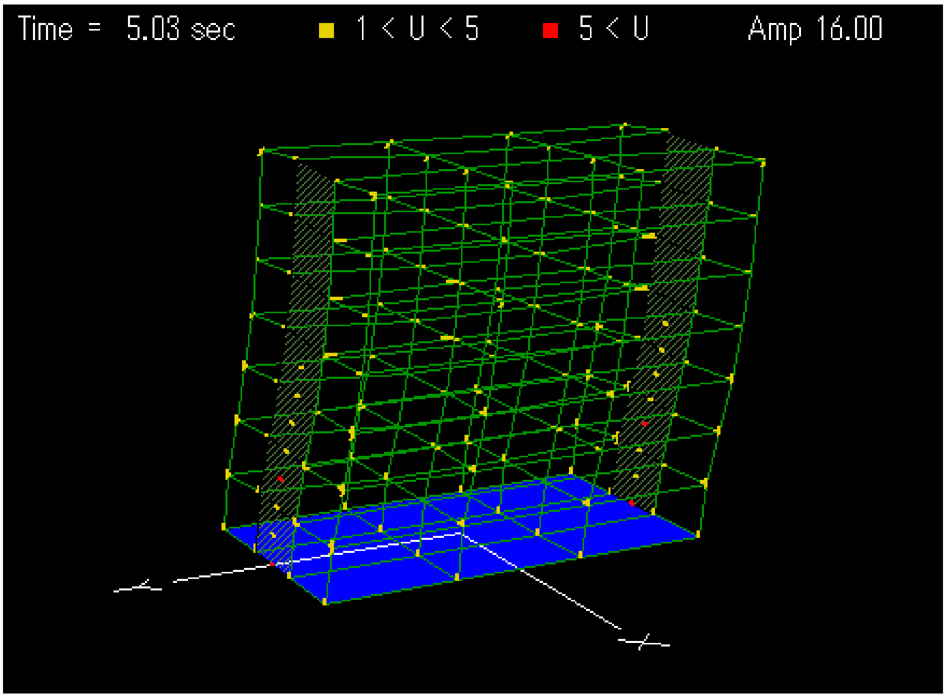
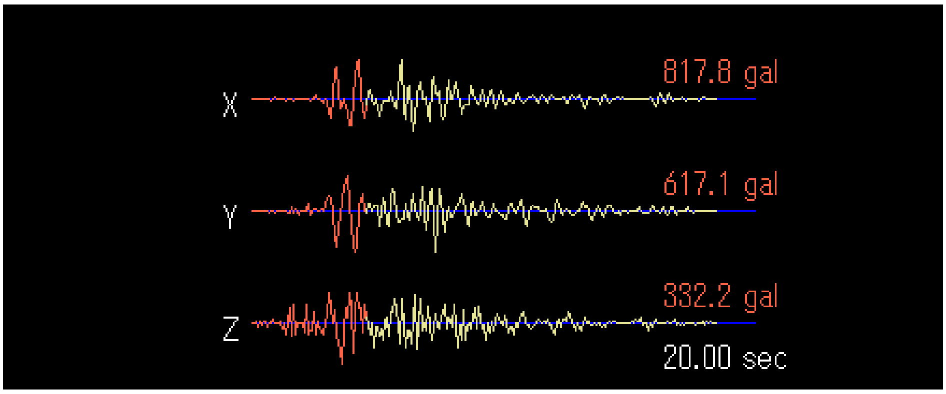
STERA 3D *ver.11.5*

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
Professor, Dr. of Engineering,
Toyohashi University of Technology,
Japan

Update history

2016/11/12 STERA_3D Ver.8.5 is uploaded.

- Minor change of input data for Wall (direct input)

2016/12/03 STERA_3D Ver.8.6 is uploaded.

- Minor error in static analysis using Mode distribution is fixed.

2016/12/11 STERA_3D Ver.8.7 is uploaded.

- Modification of stiffness degradation factor at the yield point of beam elements is adopted (see the detail in Technical Manual).

2016/12/25 STERA_3D Ver.8.8 is uploaded.

- Bugs of masonry element and connection panel were fixed.

2017/01/18 STERA_3D Ver.8.9 is uploaded.

- Effective modal mass is shown in Modal analysis
- You can set a different mass value in each node

2017/03/20 STERA_3D Ver.9.0 is uploaded.

- You can set the parameters of band-pass filter to get the ground displacement.

2017/08/01 STERA_3D Ver.9.1 is uploaded.

- Minor change of the format of output file

2017/09/11 STERA_3D Ver.9.2 is uploaded.

- Hardening model is added for NRB of Seismic Isolator

2017/10/08 STERA_3D Ver.9.3 is uploaded.

- Ground spring is added.

2017/10/24 STERA_3D Ver.9.4 is uploaded.

- The default setting of “upper beam” of damper and masonry is changed to be “rigid” beam instead of “none”.

2017/11/27 STERA_3D Ver.9.6 is uploaded.

- Fixed a mistake in mass setting of Ver. 9.4. Note that mass is set correctly in Ver. 9.3.

2019/02/03 STERA_3D Ver.10.0 is uploaded.

- You can set a vibrator on a floor to shake the building.
- You can execute the program from command line.
- Automatic generator for lumped mass model is implemented.
- You can set an original load distribution for static analysis
- A new nonlinear spring is added for passive damper

2019/05/20 STERA_3D Ver.10.1 is uploaded.

- You can consider radiation damping for ground spring.

2019/07/25 STERA_3D Ver.10.2 is uploaded.

- You can apply dynamic wind forces to the building.

2019/10/08 STERA_3D Ver.10.3 is uploaded.

- The buckling of steel members can be considered.
- You can conduct continuous analysis for dynamic inputs (earthquake and wind).

2020/03/16 STERA_3D Ver.10.4 is uploaded.

- Pile can be considered for ground spring.
- Air spring has been added to the vertical spring.

2020/04/14 STERA_3D Ver.10.5 is uploaded.

- You can set part of the floor to be rigid.

2020/06/11 STERA_3D Ver.10.6 is uploaded.

2020/08/04 STERA_3D Ver.10.7 is uploaded.

- You can select rebar size from the table.

2020/09/24 STERA_3D Ver.10.8 is uploaded.

2021/10/10 STERA_3D Ver.11.0 is uploaded.

- For RC column and RC wall, the nonlinear bending springs independent in x and y directions are introduced.
- For Steel beam, the nonlinear shear spring for hysteresis damper is introduced.
- Damage indices of members are introduced.

2022/08/22 STERA_3D Ver.11.1 is uploaded.

- The input windows for columns and beams (direct input for parameters of hysteresis model) are changed.
- “Base Plate" and "Pendulum Spring" are added to External Spring.

2022/12/14 STERA_3D Ver.11.2 is uploaded.

- FPB (Friction Pendulum Bearing) is added to the seismic isolation elements.

2023/03/10 STERA_3D Ver.11.3 is uploaded.

- The formula of compression strength of Masonry element is changed.

2023/06/06 STERA_3D Ver.11.4 is uploaded.

- Viscoelastic damper is added to the passive damper.
- You can set the restrained freedom of the partial rigid slab at the center of gravity.

2024/07/15 STERA_3D Ver.11.5 is uploaded.

- Viscoelastic damper is added to the shear spring of direct beams.
- Some output files are in csv format.

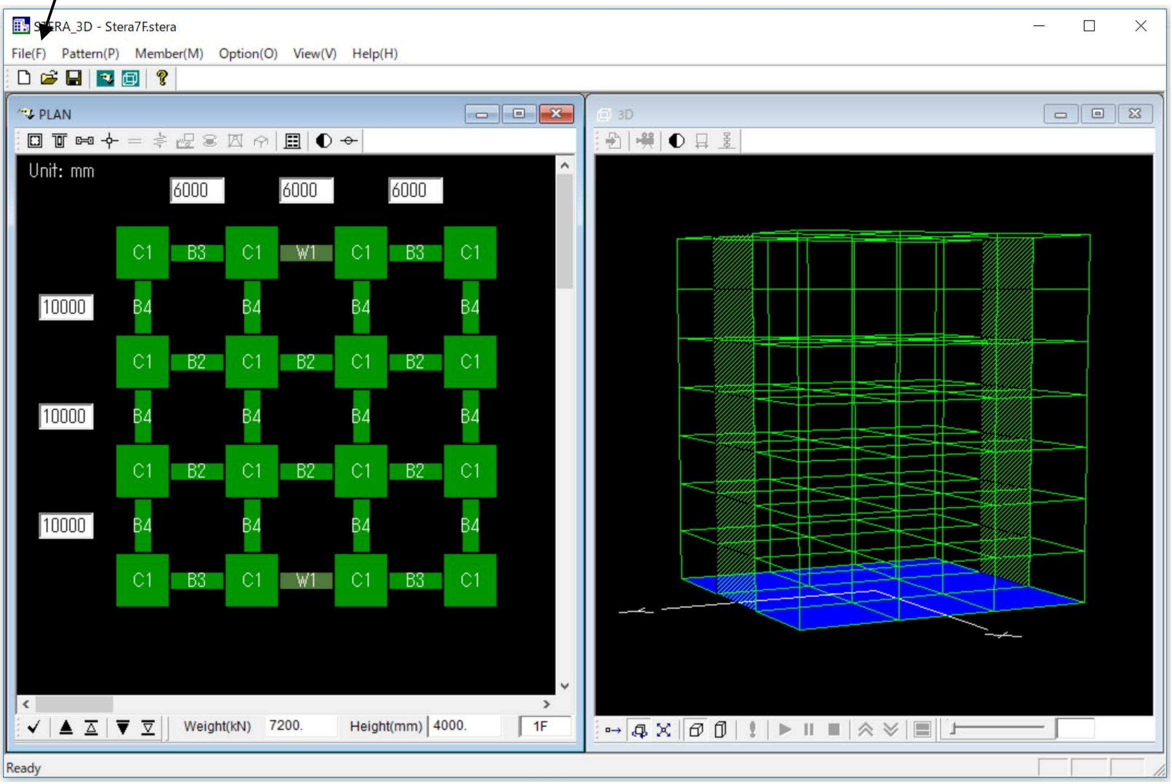
Quick User Manual

READ BUILDING DATA

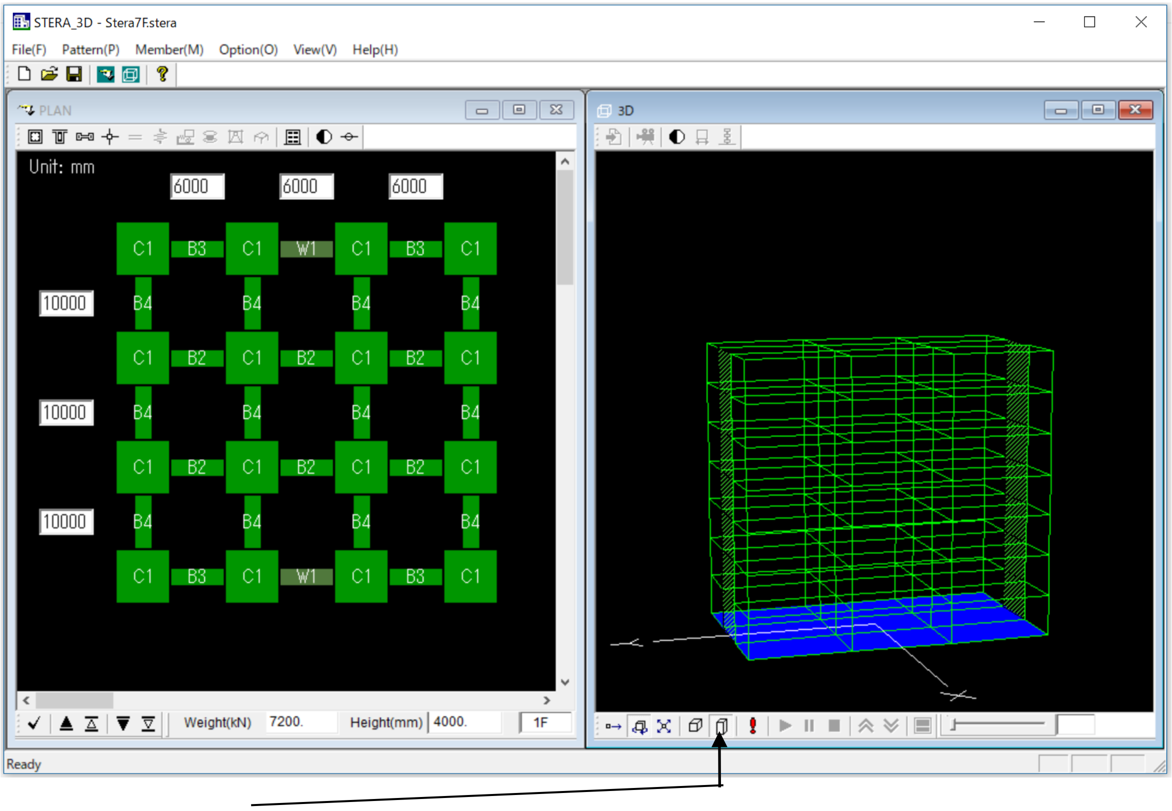
① Double Click

② "File" → "Open"

Select an example building "Structure7F.stera"



MOVE THE BUILDING

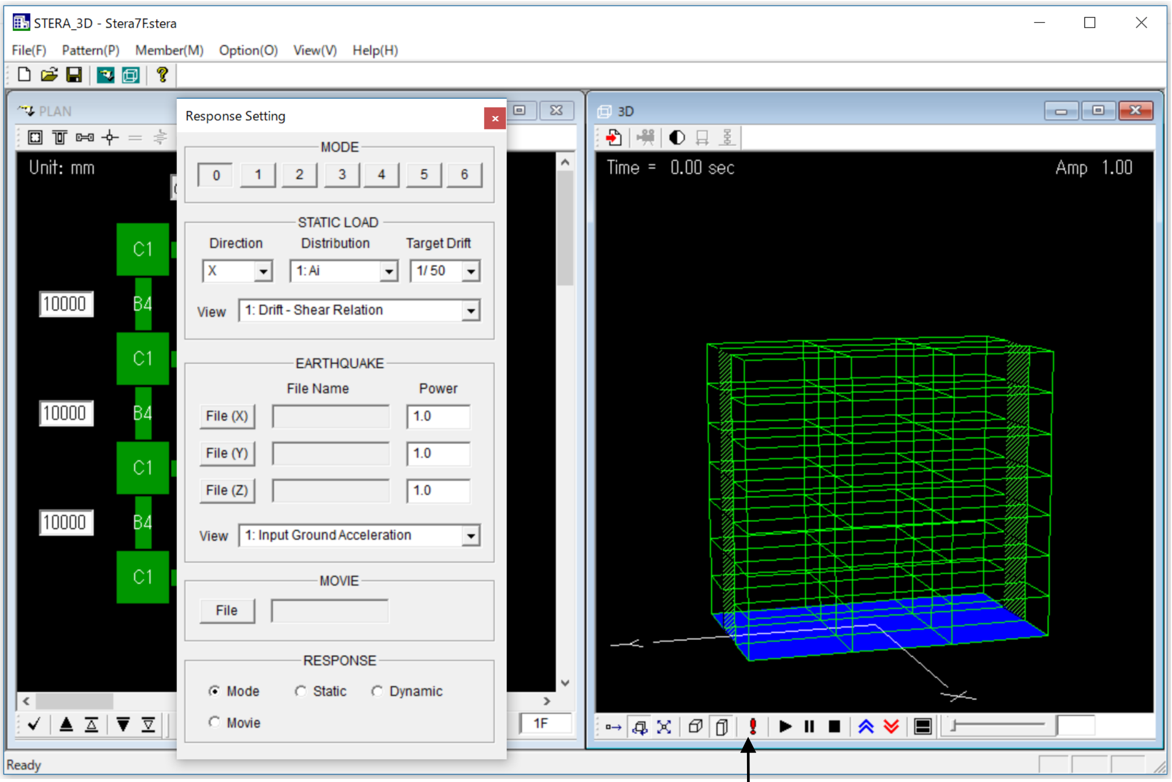


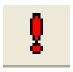
① Click  to be actual size.

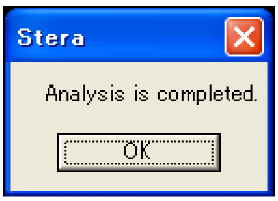
② Drag the right mouse on the image to rotate the building.

③ Drag the left mouse on the image to enlarge and reduce.

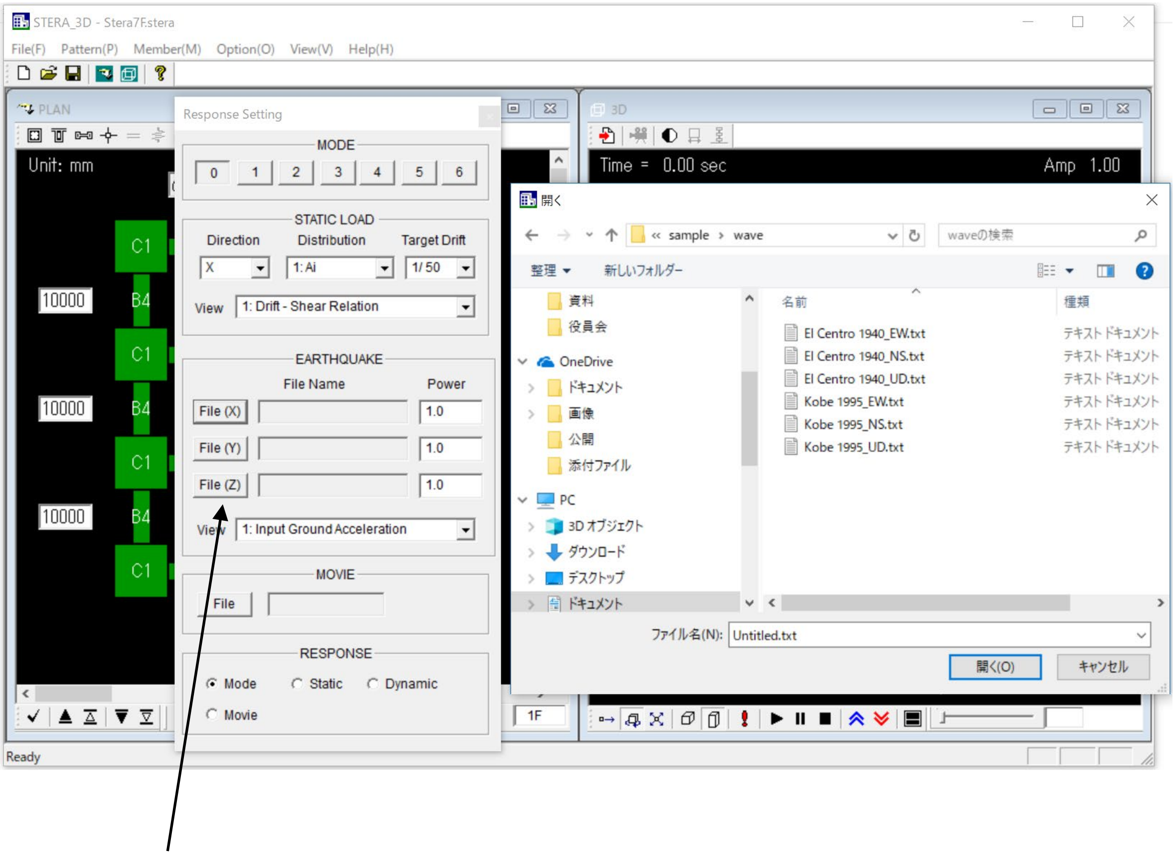
EARTHQUAKE RESPONSE



① Click  to analyze the building.

② After the message, 

"Response Setting Dialog" will appear.



③ 

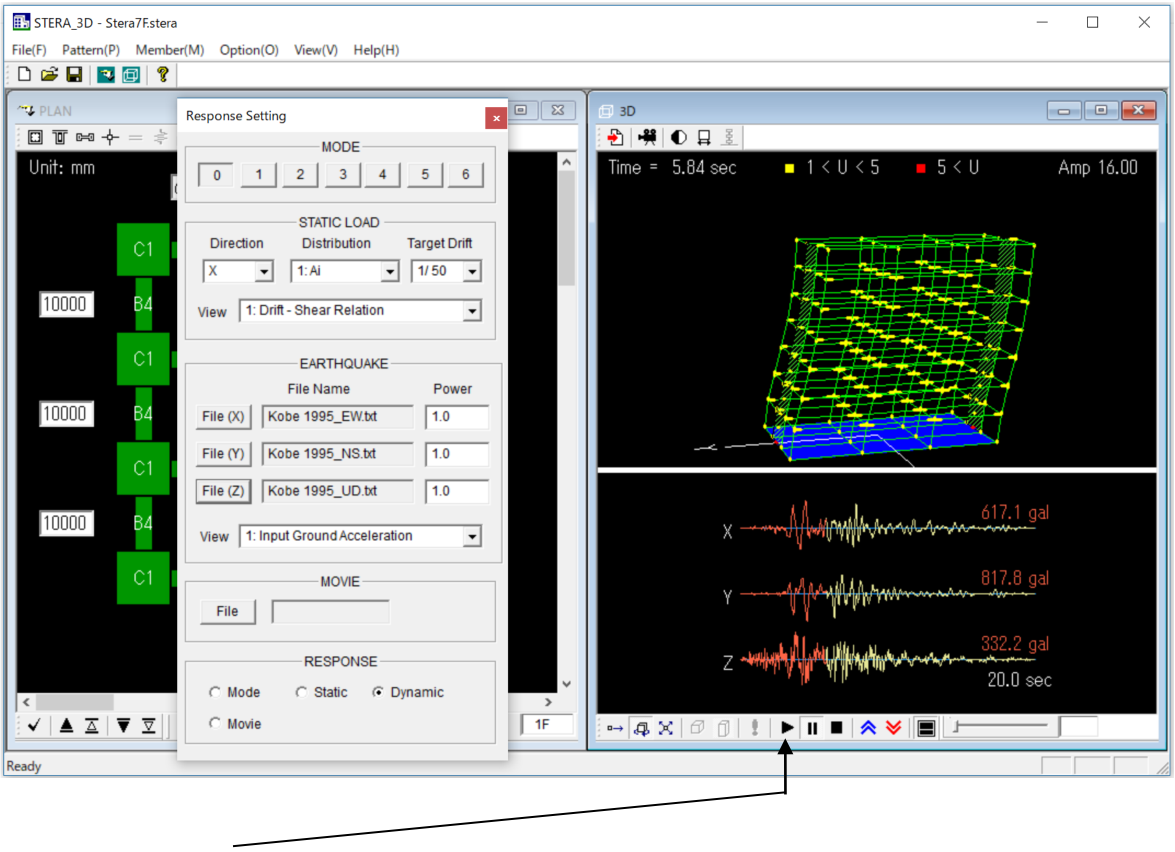
: Select X-direction earthquake data file.
For example, "Kobe_1995_EW".

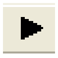
④ 

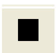
: Select Y-direction earthquake data file.
For example, "Kobe_1995_NS".

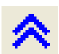
⑤ 

: Select Z-direction earthquake data file.
For example, "Kobe_1995_UD".

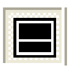


⑥  : Start the response

 : Stop the response

 : Amplify the response

 : Reduce the response

 :: Change the view from double screen to single screen

User Manual

Index

1 Basic Assumptions

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

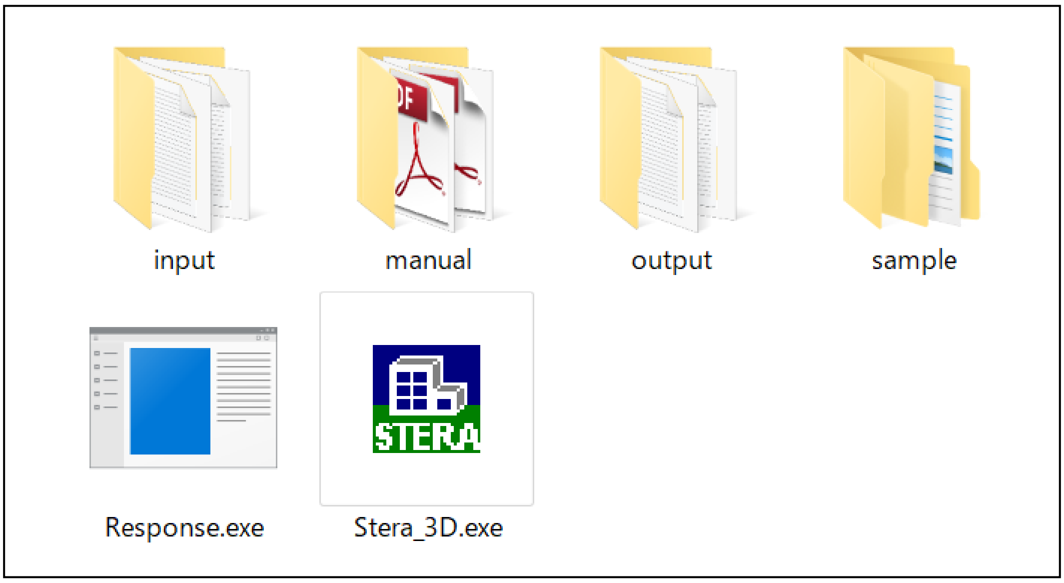
  STERA_user_manual

  STERA_technical_manual

sample/ … Folder for sample

  building/ … Folder for sample building for STERA

  wave/ … Folder for sample input waves

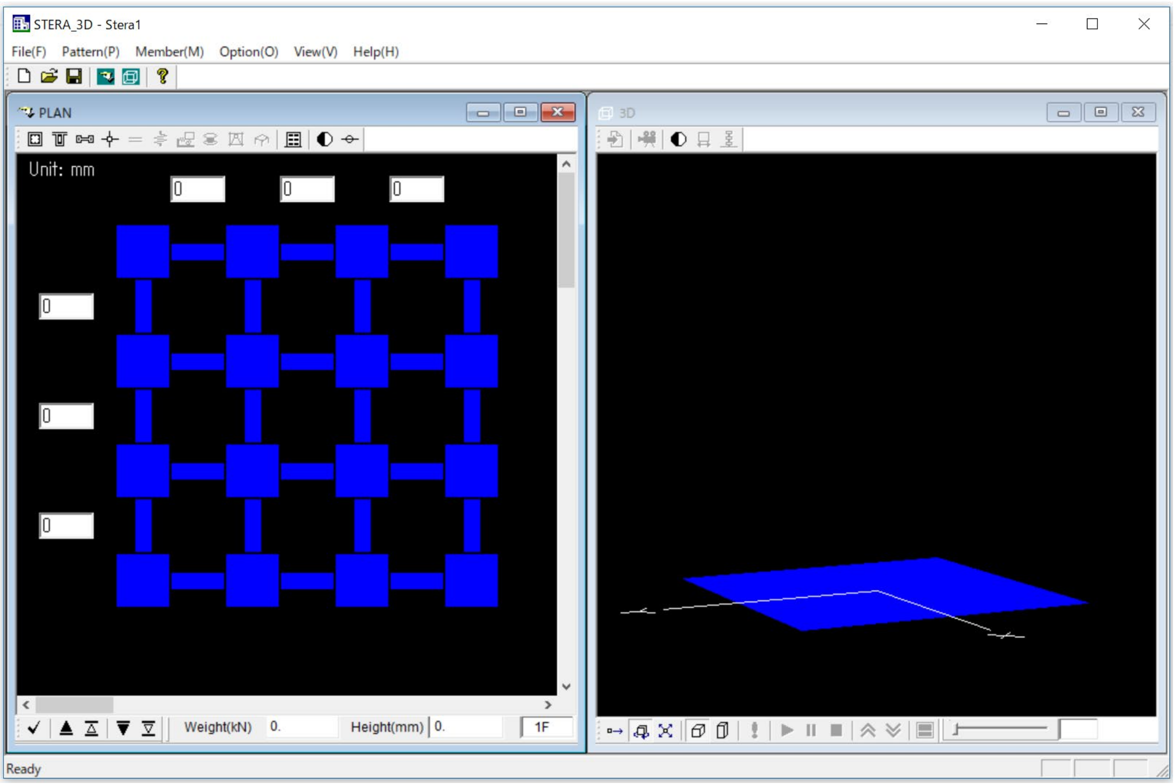


3 Initial View

Please double crick "Stera 3D.exe"

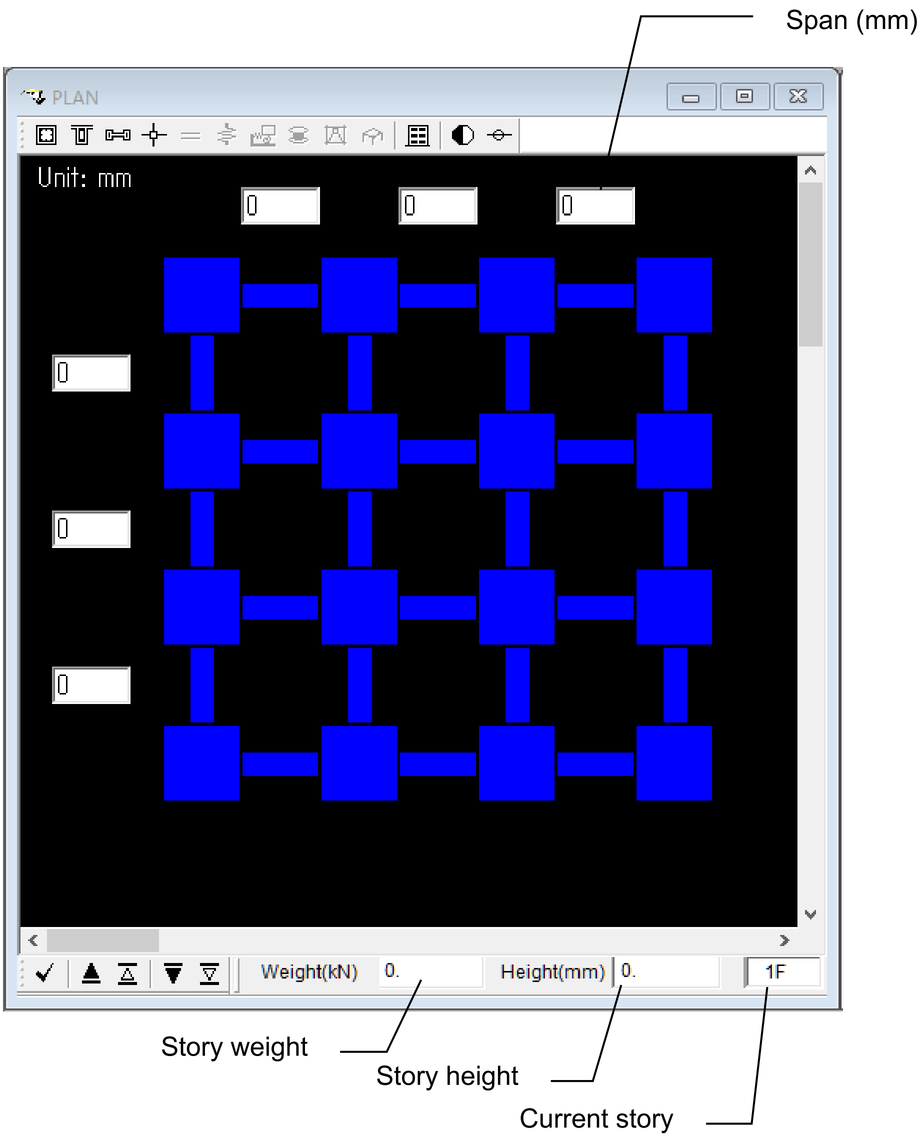
The left view is "PLAN EDIT VIEW" where you input building plan data, and the right view is "3D VIEW" where you can see the building shape and its response after the analysis.

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 1st 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)

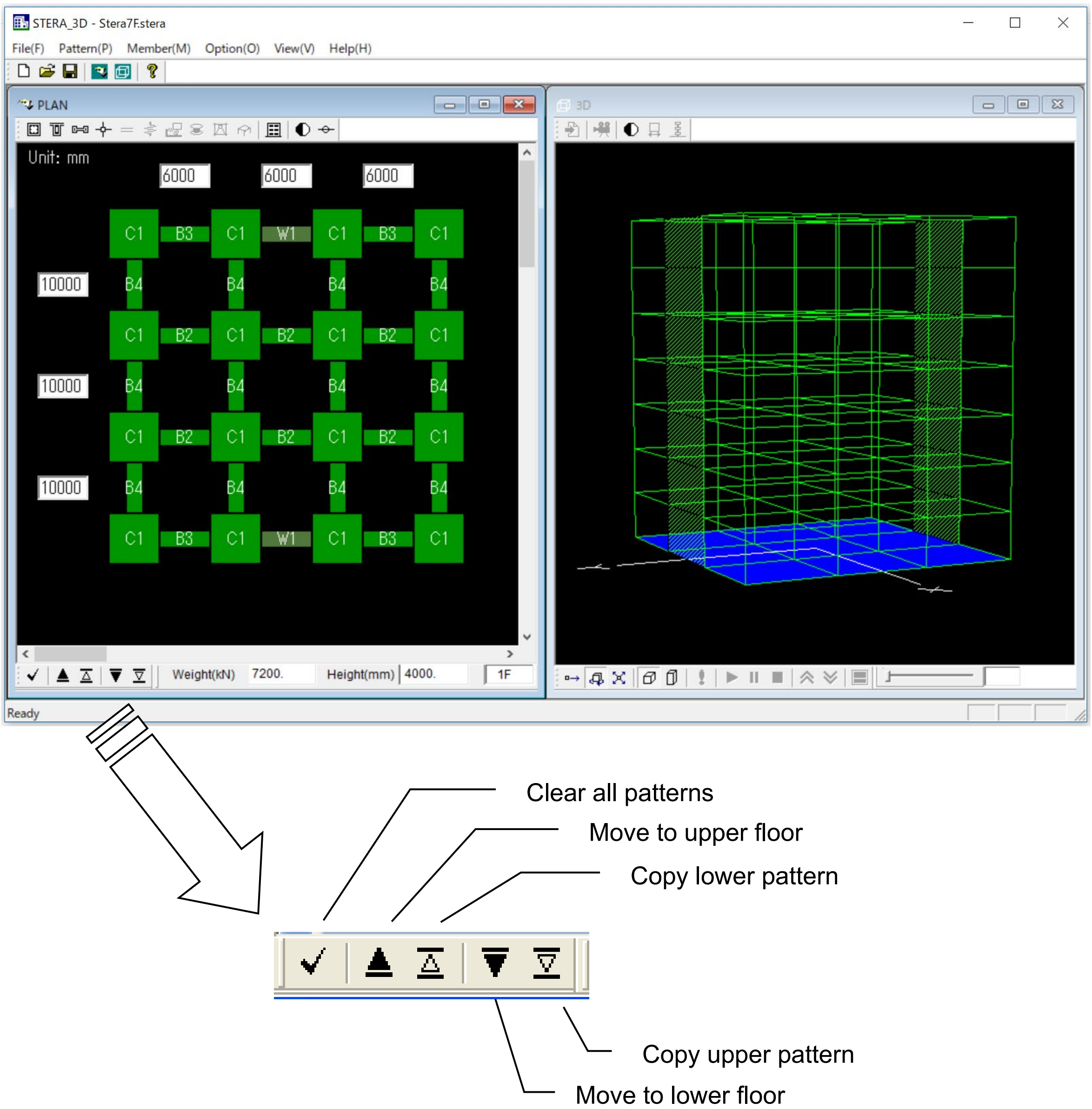
  But, in case of the basement floor (BF), the order is changed as:
  - ✧ Base Spring (brown) → Empty → Base Spring (brown)

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.

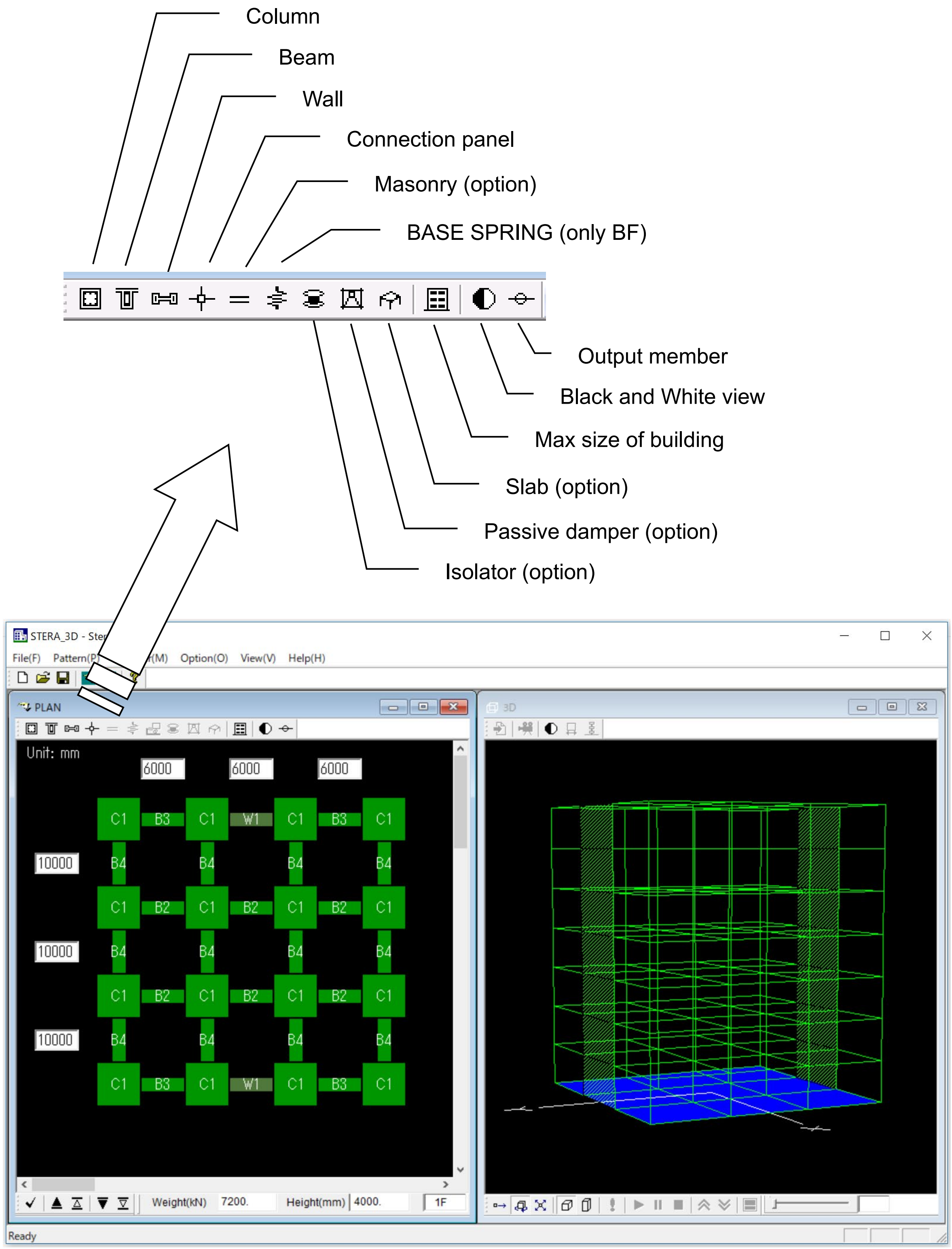
- By dragging your mouse in a region, you can set all the elements in the region at once.
- By clicking the right button of your mouse, you can change the number of element type for column (C1-C100), for beam (B1-B100), and for wall (W1-W100) etc.
- To move to another floor and copy or clear the member 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

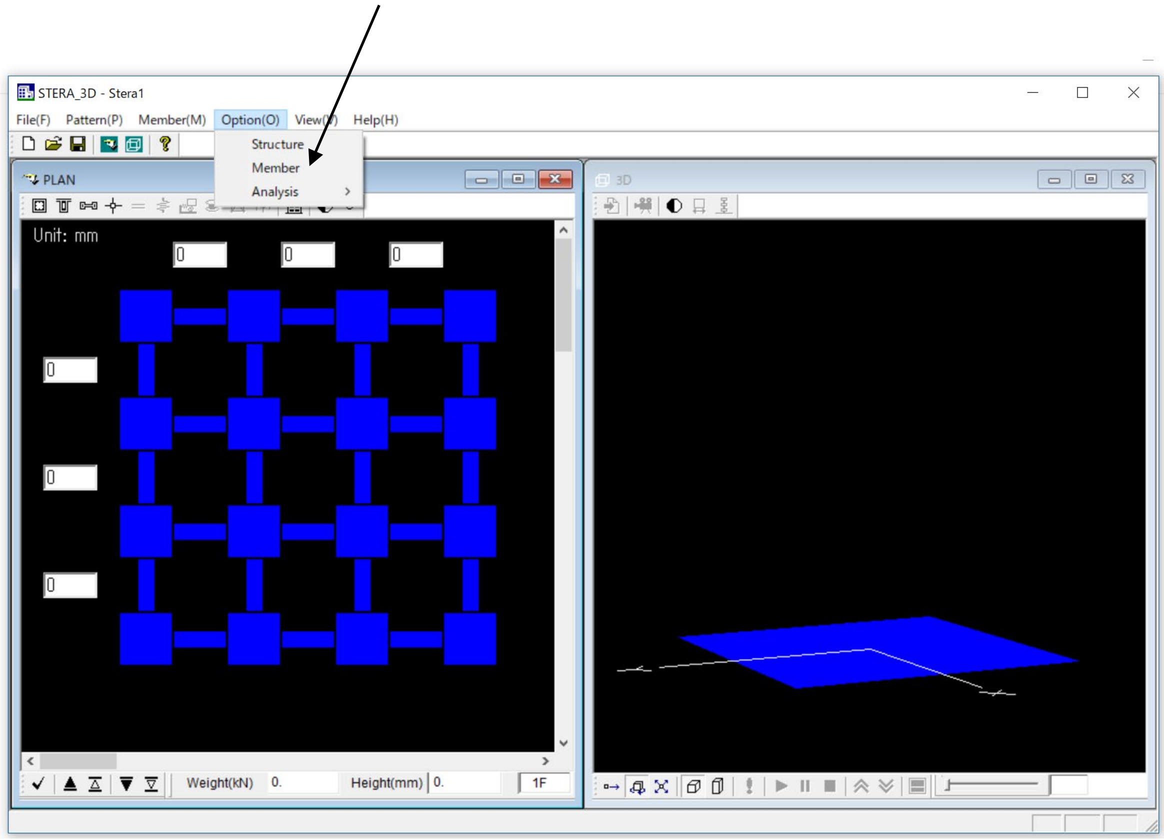


5.2 Activate Element

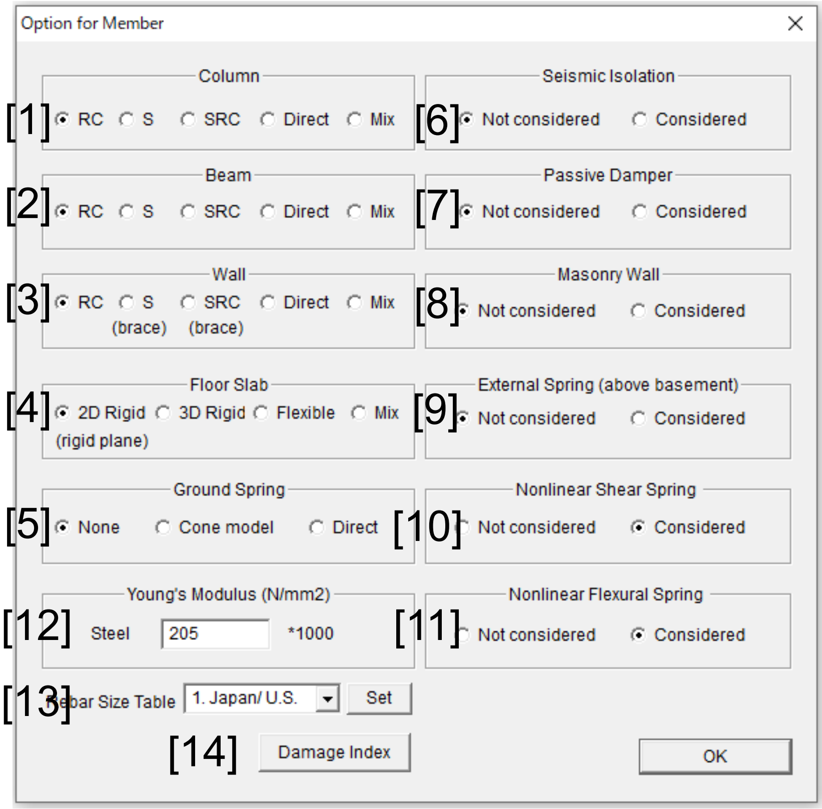
In the default condition,

- All columns, beams and walls are reinforced concrete elements,
- Other elements are non-active in the element menu.

