# FPHM & FPHM-D v5.10
# User's Manual

Shugyo, Minoru

(Prof. Emeritus, Nagasaki Univ., Japan, Dr. Eng.)

March 5, 2024

Contents

The author shall not be responsible for any loss, damages and troubles caused by use of the programs. If you have any questions about the programs, please send an e-mail to the following address: shugyo@nagasaki-u.ac.jp .

## 1. Introduction

The Fibered Plastic Hinge Model (FPHM) program is an analysis code for quasi-static three-dimensional (3D) elastoplastic large deflection analysis of frames that contain steel members, reinforced concrete (RC) members, steel reinforced concrete (SRC) members, concrete filled tube (CFT) members, prestressed concrete (PC) members, composite beams, tension braces, steel damper braces (buckling restrained braces) and truss members. The concept for the model was originally presented by Shugyo (2003a). Since its numerical procedure precisely accounts for $P\Delta$-effects, the ultimate lateral strength of a frame can be obtained accurately.

The model formulations have been done for steel members (Shugyo, Shimazu, and Sakumoto 2005; Shugyo, et al. 2006a; Shugyo, Shimazu, and Hayashida 2007; Shugyo and Shimazu 2010), RC and SRC members (Shugyo, et al. 2006b; Shugyo, et al. 2008), CFT members (Shugyo and Li 1998), and PC members (Shimazu and Shugyo 2014) under almost identical assumptions. The present element model has a semi-rigid function which adds connection compliances to both ends of the element (Shugyo 2003b; Shugyo, Oka, and Li 1996). With this function, it is easy both to treat pin connected members and to introduce rotation stiffnesses and strengths of member connections and exposed column-bases.

The FPHM-dynamic (FPHM-D) program is the analysis code for dynamic 3D elastoplastic large deflection analysis of frames in which the restoring force characteristics of frames are estimated by the FPHM program (Shugyo and Shimazu 2014a). Therefore, it can perform 3D seismic response analysis of frames containing steel members, RC members, SRC members, CFT members, PC members, composite beams, tension braces, steel damper braces and truss members. Since the form of the frame structure input data is almost the same as that for the FPHM program, it is recommended that the input data file for the FPHM-D program is created by adding the data concerned with dynamic behaviors of the frame to the input data file for FPHM after quasi-static analysis to confirm the correctness of the frame structure input data.

The executable files, ‘s64.exe’, ‘ms64.exe’, ‘d64.exe’, ‘md64.exe’ and ‘ft64.exe’, are provided as the zip-file ‘FPHM & FPHM-D v5.10-64E.zip’ with this manual. These executable files are the outputs of compilation by ‘gfortran’ on a Windows 11 64bitOS machine.

If your PC requires .dll file at the beginning of the computation, the installation of tdm-gcc package ( http://jmeubank.github.io/tdm-gcc/ ) is necessary.

The executable files ‘s64.exe’ and ‘d64.exe’ are capable of analyzing a frame model with 1200 nodes. Other executable files ‘ms64.exe’ and ‘md64.exe’ are attached for a compact frame with fewer than 600 nodes.

## 2. Characteristics of the Fibered Plastic Hinge Model (FPHM)

Finite element models for elastoplastic analysis of frames can be categorized into two types of beam-column elements: fiber model and plastic hinge model.

Figure 1 (a) shows a typical fiber model element. A frame member is replaced by an equivalent assembly of fibers obtained by dividing the member in both the longitudinal direction and the cross-sectional direction. This allows the model to accurately treat members with arbitrary cross-sectional shape, the interaction effects of varying axial force and bending moments about two cross-sectional axes, and the influence of the warping moment in an H-shaped steel member. However, the model has a long calculation time because it requires numerical integration for all fibers in every step of the incremental analysis. In addition, the author is concerned that the model has the more serious problem that the management of the constitutive equation of each fiber is very difficult in an iterative process to obtain the balance state of the frame, because the iteration uses the total strains of fibers without separating the plastic strains from the total strains.

By contrast, in a plastic hinge model element, plastic deformations of a frame member are concentrated to a few plastic hinge points. Although the model has the advantages of simplicity and a short calculation time, available analysis codes based on conventional plastic hinge model theory may not assess a strength degradation caused by the large deflection of a frame, exact interaction effects of varying axial force and two bending moments, or a semi-rigidity of member connections. Further, application of this approach to a frame with asymmetrical cross section members is quite difficult.

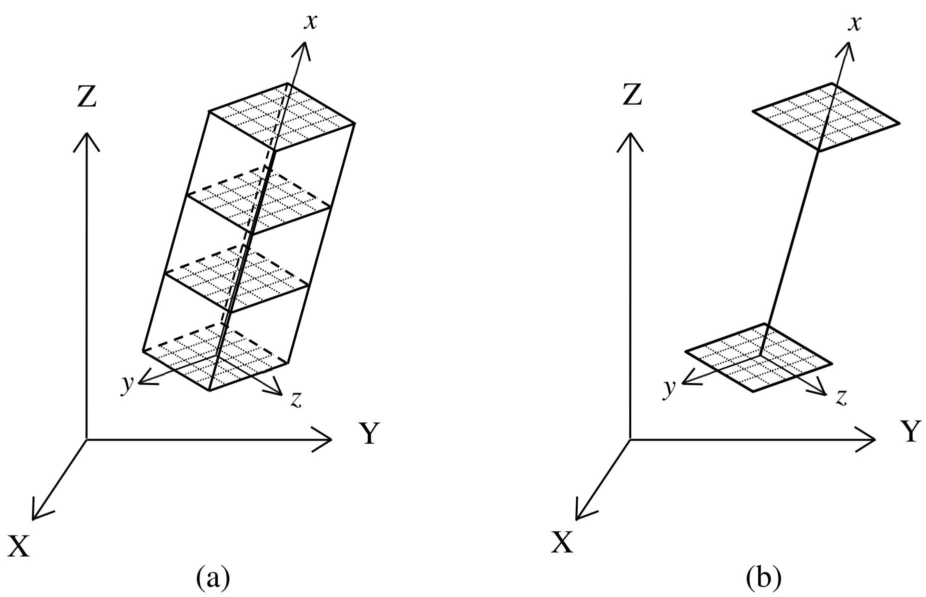



Fig. 1 Schema of (a) a fiber model and (b) the Fibered Plastic Hinge Model

Fibered Plastic Hinge Model (FPHM) is an advanced plastic hinge model. The model element has finely partitioned (fibered into zero-length fibers) cross sections at both ends, i.e., plastic hinges, as shown in Fig. 1 (b). The plastic deformation increments at the plastic hinges are estimated by tangent plastic compliance matrices based on the constitutive equations of those zero-length fibers.

The following assumptions are made in the model formulation for an H-shaped steel member:

(1) The member has a thin-walled H-shaped cross section, and it remains plane and does not distort in the absence of cross-sectional warping,
(2) Deflection is large, but elastic strain is small,
(3) Only axial stress participates in the yielding of fibers of the member,
(4) Plastic deformation consists of only four components that correspond to an axial force, biaxial bending moments, and a warping moment,
(5) There is no local buckling,
(6) Although actual generalized plastic strain increments in a short element generally distribute nonlinearly, this situation is idealized as generalized plastic strain increments distributed linearly with the values at element nodes i and j, and
(7) Incremental plastic deformations in the two half portions occur concentrically in the plastic hinges of zero length at element nodes i and j, respectively.

For a steel member with a hollow circular cross section or hollow square cross section, the assumptions are as follows:

(1) The member has a thin-walled closed cross section and it remains plane,
(2) Deflection is large, but elastic strain is small,
(3) Axial stress and the shearing stress due to St. Venant torsion participate in the yielding of fibers of the member,
(4) Plastic deformation consists of only four components that correspond to an axial force, biaxial bending moments, and a torsional moment,
(5) There is no local buckling,
(6) Although actual generalized plastic strain increments in a short element generally distribute nonlinearly, this situation is idealized as generalized plastic strain increments distributed linearly with the values at element nodes i and j, and
(7) Incremental plastic deformations in the two half portions occur concentrically in the plastic hinges of zero length at element nodes i and j, respectively.

For other steel members and composite members, the assumptions are as follows:

(1) The member cross section remains plane,
(2) Deflection is large, but elastic strain is small,
(3) Only axial stress participates in the yielding of fibers of the member,
(4) Plastic deformation consists of only three components that correspond to an axial force and

biaxial bending moments,

(5) There is no local buckling,

(6) Although actual generalized plastic strain increments in a short element generally distribute nonlinearly, this situation is idealized as generalized plastic strain increments distributed linearly with the values at element nodes i and j,

(7) Incremental plastic deformations in the two half portions occur concentrically in the plastic hinges of zero length at element nodes i and j, respectively, and

(8) Cross-sectional warping is negligible.

The constitutive equation of a steel fiber is derived from the associated flow rule using von Mises yield criterion and Ziegler's kinematic hardening rule under the condition that the post-yield strain hardening coefficient is $\beta E$, where $\beta$ is a non-dimensional strain hardening coefficient and $E$ is Young's modulus. This equation is then a bilinear type equation except for pure steel pipe members subjected to both bending and torsional loads. Figure 2 shows the assumed constitutive equation of a concrete fiber. It is a trilinear type equation including initial tensile strength. Degrading behavior after yielding is not considered.

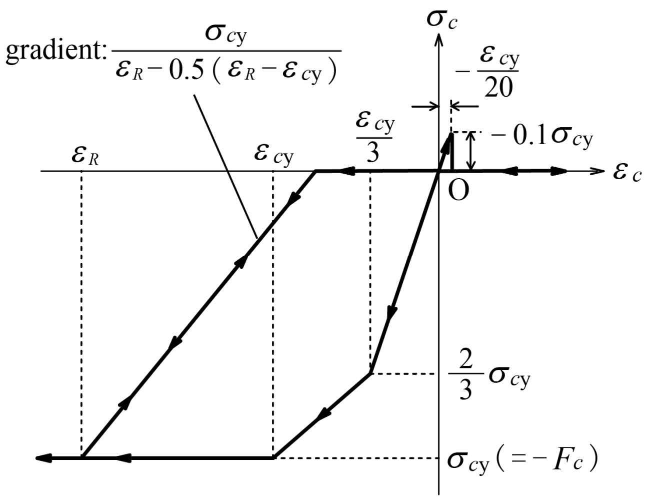



Fig. 2 Constitutive equation of concrete fibers

Characteristics of the present model are as follows.

(1) The model is essentially a plastic hinge model, so it has the benefits of conventional plastic hinge models, i.e. the model formulation process is relatively simple and the required time to calculate elastoplastic problems is not too large.

(2) Any database on the yield surfaces represented by resultant forces is not necessary because

the hysteretic plastic behaviors of plastic hinges are traced utilizing the Newton-Raphson procedure using numerical integration of fiber stresses and stiffnesses. As mentioned above, the element end cross sections are finely partitioned into small areas (zero-length fibers) and hence, the cross-sectional constants, the centroid and the principal axes of the cross section can be obtained in the program using the cross-sectional shape and dimension of each part of the cross section given as the input data.

(3) The influence of the warping moment in an H-shaped steel member, and the interaction effect between normal stress and shearing stress due to the torsional moment in pure steel pipes are considered with respect to the behaviors of plastic hinges. Therefore, good accuracy can be expected in the case where such members are subjected to a large torsional moment.

(4) The model has useful semi-rigid function. Using this function, the semi-rigidity of member connections and column bases can be introduced easily into the frame analysis as element-end compliance. The strength of such connections can also be set.

(5) The computation is very stable, even in the 3D elastoplastic large deflection analysis of a frame, because the use of a modified incremental stiffness method with a small step size does not require iteration.

(6) In the incremental analysis, the internal force vector of a frame in each incremental step is estimated as the gradient of the current total elastic strain energy of the frame. The obtained internal force vector makes it possible to precisely correct the unbalanced force vector of the frame.

## 3. Verification of reliability of the FPHM program

The verification of the reliability of the FPHM program for quasi-static analyses can be seen in the references described in Chapter 1. It has been confirmed through examples that an approximation by four elements for a member gives excellent results. Further, a one element approximation for a member using the modified model, which is obtained by modifying the common assumption (6) stated in Chapter 2, also gives good results. Further detail on the modified model is presented in the latter part of Chapter 4.

The present model can theoretically treat arbitrary members which have asymmetric cross-sectional shapes or contain multiple materials; however, only 16 kinds of members are supported at the present time.

The stiffnesses and strengths of member connections and column bases can be easily introduced into the analysis if these values are known. RC walls must be replaced with equivalent braces, beams and columns. Any shearing fracture of a member is not detected in the program; therefore, it is necessary to examine the obtained shearing force of each member of the frame that contains short-length members.

The verification of the reliability of the FPHM-D program for dynamic analysis was performed through the 3D seismic response analyses of a full-scale four-story steel frame (Shugyo and Shimazu 2014a), and full-scale five-story steel frame with steel damper used in the 'E-defense Blind Analysis Contest 2009'. The author was the third-place winner of the 'E-defense Blind Analysis Contest 2009' in the category of 3D analysis, steel damper
( https://www.bosai.go.jp/hyogo/blind-analysis/2008/index_e.html )

## 4. Basic form of input data file for the FPHM program and examples

### (1) Frame analysis by the standard model

The one-bay one-story plane portal frame shown in Fig. 3 is analyzed to show a basic form of the input data file. The frame consists of two square steel pipe columns and an H-shaped steel beam. The square steel pipe is □-300×300×9, and the H-shaped steel is H-400×200×8×13. Two element-end cross sections of each element are automatically partitioned into small areas (zero-length fibers) in the program using the input data: the cross-sectional shape and dimension of each part. Figure 4 shows the partitioning of cross sections.

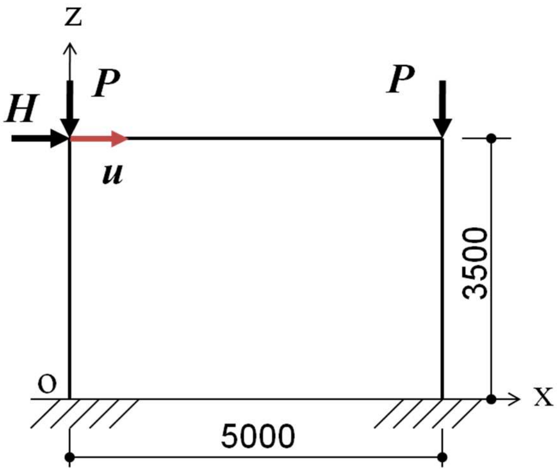



Fig. 3 Global coordinate axes, sizes of the frame and loading condition

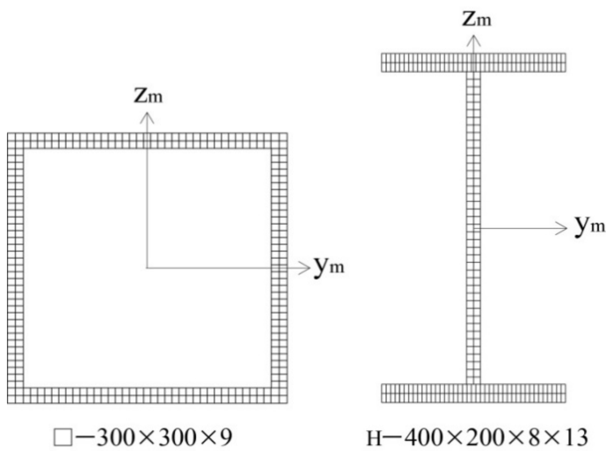



Fig. 4 Partitioning of element-end cross sections and element coordinate axes

To examine the influences of finite element modeling of the frame, analyses are performed for three cases of approximation: one element for a member, two elements for a member and four elements for a member. Since the FPHM program has a remodeling function that divides an element into two, three or four elements, the input data must be prepared for only a one element approximation per member as shown in Fig. 5. However, other nodes are required when the loading points exist in addition to the member connection points.

In Fig. 5, normal numbers are node numbers and circled numbers represent element numbers. The coordinates of nodes and pairs of node numbers belonging to the elements can be obtained from the figure. The coordinates of nodes in the Y-axis direction are needed, even in a plane frame analysis. Figures 6 and 7 show the automatically generated node numbers and element numbers when the remodeling number is 2 or 4, respectively.

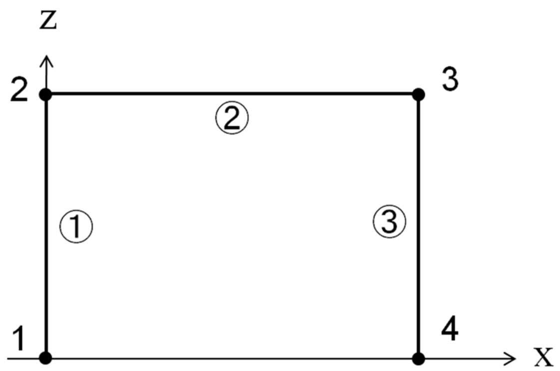



Fig. 5 One element approximation model of the frame (from input data)

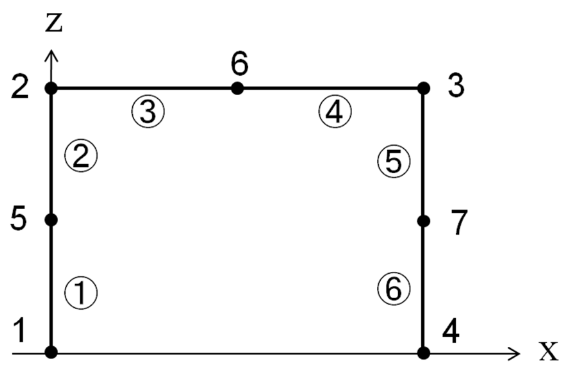



Fig. 6 Two element approximation model of the frame (remodeling number is 2)

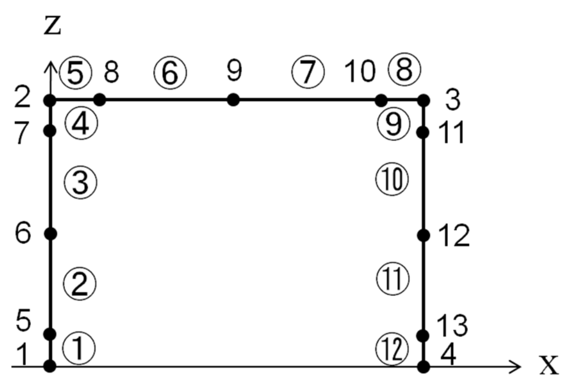



Fig. 7 Four element approximation model of the frame (remodeling number is 4)

When the remodeling number is 4, three new nodes are added to all the elements at the points of 0.1$L$, 0.5$L$ and 0.9$L$, where $L$ is the length of the element. Total numbers of both nodes and elements increase, and element numbers are renumbered. Those new numbers and coordinates are written in the output file 's6.txt' (see Chapter 6). Examination of the analysis results must be done using the information in 's6.txt'.

Preparation of the input data to define each element, such as cross-sectional shape, cross-sectional dimensions, material properties and plastic deformation reduction coefficient $r$ (see Section (2) of this Chapter), is described in Section (3) of Chapter 5.

The present program always treats a frame three-dimensionally, so the chord angle of each element is necessary even in a plane frame analysis. The definition of a chord angle is given in Section (4) of Chapter 5. In a plane frame analysis, almost all chord angles of the elements become 0 provided that the global coordinate system (X, Y, Z) is set up as shown in Fig. 3.

The present program has a semi-rigid function for member connection points and the column base. Detailed explanation of the semi-rigid function is presented in Section (5) of Chapter 5. The input data concerned with the semi-rigid function must be written in the input data file even if any element does not use the semi-rigid function (see Section (5) of Chapter 5).

