```
+-----------------------------------------------------------------------+
|                                                                       |
|   User's Manual                                                       |
|                                                                       |
|       PLATE : Plate Bending Analysis by Finite Element Method         |
|                                                                       |
|                Programmed by Toshimi Taki                             |
|                            17 Yakushi Ushirohibo                      |
|                            Konan, Aichi                               |
|                            483 Japan                                  |
|                                                                       |
|                                                                       |
|                                                                       |
+-----------------------------------------------------------------------+
```

CONTENTS

1. What is 'PLATE'?

'PLATE' is a program of plate bending analysis by Finite Element Method (FEM in short). The program calculates deformation of a flat plate with arbitrary shape subjected to out-of-plane loads (lateral pressure and lateral concentrated forces). It runs on IBM-PC compatible machines.

The program was developed by Toshimi Taki in order to analyze deformation of astronomical telescope mirrors installed in various cell designs. A prototype program was used to write the article titled 'Mirror Support: 3 or 9 Points?' in Sky & Telescope, September 1994, p.84.

The prototype is applicable only to 'thin plate *' and the error becomes larger when it is applied to 'thick plate **'. This new version has been developed in order to analyze 'thick plate'. It is based on 'theory of plate bending with transverse shear deformation'.

Notes: * 'Thin plate' means a plate which support span (l) is much larger than the thickness (t). t/l<<1.

** 'Thick plate' means a plate which thickness is comparable to the support span. t/l≒1.

2. Procedure of FEM analysis

2.1 Preparation -- Defining problem

Before you make an FEM model, study your subject and define a problem which you can solve by 'PLATE'. Some simplifications may be required.

(1) Define dimensions
(2) Define material(s)
(3) Define external load(s)
(4) Define support condition(s)

2.2 Making FEM model

(1) Define region to be analyzed

You can reduce a region of the model by utilizing symmetry of the problem. For example 1 to 4, only one quarter of the plate is modeled in FEM analysis.

(2) Divide the model into small triangular elements.

It is difficult to say how many elements are needed. Follow your 'engineering sense'.

(3) Set identification numbers to elements and nodes

The element numbers and the node numbers begin from '1' to total number of elements (or nodes) in ascendant order without skip. Total number of elements should not exceed 300 and total number of nodes should not exceed 170 in 'PLATE'.

(4) Define location (x,y coordinates) of nodes

(5) Define properties of elements

Three nodes which define a element, thickness, material should be defined. The thickness is considered to be uniform in the element.

(6) Define boundary conditions of model

Study the support condition of the model at particular nodes. Constraints of movement along z-axis, rotation around x-axis and rotation around y-axis are used in 'PLATE'. For expressions of constraint, see p.8.

(7) Define distributed pressure on elements

In 'PLATE', pressure is uniform over a element.

(8) Define concentrated forces at nodes

2.3 Making input data file

See ~~xxx~~ p.6 for data format and its explanation.

2.4 Running 'PLATE'

The name of the input data file should be 'INPUT.DAT'. The input data file should be in the same directory of 'PLATE'. If the name is not 'INPUT.DAT', rename the input data file before you run the program. When you run the program, there should not be a file named 'OUTPUT.DAT' in the directory. It is because the program will make 'OUTPUT.DAT' as a result. If there is a file named 'OUTPUT.DAT', rename it before you run the program.

Type 'PLATE' in the directory in which 'PLATE.EXE' and 'INPUT.DAT' exist. After a while, 'PLATE.EXE' makes a file named 'OUTPUT.DAT' as a result of analysis.

2.5 Output data

(1) Explanation of output data

See p.9 for the explanation of the output data.

(2) Check it

You should remind yourself of a famous saying, 'Garbage in, garbage out'. Before you accept the result of FEM analysis, check out your input data and output data thoroughly. Check that the deformed shape is reasonable or not.

3. Limitations of 'PLATE'

'PLATE' has following limitations.

(1)	Maximum number of nodes	: 170
(2)	Maximum number of elements	: 300
(3)	Type of element	: Triangular element only.
(4)	Maximum number of materials	: 25
(5)	Type of materials	: Elastically isotropic materials only
(6)	Maximum number of constraint data	: 60
(7)	Type of constraint	: Oblique constraint is not considered.

4. Files included in package

The following 15 files are included in 'PLATE' Version 1.0.

File name	Description
PLATE.EXE	FEM program
INPUT1.DAT	Example of input data No.1 Square panel, simply supported edge, uniform pressure
INPUT2.DAT	Example of input data No.2 Square panel, simply supported edge, center force
INPUT3.DAT	Example of input data No.3 Square panel, clamped edge, uniform pressure

INPUT4.DAT	Example of input data No.4 Square panel, clamped edge, center force
INPUT5.DAT	Example of input data No.5 Circular panel, simply supported edge, uniform pressure
INPUT6.DAT	Example of input data No.6 15-point astatic system, 680mm dia., F/4.4, 60mm thick.
INPUT7.DAT	Example of input data No.7 18-point floatation system, 680mm dia., F/4.4, 60mm thick.
OUTPUT1.DAT	Example of output data No.1 Square panel, simply supported edge, uniform pressure
OUTPUT2.DAT	Example of output data No.2 Square panel, simply supported edge, center force
OUTPUT3.DAT	Example of output data No.3 Square panel, clamped edge, uniform pressure
OUTPUT4.DAT	Example of output data No.4 Square panel, clamped edge, center force
OUTPUT5.DAT	Example of output data No.5 Circular panel, simply supported edge, uniform pressure
OUTPUT6.DAT	Example of output data No.6 15-point astatic system, 680mm dia., F/4.4, 60mm thick.
OUTPUT7.DAT	Example of output data No.7 18-point floatation system, 680mm dia., F/4.4, 60mm thick.

