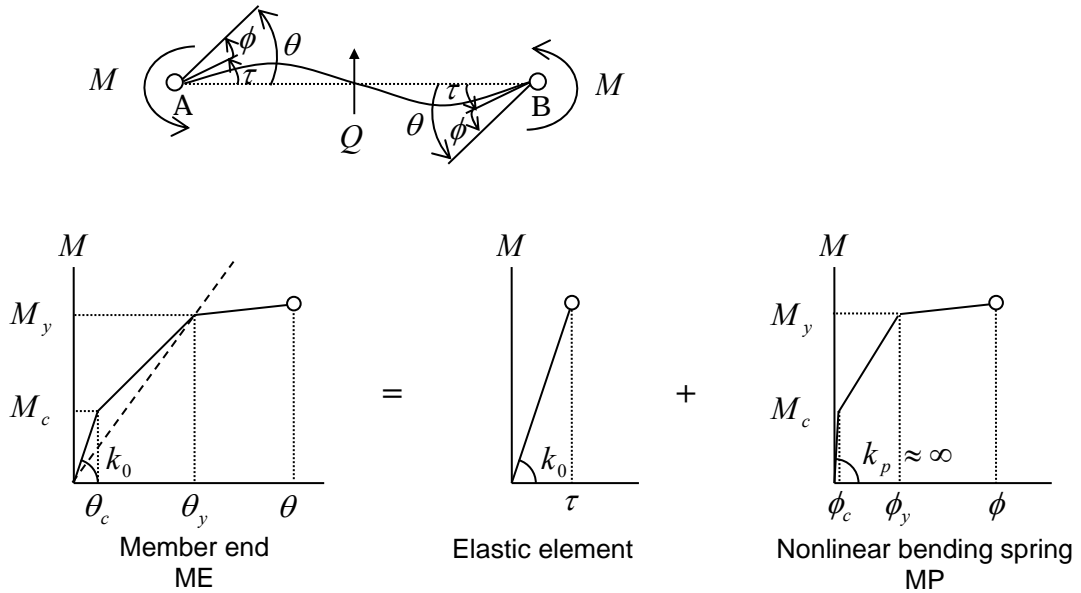
[5] "max\_beam.csv"

Unit (kN,cm)

EL.NO.=		33	disp	force	Um	Uh	D.I
ME	1	-0.2621E-02	-0.1034E+06	-0.62	0.093	0.043	
MP	1	0.2403E-02	-0.1034E+06	0.76	0.000		
ME	2	-0.3307E-02	0.1065E+06	-0.79	0.089	0.054	
MP	2	-0.3050E-02	0.1065E+06	-0.96	0.000		
Q	1	-0.5575E-01	-0.3563E+03	-0.03			

ME: Member end A  
 MP: Nonlinear bending spring at end A  
 ME: Member end B  
 MP: Nonlinear bending spring at end B  
 Q: Nonlinear shear spring

Um: Ductility factor (= Dm / Dy)  
 (Dm: max disp., Dy: yield disp.)  
 Uh: Cumulative ductility factor (= Eh / QyDy)  
 (Eh: hysteresis energy, Qy: yield force)  
 D.I.: Damage Index  
 (RC: Park and Ang, S: Fatigue)



Ductility factor is the ratio of the maximum deformation divided by the yield deformation as,

$$\mu = \frac{\theta_{\max}}{\theta_y} \text{ for member end}$$

$$\mu = \frac{\phi_{\max}}{\phi_y} \text{ for nonlinear bending spring}$$



[6] "max\_column.csv"

Unit (kN,cm)

EL. NO. =	1	disp	force	Umy		MX	1	disp	force	Umx	Uh	D. I
MY	1	0.2585E-02	-0.1348E+06	0.00		ST	1	0.2098E-05	-0.2680E+04	0.00	0.040	0.001
CO	1	0.9620E-01	-0.1766E+04	-0.76 /		ST	1	0.9620E-01	0.9981E+03	0.76		
CO	2	0.2662E-01	-0.6613E+03	-0.21 /		ST	2	0.2662E-01	0.4679E+03	0.21		
CO	3	0.9647E-01	-0.1763E+04	-0.76 /		ST	3	0.9647E-01	0.1000E+04	0.76		
CO	4	0.2683E-01	-0.6391E+03	-0.21 /		ST	4	0.2683E-01	0.4695E+03	0.21		
CO	5	0.4591E-01	-0.3200E+03	-0.36 /		ST	5	0.0000E+00	0.0000E+00	0.00		
MY	2	0.5071E-03	0.2786E+05	0.00		MX	2	0.1515E-04	0.1793E+04	0.00	0.062	0.001
CO	1	-0.6212E-02	-0.7677E+03	0.05 /		ST	1	-0.6212E-02	-0.1337E+03	-0.05		
CO	2	0.1827E-01	-0.9443E+03	-0.14 /		ST	2	0.1827E-01	0.3933E+03	0.14		
CO	3	-0.6243E-02	-0.7715E+03	0.05 /		ST	3	-0.6243E-02	-0.1344E+03	-0.05		
CO	4	0.1808E-01	-0.9475E+03	-0.14 /		ST	4	0.1808E-01	0.3892E+03	0.14		
CO	5	0.8815E-02	-0.2104E+03	-0.07 /		ST	5	0.0000E+00	0.0000E+00	0.00		
QX		-0.2722E-01	-0.5106E+03	-0.02		QY		-0.4822E-03	-0.9044E+01	-0.00		
N		-0.7363E-01	-0.4163E+04									

MY: moment around Y axis, end A

MX: moment around X axis, end A

CO : concrete spring

ST: steel spring

MY: moment around Y axis, end B

MX: moment around X axis, end B

CO : concrete spring

ST: steel spring

QX : shear force in X axis

QY: shear force in Y axis

N: axial force

Umx, Umy: Ductility factor (= Dm / Dy)

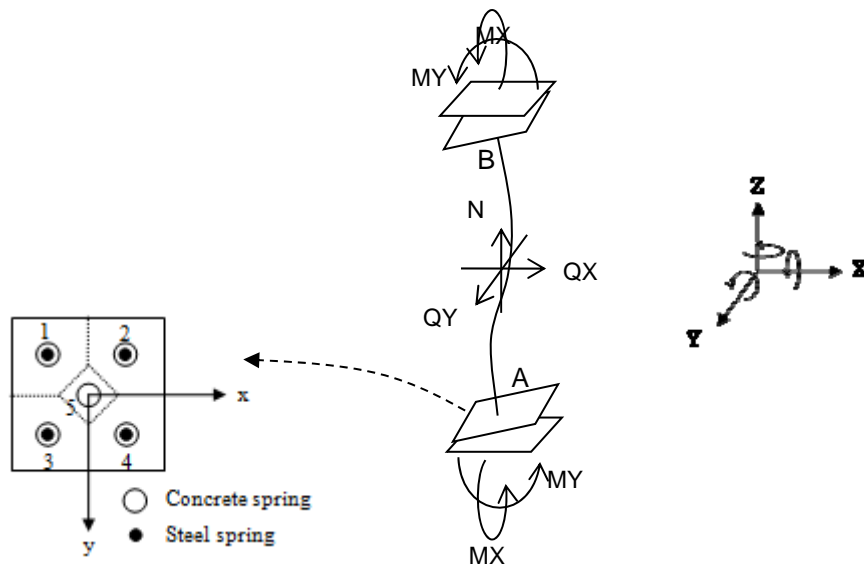
(Dm: max disp., Dy: yield disp.)