Fixed boundary conditions must be included in the input data. To make the boundary conditions input data, the degrees of freedom of a node in the analysis must be known. As mentioned above, the present program always treats a frame three-dimensionally. In addition, the cross-sectional warping of an H-shaped steel element caused by an axial torsion is considered. Therefore, in the present program, there are 7 degrees of freedom. The components are displacements $U$, $V$, $W$, rotation angles $\theta_X$, $\theta_Y$, $\theta_Z$, and $\psi$, which corresponds to warping. In general, the seventh component $\psi$ must be fixed for all nodes, otherwise the global stiffness matrix of the frame becomes singular. In the present example, the fixed boundary conditions are as follows from Fig. 3: $U$ of nodes 1 and 4, $V$ of every node, $W$ of nodes 1 and 4, $\theta_X$ of every node, $\theta_Y$ of nodes 1 and 4, $\theta_Z$ of every node, and $\psi$ of every node. Regarding the fixed boundary conditions, the program first reads the numbers of fixed nodes in one line in order of $U$, $V$, $W$, $\theta_X$, $\theta_Y$, $\theta_Z$, $\psi$, and then reads the corresponding node numbers in the following lines. If any number of fixed nodes in the first line is -1, then all the corresponding nodes are fixed and the input of node numbers can be omitted.

A frame is usually subjected to two kinds of loads: an initially loaded constant load set (initial loads) and a gradually loaded incremental load set (incremental loads). The components of initial loads have the following directions and actual values, respectively: forces $F_X$, $F_Y$, $F_Z$, and bending moments $M_X$, $M_Y$, $M_Z$. The components of incremental loads have the following directions and relative values, respectively: force increments $dF_X$, $dF_Y$, $dF_Z$, and bending moment increments $dM_X$, $dM_Y$, $dM_Z$. Regarding the loading conditions, the program first reads the numbers of initial loads in one line in order of $F_X$, $F_Y$, $F_Z$, $M_X$, $M_Y$, $M_Z$, and then successively reads the corresponding node numbers and the actual values of loads in the following lines. Next, the program reads the numbers of incremental loads in one line in order of $dF_X$, $dF_Y$, $dF_Z$, $dM_X$, $dM_Y$, $dM_Z$, and then reads the corresponding node numbers and the relative values of incremental loads in the following lines successively. In the present example, the two *P*s and *H* shown in Fig. 3 are the initial vertical loads and an integrated value of the horizontal incremental load, respectively.

The numerical analysis by the present program is done through a 'displacement-controlled load increment method'; hence, the displacement to control the computation (control-displacement) must be specified by the input data. In the present example, the displacement in the X-axis direction of node 2, which is the same direction as the aforementioned incremental load, is selected as the control-displacement. The direction of control-displacement is specified using its flexibility number in the degrees-of-freedom table of the whole frame. For example, the flexibility number of the Z-axis direction of node 1 is 3, and that of the X-axis direction of node 2 is 8, because the degrees of freedom per node is 7 in the present program. The magnitude of the control-displacement is also needed. If the specified magnitude is less than $10^{-8}$, a suitable value is automatically set in the program.

The present program covers cyclic loading problems, so the number of unloading points and the value of the control-displacement at each unloading point are required. In the present example, the number of unloading points is 1 because the loading is monotonic, and the value of the control-displacement at the unloading point is set as 100mm (0.029 rad in story drift angle).

To obtain the load-displacement relationships as the results of analysis, five pairs of flexibility numbers of load and displacement can be assigned. In the present example, the frame shown in Fig.5, the author selected the horizontal load *H* (flexibility number is 8) as the load for every pair, and the horizontal and vertical displacements at node 2 (flexibility numbers are 8 and 10, respectively) and node 3 (flexibility numbers are 15 and 17, respectively) as the displacement.

The contents of the input data file written according to the above explanation for the present example are shown below. The first input data is the remodeling number. Although the remodeling number used here is 1, the numbers 1 to 5 can be used as the remodeling number, and 5 specifies the use of the modified model with a one element approximation for a member (see Chapter 4). The input data file is followed by the total numbers of nodes and elements, coordinates of each node and so forth. The comment lines within quotation marks are also required. The unit system is free. Although the SI unit system is recommended, the author generally uses the (mm, N, s) unit system. The value of the initial load *P* in Fig. 3 is 1,040,000N, which is obtained as 0.3*Py* where *Py* is the yield axial force of the column.

-------- from here --------

```
 ‘remodeling number (5 indicates modified model)’
1
 ‘numbers of nodes and elements’
4, 3
 ‘node number and coordinates’
1, 0, 0, 0
2, 0, 0, 3500
3, 5000, 0, 3500
4, 5000, 0, 0
 ‘element number and two node numbers at element-ends’
1, 1, 2
2, 2, 3
3, 4, 3
 ‘number of kinds of members’
2
 ‘section-number of each element (in order of element number)’
1, 2, 1
 ‘data of cross section 1’
 ‘S’, 300, 300, 9, 9, 330, 210000, 0.01, 1.0
0, 0, 0, 0
0, 0, 0
0, 0, 0
’data of cross section 2’
 ‘H’, 400, 200, 8, 13, 326, 210000, 0.01, 1.0
0, 0, 0, 0
0, 0, 0
0, 0, 0
 ‘chord angle of each element (in order of element number)’
0, 0, 0
 ‘number of elements with semi-rigid node i’
0
 ‘number of elements with semi-rigid node j’
0
```

```
'number of tension brace elements'
0
'number of steel damper brace and truss elements, and element numbers'
0
'number of slab equivalent brace elements and element numbers'
0
'numbers of fixed nodes (in order of U, V, W, θX, θY, θZ, ψ), and node numbers'
2, -1, 2, -1, 2, -1, -1
1, 4
1, 4
1, 4
'numbers of initial loads (in order of FX, FY, FZ, MX, MY, MZ), and data'
0, 0, 2, 0, 0, 0
2, -1040000
3, -1040000
'numbers of incremental loads (in order of dFX, dFY, dFZ, dMX, dMY, dMZ), and data'
1, 0, 0, 0, 0, 0
2, 1
'flexibility number of control-displacement and displacement increment'
8, 1E-8
'number of unloading points and displacement at those points'
1
100
'five pairs of flexibility numbers of load and displacement for output'
8,8,  8,15,  8,10,  8,17,  8,17
```

-------- to here --------

The above input data file is attached under the name 'frame-s-2d-e.txt', where 's' and '2d' refer to static two-dimensional (2D) analysis and 'e' indicates the English version. The element model used in the above analysis is the 'standard model' because the remodeling number is 1, which is less than 5. If the remodeling number is 5, this indicates the use of the modified model.

Figures 8-1, 8-2, and 8-3 show the obtained relationships between the horizontal load $H$ and the horizontal displacement $u$ at node 2 under the vertical initial loads 0, $0.3Py$ and $0.5Py$, respectively. As mentioned in Chapter 3, it has been confirmed that a four element approximation for a member (remodeling number is 4) gives excellent results through various examples. It is obvious from the figures that the restoring forces obtained by a one or two element approximation for a member are lower than those obtained by a four element approximation for a member. Assumption (6) in Chapter 2 is the cause of these decreases of restoring forces. In other words, assumption (6) causes an overestimation of plastic deformations of a frame if the remodeling number is 1 in the analysis using the standard model.

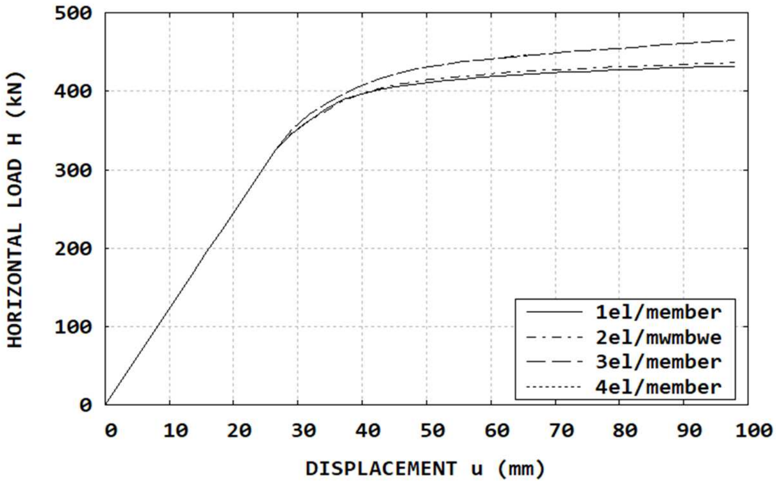



Fig. 8-1 Comparison of horizontal load-displacement relations (standard model : *P*=0)

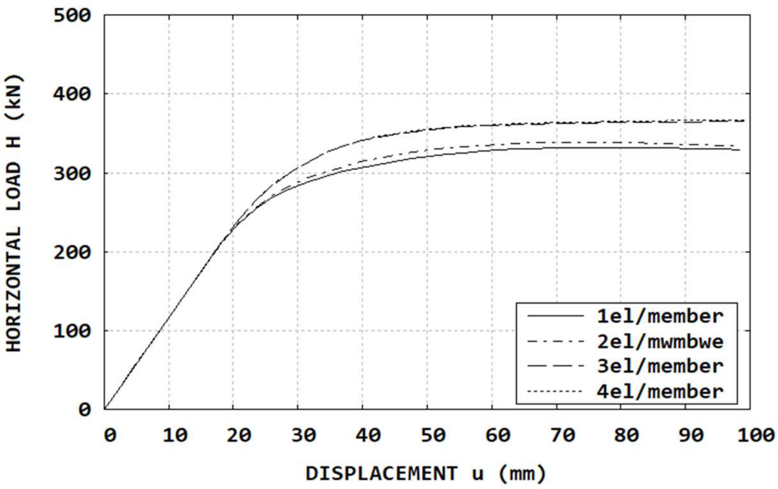



Fig. 8-2 Comparison of horizontal load-displacement relations (standard model : *P*=0.3*Py*)

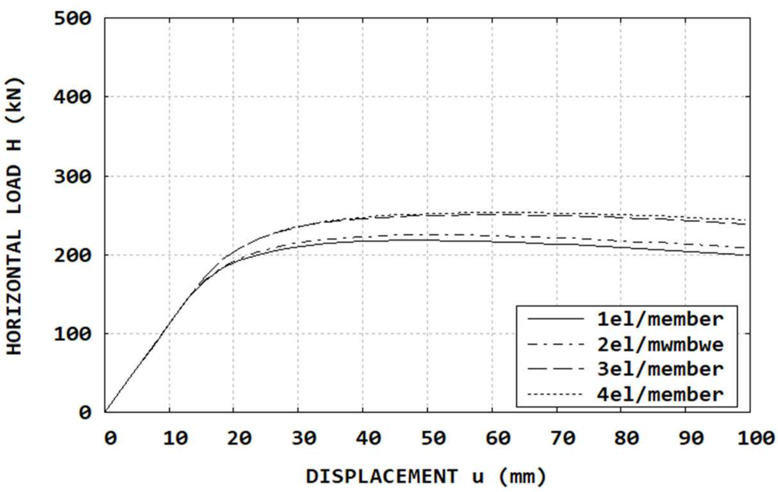



Fig. 8-3 Comparison of horizontal load-displacement relations (standard model : *P*=0.5*Py*)

### (2) The same analysis by the modified model

As mentioned above, assumption (6) in Chapter 2 causes the overestimation of plastic deformations of a frame under the one element approximation for a member by the standard model, and usually a four element approximation for a member is necessary. This means a great increase in computation time for large-scale frames. For practical use, it is desirable that the one element approximation for a member possesses sufficient accuracy. From this perspective, Figures 8-1, 8-2, and 8-3 suggest that the introduction of a plastic deformation reduction coefficient $r$ is effective, at least for rigid frame members.

The optimum value of the plastic deformation reduction coefficient $r$ (variable name in the program is VARSP) was examined by Shugyo and Shimazu (2016) through quasi-static analysis of four kinds of one-bay one-story plane portal frames with the same frame-size (see Fig. 3), 3D quasi-static analysis of a two-bay four-story steel frame containing composite beams and semi-rigid column bases, and 3D quasi-static analysis of a twenty-story eccentric steel frame with H-shaped steel columns. The results suggest the optimum value of $r$ is 0.3 (VARSP=0.3).

Figures 8-4, 8-5, 8-6, 8-7, 8-8, and 8-9 are citations from Shugyo and Shimazu (2016). Figures 8-4, 8-5, and 8-6 show the comparisons of the horizontal load-displacement relationships at node 2 of the frame shown in Fig. 3 under the vertical initial loads 0, $0.3Py$ and $0.5Py$, respectively. The frame is a weak column type frame. Figures 8-7, 8-8, and 8-9 are the same comparisons for the frame where the beam is changed from H-400×200×8×13 to H-350×175×7×11: a weak beam type frame. It is obvious from the figures that the maximum difference in horizontal load between a precise result by the four element approximation for a member and the result by a one element approximation for a member with $r$=0.3 is 4% at the loading end point, $u$=100mm, of the 'curve of $r$=0.3' in Fig. 8-6. This difference may be allowable for practical use.

Here, the modified model is defined as the standard model with the plastic deformation reduction coefficient $r$=0.3. Conversely, the standard model can be defined as the modified model with the plastic deformation reduction coefficient $r$=1.0. It is important to recall that the modified model with $r$=0.3 can only be used for the members in which an inflexion point exists under the applied loads. For other members, such as compression braces in which an axial force is major and a buckling may occur, the four element approximation for a member using the modified model with $r$=1.0 is needed.

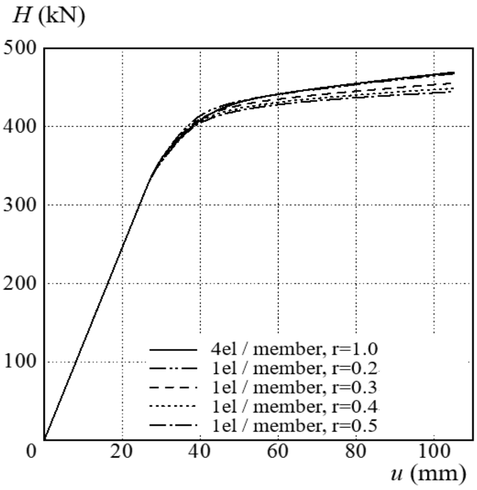



Fig. 8-4 Comparison of *H*-*u* relations (weak column frame, *P*=0)

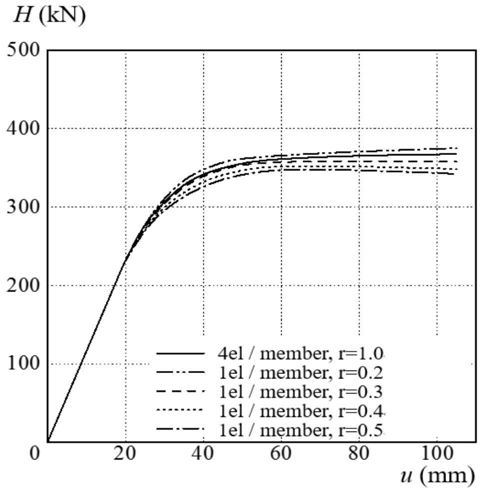



Fig. 8-5 Comparison of *H*-*u* relations (weak column frame, *P*=0.3*Py*)

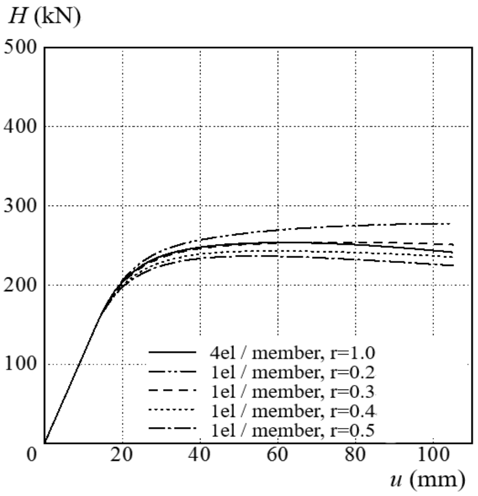



Fig. 8-6 Comparison of *H*-*u* relations (weak column frame, *P*=0.5*Py*)

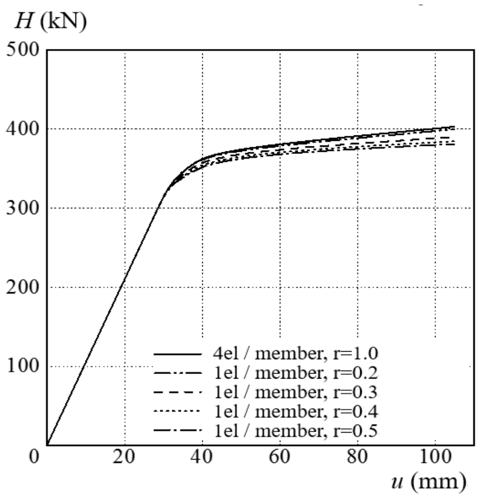



Fig. 8-7 Comparison of *H*-*u* relations (weak beam frame, *P*=0)

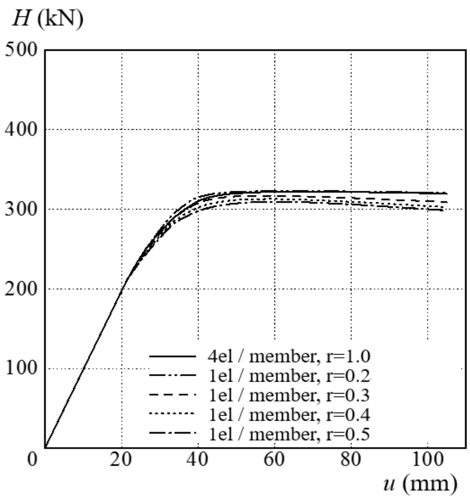



Fig. 8-8 Comparison of *H*-*u* relations (weak beam frame, *P*=0.3*Py*)

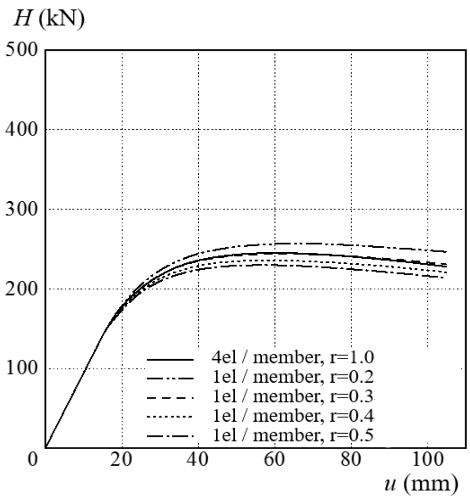



Fig. 8-9 Comparison of *H*-*u* relations (weak beam frame, *P*=0.5*Py*)

## 5. Preparation of input data

At the beginning of input data file, the remodeling number (1 or 2 or 3 or 4) or the use of modified model (5) must be specified.

### (1) Division of frame into finite elements and acquisition of node coordinates

Since the present program always treat a frame three-dimensionally, the chord angle of each element is required for coordinate transformation. Thus, it is recommended to set the global X-axis as the horizontal right direction, and the global Z-axis as the vertical upper direction as shown in Fig. 3 because most chord angles of elements become 0 under that setting up of the global coordinate system.

The frame is divided into elements with nodes that are made at the member connections and loading points in principle. The nodes and elements are assigned their numbers, respectively. The order of the assignment is not restricted. Then, the node coordinates are acquired. The form of input data of node coordinates (Xi, Yi, Zi, i=1,2,…) are as follows for the present example shown in Fig. 3:

```
'node number and coordinates'
1,0,0,0
2,0,0,3500
3,5000,0,3500
4,5000,0,0
```

These data must be written in order of node number. Missing numbers are not allowed. If unnecessary lines are found in the input data, let all components of the displacement vector of the corresponding nodes be fixed (see Section (6) of this Chapter).

### (2) Node numbers at both ends of an element

The form of input data of node numbers at both ends i and j of elements are as follows for the present example:

```
'element number and two node numbers at element-ends'
1,1,2
2,2,3
3,4,3
```

These data must be written in order of element number. Missing numbers are not allowed. If unnecessary lines are found in the input data, let the yield stress and Young's modulus of corresponding elements be $10^8$ and 0, respectively (see Section (3) of this Chapter). It is recommended to assign the lower-end node number for a column-element, and the left-end node number for a beam-element to the number of node i of the element for convenience of the analysis result examination.