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

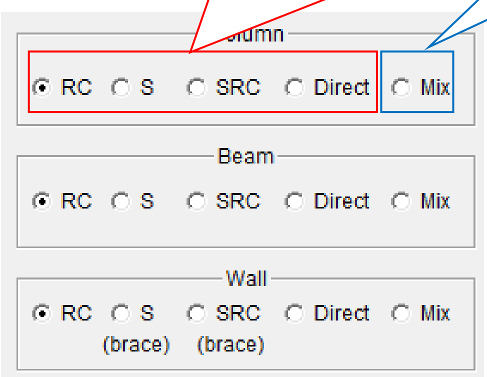


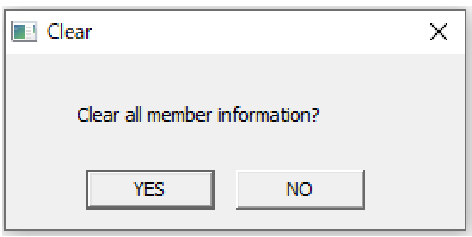
[1] Column, [2] Beam, [3] Wall

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.

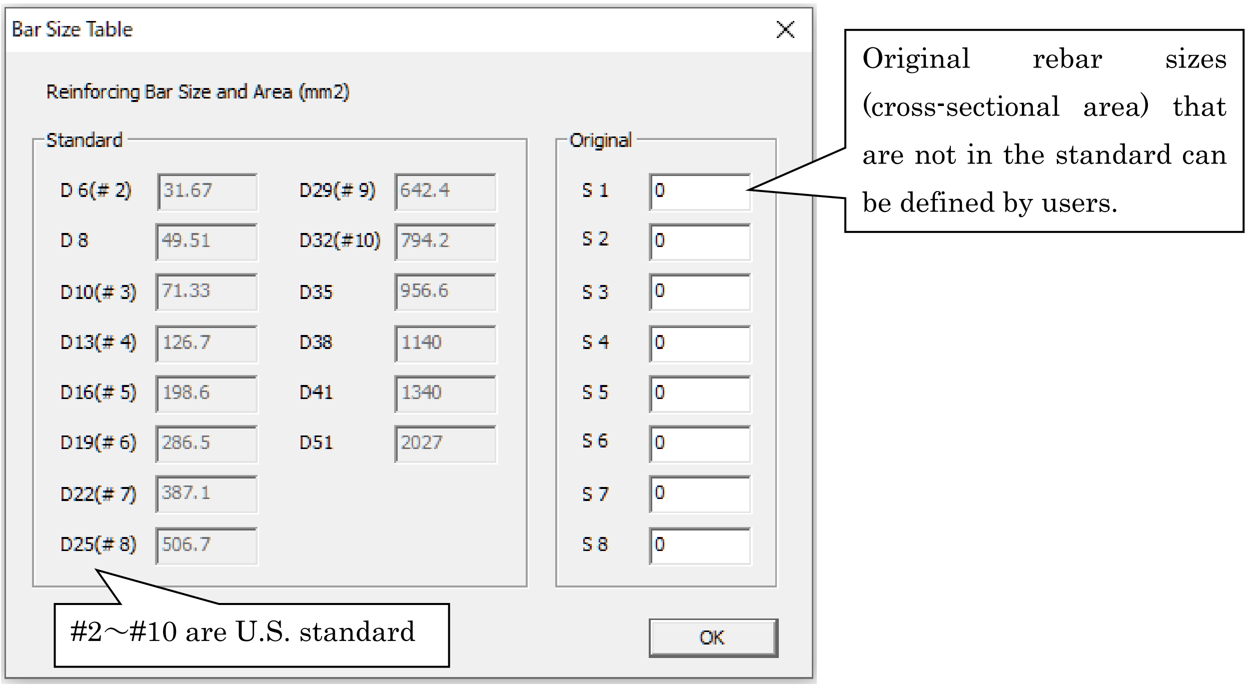
[11] Nonlinear Flexural Spring

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

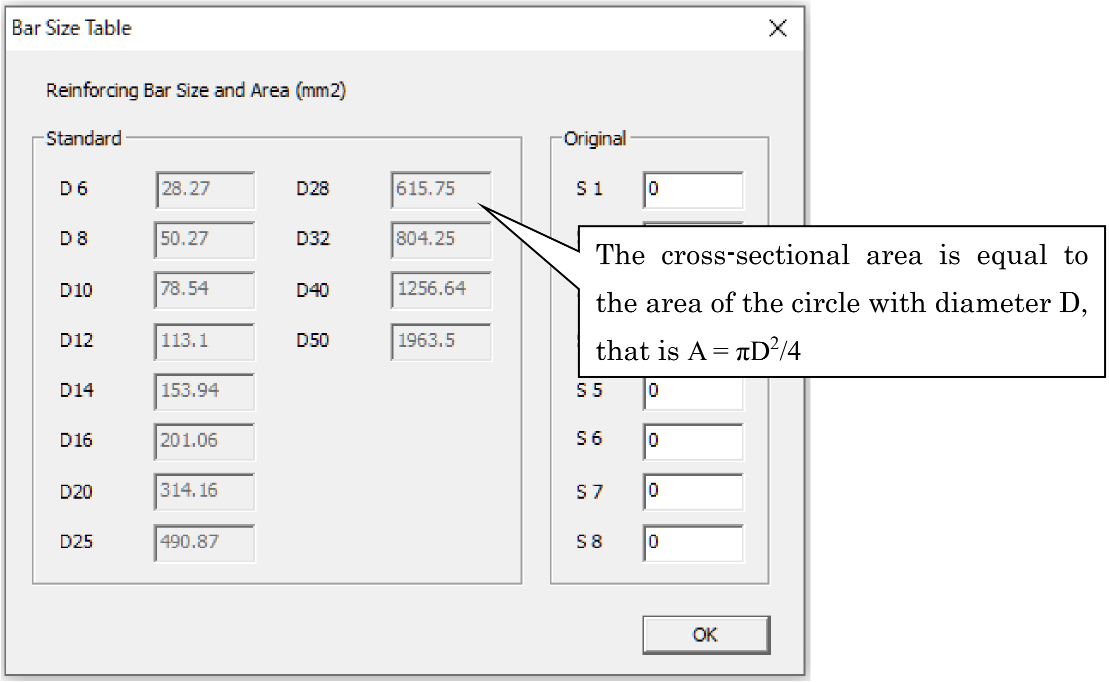
[12] Young's Modulus of Steel

[13] Rebar Size Table

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

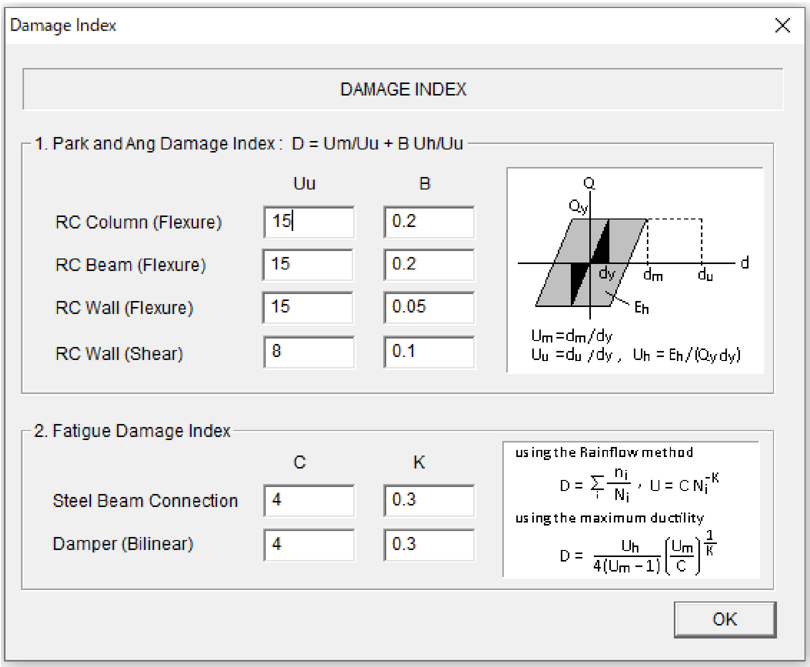


You can select Euro (Eurocode) from the pull-down menu.



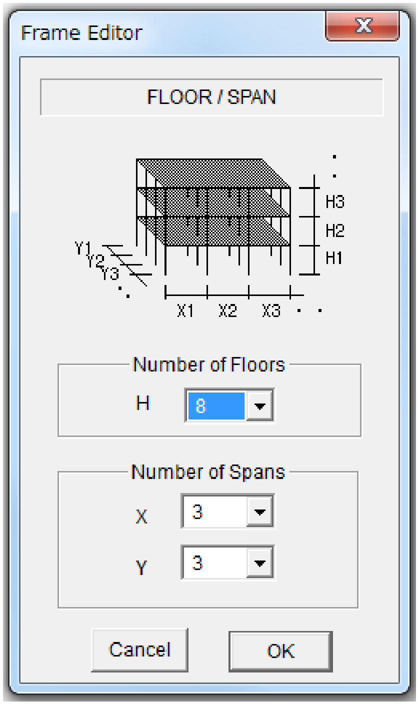
[14] Damage Index

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 ()

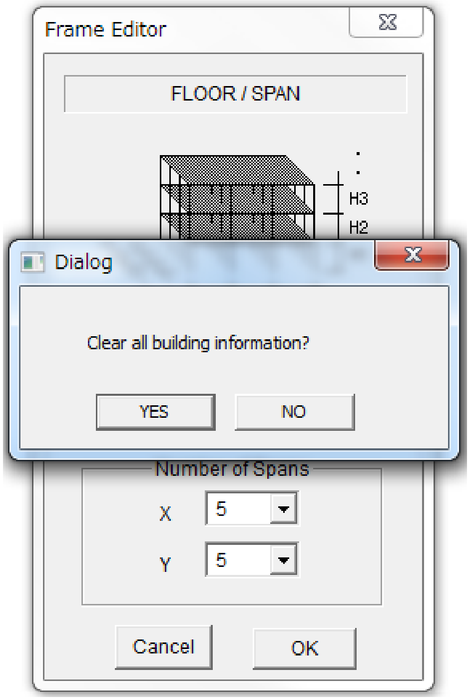


You can change the maximum number of spans and stories of the building. The default setting is

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

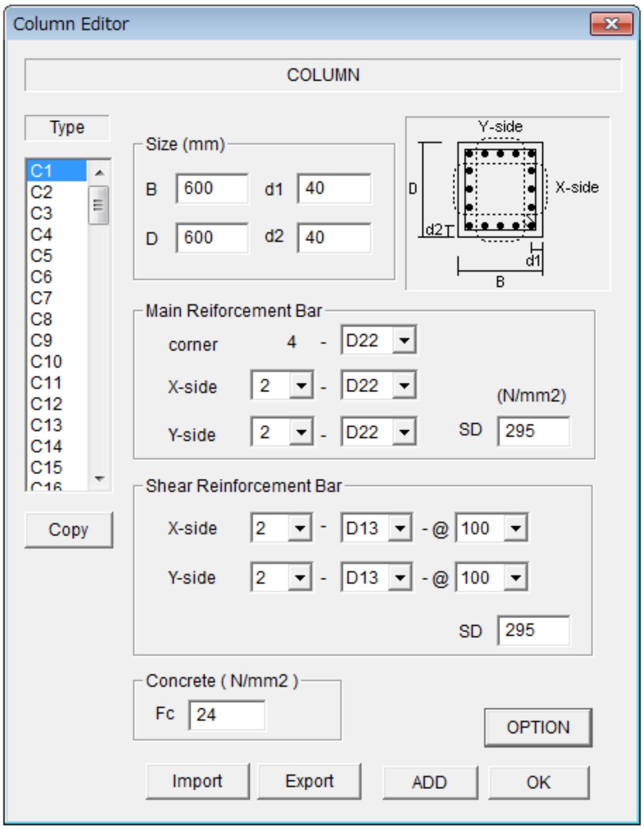


After you select new numbers, a dialog will appear asking if you clear all building information or not. If you select “NO”, you can keep the same building information.

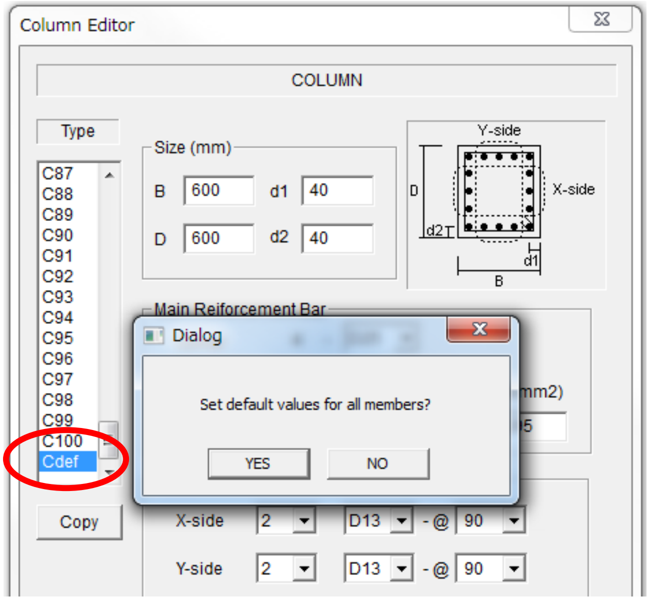
6 Input Element Information

6.1 RC Column

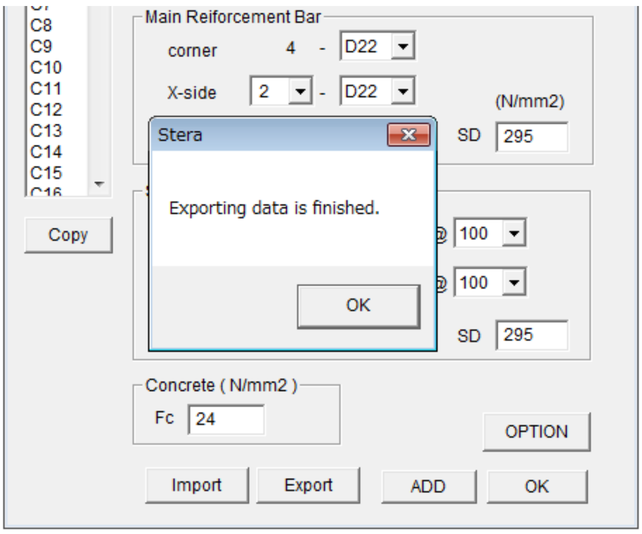
COLUMN ()



- 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.



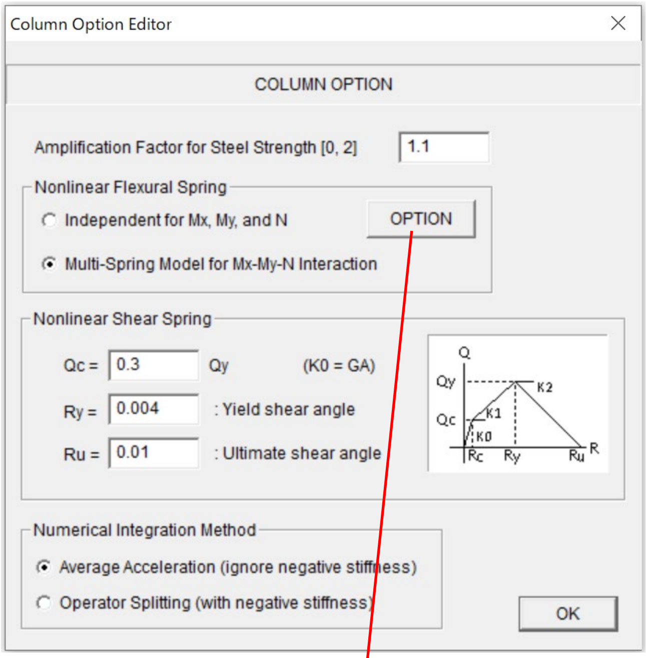
- You can set the default values for all members by selecting the last member type “Cdef”.



- 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.

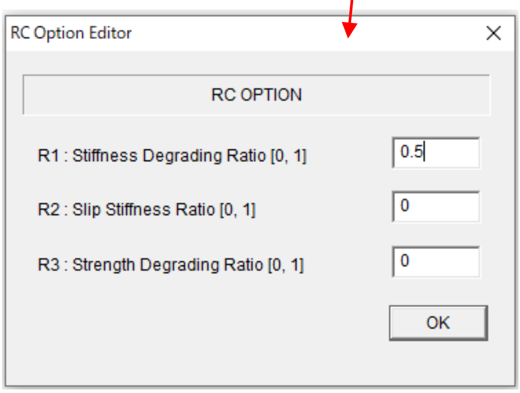
n	Width(mm)	Height(mm)	d1	d2	vsize_C	vno_X
1	600	600	40	40	9	1
2	600	600	40	40	9	1
3	600	600	40	40	9	1
4	600	600	40	40	9	1
5	600	600	40	40	9	1
6	600	600	40	40	9	1
7	600	600	40	40	9	1
8	600	600	40	40	9	1
9	600	600	40	40	9	1
10	600	600	40	40	9	1
11	600	600	40	40	9	1
12	600	600	40	40	9	1



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 Qc is defined as the ratio of the yield strength Qy. The default value is 0.3.
- The default values of yield and ultimate shear deformation angles, Ry and Ru 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.

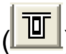


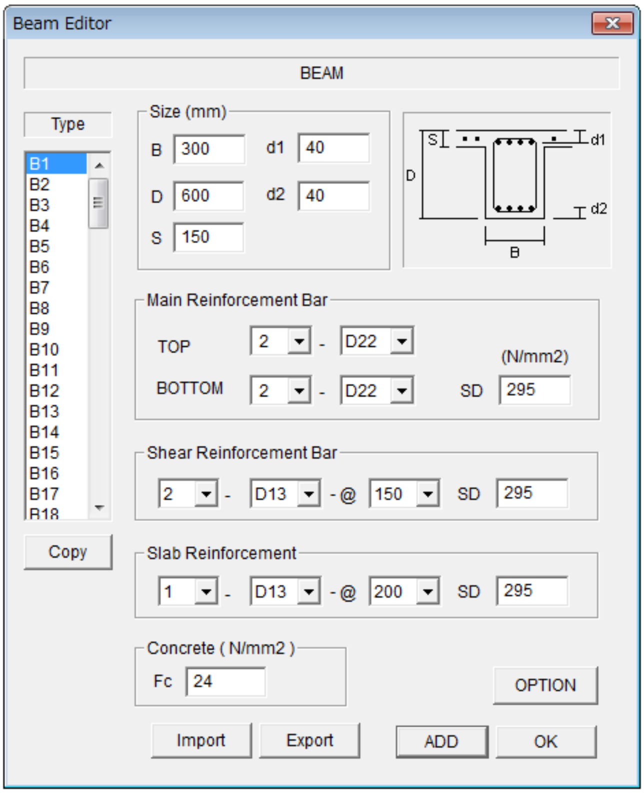
In [OPTION] menu, the parameters to control the shape of hysteresis model are defined as follows:

- 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.

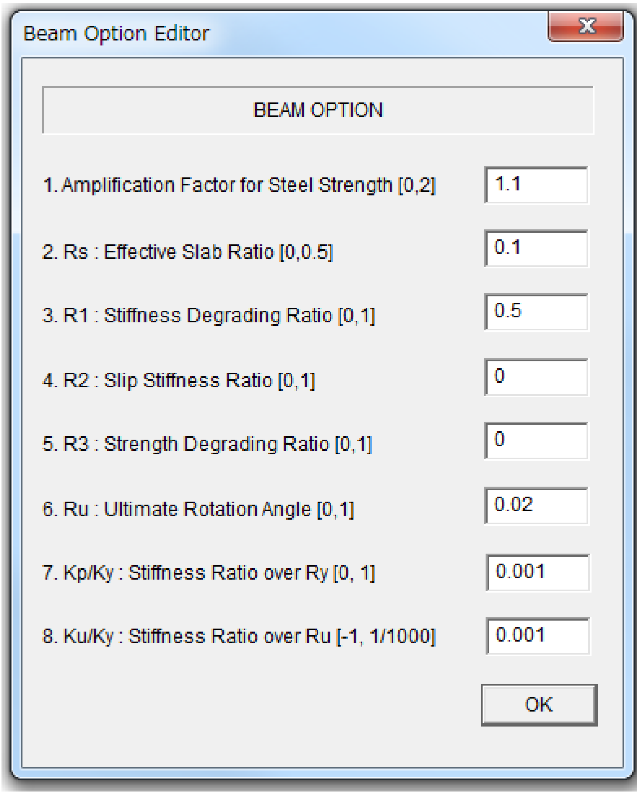
Please refer “Technical Manual” for the detail.

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.



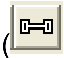
In [OPTION] menu,

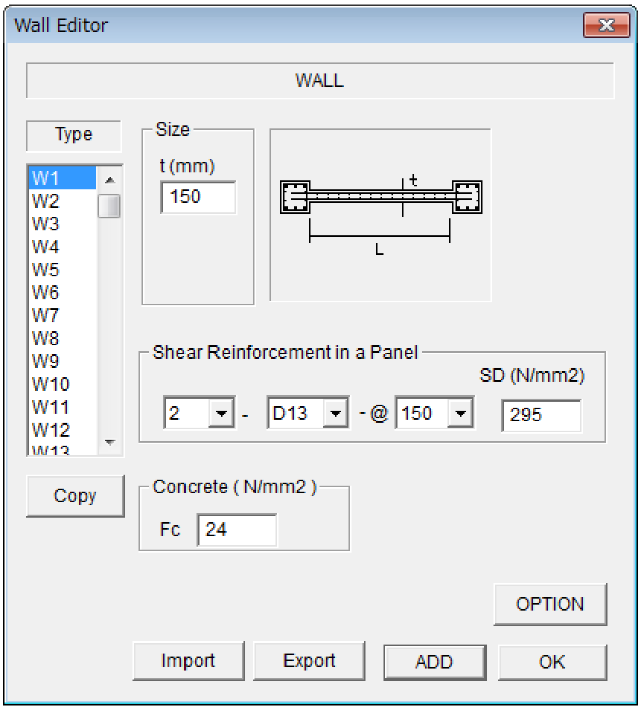
- The default steel strength is assumed to be 1.1 times larger than the nominal strength.
- Rs: the effective slab width to contribute flexural behavior of beam is assumed to be 0.1 times beam length.

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

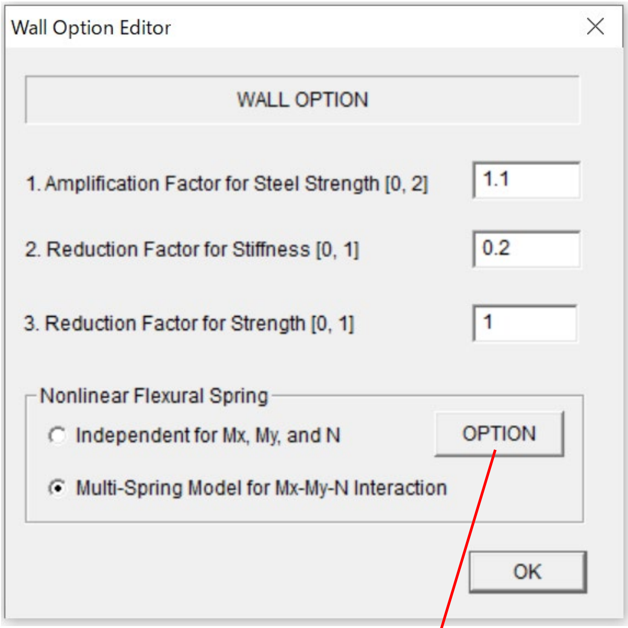
- R1: the default value of stiffness degrading ratio in the trilinear hysteresis is 0.5. (0: no degradation)
- R2: the default value of slip stiffness ratio in the trilinear hysteresis is 0.0 (0: no slip).
- R3: the default value of strength degrading ratio in the trilinear hysteresis is 0.0.
- Ru: the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

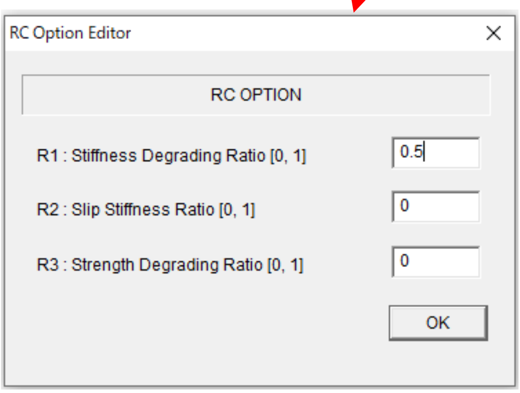


- Please input the section size.
- For the number of reinforcing bars and their size, please select the values from the popup windows.
- For the material strength, SD and Fc, you can input values by changing the default values.
- To move to the next element type, please click [ADD] button.
- You can 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.

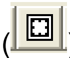


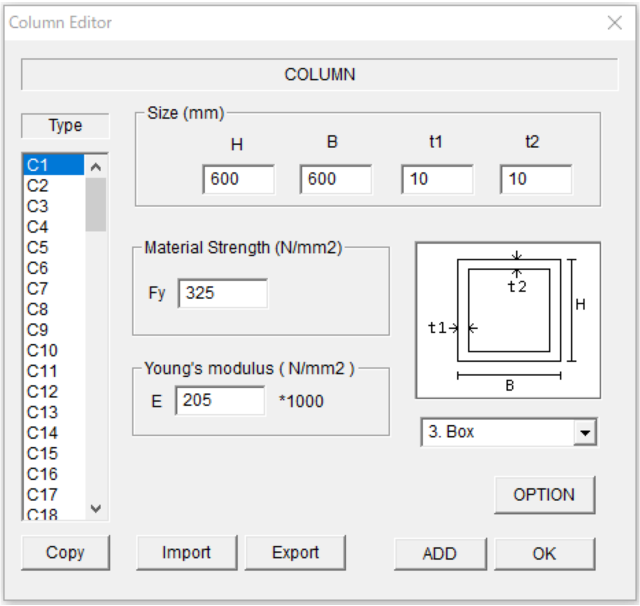
In [OPTION] menu, the parameters to control the shape of hysteresis model are defined as follows:

- 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.

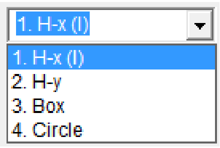
Please refer “Technical Manual” for the detail.

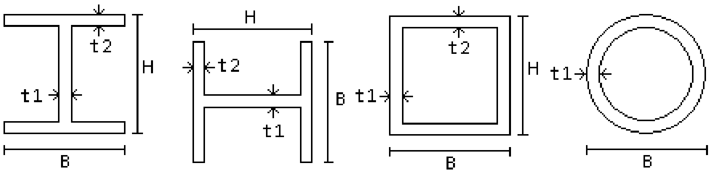
6.4 Steel Column

COLUMN ()

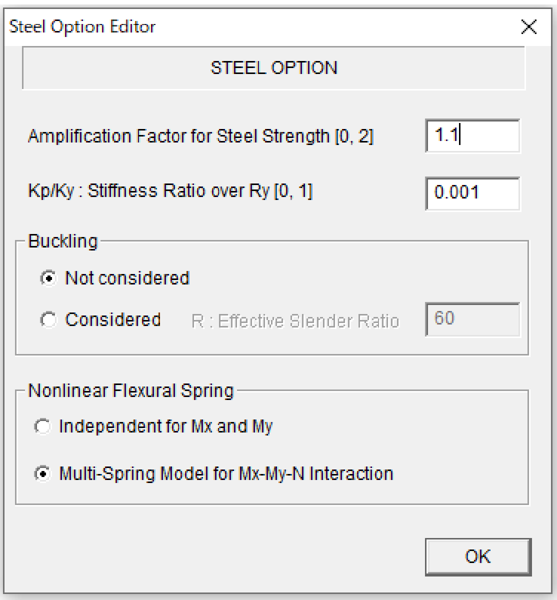


- Please select the shape of section from the pull-down menu.
- Please input the section size.
- For the material strength, Fy, and Young' s modulus, E, you can change the default values.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- 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

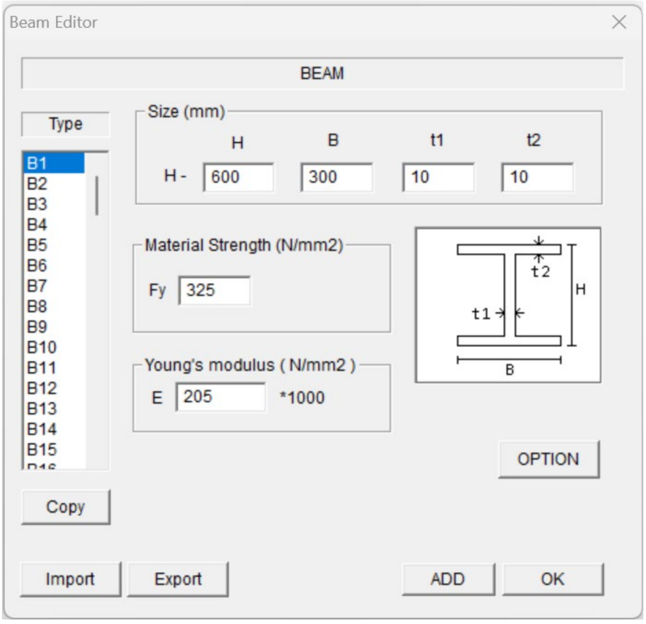


In [OPTION] menu,

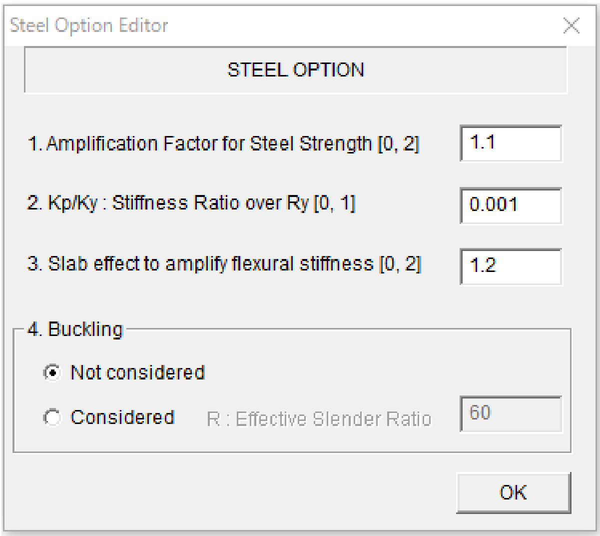
- The default steel strength is assumed to be 1.1 times larger than the nominal strength and the stiffness ratio after yielding is 0.001.
- Non-linear hysteresis due to buckling can be considered. The initial setting is "Not considered". To consider the buckling, you must enter an effective slender ratio.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

6.5 Steel Beam

BEAM ()



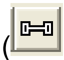
- Please input the section size.
- For the material strength, Fy, and Young' s modulus, E, you can change the default values.
- In [OPTION], you can enter the material properties of the steel beam, slab effects, and buckling considerations.
- To move to the next element type, please click [ADD] button.
- 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.

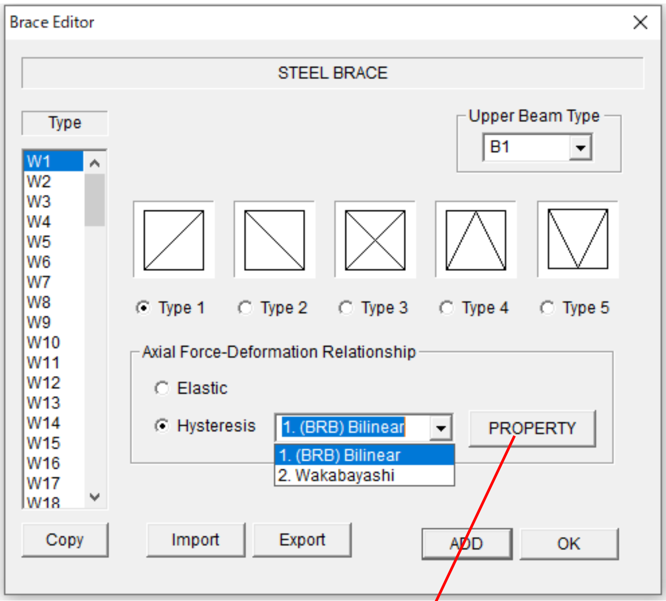


In [OPTION] menu,

- 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.

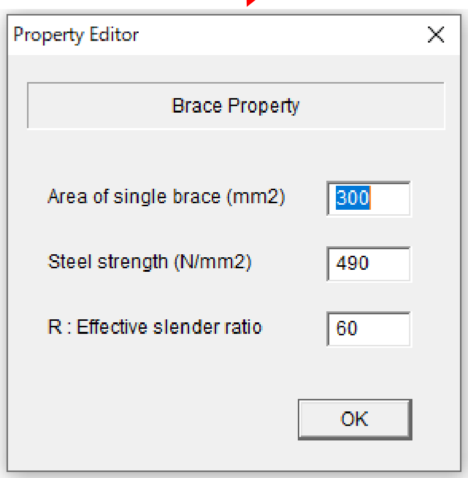
6.6 Steel Wall (Brace)

WALL ()



*BRB: buckling restrained brace

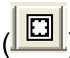
- If there is a beam upper of Brace element, please select the beam type number from the pop-up menu.
- Please select brace types from the pictures (Type1, 2, 3, 4, 5).
- You can select Axial force deformation relationship from "Elastic" and "Hysteresis". Non-linear hysteresis due to buckling can be 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.

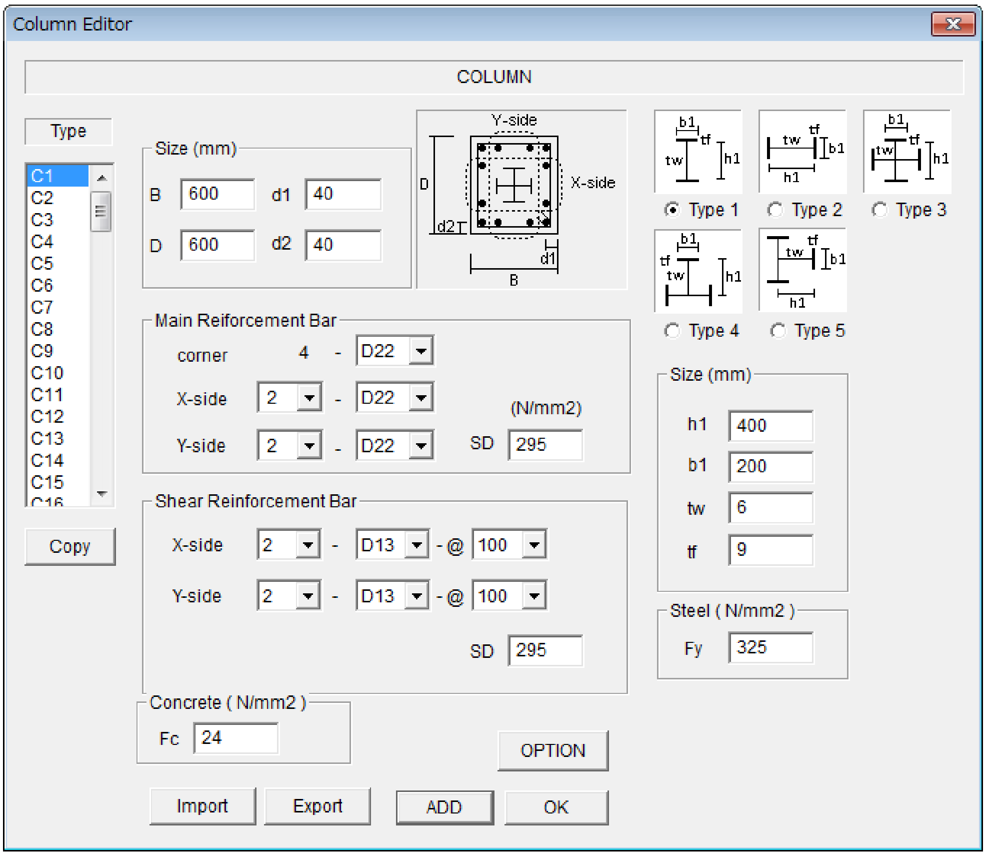


In [PROPERTY] menu,

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

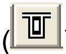
6.7 SRC (Steel Reinforced Concrete) Column

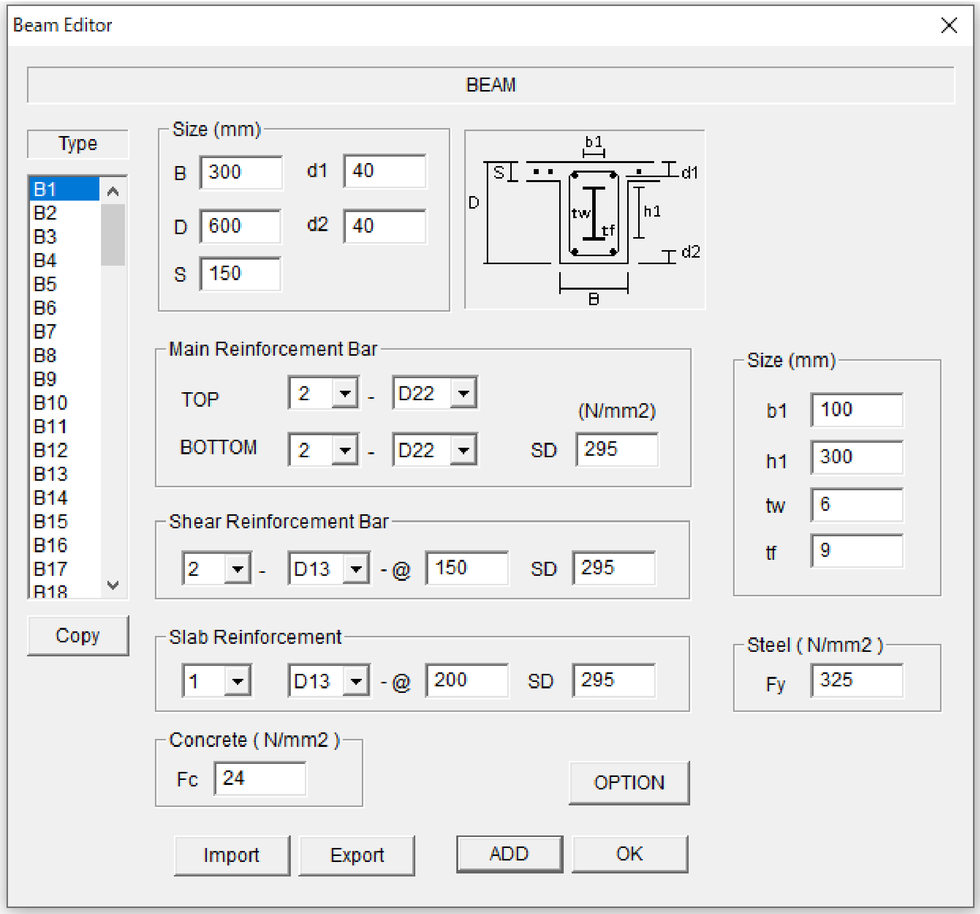
COLUMN ()



- 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.