==

5. References

[1] O. C. Zienkiewicz, The Finite Element Method in Engineering Science, McGraw-Hill Publishing Company Limited, 1971.

[2] Y. Ando, K. Iida, T. Kawai, M. Yagawa and H. Kikuchi, "Plate Bending Analysis by Matrix Method," Proceedings of Structural Analysis by Matrix Method Conference, 1979, in Japanese.

```
==================
 Input Data Format
==================

1. File Name
   INPUT.DAT

2. Data Format
-----------------------------------------------------------------------------
/ TITLE /
(72 characters maximum)
/ NODES / ELEMENTS / CONSTRAINED NODES / MATERIALS /
   **N       **E          **C              **M
/ MATERIAL NO. / YOUNG'S MODULUS / POISSON'S RATIO /
       1               #.#            #.#
       2               #.#            #.#
       :                :              :
       :                :              :
       :                :              :
     **M               #.#            #.#
/ NODE NO. /   X   /   Y   /
      1       #.#     #.#
      2       #.#     #.#
      3       #.#     #.#
      :        :       :
      :        :       :
      :        :       :
    **N       #.#     #.#
/ ELEM. NO. / NODE1 / NODE2 / NODE3 / THICKNESS / MATERIAL NO. / SHEAR FACTOR /
      1        **      **      **       #.#           **          0.83333
      2        **      **      **       #.#           **          0.83333
      3        **      **      **       #.#           **          0.83333
      :         :       :       :        :             :             :
      :         :       :       :        :             :             :
      :         :       :       :        :             :             :
    **E        **      **      **       #.#           **          0.83333
```

```
BOUNDARY CONDITION / NODE NO. / CONSTRAINT  /
                       **        %%%   )
                        :         :    |
                        :         :     > **C Lines
                        :         :    |
                       **        %%%   )
FORCES ON NODES / NODE NO. /  FZ  /  MX  /  MY  /
                     **      #.#    #.#    #.#
                      :       :      :      :
                      :       :      :      :
                      :       :      :      :
                    **N      0.0    0.0    0.0   <- This line is required even if
PRESSURE ON ELEMENTS / ELEMENT NO. / PRESSURE /      there are no external forces.
                            1          #.#   )
                            2          #.#   |
                            3          #.#    > **E Lines
                            :           :    |
                            :           :    |
                            :           :    |
                          **E          #.#   )
--------------------------------------------------------------------------------
```

Note 1.

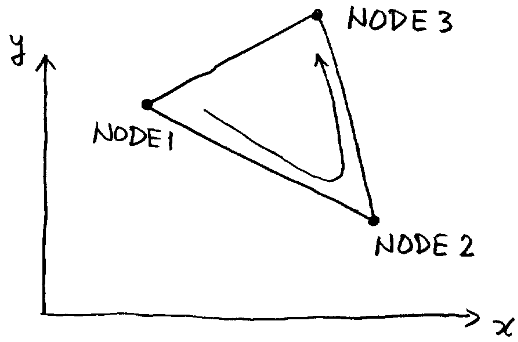
```
**   : Integer
**N  : Integer, Total number of nodes, max. 170
**E  : Integer, Total number of elements, max. 300
**C  : Integer, Total number of constrained nodes, max. 60
**M  : Integer, Total number of materials, max. 25
#.#  : Real
%%%  : %=0 or 1
       example: 101
```

Note 2.

Free format system is used. Divide each number by comma or blank(s).

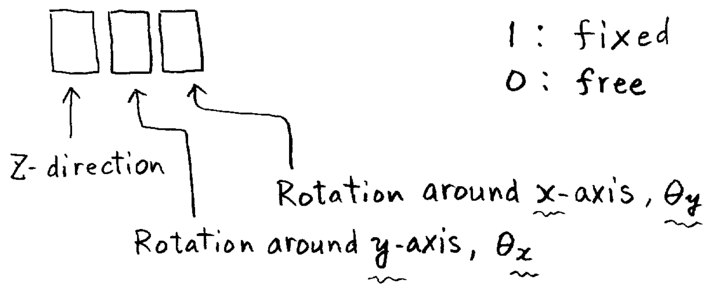
Note 3. Element Data

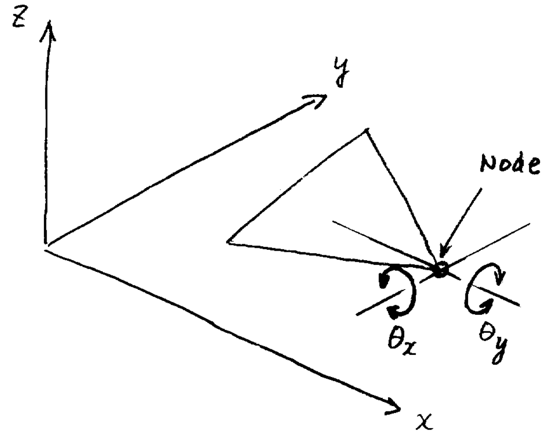
Pick surrounded nodes counterclockwise.



