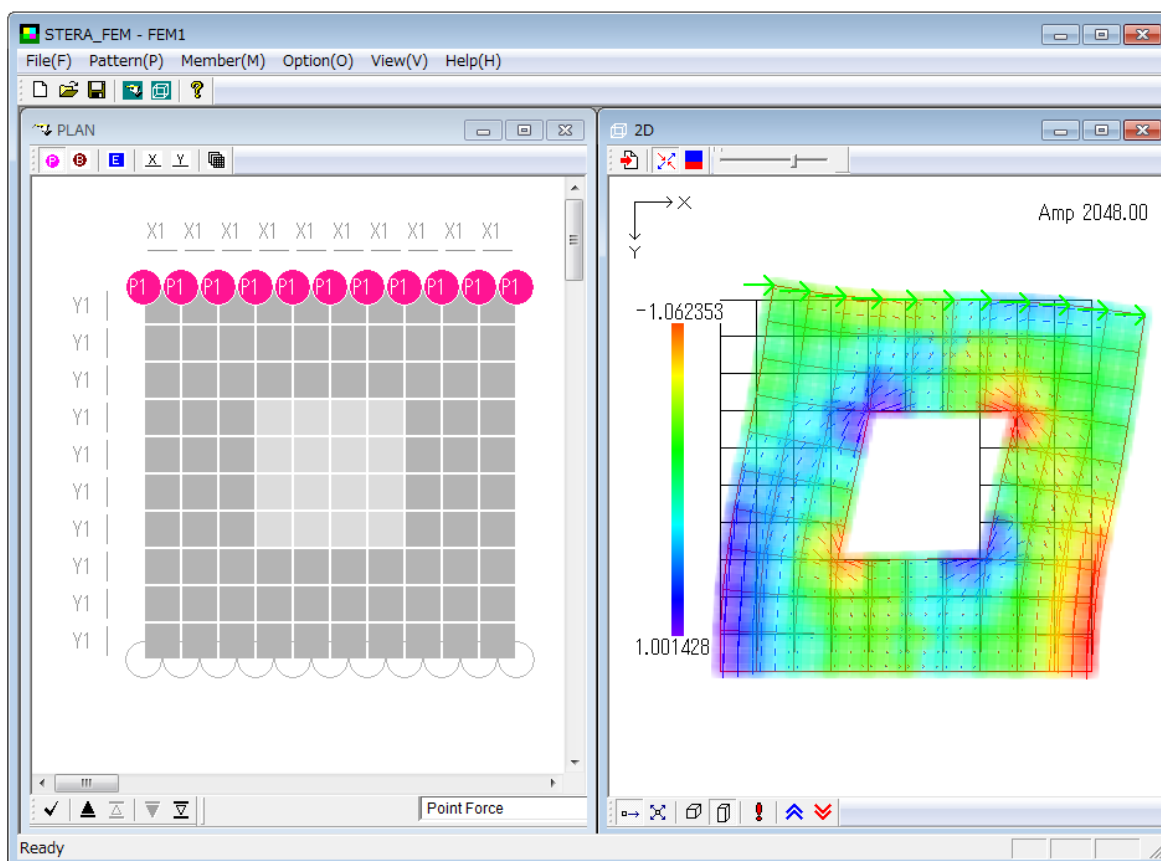# *STERA FEM* ver.2.2

STructural Earthquake Response Analysis FEM

Dr. / Prof. Taiki SAITO

TOYOHASHI UNIVERSITY OF TECHNOLOGY (TUT), JAPAN

Preface

This software is developed for the following analyses of two dimensional FEM structure under plain stress assumption:

- Elastic static analysis under point forces

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.

11 June, 2015

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

Update history

2015.06.11	STERA_FEM Ver.2.0 is uploaded.
2015.12.28	STERA_FEM Ver.2.1 is uploaded
2016.04.06	STERA_FEM Ver.2.2 is uploaded
	Color map is added for stress distribution.

User Manual

Index

1. Basic Assumptions

- Four nodes Isoparametric FEM element is used. Incompatible linear element is adopted to avoid shear rocking effect.

- Three points Gaussian quadrature rule is adopted. Therefore, there are nine Gaussian points in one element.
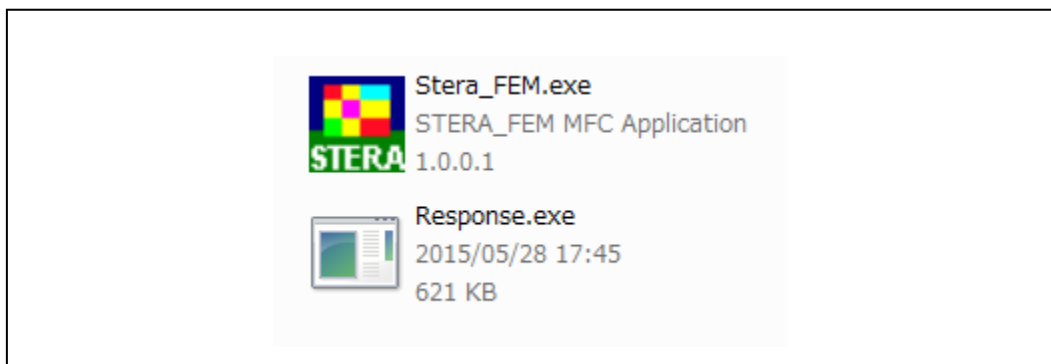
Other assumptions and details are written in "Technical Manual".

2. File Arrangement

Please check if you have the following files and folders in the folder "STERA FEM V*.*":

Stera_FEM.exe	… Main program
Response.exe	… Sub-program for response output

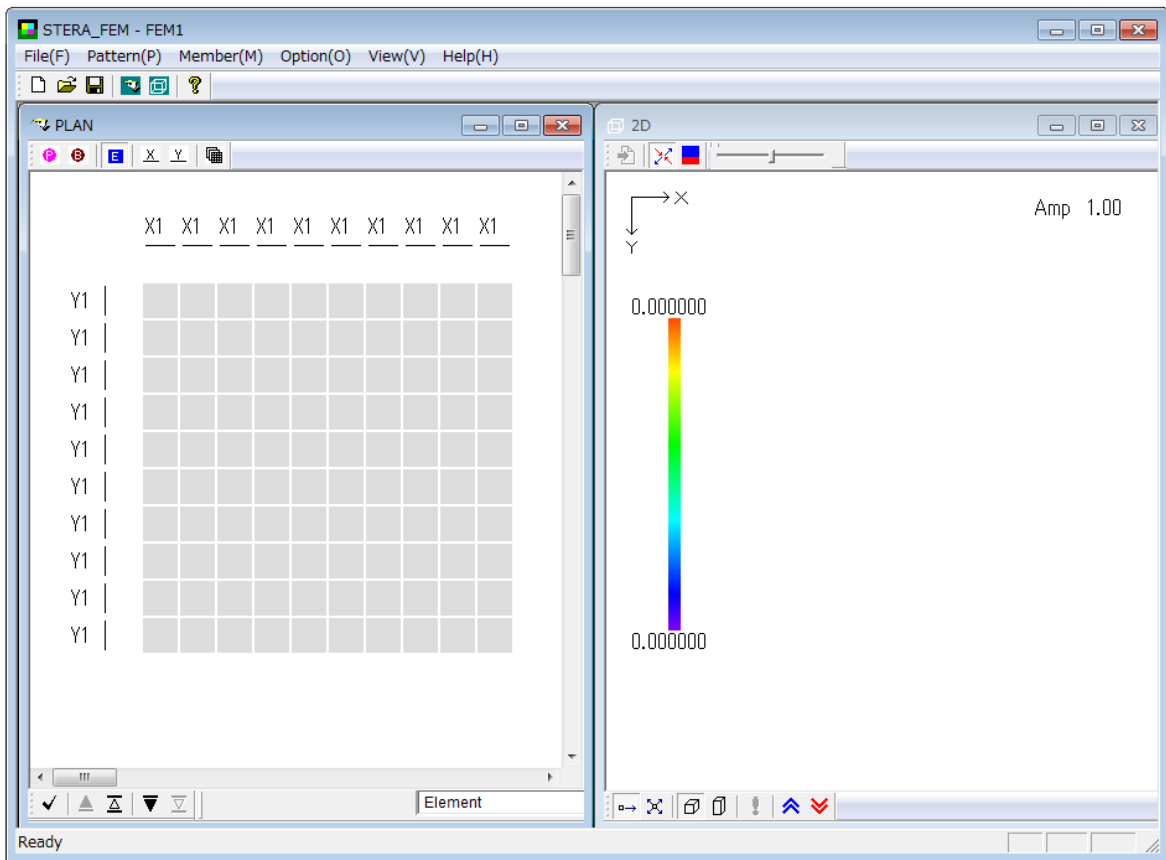Please put two executable files in a same folder.