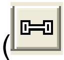
6.8 SRC (Steel Reinforced Concrete) Beam

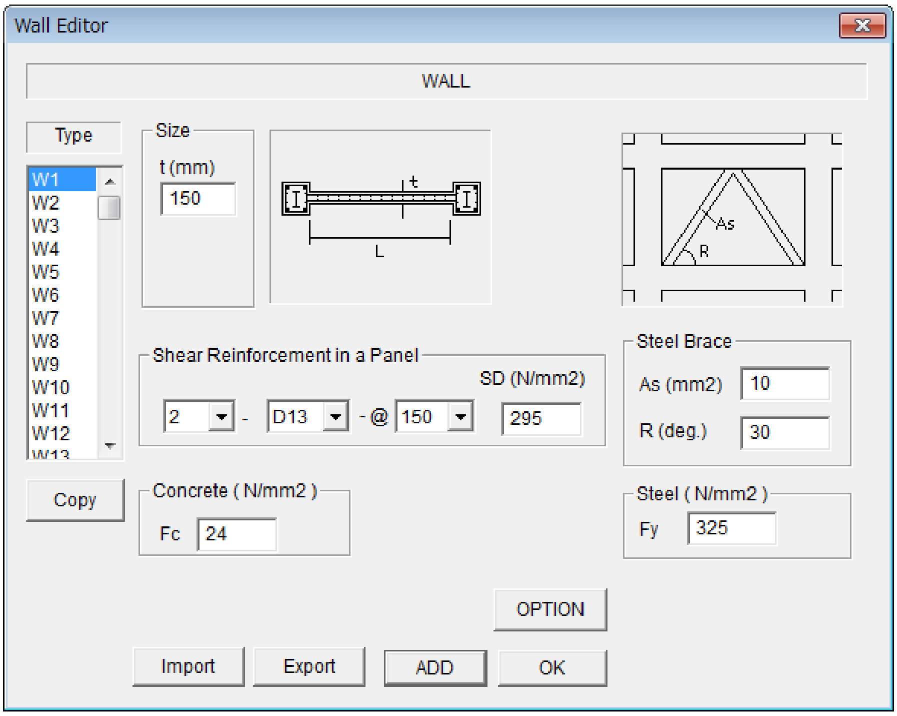
BEAM ()



- 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.

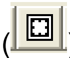
6.9 SRC (Steel Reinforced Concrete) Wall

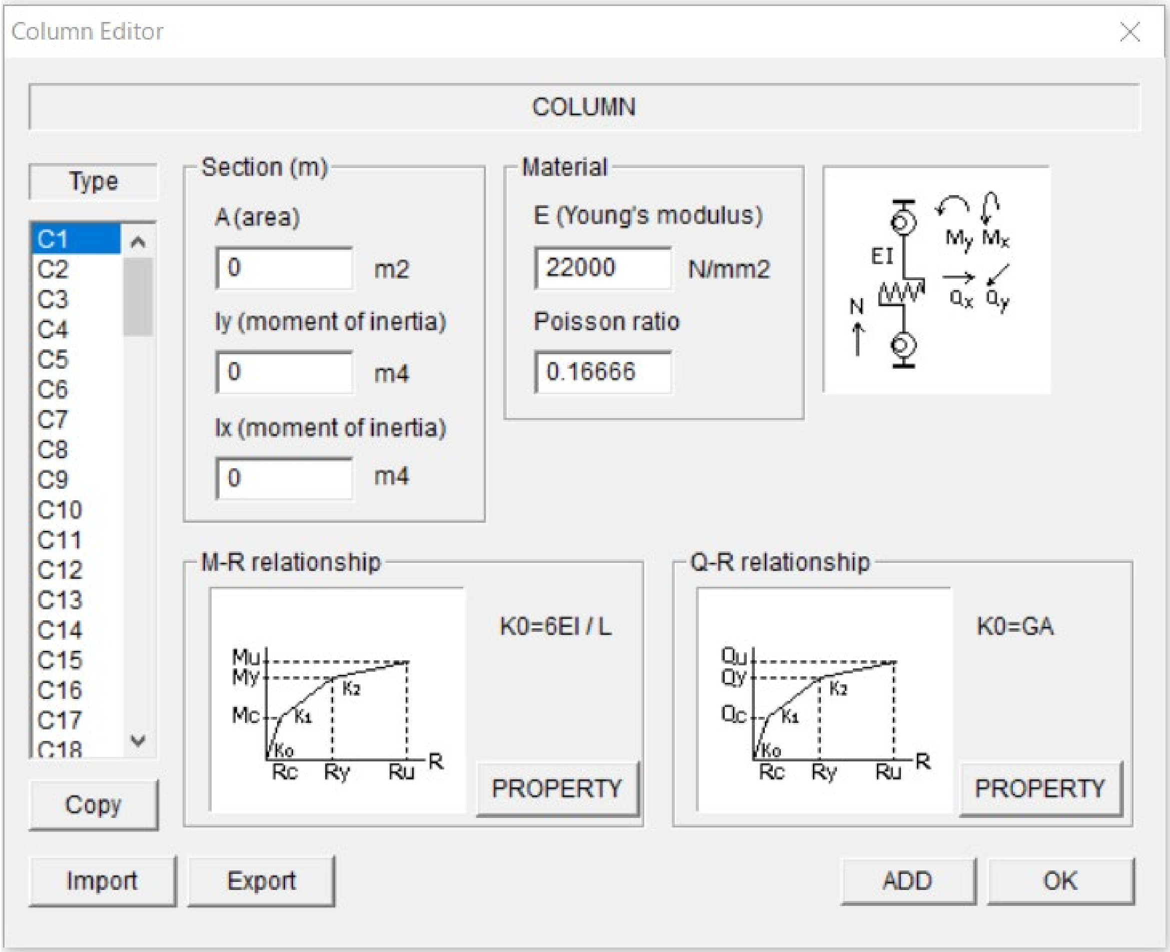
WALL ()



- RC part is the same as RC Wall.
- [OPTION] menu is the same as RC Wall.
- Please input area (As) and angle (R) of the steel brace.
- 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 “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.

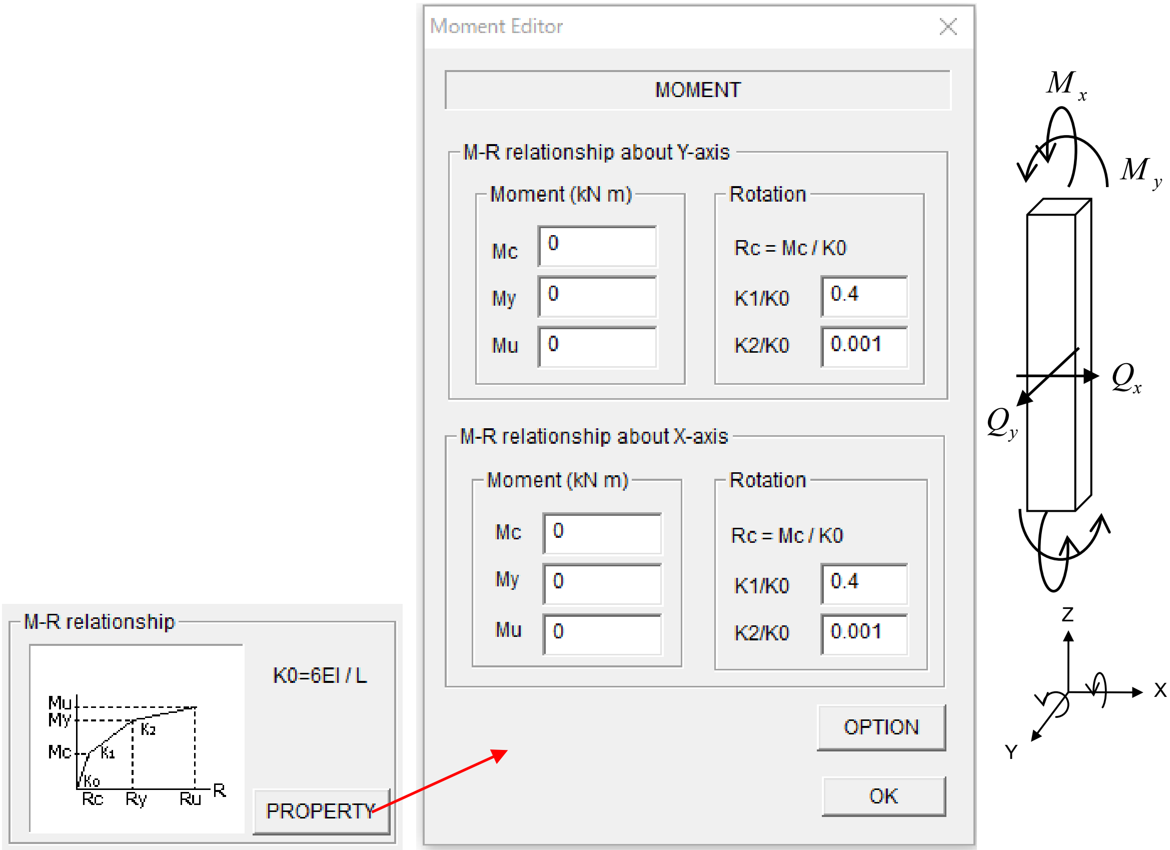
6.10 Column (Direct input for parameters of hysteresis model)

COLUMN ()



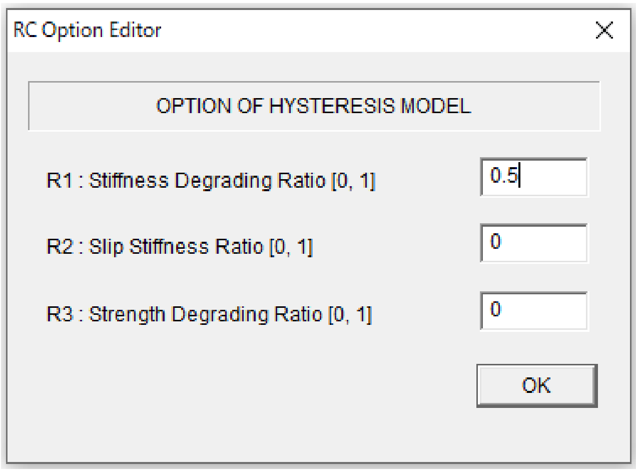
- Please input the area of section (A), the moment of inertia around x and y directions (Iy and Ix), Young's modulus (E), and Poisson's ratio (ν) of material that is used for
  - Axial stiffness of the section EA
  - Initial flexural stiffness K_0 = 6EI/L (L is the member length)
  - Initial shear stiffness K_0 =GA/L (G is the shear modulus = 0.5E/(1+ν))
- 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.

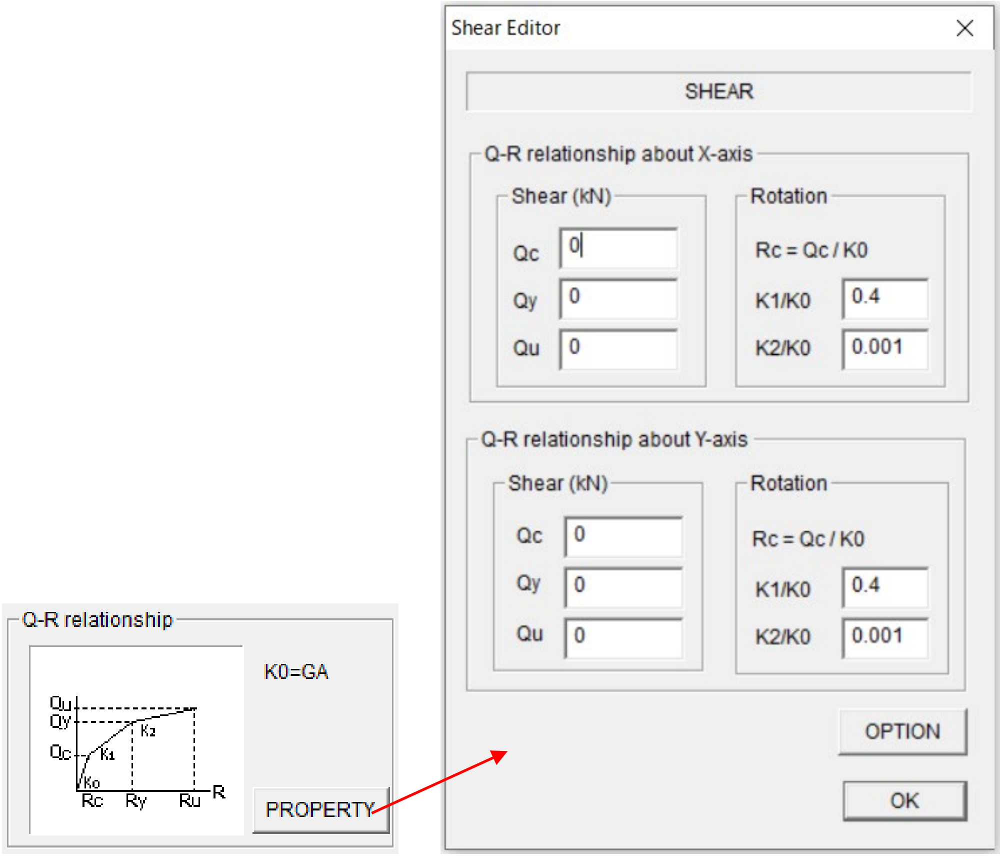


- Please input parameters for moment (M_y)-rotation relationship about Y axis.
- Please input parameters for moment (M_x)-rotation relationship about X axis.

- [OPTION] button is activated for RC structural type. From [OPTION] dialog, you can set the parameters for hysteresis model as follows:

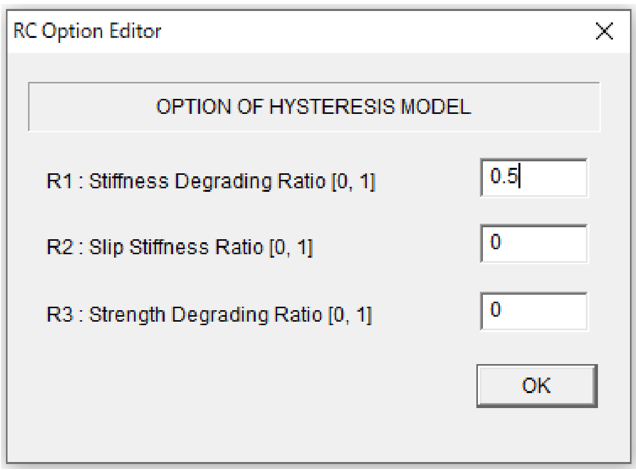


- Please input the parameters of Q-R (shear-rotation) relationship by [PROPERTY] bottom.

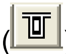


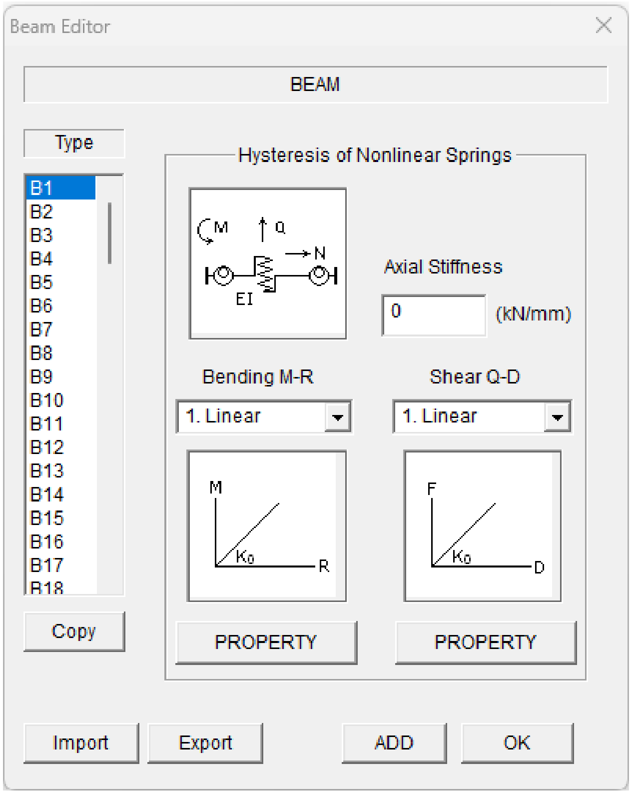
  - Please input parameters for shear (Q_x)-rotation relationship in X axis.
  - Please input parameters for shear (Q_y)-rotation relationship in Y axis.

- [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)

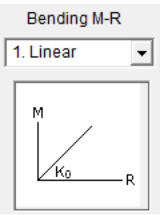
BEAM ()

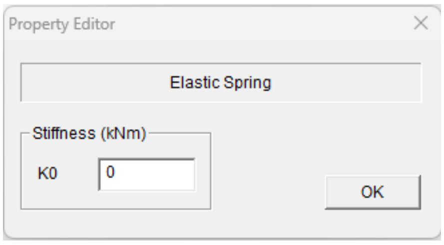


- It consists of bending springs, shear spring, and axial spring.
- Please input the axial stiffness directly.
- Please select hysteresis types for shear and bending springs from the pull-down menu.
- You can input the parameters of the hysteresis in [PROPERTY] view.
- You can set default values for 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.

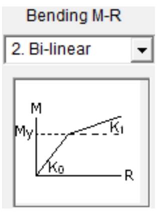
[1] Bending M-R

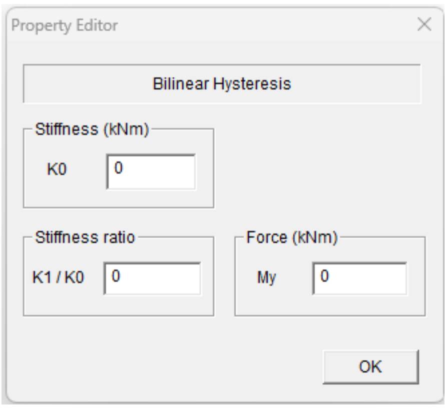
Linear



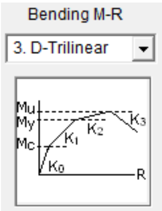


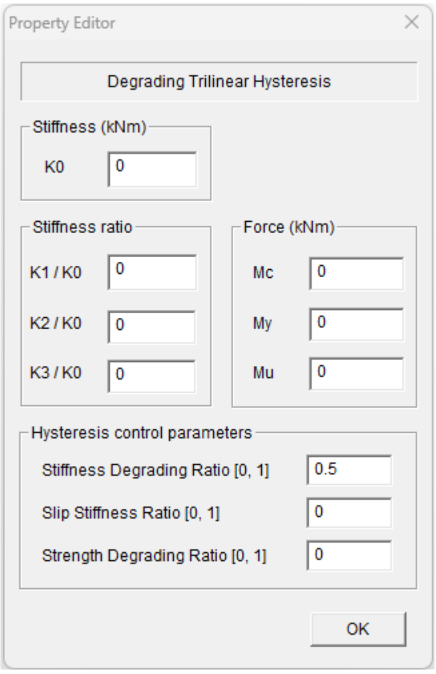
Bi-linear





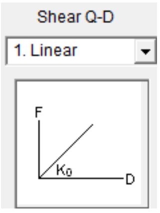
D-Trilinear

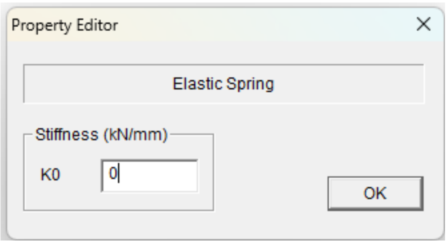




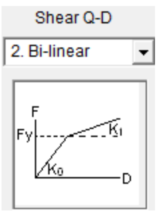
[2] Shear Q-D

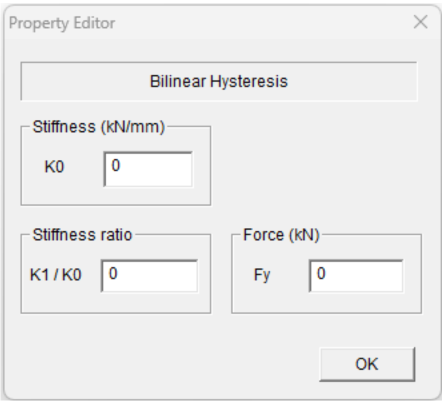
Linear



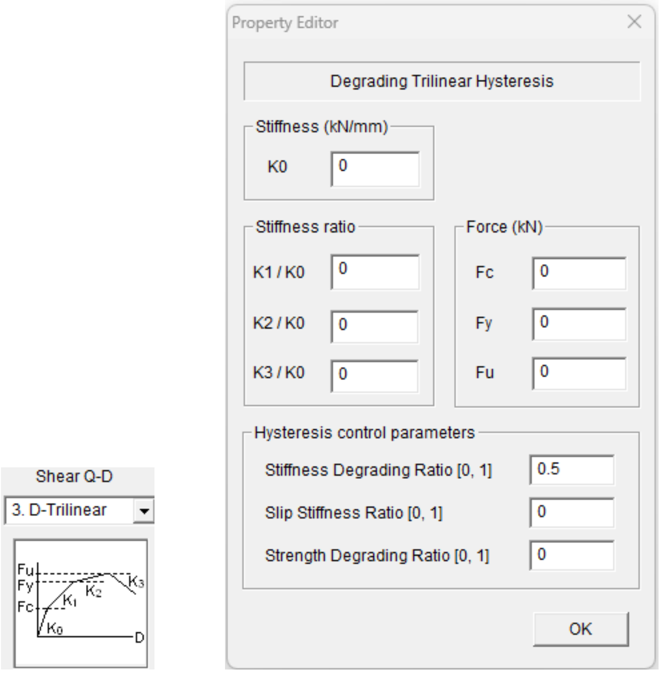


Bi-linear

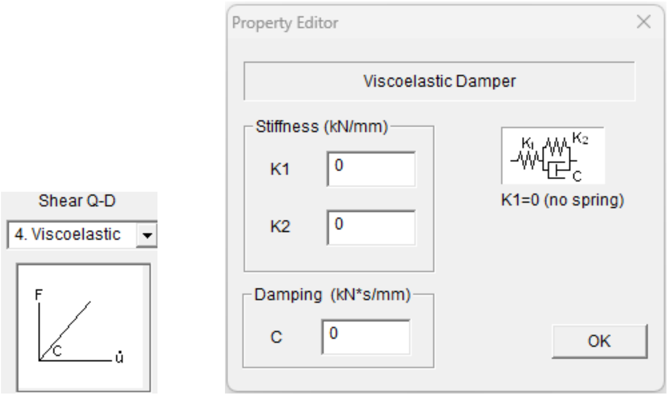




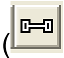
D-Trilinear

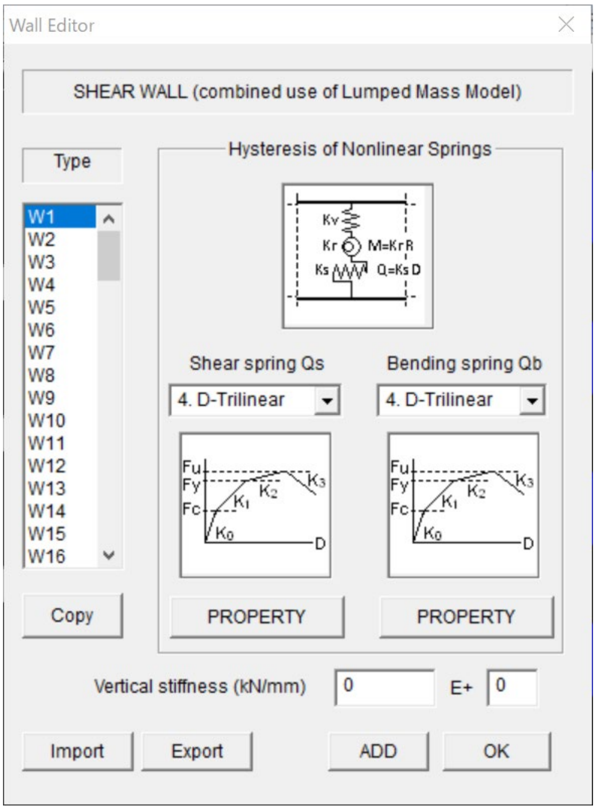


Viscoelastic



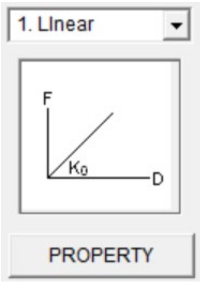
6.12 Wall (Direct input for parameters of hysteresis model)

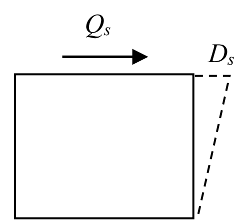
WALL ()

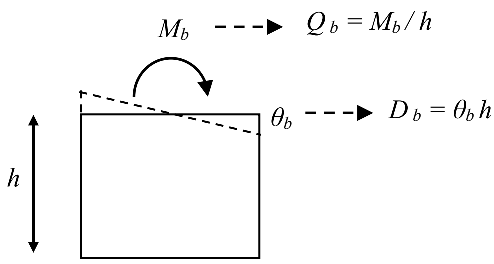


- It consists of shear spring, bending spring and axial spring.
- Please select hysteresis types for shear and bending springs from the pull-down menu.
- You can input the parameters of the hysteresis in [PROPERTY] view.
- Please input vertical stiffness directly.
- You can set default values for all 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

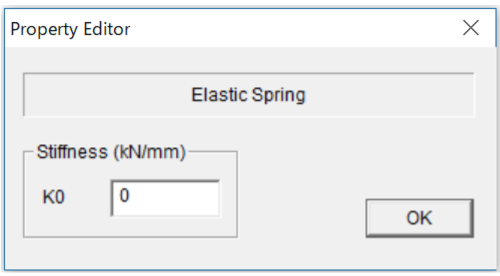


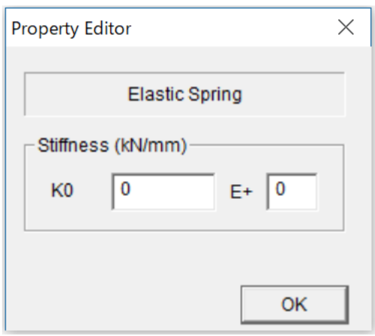




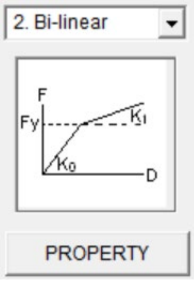
Bending spring (Q_b -D_b)

Shear spring (Q_s -D_s)

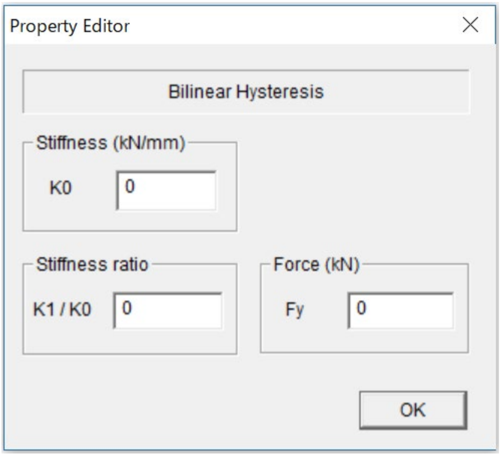


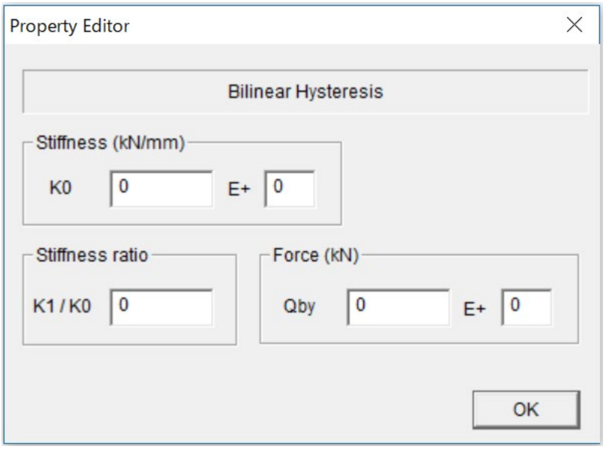


[2] Bi-linear model

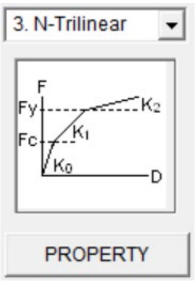


Shear spring $(Q_s - D_s)$ Bending spring $(Q_b - D_b)$

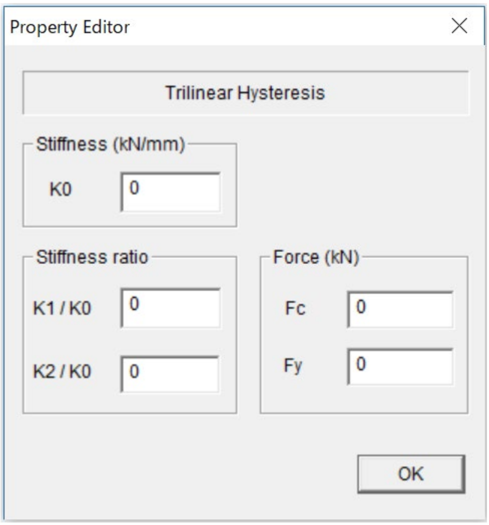


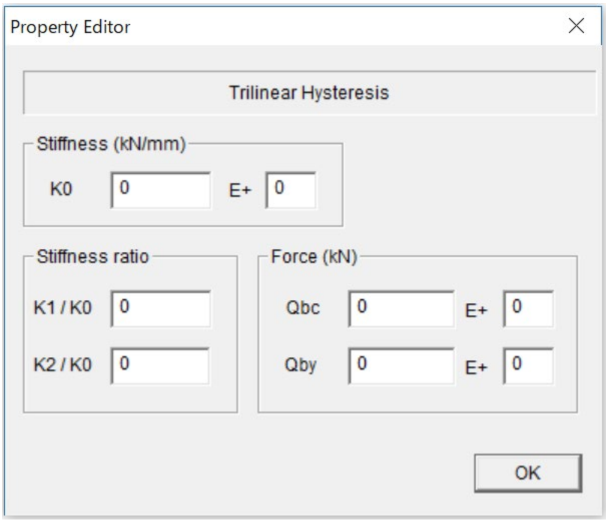


[3] Normal-Trilinear model

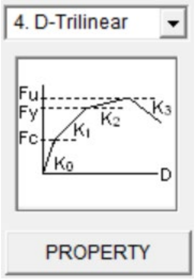


Shear spring $(Q_s - D_s)$ Bending spring $(Q_b - D_b)$

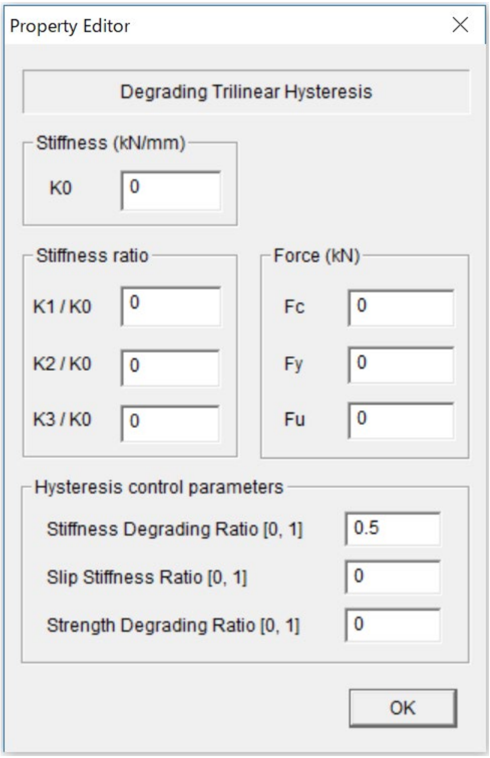




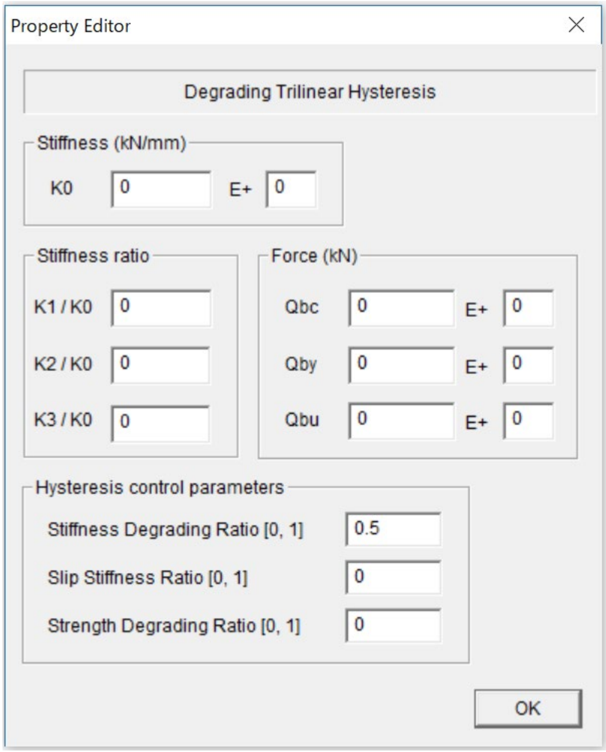
[4] Degrading Trilinear model



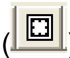
Shear spring (Q_s -D_s)

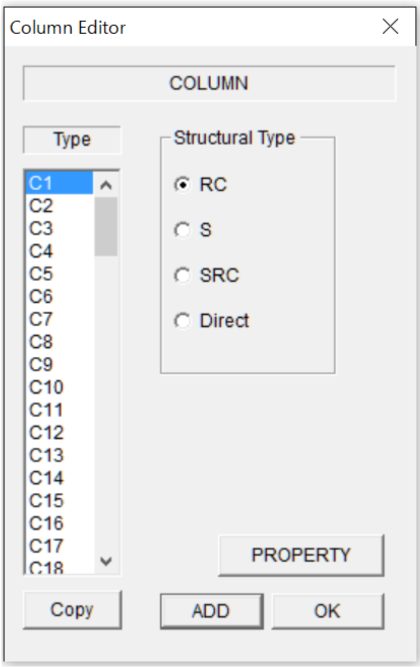


Bending spring (Q_b -D_b)



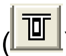
6.13 Column (Mixed mode)

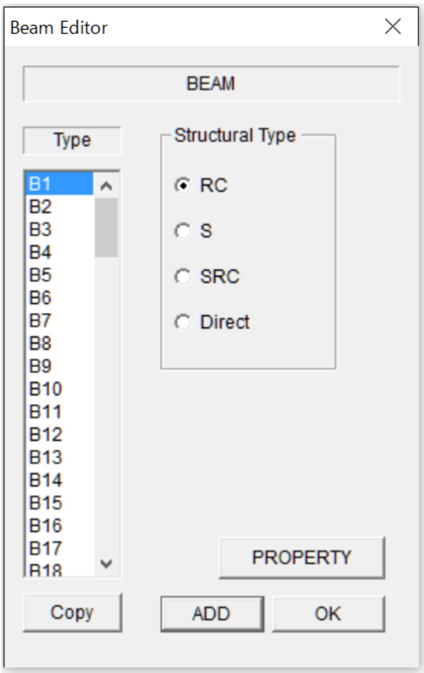
COLUMN ()



- 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".

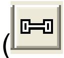
6.14 Beam (Mixed mode)

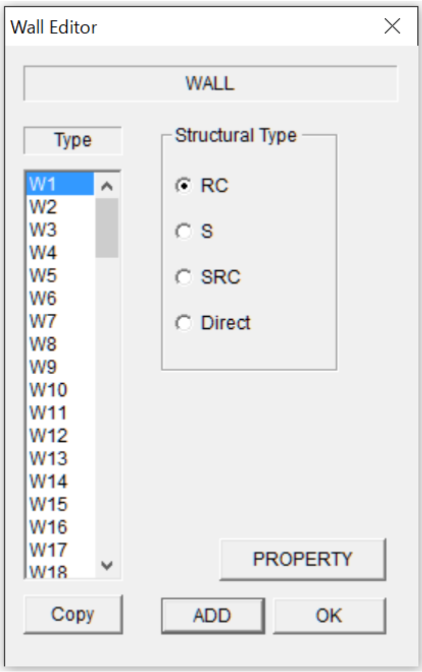
BEAM ()



- 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 ()

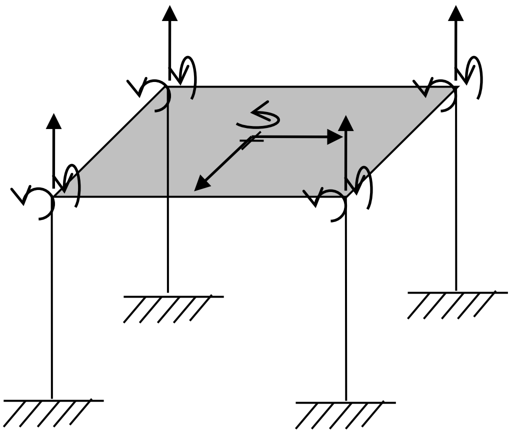


- You can select different “Structural Type” for each member type (W1, W2, ...).
- You can input member properties by [PROPERTY] button.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.
- You can set default values for all members by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

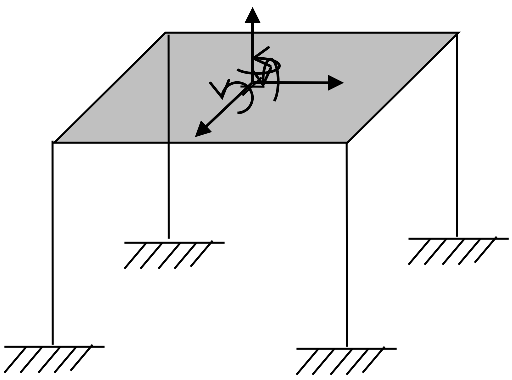
(a) In plane deformation (b) Out of plane deformation



(c) Independent freedom for 2D Rigid

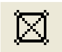
6.17 Floor Slab (3D Rigid)

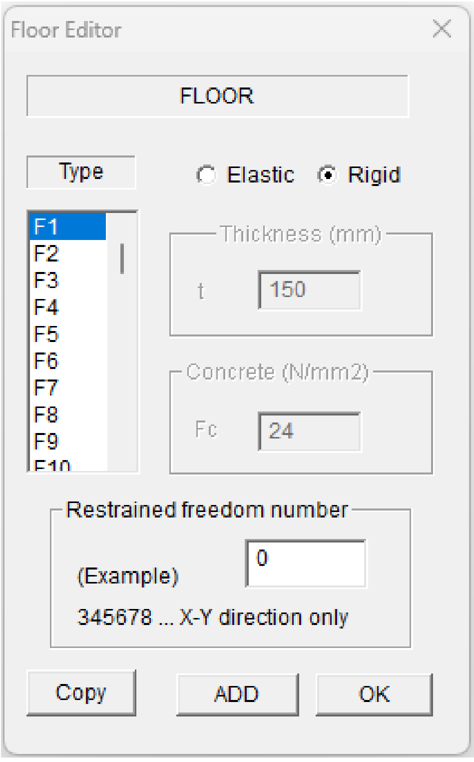
- Rigid for all directional deformation. Deformations at nodes are dependent to those of center of gravity.



(d) Independent freedom for 3D Rigid

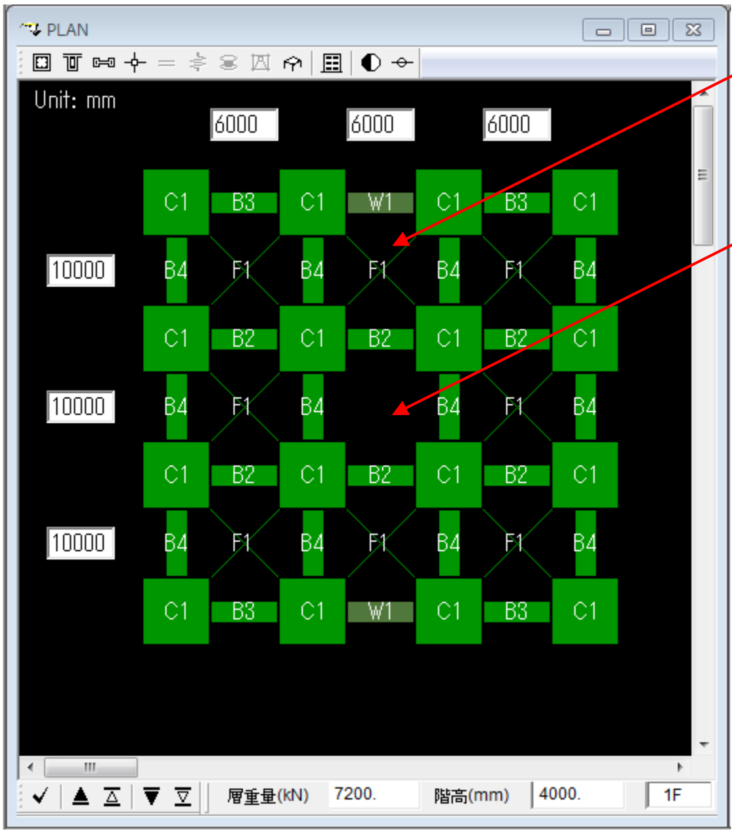
6.18 Floor Slab (Flexible)

SLAB ()



- **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”.

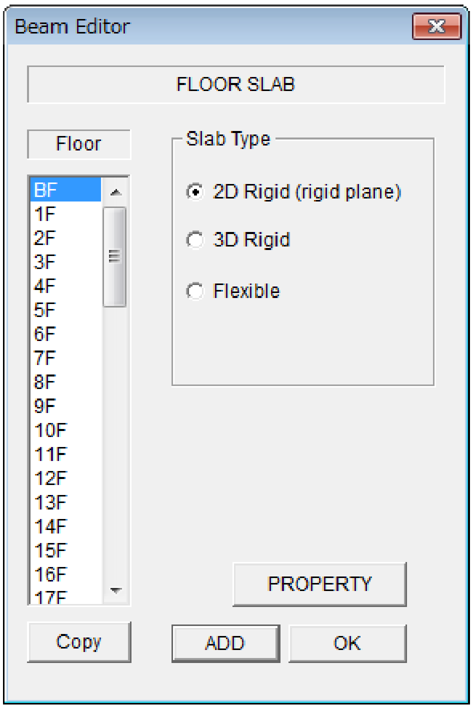


You can set member number (F1～F100).