Uh: Cumulative ductility factor (=Eh / QyDy)

(Eh: hysteresis energy, Qy: yield force)

D.I.: Damage Index

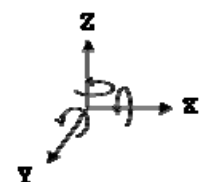
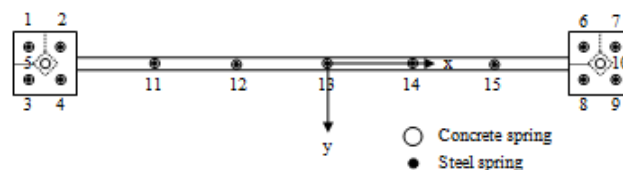
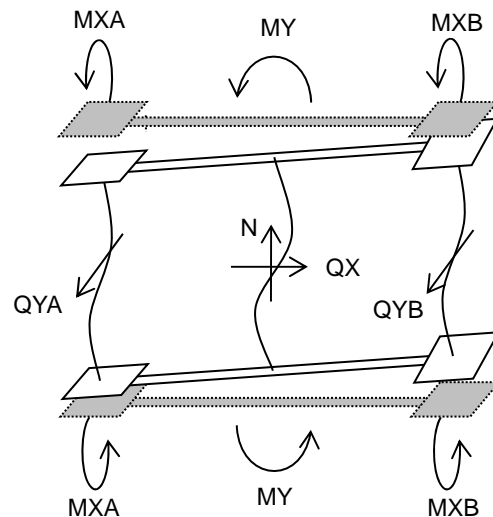
(RC: Park and Ang)



[7] "max\_wall.csv"

Unit (kN,cm)

EL.NO.=	1	disp	force	Um		disp	force	Um		Uh	D.I
MY	1	0.2818E-02	-0.6502E+07	2.38						6.577	0.181
MXA	1	0.4383E-04	-0.9028E+04	0.01	MXB	1	0.4901E-04	-0.2265E+05	0.01		
CO	1	0.1792E+00	-0.2706E+04	-1.29 /	ST	1	0.1792E+00	0.1241E+04	1.29		
CO	2	0.1595E+00	-0.2202E+04	-1.15 /	ST	2	0.1595E+00	0.1242E+04	1.15		
CO	3	0.1860E+00	-0.2684E+04	-1.34 /	ST	3	0.1860E+00	0.1236E+04	1.34		
CO	4	0.1663E+00	-0.2180E+04	-1.20 /	ST	4	0.1663E+00	0.1245E+04	1.20		
CO	5	0.1728E+00	-0.7880E+03	-1.24 /	ST	5	0.0000E+00	0.0000E+00	0.00		
CO	6	0.2655E+00	-0.1733E+04	-1.91 /	ST	6	0.2655E+00	0.1276E+04	1.91		
CO	7	0.2973E+00	-0.2031E+04	-2.14 /	ST	7	0.2973E+00	0.1272E+04	2.14		
CO	8	0.2686E+00	-0.1759E+04	-1.93 /	ST	8	0.2686E+00	0.1284E+04	1.93		
CO	9	0.3004E+00	-0.2048E+04	-2.16 /	ST	9	0.3004E+00	0.1278E+04	2.16		
CO	10	0.2829E+00	-0.6032E+03	-2.04 /	ST	10	0.0000E+00	0.0000E+00	0.00		
CO	11	0.1241E+00	-0.2784E+04	-0.89 /	ST	11	0.1241E+00	0.1153E+04	0.89		
CO	12	0.9080E-01	-0.1037E+04	-0.65 /	ST	12	0.9080E-01	0.9201E+03	0.65		
CO	13	0.9653E-01	-0.8239E+03	-0.70 /	ST	13	0.9653E-01	0.9602E+03	0.70		
CO	14	0.1502E+00	-0.9546E+03	-1.08 /	ST	14	0.1502E+00	0.1260E+04	1.08		
CO	15	0.2041E+00	-0.2528E+04	-1.47 /	ST	15	0.2041E+00	0.1265E+04	1.47		
MY	2	0.1919E-02	0.5459E+07	-1.25						5.934	0.103
MXA	2	0.3166E-03	0.7015E+04	-0.01	MXB	2	0.3540E-03	0.8920E+04	0.07		
CO	1	0.1171E+00	-0.2101E+04	-0.84 /	ST	1	0.1171E+00	0.1082E+04	0.84		
CO	2	0.1042E+00	-0.1851E+04	-0.75 /	ST	2	0.1042E+00	0.9938E+03	0.75		
CO	3	0.1056E+00	-0.2040E+04	-0.76 /	ST	3	0.1056E+00	0.1003E+04	0.76		
CO	4	0.9271E-01	-0.1789E+04	-0.67 /	ST	4	0.9271E-01	0.9149E+03	0.67		
CO	5	0.1049E+00	-0.6239E+03	-0.76 /	ST	5	0.0000E+00	0.0000E+00	0.00		
CO	6	0.1260E+00	-0.1641E+04	-0.91 /	ST	6	0.1260E+00	0.1143E+04	0.91		
CO	7	0.1417E+00	-0.1851E+04	-1.02 /	ST	7	0.1417E+00	0.1232E+04	1.02		
CO	8	0.1115E+00	-0.1547E+04	-0.80 /	ST	8	0.1115E+00	0.1043E+04	0.80		
CO	9	0.1272E+00	-0.1758E+04	-0.92 /	ST	9	0.1272E+00	0.1151E+04	0.92		
CO	10	0.1266E+00	-0.5423E+03	-0.91 /	ST	10	0.0000E+00	0.0000E+00	0.00		
CO	11	0.7293E-01	-0.2757E+04	-0.53 /	ST	11	0.7293E-01	0.7952E+03	0.53		
CO	12	0.5109E-01	-0.1447E+04	-0.37 /	ST	12	0.5109E-01	0.6426E+03	0.37		
CO	13	0.3447E-01	-0.8293E+03	-0.25 /	ST	13	0.3447E-01	0.5264E+03	0.25		
CO	14	0.6098E-01	-0.1242E+04	-0.44 /	ST	14	0.6098E-01	0.7117E+03	0.44		
CO	15	0.8762E-01	-0.2442E+04	-0.63 /	ST	15	0.8762E-01	0.8979E+03	0.63		
QX		0.6429E+00	0.3965E+04	0.57						0.090	0.073
QYA		-0.1663E-02	-0.3120E+02	-0.00	QYB		-0.4549E-02	-0.8533E+02	-0.00		
N		0.1084E+00	-0.8350E+04								



[8] "max\_node.csv"

Unit (kN,cm)

Maximum Nodal Response

0F									
	1	2	3	4					
	5	6	7	8					
	9	10	11	12					
	13	14	15	16					
Center of gravity:				17					
1F									
	18	19	20	21					
	22	23	24	25					
	26	27	28	29					
	30	31	32	33					
Center of gravity:				34					
2F									
	35	36	37	38					
	39	40	41	42					
	43	44	45	46					
	47	48	49	50					
Center of gravity:				51					
node	X	Y	Z	dx	dy	dz	rx	ry	rz
1	0.00	0.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	600.00	0.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
3	1200.00	0.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
4	1800.00	0.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
5	0.00	1000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
6	600.00	1000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
7	1200.00	1000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
8	1800.00	1000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
9	0.00	2000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
10	600.00	2000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
11	1200.00	2000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
12	1800.00	2000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
13	0.00	3000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
14	600.00	3000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
15	1200.00	3000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
16	1800.00	3000.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
17	900.00	1500.00	0.00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
18	0.00	0.00	400.00	0.1021E+02	0.9873E-13	0.1164E+01	0.2864E-03	0.2045E-01	0.1325E-15
19	600.00	0.00	400.00	0.1021E+02	0.1921E-13	0.6651E+01	0.4259E-02	0.1212E-01	0.1325E-15
20	1200.00	0.00	400.00	0.1021E+02	0.6032E-13	0.6237E+00	0.3035E-03	0.1212E-01	0.1325E-15
21	1800.00	0.00	400.00	0.1021E+02	0.1398E-12	0.4719E+00	0.9053E-04	0.3001E-01	0.1325E-15
22	0.00	1000.00	400.00	0.1021E+02	0.9873E-13	0.8802E+00	0.4683E-04	0.2883E-01	0.1325E-15

node	node number
<Coordinate>	
X	X coordinate (cm)
Y	Y coordinate (cm)
Z	Z coordinate (cm)
<Maximum nodal displacement>	
dx	displacement in X-direction (cm)
dy	displacement in Y-direction (cm)
dz	displacement in Z-direction (cm)
rx	rotational angle around X-direction
ry	rotational angle around Y-direction
rz	rotational angle around Z-direction

[9] "max\_structure.csv"

Unit (kN,cm)