Note 4. Boundary Condition — Expression of Constraint

Example of Boundary Condition Code : 010





```
===================
 Output Data Format
===================
```

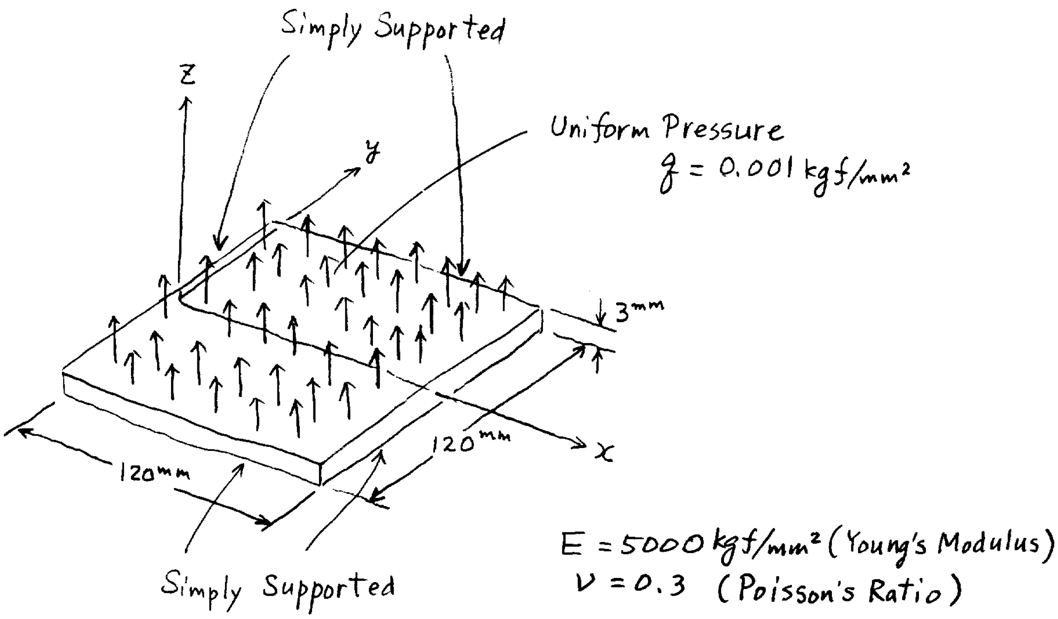
1. File Name
   OUTPUT.DAT

2. Data Format

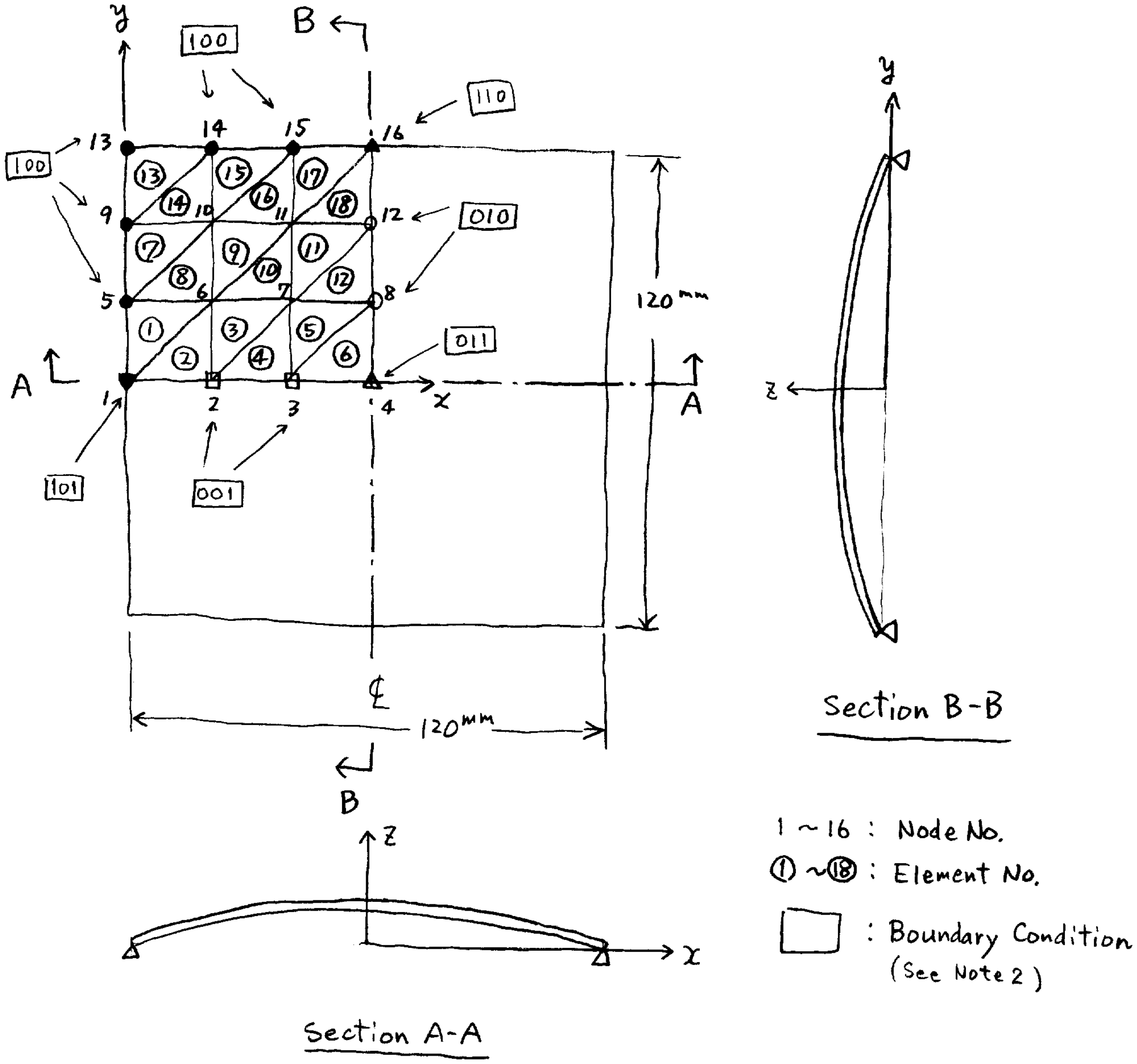
```
----------------------------------------------------------------------------
 CHECK DATA --- SQUARE PANEL, SIMPLY SUPPORTED EDGE, FORCE AT CENTER   <- Title
   16   18   12    1
 MATERIAL PROPERTIES
         1  5000.00      0.30
 NODAL POINTS
         1     0.000     0.000
         2    20.000     0.000
         :       :         :
         :       :         :
         :       :         :
        16    60.000    60.000
 ELEMENTS
    1    1    6    5     3.000    1     0.833
    2    1    2    6     3.000    1     0.833
    :    :    :    :       :      :       :
    :    :    :    :       :      :       :
    :    :    :    :       :      :       :
   18   11   12   16     3.000    1     0.833
 BOUNDARY CONDITIONS
    1  101
    :   :
    :   :
    :   :
   16  110
         4      0.25      0.00      0.00   }  External forces
        16      0.00      0.00      0.00   }
         1        0.00000
```

```
      2         0.00000
      :            :
      :            :
      :            :
     18         0.00000
PTOTAL=  0.00000E+00
============================================
 GRID NO.      W         THETA-X      THETA-Y
============================================
     1    0.00000E+00-0.30491E-03 0.00000E+00
     2    0.54193E-02-0.24863E-03 0.00000E+00
     :     :             :            :
     :     :             :            :
     :     :             :            :
    16    0.00000E+00 0.00000E+00 0.30492E-03
=====================================================
 ELEMENT NO.   X           Y          AREA        W
=====================================================
     1       6.6667      13.333      200.00     0.14733E-02
     2       13.333      6.6667      200.00     0.32797E-02
     :         :           :           :          :
     :         :           :           :          :
     :         :           :           :          :
    18       53.333      46.667      200.00     0.32797E-02
-------------------------------------------------------------------------
```