3. Initial Setting

3-1. Initial view

Please double crick "Stera FEM.exe"

The left view is "PLAN EDIT VIEW" where you input location data, and the right view is "2D VIEW" where you can see the structure shape and its response after the analysis..

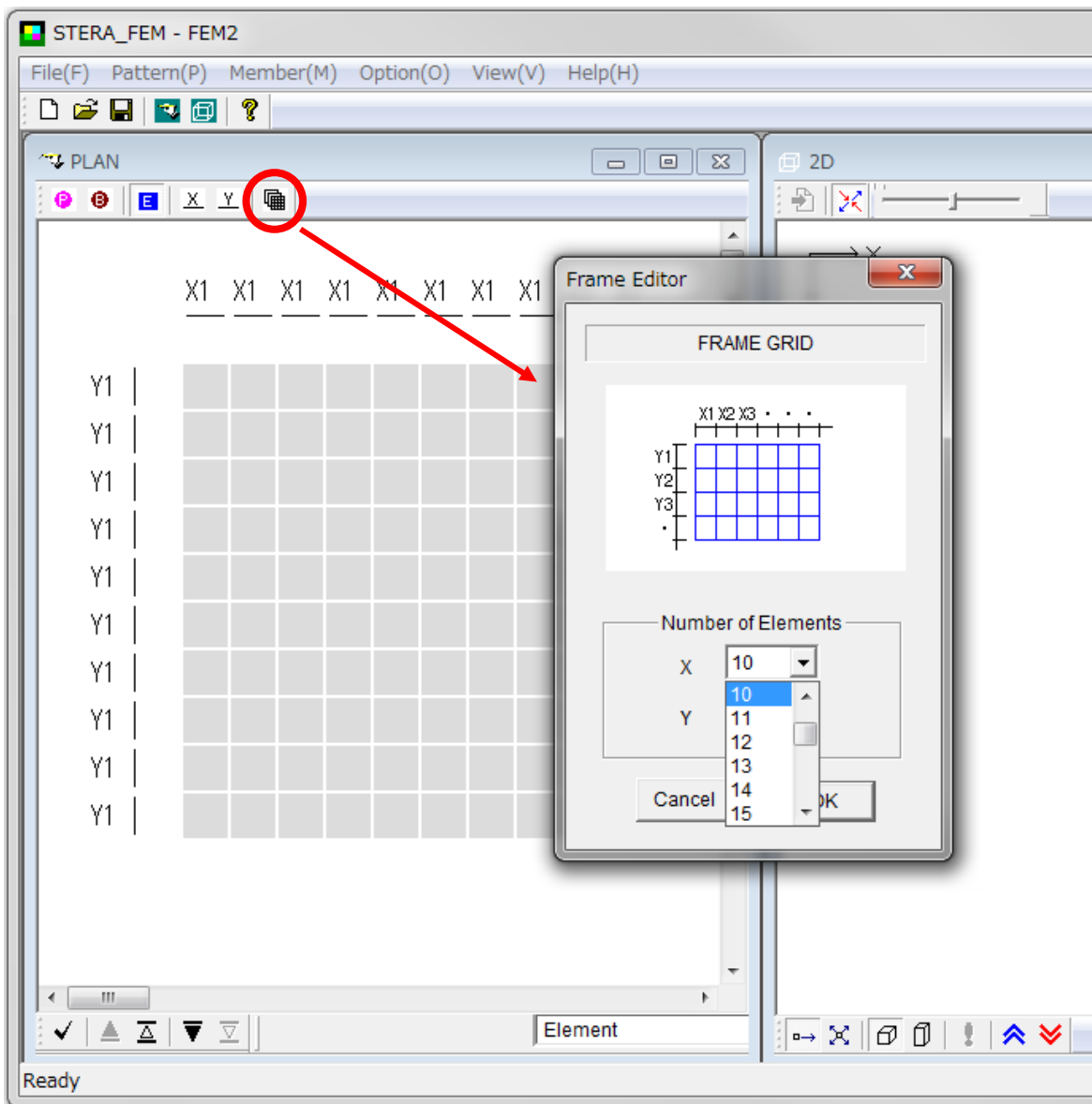To open the building data already saved, [File]→ [Open], and select the file.

PLAN EDIT VIEW   2D VIEW

3-1. Change the number of elements

The maximum numbers of elements in both X and Y directions are 10 in the initial setting.

By clicking the icon ⬚, you can increase the number of elements up to 60 in each direction.
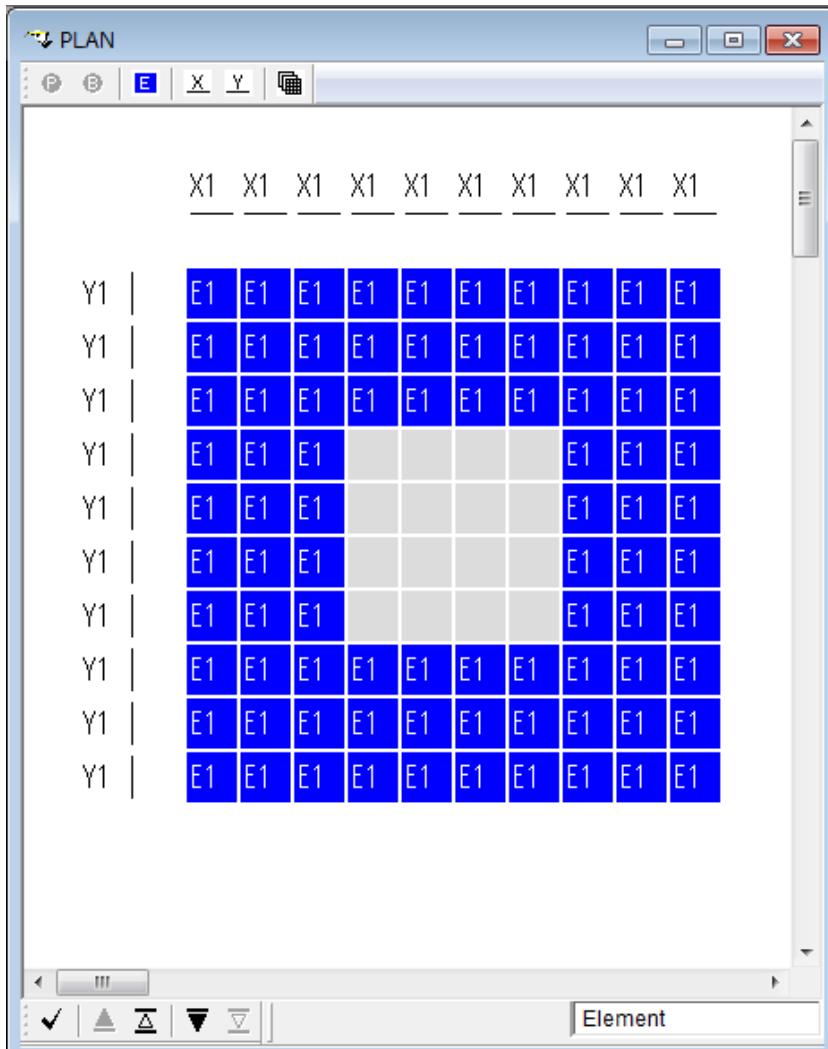

However, there are following conditions:

- The total number of members must be less than 2000.
- The total number of nodes must be less than 2000.