F	h (cm)	sdx (cm)	sdv (cm)	drx	dry	sfx (kN)	sfy (kN)		
7	0.4000E+03	0.1135E+01	0.1677E-13	0.2837E-02	0.4192E-16	0.5144E+04	0.9031E-11		
6	0.4000E+03	0.1227E+01	0.1451E-13	0.3068E-02	0.3628E-16	0.8306E+04	0.1748E-10		
5	0.4000E+03	0.1339E+01	0.1210E-13	0.3348E-02	0.3024E-16	0.1073E+05	0.2489E-10		
4	0.4000E+03	0.1481E+01	0.8657E-14	0.3704E-02	0.2164E-16	0.1243E+05	0.2757E-10		
3	0.4000E+03	0.1502E+01	0.5875E-14	0.3756E-02	0.1469E-16	0.1322E+05	0.3404E-10		
2	0.4000E+03	0.1400E+01	0.3609E-14	0.3500E-02	0.9023E-17	0.1368E+05	0.3691E-10		
1	0.4000E+03	0.8303E+00	0.7058E-14	0.2076E-02	0.1764E-16	0.1450E+05	0.3727E-10		
0	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.1450E+05	0.3727E-10		
	dx (cm)	dy (cm)	rz (rad)	vx (cm/s)	vy (cm/s)	ax (gal)	ay (gal)	D. I (F)	D. I (F+W)
	0.8901E+01	0.5822E-13	0.2971E-15	0.7102E+02	0.1467E-12	0.7109E+03	0.1276E-11	0.030	0.028
	0.7767E+01	0.4288E-13	0.2600E-15	0.6280E+02	0.1251E-12	0.5557E+03	0.1099E-11	0.047	0.023
	0.6540E+01	0.2989E-13	0.2190E-15	0.5418E+02	0.1137E-12	0.4624E+03	0.9420E-12	0.037	0.030
	0.5202E+01	0.2070E-13	0.1738E-15	0.4483E+02	0.9799E-13	0.4306E+03	0.8408E-12	0.041	0.008
	0.3723E+01	0.1435E-13	0.1287E-15	0.3356E+02	0.7625E-13	0.3914E+03	0.6853E-12	0.040	0.011
	0.2224E+01	0.9603E-14	0.7822E-16	0.2101E+02	0.4645E-13	0.3268E+03	0.5067E-12	0.041	0.093
	0.8303E+00	0.5049E-14	0.2854E-16	0.8071E+01	0.2068E-13	0.3259E+03	0.3338E-12	0.026	0.142
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.000	0.000
						total		0.037	0.093

F	story number
h	story height (cm)
<Maximum relative story displacement>	
sdx	story drift in X-direction (cm)
sdv	story drift in Y-direction (cm)
drx	story drift ratio in X-direction (rotational component)
dry	story drift ratio in Y-direction (rotational component)
<Maximum story shear force>	
sfx	story shear force in X-direction (kN)
sfy	story shear force in Y-direction (kN)
<Maximum displacement from the ground at the center of gravity in each floor>	
dx	displacement in X-direction (cm)
dy	displacement in Y-direction (cm)
rz	rotational angle around Z-direction
<Maximum relative velocity>	
vx	velocity in X-direction (cm/sec)
vy	velocity in Y-direction (cm/sec)
<Maximum absolute acceleration>	
ax	acceleration in X-direction (cm/sec <sup>2</sup> )
ay	acceleration in Y-direction (cm/sec <sup>2</sup> )
<Damage Index>	
D.I.(F)	Damage Index of each floor (beam and column)
D.I.(F+W)	Damage Index of each floor (beam, column, and wall)
total:	average damage index of entire building

[10] "response\_eigen.csv"

In this file, the results of modal analysis including natural periods, mode vectors, and stimulus functions are saved.

=== natural period and mode ===					
++ 1-mode ++					
natural period					
	T (sec)				
	0.76562				
participation factor					
	bx	by	bz		
	0	6.36038	0		
effective mass ratio					
	mx	my	mz		
	0	0.78661	0		
mode vector					
		mode	bx{v}	by{v}	bz{v}
X-component					
	0F	0	0	0	0
	1F	0	0	0	0
	2F	0	0	0	0
	3F	0	0	0	0
	4F	0	0	0	0
	5F	0	0	0	0
	6F	0	0	0	0
	7F	0	0	0	0
Y-component					
	0F	0	0	0	0
	1F	0.01921	0	0.12215	0
	2F	0.05744	0	0.36532	0
	3F	0.09667	0	0.61486	0
	4F	0.13247	0	0.84253	0
	5F	0.16294	0	1.03637	0
	6F	0.1888	0	1.20082	0
	7F	0.2082	0	1.32425	0
Z-rotation					
	0F	0	0	0	0
	1F	0	0	0	0
	2F	0	0	0	0
	3F	0	0	0	0
	4F	0	0	0	0
	5F	0	0	0	0
	6F	0	0	0	0
	7F	0	0	0	0
++ 2-mode ++					

[11] "response\_structure.csv"

In case of nonlinear static analysis, the following data are saved for each story.

kstep	Sd(cm)	Sa(gal)	max drift			
0	0.00E+00	0.00E+00	0			
1	1.20E-02	3.33E+01	0.00004			
2	2.40E-02	6.67E+01	0.00008			
3	3.60E-02	1.00E+02	0.00012			
4	4.80E-02	1.26E+02	0.00016			
F	sdx(cm)	sdycm)	ssx(cm)	ssy(cm)	sfx(kN)	sfy(kN)
0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	3.40E+01	-1.54E-17
0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	6.80E+01	-1.54E-17
0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	1.02E+02	-1.54E-17
0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	1.28E+02	-3.84E-17
sbx(cm)	sby(cm)	smx(kN)	smy(kN)	dx(cm)	dy(cm)	rz(rad)
0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
0.00E+00	0.00E+00	-5.10E+03	-2.31E-15	0.00E+00	0.00E+00	0.00E+00
0.00E+00	0.00E+00	-1.02E+04	-2.31E-15	0.00E+00	0.00E+00	0.00E+00
0.00E+00	0.00E+00	-1.53E+04	-2.31E-15	0.00E+00	0.00E+00	0.00E+00
0.00E+00	0.00E+00	-1.93E+04	-5.75E-15	0.00E+00	0.00E+00	0.00E+00

kstep      calculation step in static analysis  
 < Equivalent 1DOF system>  
     sd          displacement (cm)  
     sa          acceleration (gal)  
     max drift   maximum drift among all stories  
 <Relative story displacement>  
     F          story number  
     sdx          story drift in X-direction (cm)  
     sdy          story drift in Y-direction (cm)  
 <Relative story displacement (shear component)>  
     ssx          story drift in X-direction (shear component) (cm)  
     ssy          story drift in Y-direction (shear component) (cm)  
 <Story shear force>  
     sfx          story shear force in X-direction (kN)  
     sfy          story shear force in Y-direction (kN)  
 <Relative story displacement (rotational component)>  
     sbx          story drift in X-direction (rotational component)  
     sby          story drift in Y-direction (rotational component)  
 <Story moment>  
     smx          story moment in X-direction (kNcm)  
     smy          story moment in Y-direction (kNcm)  
 <Displacement from the ground at the center of gravity in each floor>  
     dx          displacement in X-direction (cm)  
     dy          displacement in Y-direction (cm)  
     rz          rotational angle around Z-direction

In case of earthquake response analysis, the following data are saved for each story:

kstep	t	a0x	a0y	a0z	d0x	d0y	d0z
0	0	-1.40E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
5	0.02	-1.08E+01	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
10	0.04	-1.01E+01	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
15	0.06	-8.80E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
20	0.08	-9.50E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
F	sdx(cm)	sdv(cm)	sfx(kN)	sfy(kN)	dx(cm)	dy(cm)	rz(rad)
0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
0	0.00E+00	0.00E+00	2.35E+00	-1.53E-17	0.00E+00	0.00E+00	0.00E+00
0	0.00E+00	0.00E+00	1.10E+01	-1.31E-17	0.00E+00	0.00E+00	0.00E+00
0	0.00E+00	0.00E+00	1.81E+01	-1.39E-17	0.00E+00	0.00E+00	0.00E+00
0	0.00E+00	0.00E+00	1.61E+01	-1.48E-17	0.00E+00	0.00E+00	0.00E+00
vx(cm/s)	vy(cm/s)	ax(gal)	ay(gal)				
0.00E+00	0.00E+00	0.00E+00	0.00E+00				
0.00E+00	0.00E+00	0.00E+00	0.00E+00				
0.00E+00	0.00E+00	0.00E+00	0.00E+00				
0.00E+00	0.00E+00	0.00E+00	0.00E+00				
0.00E+00	0.00E+00	0.00E+00	0.00E+00				

t time step in dynamic analysis

<Input ground acceleration>

a0x acceleration in X-direction (cm/sec<sup>2</sup>)

a0y acceleration in Y-direction (cm/sec<sup>2</sup>)

a0z acceleration in Z-direction (cm/sec<sup>2</sup>)

<Input ground displacement (cm)>

d0x displacement in X-direction (cm)

d0y displacement in Y-direction (cm)

d0z displacement in Z-direction (cm)

<Relative story displacement>

F story number

sdx story drift in X-direction (cm)

sdv story drift in Y-direction (cm)

< Story shear force>

sfx story shear force in X-direction (kN)

sfy story shear force in Y-direction (kN)

<Displacement from the ground at the center of gravity in each floor>

dx displacement in X-direction (cm)

dy displacement in Y-direction (cm)

rz rotational angle around Z-direction

<Relative velocity>

vx velocity in X-direction (cm/sec)

vy velocity in Y-direction (cm/sec)

<Absolute acceleration>

ax acceleration in X-direction (cm/sec<sup>2</sup>)

ay acceleration in Y-direction (cm/sec<sup>2</sup>)

[12] "response\_member01.csv ..."