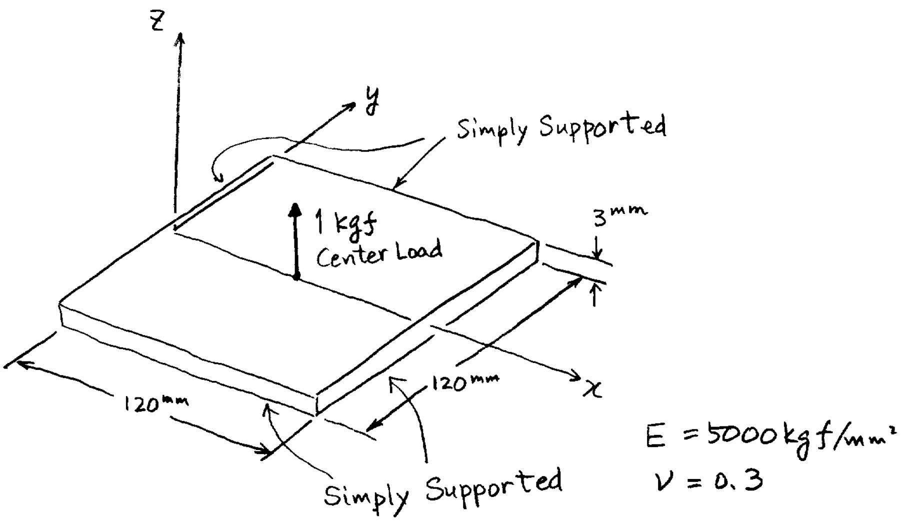
- PTOTAL ← Total pressure $= \sum_{ELEMENT} q_i A_i$, where q_i: Pressure on Element, A_i: Area of Element
- Under the grid table headings: W (deflection), THETA-X (θx), THETA-Y (θy)
- W : Deflection
- θ_x : Rotation around y-Axis
- θ_y : Rotation around x-Axis
- X, Y: Location of Element Center
- AREA: Area of Element
- W: Deflection at Element Center