4. Location Setting

4-1. Location of elements

"PLAN EDIT VIEW" starts from the first layer. It is noted as "Element" at the right bottom.
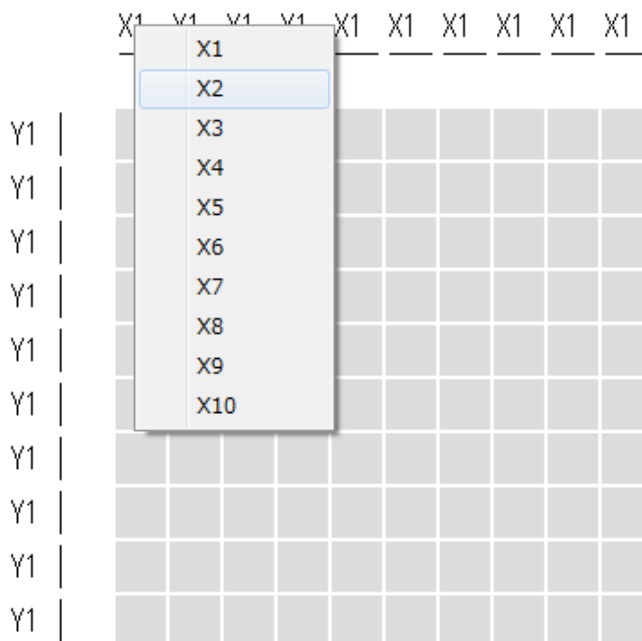
- Please click the place you want to set the element. The element is shown in blue color.
- If you click it again, it will be empty.
- By dragging your mouse in a region, you can set all the elements in the region at once.
- By clicking the right button of your mouse, you can change the number of element type from (E1-E10).

- The number of color of the element by default is set to five.
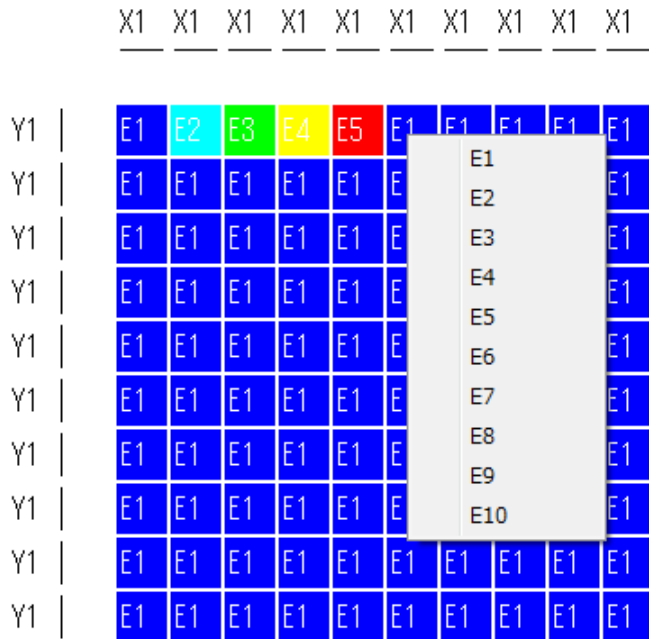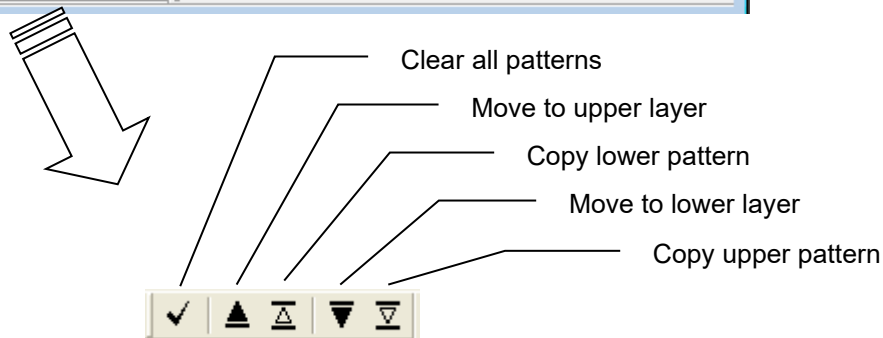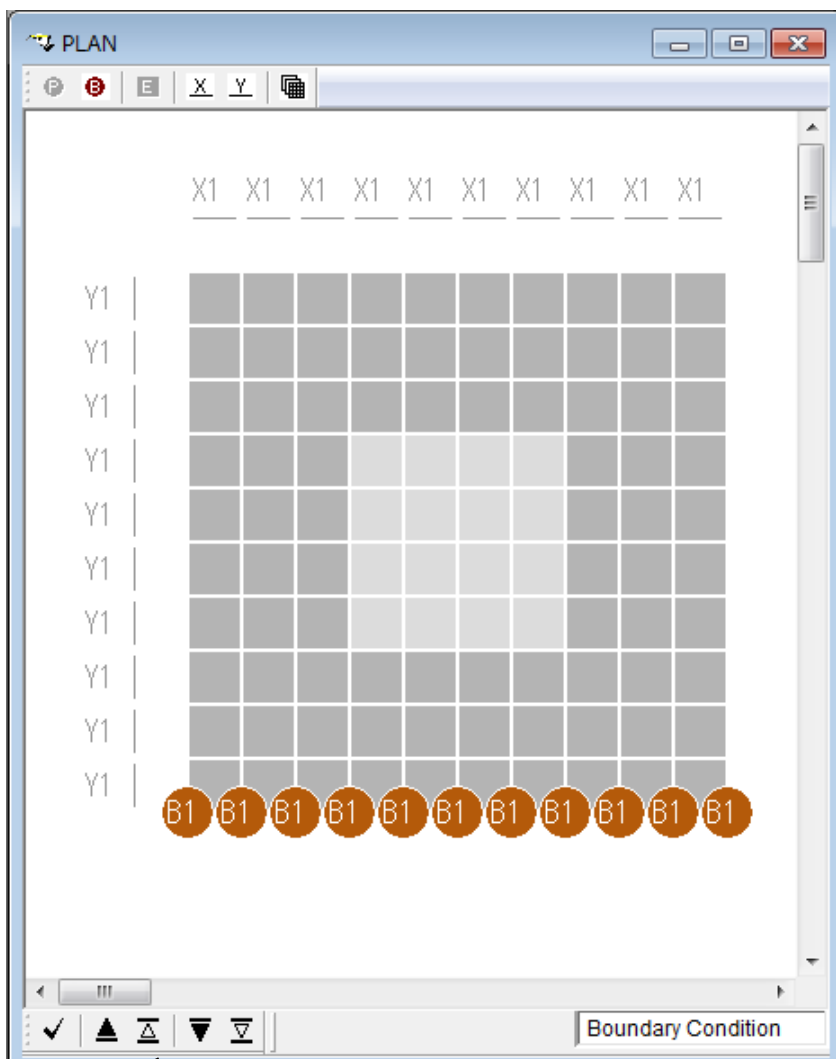

4-2. Size of elements

- The size of element (X1, Y1) can be changed by clicking the right button of your mouse, and select the number of size type from (X1～X10) or (Y1～Y10).