### (3) Cross-sectional shapes, dimensions and material constants

#### (a) Standard members

Regarding the mechanical properties that characterize each member of the frame, the program first reads how many kinds of members exist in the frame. Each kind of member is assigned a 'member-number'. The author uses the word 'section-number' herein instead of 'member-number' to avoid confusion. Next, the program reads the section-number of each element in order of element number. Then, the program reads the data concerned with the mechanical properties of each kind of member. Table 1 shows the variable names used in the program and their meanings.

Table 1: Variable names and their meanings for a member

| Variable name | Meaning |
|---|---|
| *CSEC* | name of cross-sectional shape (character variable) |
| *D,B,WTHK,FTHK* | dimensions of steel cross section |
| *SYSS* | yield stress of steel |
| *E* | Young's modulus of steel |
| *HARD* | non-dimensional strain hardening coefficient $\beta$ |
| *VARSP* | plastic deformation reduction coefficient $r$ |
| *CD,CB* | dimensions of concrete cross section |
| *CYSS* | yield stress of concrete |
| *CE* | initial elastic modulus of concrete |

*CSEC* (character variable: data is a character within quotation marks) and the dimensions of the cross section (*D, B, WTHK, FTHK, CD, CB)* for each kind of member are shown in Fig. 9-1. The axes of the local coordinate system ($x_m$, $y_m$, $z_m$) are also shown in the figure. The longitudinal direction of each member is set as the $x_m$-axis. *CSEC*='C' and *CSEC*='S' indicate circular pure steel pipe and square pure steel pipe, respectively, when *CYSS* is 0; however, *CSEC*='C' and *CSEC*='S' indicate concrete filled circular steel pipe and concrete filled square steel pipe, respectively, if *CYSS* is not 0.

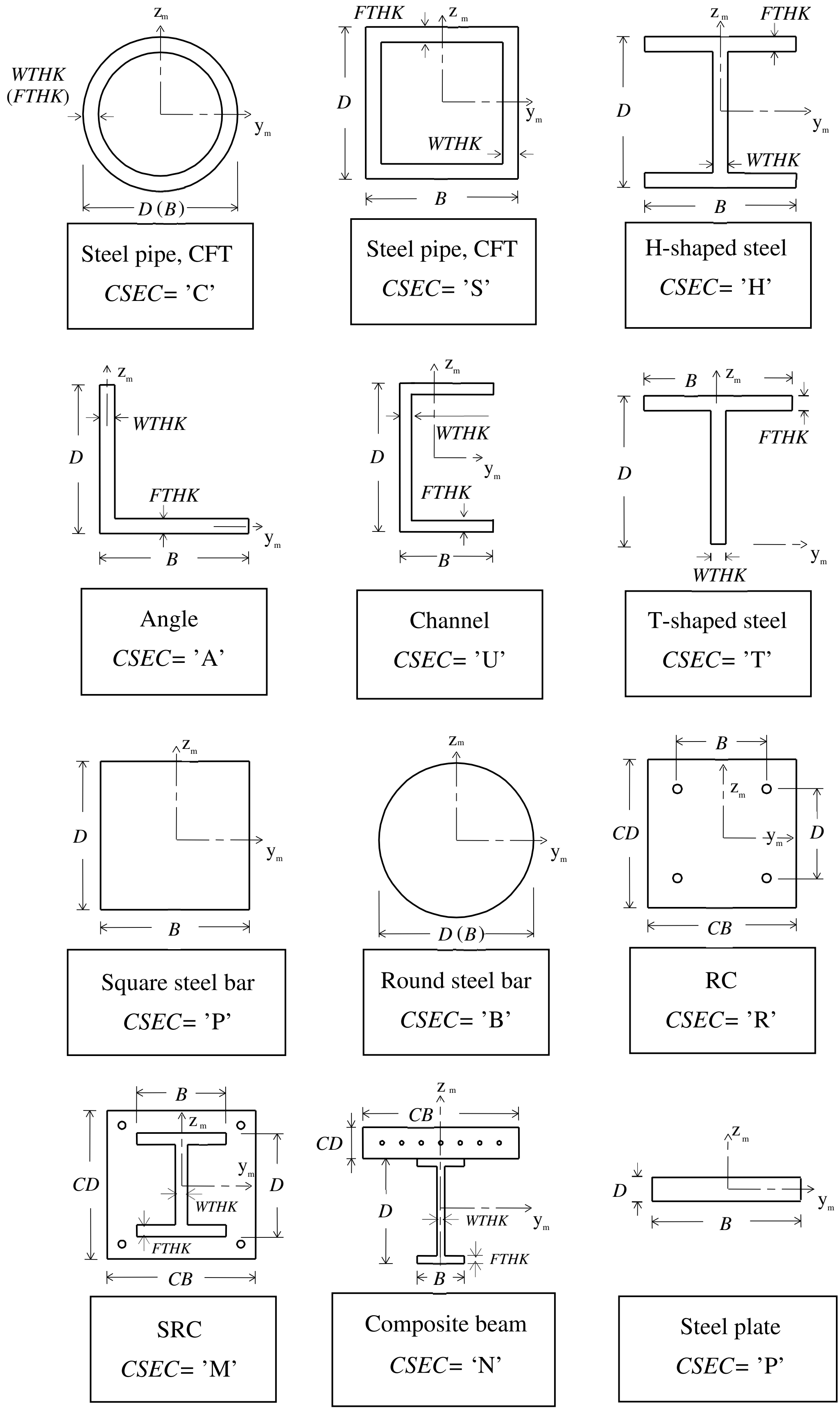


Fig. 9-1 Names of cross-sectional shapes, dimensions and local coordinate axes of members

The coordinate axes, $y_m$ and $z_m$, in Fig. 9-1 are not always the principal axes of the cross section. The principal axes of the cross section and the cross-sectional constants, e.g. cross-sectional area and cross-sectional secondary moments about the two principal axes, are obtained using the above input data and written into the output file 's6.txt' (see Chapter 6). Users must determine the chord angles (see Chapter 4) by using the $z_m$-axis shown in Fig. 9-1. The actual chord angles are recalculated in the program.

The data of the mechanical properties of each kind of member must be written in the input data file successively in order of section-number. The data are divided into several data lines. The first line is a comment line such as 'data of cross section 1'. The data on the next line correspond to *CSEC*, *D, B, WTHK, FTHK,* and *VARSP,* respectively, and those on the third line correspond to *CD, CB, CYSS,* and *CE,* respectively. The author usually uses the compressive strength of concrete $F_c$ as the data of *CYSS*. The initial elastic modulus of concrete $E_c$ may be estimated by the following equation:

$$E_c = 20580\sqrt{0.05Fc} \qquad (\mathrm{N/mm^2})$$

The input data file is followed by the data about reinforcing steels and reinforcing fiber sheets. In the present example, the assigned section-number of square pure steel pipe and H-shaped steel are 1 and 2, respectively, and the members do not contain any other materials, i.e. concrete, reinforcing steels or reinforcing fibers. Therefore, the form of input data of the mechanical properties are as follows for the present example:

```
'number of kinds of members'
2
'section-number of each element (in order of element number)'
1,2,1
'data of cross section 1'
'S',300,300,9,9,330,210000,0.01,1.0   (CSEC,D,B,WTHK,FTHK,SYSS,E,HARD,VARSP )
0,0,0,0                               (CD,CB,CYSS,CE )
0,0,0                                 (NELMR,RYSS,RE )
0,0,0                                 (NELMF,FYSS,FE )
'data of cross section 2'
'H',400,200,8,13,326,210000,1.0       (CSEC,D,B,WTHK,FTHK,SYSS,E,VARSP )
0,0,0,0                               (CD,CB,CYSS,CE )
0,0,0                                 (NELMR,RYSS,RE )
0,0,0                                 (NELMF,FYSS,FE )
```

If the value for *CYSS* in the data block of 'data of cross section 1' is not 0, e.g. -20, the elements assigned section-number=1 are treated as CFT members with filled concrete of strength 20N/mm$^2$, and *CE* must have a value, e.g. 20580. The values for *CD* and *CB* are free.

Table 2 shows the variable names for reinforcing steels and reinforcing fibers used in the program and their meanings. The variable names for reinforcing steels, *NELMR, RYSS, RE, RU, RV* and *RAREA*, are used only for RC members, SRC members, composite beams and CFT members with reinforcing steels. The variable names for reinforcing fibers, *NELMF, FYSS, FE, FU, FV* and *FAREA,* are supported only for RC members at this time.

Table 2: Variable names and their meanings for reinforcing steels and reinforcing fibers

| Variable name | Meaning |
|---|---|
| *NELMR* | number of reinforcing steels |
| *RYSS* | yield stress of reinforcing steel |
| *RE* | Young's modulus of reinforcing steel |
| *RU* | $y_m$-coordinate of the centroid of reinforcing steel |
| *RV* | $z_m$-coordinate of the centroid of reinforcing steel |
| *RAREA* | cross-sectional area of reinforcing steel |
| *NELMF* | number of reinforcing fibers |
| *FYSS* | yield stress of reinforcing fiber |
| *FE* | Young's modulus of reinforcing fiber |
| *FU* | $y_m$-coordinate of the centroid of reinforcing fiber |
| *FV* | $z_m$-coordinate of the centroid of reinforcing fiber |
| *FAREA* | cross-sectional area of reinforcing fiber |

An example of the form of input data of the mechanical properties for an RC member is shown in the following, assuming section-number=3. *D* and *B* are the distances between the centroids of the outer reinforcing steels in the $z_m$ and $y_m$ directions, respectively. For the values for *SYSS* and *E*, the respective values of *RYSS* and *RE* must be used. The values for *WTHK* and *FTHK* are both free. The last line means that any reinforcing fiber is not used.

```
'data of section 3'
'R',780,280,1,1,294,206000,0.01,1.0     (CSEC,D,B,WTHK,FTHK,SYSS,E,HARD,VARSP )
900,400,-18,19500                       (CD,CB,CYSS,CE )
14,294,206000                           (NELMR,RYSS,RE )
140,390,491                             (RU,RV,RAREA )
46.7,390,491                               (14 lines are necessary)
-46.7,390,491
```

…………..
46.7,-390,491
-46.7,-390,491
-140,-390,491
0,0,0 (*NELMF,FYSS,FE* )

**(b) PC beam (*CSEC*='D')**

A PC member can be considered as an RC member with high strength PC steel bars. Therefore, the basic form of input data is almost the same as that of a standard RC member, but some additional data concerning the PC steel bars are required. The necessary additional data are: number of PC steel bars *NELMP*, yield stress *PYSS*, Young's modulus *PE*, coordinates on the $y_m$-axis *PU* and on the $z_m$-axis *PV*, cross-sectional area *PAREA,* and tensile prestress ratio to the yield stress of PC steel bar *PRETEN* .

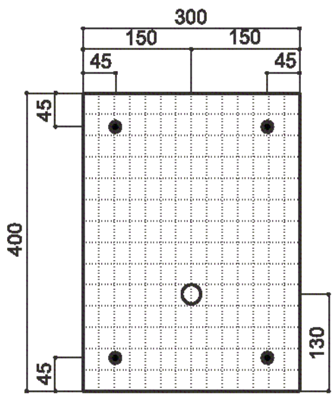



Fig. 9-2 Cross section of a PC member

Figure 9-2 shows a cross section of a simple PC member. In the figure, ● and ○ indicate reinforcing steels and PC steel bar, respectively. An example of the form of input data of the mechanical properties for a PC member is shown in the following, assuming section-number=4. For the values for *SYSS* and *E*, the respective values for *PYSS* and *PE* must be used. Red indicates additional data.

| | |
|---|---|
| 'data of section 4' | |
| 'D',310,210,1,1,1004,206000,0.01,1.0 | (*CSEC,D,B,WTHK,FTHK,SYSS,E,HARD,VARSP* ) |
| 1,1004,206000 | (*NELMP, PYSS, PE* ) |
| 0,-70,506.7,0.8 | (*PU, PV, PAREA, PRETEN* ) |
| 400,300,-24.7,22887 | (*CD,CB,CYSS,CE* ) |
| 4,328,177000 | (*NELMR,RYSS,RE* ) |

105,-155,126.7 (*RU,RV,RAREA* )
-105,-155,126.7 (4 lines are necessary)
105,155,126.7
-105,155,126.7
0,0,0 (*NELMF,FYSS,FE* )

**(c) SRC members with cross-H-shaped steel and T-shaped steel (*CSEC*='O', *CESC*='Q')**

A standard SRC member with an H-shaped core steel (*CSEC*='M') is already supported as shown in Fig. 9-1. Here, two other SRC members are added. The cross-sectional shapes of their core steels are cross-H and T, respectively. Figures 9-3 and 9-4 show their cross sections partitioned into small areas (zero-length fibers). The values for *CSEC* are 'O' and 'Q', respectively. There are some restrictions on the dimensions of the cross sections to ensure the form of input data is the same as for a standard SRC.

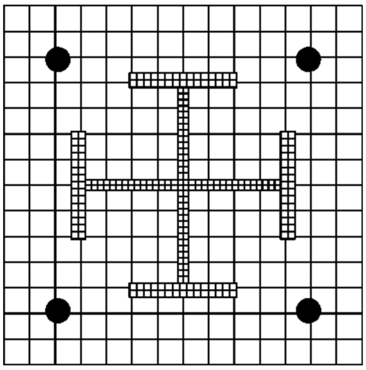


Fig. 9-3 Cross-H-shaped steel SRC

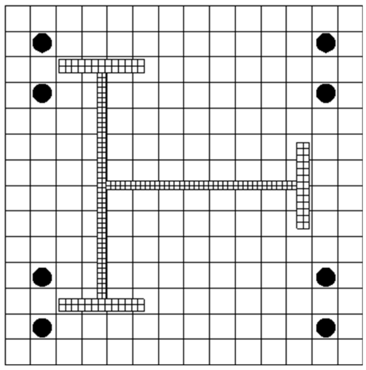


Fig. 9-4 T-shaped steel SRC

These restrictions are as follows:

1. For both members, the cross-sectional shape of concrete is square, and its centroid is the origin of the $y_m$ and $z_m$ axes.
2. A cross-H-shaped steel is made by connecting the same two H-shaped steels at their web centers.
3. A T-shaped steel is made by connecting the webs of the same two H-shaped steels such that the distance between the two edges of core steel in the $y_m$ direction equals the depth of the original H-shaped steel as shown in Fig. 9-3.

The number of reinforcing steels is free. Young's moduli for core steel and reinforcing steel are reset to the mean value of those of these two materials in the program for rapid convergence of the computation.

### (d) Slab-added RC beam (*CSEC*='F')

A standard RC member (*CSEC*='R') is already supported as shown in Fig. 9-1. However, the influences of an RC slab on the bending behaviors of an RC beam cannot be considered by the standard RC member. Here, a slab-added RC beam is defined. The slab-added RC beam can be considered as an RC member with an RC slab of effective width on its top. Therefore, the basic form of input data is almost the same as for the standard RC member, but some additional data concerned with the RC slab are needed. The necessary additional data are as follows: thickness of the RC slab, *SD,* and effective width of the RC slab, *SB*. The value for *CSEC* is 'F'.

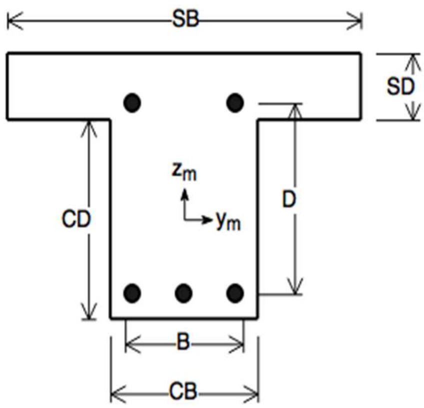



Fig. 9-5 Cross section of a slab-added RC beam

Figure 9-5 shows the cross section of a slab-added RC beam. An example of the form of input data of the mechanical properties for a slab-added RC beam is shown in the following, assuming section-number=5. Red indicates additional data.

| | |
|---|---|
| 'data of section 5' | |
| 'F',310,210,1,1,370,175000,0.01,1.0 | (*CSEC,D,B,WTHK,FTHK,SYSS,E,HARD,VARSP* ) |
| 400,300,-24.7,22887 | (*CD,CB,CYSS,CE* ) |
| 80,400 | (*SD,SB* ) |
| 5,370,175000 | (*NELMR,RYSS,RE* ) |
| 105,-155,506.7 | (*RU,RV,RAREA* ) |
| 0,-155,506.7 | (5 lines are necessary) |
| -105,-155,506.7 | |
| 105,220,506.7 | |
| -105,220,506.7 | |
| 0,0,0 | (*NELMF,FYSS,FE* ) |

### (e) Virtual element for eccentricity and element exclusion

[1] Virtual element for eccentricity

To set a slight eccentricity to an element, a short rigid element whose length equals the eccentricity is generally used. For this virtual element, a square-shaped steel bar (*CSEC*='P') with a high elastic modulus and high strength is recommended. If the yield stress is set to greater than $10^8$, the program does not check the yielding of the element. This makes the computation stable.

[2] Element exclusion

If unnecessary lines are then found in the input data regarding the node numbers at both ends of the elements (see Section (2) of this Chapter), let the yield stress and Young's modulus of corresponding elements be $10^8$ and 0, respectively. This can be done by making a new member in which *CSEC*='P', *SYSS*=$10^8$, and *E*=0. The element assigned this member absolutely does not contribute to the frame analysis.

## (4) Chord angle

To construct the tangent stiffness matrix of the whole frame, it is necessary to perform coordinate transformation of the tangent stiffness matrix of each element from the local coordinate system ($x_m$, $y_m$, $z_m$) to the global coordinate system (X, Y, Z). The coordinate transformation matrix contains a chord angle α which informs the direction of the $z_m$-axis in the global space. This section explains how to get the chord angle of each element. The procedure is as follows:

1. Suppose the plane *P* which is perpendicular to the $x_m$-axis. Draw on *P* the new coordinate axis Z' which is parallel to the plane made by the X-axis and the Z-axis. The positive direction of the Z'-axis is unknown at this time.
2. Define the Y'-axis which forms a right-handed system with the $x_m$-axis and the Z'-axis and where the positive direction is the same as the Y-axis. This decides the positive direction of the Z'-axis.
3. The angle measured counterclockwise from the Z'-axis to the $z_m$-axis on the plane *P* is chord angle α.

As mentioned in Chapter 3, almost all chord angles of the elements become 0 in a plane frame analysis, provided the global coordinate system (X, Y, Z) is set up as shown in Fig. 3. When the $x_m$-axis agrees with the Y-axis, the Z'-axis becomes indeterminate. In this case, the angle measured counterclockwise from the Z-axis to the $z_m$-axis is chord angle α.

### (5) Semi-rigid function and designation of special member

#### (a) Semi-rigid connection

The present model has a distinctive semi-rigid function. The function makes it possible to set a zero-length semi-rigid connection element with arbitrary rotational compliance and rotational strength to the element-end. This is a modification of the fundamental properties of the element determined in Section (3) of this Chapter.

For the both element-end nodes i and j, two cases of settings can be specified as follows:

| | |
|---|---|
| function no. 0 | setting of rotational compliance only is possible |
| function no. 1 | settings of rotational compliance and strength are possible |

For convenience of explanation, the four physical quantities $M_{cy1}$, $M_{cy2}$, $C_1$, and $C_2$ that characterize function no. 1 are shown in Fig. 10. Figure 10 shows typical double flag type restoring force characteristics. The vertical axis is the bending moment $M$, and the horizontal axis is the rotation angle $\theta$. The terms $C_1$ and $C_2$ are the compliances for the respective state in Fig. 10. These four physical quantities must be known in advance.