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

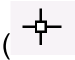
6.19 Floor Slab (Mix)

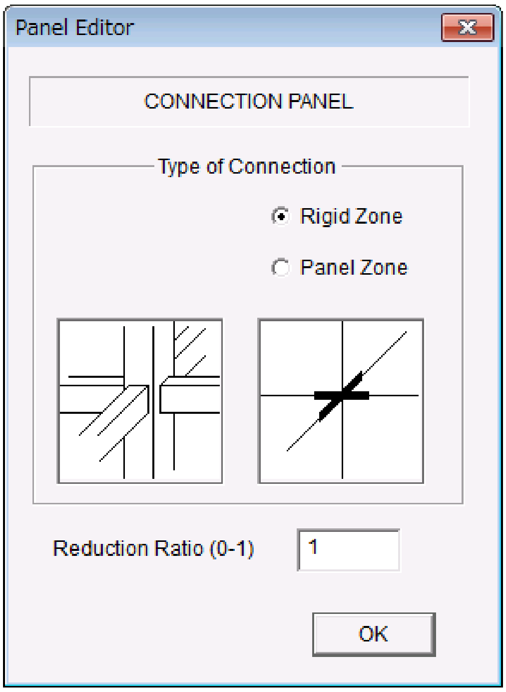
SLAB ()



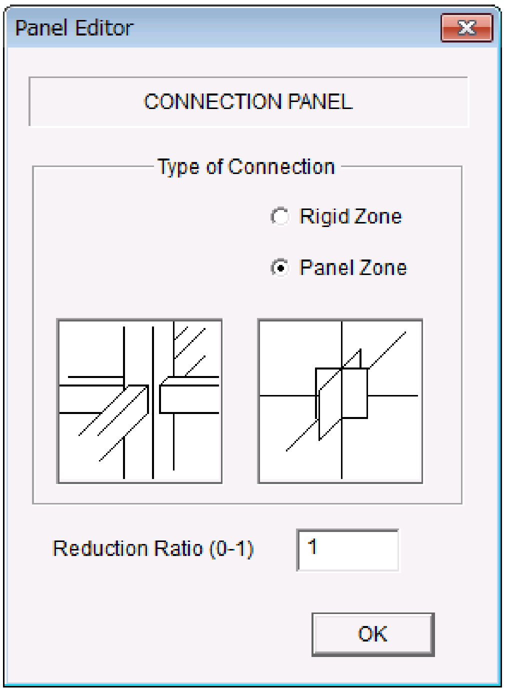
- You can select different "Slab Type" for each floor (BF, 1F, 2F, …).
- In case of "Flexible", you can input member properties by [PROPERTY] button.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element by [COPY] button.

6.20 Connection Panel

CONNECTION PANEL ()



Rigid Zone

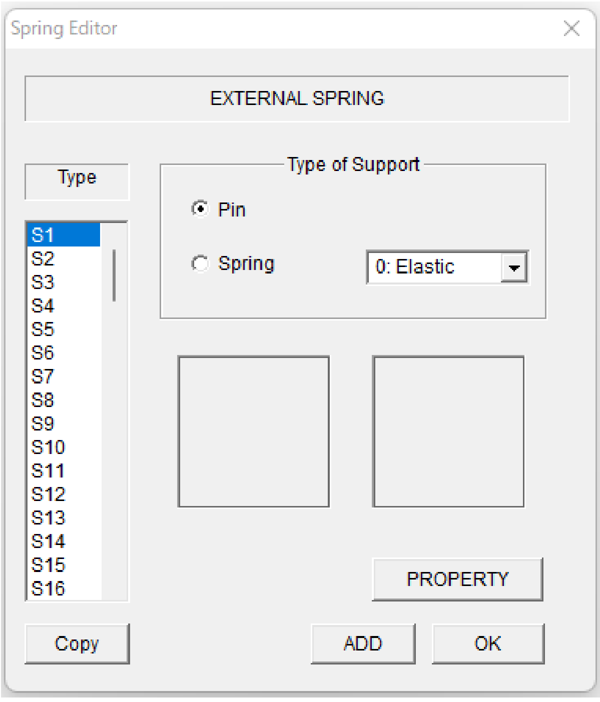


Panel Zone

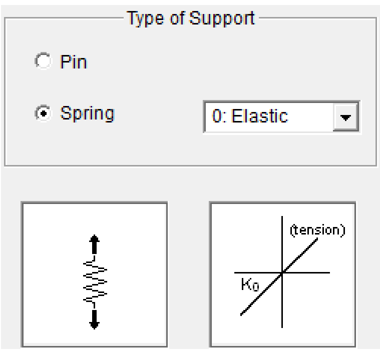
You can set the ratio of the length of rigid zone or panel zone inside connection area. The default value is 1.0 (to the member face).

6.21 External Spring

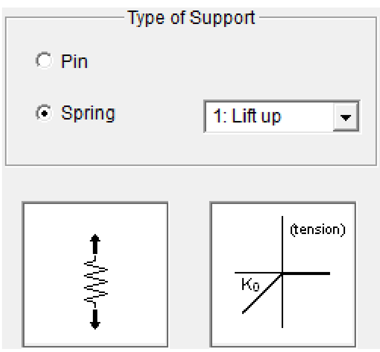
EXTERNAL SPRING () (NOTE: only available at the Basement Floor or you select External Spring in Option menu. Default is PIN)



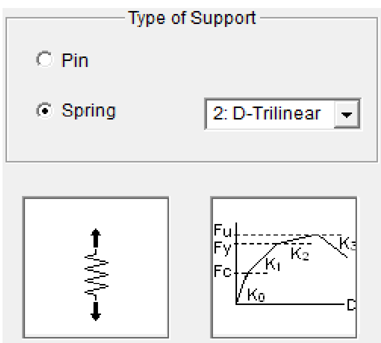
- You can select “Pin” support or “Spring” support. The default setting is “Pin”.
- As for “Spring”, you can select from “0. Elastic, 1. Lift up, 2: D-Trilinear, 3: Air Spring, 4: Base Plate, 5: Pendulum”.
- To move to the next element type, please click [ADD] button.
- You can copy the previous element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



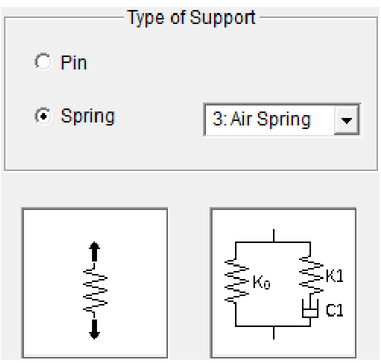
Elastic



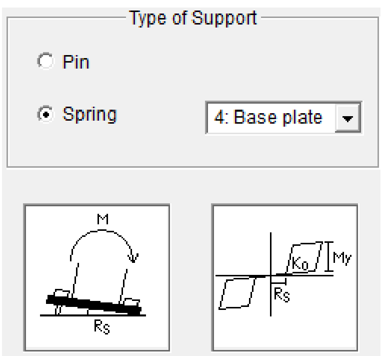
Lift up



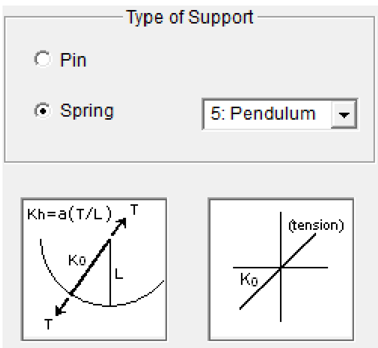
D-Trilinear



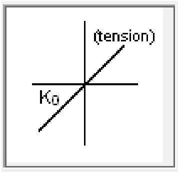
Air Spring

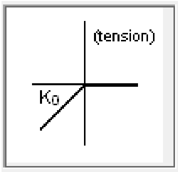


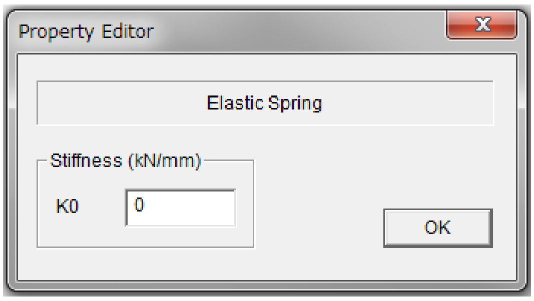
Base plate



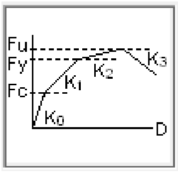
Pendulum

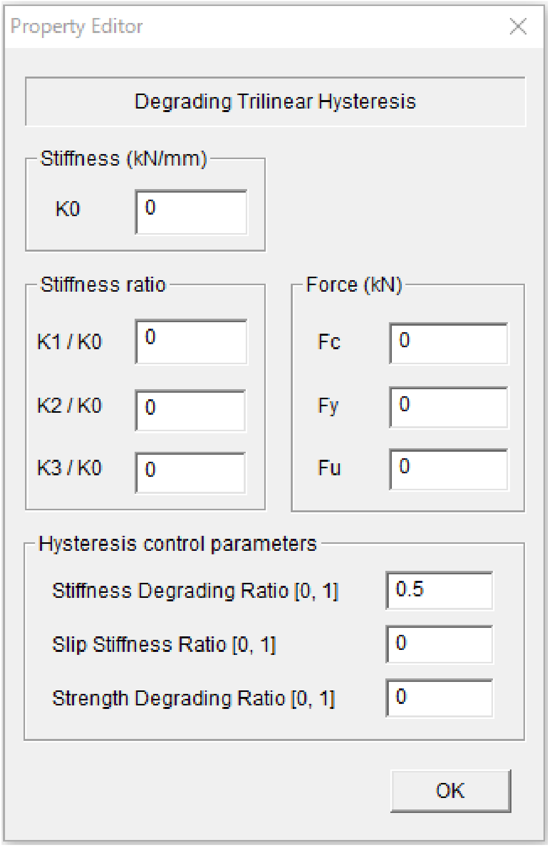
[0] In case of "Elastic" 

and [1] "Lift Up" 



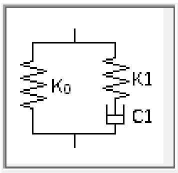
- Please input spring stiffness from the [PROPERTY] menu.

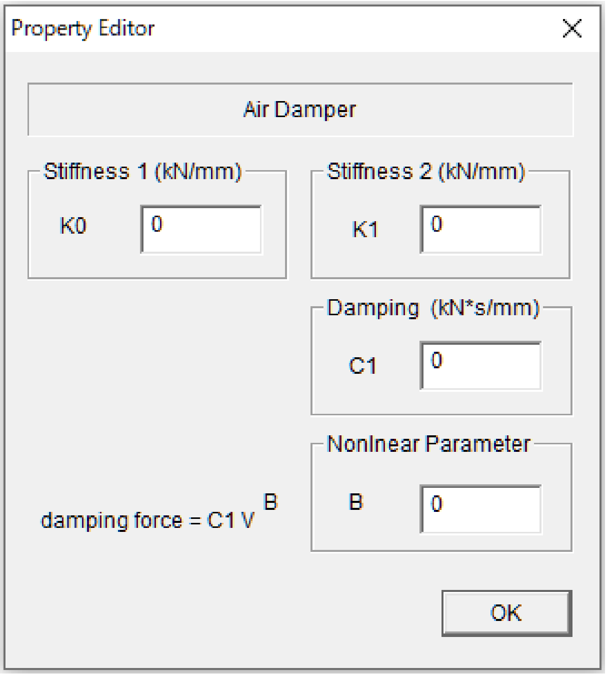
[2] In case of "D-Trilinear" 



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

[3] In case of "Air Spring"





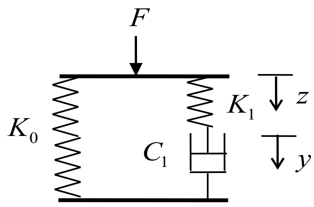
- Please input spring properties from the [PROPERTY] menu.

The air spring force is expressed as

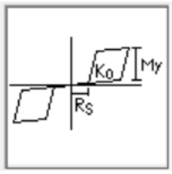
$$F = K_1 (z - y) + K_0 z$$

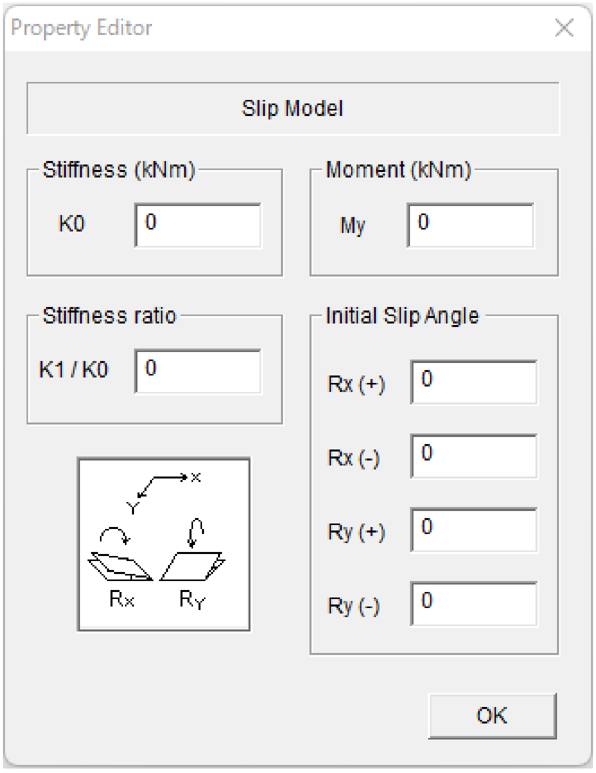
$$K_1 (z - y) = C_1 \cdot \dot{y}^B$$

Please refer "Technical Manual" for the detail.



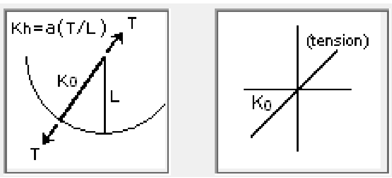
[4] In case of "Base Plate"

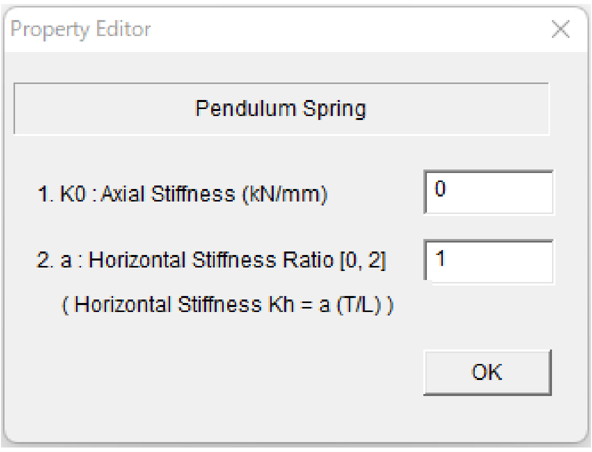




- Please input spring properties from the [PROPERTY] menu.

[5] In case of "Pendulum"

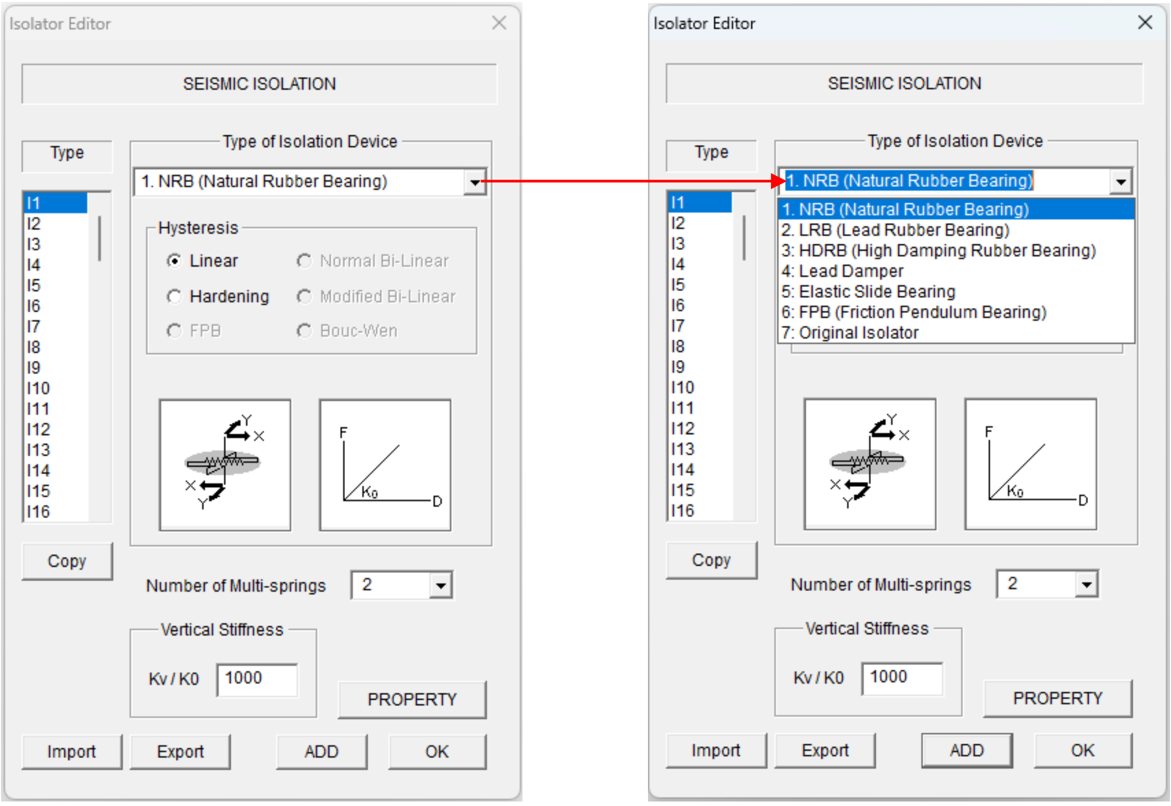




- Please input spring properties from the [PROPERTY] menu.

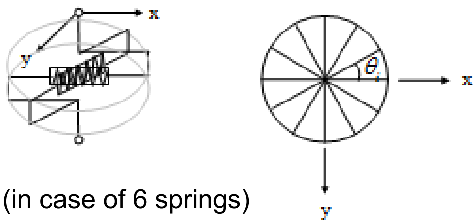
6.22 Seismic Isolator

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



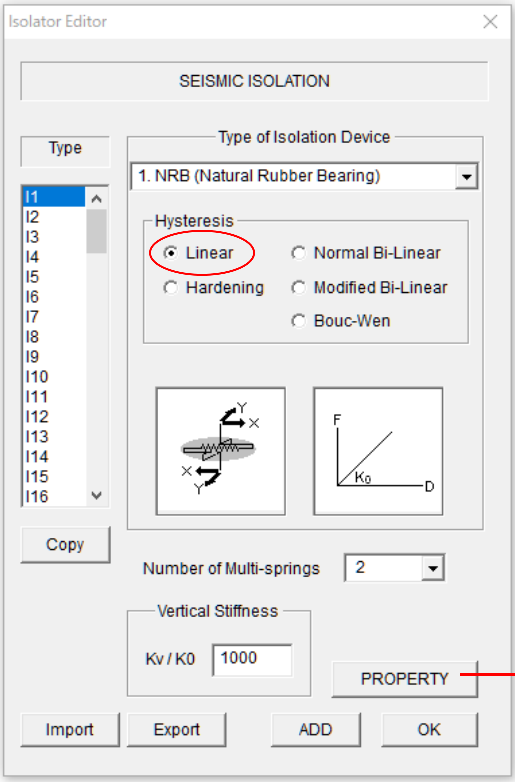
(in case of 6 springs)

Please read "STERA3D_technical_manual" about the detail of each device.

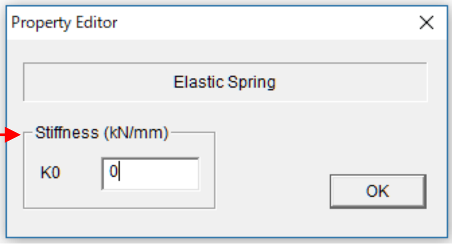
- The default value of the ratio between vertical stiffness, Kv, and the horizontal stiffness, K_0, is 1000.
- You can select the number of multi-springs from the pull-down menu (2, 4, 6, 8, 10).
- You can set default values for all members by selecting the last member type "Idef".
- You can export member 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.

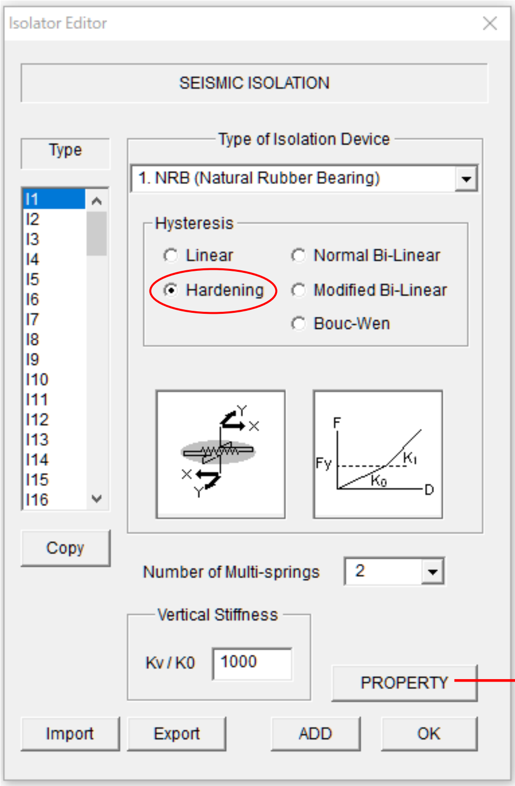
[1] NRB Isolator

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

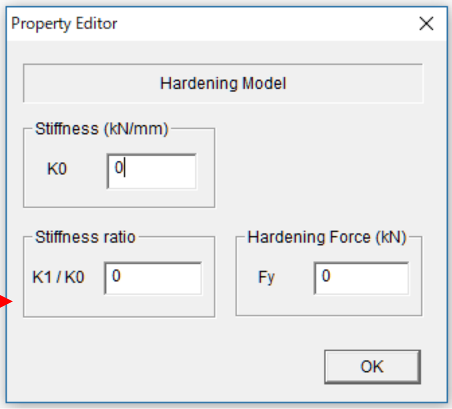


In case of “Linear”

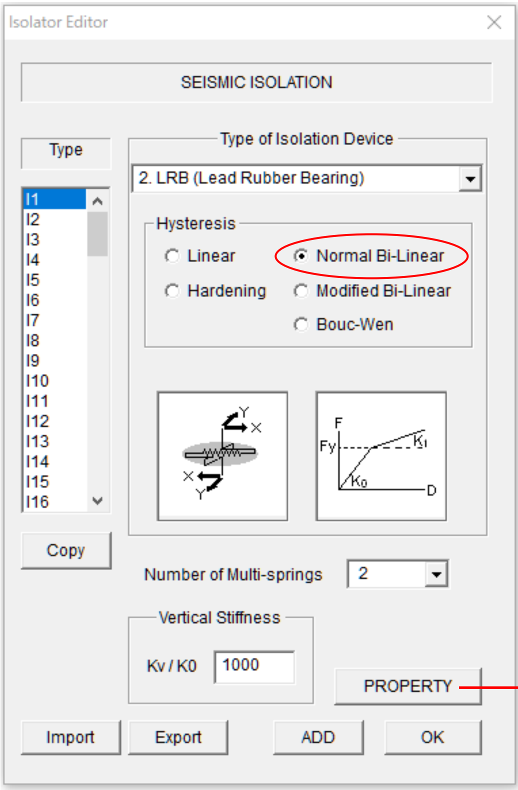




In case of “Hardening”

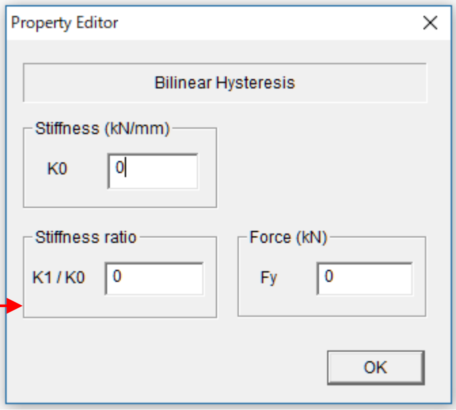


[2] LRB Isolator



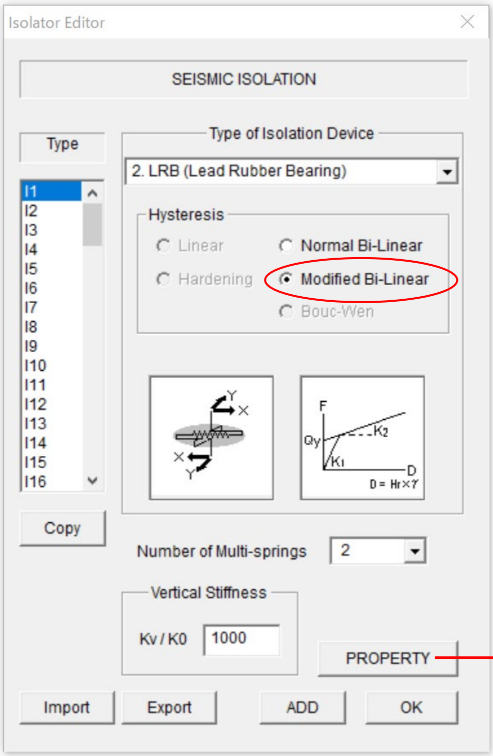
- For LRB isolator, you can select “Normal Bi-Linear” or “Modified Bi-Linear” hysteresis.
- Please input spring property by [PROPERTY] button.

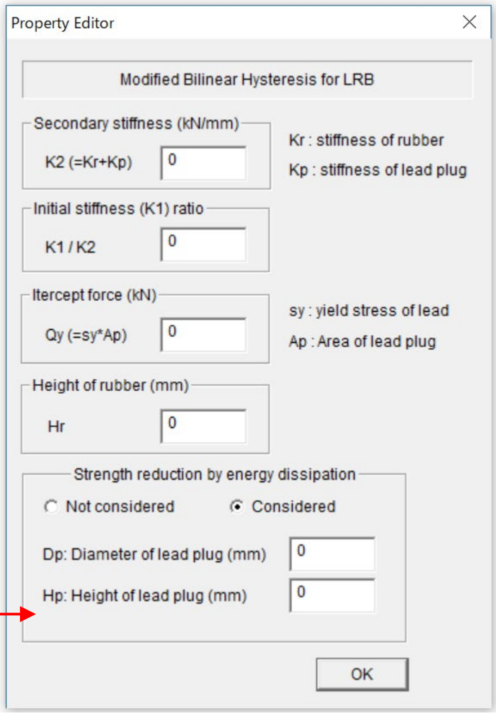
In case of “Normal Bi-Linear”



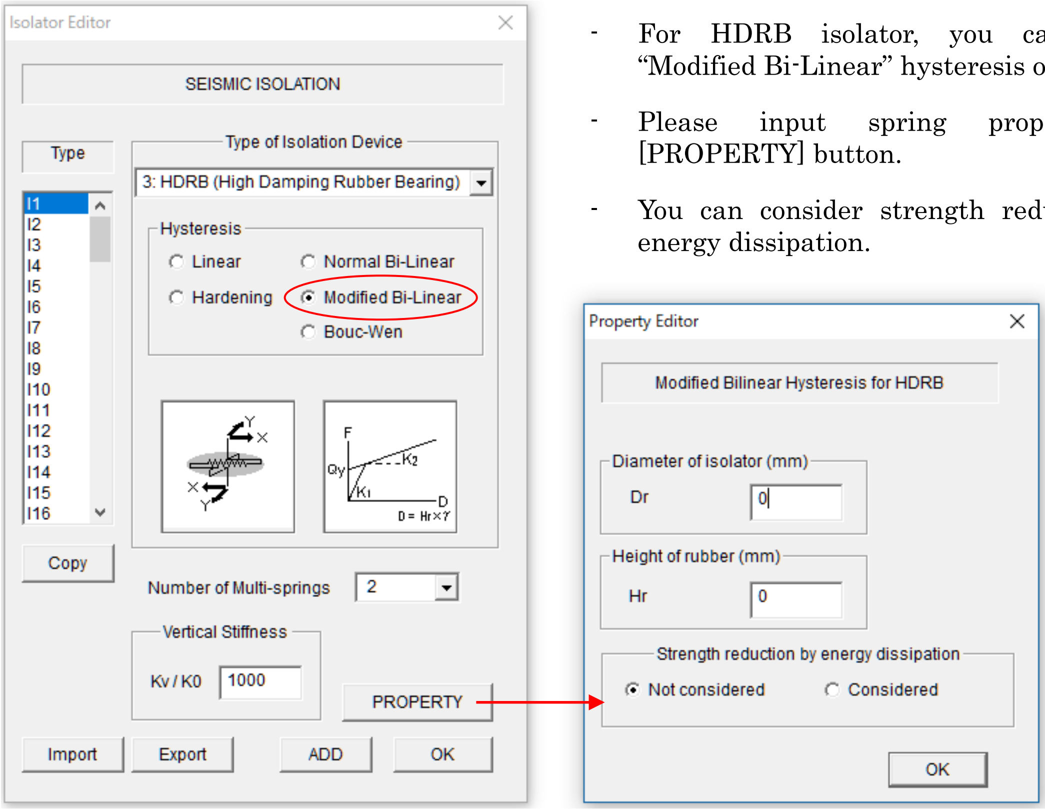
- In case of “Modified Bi-Linear” hysteresis, you can consider strength reduction by energy dissipation.

In case of “Modified Bi-Linear”



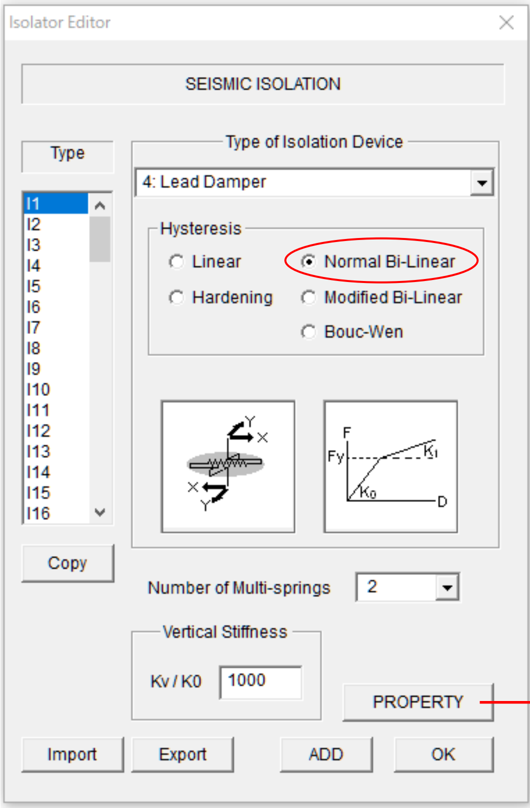


[3] HDRB (High Damping Rubber Bearing) Isolator

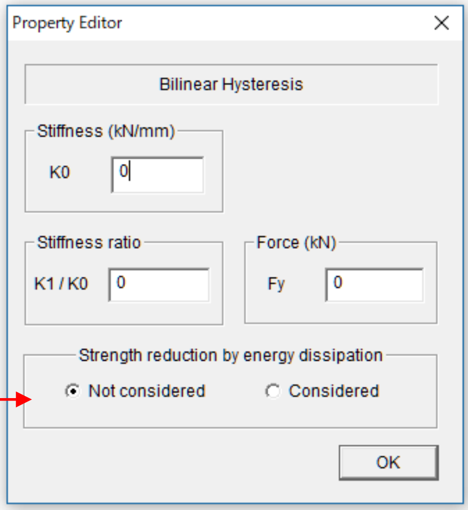


- For HDRB isolator, you can select "Modified Bi-Linear" hysteresis only.
- Please input spring property by [PROPERTY] button.
- You can consider strength reduction by energy dissipation.

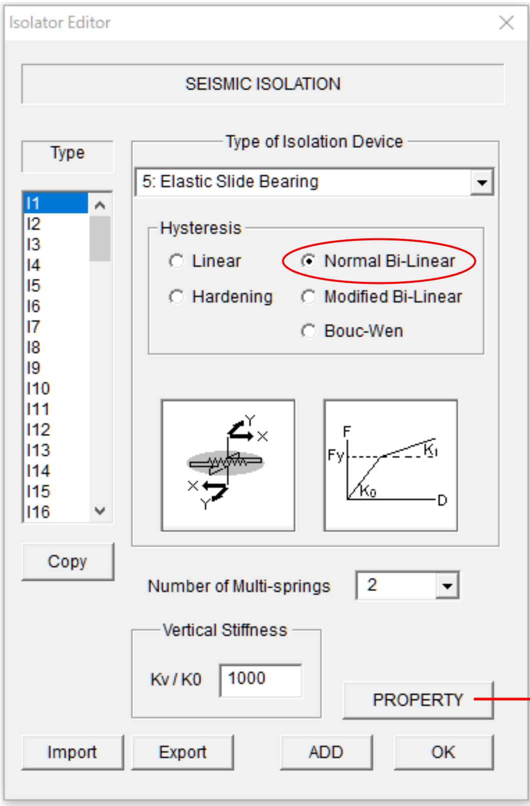
[4] Lead Damper



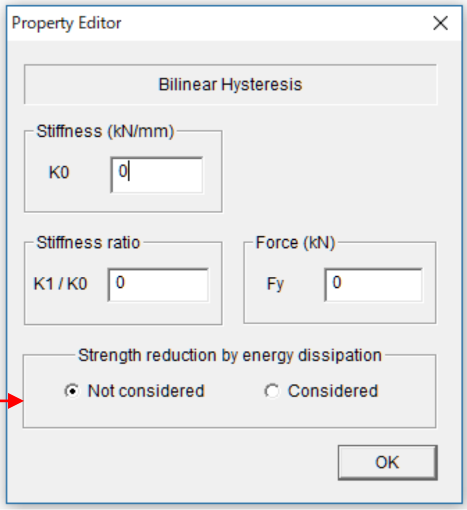
- For Lead Damper, you can select "Normal Bi-Linear" hysteresis only.
- Please input spring property by [PROPERTY] button.
- You can consider strength reduction by energy dissipation.



[5] Elastic Slide Bearing

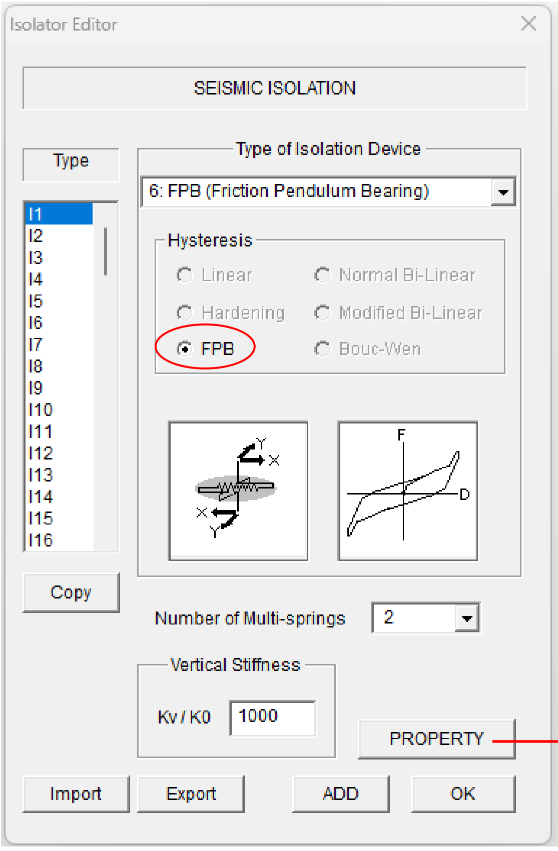


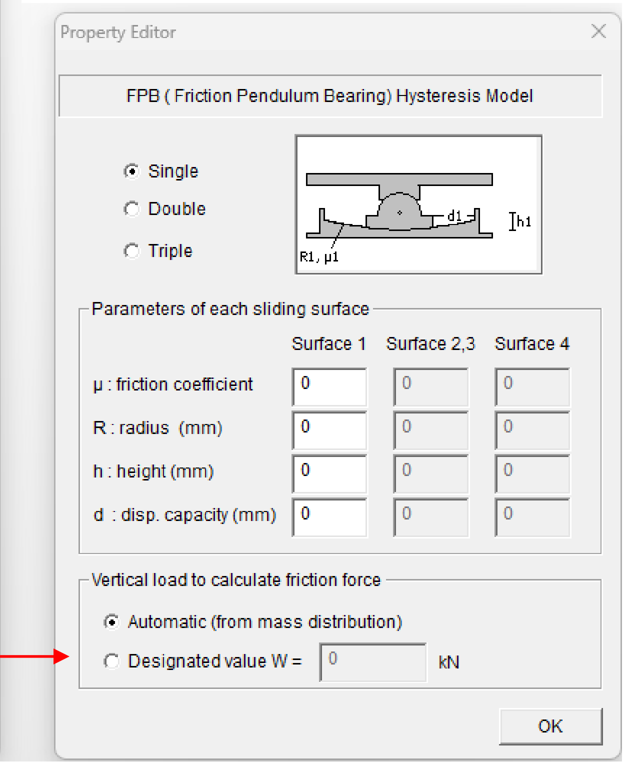
- For Elastic Slide Bearing, you can select "Normal Bi-Linear" hysteresis only.
- Please input spring property by [PROPERTY] button.
- You can consider strength reduction by energy dissipation.



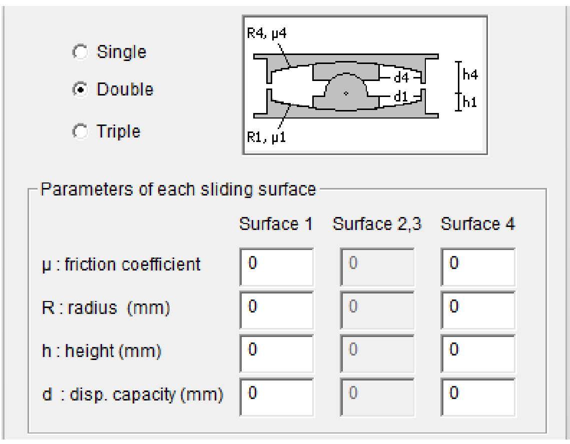
[6] FPB (Friction Pendulum Bearing)

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

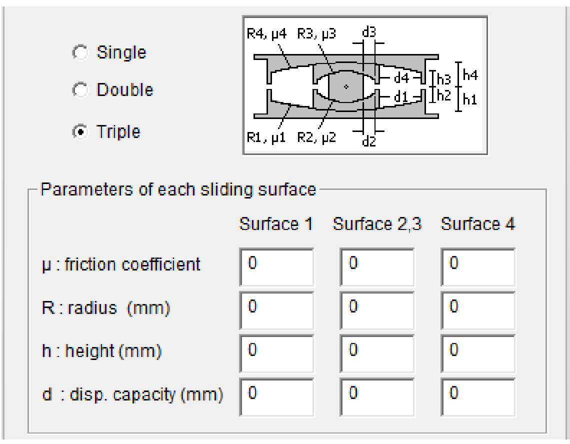




In case of Double,

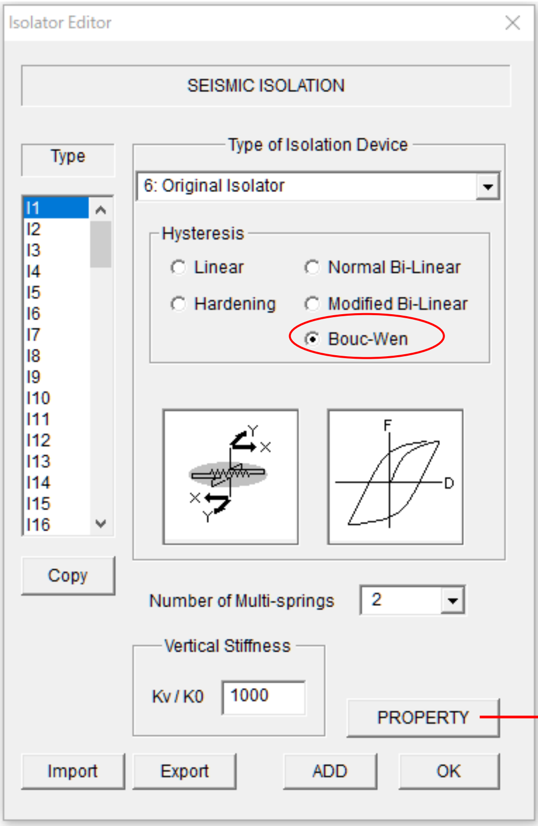


In case of Triple

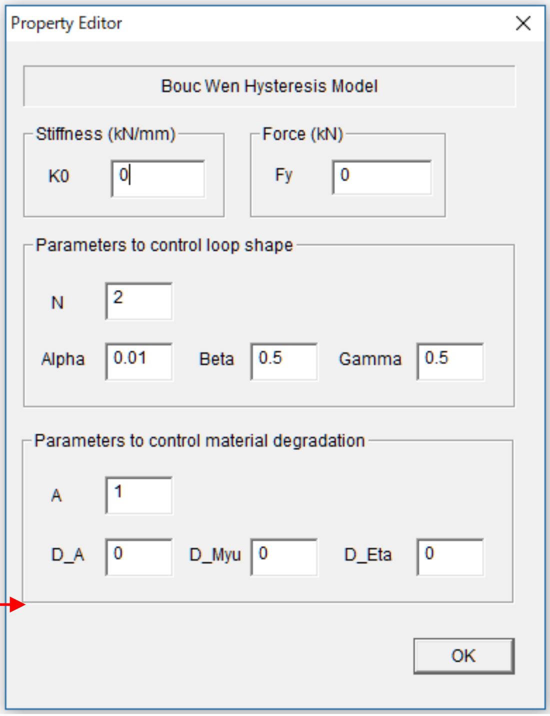


[7] Original Isolator

- 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.