4-3. Location of restraint nodes

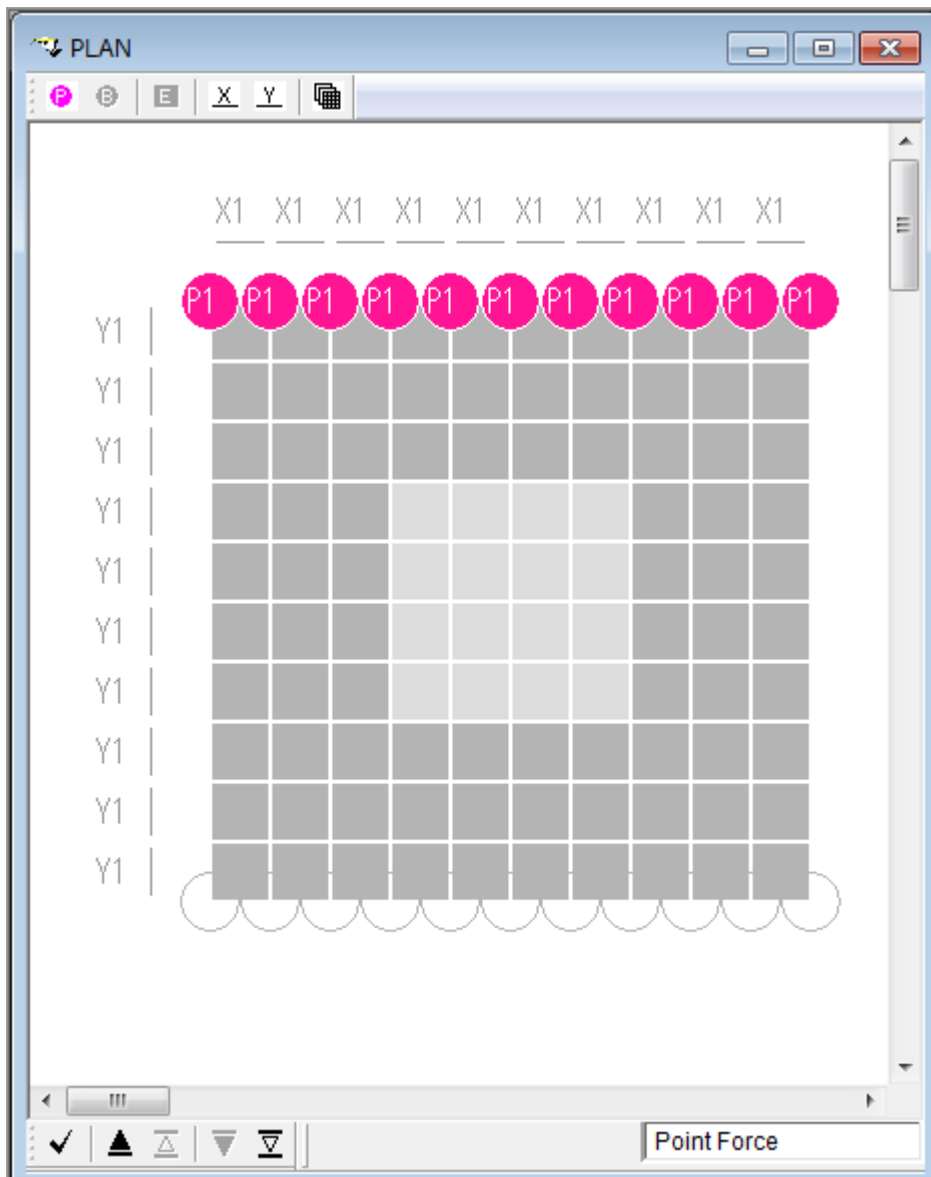By clicking the button ▼ , or by clicking the icon Ⓑ , you can move the layer of Boundary Condition.

- The squares with elements are shown in gray color.
- Please click the corner of square you want to set the restraint condition. The restraint node is shown as a circle in brown color.
- By clicking the right button of your mouse, you can change the number of restraint condition from (B1-B10).

Clear all patterns
Move to upper layer
Copy lower pattern
Move to lower layer
Copy upper pattern

4-4. Location of point forces

By clicking the button $\blacktriangledown$, or by clicking the icon ![P], you can move the layer of Point Force.

- The squares with elements are shown in gray color.
- The corners with boundary conditions are shown in gray color circle.
- Please click the corner of square you want to set the point force. The node is shown as a circle in pink color.
- By clicking the right button of your mouse, you can change the number of force condition from (P1-P10).

5. Input Properties

5-1. Element properties

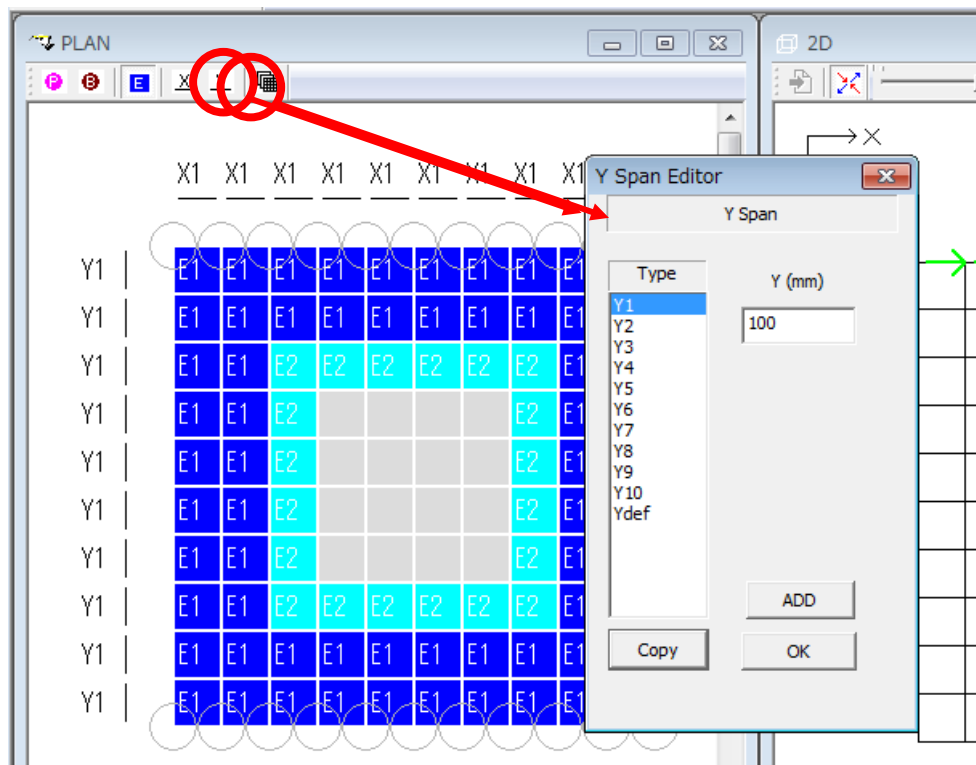By clicking the icon **E** , the edit view of element properties is appeared.
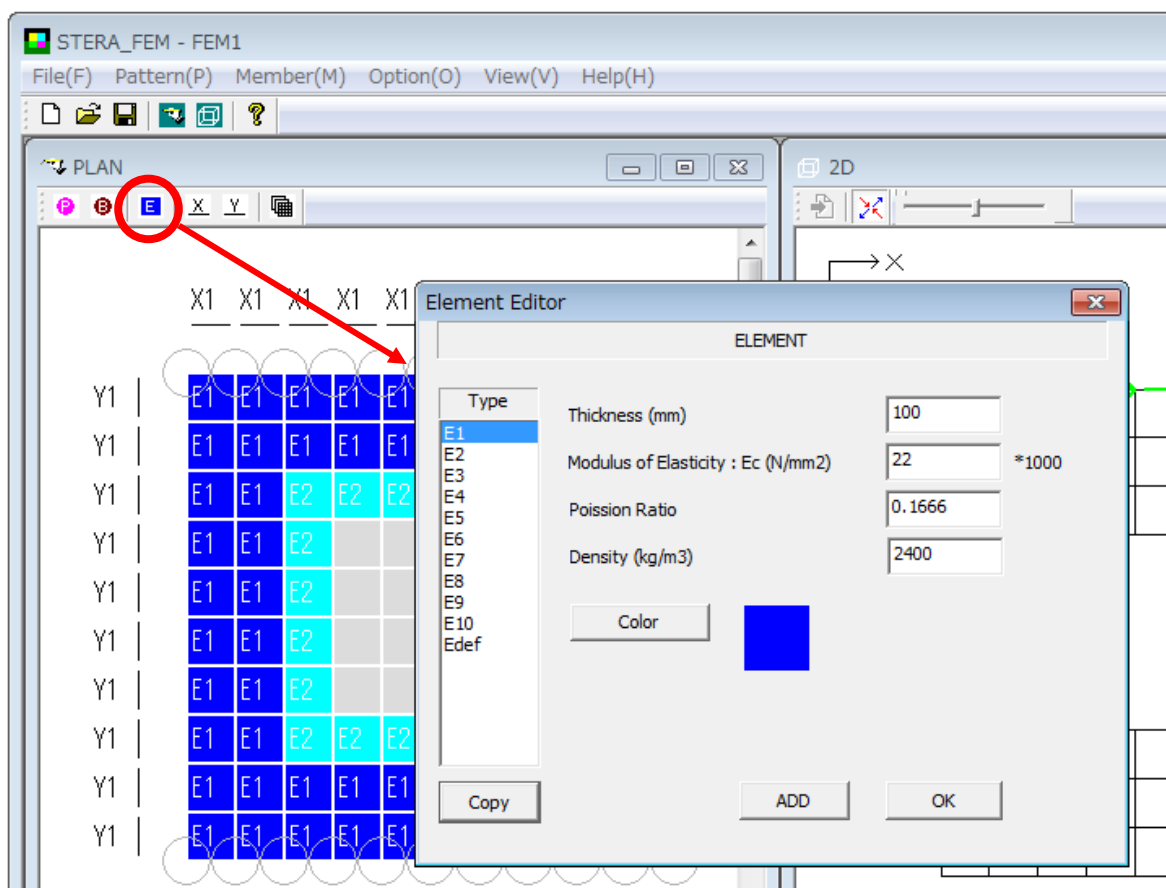
- Concrete properties are used as the default values. Please change those values if necessary.
- You can change the color of the element by clicking [Color] button.
- To move to the next element type, please click [ADD] button.
- You can set default values for all element types by selecting the last element type "Edef".
- Please click [OK] to finish.