In this file, the responses of the designated members are saved.

In case of Beam

BE No.	1						
		Rya	Mya	Uya	Rpa	Mpa	Upa
0.000		-0.7362E-21	-0.1249E-13	0.000	-0.3669E-21	-0.1249E-13	0.000
0.020		0.1087E-06	0.2127E+01	0.000	0.6252E-07	0.2127E+01	0.000
0.040		0.8696E-06	0.1702E+02	0.000	0.5001E-06	0.1702E+02	0.000
0.060		0.4024E-05	0.7874E+02	0.002	0.2314E-05	0.7874E+02	0.001
0.080		0.9845E-05	0.1927E+03	0.004	0.5662E-05	0.1927E+03	0.003
		Ryb	Myb	Uyb	Rpb	Mpb	Upb
		-0.4351E-21	-0.1044E-13	0.000	-0.3067E-21	-0.1044E-13	0.000
		0.1087E-06	0.2127E+01	0.000	0.6252E-07	0.2127E+01	0.000
		0.8696E-06	0.1702E+02	0.000	0.5001E-06	0.1702E+02	0.000
		0.4024E-05	0.7874E+02	0.002	0.2314E-05	0.7874E+02	0.001
		0.9845E-05	0.1927E+03	0.004	0.5662E-05	0.1927E+03	0.003
		Dsz	Qsz	Usz	Dx	Nx	
		-0.1311E-19	-0.4245E-16	0.000	0.0000E+00	0.0000E+00	
		0.2433E-05	0.7879E-02	0.000	0.0000E+00	0.0000E+00	
		0.1946E-04	0.6303E-01	0.000	0.0000E+00	0.0000E+00	
		0.9005E-04	0.2916E+00	0.000	0.0000E+00	0.0000E+00	
		0.2203E-03	0.7136E+00	0.000	0.0000E+00	0.0000E+00	

Disp.	Force.	Ductility factor	(kN, cm)
< Moment >			
Rya	Mya	Uya	Member end A
Rpa	Mpa	Upa	Member end B
Ryb	Myb	Uyb	Nonlinear rotational spring at end A
Rpb	Mpb	Upb	Nonlinear rotational spring at end B
< Shear Force >			
Dsz	Qsz	Usz	Nonlinear shear spring
< Axial Force >			
Dx	Nx		Axial spring



In case of Column

CO No.	1						
	Rya	Mya	Uya	Ryb	Myb	Uyb	
0.000	-0.2659E-21	0.3082E-15	0.000	0.5811E-21	0.1816E-13	0.000	
0.020	-0.9682E-07	-0.3822E+01	0.000	0.1773E-08	-0.1745E+01	0.000	
0.040	-0.7746E-06	-0.3058E+02	0.000	0.1425E-07	-0.1396E+02	0.000	
0.060	-0.3584E-05	-0.1415E+03	0.000	0.6608E-07	-0.6459E+02	0.000	
0.080	-0.8772E-05	-0.3463E+03	0.000	0.1627E-06	-0.1580E+03	0.000	
	Rxa	Mxa	Uxa	Rxb	Mxb	Uxb	
	-0.5793E-21	-0.2197E-13	0.000	0.5866E-22	-0.8529E-14	0.000	
	-0.4518E-06	-0.1784E+02	0.000	0.8275E-08	-0.8144E+01	0.000	
	-0.4014E-05	-0.1585E+03	0.000	0.7376E-07	-0.7234E+02	0.000	
	-0.1407E-04	-0.5555E+03	0.000	0.2606E-06	-0.2535E+03	0.000	
	-0.2144E-04	-0.8464E+03	0.000	0.3975E-06	-0.3863E+03	0.000	
	Dsx	Qsx	Usx	Dsy	Qsy	Usy	
	0.6335E-20	0.6838E-16	0.000	-0.1046E-19	-0.1130E-15	0.000	
	-0.1910E-05	-0.2062E-01	0.000	-0.8914E-05	-0.9623E-01	0.000	
	-0.1528E-04	-0.1650E+00	0.000	-0.7919E-04	-0.8548E+00	-0.001	
	-0.7071E-04	-0.7633E+00	-0.001	-0.2776E-03	-0.2996E+01	-0.004	
	-0.1730E-03	-0.1868E+01	-0.003	-0.4229E-03	-0.4566E+01	-0.006	
	Dz	Nz	Rz	Tz			
	-0.7726E-02	-0.2500E+03	-0.1983E-21	-0.1721E-14			
	-0.7725E-02	-0.2500E+03	-0.1974E-21	-0.1713E-14			
	-0.7718E-02	-0.2497E+03	-0.1883E-21	-0.1634E-14			
	-0.7699E-02	-0.2491E+03	-0.1013E-21	-0.8791E-15			
	-0.7694E-02	-0.2490E+03	0.1517E-21	0.1317E-14			
	C1D(a)	C1F(a)	C1U(a)	S1D(a)	S1F(a)	S1U(a)	
	-0.3072E-02	-0.2994E+03	0.032	-0.3072E-02	-0.5215E+02	-0.032	
	-0.3090E-02	-0.3012E+03	0.032	-0.3090E-02	-0.5245E+02	-0.032	
	-0.3111E-02	-0.3033E+03	0.032	-0.3111E-02	-0.5282E+02	-0.032	
	-0.3112E-02	-0.3033E+03	0.032	-0.3112E-02	-0.5283E+02	-0.032	
	-0.3091E-02	-0.3012E+03	0.032	~ -0.3091E-02	-0.5247E+02	-0.032	
	C1D(b)	C1F(b)	C1U(b)	S1D(b)	S1F(b)	S1U(b)	
	-0.2987E-02	-0.2912E+03	0.031	-0.2987E-02	-0.5071E+02	-0.031	
	-0.3003E-02	-0.2927E+03	0.031	-0.3003E-02	-0.5098E+02	-0.031	
	-0.3028E-02	-0.2950E+03	0.032	-0.3028E-02	-0.5137E+02	-0.032	
	-0.3028E-02	-0.2952E+03	0.032	-0.3028E-02	-0.5141E+02	-0.032	
	-0.3008E-02	-0.2932E+03	0.032	~ -0.3008E-02	-0.5108E+02	-0.032	
	Disp.	Force.	Ductility factor	(kN, cm)			
	< Moment >						
	Rya	Mya	Uya	End A	(Bottom)	Y-direction	
	Ryb	Myb	Uyb	End B	(Bottom)	Y-direction	
	Rxa	Mxa	Uxa	End A	(Bottom)	X-direction	
	Rxb	Mxb	Uxb	End B	(Bottom)	X-direction	
	< Shear Force >						
	Dsx	Qsx	Usx	Nonlinear shear spring		X-direction	
	Dsy	Qsy	Usy	Nonlinear shear spring		Y-direction	
	< Axial Force >						
	Dz	Nz		Axial spring			
	< Torque Force >						
	Rz	Tz		Torque spring			
	< Multi-spring >						
	C1D(a)	C1F(a)	C1U(a)	End A	Concrete Spring 1		
	C2D(a)	C2F(a)	C2U(a)	End A	Concrete Spring 2		
	C3D(a)	C3F(a)	C3U(a)	End A	Concrete Spring 3		
	C4D(a)	C4F(a)	C4U(a)	End A	Concrete Spring 4		
	C5D(a)	C5F(a)	C5U(a)	End A	Concrete Spring 5		
	S1D(a)	S1F(a)	S1U(a)	End A	Steel Spring 1		
	S2D(a)	S2F(a)	S2U(a)	End A	Steel Spring 2		
	S3D(a)	S3F(a)	S3U(a)	End A	Steel Spring 3		
	S4D(a)	S4F(a)	S4U(a)	End A	Steel Spring 4		
	S5D(a)	S5F(a)	S5U(a)	End A	Steel Spring 5		
	C1D(b)	C1F(b)	C1U(b)	End B	Concrete Spring 1		
	C2D(b)	C2F(b)	C2U(b)	End B	Concrete Spring 2		
	C3D(b)	C3F(b)	C3U(b)	End B	Concrete Spring 3		
	C4D(b)	C4F(b)	C4U(b)	End B	Concrete Spring 4		
	C5D(b)	C5F(b)	C5U(b)	End B	Concrete Spring 5		
	S1D(b)	S1F(b)	S1U(b)	End B	Steel Spring 1		
	S2D(b)	S2F(b)	S2U(b)	End B	Steel Spring 2		
	S3D(b)	S3F(b)	S3U(b)	End B	Steel Spring 3		
	S4D(b)	S4F(b)	S4U(b)	End B	Steel Spring 4		
	S5D(b)	S5F(b)	S5U(b)	End B	Steel Spring 5		