EXAMPLE 1



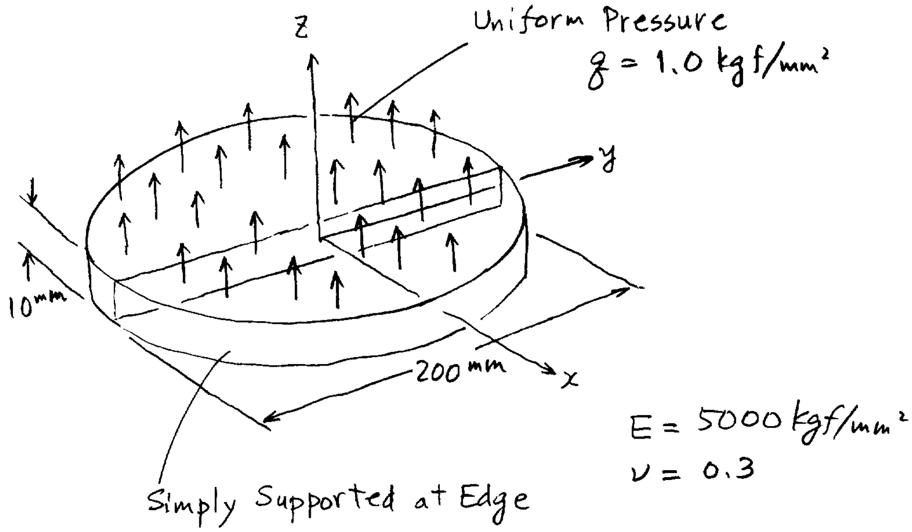
MODELING OF EXAMPLE 1



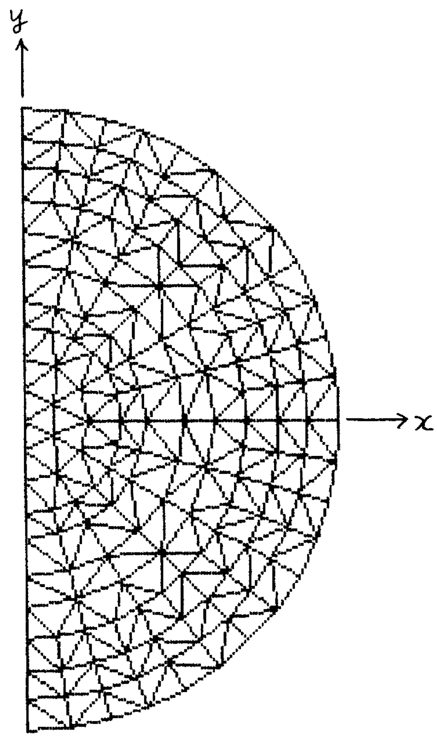
EXAMPLE 2

SAME AS EXAMPLE 1

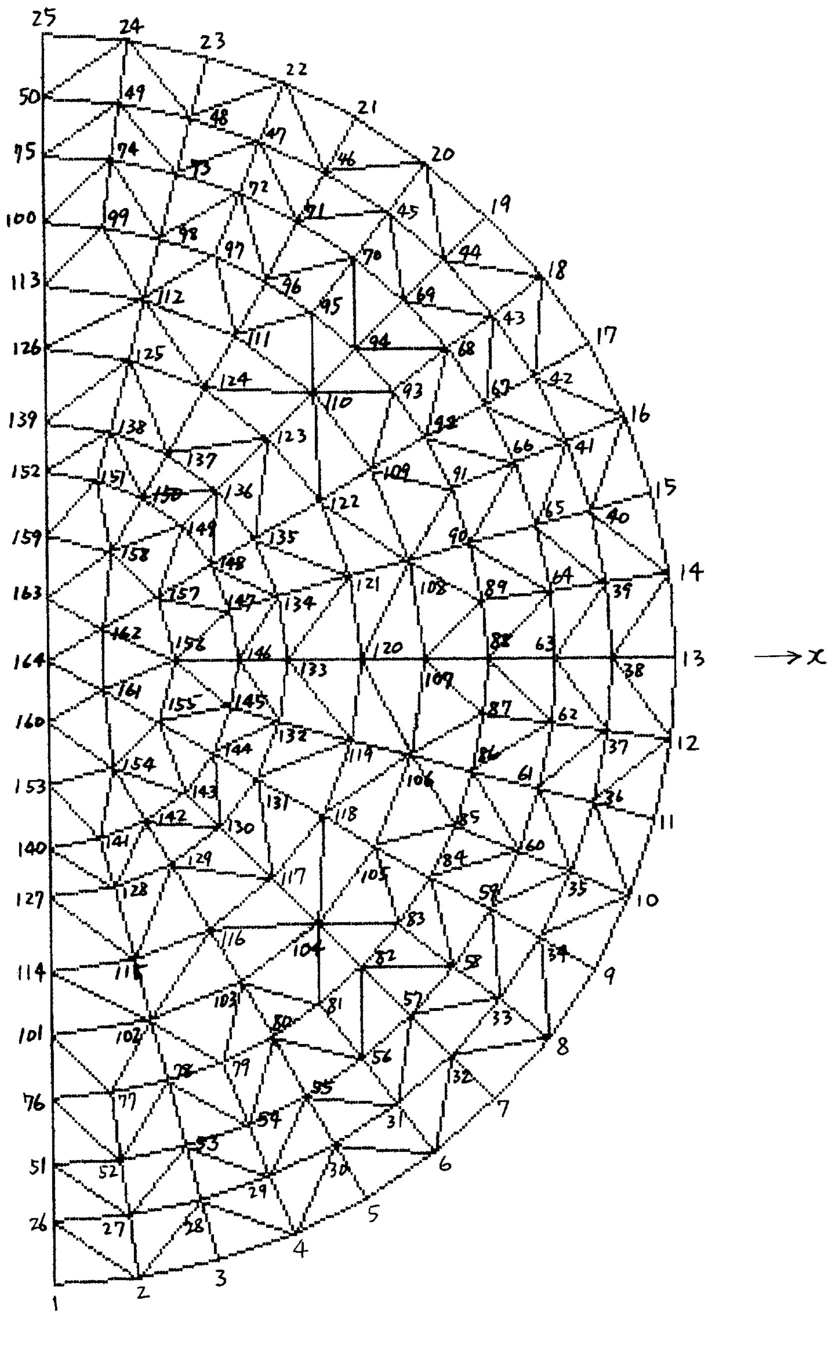
MODELING OF EXAMPLE 2



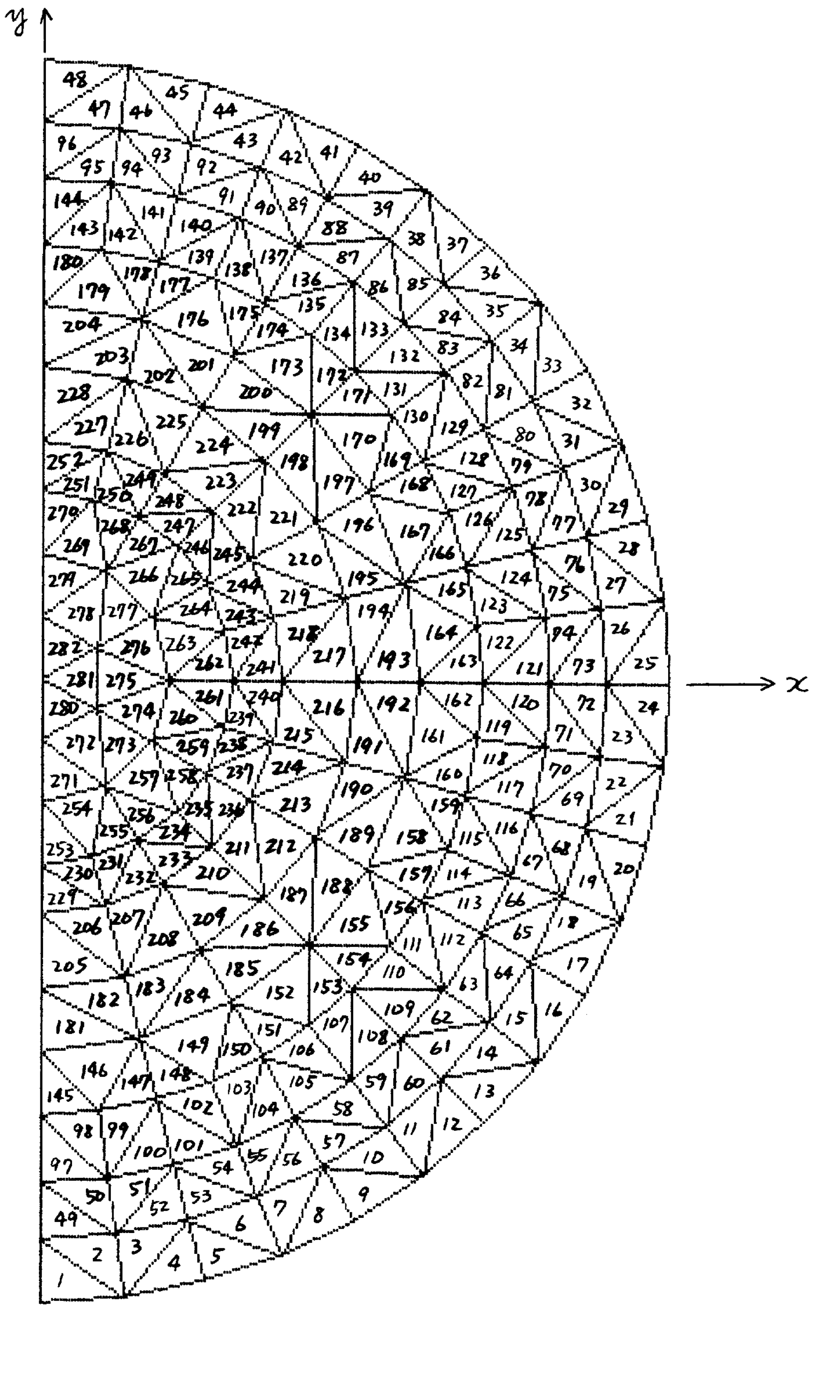