By clicking the icon $\overline{\underline{X}}$, you can input the element size in X-direction.
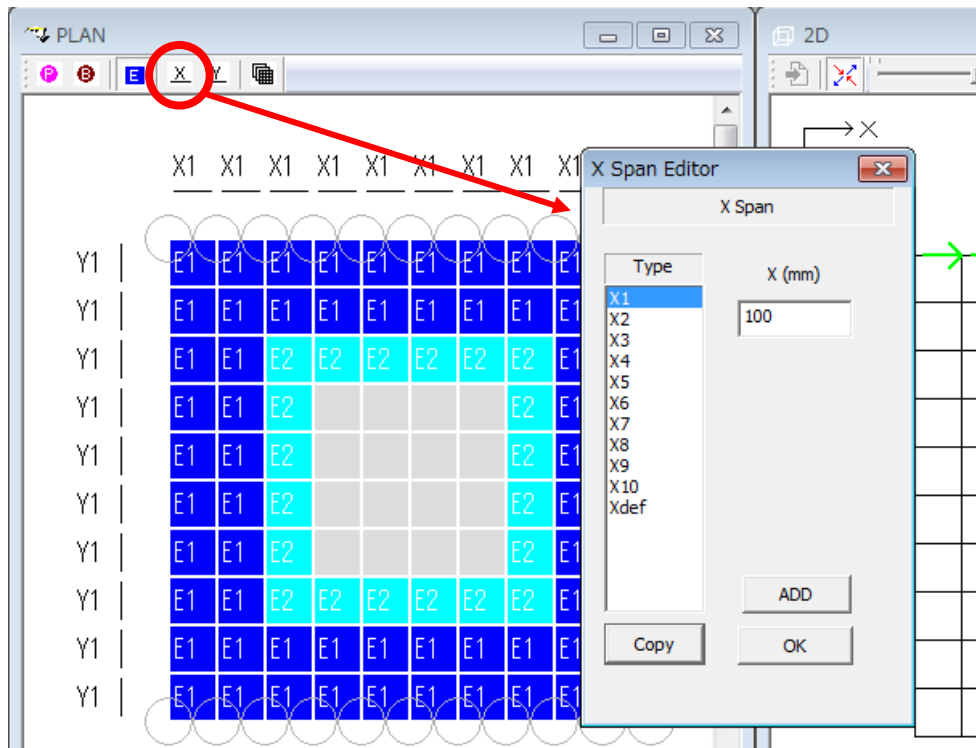
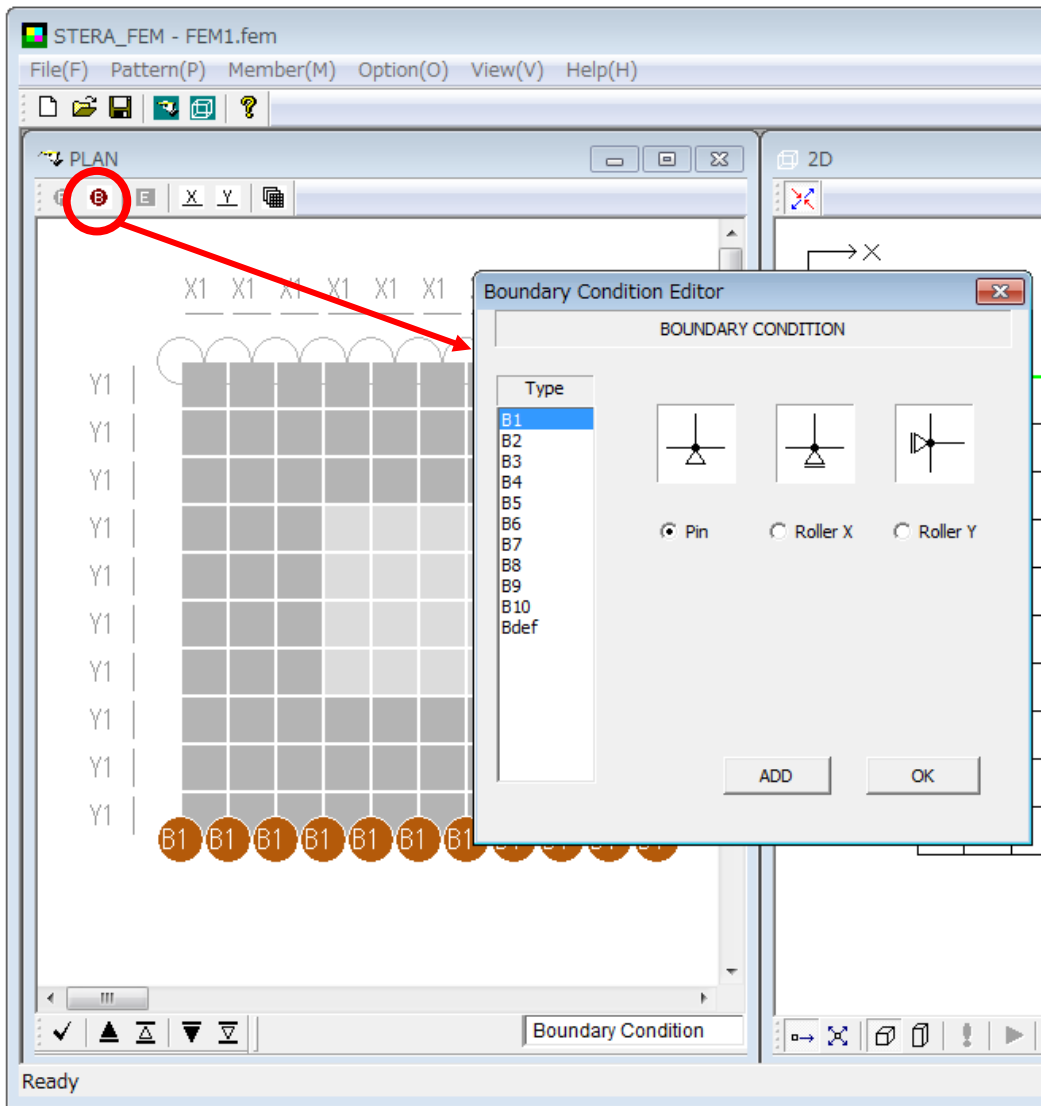By clicking the icon $\underline{Y}$, you can input the element size in Y-direction.

5-2. Restraint condition

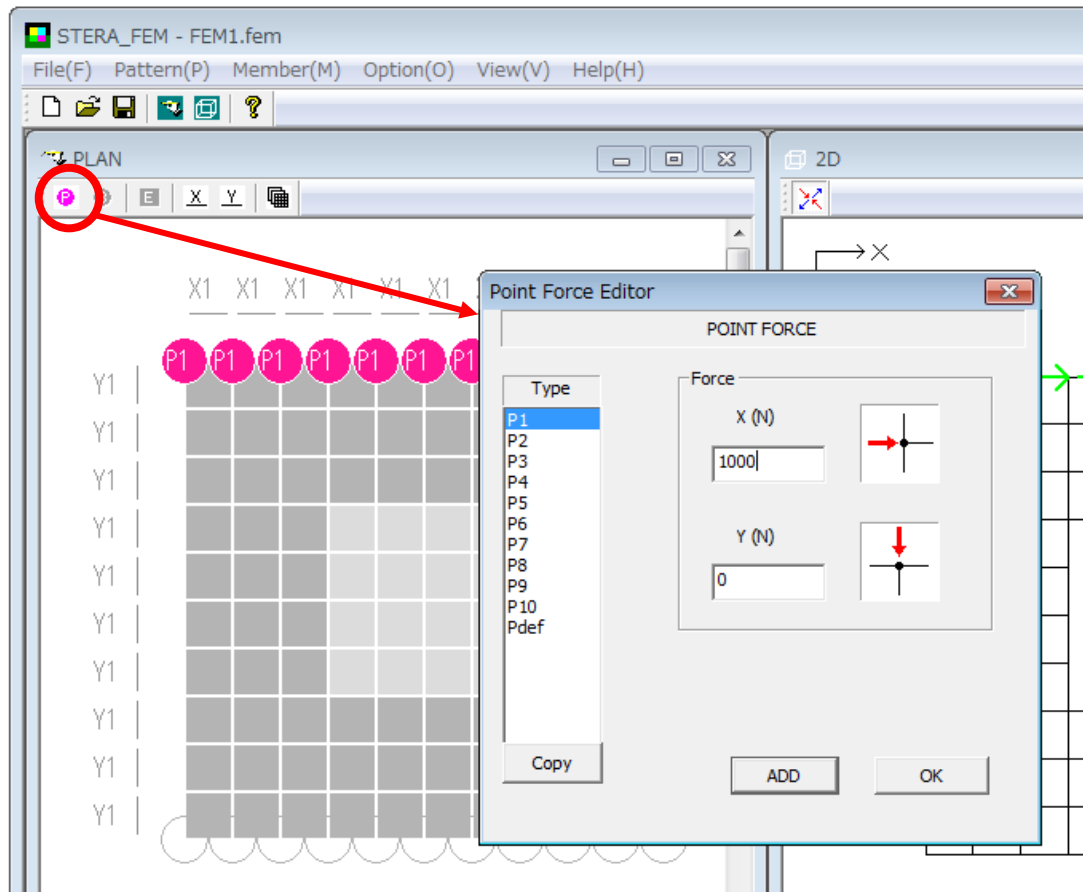By clicking the icon , the edit view of boundary condition is appeared.