## STERA 3D User Manual

### In case of Wall

WA No.	1						
		Rya	Mya	Uya	Ryb	Myb	Uyb
0.004	-0.1187E-07	-0.6907E+02	0.000	-0.9263E-08	-0.1300E+02	0.000	
0.024	-0.2198E-06	-0.1732E+04	0.000	-0.1199E-06	0.4134E+03	0.000	
0.044	-0.2985E-06	-0.3039E+04	0.000	-0.8521E-07	0.1549E+04	0.000	
0.064	0.8330E-07	-0.5840E+03	0.000	0.1862E-06	0.1631E+04	0.000	
0.084	0.9203E-06	0.6774E+04	0.000	0.5574E-06	-0.1035E+04	0.000	
	Rsx	Qsx	Usx	Dz	Nz		
-0.9919E-08	-0.2312E+00	-0.000	-0.2937E-01	-0.5986E+04			
-0.1594E-06	-0.3714E+01	-0.000	-0.2952E-01	-0.8018E+04			
-0.1801E-06	-0.4197E+01	-0.000	-0.2973E-01	-0.8060E+04			
0.1265E-06	0.2948E+01	-0.000	-0.2975E-01	-0.8064E+04			
0.6936E-06	0.1617E+02	0.000	-0.2956E-01	-0.8026E+04			
	C11D(a)	C11F(a)	C11U(a)	S11D(a)	S11F(a)	S11U(a)	
-0.2905E-02	-0.5887E+03	0.030	-0.2905E-02	-0.5032E+02	-0.030		
-0.2923E-02	-0.5903E+03	0.030	-0.2923E-02	-0.5063E+02	-0.030		
-0.2946E-02	-0.5950E+03	0.031	-0.2946E-02	-0.5103E+02	-0.031		
-0.2943E-02	-0.5945E+03	0.031	-0.2943E-02	-0.5099E+02	-0.031		
-0.2913E-02	-0.5883E+03	0.031	~ -0.2913E-02	-0.5046E+02	-0.031		

Disp.	Force.	Ductility factor	(kN, cm)		
< Moment >					
Rya	Mya	Uya	End A	(Bottom)	Y-direction
Ryb	Myb	Uyb	End B	(Bottom)	Y-direction
Rxa	Mxa	Uxa	End A	(Bottom)	X-direction
Rxb	Mxb	Uxb	End B	(Bottom)	X-direction
< Shear Force >					
Rsx	Qsx	Usx	Nonlinear shear spring X-direction		
< Axial Force>					
Dz	Nz	U			
< Multi-spring > (springs 11-15 in a wall panel)					
C11D(a)	C11F(a)	C11U(a)	End A	Concrete Spring 11	
C12D(a)	C12F(a)	C12U(a)	End A	Concrete Spring 12	
C13D(a)	C13F(a)	C13U(a)	End A	Concrete Spring 13	
C14D(a)	C14F(a)	C14U(a)	End A	Concrete Spring 14	
C15D(a)	C15F(a)	C15U(a)	End A	Concrete Spring 15	
S11D(a)	S11F(a)	S11U(a)	End A	Steel Spring 11	
S12D(a)	S12F(a)	S12U(a)	End A	Steel Spring 12	
S13D(a)	S13F(a)	S13U(a)	End A	Steel Spring 13	
S14D(a)	S14F(a)	S14U(a)	End A	Steel Spring 14	
S15D(a)	S15F(a)	S15U(a)	End A	Steel Spring 15	
C11D(b)	C11F(b)	C11U(b)	End B	Concrete Spring 11	
C12D(b)	C12F(b)	C12U(b)	End B	Concrete Spring 12	
C13D(b)	C13F(b)	C13U(b)	End B	Concrete Spring 13	
C14D(b)	C14F(b)	C14U(b)	End B	Concrete Spring 14	
C15D(b)	C15F(b)	C15U(b)	End B	Concrete Spring 15	
S11D(b)	S11F(b)	S11U(b)	End B	Steel Spring 11	
S12D(b)	S12F(b)	S12U(b)	End B	Steel Spring 12	
S13D(b)	S13F(b)	S13U(b)	End B	Steel Spring 13	
S14D(b)	S14F(b)	S14U(b)	End B	Steel Spring 14	
S15D(b)	S15F(b)	S15U(b)	End B	Steel Spring 15	

### In case of Vertical Spring

< Axial Force >		
Disp.	Force.	Ductility Factor
Dz	Fz	Uz

### In case of Base Isolator

< Shear Force and Axial Force >		
Disp.	Force.	Ductility Factor
Dx	Qx	Ux
Dy	Qy	Uy
Dv	Fv	
		X-direction (Shear)
		Y-direction (Shear)
		Z-direction (Axial)

### In case of Damper and Nonstructural Wall

< Shear Force >		
Disp.	Force.	Ductility Factor
Dx	Qx	Ux
		X-direction

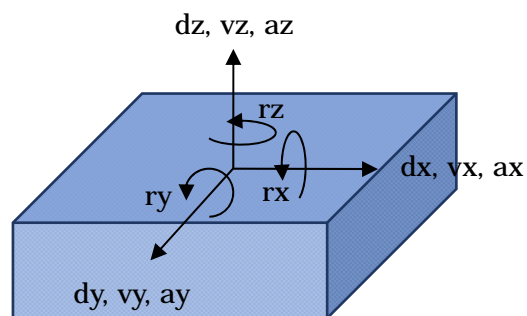
[13] "response\_floor01.csv ..."

In this file, the responses of the center of gravity of 3D rigid floor are saved.


The 3D rigid floor is used in the following cases:

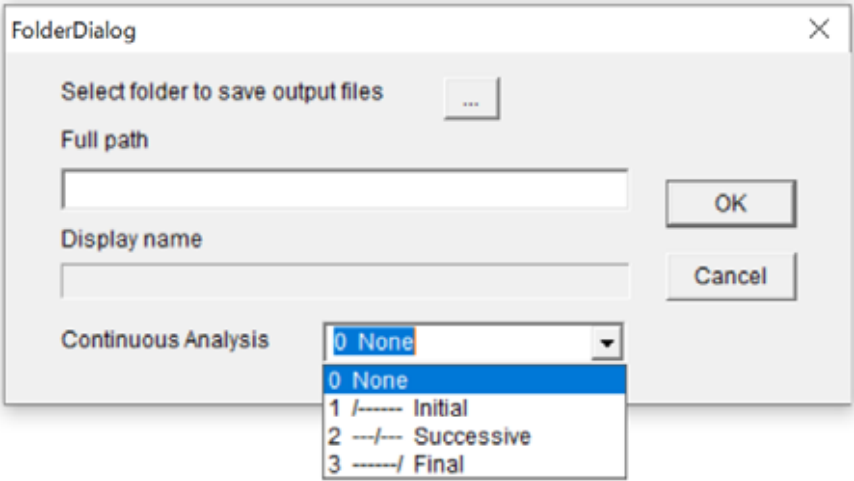
- In case "Ground Spring" (Sway and Rocking) is considered in "Option" > "Member" menu, automatically, the foundation floor is considered 3D rigid.
- In case 3D rigid is selected in "Option" > "Member" menu.