In function no. 0, only the initial compliance $C_1$ can be set.

In the input data file, the input data for all the elements with semi-rigid node i must be written first. Then, the input data for all the elements with semi-rigid node j must be added.

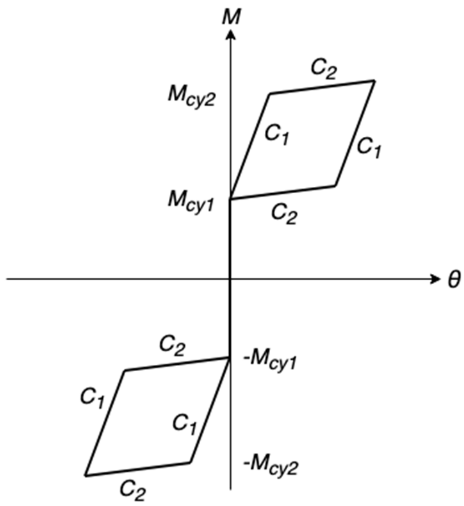



Fig. 10 Double flag type restoring force characteristics ($C_1$ and $C_2$ are compliances)

**(b) Tension brace**

The tension brace here is defined as the element having only tensile axial stiffness and almost zero compressive resistance. It is assumed that the tensile axial stiffness is *EA/L* in the elastic region and softens to 1/20~1/200 of *EA/L* after yielding, where *E* is Young's modulus, *A* is the cross-sectional area, and *L* is the length of the brace. As additional input data, the axial plastic compliance after yielding, $C_3$ ,is needed.

Figure 11-1 shows the hysteretic change of axial plastic compliance. In the figure, *N* is an axial force, $\delta p$ is a plastic elongation, and *Np* is the axial force at the preceding unloading point of the element. The yield axial force *Ny* is calculated in the program using the input data described in Section (3) of this Chapter. As the input data for $C_3$, the inverse of 1/20~1/200 of *EA/L* can be used. This tension brace must be used by the one element approximation for a brace member. If some nodes exist in the brace member, the global tangent stiffness matrix becomes singular.

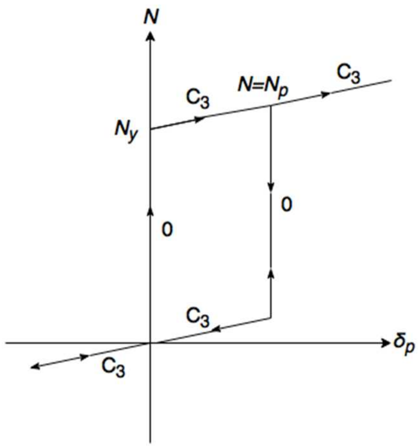



Fig. 11-1 Axial plastic compliances of the tension brace

The results of the incremental analysis of a frame that contains the tension brace are strongly affected by the displacement increment. The use of a displacement increment of 1/500~1/1000 of the initial yield displacement is recommended.

Figure 11-2 is an example of a hysteretic axial load-elongation relationship of a round bar subjected to a cyclic loading. The analysis was done using the tension brace element with a one element approximation. The dimensions and mechanical properties of the round bar are as follows: diameter *D*=22mm, length *L*=2280mm, yield stress $\sigma_y$ =253.8N/mm$^2$, Young's modulus *E*=217600N/mm$^2$, and axial plastic compliance after yielding $C_3$=0.00275mm/N. The value of $C_3$ is the inverse of 0.01*EA/L*. The elongations at the unloading points are 3, -3, 6, -2, and 8 (mm). The numbers in the figure indicate the loading path. The gradient of the elastic range is *EA/L*, which is calculated in the program.

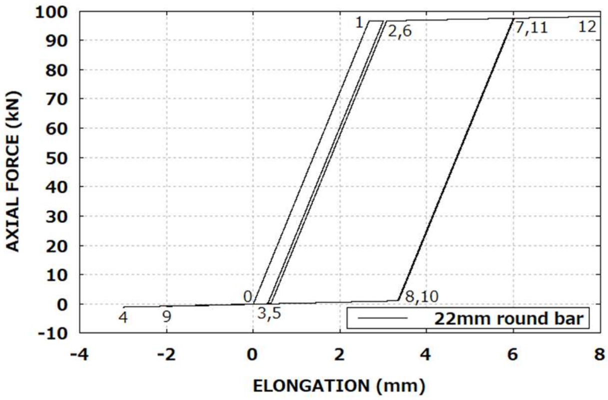


Fig. 11-2 Axial load-elongation relationship of a round bar

**(c) Steel damper brace (Buckling restrained brace) and truss member**

Here, the steel damper brace is defined as the element which has an equivalent uniform cross-sectional shape, dimensions, yield stress, and Young's modulus, neglecting the details of frame-brace connections. Therefore, the experimental result of the cyclic loading test for the whole steel damper device is needed to estimate the abovementioned physical quantities. The author uses *CSEC*='P' as the steel damper brace element. The equivalent cross-sectional area can be obtained by using an experimental result of the axial force-elongation relationship in the elastic range. Then, the dimensions of the cross section can be determined. Further, the yield stress can be set so that the yield axial force is equivalent to that of the experiment (see Section (3) of this Chapter). This steel damper brace element must be used by a one element approximation for a brace member.

Figure 12 is the hysteretic axial load-elongation relationship of a steel damper of the full-scale 5-story steel frame used in the 'E-defense Blind Analysis Contest 2009' (see Chapter 3). The axial stiffness after yielding is set to 1/100 of the elastic stiffness in the program.

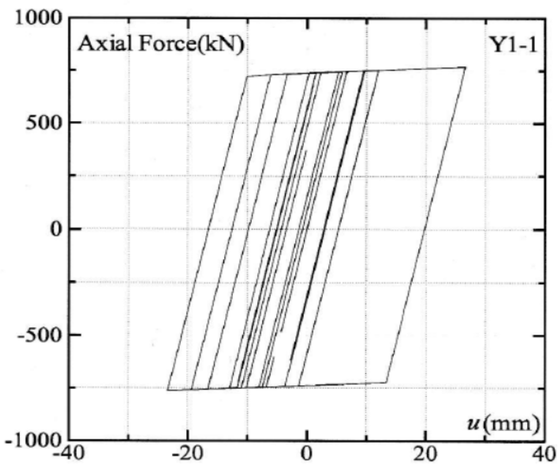


Fig. 12 Axial load-elongation relationship of a steel damper

Since the designated element is set to an element with only axial stiffness, this designation can also be used to set an element to a general truss member.

### (d) Brace to replace the in-plane stiffness of an RC slab

This replacement is done to approximately realize a rigid floor assumption. The brace to replace the in-plane stiffness of the RC slab is hereinafter called a slab equivalent brace. For the element designated as a slab equivalent brace, the bending stiffnesses are neglected and any yielding is not checked in the program. This makes the computation stable. For convenience, *CSEC*='P' is used as a slab equivalent brace. It is recommended to set the axial stiffness to less than 10% of that of the nearest beam. This slab equivalent brace must be used by a one element approximation for a brace member.

### (e) Example of input data for semi-rigid function

The input data must be written in order of semi-rigid connections, tension braces, steel damper braces (buckling restrained braces), and slab equivalent braces. As mentioned above, the forms of input data are different from one another.

To show an example of input data, the following situations are assumed for several elements of a frame:

1. Node i of the 15th element is a pin connection.
2. The bending strength at node i of the 18th element is 50kNm.
3. Node j of the 15th element is a pin connection.
4. The 8th and 13th elements are tension braces.
5. The 10th and 12th elements are steel damper braces.
6. The 8 elements from 21st to 28th are slab equivalent braces.

The input data in this case are as follows, noting that all the comment lines are required. For steel damper braces, truss members and slab equivalent braces, a condensed writing of the element numbers is possible when the element numbers are consecutive. For example, if the relevant number of elements is -1 and the data for element numbers are written as '21,28', the specified elements are the 8 elements from 21st to 28th.

```
'number of elements with semi-rigid node i'
2
'element number and data'
15                                  (element number)
```

| | |
|---|---|
| 0 | (function number) |
| 1E-3 | ($C_1$ about $y_m$-axis) |
| 1E-3 | ($C_1$ about $z_m$-axis) |

'element number and data'

| | |
|---|---|
| 18 | (element number) |
| 1 | (function number) |
| 0, 50E6, 0, 1E-3 | ($M_{cy1}$, $M_{cy2}$, $C_1$, $C_2$ about $y_m$-axis) |
| 0, 50E6, 0, 1E-3 | ($M_{cy1}$, $M_{cy2}$, $C_1$, $C_2$ about $z_m$-axis) |

'number of elements with semi-rigid node j'

1

'element number and data'

| | |
|---|---|
| 15 | (element number) |
| 0 | (function number) |
| 1E-3 | ($C_1$ about $y_m$-axis) |
| 1E-3 | ($C_1$ about $z_m$-axis) |

'number of tension braces'

2

'element number and data'

| | |
|---|---|
| 8 | (element number) |
| 6E-3 | ($C_3$ in $x_m$-axis (axial direction)) |

'element number and data'

| | |
|---|---|
| 13 | (element number) |
| 6E-3 | ($C_3$ in $x_m$-axis (axial direction)) |

'number of steel dampers and truss members'

| | |
|---|---|
| 2 | (number of elements) |
| 10, 12 | (element numbers) |

'number of slab equivalent braces'

| | |
|---|---|
| -1 | (number of elements) |
| 21, 28 | (element numbers) |

For the semi-rigid connection element, the program always requires data about the $y_m$-axis and the $z_m$-axis, even in a plane frame analysis. On these two coordinate axes, different physical quantities can be used. To set a pin connection, it is recommended to use the inverse of $1/10^6 \sim 1/10^8$ of the bending stiffness *EI* of the relevant element as the bending compliance. Zero compliance indicates a rigid connection.

### (6) Fixed boundary conditions

As mentioned in Section (1) of Chapter 4, the present program always treats a frame three-dimensionally, and a cross-sectional warping of an H-shaped steel element caused by an axial torsion is considered. Therefore, in the present program, the degrees of freedom is 7. The components are displacements $U$, $V$, $W$, rotation angles $\theta_X$, $\theta_Y$, $\theta_Z$, and $\psi$, which corresponds to warping. In general, the seventh component $\psi$ must be fixed for all nodes; otherwise, the global stiffness matrix of the frame becomes singular.

For the example shown in Fig. 3, the fixed boundary conditions are as follows: $U$ of nodes 1 and 4, $V$ of every node, $W$ of nodes 1 and 4, $\theta_X$ of every node, $\theta_Y$ of nodes 1 and 4, $\theta_Z$ of every node, and $\psi$ of every node. For the fixed boundary conditions, the program first reads the numbers of fixed nodes in one line in order of $U$, $V$, $W$, $\theta_X$, $\theta_Y$, $\theta_Z$, $\psi$, and then reads the corresponding node numbers in the following lines. If any number of fixed nodes in the first line is -1, then all the corresponding nodes are fixed and writing the node numbers is not necessary. The corresponding input data are as follows:

‘numbers of fixed nodes and element numbers’

| | |
|---|---|
| 2, -1, 2, -1, 2, -1, -1 | (numbers of fixed nodes of $U, V, W, \theta_X, \theta_Y, \theta_Z, \psi$) |
| 1, 4 | (node numbers for $U$ ) |
| (unnecessary) | (node numbers for $V$ ) |
| 1, 4 | (node numbers for $W$ ) |
| (unnecessary) | (node numbers for $\theta_X$ ) |
| 1, 4 | (node numbers for $\theta_Y$ ) |
| (unnecessary) | (node numbers for $\theta_Z$ ) |
| (unnecessary) | (node numbers for $\psi$ ) |

**(7) Initial loads (constant loads such as sustained loads)**

The components of the initial loads have the following directions and actual values, respectively: forces $F_X$, $F_Y$, $F_Z$, and bending moments $M_X$, $M_Y$, $M_Z$. The program first reads the numbers of initial loads in one line in order of $F_X$, $F_Y$, $F_Z$, $M_X$, $M_Y$, $M_Z$, and then successively reads the corresponding node numbers and the actual values of loads in the following lines. For the example shown in Fig. 3, the input data are as follows:

| | |
|---|---|
| 'numbers of initial loads (in order of $F_X$, $F_Y$, $F_Z$, $M_X$, $M_Y$, $M_Z$), and data' | |
| 0, 0, 2, 0, 0, 0 | (numbers of loads $F_X$, $F_Y$, $F_Z$, $M_X$, $M_Y$, $M_Z$) |
| 2, -1040000<br>3, -1040000 | (node numbers and actual values of loads) |

The results for the initial loadings are written into the output file 's6.txt' (see Chapter 6). Hence, the situation of the frame under the initial loads can be checked.

**(8) Incremental loads**

The components of incremental loads have the following directions and relative values, respectively: force increments $dF_X$, $dF_Y$, $dF_Z$, and bending moment increments $dM_X$, $dM_Y$, $dM_Z$. The program first reads the numbers of incremental loads in one line in order of $dF_X$, $dF_Y$, $dF_Z$, $dM_X$, $dM_Y$, $dM_Z$, and then successively reads the corresponding node numbers and the relative values of incremental loads in the following lines. For the example shown in Fig. 3, the input data are as follows:

| | |
|---|---|
| 'numbers of incremental loads (in order of $dF_X$, $dF_Y$, $dF_Z$, $dM_X$, $dM_Y$, $dM_Z$), and data' | |
| 1, 0, 0, 0, 0, 0 | (numbers of loads $dF_X$, $dF_Y$, $dF_Z$, $dM_X$, $dM_Y$, $dM_Z$) |
| 2, 1 | (node number and relative value of incremental load) |

When all the numbers of loads are zero, the incremental analysis is not performed. However, the program requires the input data of the following Sections from (9) to (11). These input data are free, and the attached input data file 'frame-s-2d-e.txt' can be used without change.

**(9) Displacement to control the frame analysis**

As mentioned in Section (1) of Chapter 4, a numerical analysis by the present program is performed through a displacement-controlled load increment method. Thus, the displacement to control the computation (control-displacement) must be specified by the input data. For the example shown in Fig. 3, the author selected the displacement in the X-axis direction of node 2, which is the same direction as for the incremental load. The direction of the control-displacement is specified using its flexibility number in the degrees-of-freedom table of the whole frame. For example, the flexibility number of the Z-direction of node 1 is 3, and that of the X-direction of node 2 is 8 because the degrees of freedom per node is 7 in the present program. The value of the control-displacement is also needed. A standard value is 1/50 ～1/500 of the elastic limit displacement. When the absolute value of the specified control-displacement is less than $10^{-8}$, a suitable value may be automatically set in the program. The sign of the control-displacement must be consistent with that of the displacement at the first unloading point (see Section (10) of this Chapter). For the example shown in Fig. 3, the input data are as follows:

```
'flexibility number of control-displacement, and its increment'
8, 1E-8                  (flexibility number and displacement increment)
```

**(10) Number of unloading points and displacements at the unloading points**

The present program covers cyclic loading problems, so the number of unloading points and the value of the control-displacement at each unloading point are required. For the example shown in Fig. 3, the number of unloading points is 1 because the loading is monotonic, and the value of the control-displacement at the unloading point was set as 100mm (0.029 rad in story drift angle). The input data are as follows:

```
'number of unloading points and displacements at those points'
1                     (number of unloading points)
100                   (displacement at unloading point)
```

**(11) Loads and displacements for output**

To obtain the load-displacement relationships as the results of analysis, five pairs of flexibility numbers of load and displacement can be assigned. For the example shown in Fig. 3, the author selected the horizontal load *H* (flexibility number is 8) as the load for every pair, and the horizontal and vertical displacements at nodes 2 (flexibility numbers are 8 and 10) and 3 (flexibility numbers are 15 and 17) as the displacement. The input data are as follows:

```
'five pairs of flexibility numbers of load and displacement for output'
8, 8,   8, 15,   8, 10,   8, 17,   8,17     (load-1, disp-1, load-2, disp-2, load-3, disp-3,
                                              load-4, disp-4, load-5, disp-5)
```

The program always requires five pairs of flexibility numbers. If a flexibility number in the input data has a negative sign, e.g. -8, the signs of the corresponding output data are inverted. The output data of load and displacement are written into the file ‘s7.txt’ (see Chapter 7).

## 6. Execution of the FPHM program and output of the results

An execution of the FPHM program is performed using a batch file. A batch file is a text file that contains a series of commands. When a batch file is run, the command processor reads the file and successively executes its commands.

The name of the executable file of the FPHM program is ‘s64.exe’, which is capable of analyzing a frame model with 1200 nodes. Another executable file ‘ms64.exe’ is attached for a compact frame with fewer than 600 nodes. This manual uses ‘ms64.exe’ as the default. Data input to an executable file can be done by a redirection. For example, the attached batch file for static 2D analysis, ‘frame-s-2d.bat’, contains one command as follows:

ms64.exe<frame-s-2d-e.txt

where ‘frame-s-2d-e.txt’ is an input data file. When ‘ms64.exe’ is in a different folder, e.g. C:¥study¥base, the command must be written as follows:

C:¥study¥base¥ms64.exe<frame-s-2d-e.txt

The computation begins by double-clicking ‘frame-s-2d.bat’. Simultaneously, a command prompt window opens to show the status of computation. When the computation is completed or any error is detected during the computation, the command prompt window closes and four output files, ‘s6.txt’, ‘s7.txt’, ‘s8.txt’ and ‘s9.txt’, are created. The contents of each output file are as follows:

s6.txt: confirmation of input data and the results for the initial loads
s7.txt: load-displacement relationships (five pairs of load and displacement)
s8.txt: yielded elements, and yielded nodes and connections
s9.txt: resultant forces of each element at important points of loading

In the following, the contents of the output files from the input data file ‘frame-s-2d-sr-e.txt’, where ‘sr’ indicates the use of the semi-rigid function, are explained with the comments in red. ‘frame-s-2d-sr-e.txt’ is also attached.

