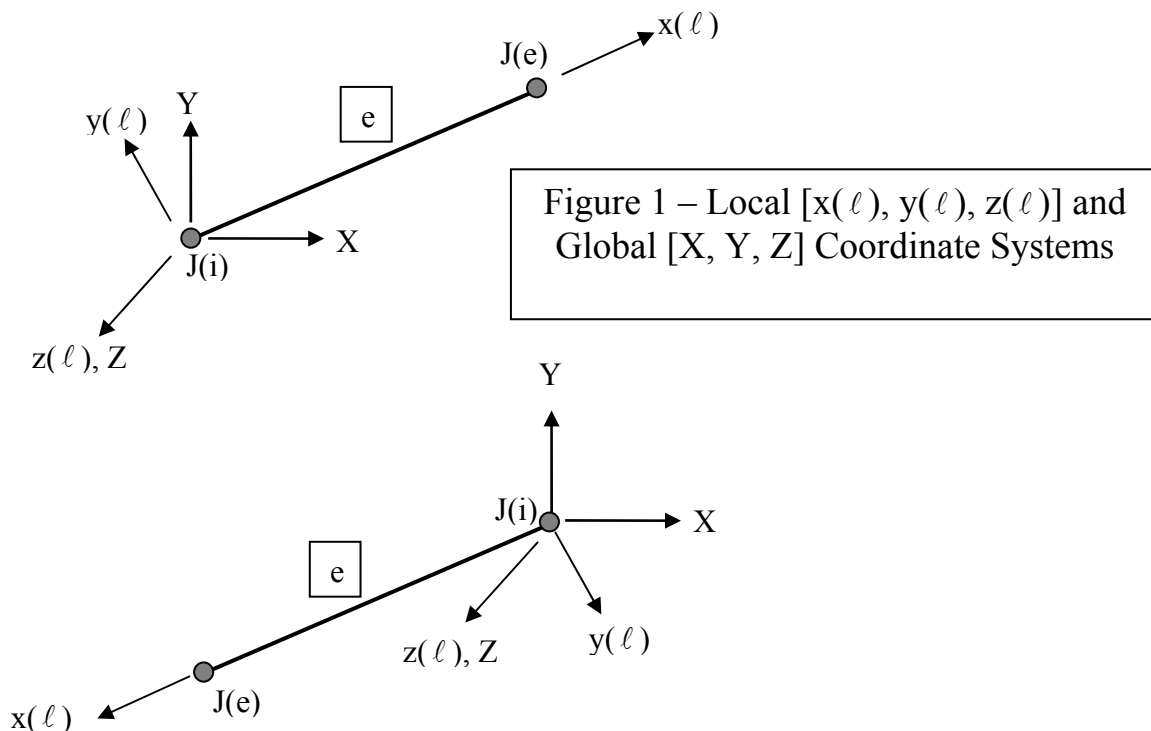


I – Introduction

KYFrame has been designed and developed to be a user friendly, two-dimensional structural analysis tool. Detailed on-screen instructions allow the user the freedom to solve structural systems without consulting a user's manual for every stage of the program operation. A step-by-step procedure is followed whereby the user is guided through the data input and solution of plane frame or truss problems of small order.

KYFrame analyzes two-dimensional structural systems consisting of beam and/or truss elements in either the English or Metric system of units. Since the program handles only small systems, the maximum number of elements (members), nodes (joints), support nodes, property sets, and load cases are: 50, 25, 10, 5, and 15, respectively. The program utilizes both local coordinate and global coordinate systems, as shown in Fig. 1.



While knowledge of both is a plus, it is not absolutely necessary to use the program. Basically the local coordinate system is measured relative to the element or member with the local x-axis, $x(\ell)$, running along the length of the member and the local y-axis, $y(\ell)$, oriented at 90 degrees to $x(\ell)$ such that a right-hand coordinate system is formed with the z-axis always coming out of the plane of the paper, i.e., directly towards the reader. Global axes are defined in a similar manner, except that the global X and Y axes are fixed with the origin of the axes acting as a reference point for measuring distances. Again, the X and Y axes should be arranged such that a right-hand coordinate system is formed with the global Z axis coming out of the plane of the paper towards the reader.