You can select the boundary condition from (Pin), (Roller X) and (Roller Y).
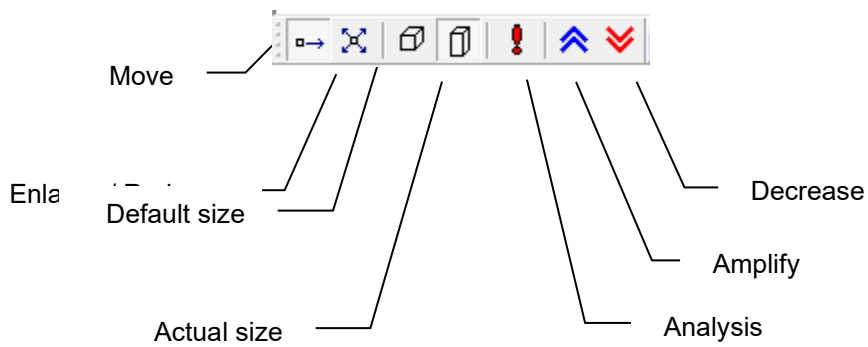
5-3. Force condition

By clicking the icon , the edit view of force condition is appeared.

Please input forces (N) in X and Y directions.

6. 2D View of Structure and Response
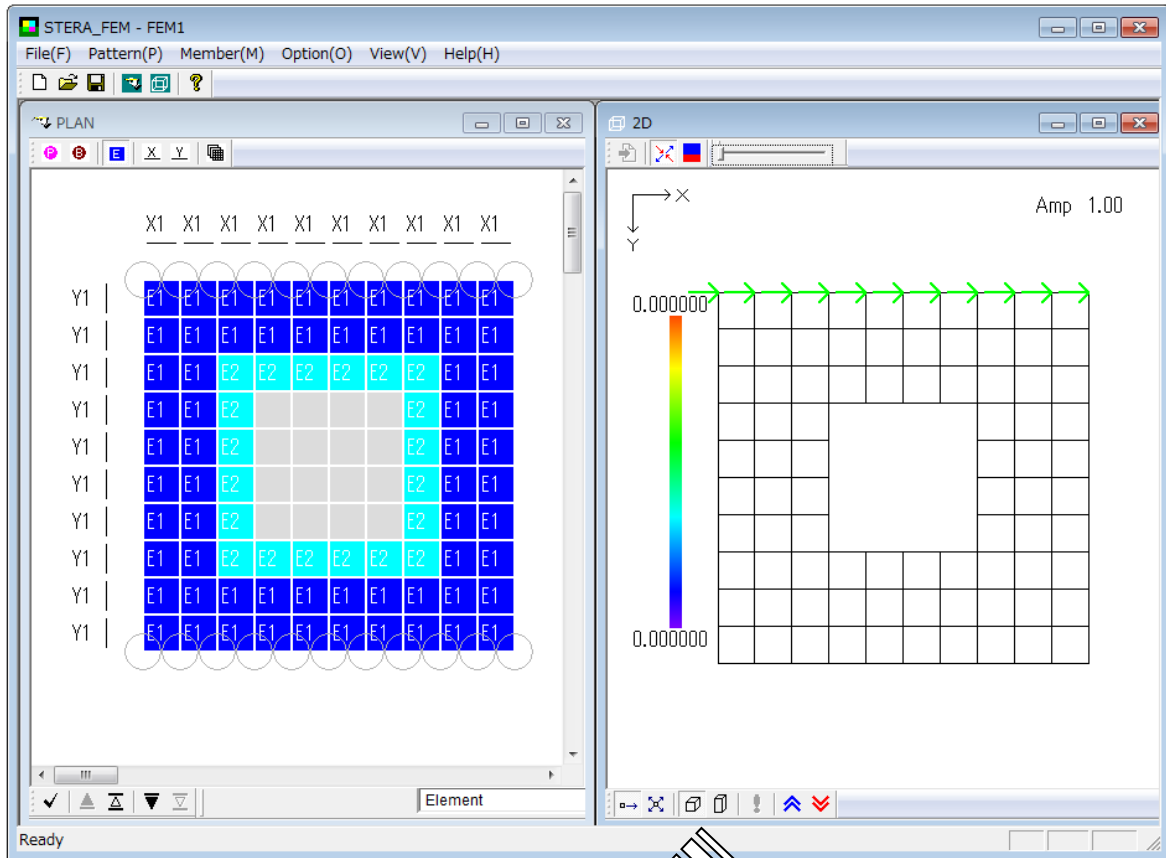
[1] [Default] ( ) set the default size. [Actual] ( ) use the actual size based on input data.

[2] If the [Analyze] ( ) is activated, please click it to start analysis.