### (1) Confirmation of input data and results for initial loads: s6.txt

The comments in red are explanations of the following outputs. The first line of s6.txt is the version number of the FPHM program and the day of update. The first line is followed by comments of the status of the input data file reading. If the reading fails and the execution stops, the user may identify the error in the input data by checking the input data file near the interrupted point.

```
 mini-FPHM v5.10 (C)SHUGYO, M. 3/5/2024

*** FPHM DATA-INPUT STARTED ***
    (GRADIENT OF ELASTIC STRAIN ENERGY)
    INPUT OF NUMBERS OF NODES AND ELEMENTS
    INPUT OF COORDINATES OF EACH NODE
    INPUT OF NODE NUMBERS OF EACH ELEMENT
    INPUT OF NUMBER OF KINDS OF MEMBERS
    INPUT OF SECTION-NUMBER OF EACH ELEMENT
    INPUT OF DATA OF CROSS SECTION-    1
    INPUT OF DATA OF CROSS SECTION-    2
    INPUT OF CHORD ANGLE OF EACH ELEMENT
    INPUT OF DATA OF SEMI-RIGID CONNECTIONS
     I-SECTIONS
     J-SECTIONS
    INPUT OF DATA OF TENSION BRACES
    INPUT OF DATA OF STEEL DAMPERS AND TRUSS MEMBERS
    INPUT OF DATA OF SLAB EQUIVALENT BRACES
    INPUT OF NUMBERS, NODE NOS OF FIXED POINTS
    INPUT OF NUMBERS, NODE NOS AND INITIAL LOADS
    INPUT OF NUMBERS, NODE NOS AND INCRMTL LOADS
    INPUT OF DIRECTION, MAG OF CONTROLLED DISP
    INPUT OF NUMBERS, DISPS OF UNLOADING POINTS
    INPUT OF DRCTIONS OF LOAD AND DISP (4-PAIRS)
*** FPHM DATA-INPUT COMPLETED ***
```

<remodeling number>

NDIV= 1 (one element approximation using the standard model)

-----------------------------------------------

<total number of nodes and total number of elements>

```
 NNOD=    4 NELM=    3
 ------------------------------------------------
```

<node numbers and coordinates>

```
  NODE          X(I)        Y(I)        Z(I)
     1       0.0000D+00  0.0000D+00  0.0000D+00
     2       0.0000D+00  0.0000D+00  0.3500D+04
     3       0.5000D+04  0.0000D+00  0.3500D+04
     4       0.5000D+04  0.0000D+00  0.0000D+00
 ------------------------------------------------
```

<node numbers at both ends, section-number, cross-sectional shape, and cross-sectional constants of each element>

ELM: element number, NODES: node numbers at both ends, SECT-NO: section-number, SHAPE: cross-sectional shape, A: cross-sectional area of steel, YI: cross-sectional secondary moment of steel about the $y_m$-axis, ZI: cross-sectional secondary moment of steel about the $z_m$-axis, AKT: torsional constant, WI: warping constant, RI: higher order cross-sectional constant, CEA: axial stiffness of concrete, CEYI: bending stiffness of concrete about the $y_m$-axis, CEZI: bending stiffness of concrete about the $z_m$-axis.

```
  ELM    NODES   SECT-NO SHAPE     A          YI          ZI          AKT         WI
RI          CEA         CEYI        CEZI
    1     1     2      1      S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    2     2     3      2      H 0.8192D+04 0.2296D+09 0.1735D+08 0.3568D+06 0.6490D+12
0.8692D+13 0.0000D+00 0.0000D+00 0.0000D+00
    3     4     3      1      S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
 ------------------------------------------------
```

<section-number and corresponding data>

NO: section-number, D, B, WT(=WTHK), FT(=FTHK): dimensions of steel, SYSS: yield stress of steel, E: Young's modulus of steel, HARD: post-yield non-dimensional strain hardening coefficient, VARSP: plastic deformation reduction coefficient *r*, CD, CB: dimensions of concrete, CYSS: yield stress of concrete, CE: Young's modulus of concrete, Y0: $y_m$ coordinate of centroid, Z0: $z_m$ coordinate of centroid, PANG: angle measured counterclockwise from the $y_m$-axis to the principal strong axis (rad), NELMR: number of reinforcing bars, RYSS: yield stress of a reinforcing bar, RE: Young's modulus of a reinforcing bar, RU, RV: $y_m$ and $z_m$ coordinates of the centroid of a reinforcing bar, RAREA: cross-sectional area of a reinforcing bar.

```
SECTION-NUMBER AND CORREPONDING DATA
NO=    1 SECTION=S D= 0.3000D+03 B= 0.3000D+03 WT= 0.9000D+01 FT= 0.9000D+01
  SYSS= 0.3300D+03   E= 0.2100D+06 HARD= 0.1000D-01 VARSP= 0.1000D+01
    CD= 0.0000D+00 CB= 0.0000D+00
  CYSS=-0.0000D+00 CE= 0.0000D+00
     Y0          Z0         PANG
 0.0000D+00 0.0000D+00 0.0000D+00
NO=    2 SECTION=H D= 0.4000D+03 B= 0.2000D+03 WT= 0.8000D+01 FT= 0.1300D+02
  SYSS= 0.3260D+03   E= 0.2100D+06 HARD= 0.1000D-01 VARSP= 0.1000D+01
    CD= 0.0000D+00 CB= 0.0000D+00
  CYSS=-0.0000D+00 CE= 0.0000D+00
     Y0          Z0         PANG
 0.0000D+00 0.0000D+00 0.0000D+00
------------------------------------------------
```

<chord angles>

Five chord angles a line are written in order of element number. The left end data indicates the number of elements given in the line.

```
CHORD ANGLES
 ELEMENTS
      3 0.0000D+00 0.0000D+00 0.0000D+00
------------------------------------------------
```

<data of semi-rigid connections>

NISR: number of elements with semi-rigid node i, ELM: element number, ISRST: function number, NJSR: number of elements with semi-rigid node j, JSRST: function number, MCY1-, C1-, etc.: see Section (5) of Chapter 5. In this example, the bending strength 200kNm is set to both element-ends.

```
DATA OF SEMI-RIGID CONNECTIONS
NISR=   1
 ELM=   2
  ISRST=   1
   MCY1-Y-I= 0.0000D+00  MCY2-Y-I= 0.2000D+09
     C1-Y-I= 0.0000D+00    C2-Y-I= 0.1000D-04
```

```
    MCY1-Z-I= 0.0000D+00  MCY2-Z-I= 0.2000D+09
      C1-Z-I= 0.0000D+00    C2-Z-I= 0.1000D-04
 ------------------
 NJSR=    1
  ELM=    2
   JSRST=    1
    MCY1-Y-J= 0.0000D+00  MCY2-Y-J= 0.2000D+09
      C1-Y-J= 0.0000D+00    C2-Y-J= 0.1000D-04
    MCY1-Z-J= 0.0000D+00  MCY2-Z-J= 0.2000D+09
      C1-Z-J= 0.0000D+00    C2-Z-J= 0.1000D-04
 ------------------------------------------------


  NUMBER OF TENSION BRACES
    0
  NUMBER OF STEEL DAMPERS AND TRUSS MEMBERS
    0
  NUMBER OF SLAB EQUIVALENT BRACES
    0
 ------------------------------------------------
```

<numbers of fixed nodes and corresponding node numbers>

NDX, NDY, NDZ, NRZ, NRY, NRZ, NRXX: respective numbers of fixed nodes for *U, V, W*, $\theta_X$, $\theta_Y$, $\theta_Z$, ψ.

```
 NUMBERS OF FIXED NODES
  NDX=    2  NDY=    4  NDZ=    2  NRX=    4  NRY=    2  NRZ=    4  NRXX=    4
  FIXED NODES FOR NDX
    1    4
  FIXED NODES FOR NDY
    1    2    3    4
  FIXED NODES FOR NDZ
    1    4
  FIXED NODES FOR NRX
    1    2    3    4
  FIXED NODES FOR NRY
    1    4
  FIXED NODES FOR NRZ
    1    2    3    4
```

```
 FIXED NODES FOR NRXX
   1    2    3    4
-----------------------------------------------
```

<numbers of initially loaded nodes and actual values of loads>

NFXI, NFYI, NFZI, NMXI, NMYI, NMZI: respective numbers of initially loaded nodes for $F_X$, $F_Y$, $F_Z$, $M_X$, $M_Y$, $M_Z$.

```
INITIAL LOADS (ACTUAL VALUES)
 NFXI=   0  NFYI=   0  NFZI=   2  NMXI=   0  NMYI=   0  NMZI=   0
 NODE       FXI(I)     FYI(I)     FZI(I)     FMXI(I)    FMYI(I)    FMZI(I)
   1      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   2      0.0000D+00 0.0000D+00-0.1040D+07 0.0000D+00 0.0000D+00 0.0000D+00
   3      0.0000D+00 0.0000D+00-0.1040D+07 0.0000D+00 0.0000D+00 0.0000D+00
   4      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
-----------------------------------------------
```

<numbers of incrementally loaded nodes and relative values of incremental loads>

NFX, NFY, NFZ, NMX, NMY, NMZ: respective numbers of incrementally loaded nodes for $dF_X$, $dF_Y$, $dF_Z$, $dM_X$, $dM_Y$, $dM_Z$.

```
INCREMENTAL LOADS (RELATIVE VALUES)
 NFX=   1   NFY=   0   NFZ=   0   NMX=   0   NMY=   0   NMZ=   0
 NODE       FX(I)      FY(I)      FZ(I)      FMX(I)     FMY(I)     FMZ(I)
   1      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   2      0.1000D+01 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   3      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   4      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
-----------------------------------------------
DIRECTION AND VALUE OF CONTROL-DISPLACEMENT
 IDISP=   8 DINC= 0.1000D-07
-----------------------------------------------
DISPLACEMENTS AT UNLOADING POINTS
  0.1000D+03
-----------------------------------------------
DIRECTIONS OF LOAD AND DISPLACEMENT FOR OUTPUT
 L1=    8  D1=    8
 L2=    8  D2=   15
```

```
 L3=     8   D3=    10
 L4=     8   D4=    17
 L5=     8   D5=    17
------------------------------------------------
(RESIDUAL STRESSES IN H-SECTION NOT CONSIDERED)
------------------------------------------------
```

From here, the results for the initial loading are written.

```
================================================
RESULTS FOR INITIAL LOADING
DISPLACEMENTS
NODE         U           V           W        THETAX      THETAY      THETAZ
DTHETAX
    1     0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00
0.0000D+00
    2    -0.1952D-16  0.0000D+00 -0.1654D+01  0.0000D+00 -0.3586D-20  0.0000D+00
0.0000D+00
    3     0.1150D-16  0.0000D+00 -0.1654D+01  0.0000D+00  0.6461D-20  0.0000D+00
0.0000D+00
    4     0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00
0.0000D+00
RESULTANT FORCES
ELEM NODE     RX          RY          RZ          MX          MY          MZ
MOMEGA
    1    1   0.1040D+07  0.1594D-46 -0.1088D-12  0.1678D-64  0.2312D-09  0.2789D-43
0.0000D+00
    1    2  -0.1040D+07 -0.1594D-46  0.1088D-12 -0.1678D-64  0.1697D-09  0.2789D-43
0.0000D+00
    2    2  -0.7590D-32 -0.5043-308 -0.3327D-13  0.0000D+00 -0.1372D-10 -0.8405-305 -
0.1334-321
    2    3   0.7590D-32  0.5043-308  0.3327D-13  0.0000D+00  0.1801D-09 -0.1681-304
0.4842-321
    3    4   0.1040D+07  0.1594D-46 -0.9777D-13 -0.3024D-64  0.1156D-09  0.2789D-43
0.0000D+00
    3    3  -0.1040D+07 -0.1594D-46  0.9777D-13  0.3024D-64  0.2264D-09  0.2789D-43
0.0000D+00
================================================
```

```
COMPUTATION COMPLETED
```

( The comment means the computation has successfully finished. )

<state of each member, tension or compression, after the computation>

ELM: element number, SECT-NO: section-number, SHAPE: cross-sectional shape, T: tension, C: compression.

```
ELM SECT-NO SHAPE TEN-OR-COMP
 1      1     S       C
 2      2     H       C
 3      1     S       C
```

**(2) Load-displacement relationships: s7.txt**

Five pairs of load and displacement for the flexibility numbers assigned in the input data file are written. L means load, D means displacement, and the values in parentheses are the flexibility numbers.

```
   L(     8)    D(     8)    L(     8)    D(    15)    L(     8)    D(    10)
L(     8)    D(    17)    L(     8)    D(    17)
   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
   0.1164D-03   0.1000D-07   0.1164D-03   0.9832D-08   0.1164D-03  -0.1654D+01
0.1164D-03  -0.1654D+01   0.1164D-03  -0.1654D+01
   0.8197D-01   0.7044D-05   0.8197D-01   0.6926D-05   0.8197D-01  -0.1654D+01
0.8197D-01  -0.1654D+01   0.8197D-01  -0.1654D+01
   0.1088D+03   0.9347D-02   0.1088D+03   0.9190D-02   0.1088D+03  -0.1654D+01
0.1088D+03  -0.1654D+01   0.1088D+03  -0.1654D+01
   0.3223D+04   0.2770D+00   0.3223D+04   0.2724D+00   0.3223D+04  -0.1652D+01
0.3223D+04  -0.1656D+01   0.3223D+04  -0.1656D+01
   0.6337D+04   0.5446D+00   0.6337D+04   0.5354D+00   0.6337D+04  -0.1651D+01
0.6337D+04  -0.1657D+01   0.6337D+04  -0.1657D+01
   0.9450D+04   0.8122D+00   0.9450D+04   0.7985D+00   0.9450D+04  -0.1649D+01
0.9450D+04  -0.1659D+01   0.9450D+04  -0.1659D+01
```

--------- the rest is omitted -----------

**(3) Yielded elements: s8.txt**

When a new yielding of element-end or semi-rigid connection is detected, the following are written: the number of steps of incremental calculation, five pairs of load and displacement for the flexibility numbers specified in the input data file, and node-number and element-number of yielded elements. The data for these parameters at the end of the analysis are also written.

```
  STEP       LOAD-1       DISP-1       LOAD-2       DISP-2       LOAD-3       DISP-3
LOAD-4       DISP-4       LOAD-5       DISP-5
     85     0.2422D+06  0.2168D+02  0.2422D+06  0.2133D+02  0.2422D+06 -0.1655D+01
0.2422D+06 -0.1941D+01  0.2422D+06 -0.1941D+01
  NUMBER OF YIELDED POINTS=     1              (*1)
   CONNECTION     2 OF ELM     2 YIELDED.     (*2)

  STEP       LOAD-1       DISP-1       LOAD-2       DISP-2       LOAD-3       DISP-3
LOAD-4       DISP-4       LOAD-5       DISP-5
     87     0.2439D+06  0.2222D+02  0.2439D+06  0.2186D+02  0.2439D+06 -0.1669D+01
0.2439D+06 -0.1963D+01  0.2439D+06 -0.1963D+01
  NUMBER OF YIELDED POINTS=     2
   CONNECTION     2 OF ELM     2 YIELDED.
   CONNECTION     3 OF ELM     2 YIELDED.

  STEP       LOAD-1       DISP-1       LOAD-2       DISP-2       LOAD-3       DISP-3
LOAD-4       DISP-4       LOAD-5       DISP-5
    118     0.2505D+06  0.3052D+02  0.2505D+06  0.3015D+02  0.2505D+06 -0.1939D+01
0.2505D+06 -0.2243D+01  0.2505D+06 -0.2243D+01
  NUMBER OF YIELDED POINTS=     3
   CONNECTION     2 OF ELM     2 YIELDED.
   CONNECTION     3 OF ELM     2 YIELDED.
   NODE     1 OF ELM     1 YIELDED.            (*3)

  STEP       LOAD-1       DISP-1       LOAD-2       DISP-2       LOAD-3       DISP-3
LOAD-4       DISP-4       LOAD-5       DISP-5
    120     0.2508D+06  0.3105D+02  0.2508D+06  0.3069D+02  0.2508D+06 -0.1959D+01
0.2508D+06 -0.2263D+01  0.2508D+06 -0.2263D+01
  NUMBER OF YIELDED POINTS=     4
   CONNECTION     2 OF ELM     2 YIELDED.
   CONNECTION     3 OF ELM     2 YIELDED.
   NODE     1 OF ELM     1 YIELDED.
   NODE     4 OF ELM     3 YIELDED.
```

```
  STEP      LOAD-1      DISP-1      LOAD-2      DISP-2      LOAD-3      DISP-3
LOAD-4      DISP-4      LOAD-5      DISP-5
    378    0.2456D+06  0.9982D+02  0.2456D+06  0.9947D+02  0.2456D+06 -0.5675D+01
0.2456D+06 -0.6224D+01  0.2456D+06 -0.6224D+01
  NUMBER OF YIELDED POINTS=    4
   CONNECTION    2 OF ELM    2 YIELDED.
   CONNECTION    3 OF ELM    2 YIELDED.
   NODE    1 OF ELM    1 YIELDED.
   NODE    4 OF ELM    3 YIELDED.
```

(*1) Total number of yielded nodes and semi-rigid connections up to the current moment.
(*2) The bending moment in the semi-rigid connection on node-2 of element-2 has reached its strength.
(*3) Node-1 of element-1 has yielded.

**(4) Resultant forces of each element: s9.txt**

When a new yielding of element-end or semi-rigid connection is detected, the following are written: the number of steps of incremental calculation, five pairs of load and displacement for the flexibility numbers assigned in the input data file, the total number of yielded nodes and semi-rigid connections, and the resultant forces of all the elements. The data for these parameters at the end of the analysis are also written.

RX: axial force, RY: shearing force in the $y_m$-axis direction, RZ: shearing force in the $z_m$-axis direction, MX: torsional moment about the $x_m$-axis, MY: bending moment about the $y_m$-axis, MZ: bending moment about the $z_m$-axis, MOMEGA: warping moment.

```
  STEP      LOAD-1      DISP-1      LOAD-2      DISP-2      LOAD-3      DISP-3
LOAD-4      DISP-4      LOAD-5      DISP-5
     85    0.2422D+06  0.2168D+02  0.2422D+06  0.2133D+02  0.2422D+06 -0.1655D+01
0.2422D+06 -0.1941D+01  0.2422D+06 -0.1941D+01
  NUMBER OF YIELDED POINTS=    1
  RESULTANT FORCES
  ELEM NODE    RX          RY          RZ          MX          MY          MZ
MOMEGA
    1    1  0.9595D+06  0.1919D-11  0.1294D+06  0.7897D-11 -0.2523D+09  0.5540D-08
0.0000D+00
    1    2 -0.9595D+06 -0.1919D-11 -0.1294D+06 -0.7897D-11 -0.2005D+09  0.1176D-08
```

```
0.0000D+00
    2     2   0.1187D+06   0.0000D+00  -0.7971D+05   0.0000D+00   0.2005D+09   0.0000D+00
0.0000D+00
    2     3  -0.1187D+06   0.0000D+00   0.7971D+05   0.0000D+00   0.1980D+09   0.0000D+00
0.0000D+00
    3     4   0.1119D+07   0.2933D-11   0.1255D+06   0.1203D-10  -0.2411D+09   0.8450D-08
0.0000D+00
    3     3  -0.1119D+07  -0.2933D-11  -0.1255D+06  -0.1203D-10  -0.1980D+09   0.1811D-08
0.0000D+00
```

**---------** the rest is omitted **-----------**

### (5) Resultant forces of each element at the end of analysis: s9.txt

As mentioned above, the results for the last state of the analysis are always written in the output files. This makes it possible to get the results for an interesting point in the load-displacement relationship, if necessary, by a recomputation.

# 7. How to use the FPHM-D program

## 7.1 Outline of the FPHM-D program

The FPHM-D program is an analysis code for dynamic 3D elastoplastic large deflection analysis of frames in which the restoring force characteristics of frames are estimated by the FPHM program. A 3D seismic response analysis of frames containing steel members, RC members, SRC members, CFT members, PC members, composite beams, tension braces and steel damper braces can be performed.

The masses of members, slabs and attachments are lumped to beam-column connection points in principle for dynamic analysis, while the restoring forces of the frame are estimated by a displacement increment analysis of the whole frame using the FPHM program. The degrees of freedom of each mass is assumed to be 3 and the components are two horizontal translations in the X-axis and Y-axis directions and a vertical translation in the Z-axis direction. The influences of in-plane stiffness and bending stiffness of the RC slab are considered by replacing the former with a slab equivalent brace element (see Section (5) of Chapter 5) and by using a composite beam element for the latter. Story displacement is calculated as a mean value of displacements of four representative masses near the gravity center of the story.

## 7.2 Basic form of input data file for the FPHM-D program and examples

### (1) Preparation of input data

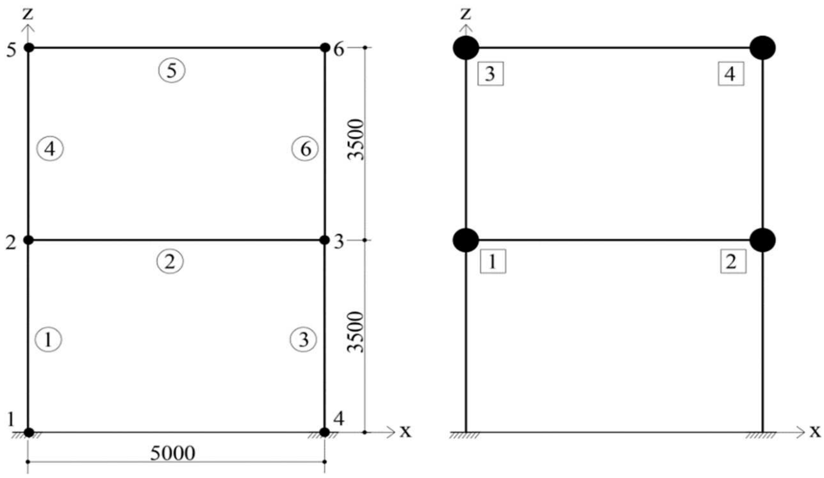
The one-bay two-story plane portal frame shown in Fig. 13 is analyzed to show the basic form of the input data file. The frame consists of the same four square steel pipe columns and the same two H-shaped steel beams. The square steel pipe is □-300×300×9, and the H-shaped steel is H-400×200×8×13 (see Fig. 4 in Chapter 4). It is assumed that the quantities of the four lumped masses are the same.