	displacement	rotation	velocity	acceleration
t	dx(cm) dy(cm) dz(cm)	rx(rad) ry(rad) rz(rad)	vx(cm) vy(cm) vz(cm)	ax(gal) ay(gal) az(gal)



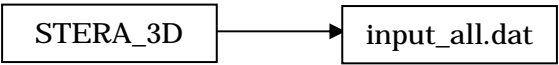
11 Continuous Analysis

When you push the “Save Data” button () during Dynamic



Please select condition of continuous analysis from the menu:

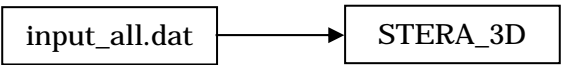
- 0 None                      No continuous analysis (default)
- 1 /----- Initial              Initial analysis  
   (save building status after the analysis)



- 2 ---/--- Successive              Successive analysis  
   (read previous building status, then save it after the analysis)



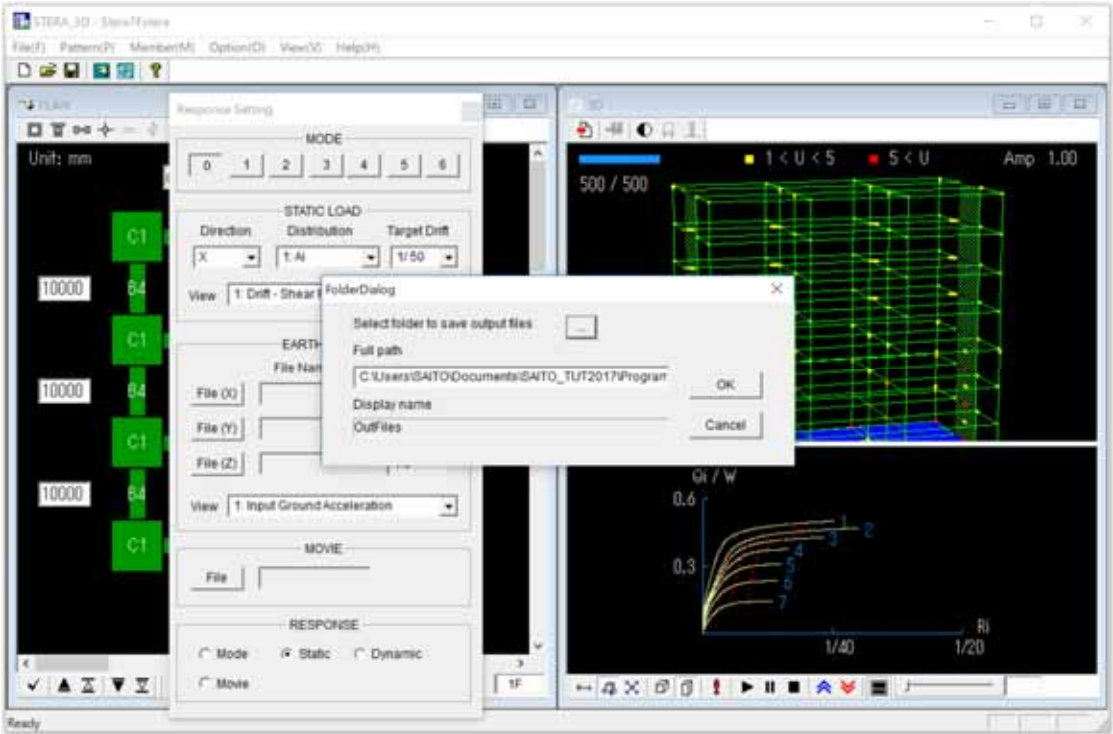
- 3 -----/ Final              Final analysis  
   (read building status)




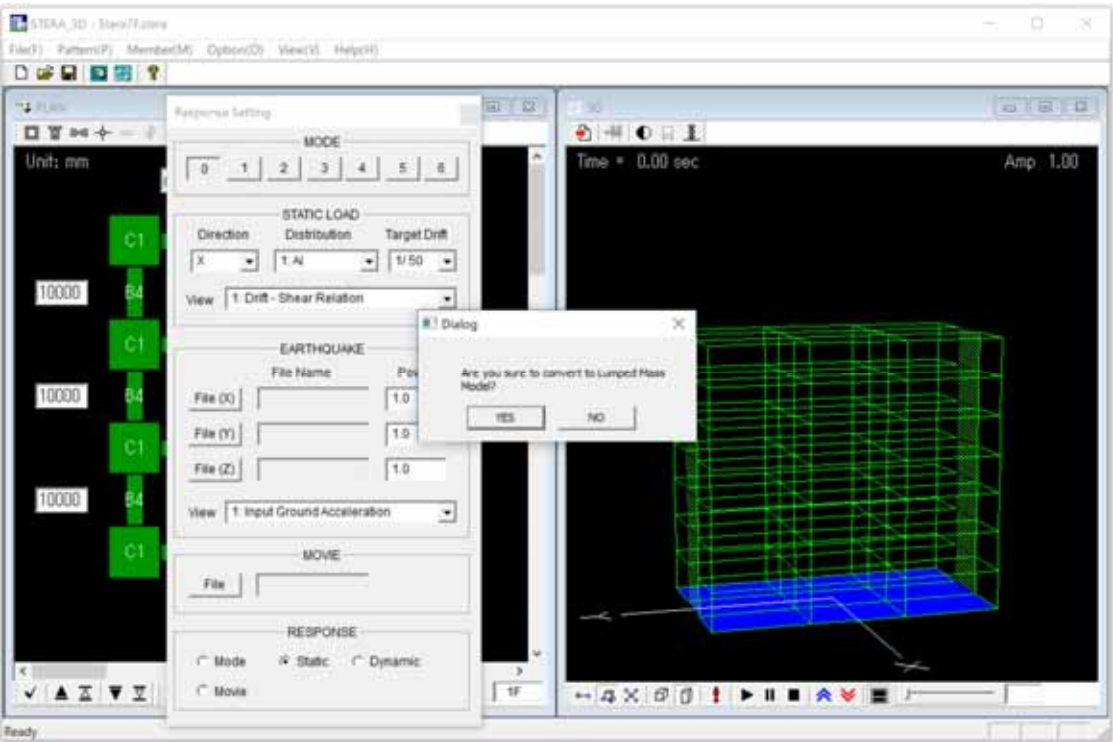
In continuous analysis, it is recommended to change the folder to save output files in each analysis, since all output files will be overwritten.

12 Automatic generation of Lumped Mass Model (LMM)

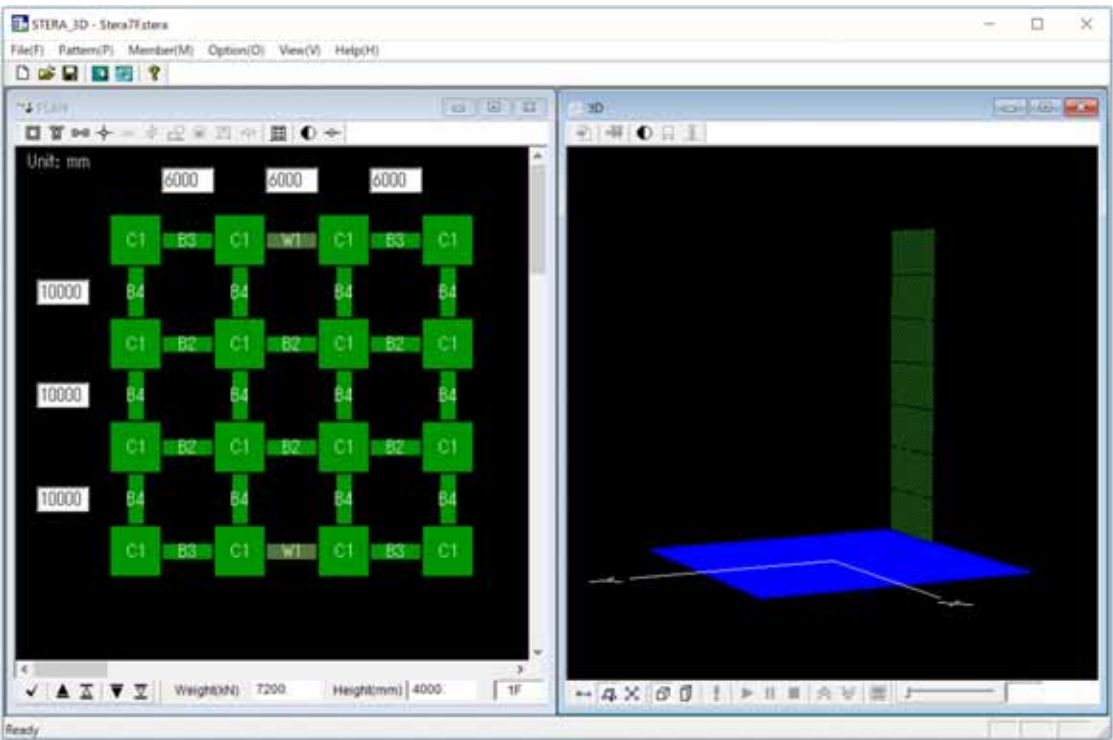
Save results of static push-over analysis.