[3] If you have the message "Calculation is completed" in the console window, please press any key to close it.
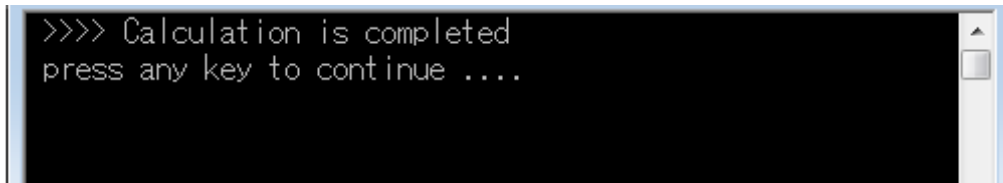
[4] On the 2D view, you can see the **deformation and principal stress under static loading**. ( ) amplifies the response. ( ) reduces the response. The factor of amplification is shown in the right top of the window.
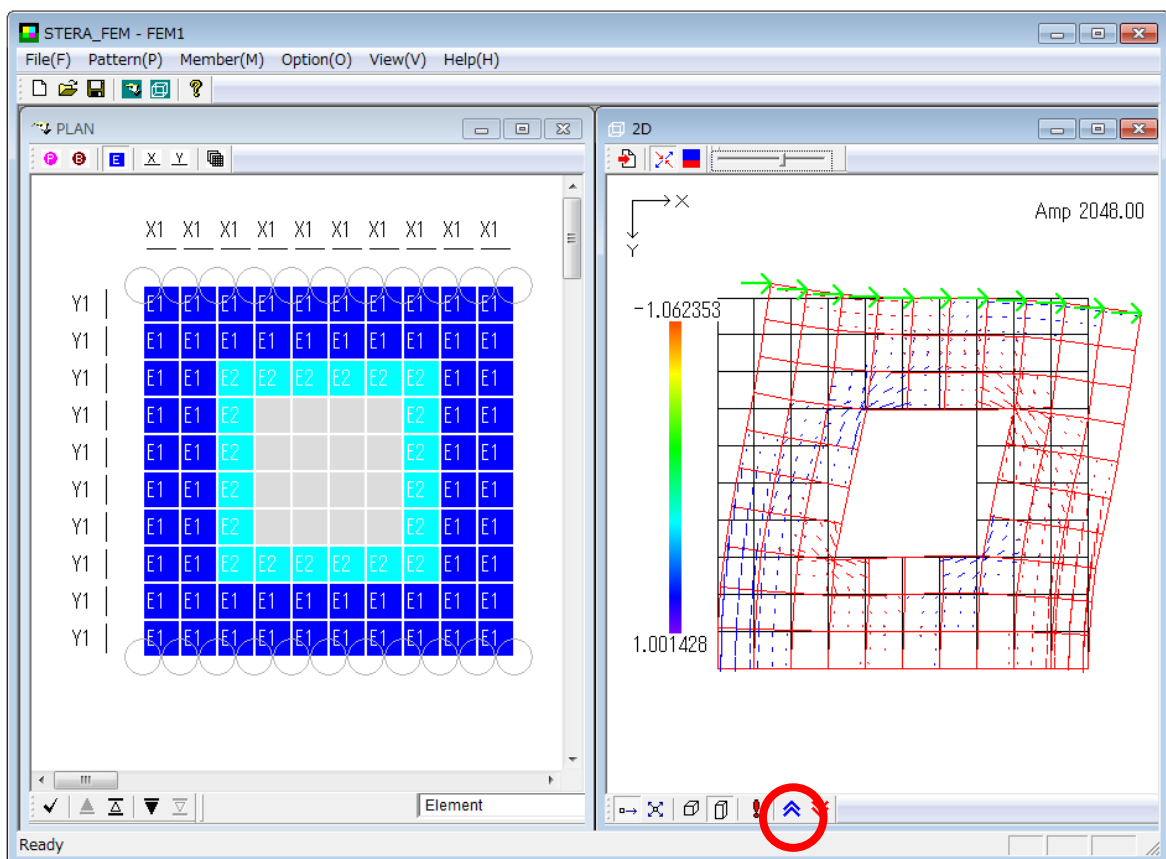
[5] If you move the slider [slider], you can amplify the stress. By clicking [icon], the stress will disappear.
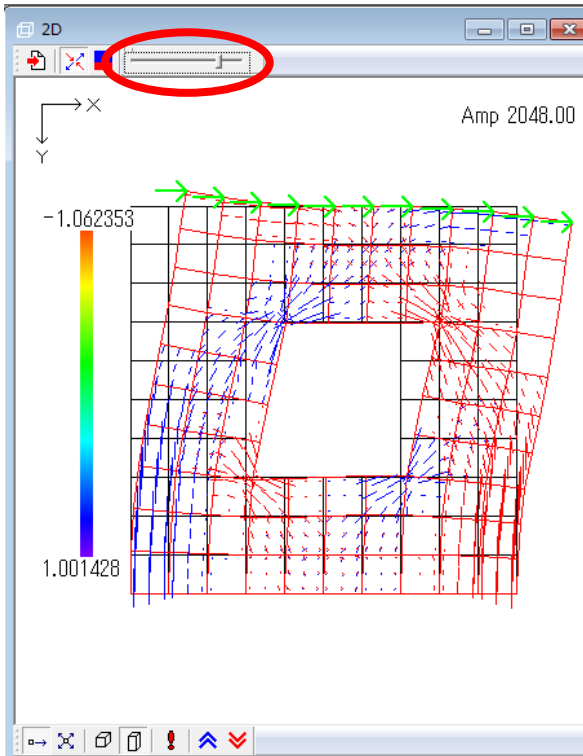
[6] By dragging your mouse with left click, you can magnify the picture. By dragging with right click, you can move the picture.
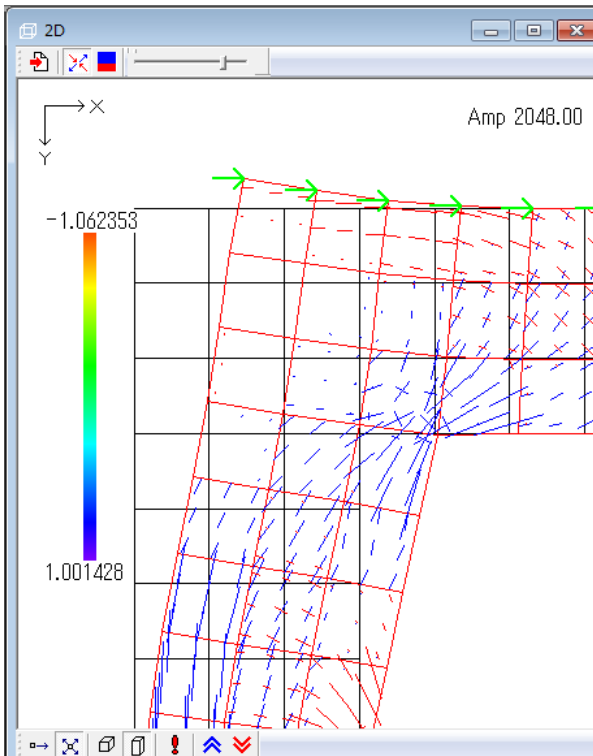
[7] By clicking , you can see the stress distribution by color.
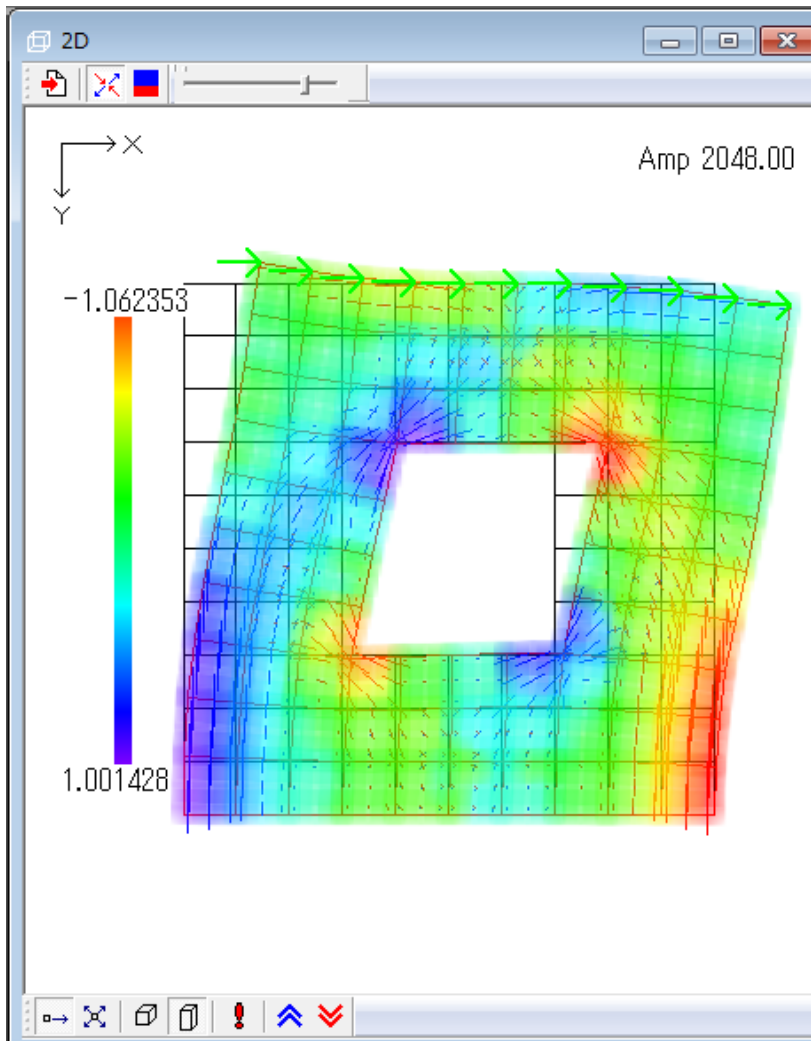
7. Output Files