In case of “Bouc-Wen”



Definition of Bouc-Wen Model
(Please refer “Technical Manual” for the detail.)

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

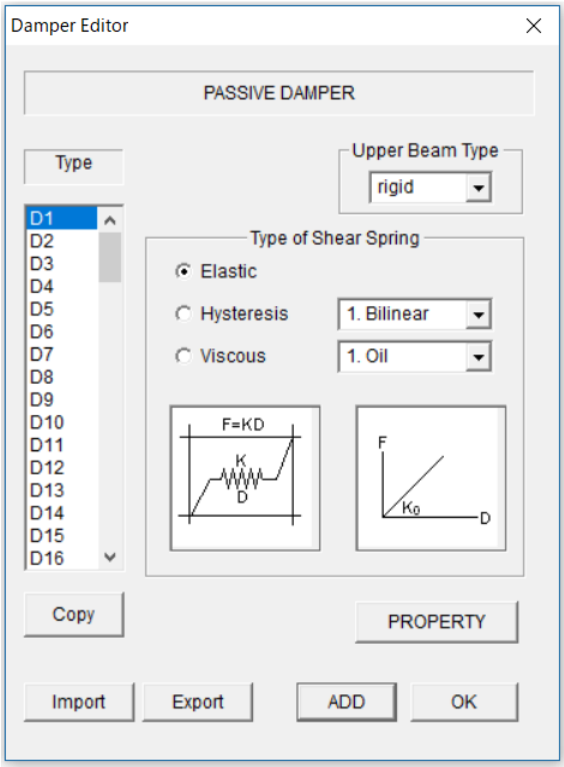
$$\dot{z} = \frac{A\dot{x} - \left\{\beta |\dot{x}| |z|^{N-1} z + \gamma \dot{x} |z|^{N}\right\} \nu}{\eta}$$

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

Alpha = α , Beta = β , Gamma = γ
A = A_0
D_A = δ_A , D_Myu = δ_ν , D_Eta= δ_η ,

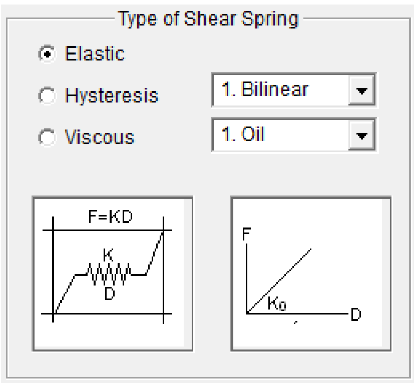
6.23 Passive Damper

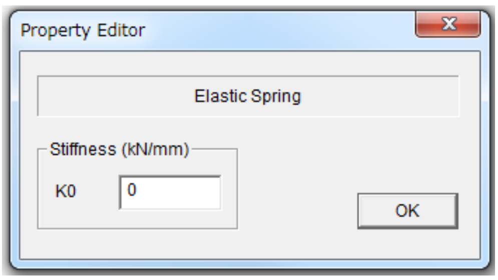
PASSIVE DAMPER () (NOTE: only available when you select Passive Damper in Option menu)



- 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.

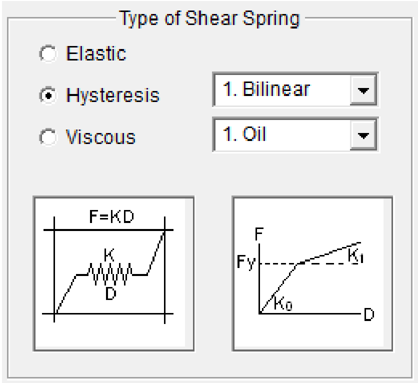
[1] Elastic spring

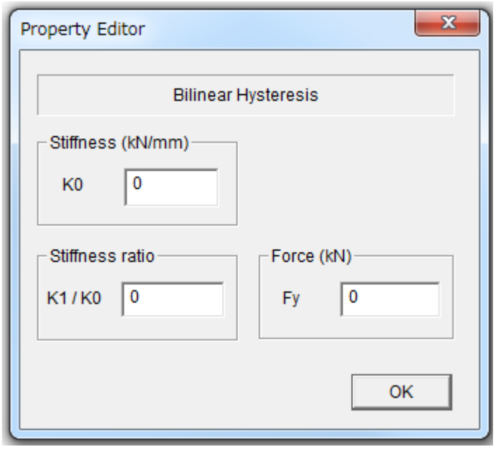




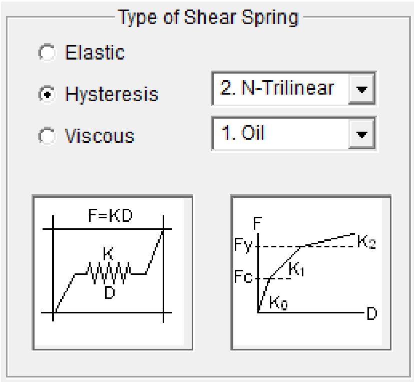
[2] Hysteresis Dampers

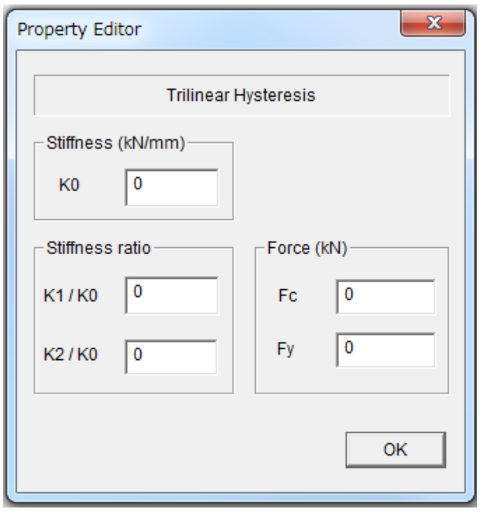
Bilinear



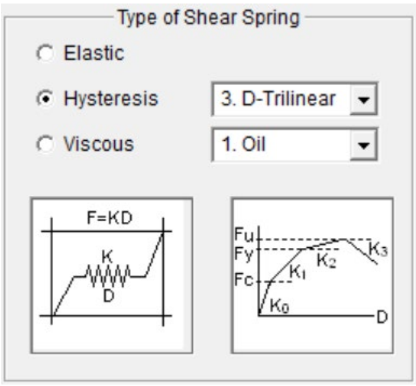


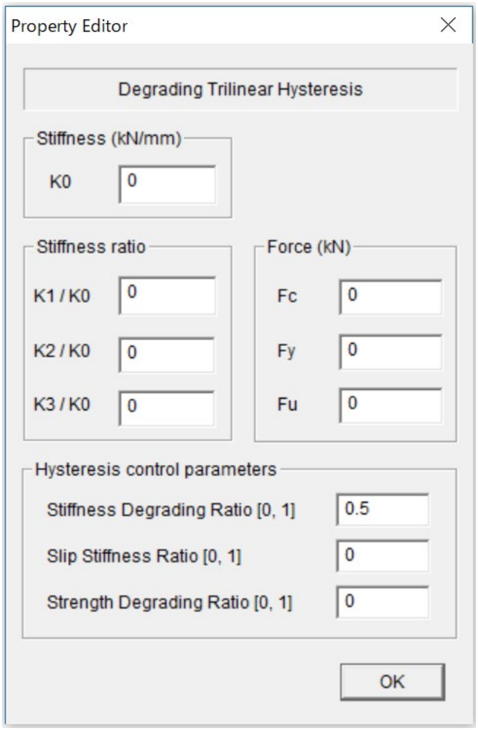
Normal Trilinear



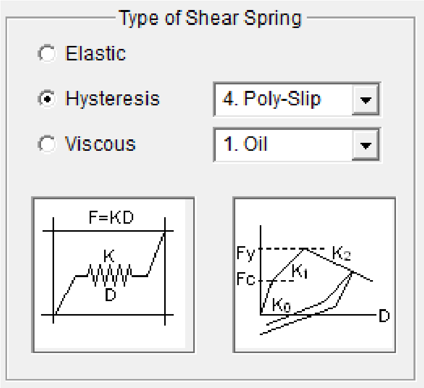


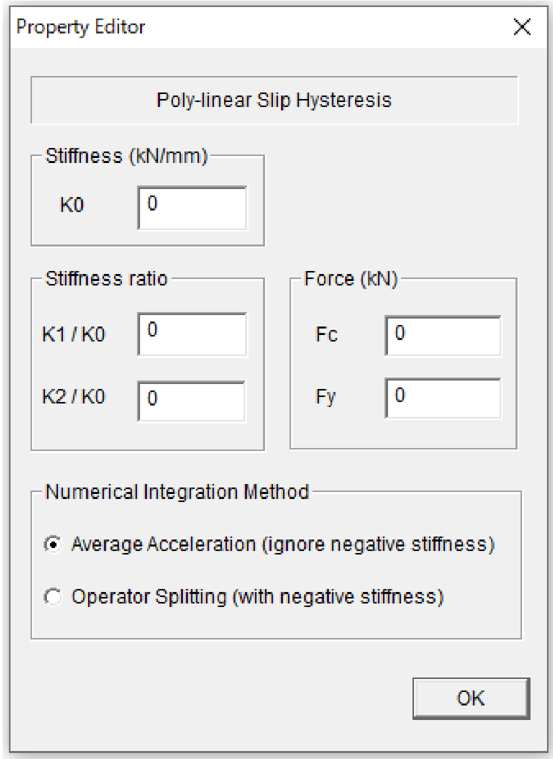
Degrading Trilinear



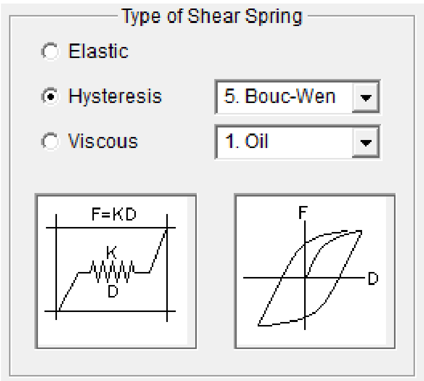


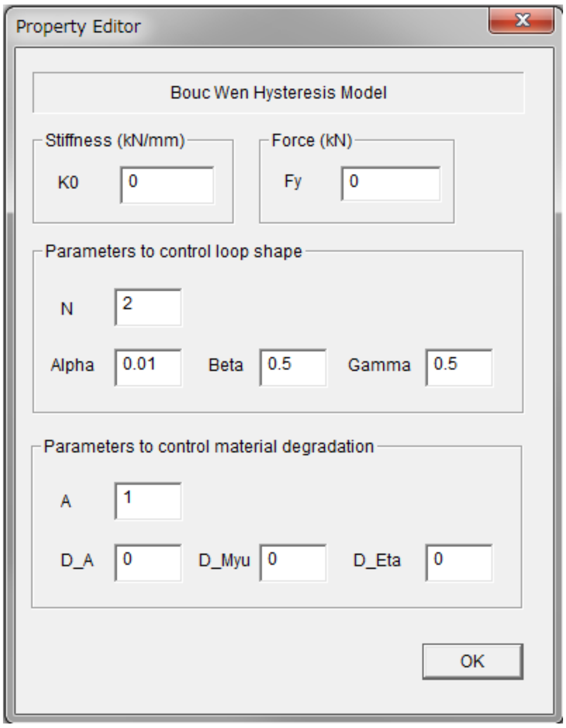
Poly-linear Slip



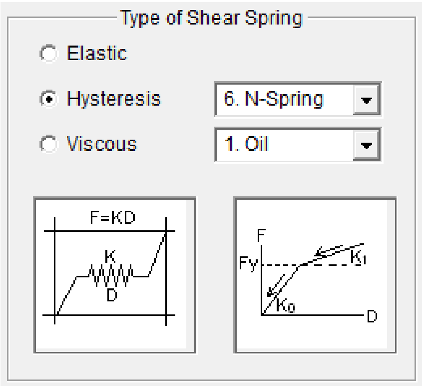


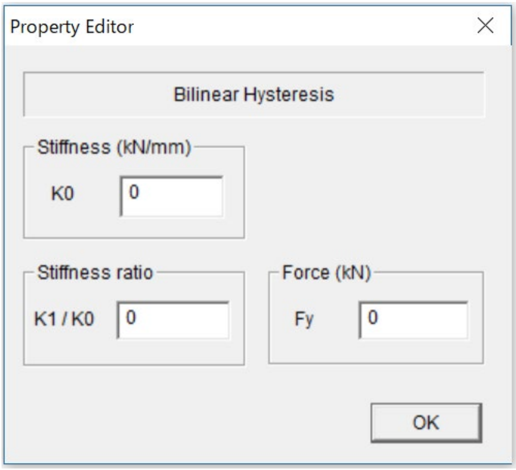
Bouc-Wen





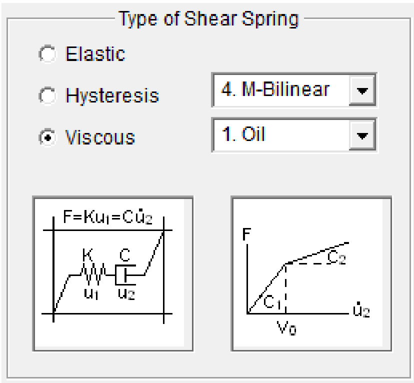
Nonlinear Spring (without hysteresis)

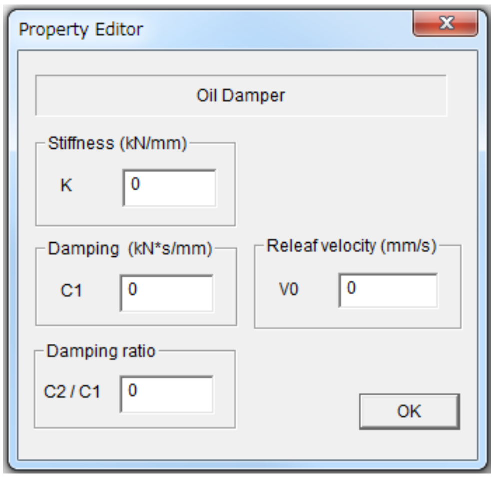




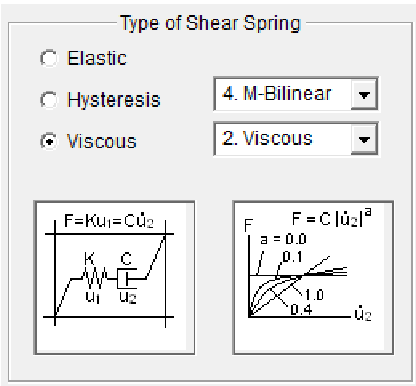
[3] Viscous Dampers

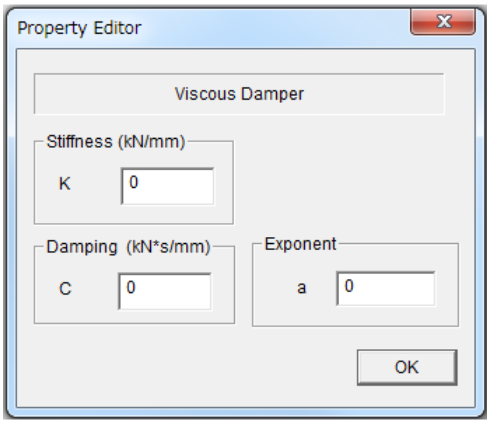
Oil damper



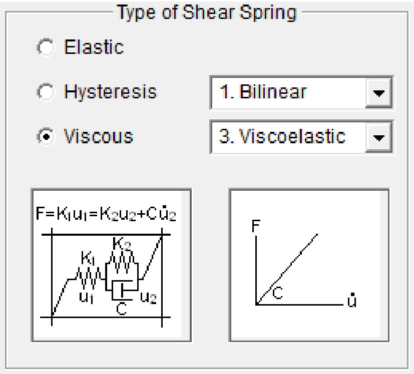


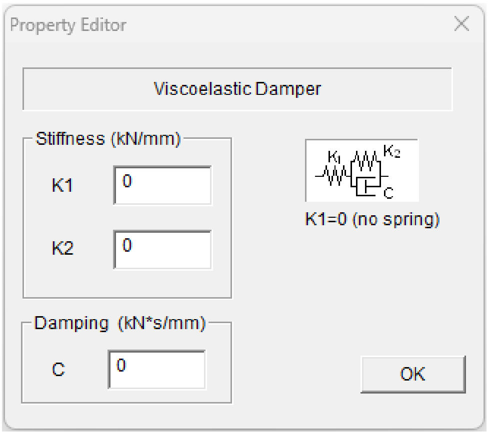
Viscous damper



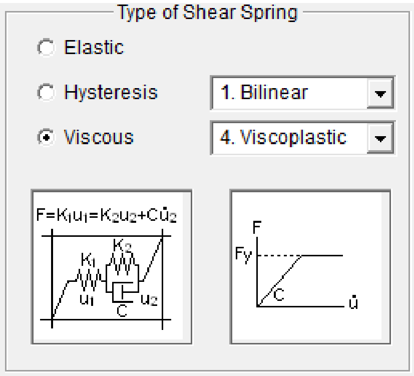


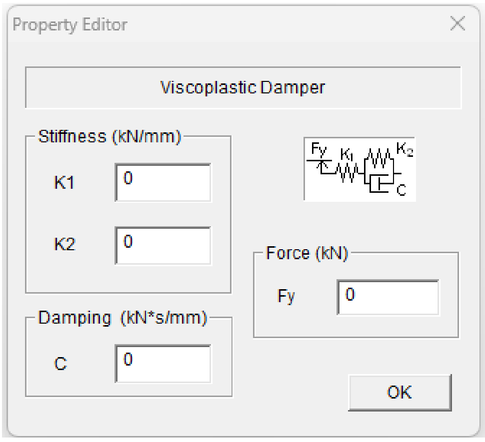
Viscoelastic damper





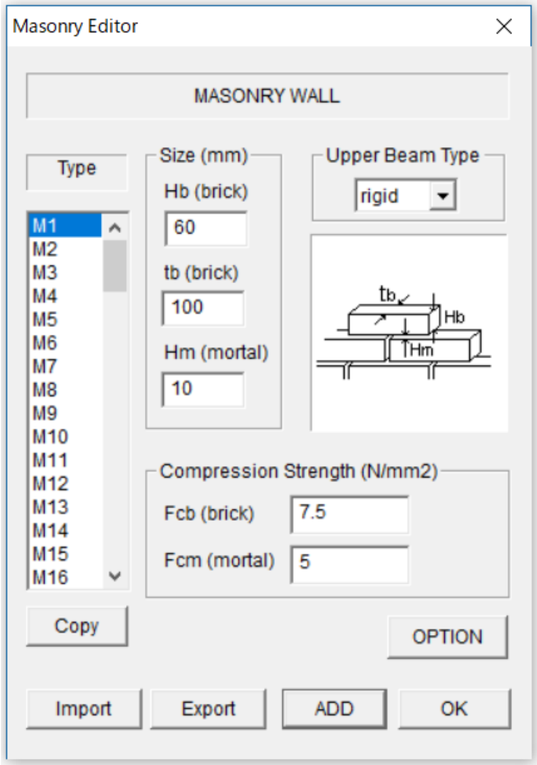
Viscoplastic damper



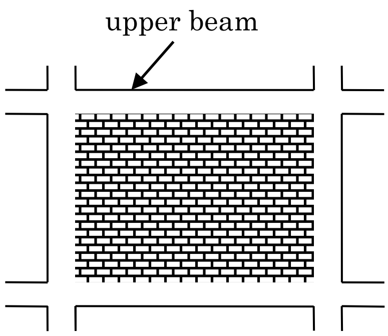


6.24 Masonry Wall

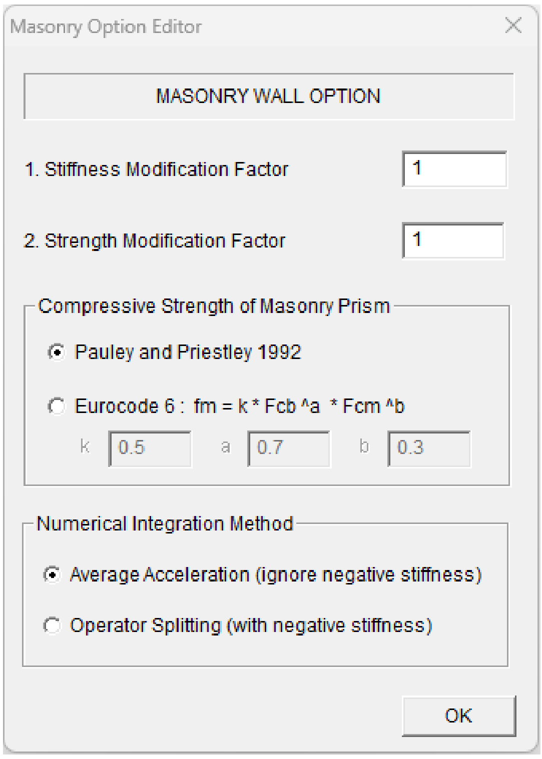
MASONRY ( ) (NOTE: only available when you select Masonry in Option menu)

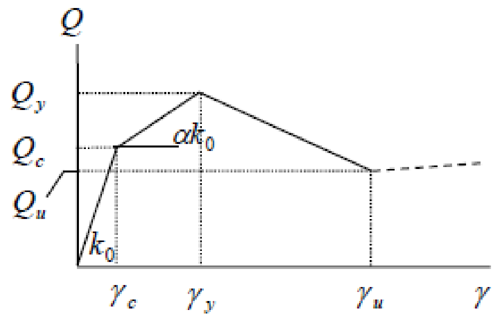


- Please input the size of brick unit and thickness of mortal and compression strength of these materials.
- If there is a reinforcement concrete beam upper of Masonry Wall as shown below, please select the upper beam type number from the pop-up menu. The default is “rigid beam”.



- 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.



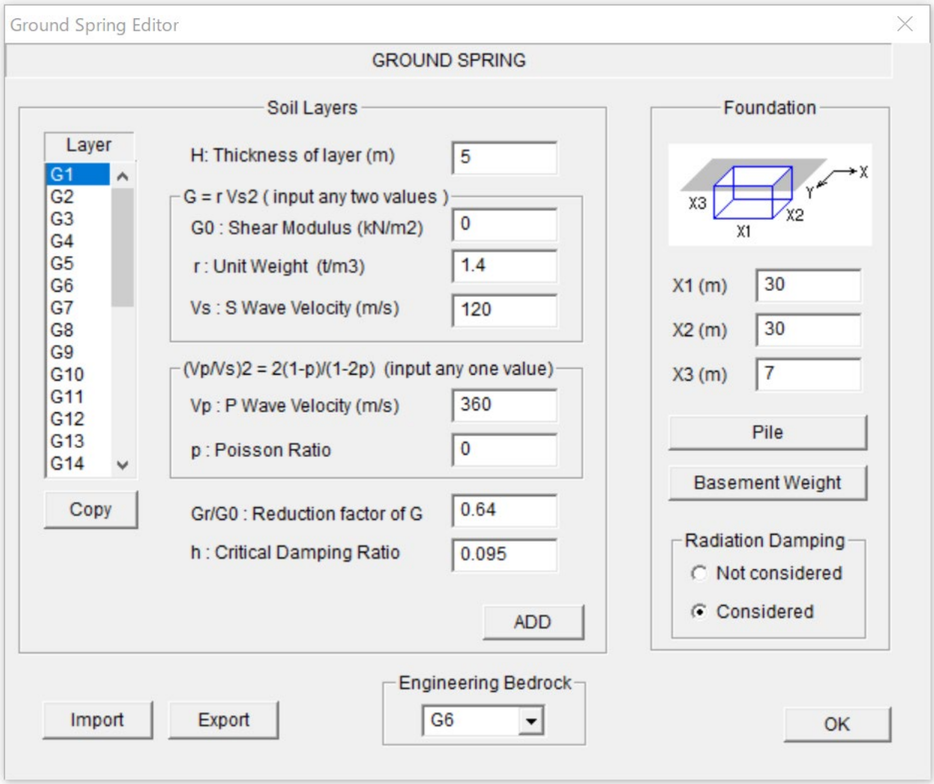


- You can modify the stiffness and shear strength by multiplying modification factors in [OPTION] menu. The default values are 1.0.
- You can select the formula of compression strength of masonry prism from:
  - Pauley and Priestley 1992
  - Eurocode 6

  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.

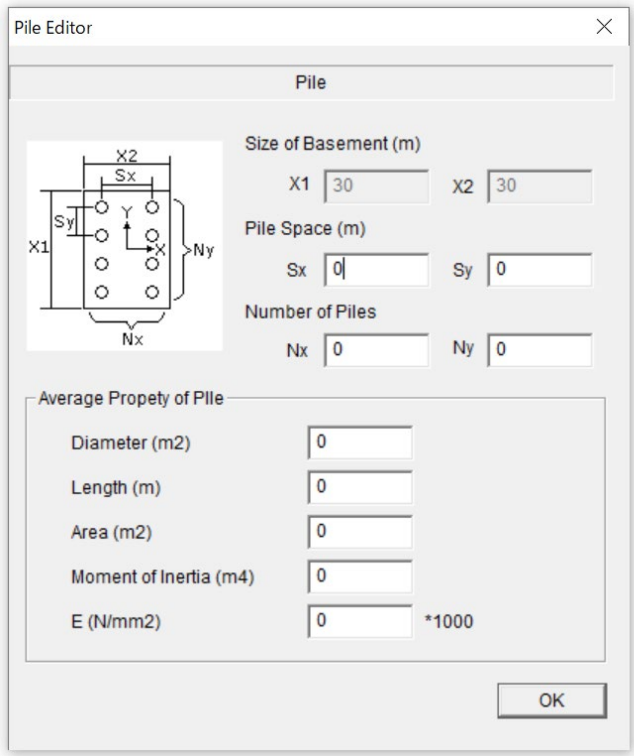
6.25 Ground Spring (Cone model)

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

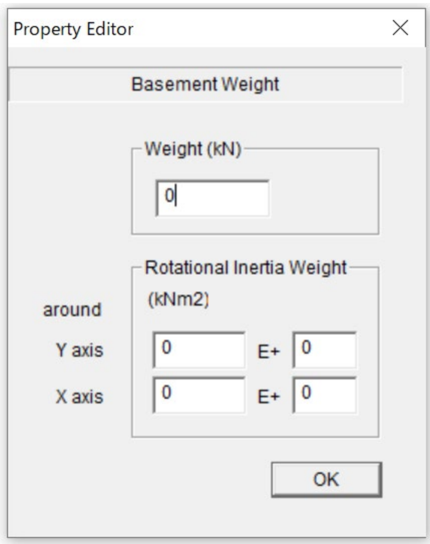


- Please input thickness and soil properties in each layer.
- Since the relationship $G_0 = rV_S^2$ holds, if any two variables are input, the remaining variable is automatically calculated.
- Also, since the relationship $\frac{V_P^2}{V_S^2} = \frac{2(1-p)}{(1-2p)}$ holds, if any one variable except 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.

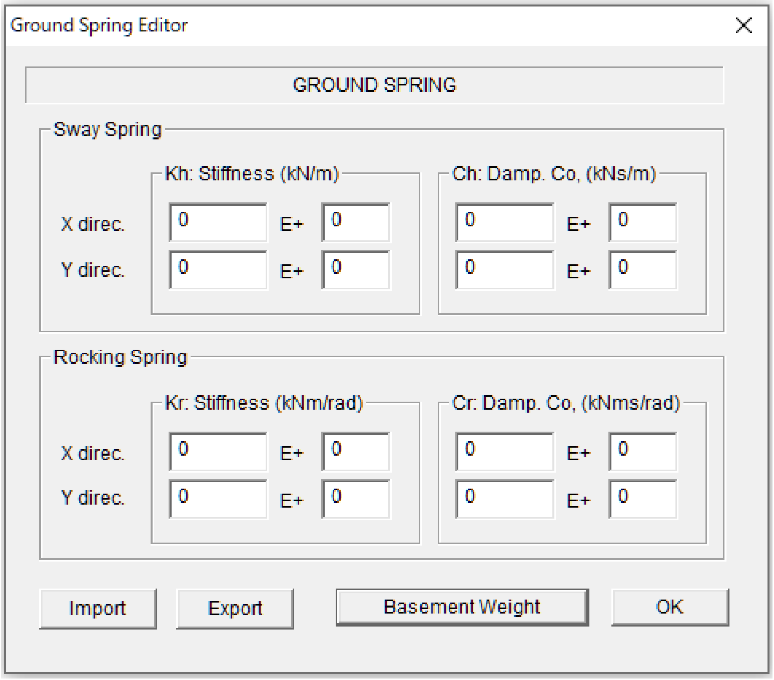


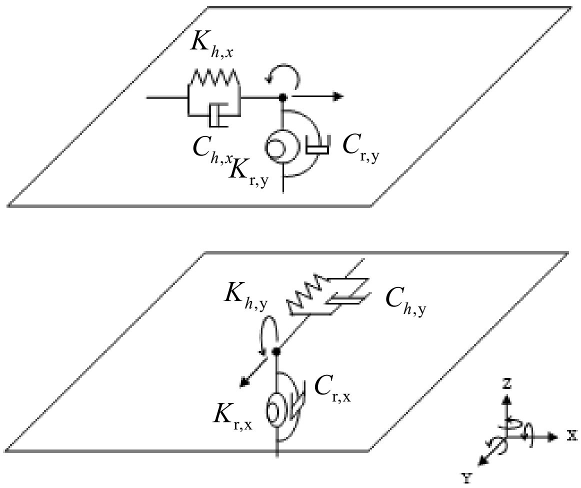
When you click [Basement Weight] Button, the input window of basement weight appears.



6.26 Ground Spring (Direct)

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





- 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.

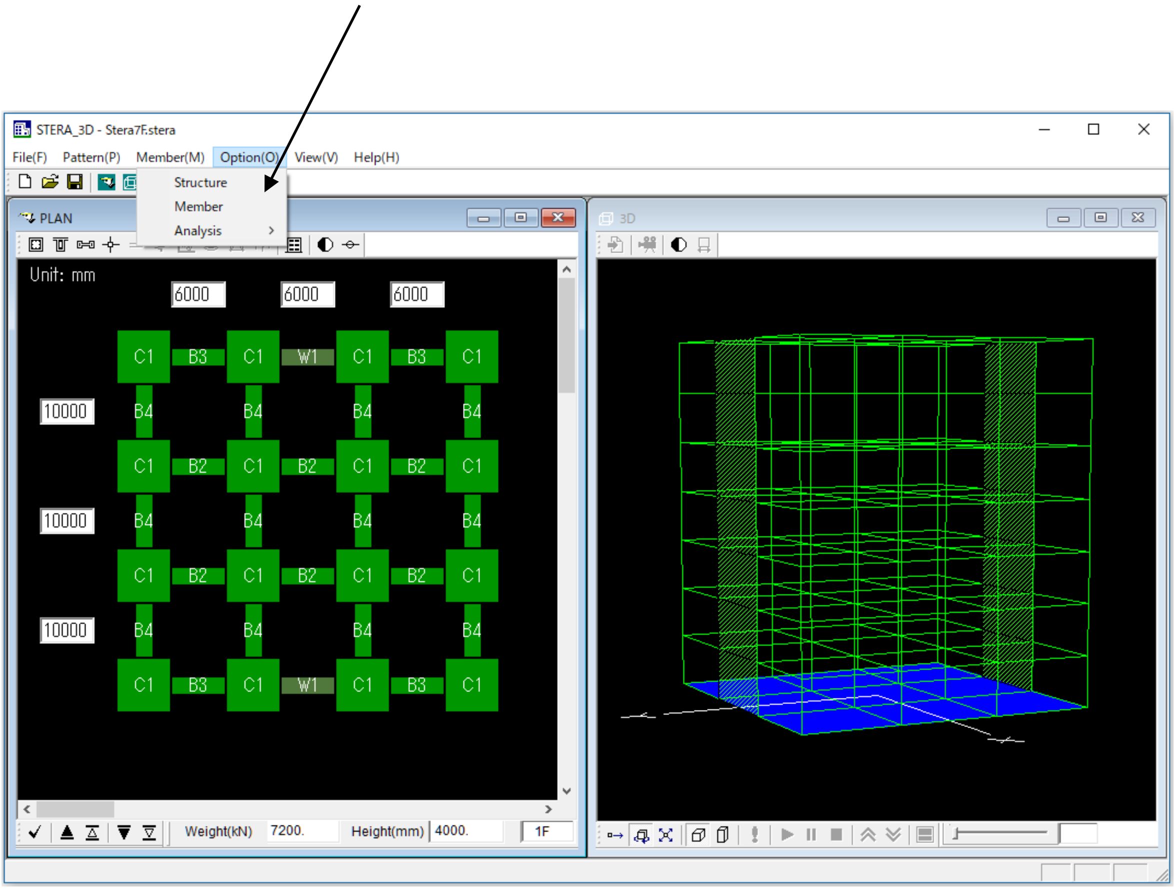
7 Initial Setting of Analysis Condition

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

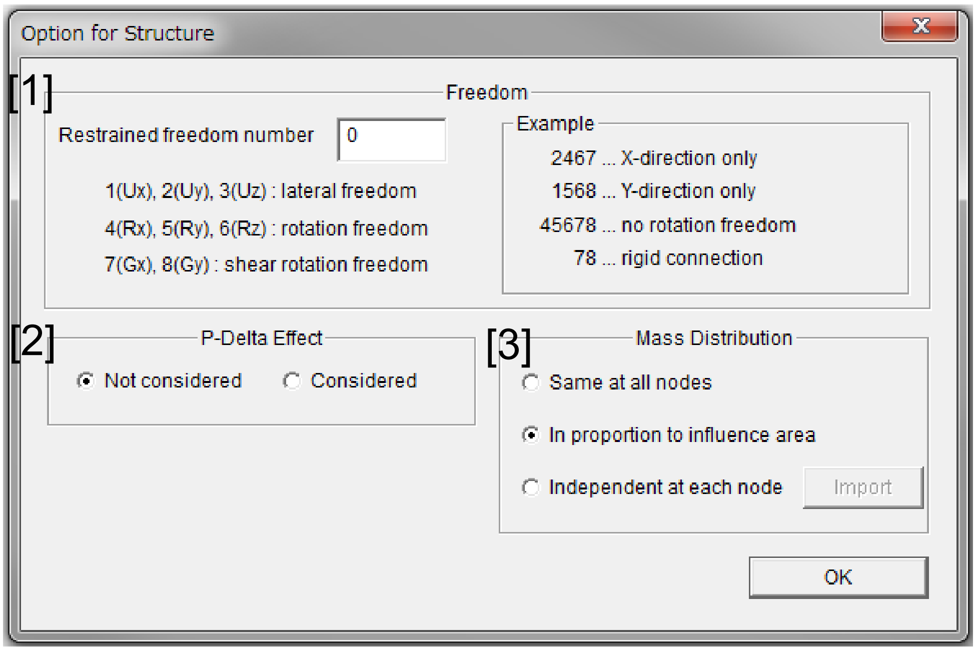
In the default condition,

- 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] Restrained freedom number

Please indicate the freedom numbers to be restrained.

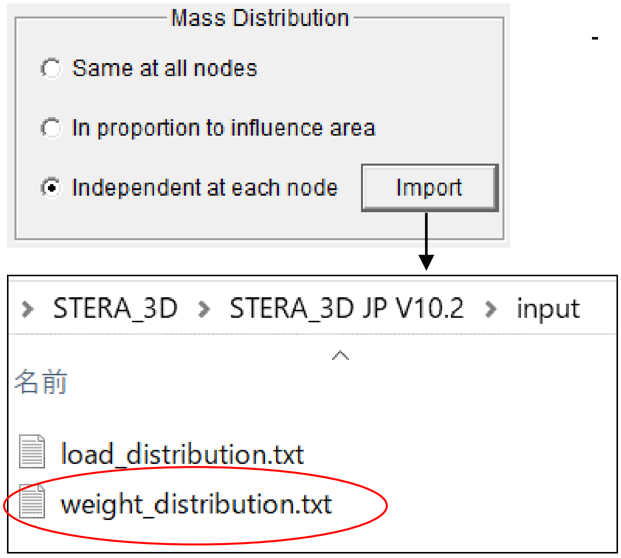
[2] P-Delta Effect

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

[3] Mass distribution at nodes in a floor

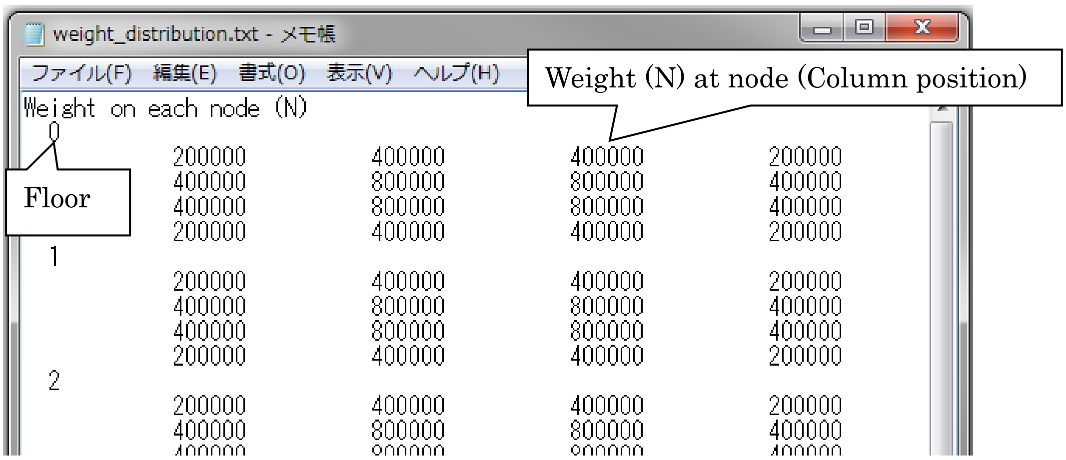
Please select from:

- 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.

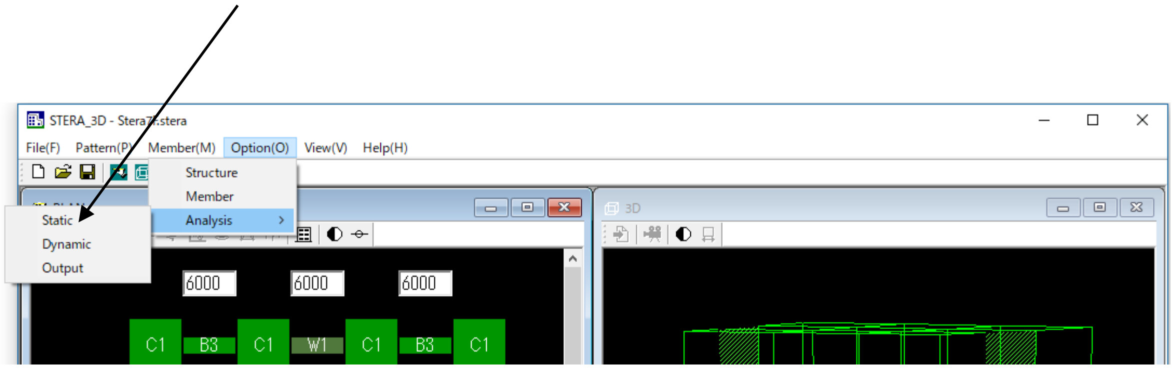


7.2 Condition of static analysis

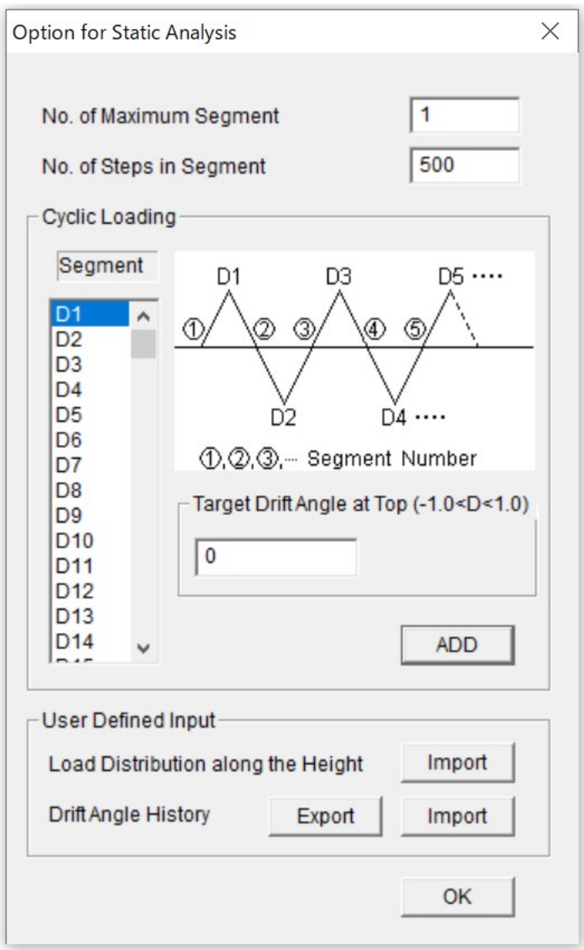
In the default condition,

- 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

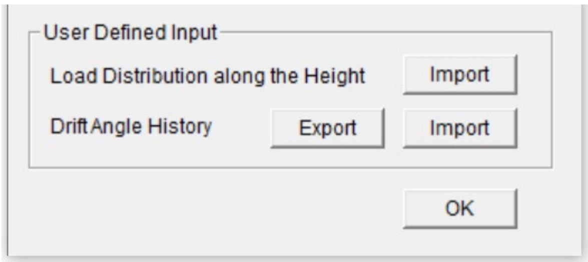
The total number of segments in cyclic loading,

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]
[2]



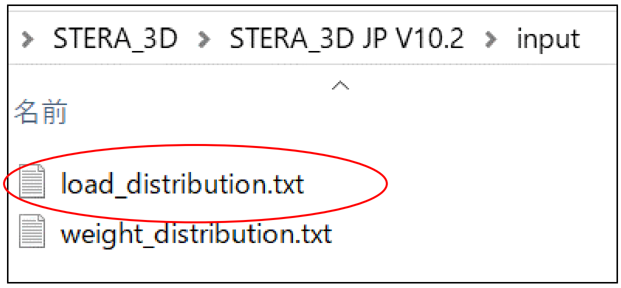
[1] Load Distribution along the Height

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

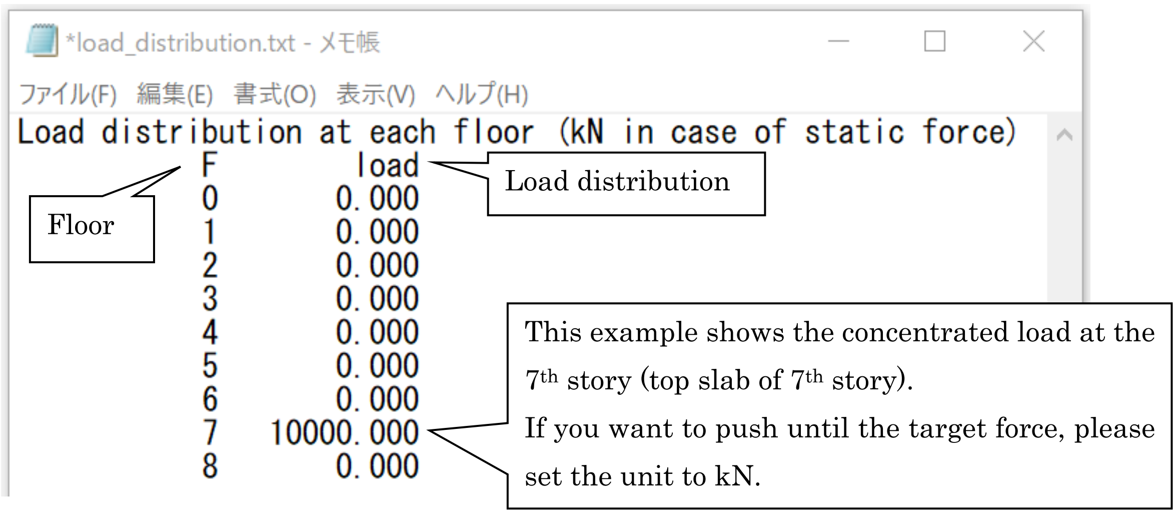
1. Ai 2. Triangular 3. Uniform 4. UBC 5. ASCE 6. Mode 7. User defined

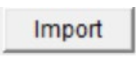
If you want to use [7. User defined] distribution, please create the load distribution file as follows.