The key points of the numerical modeling of a frame for dynamic analysis are as follows:

1) The masses of members, slabs and attachments are lumped to beam-column connection points in principle. The degrees of freedom of each mass is 3 and the components are two horizontal translations in the X-axis and Y-axis directions and a vertical translation in the Z-axis direction.

2) Story displacement is calculated as the mean value of displacements of four representative masses near the gravity center of the story.

3) To save computation time, the use of the modified model is prioritized.




(a) Node numbers and element numbers (b) Mass numbers

Fig. 13 Global coordinate axes and sizes of the frame

The following explains how to prepare the input data file for the FPHM-D program. Note that the FPHM-D program only supports the (mm, N, s) unit system.

1) Divide the frame into elements with the nodes, which are also the mass points, at member connection points, and other nodes for necessary mass points. Then, make the data file ‘fd.txt’ according to the descriptions from Section (1) to Section (7) of Chapter 5 using the modified model (see Section (2) of Chapter 4). Consequently, the numbers of nodes and elements may be determined as shown in Fig. 13 (a), and the global stiffness matrix of the frame is obtained. The damping coefficient matrix of the frame is estimated by using the components of this stiffness matrix.

2) Based on the aforementioned data file ‘fd.txt’, another data file ‘fr.txt’ can be made by adding nodes for the initial loading points except the mass point and three inner nodes of every compression brace element that may buckle. The restoring force characteristics are estimated by a displacement increment analysis of the whole frame using the data of ‘fr.txt’. If any additional node is not necessary, the contents of both ‘fd.txt’ and ‘fr.txt’ are the same. When precise results are needed, the use of the standard model with a remodeling number of 4 (see Section (1) of Chapter 4) can be specified, provided a considerable increase of computation time is acceptable.

3) The input data file for the FPHM-D program contains the data files ‘fd.txt’ and ‘fr.txt’, and begins with the data on dynamic behaviors of the frame. First, a mass numbering must be done as shown in Fig. 13 (b).

The following data are needed for dynamic analysis of a frame.

(a) Number of masses
(b) Mass number and the corresponding node number
(c) Mass number and magnitude of mass (N/(mm/s$^2$))
(d) Direction of vibration
(e) Two natural frequencies and two damping constants
(f) Initial displacements in the X-, Y-, and Z-axis directions of each mass (mm)
(g) Initial velocities in the X-, Y-, and Z-axis directions of each mass (mm/s)
(h) Initial accelerations in the X-,Y-, and Z-axis directions of each mass (mm/s$^2$)
(i) The maximum accelerations of earthquake motion in the X-, Y-, and Z-axis directions, time increment (sec), and time range (sec)
(j) Number of stories
(k) Story number and four representative mass numbers of each story
(l) Story number and story height (mm) of each story
(m) Story number and story mass (N/(mm/s$^2$)) of each story
(n) Five story numbers for output of results

Some additional detailed explanations concerned mainly with the present example are given below.

(c) Mass number and magnitude of mass (N/(mm/s$^2$))

In the present example, the initial downward axial force of each column of the first story is assumed as the same value of 693,000N, one fifth of the yield axial force of the column. Since the axial force of the column is caused by a share of weight of the whole frame, the magnitude of each mass is half of 70.7, 693,000/9,800, considering the magnitudes of the four masses are the same. Consequently, the magnitude of each mass is 35.4 (N/(mm/s$^2$))

(d) Direction of vibration

The direction of vibration is specified by a character constant, 'X', 'Y', 'B' or 'T'. 'B' means biaxial vibration, and 'T' means triaxial vibration. Vertical vibration can only be treated by the specification of 'T'.

(e) Two natural frequencies $f_1$(Hz), $f_2$(Hz) and two damping constants $h_1$, $h_2$.

These are the data for estimating a damping coefficient matrix. A method to identify the natural frequencies is presented in Section (2) of this Chapter. FPHM-D judges the type of damping by the sign of $h_2$. Positive $h_2$ indicates Rayleigh damping, zero $h_2$ indicates mass proportional damping, and negative $h_2$ indicates initial-stiffness proportional damping. The present example assumes Rayleigh damping with the values $h_1$=0.03 and $h_2$=0.03.

(i) The maximum accelerations of earthquake motion in the X-, Y-, and Z-axis directions, time increment (sec), and time range (sec)

The dynamic response analysis is performed under available earthquake acceleration records in general. The first three data force the maximum accelerations in the X-, Y-, and Z-axis directions to be the specified values. Because the present example is an in-plane vibration in the X-axis direction, the values are set as 4000(mm/s$^2$), 0, 0, respectively. Regarding the time increment, a value from 0.001 to 0.002 is suitable for building frames.

------

The input data file for the present example is constructed by adding the data of files 'fd.txt' and 'fr.txt' to the aforementioned data for dynamic analysis. The contents of the input data file are as follows. The accompanying comment lines within quotation marks are required. The comments in red are points to be noted. The input data file is named 'frame-d-2d-mod-e.txt' and attached, where 'd', '2d', and 'mod' indicate a dynamic 2D analysis using the modified model.

-------- from here --------

```
'number of masses'
4
'mass number and the corresponding node number'
1, 2
2, 3
3, 5
4, 6
'mass number and magnitude of mass (N/(mm/s2))'
1, 35.4
2, 35.4
3, 35.4
4, 35.4
'direction of vibration'
'X'
'two natural frequencies(Hz) and two damping constants'
1.1, 3.6, 0.03, 0.03
'initial displacements in the X-axis direction(mm)'
0, 0, 0, 0
'initial displacements in the Y-axis direction(mm)'
0, 0, 0, 0
'initial displacements in the Z-axis direction(mm)'
0, 0, 0, 0
'initial velocities in the X-axis direction(mm/s)'
```

```
0, 0, 0, 0
'initial velocities in the Y-axis direction(mm/s)'
0, 0, 0, 0
'initial velocities in the Z-axis direction(mm/s)'
0, 0, 0, 0
'initial accelerations in the X-axis direction(mm/s2)'
0, 0, 0, 0
'initial accelerations in the Y-axis direction(mm/s2)'
0, 0, 0, 0
'initial accelerations in the Z-axis direction(mm/s2)'
0, 0, 0, 0
'maximum accelerations in the X-, Y-, and Z-axis directions, time inc. and range'
4000, 0, 0, 0.002, 20
'number of stories'
2
'story number and four representative mass numbers of each story'
1, 1, 1, 2, 2   ( The same mass number can be used. )
2, 3, 3, 4, 4
'story number and story height (mm) of each story'
1, 3500
2, 3500
'story number and story mass (N/(mm/s2)) of each story'
1, 70.7
2, 70.7
'five story numbers for output of results'
1, 2, 2, 2, 2
   ( The data from here are those of 'fd.txt' .)
'remodeling number (5 indicates modified model)'
5  ( The number must be 1 or 5.)
'numbers of nodes and elements'
6, 6
'node number and coordinates'
1, 0, 0, 0
2, 0, 0, 3500
3, 5000, 0, 3500
4, 5000, 0, 0
5, 0, 0, 7000
6, 5000, 0, 7000
```

```
'element number and two node numbers at element-ends'
1, 1, 2
2, 2, 3
3, 4, 3
4, 2, 5
5, 5, 6
6, 3, 6
'number of kinds of members'
2
'section-number of each element (in order of element number)'
1, 2, 1, 1, 2, 1
'data of cross section 1'
'S', 300, 300, 9, 9, 330, 210000, 0.01, 1.0
0, 0, 0, 0
0, 0, 0
0, 0, 0
'data of cross section 2'
'H', 400, 200, 8, 13, 326, 210000, 0.01, 1.0
0, 0, 0, 0
0, 0, 0
0, 0, 0
'chord angle of each element (in order of element number)'
0, 0, 0, 0, 0, 0
'number of elements with semi-rigid node i'
0
'number of elements with semi-rigid node j'
0
'number of tension brace elements'
0
'number of steel damper brace and truss elements, and element numbers'
0
'number of slab equivalent brace elements and element numbers'
0
'numbers of fixed nodes (in order of U, V, W, θX, θY, θZ, ψ), and node numbers'
2, -1, 2, -1, 2, -1, -1
1, 4
1, 4
1, 4
```

'numbers of initial loads (in order of $F_X$, $F_Y$, $F_Z$, $M_X$, $M_Y$, $M_Z$), and data'

0, 0, 4, 0, 0, 0

2, -346500 ( one half of 693,000 (N) )

3, -346500

5, -346500

6, -346500

( The data from here are those of 'fr.txt'. In the present example, the data in 'fd.txt' and 'fr.txt' are the same. )

'remodeling number (5 indicates modified model)'

5 ( If this value is 4, this indicates the four element approximation with the standard model. )

'numbers of nodes and elements'

6, 6

'node number and coordinates'

1, 0, 0, 0

2, 0, 0, 3500

3, 5000, 0, 3500

4, 5000, 0, 0

5, 0, 0, 7000

6, 5000, 0, 7000

'element number and two node numbers at element-ends'

1, 1, 2

2, 2, 3

3, 4, 3

4, 2, 5

5, 5, 6

6, 3, 6

'number of kinds of members'

2

'section-number of each element (in order of element number)'

1, 2, 1, 1, 2, 1

'data of cross section 1'

'S', 300, 300, 9, 9, 330, 210000, 0.01, 0.3

0, 0, 0, 0

0, 0, 0

0, 0, 0

'data of cross section 2'

'H', 400, 200, 8, 13, 326, 210000, 0.01, 0.3

0, 0, 0, 0

```
0, 0, 0
0, 0, 0
'chord angle of each element (in order of element number)'
0, 0, 0, 0, 0, 0
'number of elements with semi-rigid node i'
0
'number of elements with semi-rigid node j'
0
'number of tension brace elements'
0
'number of steel damper brace and truss elements, and element numbers'
0
'number of slab equivalent brace elements and element numbers'
0
'numbers of fixed nodes (in order of U, V, W, θX, θY, θZ, ψ), and node numbers'
2, -1, 2, -1, 2, -1, -1
1, 4
1, 4
1, 4
'numbers of initial loads (in order of FX, FY, FZ, MX, MY, MZ), and data'
0, 0, 4, 0, 0, 0
2, -346500
3, -346500
5, -346500
6, -346500
```
-------- to here --------

### (2) Calculation of natural frequencies of frame

As mentioned in the previous section, it is necessary to identify two natural frequencies of a frame, generally the first and the second ones, to estimate a damping coefficient matrix of the frame. The author usually obtains these natural frequencies by utilizing a Fourier transformation of the displacement time history from an elastic free vibration analysis of the frame. The input data file for elastic free vibration analysis 'frame-d-2d-free-e.txt' can be obtained by the following revision of 'frame-d-2d-mod-e.txt'.

1) Change the values for the damping constants, the maximum accelerations of motion, and the initial loads to zero.

2) Set a small initial acceleration, e.g. 200mm/s$^2$, to every mass belonging to the top floor of the frame, and set the time range as greater than 20 sec.

A simple executable file ‘ft64.exe’ is attached for Fourier transformation of the output file ‘s7.txt’ made from the elastic free vibration analysis. Double clicking the attached batch file ‘ft64.bat’, the displacement response spectrum data in the X-, Y-, Z-axis directions are written in the output files ‘fsx.txt’, ‘fsy.txt’, and ‘fsz.txt’, respectively. The first column of those files is frequency (Hz) and the second column is relative response of displacement. The frequencies corresponding to the peak of relative displacement are the natural frequencies.

**(3) Preparation of ground acceleration data file**

Usually, ground acceleration data are provided with the time increment and the unit of data. The FPHM-D program assumes the ground acceleration data in three directions are written in the files ‘x.acc’, ‘y.acc’, and ‘z.acc’, respectively, and expects the following two lines are at the top of each file. These lines set the time increment and the unit of data.

```
'time increment (sec)=', 0.02
'unit ([-m] or [cm] or [mm])=', '-m'
```

The character constant ‘-m’ in the second line means the unit of data is m/s$^2$. Some ground acceleration data files made by using downloaded data from the site below are attached to demonstrate the file style.

The Earthquake Engineering Online Archive, http://nisee.berkeley.edu/

The names of the attached files are ‘el-ew.acc’, ‘el-ns.acc’, and ‘el-ud.acc’. These are acceleration records in three directions for the El Centro earthquake. The file name is free, but it is necessary to write a command such as ‘copy el-ew.acc x.acc’ at the top of the batch file for execution.

### 7.3 Execution of the FPHM-D program and output of the results

The FPHM-D program is similarly executed using a batch file (see Chapter 6). The name of the executable file of the FPHM-D program is ‘d64.exe’, which is capable of analyzing a frame model with 1200 nodes. Another executable file ‘md64.exe’ is also attached for a compact frame with fewer than 600 nodes. This manual uses ‘md64.exe’ as the default. Data input to an executable file can be done by a redirection. For example, the attached batch file for dynamic 2D analysis of a frame ‘frame-d-2d.bat’ contains one command as follows:

md64.exe<frame-d-2d-mod-e.txt

where ‘frame-d-2d-mod-e.txt’ is an input data file. When ‘md64.exe’ is in a different folder, e.g. C:¥study¥base, the command must be written as follows:

C:¥study¥base¥md64.exe<frame-d-2d-mod-e.txt

The computation begins by double-clicking ‘frame-d-2d.bat’. Simultaneously, a command prompt window opens to show the status of computation. When the computation is completed or any error is detected during the computation, the command prompt window closes and five output files, ‘d6.txt’, ‘d7.txt’, ‘d8.txt’, ‘d9.txt’ and ‘d10.txt’ are created. The contents of each output file are as follows:

d6.txt: confirmation of the input data
d7.txt: time histories of story shearing forces and story drift angles of the first assigned two stories
d10.txt: time histories of story shearing forces and story drift angles of the remaining three stories
d8.txt: yielded elements and current coordinates of each node at every 200 steps, and the maximum values of displacement, drift angle, acceleration, and shearing force of each story up to the current moment
d9.txt: resultant forces of each element at every 200 steps

In the following, the contents of the output files from the input data file ‘frame-d-2d-mod-e.txt’, where ‘d’, ‘2d’, and ‘mod’ mean dynamic 2D analysis using the modified model, are explained in the red comments. The unit system is (mm, N, s), so the unit of mass is N/(mm/s$^2$).