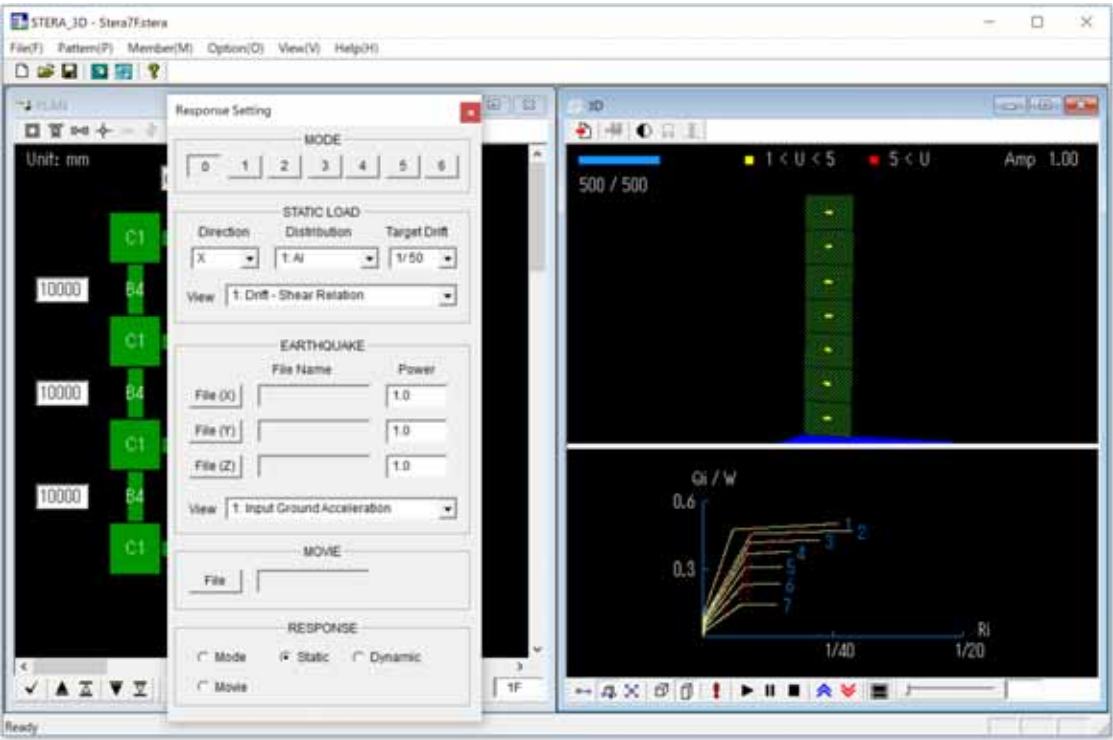
When you click the icon , a message appears to convert to Lumped Mass Model.



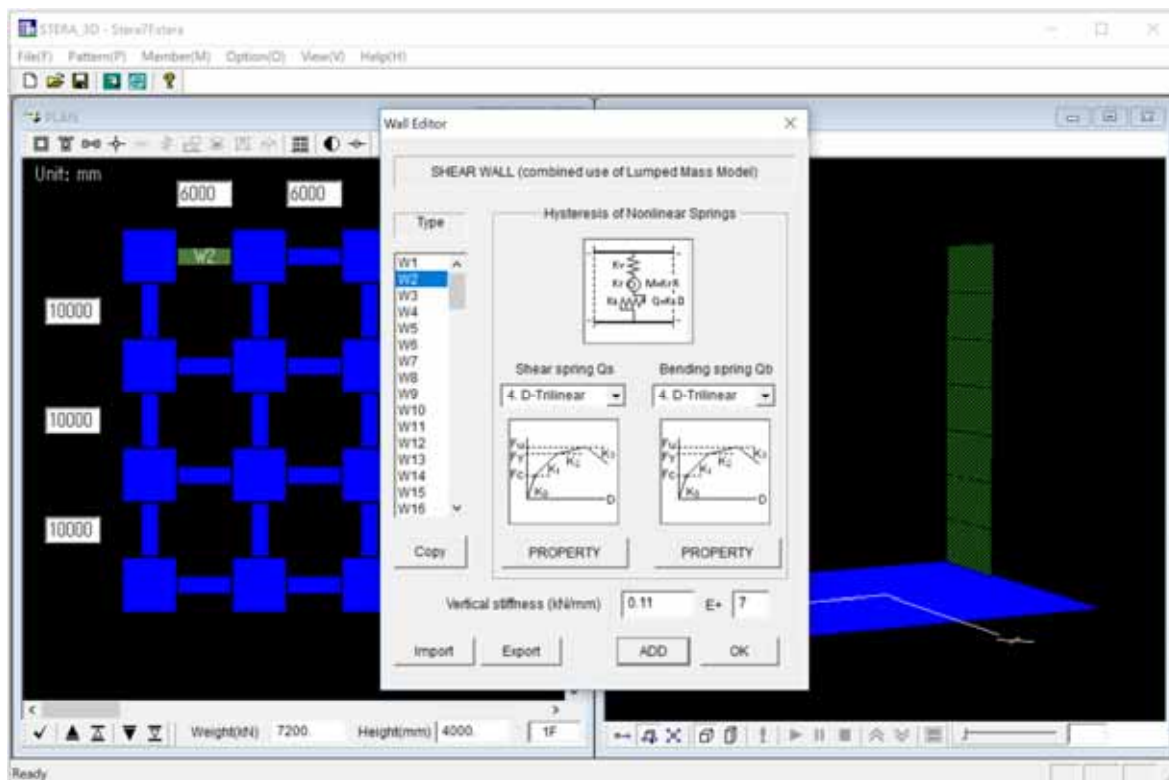
If you answer YES, an equivalent LMM will be automatically created.



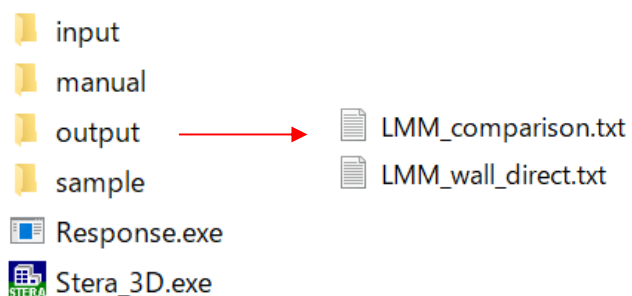
If you conduct static push-over analysis, you will see that the force-deformation relationship of each story is modeled as a tri-linear hysteresis model.



- The element model of each story is “Wall with direct input” with shear and bending springs. The hysteresis of each spring is modeled as non-linear tri-linear model.
- The element type number is “W2” for 1F, “W3” for 2F, ..., etc.
- The restrained freedom automatically set as 2467 (X-direction only).
- Floor Slab of each story is automatically set as 3D Rigid.