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.

Drift_history.txt

```
         6                  ... number of maximum segment
       200                  ... number of steps in one segment
  0.010000                  ... target drift angle at the top (D1)
 -0.010000                  ... D2
  0.020000                  ... D3
 -0.020000                  ... D4
  0.000000                  ... D5
  0.000000                  ... D6
```

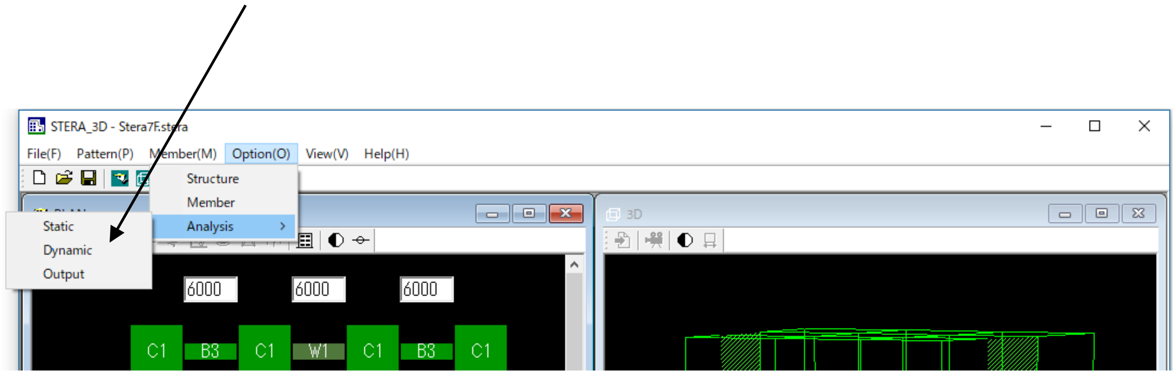
Modify the data and input a new drift angle history.

```
    9
  300
 0.02
-0.02
 0.04
-0.04
 0.06
-0.06
 0.08
-0.08
 0
```

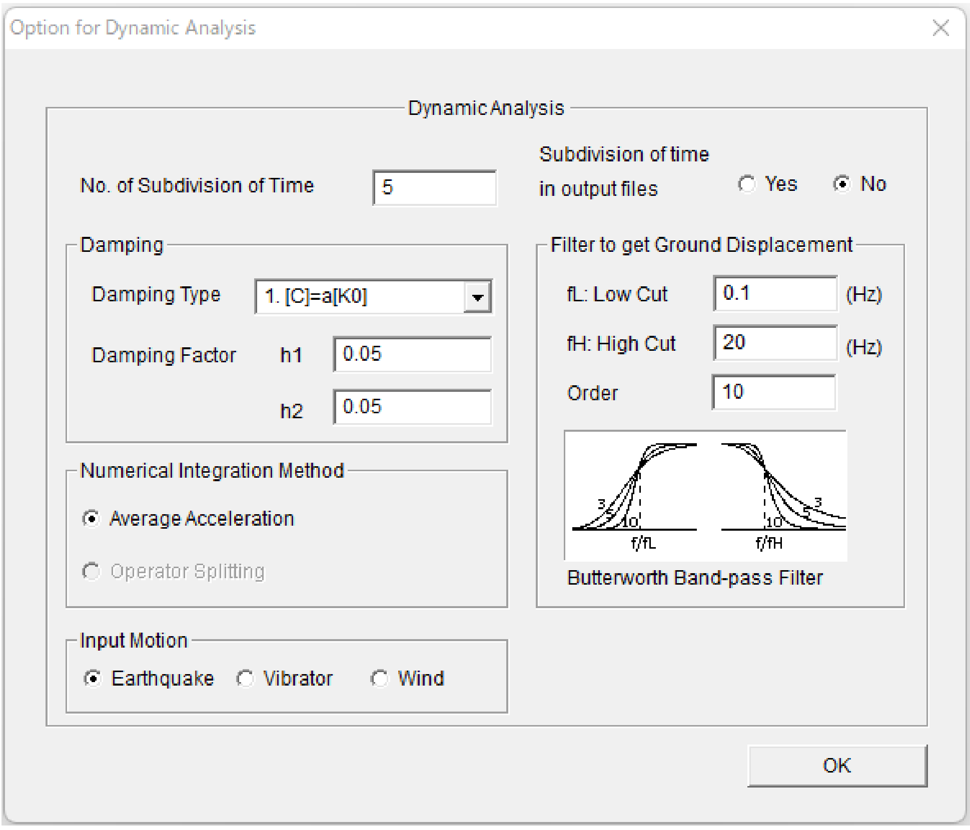
Import the modified file.

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[K0] : proportional to [K0]

  [C] = a[Kp] : proportional to [Kp]

  [C] = a[K0]+b[M] : Rayleigh damping

  The first mode damping factor, h1, is used for type 1) and 2). The second mode damping factor, h2, 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

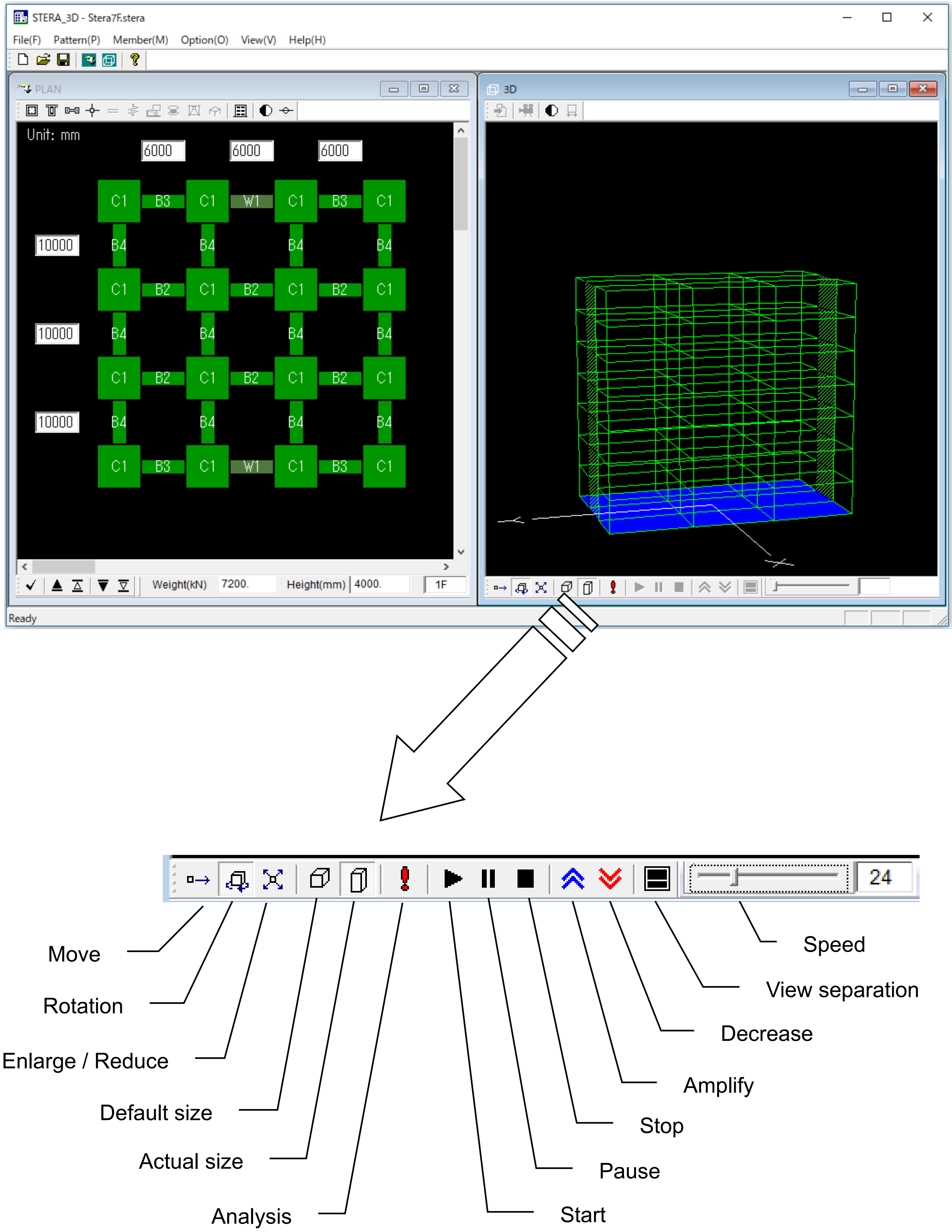
  You can set the parameters of Butterworth band-pass filter to get the ground displacement. The default values are:

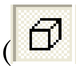
Low cut filter frequency:	0.1Hz
High cut filter frequency:	20Hz
Order of filtering:	10

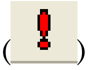
  Please check "Technical Manual" for the detail.

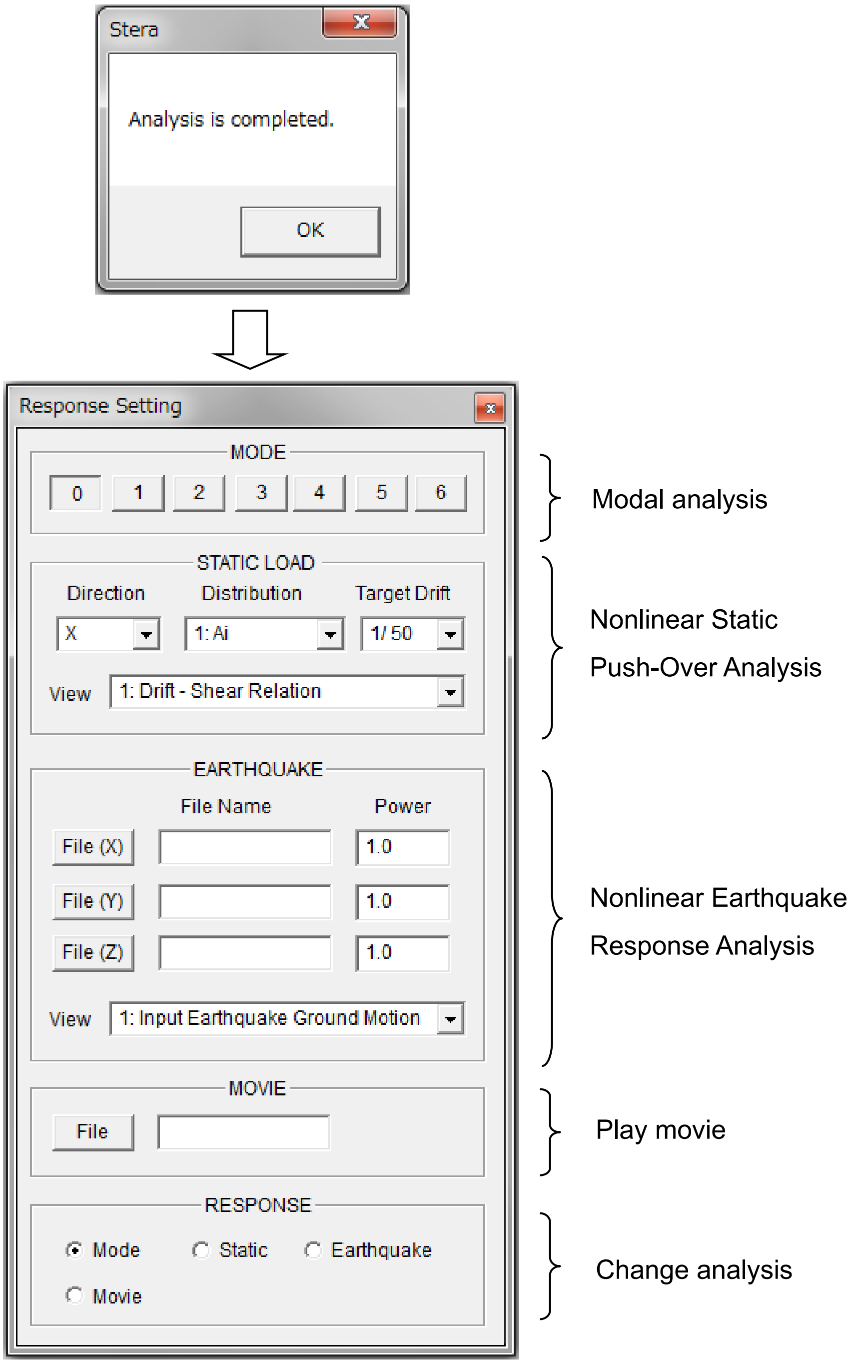
8 3D View of Building and Response

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 (Mx, My, Mz).

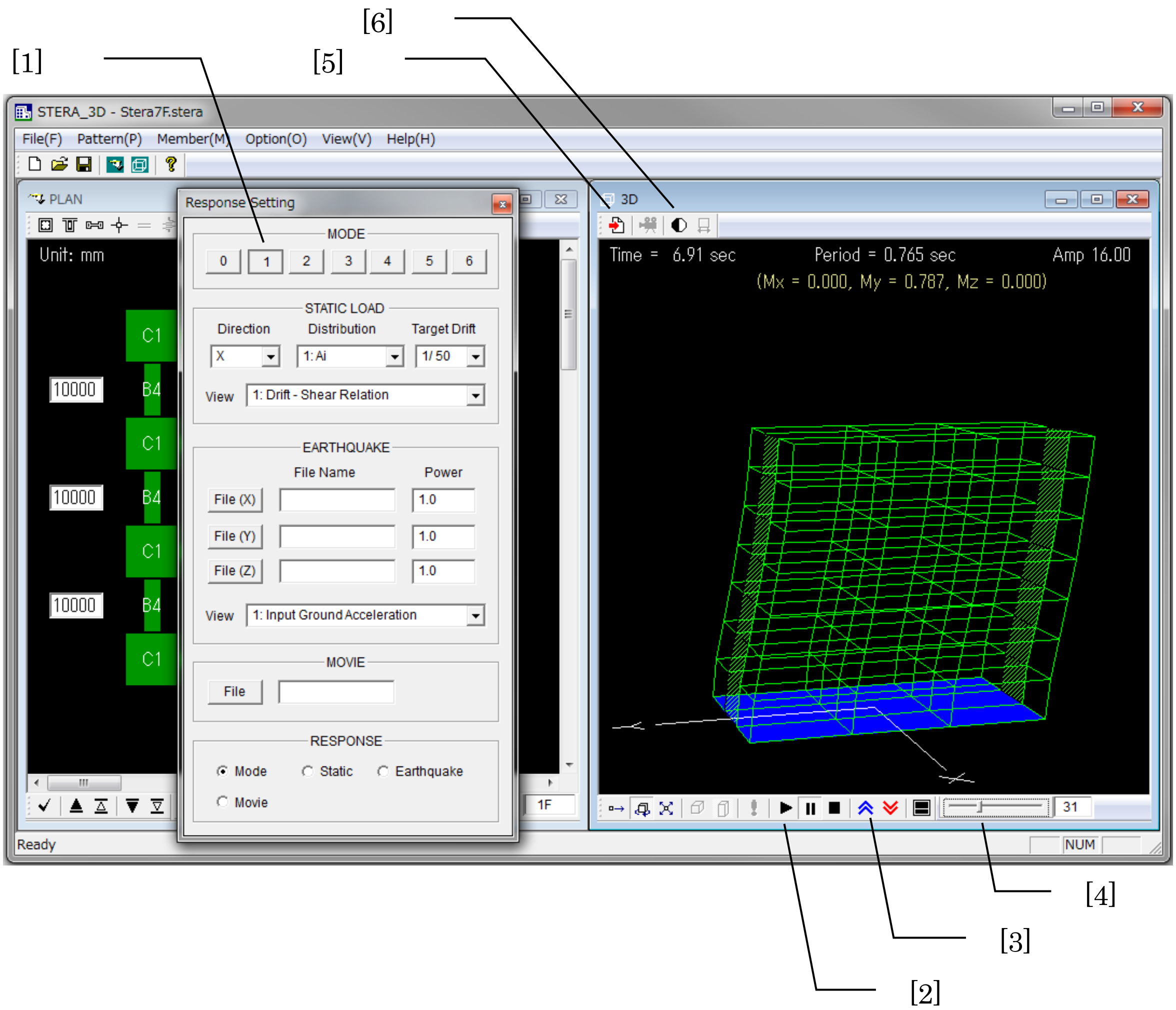
[2] On the 3D VIEW, ( ) starts the vibration of each mode, ( ) stops the vibration and ( ) pauses the vibration.

[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)

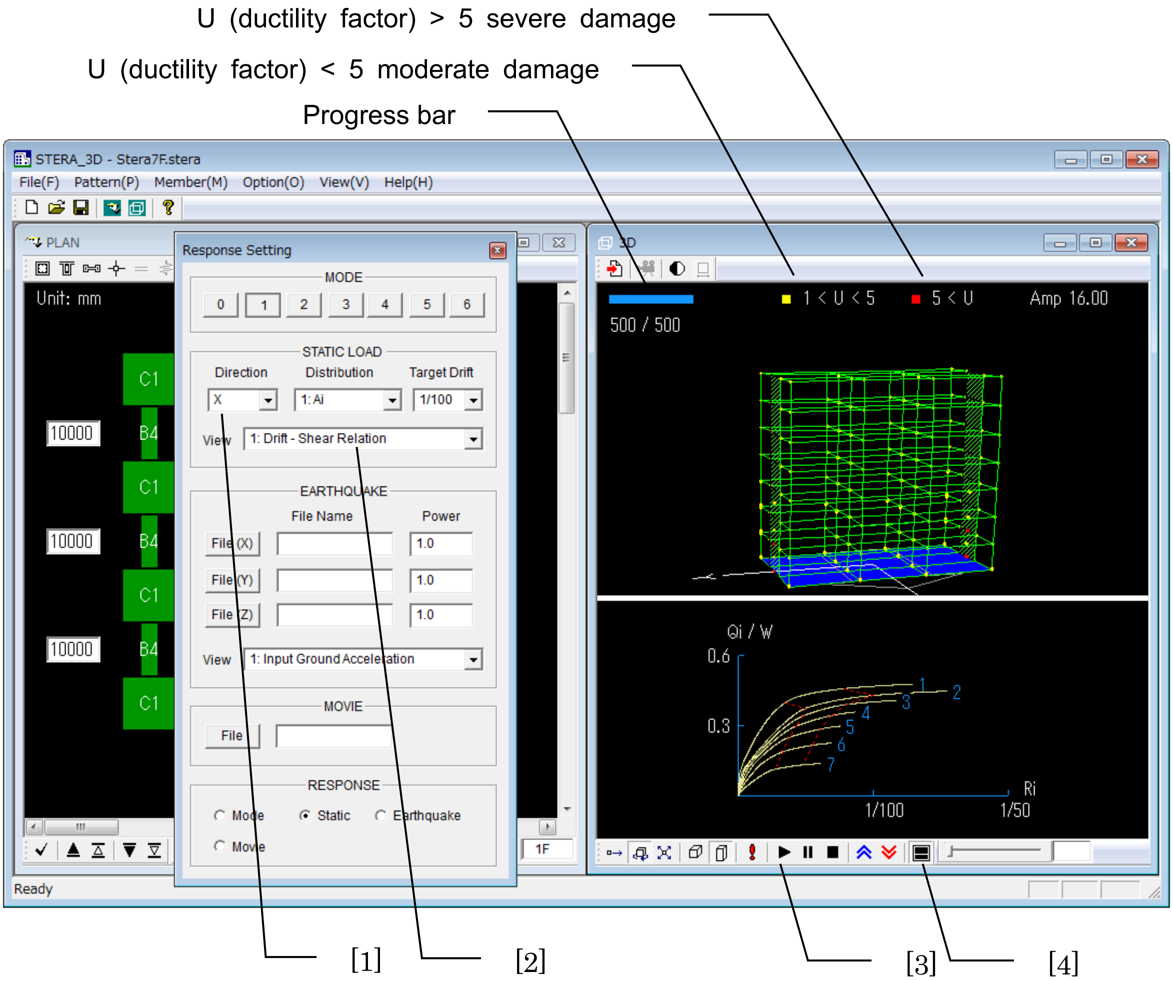
"Distribution": please select a loading distribution along the height of the building. The load is applied at the center of gravity in each floor.
1. Ai 2. Triangular 3. Uniform 4. UBC 5. ASCE
6. Mode 7. User defined

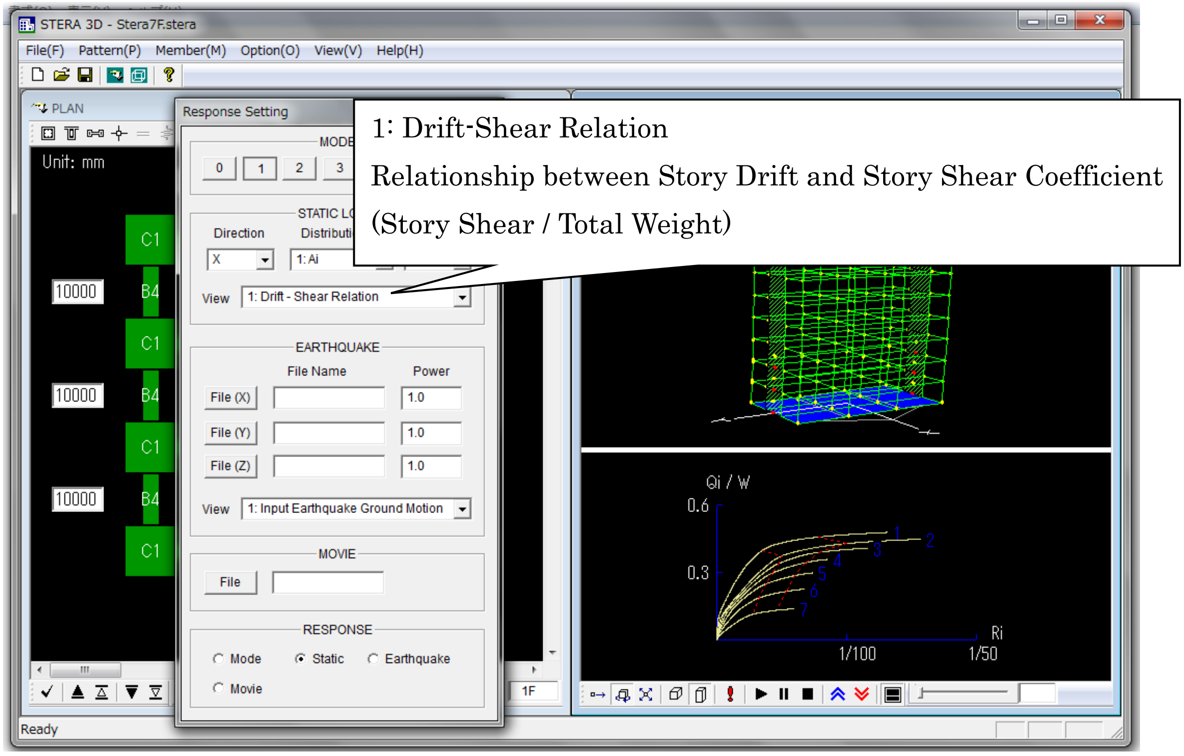
"Target Drift": please set the target drift ratio which is defined by the ratio between the top displacement and the height of the building. "Cyclic" is the cyclic loading and "Force" is the target force as described in 7.2.
1. 1/50 2. 1/100 3. 1/200 4. Cyclic 5. Force

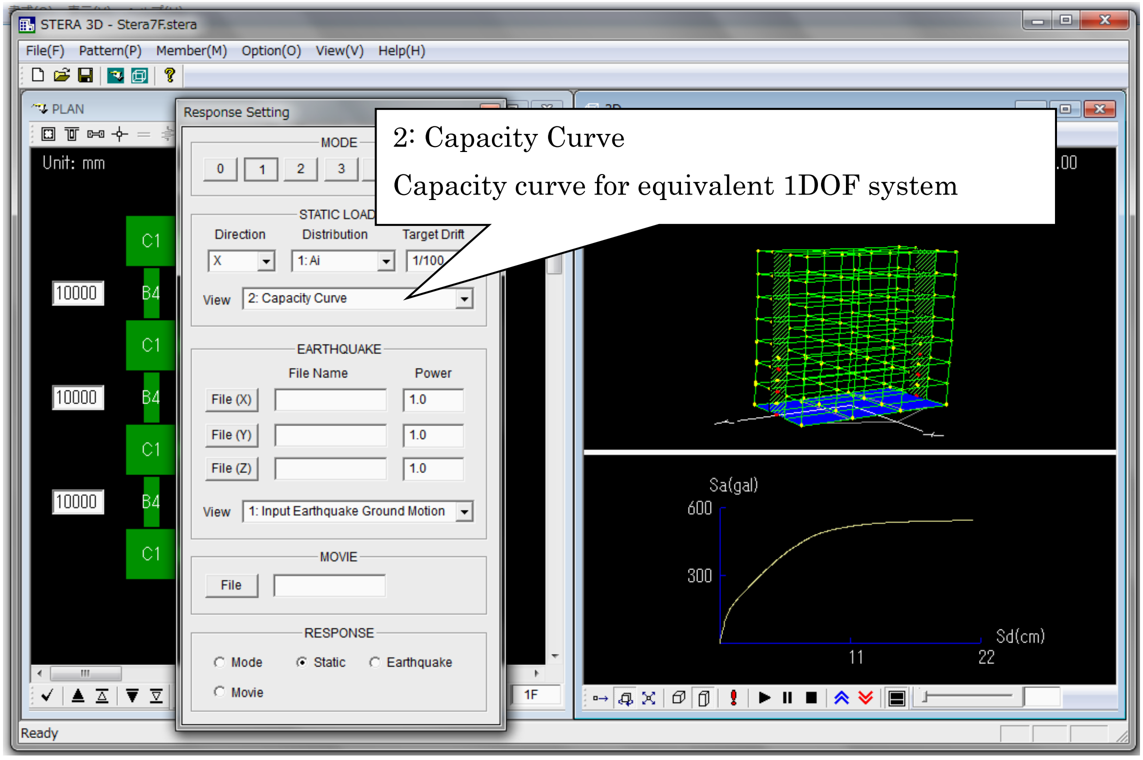
[2] Please select the response for the lower view window.

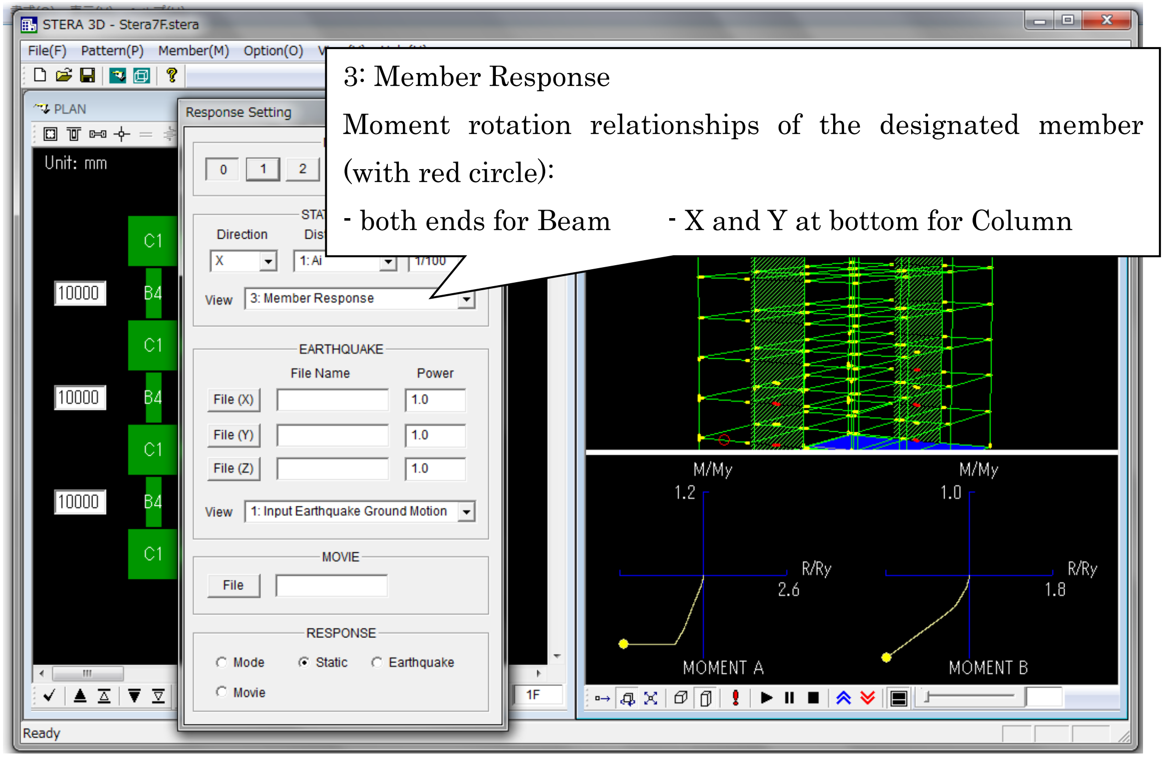
[3] On the 3D VIEW, ( ) starts, ( ) pauses and ( ) stops the response.

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




1: Drift-Shear Relation
Relationship between Story Drift and Story Shear Coefficient
(Story Shear / Total Weight)
STERA 3D - Stera7F.stera
File(F) Pattern(P) Member(M) Option(O) View(V) Help(H)
Response Setting
View 1: Drift - Shear Relation
EARTHQUAKE
View 1: Input Earthquake Ground Motion
MOVIE
RESPONSE
Mode Static Earthquake
Movie
Qi / W
0.6
0.3
Ri
1/100
1/50
Ready


2: Capacity Curve
Capacity curve for equivalent 1DOF system
STERA 3D - Stera7F.stera
File(F) Pattern(P) Member(M) Option(O) View(V) Help(H)
Response Setting
STATIC LOAD
Direction Distribution Target Drift
View 2: Capacity Curve
EARTHQUAKE
View 1: Input Earthquake Ground Motion
MOVIE
RESPONSE
Mode Static Earthquake
Movie
Sa(gal)
600
300
Sd(cm)
11
22
Ready


3: Member Response
Moment rotation relationships of the designated member (with red circle):
- both ends for Beam
- X and Y at bottom for Column
STERA 3D - Stera7F.stera
File(F)
Pattern(P)
Member(M)
Option(O)
PLAN
Unit: mm
Response Setting
Direction
View
3: Member Response
EARTHQUAKE
File Name
Power
File (X)
File (Y)
File (Z)
1: Input Earthquake Ground Motion
MOVIE
File
RESPONSE
Mode
Static
Earthquake
Movie
M/My
R/Ry
MOMENT A
MOMENT B
Ready

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:

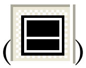
"File(X)": Please select earthquake input file for X-direction.
"File(Y)": Please select earthquake input file for Y-direction.
"File(Z)": Please select earthquake input file for Z-direction (up-down).
"Power": Set the value to amplify the original earthquake

The format of the input file is described in Section 9.1.

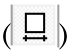
[2] Please select the response for the lower view window.

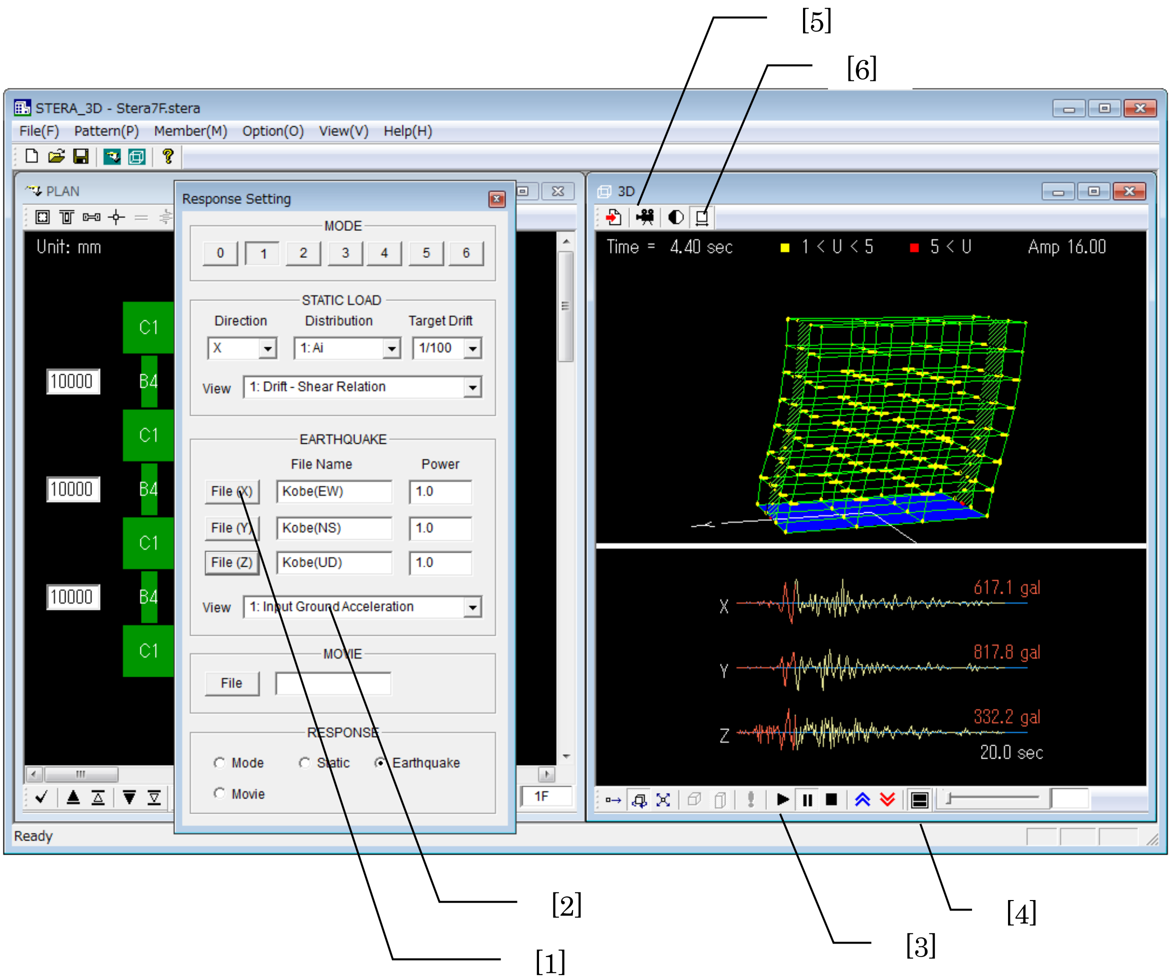
[3] On the 3D VIEW, () starts, () pauses and () stops the response.

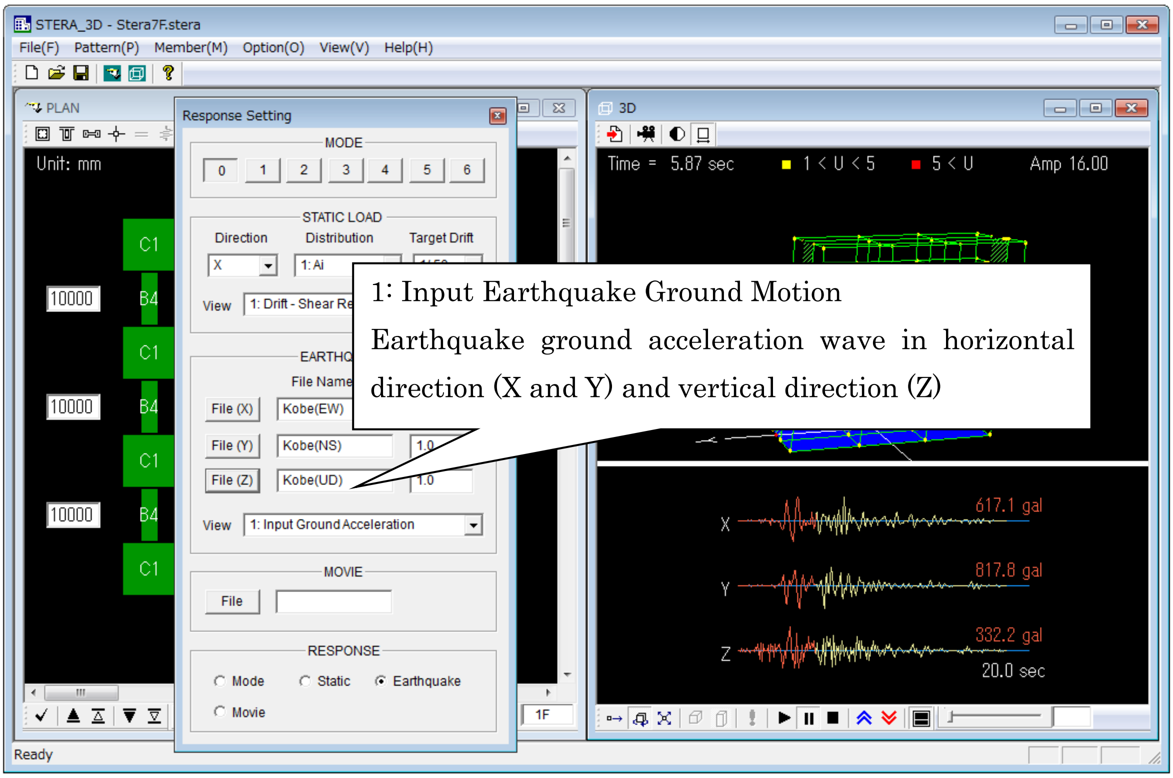
In the lower view, you can see the input earthquake wave and present status.

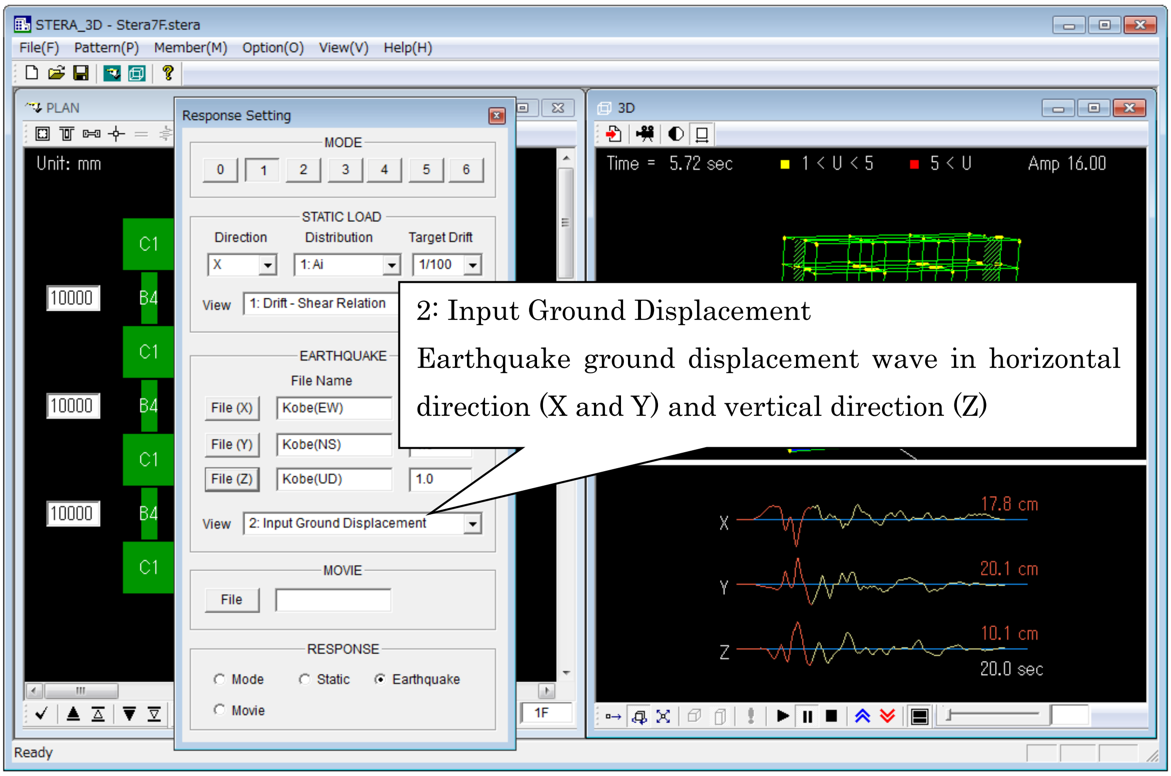
[4] () will change the view from 2-screen to 1-screen and vise versa.