EXAMPLE 5



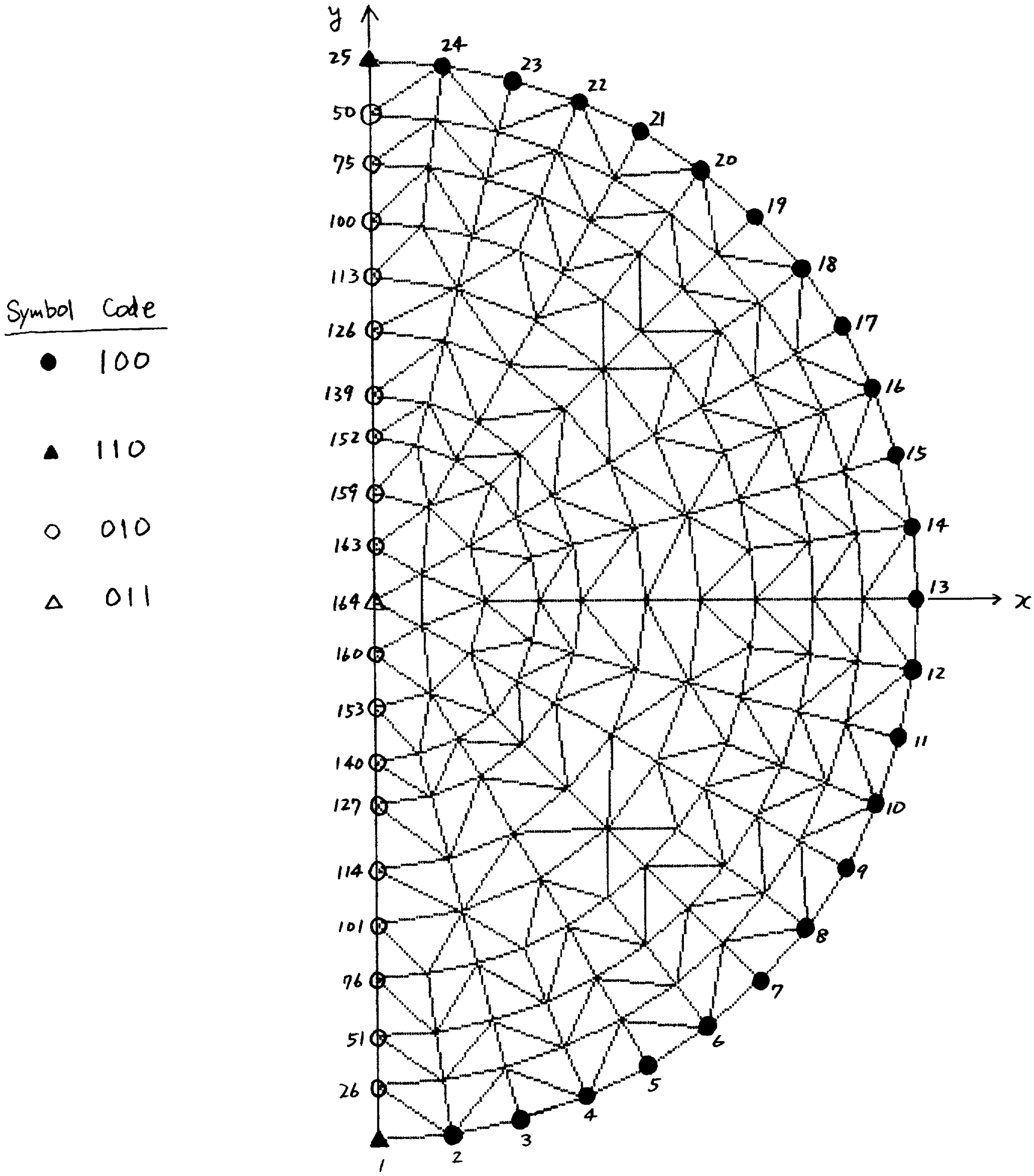
MODELING OF EXAMPLE 5



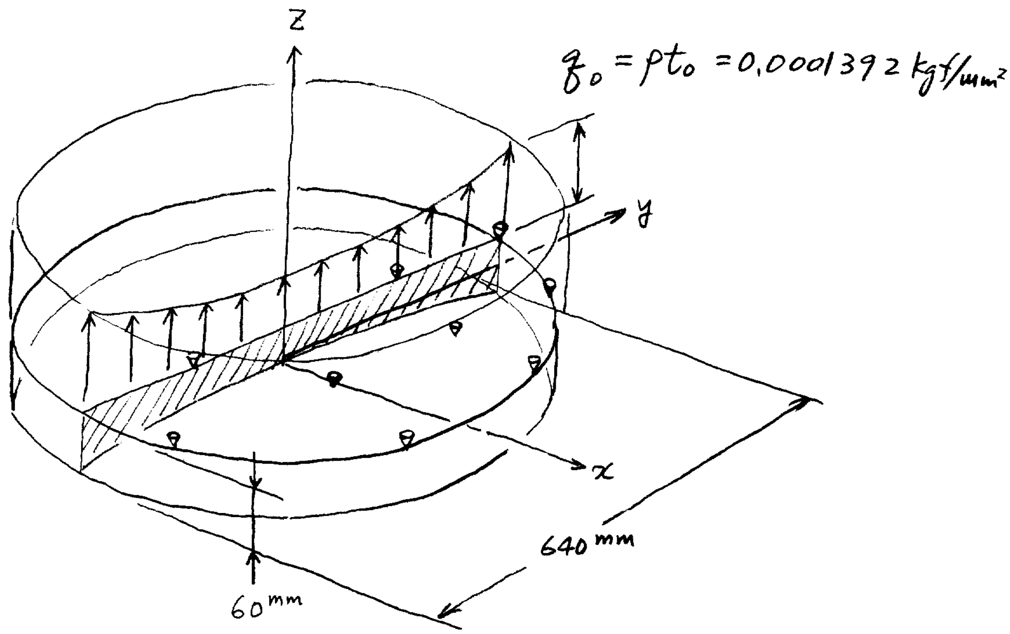
NODE NUMBER FOR EXAMPLES 5, 6, 7



ELEMENT NUMBER FOR EXAMPLES 5, 6, 7



BOUNDARY CONDITION OF EXAMPLE 5



Pyrex : $E = 6700$ kgf/mm² (Young's modulus)
$\nu = 0.2$ (Poisson's ratio)
$\rho = 2.32$ g/cm³
$= 2.32 \times 10^{-6}$ kg/mm³ (density)

EXAMPLE 6 — 15-POINT ASTATIC SYSTEM

$$P_{TOTAL} = \sum_{ELEMENT} q_i A_i = 23.163 \text{ kgf}$$ (FOR HALF MODEL)