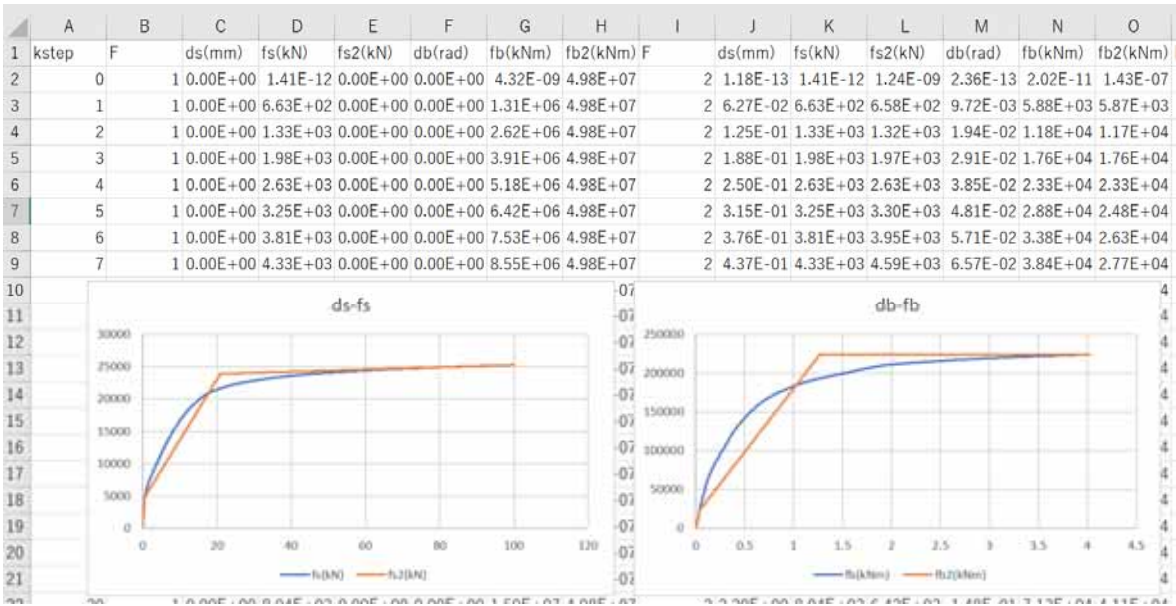


In “./output” folder, “LMM\_comparison.txt” and “LMM\_wall\_direct.txt” are automatically created.





“LMM\_comparison.txt” includes the data of story shear force and story displacement relationships both for frame model and LMM under static push-over analysis.



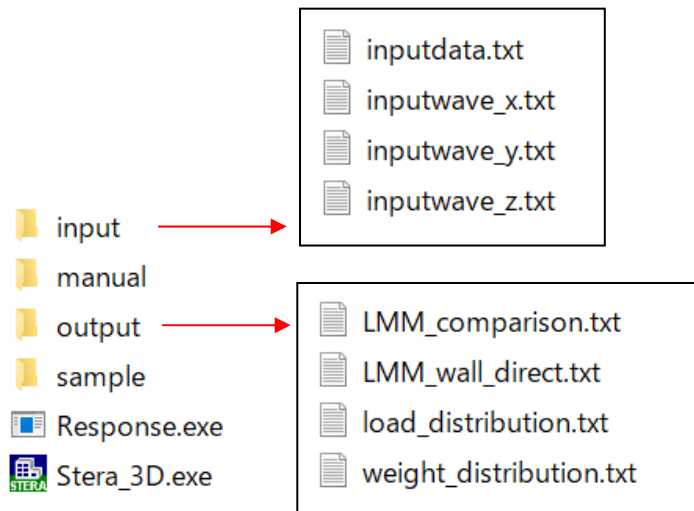
“LMM\_wall\_direct.txt” includes the parameters of hysteresis for shear and bending springs of Wall (direct input) model as described in 6.2 using the same format of “Data\_wall\_direct.txt” by [Export] button.

Please refer “Technical Manual” for the detail for how to obtain the equivalent tri-linear skelton.



## 13 Command line execution

After you save the results of analysis as described in 10.2, there are text files automatically generated in the folder of STERA\_3D as shown below:



where

inputdata.txt	Input data of structure
inputwave_x.txt	Ground acceleration data in x (horizontal)-direction (see 9.1)
inputwave_y.txt	Ground acceleration data in y (horizontal)-direction (see 9.1)
inputwave_z.txt	Ground acceleration data in z (vertical)-direction (see 9.1)

also

load_distribution.txt	Horizontal load distribution in static analysis (see 7.2)
out_comparison.txt	Comparizon of Q-D between frame and LMM (see 11)
out_parameter.txt	Q-D parameters of LMM (see 11)
weight_distribution.txt	Weight distribution in a floor node (see 7.1)

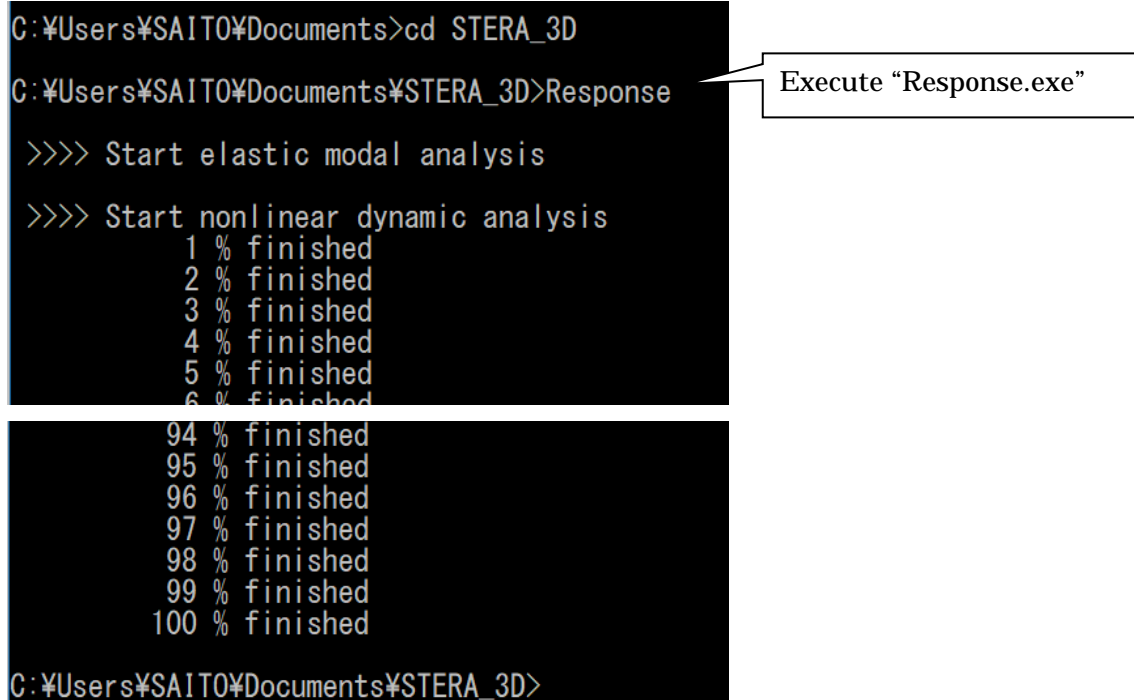
When you execute “Response.exe”, the analysis will start using the following files in “input” folder as input files:

```

inputdata.txt
inputwave_x.txt
inputwave_y.txt
inputwave_z.txt
  
```

That is, you can execute the program by command line without using STERA\_3D.

From the command prompt,



```

C:\Users\SAITO\Documents>cd STERA_3D
C:\Users\SAITO\Documents\STERA_3D>Response
>>>> Start elastic modal analysis
>>>> Start nonlinear dynamic analysis
      1 % finished
      2 % finished
      3 % finished
      4 % finished
      5 % finished
      6 % finished
      94 % finished
      95 % finished
      96 % finished
      97 % finished
      98 % finished
      99 % finished
     100 % finished
C:\Users\SAITO\Documents\STERA_3D>
  
```

For example, let's make a batch file (test.bat) to replace the earthquake ground acceleration data as

Earth\_NS.txt  
Earth\_EW.txt  
Earth\_UD.txt

test.bat

```

@echo off
copy    .\Earth_NS.txt    .\input\inputwave_x.txt
copy    .\Earth_EW.txt    .\input\inputwave_y.txt
copy    .\Earth_UD.txt    .\input\inputwave_z.txt
Response
  
```

If you double click "test.bat", the new analysis will start using new input waves.