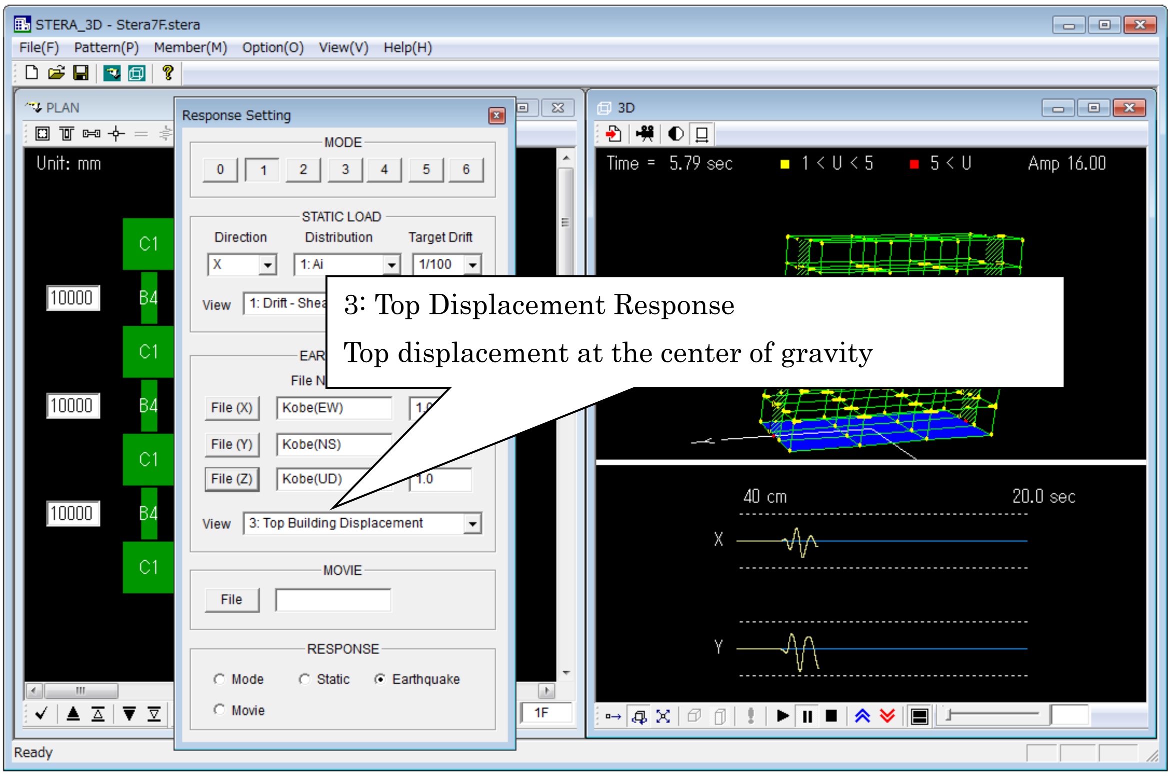
[5] () will save the response animation as a movie file (see 5-5).

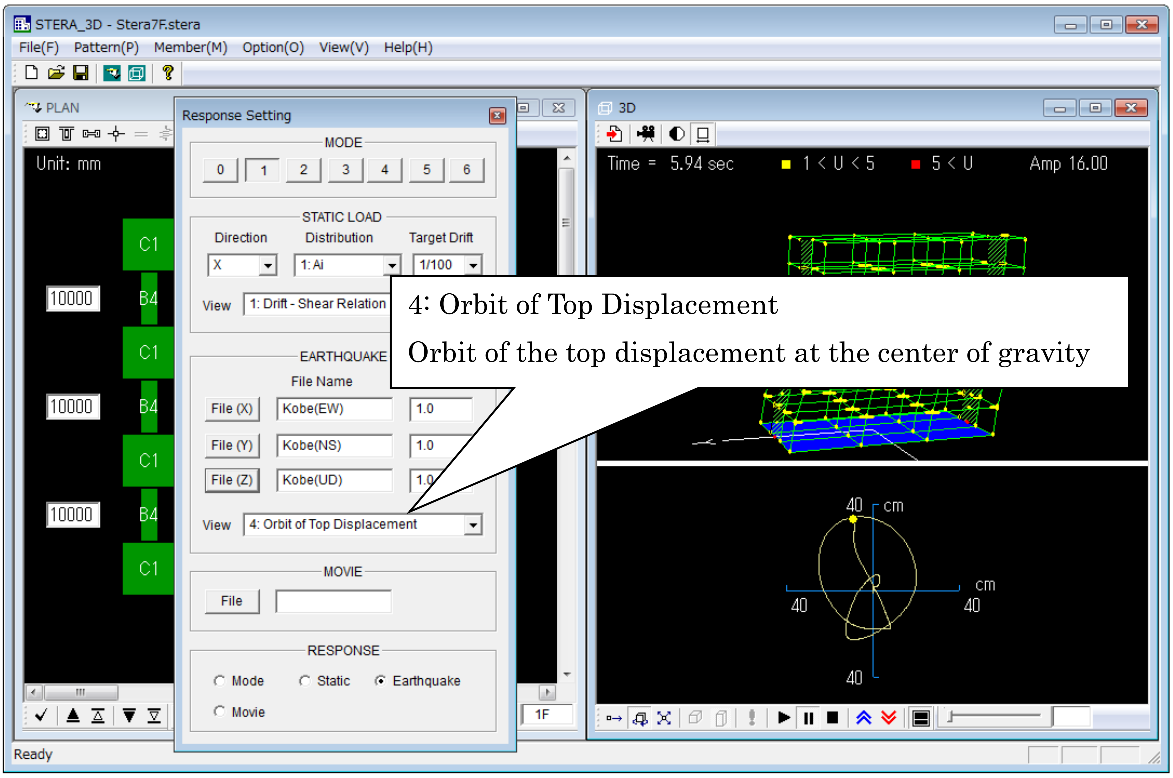
[6] () will switch on and off to include ground movement.

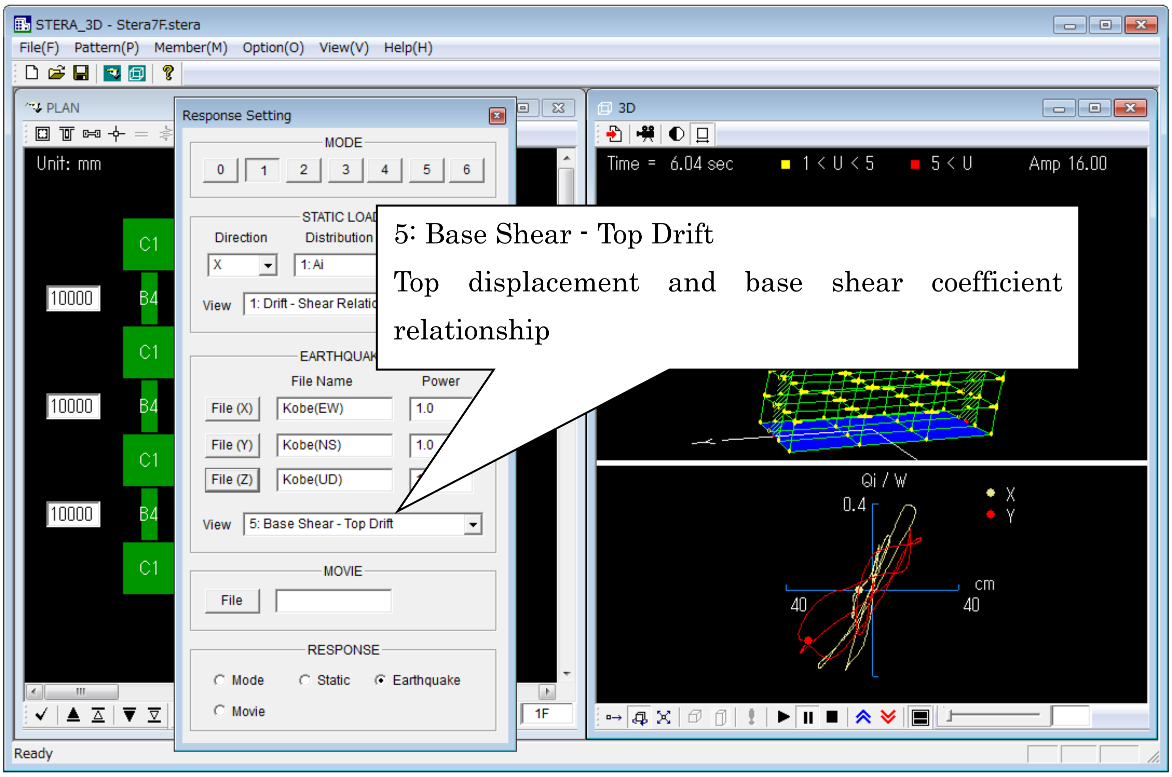


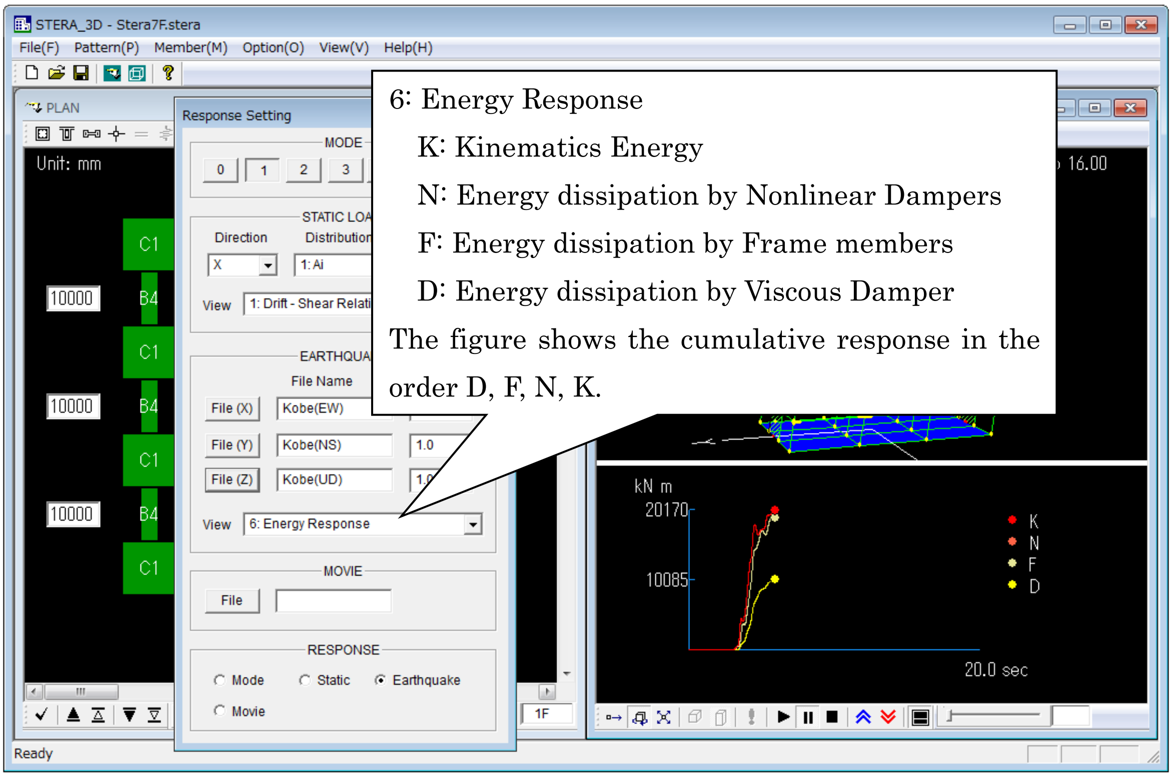

1: Input Earthquake Ground Motion
Earthquake ground acceleration wave in horizontal direction (X and Y) and vertical direction (Z)

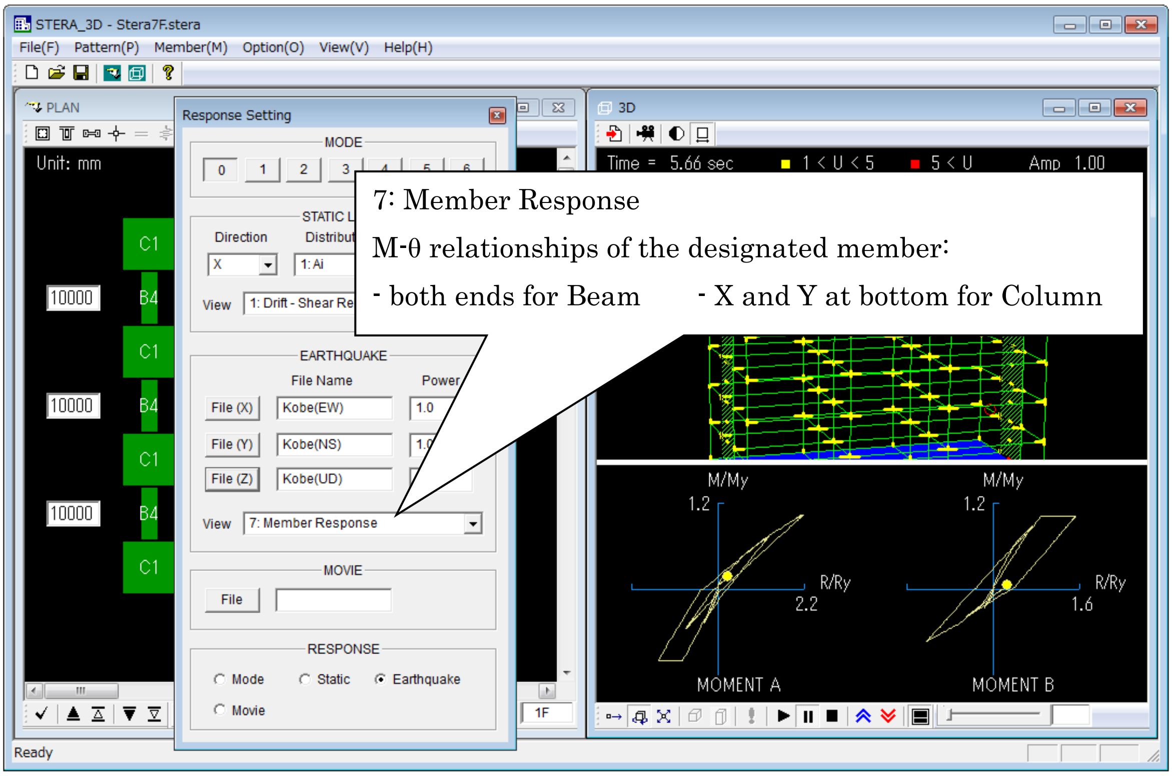

2: Input Ground Displacement
Earthquake ground displacement wave in horizontal direction (X and Y) and vertical direction (Z)


3: Top Displacement Response
Top displacement at the center of gravity


4: Orbit of Top Displacement
Orbit of the top displacement at the center of gravity

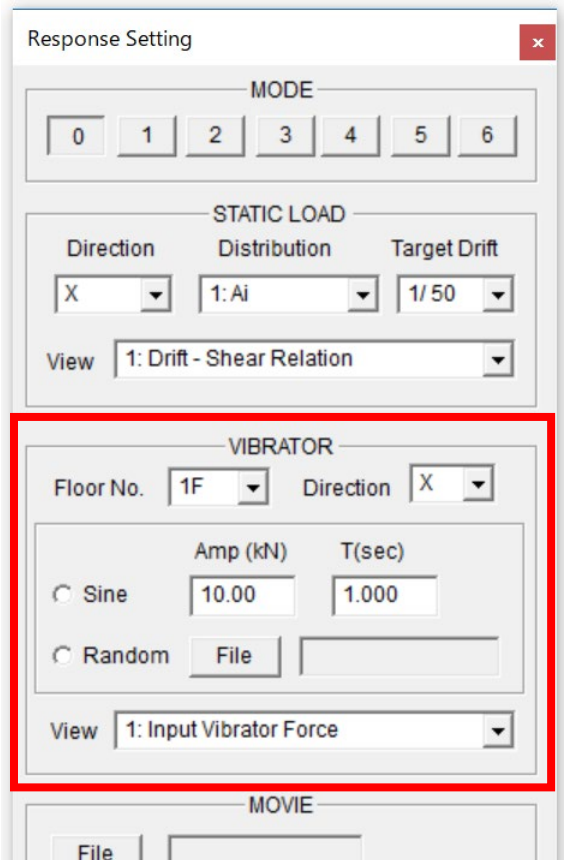

5: Base Shear - Top Drift
Top displacement and base shear coefficient relationship


6: Energy Response
K: Kinematics Energy
N: Energy dissipation by Nonlinear Dampers
F: Energy dissipation by Frame members
D: Energy dissipation by Viscous Damper
The figure shows the cumulative response in the order D, F, N, K.

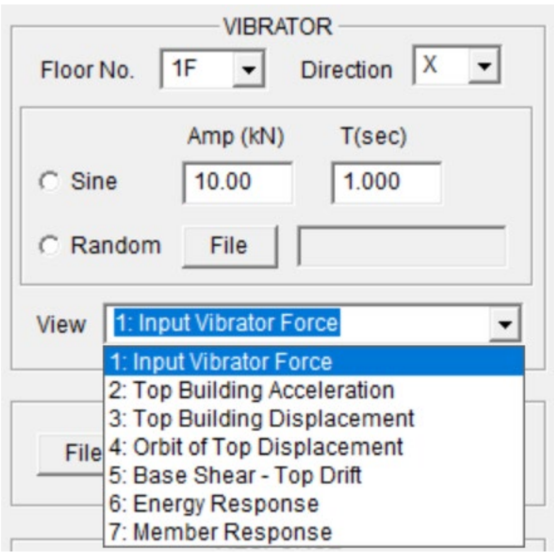

STERA_3D - Stera7F.stera
File(F) Pattern(P) Member(M) Option(O) View(V) Help(H)
PLAN
Unit: mm
C1
10000
B4
Response Setting
MODE
0 1 2 3 4 5 6
STATIC L
Direction
Distribut
X
1: Ai
View
1: Drift - Shear Re
EARTHQUAKE
File Name
Power
File (X)
Kobe(EW)
1.0
File (Y)
Kobe(NS)
File (Z)
Kobe(UD)
View
7: Member Response
MOVIE
File
RESPONSE
Mode
Static
Earthquake
Movie
1F
3D
Time = 5.66 sec
1 < U < 5
5 < U
Amp 1.00
7: Member Response
M-θ relationships of the designated member:
- both ends for Beam
- X and Y at bottom for Column
M/My
1.2
R/Ry
2.2
1.6
MOMENT A
MOMENT B
Ready

8.5 Nonlinear vibrator response analysis

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



- 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.



1. Input Vibrator Force
2. Top Building Acceleration

   Top acceleration at the center of gravity
3. Top Building Acceleration

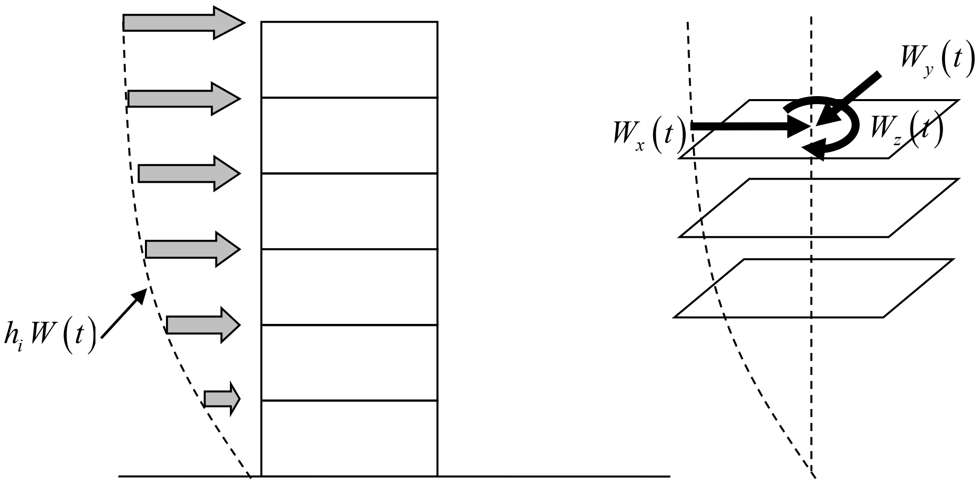
   Top displacement at the center of gravity

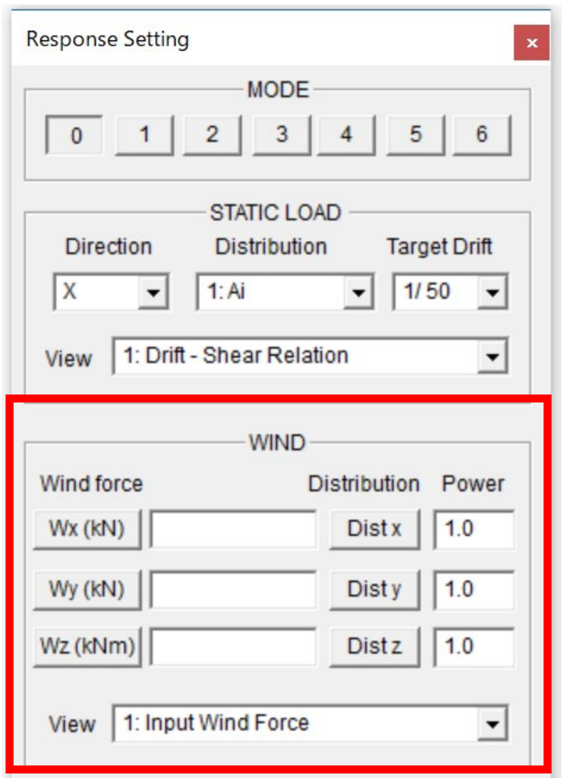
The following menu is the same as Earthquake analysis.

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.

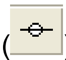




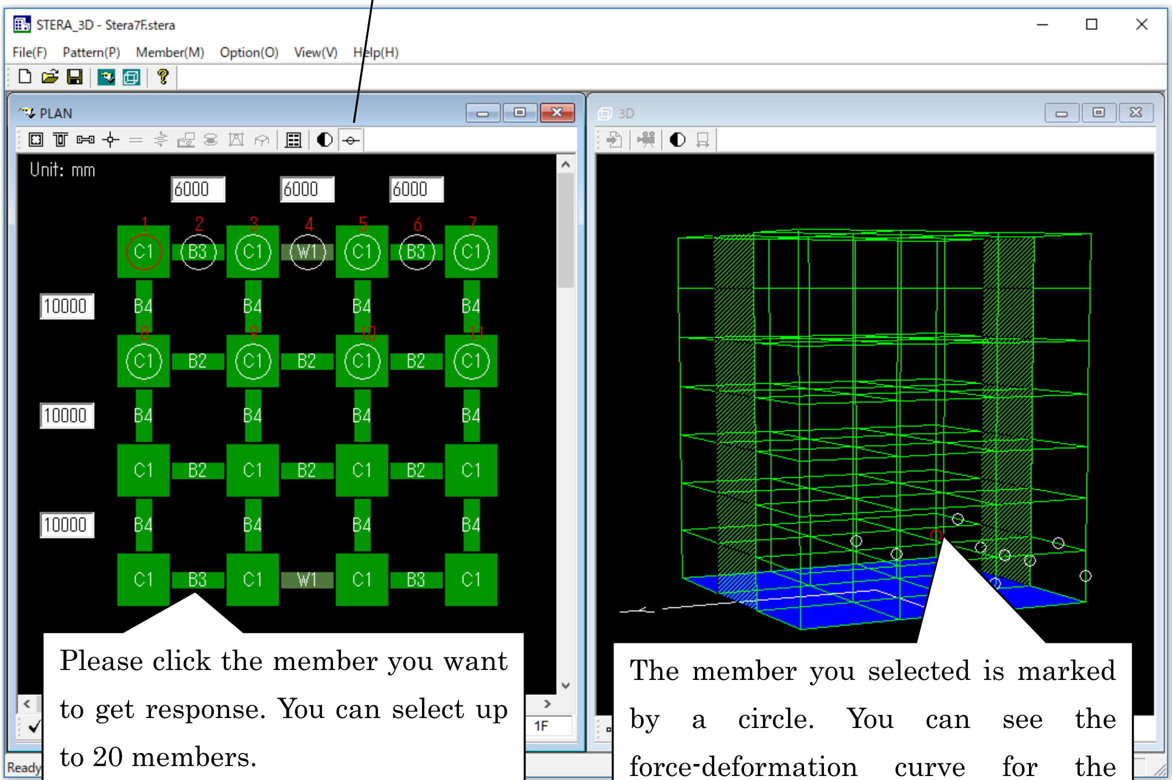
- Wx (kN): Please select an input file for wind lateral force in x-direction.
- Wy (kN): Please select an input file for wind lateral force in y-direction.
- Wz(kNm): Please select an input file for wind torque force in z-direction.
- The format of the input files of wind forces Wx, Wy and Wz are the 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 bottom, you can designate the output member. By one more click, you can cancel it.



Please click the member you want to get response. You can select up to 20 members.

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.

The member you selected is marked by a circle. You can see the force-deformation curve for the member with a red circle.

8.8 Save Nonlinear Earthquake Response as a Movie File

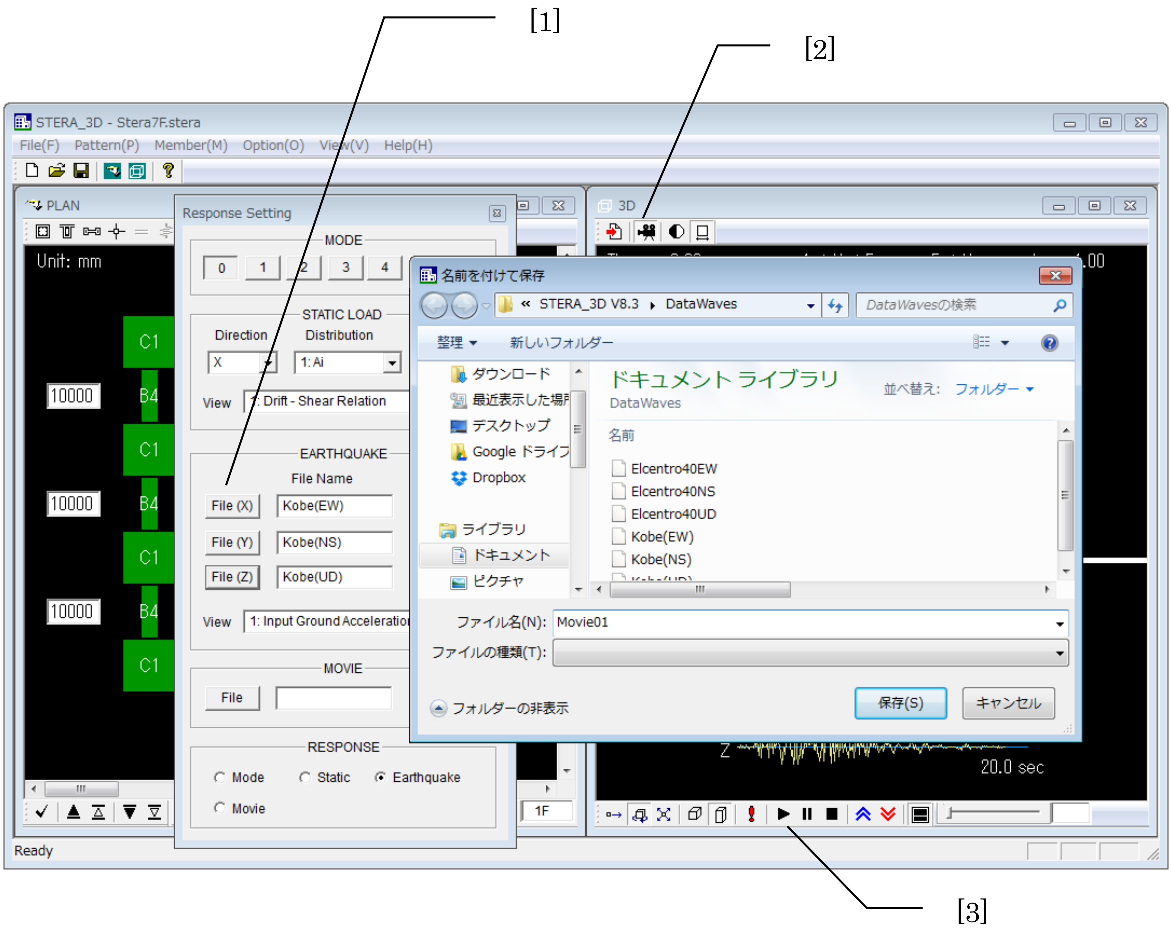
Generally, it takes long time to calculate earthquake response of a building. In this case, you can save the response of the building in a movie file and later you can play the movie to see the response quickly.

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”.

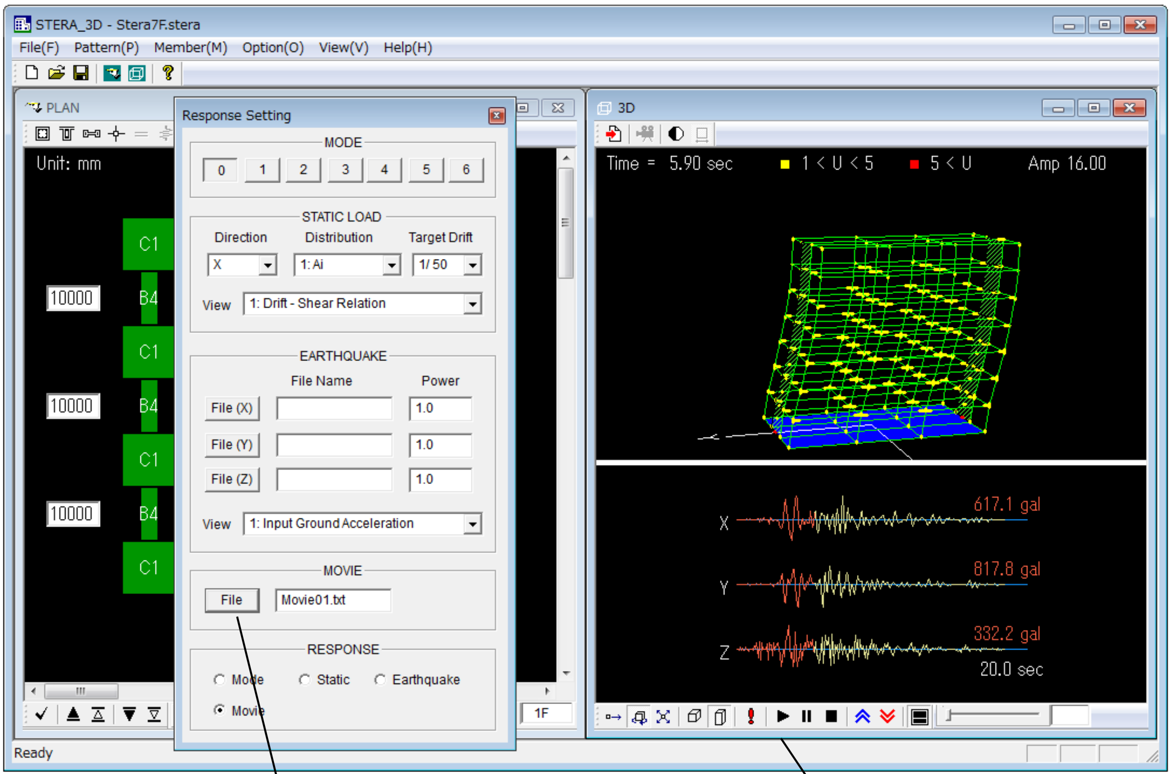
[3] Automatically the recording will start.



2) Play movie

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

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



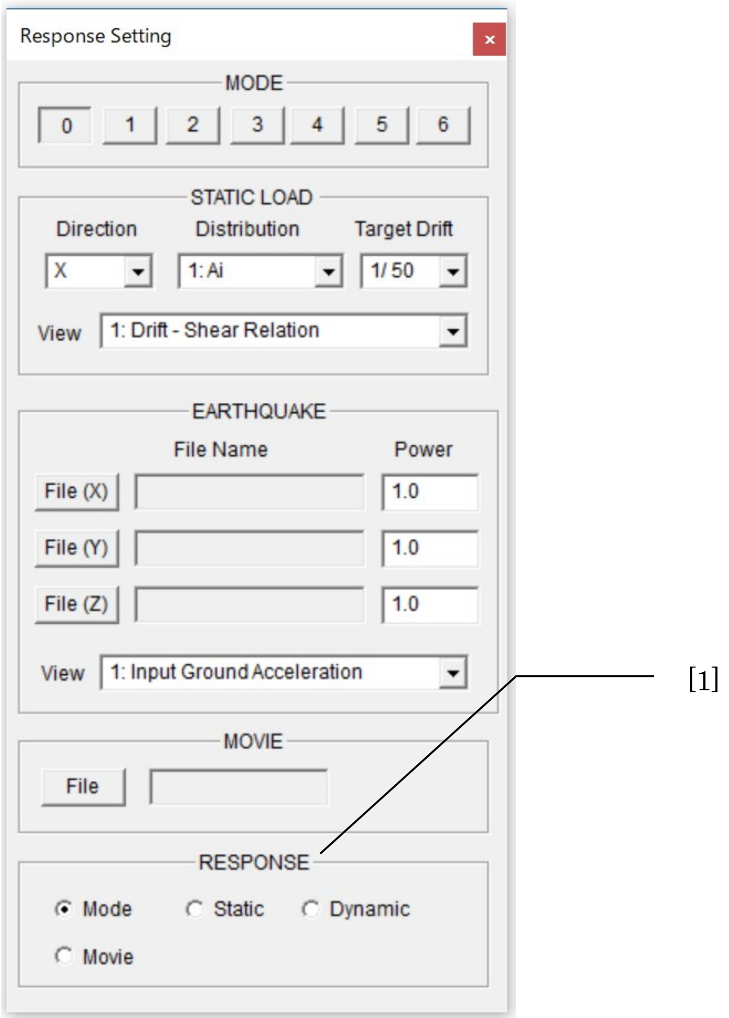
[1]

[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
1st data (NDATA)	INT	Number of data	The number of data for acceleration
2nd data (DT)	REAL	Time interval	(sec)
3rd data and later	REAL	Acceleration (cm/sec^2)	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)

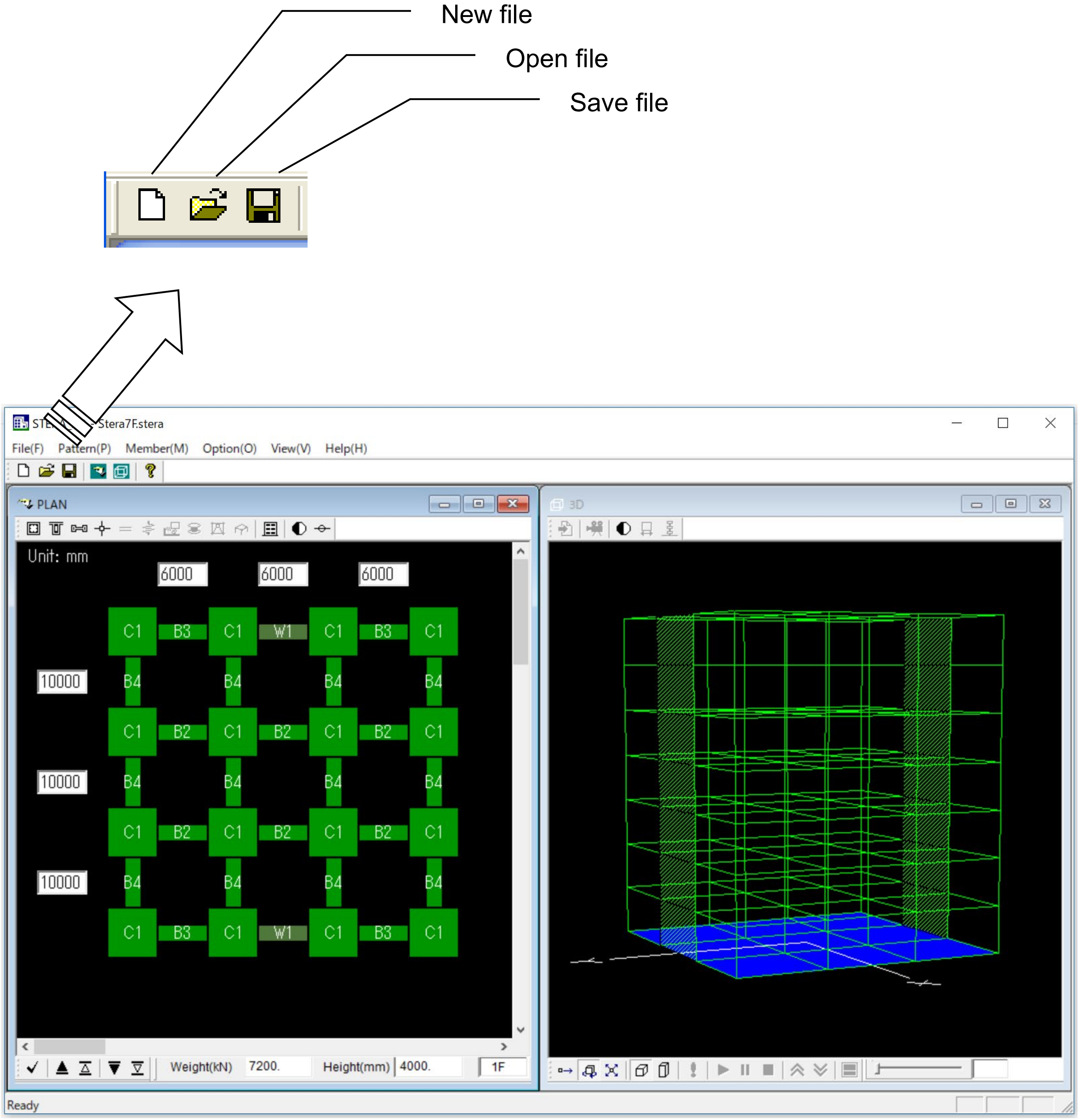
This is the earthquake data “Kobe 1995_NS.txt” in the “./sample/wave/” folder.

```
1000            …NDATA
     0.0200     …DT (time interval, sec)
     0.70    0.70   -0.30   -2.00   -2.90   -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. Acceleration data (cm/sec2)  00   -5.80  -13.50  -26.60
   -20.60   24.10   65.30   44.70    0.90  -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

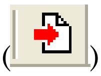
You can save the building data in a file and open it later. The file has an extension ".stera".



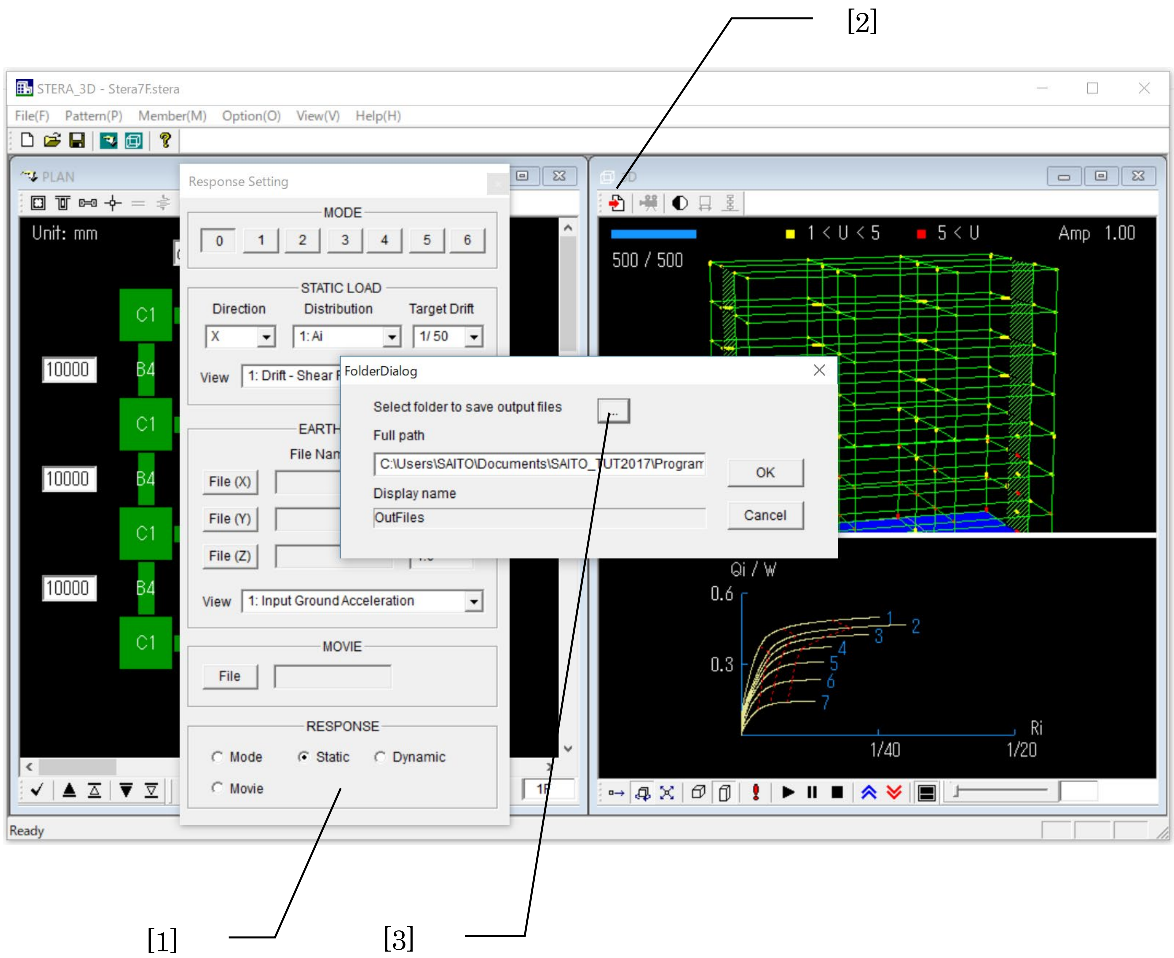
10.2 Save results of analysis in text files

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

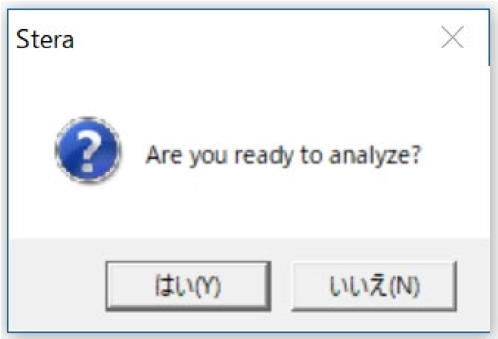
[1] On the RESPONSE SETTING DIALOG, please set the condition of analysis.

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

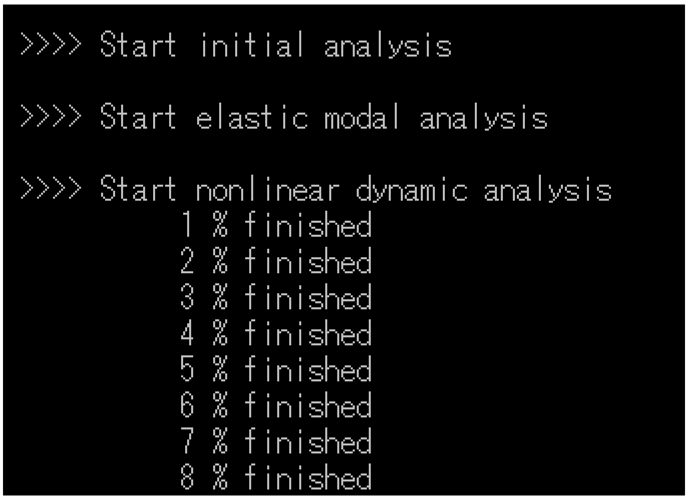
[3] Please select folder to save output text files.

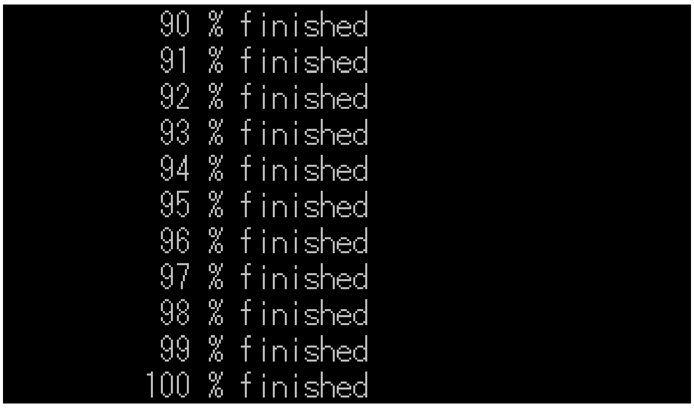


[4] When you push “OK”, a message window appears to start calculation and save output data to the designated folder.



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





10.3 Output text files

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

- data_beam.txt
- data_bi.txt
- data_column.txt
- data_damper.txt
- data_floor.txt
- data_ground.txt
- data_panel.txt
- data_pulley.txt
- data_spring.txt
- data_structure.txt
- data_wall.txt

data_***

Input data of members and building

beam	: Beam
bi	: Base Isolator
column	: Column
damper	: Damper and Nonstructural wall
floor	: Floor slab
ground	: Ground spring
node	: node
panel	: connection panel
spring	: External spring
structure	: Building
wall	: Wall

- max_beam.csv
- max_bi.csv
- max_column.csv
- max_damper.csv
- max_floor.csv
- max_ground.csv
- max_member01.csv
- max_member02.csv
- max_member03.csv
- max_node.csv
- max_panel.csv
- max_pulley.csv
- max_spring.csv
- max_structure.csv
- max_wall.csv
- mode_node.csv

max_***

Maximum response of members and building

mode_node

Mode participation vector of each node

- response_eigen.csv
- response_energy.csv
- response_member01.csv
- response_member02.csv
- response_member03.csv
- response_structure.csv

response_eigen
: Natural period and mode

response_energy
: Energy response

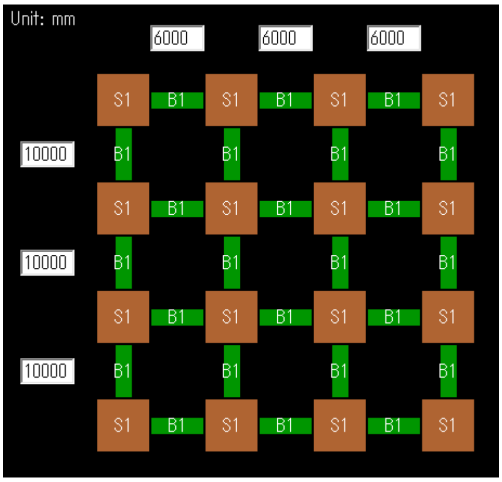
response_floor01, 02, …
: Response of 3D rigid floor (6 components)

response_member01, 02, …
: Response of designated members

response_structure
: Response of floors (horizontal components)

[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
```