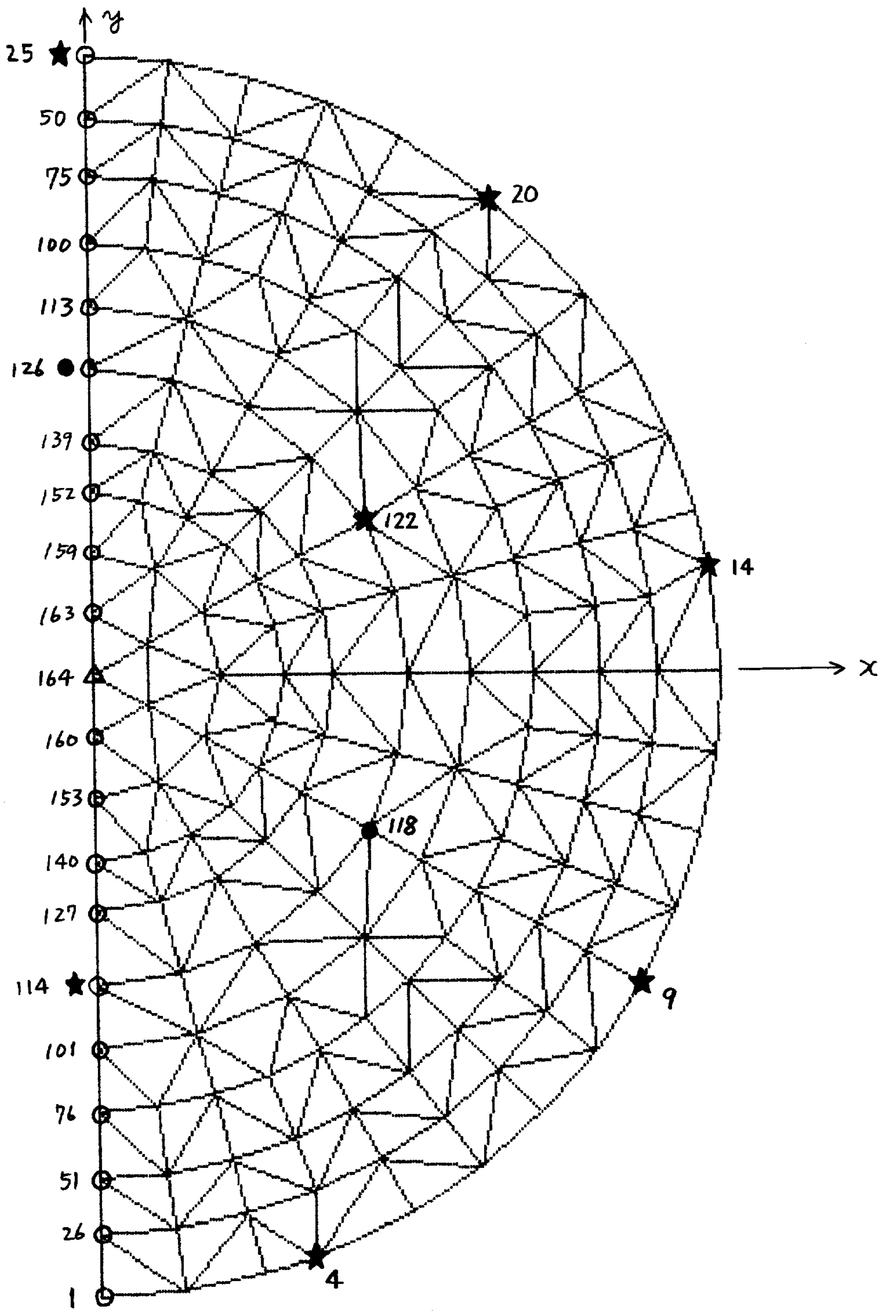
A_i: AREA OF ELEMENT i

q_i: PRESSURE ON ELEMENT i

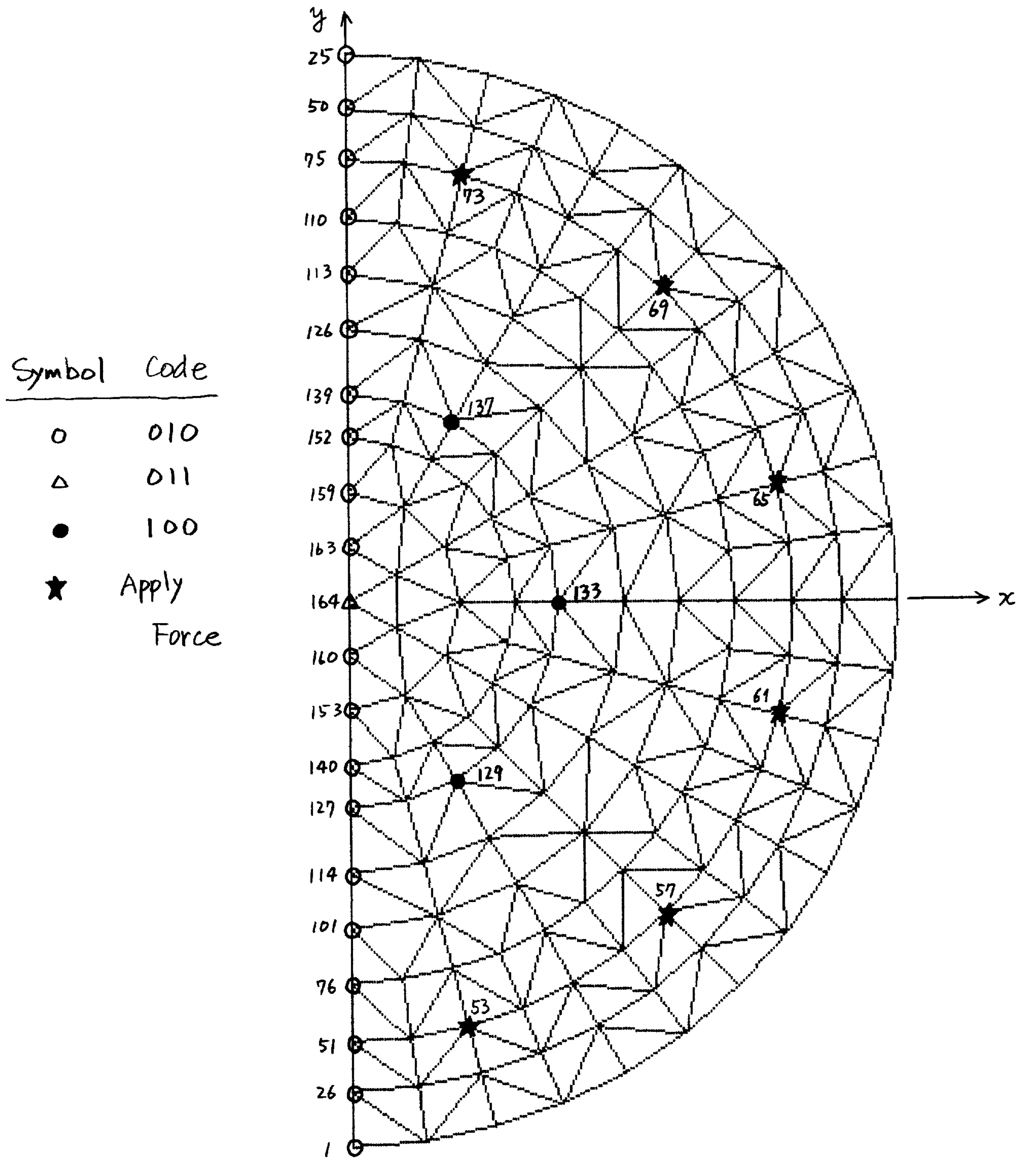
FORCE/POINT @ INNER SUPPORT (65.6% WEIGHT) $= \dfrac{23.163 \times 2 \times 0.656}{6} = 5.06498$ kgf

FORCE/POINT @ OUTER SUPPORT (34.4% WEIGHT) $= \dfrac{23.163 \times 2 \times 0.344}{9} = 1.77068$ kgf

Symbol	Code
●	100
○	010
★	Apply Force
△	011



BOUNDARY CONDITION OF EXAMPLE 6



BOUNDARY CONDITION OF EXAMPLE 7