**(1) Confirmation of input data: d6.txt**

```
 mini-FPHM-D v5.10 (C)SHUGYO, M. 3/5/2024

*** DATA-INPUT FOR DYNAMIC ANALYSIS STARTED ***

------------------------------------------------

 NUMBER OF MASSES
     4
------------------------------------------------

 NODE NUMBERS OF MASS-POINTS
 MASS-NO.  NODE-NO.
     1          2
     2          3
     3          5
     4          6
------------------------------------------------

 MAGNITUDES OF MASSES
 MASS-NOS
     4 0.3540D+02 0.3540D+02 0.3540D+02 0.3540D+02
------------------------------------------------

 DIRECTIONS OF MOTION
  UNI-AXIAL(X)
------------------------------------------------

 TWO NATURAL FREQUENCIES AND TWO DAMPING FACTORS
    H2>0 : RAYLEIGH DAMPING
    H2=0 : MASS PROPORTIONAL DAMPING AND
    H2<0 : INITIAL STIFFNESS PROPORTIONAL DAMPING
  F1=  0.1100D+01 F2=  0.3600D+01 H1=  0.3000D-01 H2=  0.3000D-01
------------------------------------------------

 INITIAL DISPLACEMENTS (X-DIRECTION)
 MASS-NOS
```

```
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
 INITIAL DISPLACEMENTS (Y-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
 INITIAL DISPLACEMENTS (Z-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
-------------------------------------------------

 INITIAL VELOCITIES (X-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
 INITIAL VELOCITIES (Y-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
 INITIAL VELOCITIES (Z-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
-------------------------------------------------

 INITIAL ACCELERATIONS (X-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
 INITIAL ACCELERATIONS (Y-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
 INITIAL ACCELERATIONS (Z-DIRECTION)
 MASS-NOS
    4 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
-------------------------------------------------

 MAX-ACC-X   MAX-ACC-Y   MAX-ACC-Z     DELTT       TEND
 0.4000D+04  0.0000D+00  0.0000D+00  0.2000D-02  0.2000D+02
-------------------------------------------------

 FOUR REPRESENTATIVE MASS NUMBERS OF EACH STORY
 NUMBER OF STORIES=   2
   1 STORY :    1    1    2    2
```

```
    2 STORY :    3    3    4    4
 ------------------------------------------------

  HEIGHT OF EACH STORY
    1 STORY :  0.3500D+04
    2 STORY :  0.3500D+04
 ------------------------------------------------

  MASS OF EACH STORY
    1 STORY :  0.7070D+02
    2 STORY :  0.7070D+02
 ------------------------------------------------

  FIVE STORY NUMBERS FOR OUTPUT OF THE RESULTS
    1    2    2    2    2
 ------------------------------------------------

*** DATA-INPUT FOR DYNAMIC ANALYSIS COMPLETED ***

*** ESTIMATION OF C-MATRIX USING ONE ELEMENT MODEL FRAME STARTED ***

*** FPHM DATA-INPUT STARTED ***
    (GRADIENT OF ELASTIC STRAIN ENERGY)
    INPUT OF NUMBERS OF NODES AND ELEMENTS
    INPUT OF COORDINATES OF EACH NODE
    INPUT OF NODE NUMBERS OF EACH ELEMENT
    INPUT OF NUMBER OF KINDS OF MEMBERS
    INPUT OF SECTION-NUMBER OF EACH ELEMENT
    INPUT OF DATA OF CROSS SECTION-   1
    INPUT OF DATA OF CROSS SECTION-   2
    INPUT OF CHORD ANGLE OF EACH ELEMENT
    INPUT OF DATA OF SEMI-RIGID CONNECTIONS
     I-SECTIONS
     J-SECTIONS
    INPUT OF DATA OF TENSION BRACES
    INPUT OF DATA OF STEEL DAMPERS AND TRUSS MEMBERS
    INPUT OF DATA OF SLAB EQUIVALENT BRACES
    INPUT OF NUMBERS, NODE NOS OF FIXED POINTS
```

```
    INPUT OF NUMBERS, NODE NOS AND INITIAL LOADS
 *** FPHM DATA-INPUT COMPLETED ***

  ------------------------------------------------

  NDIV=    5
  ------------------------------------------------

  NNOD=    6 NELM=    6
  ------------------------------------------------

   NODE         X(I)       Y(I)       Z(I)
      1       0.0000D+00 0.0000D+00 0.0000D+00
      2       0.0000D+00 0.0000D+00 0.3500D+04
      3       0.5000D+04 0.0000D+00 0.3500D+04
      4       0.5000D+04 0.0000D+00 0.0000D+00
      5       0.0000D+00 0.0000D+00 0.7000D+04
      6       0.5000D+04 0.0000D+00 0.7000D+04
  ------------------------------------------------

   ELM    NODES   SECT-NO SHAPE     A           YI           ZI           AKT          WI
RI          CEA         CEYI        CEZI
    1     1     2       1       S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    2     2     3       2       H 0.8192D+04 0.2296D+09 0.1735D+08 0.3568D+06 0.6490D+12
0.8692D+13 0.0000D+00 0.0000D+00 0.0000D+00
    3     4     3       1       S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    4     2     5       1       S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    5     5     6       2       H 0.8192D+04 0.2296D+09 0.1735D+08 0.3568D+06 0.6490D+12
0.8692D+13 0.0000D+00 0.0000D+00 0.0000D+00
    6     3     6       1       S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
  ------------------------------------------------

  SECTION-NUMBER AND CORRESPONDING DATA
  NO=    1 SECTION=S D= 0.3000D+03 B= 0.3000D+03 WT= 0.9000D+01 FT= 0.9000D+01
```

```
  SYSS= 0.3300D+03  E= 0.2100D+06 HARD= 0.1000D-01 VARSP= 0.1000D+01
    CD= 0.0000D+00 CB= 0.0000D+00
  CYSS=-0.0000D+00 CE= 0.0000D+00
      Y0          Z0         PANG
 0.0000D+00 0.0000D+00 0.0000D+00
NO=   2 SECTION=H D= 0.4000D+03 B= 0.2000D+03 WT= 0.8000D+01 FT= 0.1300D+02
  SYSS= 0.3260D+03  E= 0.2100D+06 HARD= 0.1000D-01 VARSP= 0.1000D+01
    CD= 0.0000D+00 CB= 0.0000D+00
  CYSS=-0.0000D+00 CE= 0.0000D+00
      Y0          Z0         PANG
 0.0000D+00 0.0000D+00 0.0000D+00
------------------------------------------------

CHORD ANGLES
 ELEMENTS
      5 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
      6 0.0000D+00
------------------------------------------------

DATA OF SEMI-RIGID CONNECTIONS
NISR=   0
------------------

NJSR=   0
------------------------------------------------

NUMBER OF TENSION BRACES
  0
NUMBER OF STEEL DAMPERS AND TRUSS MEMBERS
  0
NUMBER OF SLAB EQUIVALANT BRACES
  0
------------------------------------------------

NUMBERS OF FIXED NODES
 NDX=   2  NDY=   6  NDZ=   2  NRX=   6  NRY=   2  NRZ=   6  NRXX=   6
 FIXED NODES FOR NDX
   1    4
```

```
 FIXED NODES FOR NDY
   1    2    3    4    5    6
 FIXED NODES FOR NDZ
   1    4
 FIXED NODES FOR NRX
   1    2    3    4    5    6
 FIXED NODES FOR NRY
   1    4
 FIXED NODES FOR NRZ
   1    2    3    4    5    6
 FIXED NODES FOR NRXX
   1    2    3    4    5    6
------------------------------------------------

INITIAL LOADS (ACTUAL VALUES)
 NFXI=   0  NFYI=   0  NFZI=   4  NMXI=   0  NMYI=   0  NMZI=   0
 NODE       FXI(I)     FYI(I)     FZI(I)    FMXI(I)    FMYI(I)    FMZI(I)
   1      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   2      0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
   3      0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
   4      0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   5      0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
   6      0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
------------------------------------------------

 (RESIDUAL STRESSES IN H-SECTION NOT CONSIDERED)
------------------------------------------------

 DAMPING MATRIX (DIAGONAL COMPONENTS)
 0.7456D+03  0.4730D+02  0.0000D+00  0.7456D+03  0.4730D+02  0.0000D+00
 0.7280D+03  0.2963D+02  0.0000D+00  0.7280D+03  0.2963D+02  0.0000D+00

*** ESTIMATION OF C-MATRIX COMPLETED ***

*** INITIALIZATION OF THE FRAME FOR RESTORING FORCE ANALYSIS ***

*** FPHM DATA-INPUT STARTED ***
    (GRADIENT OF ELASTIC STRAIN ENERGY)
```

```
    INPUT OF NUMBERS OF NODES AND ELEMENTS
    INPUT OF COORDINATES OF EACH NODE
    INPUT OF NODE NUMBERS OF EACH ELEMENT
    INPUT OF NUMBER OF KINDS OF MEMBERS
    INPUT OF SECTION-NUMBER OF EACH ELEMENT
    INPUT OF DATA OF CROSS SECTION-    1
    INPUT OF DATA OF CROSS SECTION-    2
    INPUT OF CHORD ANGLE OF EACH ELEMENT
    INPUT OF DATA OF SEMI-RIGID CONNECTIONS
     I-SECTIONS
     J-SECTIONS
    INPUT OF DATA OF TENSION BRACES
    INPUT OF DATA OF STEEL DAMPERS AND TRUSS MEMBERS
    INPUT OF DATA OF SLAB EQUIVALENT BRACES
    INPUT OF NUMBERS, NODE NOS OF FIXED POINTS
    INPUT OF NUMBERS, NODE NOS AND INITIAL LOADS
*** FPHM DATA-INPUT COMPLETED ***

 ------------------------------------------------

 NDIV=    5      ( NDIV=5 indicates the use of the modified model )
 ------------------------------------------------

 NNOD=    6 NELM=    6
 ------------------------------------------------

  NODE         X(I)       Y(I)       Z(I)
    1       0.0000D+00 0.0000D+00 0.0000D+00
    2       0.0000D+00 0.0000D+00 0.3500D+04
    3       0.5000D+04 0.0000D+00 0.3500D+04
    4       0.5000D+04 0.0000D+00 0.0000D+00
    5       0.0000D+00 0.0000D+00 0.7000D+04
    6       0.5000D+04 0.0000D+00 0.7000D+04
 ------------------------------------------------

  ELM    NODES   SECT-NO  SHAPE     A           YI           ZI           AKT          WI
RI          CEA         CEYI        CEZI
    1     1     2       1      S  0.1048D+05  0.1480D+09  0.1480D+09  0.2218D+09  0.0000D+00
```

```
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    2     2     3      2      H 0.8192D+04 0.2296D+09 0.1735D+08 0.3568D+06 0.6490D+12
0.8692D+13 0.0000D+00 0.0000D+00 0.0000D+00
    3     4     3      1      S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    4     2     5      1      S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
    5     5     6      2      H 0.8192D+04 0.2296D+09 0.1735D+08 0.3568D+06 0.6490D+12
0.8692D+13 0.0000D+00 0.0000D+00 0.0000D+00
    6     3     6      1      S 0.1048D+05 0.1480D+09 0.1480D+09 0.2218D+09 0.0000D+00
0.8764D+13 0.0000D+00 0.0000D+00 0.0000D+00
  ------------------------------------------------

  SECTION-NUMBER AND CORRESPONDING DATA
  NO=   1 SECTION=S D= 0.3000D+03 B= 0.3000D+03 WT= 0.9000D+01 FT= 0.9000D+01
    SYSS= 0.3300D+03  E= 0.2100D+06 HARD= 0.1000D-01 VARSP= 0.3000D+00
      CD= 0.0000D+00 CB= 0.0000D+00
    CYSS=-0.0000D+00 CE= 0.0000D+00
       Y0          Z0         PANG
   0.0000D+00 0.0000D+00 0.0000D+00
  NO=   2 SECTION=H D= 0.4000D+03 B= 0.2000D+03 WT= 0.8000D+01 FT= 0.1300D+02
    SYSS= 0.3260D+03  E= 0.2100D+06 HARD= 0.1000D-01 VARSP= 0.3000D+00
      CD= 0.0000D+00 CB= 0.0000D+00
    CYSS=-0.0000D+00 CE= 0.0000D+00
       Y0          Z0         PANG
   0.0000D+00 0.0000D+00 0.0000D+00
  ------------------------------------------------

  CHORD ANGLES
   ELEMENTS
        5 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
        6 0.0000D+00
  ------------------------------------------------

  DATA OF SEMI-RIGID CONNECTIONS
  NISR=   0
  ------------------
```

```
NJSR=   0
------------------------------------------------


NUMBER OF TENSION BRACES
  0
NUMBER OF STEEL DAMPERS AND TRUSS MEMBERS
  0
NUMBER OF SLAB EQUIVALANT BRACES
  0
------------------------------------------------


NUMBERS OF FIXED NODES
 NDX=   2  NDY=   6  NDZ=   2  NRX=   6  NRY=   2  NRZ=   6  NRXX=   6
 FIXED NODES FOR NDX
   1    4
 FIXED NODES FOR NDY
   1    2    3    4    5    6
 FIXED NODES FOR NDZ
   1    4
 FIXED NODES FOR NRX
   1    2    3    4    5    6
 FIXED NODES FOR NRY
   1    4
 FIXED NODES FOR NRZ
   1    2    3    4    5    6
 FIXED NODES FOR NRXX
   1    2    3    4    5    6
------------------------------------------------


INITIAL LOADS (ACTUAL VALUES)
 NFXI=   0  NFYI=   0  NFZI=   4  NMXI=   0  NMYI=   0  NMZI=   0
 NODE        FXI(I)     FYI(I)     FZI(I)    FMXI(I)    FMYI(I)    FMZI(I)
   1       0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   2       0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
   3       0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
   4       0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00 0.0000D+00
   5       0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
   6       0.0000D+00 0.0000D+00-0.3465D+06 0.0000D+00 0.0000D+00 0.0000D+00
```

```
------------------------------------------------

 (RESIDUAL STRESSES IN H-SECTION NOT CONSIDERED)
------------------------------------------------
```

( the maximum acceleration(s) in the original acceleration data)

```
MAXIMUM VALUE IN x.acc = 0.2101D+04   TIME RANGE = 0.5352D+02
```

( the maximum accelerations specified by the input data)

```
MAXIMUM ACCELERATIONS USED IN THE ANALYSIS
 X-DIR: 0.4000D+04  Y-DIR: 0.0000D+00  Z-DIR: 0.0000D+00

TIME INCREMENT AND DURATION TIME OF ANALYSIS
 DELTT= 0.2000D-02   TEND= 0.2000D+02

DIRECTIONS OF MOTION
 UNI-AXIAL(X)

RAYLEIGH DAMPING ASSUMED
```

( monitoring of the computation every 200 steps )

```
TIME= 0.0000E+00
TIME= 0.4000E+00
TIME= 0.8000E+00
TIME= 0.1200E+01
................
................
```

**(2) Time histories of story shearing forces and drift angles of assigned five stories : d7.txt and d10.txt**

In ‘d7.txt’, the results for the first assigned two stories are written. The first column is elapsed time, from the second to the seventh columns are story shearing force in the X-axis direction, story drift angle in the X-axis direction, story shearing force in the Y-axis direction, story drift angle in the Y-axis direction, story inertial force in the Z-axis direction, and story relative displacement in the Z-axis direction, respectively. The eighth to the thirteenth columns are the results for second assigned story. The results for the remaining three assigned stories are written into ‘d10.txt’.

```
     TIME         1-F-X      1-DRIFT-X      1-F-Y      1-DRIFT-Y      1-F-Z       1-DISP-Z
2-F-X      2-DRIFT-X      2-F-Y      2-DRIFT-Y      2-F-Z      2-DISP-Z
```

```
  0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  0.1000D-01  -0.3177D+01  -0.1363D-07   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
-0.2516D+01  -0.2392D-09   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  0.2000D-01  -0.8559D+01  -0.1079D-06   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
-0.4747D+00  -0.1227D-08   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  0.3000D-01  -0.3357D+02  -0.4215D-06   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
-0.2538D+00  -0.5915D-08   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  0.4000D-01  -0.9750D+02  -0.1319D-05   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.2512D+01  -0.2557D-07   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  0.5000D-01  -0.2379D+03  -0.3359D-05   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.7894D+01  -0.8954D-07   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  0.6000D-01  -0.5176D+03  -0.7660D-05   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.2109D+02  -0.2617D-06   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
```

--------- the rest is omitted ----------

**(3) Yielded elements, coordinates of each node, maximum values of displacement, drift angle, acceleration, and shearing force of each story up to now at every 200 steps: d8.txt**

The first two lines are the information about elapsed time, story shearing force in the X-axis direction, etc., and are followed by the number of yielded points and corresponding node numbers and element numbers, current coordinates of each node, and the maximum absolute values of important physical quantities.

```
     TIME         1-F-X       1-DRIFT-X      1-F-Y      1-DRIFT-Y      1-F-Z       1-DISP-Z
2-F-X      2-DRIFT-X      2-F-Y      2-DRIFT-Y      2-F-Z      2-DISP-Z
  0.2400D+01   0.3444D+06   0.1344D-01   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.1585D+06   0.9666D-02   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
  NUMBER OF YIELDED POINTS=     6
   NODE     4 OF ELM     3 YIELDED.
   NODE     2 OF ELM     2 YIELDED.
   NODE     1 OF ELM     1 YIELDED.
   NODE     3 OF ELM     2 YIELDED.
   NODE     3 OF ELM     3 YIELDED.
   NODE     2 OF ELM     1 YIELDED.

  -------------------------------------------------

   NODE AND (X,Y,Z) COORDINATES AT THE PRESENT TIME
```

```
   2   0.4703D+02   0.0000D+00   0.3499D+04
   3   0.5047D+04   0.0000D+00   0.3497D+04
   5   0.8088D+02   0.0000D+00   0.6998D+04
   6   0.5081D+04   0.0000D+00   0.6996D+04

-------------------------------------------------
 OBSERVED MAXIMUM VALUES AT THE 1ST TO N-TH STORY

  DISPLACEMENTS IN X-DIRECTION
   0.4784D+02   0.8347D+02

  DISPLACEMENTS IN Y-DIRECTION
   0.0000D+00   0.0000D+00

  DRIFT ANGLES IN X-DIRECTION
   0.1367D-01   0.1105D-01

  DRIFT ANGLES IN Y-DIRECTION
   0.0000D+00   0.0000D+00

  ACCELERATIONS IN X-DIRECTION
   0.2708D+04   0.3939D+04

  ACCELERATIONS IN Y-DIRECTION
   0.0000D+00   0.0000D+00

  SHEAR FORCES IN X-DIRECTION
   0.3581D+06   0.2785D+06

  SHEAR FORCES IN Y-DIRECTION
   0.0000D+00   0.0000D+00
=================================================
```

**---------** the rest is omitted **-----------**

**(4) Resultant forces of all the elements for every 200 steps: d9.txt**

The first two lines are the information about elapsed time, story shearing force in the X-axis direction, etc., and are followed by the number of yielded points and the resultant forces of all the elements.

```
     TIME          1-F-X       1-DRIFT-X      1-F-Y      1-DRIFT-Y      1-F-Z      1-DISP-Z
2-F-X      2-DRIFT-X      2-F-Y      2-DRIFT-Y      2-F-Z      2-DISP-Z
   0.2400D+01   0.3444D+06   0.1344D-01   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
0.1585D+06   0.9666D-02   0.0000D+00   0.0000D+00   0.0000D+00   0.0000D+00
   NUMBER OF YIELDED POINTS=     6
   RESULTANT FORCES
   ELEM  NODE     RX             RY             RZ             MX             MY             MZ
MOMEGA
      1      1   0.4515D+06  -0.3978D-11   0.1911D+06  -0.2345D-11  -0.3596D+09  -0.7906D-08
0.0000D+00
      1      2  -0.4515D+06   0.3978D-11  -0.1911D+06   0.2345D-11  -0.3091D+09  -0.6016D-08
0.0000D+00
      2      2  -0.1082D+05  -0.1301D-13  -0.1581D+06   0.2568D-11   0.3964D+09  -0.4401D-10
0.1819D-08
      2      3   0.1082D+05   0.1301D-13   0.1581D+06  -0.2568D-11   0.3942D+09  -0.3764D-10
0.7881D-08
      3      4   0.9296D+06  -0.1360D-10   0.1792D+06  -0.4578D-10  -0.3348D+09  -0.3372D-07
0.0000D+00
      3      3  -0.9296D+06   0.1360D-10  -0.1792D+06   0.4578D-10  -0.2921D+09  -0.1387D-07
0.0000D+00
      4      2   0.2649D+06   0.0000D+00   0.8228D+05   0.0000D+00  -0.8731D+08   0.0000D+00
0.0000D+00
      4      5  -0.2649D+06   0.0000D+00  -0.8228D+05   0.0000D+00  -0.2007D+09   0.0000D+00
0.0000D+00
      5      5   0.1769D+04   0.0000D+00  -0.8082D+05   0.0000D+00   0.2007D+09   0.0000D+00
0.0000D+00
      5      6  -0.1769D+04   0.0000D+00   0.8082D+05   0.0000D+00   0.2034D+09   0.0000D+00
0.0000D+00
      6      3   0.4265D+06   0.0000D+00   0.8730D+05   0.0000D+00  -0.1021D+09   0.0000D+00
0.0000D+00
      6      6  -0.4265D+06   0.0000D+00  -0.8730D+05   0.0000D+00  -0.2034D+09   0.0000D+00
0.0000D+00
   ==================================================
```

**--------- the rest is omitted -----------**

## 8. Examples of drawing by gnuplot

Output files 's7.txt' (see Section (2) of Chapter 6) , 'd7.txt' and 'd10.txt' (see Section (2) of Chapter 7.3) can be used without change to draw lines of various relationships by gnuplot. As you might know, gnuplot is frequently used free software ( https://sourceforge.net/projects/gnuplot/ ). Two examples are shown below.

(1) Comparison of load-displacement relationships of the example frame analyzed in Chapter 4.

The input data files are as follows:

1. frame-s-2d-4div-e.txt: four element approximation for a member with the standard model.
2. frame-s-2d-4div-sr-e.txt: same as the above except that the bending strength 200kNm is set to both beam-ends by using a semi-rigid function.
3. frame-s-2d-mod-e.txt: one element approximation for a member with the modified model.
4. frame-s-2d-mod-sr-e.txt: same as the above except that the bending strength 200kNm is set to both beam-ends by using a semi-rigid function.

The output file 's7.txt' of each analysis was renamed to 's7-4div.txt', 's7-4div-sr.txt', 's7-mod.txt', and 's7-mod-sr.txt' in order.

The following command inputs into gnuplot provide Fig. 14.

```
set key bottom
set key reverse
set key box
set grid
set xlabel 'DISPLACEMENT u (mm)'
set ylabel 'HORIZONTAL LOAD H (kN)'
set xrange [0:500]
set yrange [0:100]
plot 's7-4div.txt' using 2:1 with line linetype 1 title 'static-2d-4div       ','s7-mod.txt' using 2:1 with line linetype 2 title 'static-2d-mod        ','s7-4div-sr.txt' using 2:1 with line linetype 3 title 'static-2d-4div-sr ','s7-mod-sr.txt' using 2:1 with line linetype 4 title 'static-2d-mod-sr    '
```

The above plot command can be written by an abbreviated expression, such as:

```
plot 's7-4div.txt' u 2:1 w l lt 1 t 'static-2d-4div       ', 's7-mod.txt' u 2:1 w l lt 2 t 'static-2d-mod        ', 's7-4div-sr.txt' u 2:1 w l lt 3 t 'static-2d-4div-sr ', 's7-mod-sr.txt' u 2:1 w l lt 4 t 'static-2d-mod-sr    '
```

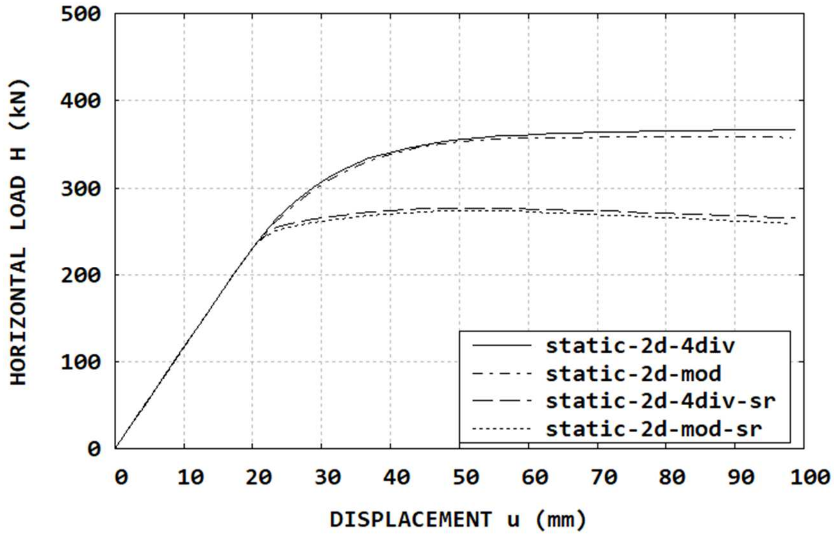



Fig. 14 Comparison of load-displacement relationships of the frame shown in Fig. 3

(2) Comparison of time history of the first story drift angle of the example frame analyzed in Chapter 7.1.