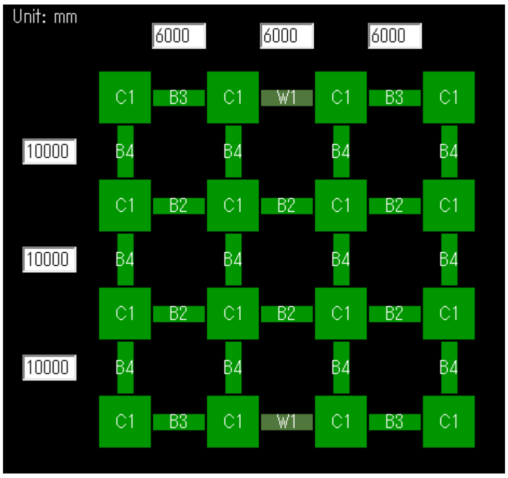



```
--- member properties (cm, kN)  member =  1 ---( type =  1 )
   b    :    60.000   d    :   150.000  slab :    15.000
   Ec   : 0.230E+04
   area : 11360.820
   Iy   : 0.284E+08
steel reinforcement
   (up)       10- at = 11.400
   (down)     10- at = 11.400
slab reinforcement
     1- at =  0.713 @  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
```

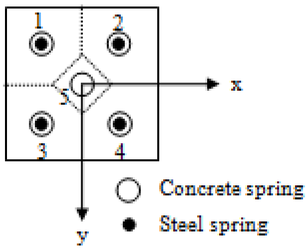
```
b: width d: height  slab: thickness
Ec: Young’s Modulus
area: Area
Iy: Moment of inertia

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)
 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
       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 =  9.566
  (X-side)       4- at =  9.566
  (Y-side)       4- at =  9.566
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 =  0.156E+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 =  0.156E+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
```



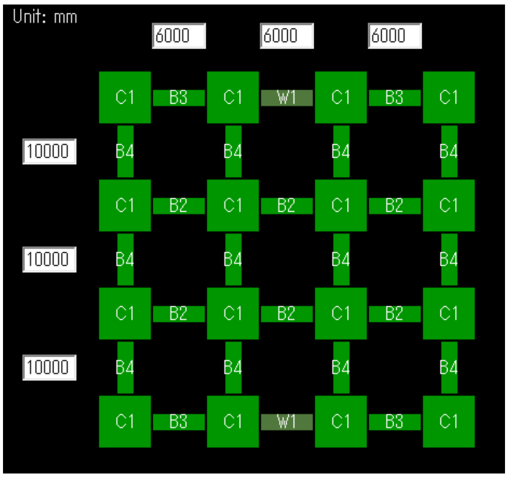
Multi-springs

multi-spring No. 1
x = -0.247E+02 y = -0.247E+02 location of spring

Fc: 1st force point Fy: yield force Dc: 1st deformation Dy: yield deformation

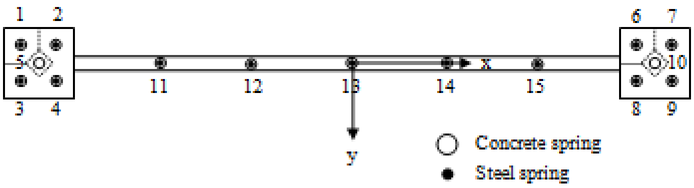
[3] "data_wall.txt"

```
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    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  ic1: 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_Chy       F_Cry       F_Crx       P_Chx       P_Chy       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
     (kNs2/cm)    (kNcms2)    (kNcms2)
     0.0000E+00  0.0000E+00  0.0000E+00

         Tx     Tswx     Trkx       Ty     Tswy     Trky
    1    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,

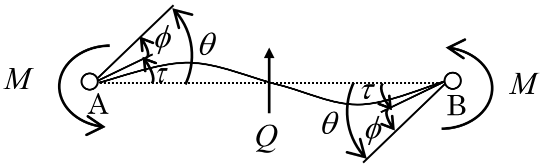
[5] “max_beam.csv”

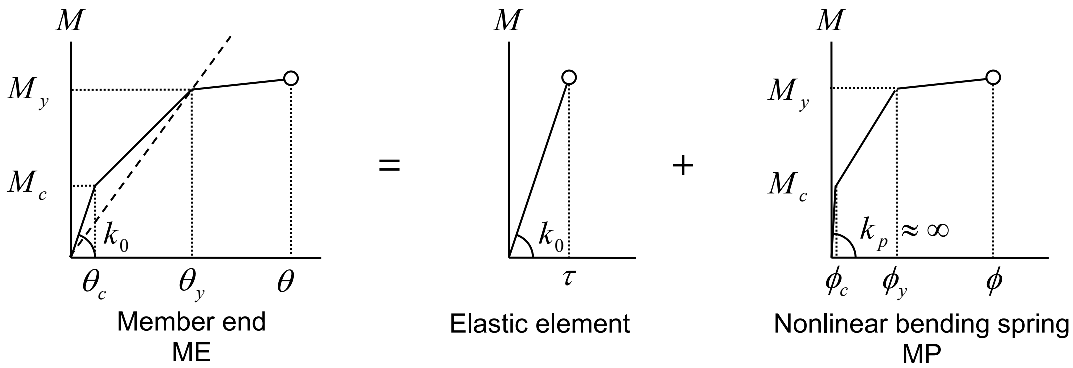
Unit (kN,cm)

```
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,

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

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

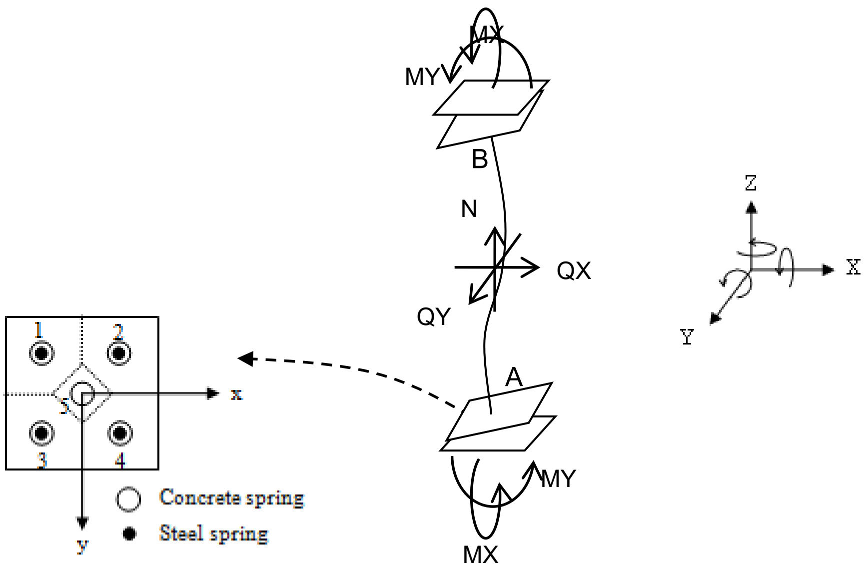
[6] "max_column.csv"

Unit (kN,cm)

EL. NO. =	1	disp	force	Umy				disp	force	Umx	Uh	D. I
MY	1	0. 2585E-02	-0. 1348E+06	0. 00		MX	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

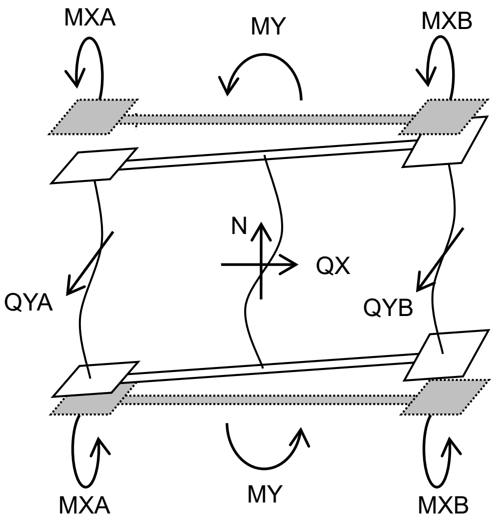
Umx, Umy: Ductility factor (= Dm / Dy)
(Dm: max disp., Dy: yield disp.)
Uh: Cumulative ductility factor (=Eh / QyDy)
(Eh: hysteresis energy, Qy: yield force)
D.I.: Damage Index
(RC: Park and Ang)

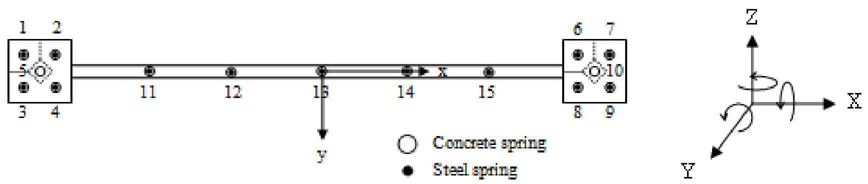


[7] “max_wall.csv”

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)	sdy(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)
- sdy 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^2)
- ay acceleration in Y-direction (cm/sec^2)

<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)	sdy(cm)	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)	sdy(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 n X-direction (cm/sec^2)
- a0y acceleration in Y-direction (cm/sec^2)
- a0z acceleration in Z-direction (cm/sec^2)

<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)
- sdy 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^2)
- ay acceleration in Y-direction (cm/sec^2)

[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
 -0.3026E-02 -0.2950E+03    0.032    -0.3026E-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 -0.5106E+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	

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   -0.2196E-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 -0.6018E+04
-0.1801E-06 -0.4197E+01   -0.000   -0.2973E-01 -0.6060E+04
 0.1265E-06  0.2948E+01   -0.000   -0.2975E-01 -0.6064E+04
 0.6936E-06  0.1617E+02    0.000   -0.2956E-01 -0.6026E+04
    C11D(a)      C11F(a)  C11U(a)       S11D(a)      S11F(a)  S11U(a)
-0.2905E-02 -0.5867E+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	X-direction (Shear)
Dy	Qy	Uy	Y-direction (Shear)
Dv	Fv		Z-direction (Axial)

In case of Damper and Nonstructural Wall

< Shear Force >

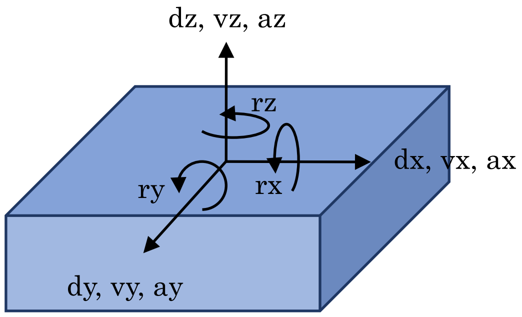
Disp.	Force.	Ductility Factor	
Dx	Qx	Ux	X-direction

[13] “response_floor01.csv …”

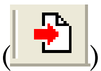
In this file, the responses of the center of gravity of 3D rigid floor are saved.
The 3D rigid floor is used in the following cases:

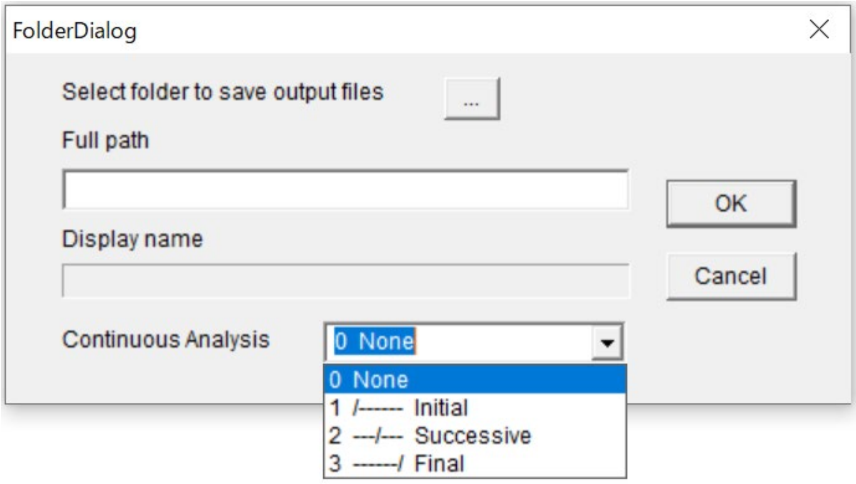
- In case “Ground Spring” (Sway and Rocking) is considered in “Option” > “Member” menu, automatically, the foundation floor is considered 3D rigid.
- In case 3D rigid is selected in “Option” > “Member” menu.

	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

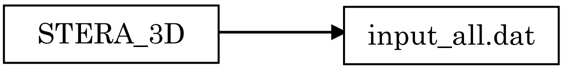
When you push the "Save Data" button () during Dynamic



Please select condition of continuous analysis from the menu:

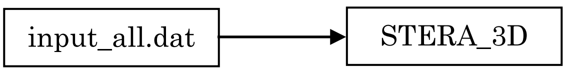
0 None — No continuous analysis (default)

1 /------ Initial — Initial analysis
(save building status after the analysis)



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

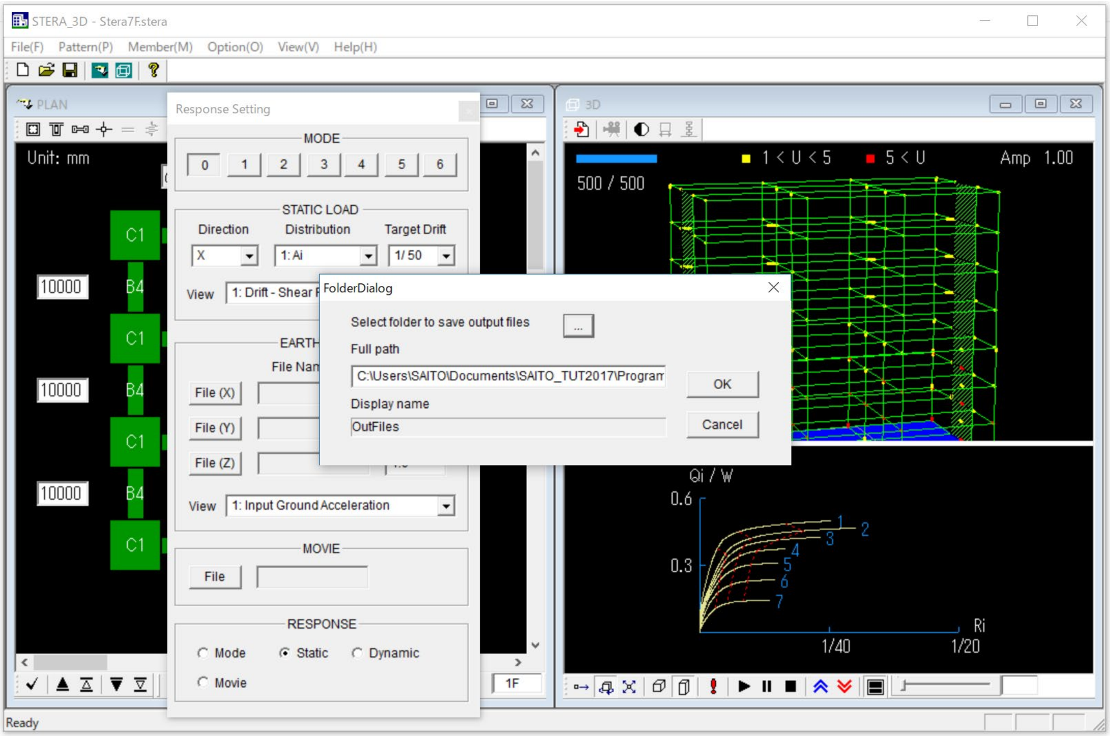
3 ------/ Final — Final analysis
(read building status)

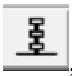


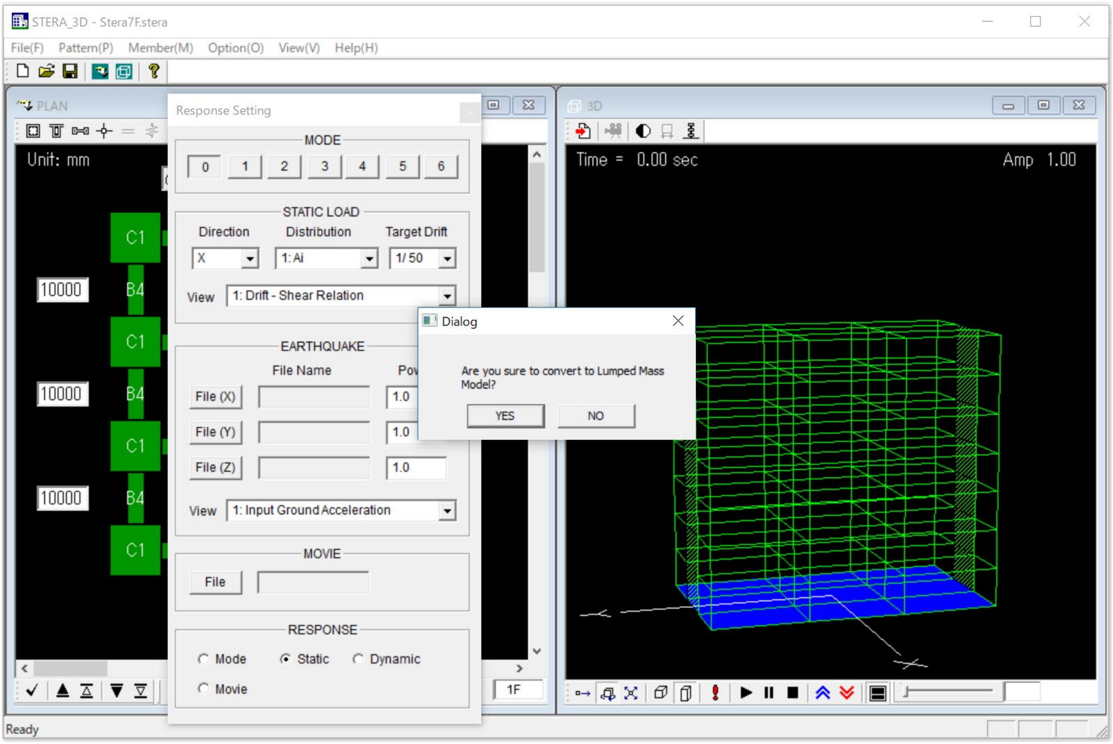
In continuous analysis, it is recommended to change the folder to save output files in each analysis, since all output files will be overwritten.

12 Automatic generation of Lumped Mass Model (LMM)

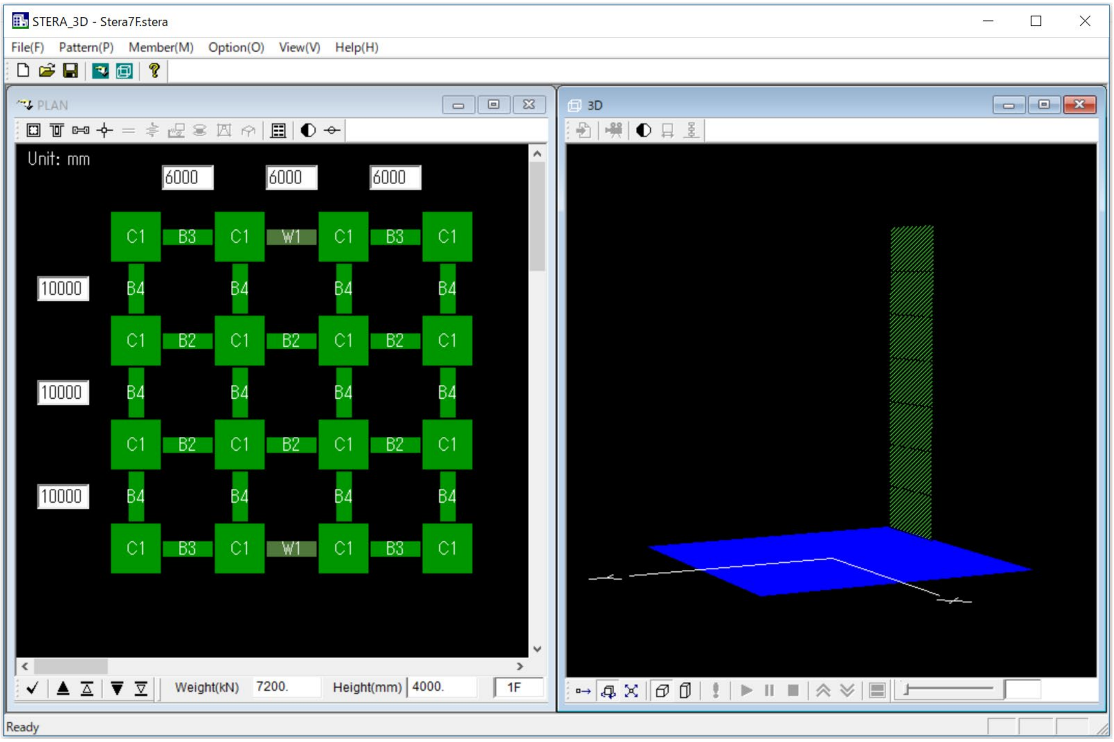
Save results of static push-over analysis.



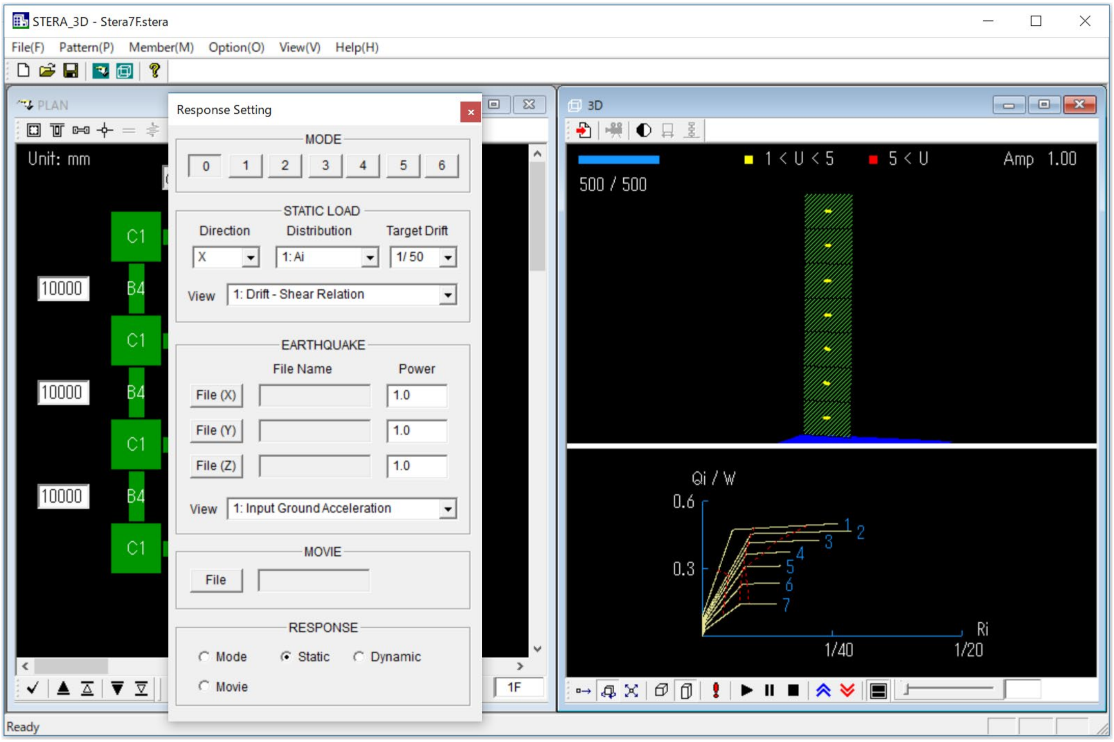
When you click the icon , a message appears to convert to Lumped Mass Model.



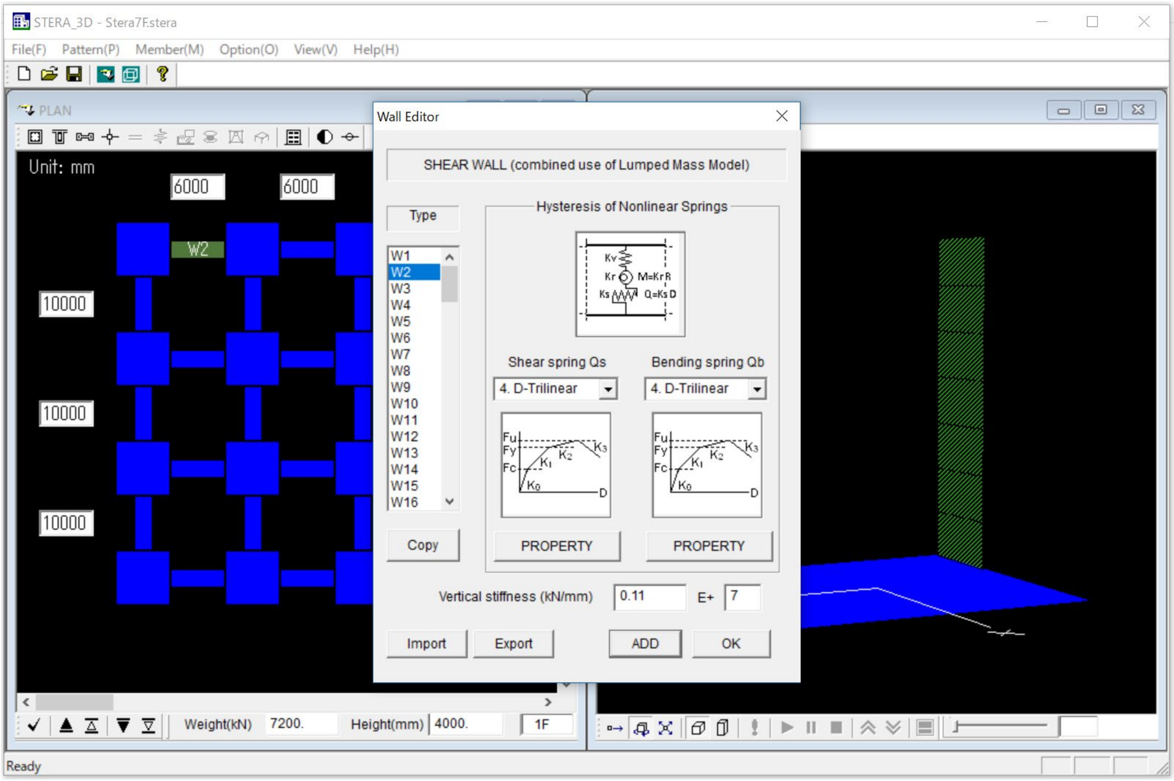
If you answer YES, an equivalent LMM will be automatically created.



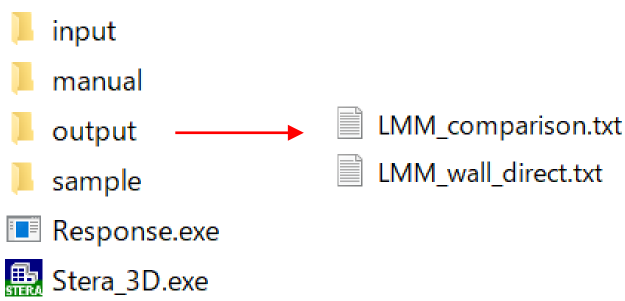
If you conduct static push-over analysis, you will see that the force-deformation relationship of each story is modeled as a tri-linear hysteresis model.



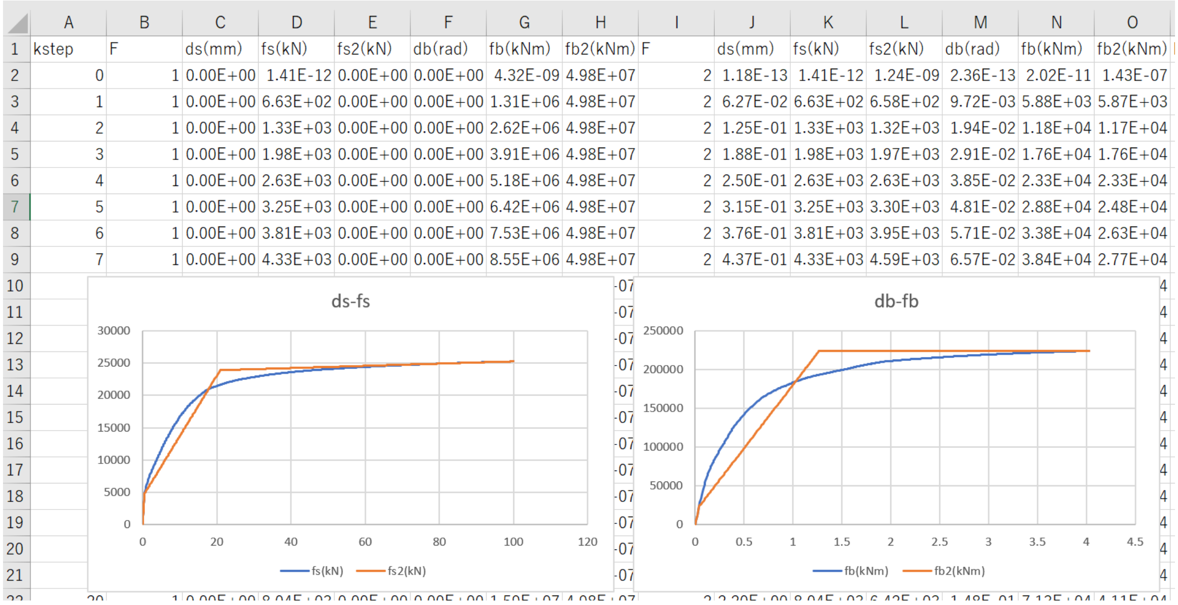
- The element model of each story is “Wall with direct input” with shear and bending springs. The hysteresis of each spring is modeled as non-linear tri-linear model.
- The element type number is “W2” for 1F, “W3” for 2F, …, etc.
- The restrained freedom automatically set as 2467 (X-direction only).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



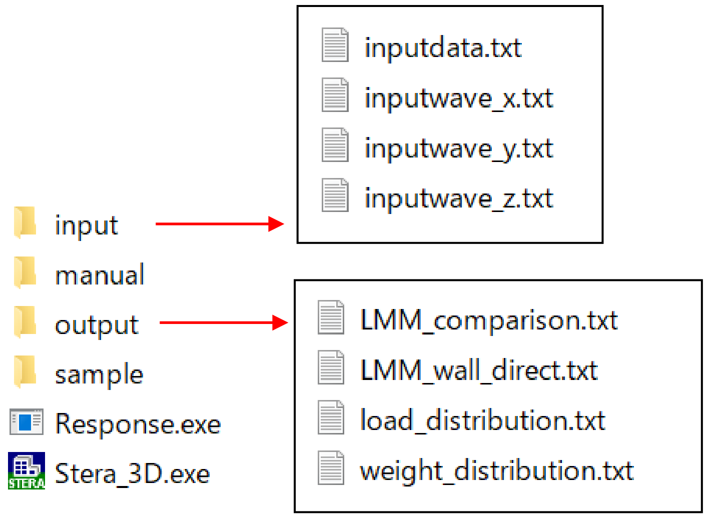
	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	kstep	F	ds(mm)	fs(kN)	fs2(kN)	db(rad)	fb(kNm)	fb2(kNm)	F	ds(mm)	fs(kN)	fs2(kN)	db(rad)	fb(kNm)	fb2(kNm)
2	0	1	0.00E+00	1.41E-12	0.00E+00	0.00E+00	4.32E-09	4.98E+07	2	1.18E-13	1.41E-12	1.24E-09	2.36E-13	2.02E-11	1.43E-07
3	1	1	0.00E+00	6.63E+02	0.00E+00	0.00E+00	1.31E+06	4.98E+07	2	6.27E-02	6.63E+02	6.58E+02	9.72E-03	5.88E+03	5.87E+03
4	2	1	0.00E+00	1.33E+03	0.00E+00	0.00E+00	2.62E+06	4.98E+07	2	1.25E-01	1.33E+03	1.32E+03	1.94E-02	1.18E+04	1.17E+04
5	3	1	0.00E+00	1.98E+03	0.00E+00	0.00E+00	3.91E+06	4.98E+07	2	1.88E-01	1.98E+03	1.97E+03	2.91E-02	1.76E+04	1.76E+04
6	4	1	0.00E+00	2.63E+03	0.00E+00	0.00E+00	5.18E+06	4.98E+07	2	2.50E-01	2.63E+03	2.63E+03	3.85E-02	2.33E+04	2.33E+04
7	5	1	0.00E+00	3.25E+03	0.00E+00	0.00E+00	6.42E+06	4.98E+07	2	3.15E-01	3.25E+03	3.30E+03	4.81E-02	2.88E+04	2.48E+04
8	6	1	0.00E+00	3.81E+03	0.00E+00	0.00E+00	7.53E+06	4.98E+07	2	3.76E-01	3.81E+03	3.95E+03	5.71E-02	3.38E+04	2.63E+04
9	7	1	0.00E+00	4.33E+03	0.00E+00	0.00E+00	8.55E+06	4.98E+07	2	4.37E-01	4.33E+03	4.59E+03	6.57E-02	3.84E+04	2.77E+04

“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

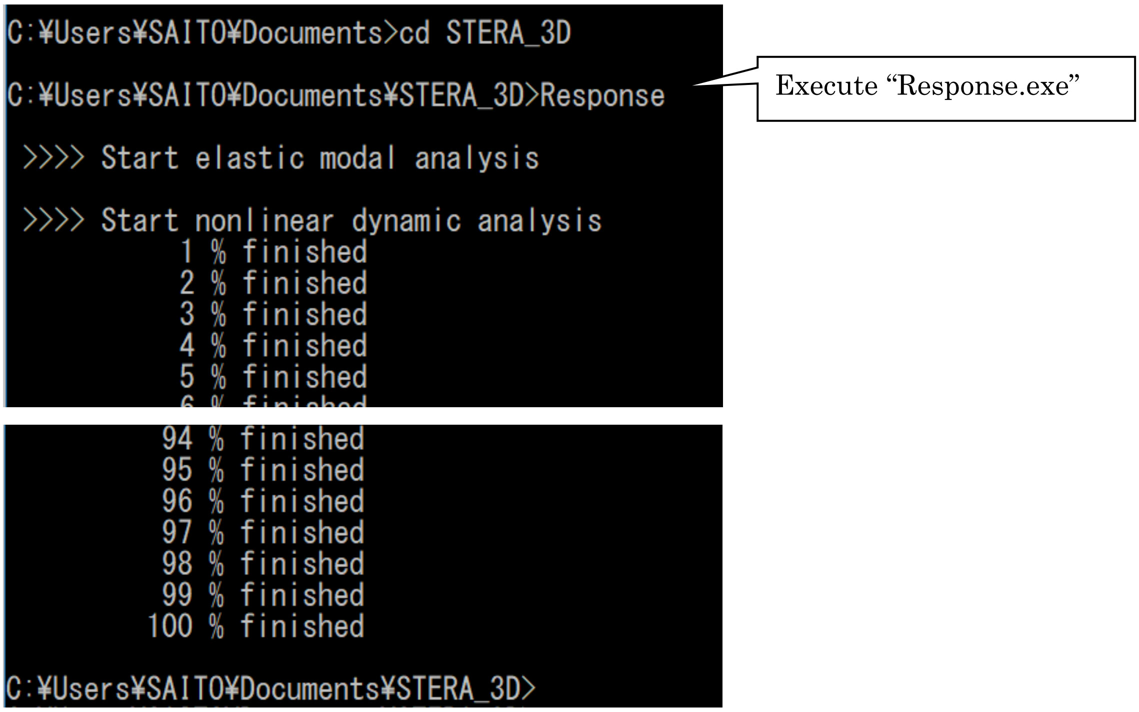
load_distribution.txt	Horizontal load distribution in static analysis (see 7.2)
out_comparison.txt	Comparizon of Q-D between frame and LMM (see 11)
out_parameter.txt	Q-D parameters of LMM (see 11)
weight_distribution.txt	Weight distribution in a floor node (see 7.1)

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,



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 anlaysis will start using new input waves.