The following files are automatically created:

input_data.txt
output_data.txt
output_mode.txt
output_response.txt

Here is an example of output files in the following analysis.
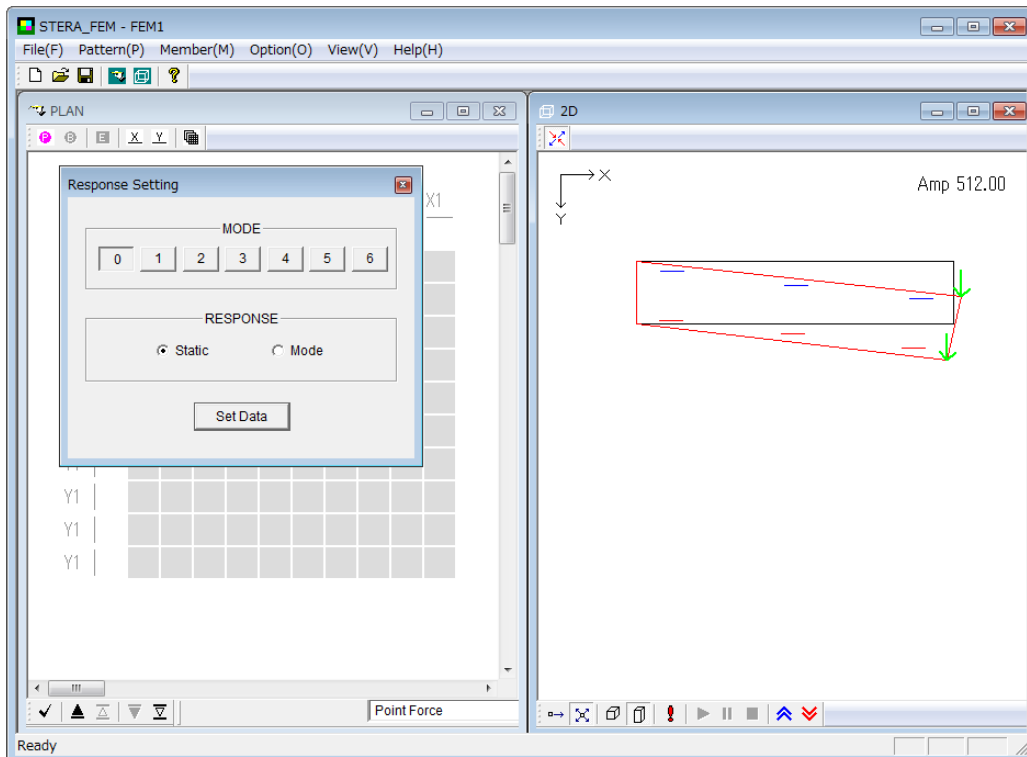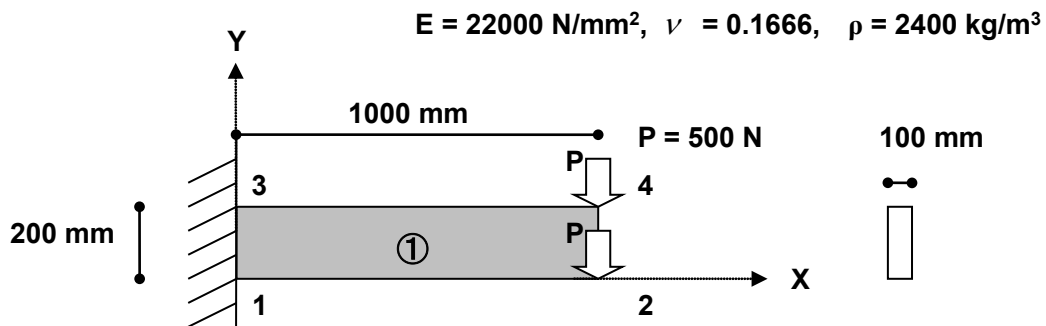
[1] "input_data.txt"

```
FEM MODEL
Type of FEM Element
2                                  (1: linear, 2: incompatible)
Total Numbers of: Nodes, Restrained Nodes and Loaded Nodes
4    2
Nodal Coordinates (n,x,y)[mm]
1    0.0      0.0             Number of node / Coordinate (x,y)
2    1000.0   0.0             ditto
3    0.0      200.0           ditto
4    1000.0   200.0           ditto
Nodal Restrains (n, 0: Free, 1: X-dir, 2: Y-dir, 12: X and Y)
1    12
3    12
Gravity Acceleration [m/sec2]
     -9.80
FEM Plate (Number of Elements, Number of Material Types)
1    1                        Number of elements / Number of types
n    Thick    Ec      v       density
     mm       N/mm2           kg/m3
1    100.0    22000.  0.1666  2400
Element Nodes Definition - Anticlockwise
1    1    2    4    3    1    Number of element, Node number(4), Type number
Load Distribution (node, Fx, Fy)
2    0.0      500.0           node number, load component (x,y) (N)
4    0.0      500.0           ditto
```

[2] "output_data.txt"

```
FEM MODEL

<Structural Parameters>

 ID of analysis mode
   idfem =    2 (1: linear, 2: incompatible)

 control parameters
   nnode =    4 (total Number of NODEs)
   nrest =    2 (total Number of RESTrained nodes)
   nlodn =    2 (total Number of LOaDed Nodes)

 nodal coordinate
 --n--    ---x---    ---y---
    1       0.00       0.00
    2    1000.00       0.00
    3       0.00     200.00
    4    1000.00     200.00

 nodal restrains
 --n--    --dir.--
    1         12
    3         12

 gravity acceleration (m/sec2)
           -9.80000

<Fem Plate>

 control parameters
   nelfp =    1
   nndfp =    4
   nmtfp =    1
   nstfp =    3
   nvafp =    8
   ngaus =    3

 material property
   mt    Thickness(mm)         E(N/mm2)   Poisson ratio  Density(kg/m3)
    1        100.0000      22000.0000          0.1666       2400.0000

 node numbers and element type
     n    1    2    3    4    t
     1    1    2    4    3    1

<LOAD>

 load distribution
 --n--    ---x---    ---y---
    2      0.000    500.000    (*3)
    4      0.000    500.000
```

Repeat of the
input data

==

```
* Freedom Vector *

including restraint conitions
      1    1    0    0    1    1    0    0

numbering freedom vector
      0    0    1    2    0    0    3    4

number of freedom =    4
```

Freedom vector

```
* Location Matrix *

Fem Plate No.  1   Number of variables = 8
      0    0    1    2    3    4    0    0
```

Location matrix

```
* Skyline Parameters *

number of diagonal element (+1)    :    5
number of stiffness matrix element :   10
band width                         :    3
--- mhigh ---
      0    1    2    3
--- mdiag ---
      1    2    4    7   11
```

Parameters for

Skyline method

```
* Gauss Point *

(number of Gauss points = 3 )
            position      weighting factor
   1  -0.774596691131592  0.555555582046509
   2   0.000000000000000  0.888888895511627
   3   0.774596691131592  0.555555582046509
```

Weighting factors of

Gauss points

```
* Static Load *

freedom      load
     1    0.000E+00
     2    0.500E+03
     3    0.000E+00
     4    0.500E+03
```

Load vector

[3]"output_response.txt"

```
Number of element
      1
Node number of each element
   EL   na   nb   nc   nd
    1    1    2    4    3
Number of node
      4
Position of all nodes
Initial coordinate
      1    0.00000E+00      0.00000E+00    ⎫
      2    0.10000E+04      0.00000E+00    ⎬   Initial coordinate of node
      3    0.00000E+00      0.20000E+03    ⎪
      4    0.10000E+04      0.20000E+03    ⎭
Static displacement
      1    0.00000E+00      0.00000E+00    ⎫
      2    0.42109E-01      0.21710E+00    ⎬   Static displacement at node
      3    0.00000E+00      0.00000E+00    ⎪
      4   -0.42109E-01      0.21710E+00    ⎭
Principal stress at gauss point
EL   1           sx           sy         theta    degree  ⎫
      1    0.52767E-02   -0.72286E+00   0.14856E+01    85.12  ⎪
      2    0.61760E-01   -0.61760E-01   0.78540E+00    45.00  ⎪
      3    0.72286E+00   -0.52767E-02   0.85231E-01     4.88  ⎪
      4    0.52767E-02   -0.72286E+00   0.14856E+01    85.12  ⎬  Principal stresses
      5    0.61760E-01   -0.61760E-01   0.78540E+00    45.00  ⎪  and angle at Gauss points
      6    0.72286E+00   -0.52767E-02   0.85231E-01     4.88  ⎪
      7    0.52767E-02   -0.72286E+00   0.14856E+01    85.12  ⎪
      8    0.61760E-01   -0.61760E-01   0.78540E+00    45.00  ⎪
      9    0.72286E+00   -0.52767E-02   0.85231E-01     4.88  ⎭
Load distribution
      n           x            y       ⎫
      1    0.00000E+00      0.00000E+00 ⎪
      2    0.00000E+00      0.61760E+03 ⎬   Nodal forces including self-weight
      3    0.00000E+00      0.00000E+00 ⎪
      4    0.00000E+00      0.61760E+03 ⎭
```