The input data files are as follows:

1. frame-d-2d-4div-e.txt: four element approximation for a member with the standard model.
2. frame-d-2d-mod-e.txt: one element approximation for a member with the modified model.

The output file 'd7.txt' of each analysis was renamed to 'd7-4div.txt' and 'd7-mod.txt' in order.

The following command inputs into gnuplot provide Fig. 15.

```
set key top
set key left
set key reverse
set key box
set grid
set xlabel 'TIME (sec)'
set ylabel 'DRIFT ANGLE (rad)'
set xrange [-0.02:0.025]
set yrange [0:20]
plot 'd7-4div.txt' u 1:3 w l lt 3 t 'dynamic-2d-4div ', 'd7-mod.txt' u 1:3 w l lt 4 t 'dynamic-2d-mod    '
```

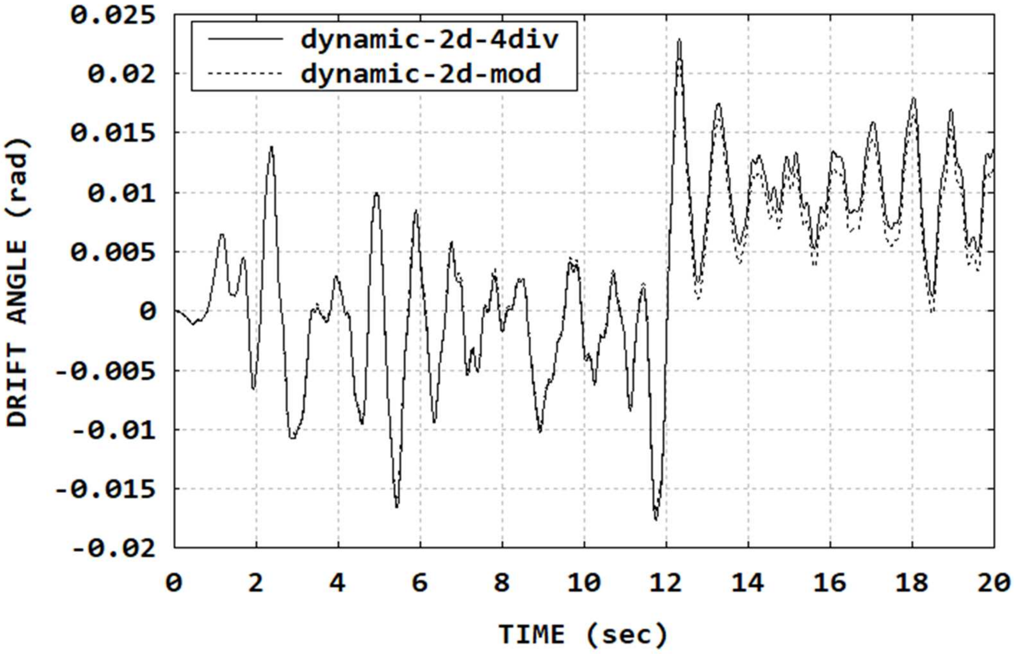



Fig. 15 Comparison of time history of the first story drift angle of the frame shown in Fig. 13

## 9. Conclusion

This manual explained how to create input data files and the content of output files for the quasi-static analysis code FPHM and the dynamic analysis code FPHM-D for elastoplastic three-dimensional frame analysis.

The following are attached: the input data files for quasi-static analysis of the plane frame in Fig. 3 in Chapter 4, a quasi-static analysis of the PC beam with the cross section in Fig. 9-2, a dynamic analysis of the plane frame in Fig. 13 in Chapter 7, and a dynamic analysis of a 3D frame. These files help the user to understand how to create the input data file and can be used as template. The names of the attached files with notes are as follows:

1. Quasi-static analysis of the plane frame in Fig. 3 in Chapter 4
   - frame-s-2d-e.txt: one element approximation with the standard model.
   - frame-s-2d-sr-e.txt: same as the above except that the bending strength 200kNm is set to both beam-ends by using a semi-rigid function.
   - frame-s-2d-4div-e.txt: four element approximation with the standard model.
   - frame-s-2d-4div-sr-e.txt: same as the above except that the bending strength 200kNm is set to both beam-ends by using a semi-rigid function.
   - frame-s-2d-mod-e.txt: one element approximation with the modified model.
   - frame-s-2d-mod-sr-e.txt: same as the above except that the bending strength 200kNm is set to both beam-ends by using a semi-rigid function.

2. Quasi-static analysis of the 3D frame with four columns per story and square plane, which satisfies a 90-degree rotational symmetry with the plane frame in Fig. 13
   - frame-s-3d-mod-e.txt: one element approximation with the modified model.

3. Quasi-static analysis of the PC beam of length 2.4m with the cross section in Fig. 9-2 in Chapter 5
   - pc-beam-s-2d-e.txt: beam is modeled by 6 elements using the standard model.

4. Dynamic analysis of the plane frame in Fig. 13 in Chapter 7
   - frame-d-2d-4div-e.txt: four element approximation with the standard model.
   - frame-d-2d-mod-e.txt: one element approximation with the modified model.
   - frame-d-2d-free-e.txt: input data file for free vibration.

5. Dynamic analysis of the 3D frame with four columns per story and square plane, which satisfies a 90-degree rotational symmetry with the plane frame in Fig. 13
   - frame-d-3d-mod-e.txt: one element approximation with the modified model.

frame-d-3d-mod-Y-e.txt: one element approximation with the modified model.
(one-directional vibration in YZ-plane)
frame-d-3d-free-e.txt: input data file for free vibration.

The use of the one element approximation for a member with the modified model is recommended, especially in dynamic analysis, because considerable computation time may be required for the use of four element approximation for a member with the standard model.

Although FPHM and FPHM-D currently only support the 16 kinds of members depicted in Figures 9-1, 9-2, 9-3, 9-4 and 9-5, it is easy to add a member with arbitrary cross-sectional shape or a composite member containing new materials. A frame with RC walls is not yet covered. The author is looking for a comprehensive experimental study on frames with various RC walls to investigate a method to accurately replace walls with beam-column elements.

## 10. References


Shugyo, M. (2003a). “Elastoplastic large deflection analysis of three-dimensional steel frames.” *Journal of Structural Engineering*, ASCE, 129(9), 1259-1267.

Shugyo, M. (2003b). “Elastoplastic large deflection analysis of curved steel I-beams.” *Advances in Structures, Hancock, et al. (Eds)*, Swets & Zeitlinger, Lisse, 485-490.

Shugyo, M., Oka, N., and Li, J. P. (1996). “Inelastic nonlinear analysis of steel frames with flexible joints.” *Journal of Structural and Construction Engineering (Transactions of AIJ),* 480, 89-94 (in Japanese).

Shugyo, M., and Li, J. P. (1998). “A numerical method for nonlinear analysis of concrete-filled tubular columns.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 505, 147-152 (in Japanese).

Shugyo, M., Shimazu, M., and Sakumoto, Y. (2005). “Collapse analysis of 3D steel frame by a fibred plastic hinge method.” *Advances in steel structures, Vol. 1, Shen, Z. I., et al (Eds.)*, Elsevier, 309-314.

Shugyo, M., Hayashida, Y., Shimazu, M., and Mineshita, Y. (2006a). “Accuracy of the fibered plastic hinge model for elastoplastic analysis of doubly asymmetric section member” *Journal of Structural and Construction Engineering (Transactions of AIJ),* 609, 97-104 (in Japanese).

Shugyo, M., Shimazu, M., and Iwanaga, H. (2006b). “Development of the fibered plastic hinge method for steel-concrete composite members.” *Proc. of the 5th International Conference on Behaviour of Steel Structures in Seismic Areas (STESSA06)*, Yokohama, 2006, 645-650.

Shugyo, M., Shimazu, M., and Hayashida, Y. (2007). “Accuracy of the fibered plastic hinge model for doubly asymmetric section members.” *Proc. of the 6th International Conference on Steel and Aluminium Structures (ICSAS07)*, Oxford, 340-347.

Shugyo, M., Shimazu, M., Hayashida, Y., and Iwanaga, H. (2008). “Elastoplastic analysis of steel-concrete composite members and frames collapsed due to bending.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 73(631), 1535-1542 (in Japanese).

Shugyo, M., and Shimazu, M. (2010). “Detailed analysis of a school gymnasium frame by the fibered plastic hinge model.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 75(651), 943-949 (in Japanese).

Shugyo, M., and Shimazu, M. (2014a). “Elastoplastic seismic response analysis of a frame with lumped mass modeling by the fibered plastic hinge method.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 79(696), 275-283 (in Japanese).

Shimazu, M., and Shugyo, M. (2014b). “Elastoplastic analysis of prestressed concrete beams and frames by the fibered plastic hinge model, *Proc. of the 6th International Conference of Asian Concrete Federation*, Seoul, 777-782.

Shugyo, M., and Shimazu, M. (2016). “Reduction of the total degrees of freedom in frame analysis by the fibered plastic hinge model.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 81(726), 1263-1270 (in Japanese).

Shugyo, M. (2019). “On cancellation of unbalanced force vector in the elastoplastic incremental analysis of a frame.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 84(760), 801-809 (in Japanese).

Shugyo, M. (2020). “Three-dimensional seismic response analysis method for a frame containing member failure.” *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 85(777), 1441-1448 (in Japanese).

**<Appendix 1>**

## Input of the reference value $\eta$ for fracture criterion of specified elements, exclusion of the fractured elements, and the cancellation of unbalanced force vector

Since the present method estimates the internal force vector of a frame as the gradient of the existing elastic strain energy of the whole frame in the incremental analysis[a1)], the unbalanced force vector of the frame can be accurately obtained. Further, the method uses the displacement-controlled load increment technique proposed by Ramm[a2)] with a small step size without iteration, so the computation is very stable even for relatively large unbalanced forces. Therefore, it is possible to exclude the element fractured suddenly at an early point of the loading process and continue the analysis with accurate cancellation of the accompanying considerable unbalanced force. Although the fracture criterion and the reference value are the most important and must be determined carefully in these analyses, the fracture criteria for various frame members are under investigation at the present time. Therefore, an example of the aforementioned frame analysis with a temporarily assumed simple fracture criterion is presented in the following.

The fracture criterion of an element used here is as follows:

$$|\varepsilon|_{max} = \eta\varepsilon_y \tag{a1}$$

where $|\varepsilon|_{max}$ is the maximum quantity of axial strain of a fiber due to varying axial force and biaxial bending moments at both ends of the element, $\varepsilon_y$ is the initial yield strain, and $\eta$ is a reference value. The default value of $\eta$ is 3000, which means a fracture is ignored. The value of $\eta$ can be changed by adding the value with the element number to the end of input date file (see attached file 'frame-s-frac-4div-e.txt').

The example frame, a two-bay two-story plane steel frame, is shown in Fig. a1. In the figure, normal numbers are node numbers and circled numbers are element numbers. Sizes and the mechanical properties of the elements are as follows:

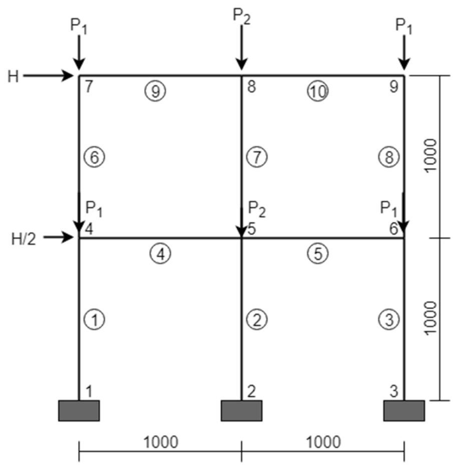



Fig. a1 Two-bay two-story plane steel frame with brittle beams

(1) elements ④,⑤,⑨,⑩ : H-100×100×6×8, yield stress $\sigma_y$=150N/mm$^2$, Young's modulus $E$ =200,000 N/mm$^2$, post-yield strain hardening coefficient $E_t$=0.01$E$

(2) the other elements : H-100×100×6×8, yield stress $\sigma_y$=300N/mm$^2$, Young's modulus $E$ =200,000 N/mm$^2$, post-yield strain hardening coefficient $E_t$=0.01$E$

That is, the beam members are set to yield at an early point of the loading process by assuming a weak yield stress.

The initial vertical loads are set as the initial axial compressive loads of the columns of the first story ① and ③ which are equal to 0.2$P_y$ and that of column ② is set to 0.4$P_y$, where $P_y$ is the initial yield axial force of the columns. Since all the columns have the same area $A$ = 2100mm$^2$ , $P_1$and $P_2$ in Fig. a1 are $P_1$=2100×300×0.1=63,000N and $P_2$=2100×300×0.2＝126,000N. The horizontal loads $H$ and $H/2$ are incrementally loaded as shown in Fig. a1. The following analyses are done by using the four element approximation for a member with the standard model. The reference value $\eta$ is set to 3 for elements ④ and ⑩, and 3000 for the other elements.

Fig. a2 compares the relationships between the horizontal load $H$ and horizontal sway in the same direction at node 7 $u_7$. Fig. a3 compares the relationships between $H$ and the vertical displacement (upward direction is positive) at node 8 $w_8$. The dotted line is the result by the proposed procedure, in which the aforementioned fracture criterion is applied with Ramm's technique. Element ④ fractures at point a in the figures, and soon the element ⑩ fractures at point c. The restoring force deteriorates drastically at both of these points. The dot chain line is the result when a conventional load increment method is employed instead of Ramm's technique. The latter analysis cannot detect the deterioration of the restoring force. The solid line is the curve when the fracture is ignored, and the broken line is that when the unbalanced force is not canceled. The two-dot chain line is the curve for the frame in which elements ④ and ⑩ are initially excluded.

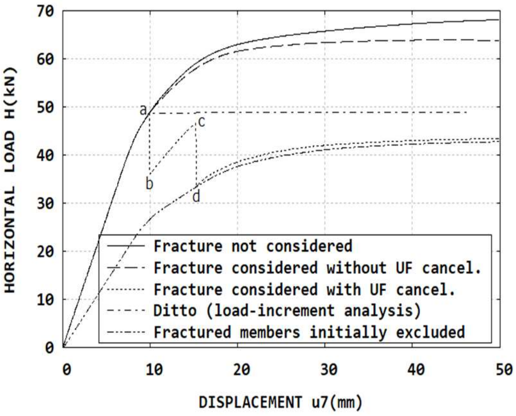



Fig. a2 Comparison of $H$-$u_7$ relationships

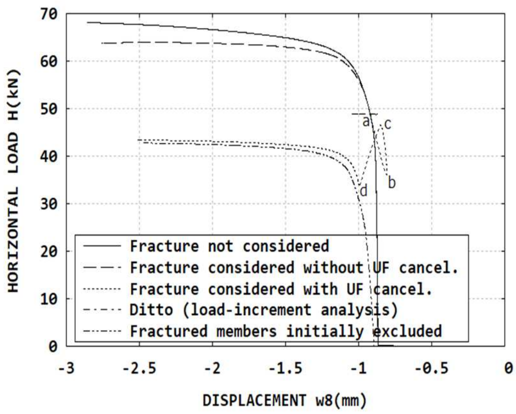
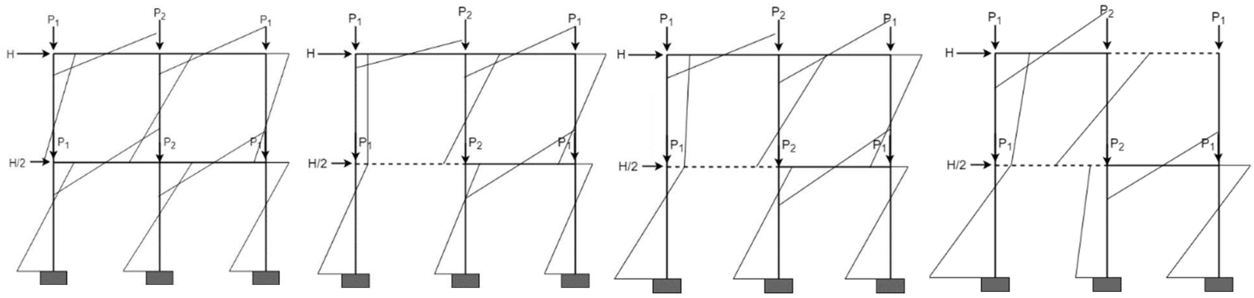



Fig. a3 Comparison of $H$-$w_8$ relationships

As shown in Fig. a2 and Fig. a3, the final behavior of the frame considering the fracture of brittle members tends approach to that of the frame initially excluding the fractured members. The restoring force characteristics of the frame having both brittle and ductile members, shown as the dotted line in Fig. a2, agrees qualitatively with the schematic diagram presented in the Commentary on Structural Regulations of the Building Standard Law of Japan 2015 Edition[a3].

Fig. a4 shows the bending moment diagrams at points a, b, c and d in the above figures obtained by the proposed procedure. The bending moment diagram at point d is almost the same as that for the frame in which elements ④ and ⑩ are initially excluded.



(a) point a (b) point b (c) point c (d) point d

Fig. a4 Bending moment diagram

The input data file, 'frame-s-frac-4div-e.txt', for the frame in Fig. a1 is also attached. The reference values are added to the end of the data file with a comment line and the corresponding element numbers and reference values . See the end of 'frame-s-2d-frac-4div-e.txt'.

**<Appendix 2>**

## Specification of how to obtain the internal force vector for estimation of the unbalanced force vector of a frame

As mentioned in Appendix 1, the present method uses the gradient of elastic strain energy to obtain the internal force vector of a frame, which is a base of the estimation of the unbalanced force vector. This approach is the default of the present method. From v4.05 onwards, the function is added to specify an alternative approach [a1] in which the integration of the nodal force increments in each step is used to obtain the internal force vector of a frame.

The specification can be done by adding the character string 'proc=2' or 'PROC=2' to the beginning of the first comment line of the input data file for FPHM (for FPHM-D, the beginning of the first comment line of 'fr.txt' (see Section 7.2 (1) of Chapter 7)). For example, the first comment line of the attached 'frame-s-2d-mod-e.txt' becomes as follows:

```
'proc=2,remodeling number (5 indicates modified model)'
```
or
```
'PROC=2,remodeling number (5 indicates modified model)'.
```

If 'proc=2' or 'PROC=2' is specified, the comment:
```
(INTEGRATION OF NODAL FORCE INCREMENT)
```
is written at the next line of the comment:
```
*** FPHM DATA-INPUT STARTED ***
```
in the output data file 's6.txt' or 'd6.txt'.

Otherwise, the comment:
```
(GRADIENT OF ELASTIC STRAIN ENERGY)
```
is written there.

The reliability of the results under the specification 'proc2' for 3D problems may slightly decrease, but the computation becomes stable.

**References**

a1) Shugyo, M.(2019). "On cancellation of unbalanced force vector in the elastoplastic incremental analysis of a frame." *Journal of Structural and Construction Engineering (Transactions of AIJ)*, 84(760), 801-809 (in Japanese).

a2) Ramm, E.(1982). "Riks/Wempner approach – an extension of the displacement control method in nonlinear analysis." Recent Advances in Nonlinear Computational Mechanics, Pineridge Press, pp.63-86 a3) Commentary on Structural Regulations of the Building Standard Law of Japan Editorial Committee.(2015) “2015nen-ban Kenchiku-butsu no Kouzoukankei-gijutsukijun-kaisetsusho (Commentary on Structural Regulations of the Building Standard Law of Japan 2015 Edition).” Official Gazette Cooperation of Japan, p.344 (in Japanese)