The user has the option of entering data directly through the keyboard or by creating a data file which is stored on disk. As data is entered on the keyboard, it is transferred to an external file, specified by the user, which can be updated by any text editor (e.g. NotePad) and read by the program in the future using the data file input option. KYFrame requires all joint properties, member properties, sectional properties, and load considerations to be entered as input. When describing the structure, the user may choose from a range of supports, member types, cross-sectional properties, loadings, and orientations.

The user has many alternatives when considering loadings, the first of which is choosing between mechanical or non-mechanical loadings. Loading possibilities supported by KYFrame include: point forces at the nodes, member trapezoidal distributed loads or concentrated loads, member thermal loading with a linear through depth variation, and support settlements. The program can consider each of these either individually or in combination as a single load case.

In addition to being used to provide on-screen plots of the original and deformed structures plus the member shear and bending moment diagrams, the results generated by KYFrame are written to an output file with output file name specified by the user. This file lists all of the input data, nodal displacements, member end forces, and support reactions of the particular structural system under investigation for each specified load case. All output files can be imported into word processors for printing, e.g., WordPad. Use a fixed-width font (e.g., Courier New). While the structure graphics (original structure, deformed structure, member shear and bending moment diagrams) are not directly written to the output file, a hard copy may be obtained by saving the screen image into Paintbrush and then printing the JPEG image. The user should note that deflections along the length of an individual member are not listed in the output file; only the nodal displacements are recorded. The deformed structure as displayed on the screen approximates the actual condition so that the user may visually comprehend the system's response to the given load case(s) and inspect whether or not the loading was input correctly.

Before beginning a session with KYFrame, having a detailed sketch of the problem to be solved on paper is strongly recommended. The first screen the user will encounter sets the structure system parameters in which the analysis will be constrained; the user is instructed to define the size of the structural system to be solved. In order to do this, the user must be familiar with the number of elements (members), number of nodes (joints), coordinate and element data, etc. for the analysis. This is extremely important! An error at these prompts can only be rectified on the appropriate input screen. For example, if the user miscounts the number of elements and subsequently enters the incorrect number, then only two alternatives exist to correct this error after advancing to the next screen: restart the program or edit the input file externally to add the missing element(s). Regardless of which alternative is chosen, a new analysis is required. A detailed sketch is useful for counting the elements and nodes, as well as knowing exactly what coordinates and loads should be input so that errors of this type can be avoided. Once the analysis is complete, the sketch is also useful to compare the original structure plotted by KYFrame versus the structural system expected by the user.

II - Getting Started

To start the program, simply double click the KYFrame icon. The first screen simply provides the program and programmer names as well as displaying a right-hand coordinate system. Pressing any key will bring up a second screen which provides some introductory comments on KYFrame. Pressing any key after this screen brings up the third screen which asks whether you desire to input the data through the **Keyboard** (input “**K**” or “**k**”) or through a **Data file** (input “**D**” or “**d**”). A fourth screen prompts for the input and output data file names. If you selected D (or d) the next screen will prompt you for displaying the input structure. This screen is followed by a prompt requesting whether you wish to display the deformed structure on top of the original structure or not and a scale factor. The last screen will prompt you for displaying the member shear and bending moment diagrams. Additional details on these screens are provided later in the documentation.

All input and output files are written to the directory or diskette from which the program is operating unless an alternate path is specified. When naming the files, a “.DAT” extension is recommended for input files and a “.OUT” extension is suggested for output files. The program prompts the user for each file name. In the event that the user wishes to store input and output files on a separate directory or diskette, the correct path and file name desired may be typed at the prompt. For example, if the program is operating from the C: drive, typing

```
a:bridge.dat
```

will save the input file bridge.dat on the a: drive (or search for the file under that path if the user has elected to read a file from diskette for input). Output files may be managed in the same manner. Remember, the default path KYFrame uses to search for files is the current directory in which the program is operating. The user should make sure that a path is specified for writing and reading files if the default path is not acceptable.

Once an analysis has been completed, the user may view the input and output files with any DOS text editor or ASCII word processing tool, NotePad is an excellent choice. The input file may be altered using either of these tools and evaluated again in KYFrame using the read file from disk option. It is important to remember that each output file should be given a unique name since the program will automatically overwrite files with the same name. In order to insure that files are not mistakenly overwritten or lost, assigning different names to every input file and every output file is recommended.

III - The Input File

KYFrame has the flexibility to allow the user manual input of all structural and load data or to read the input file from disk. This chapter is devoted to explaining the process of entering the necessary data by means of the keyboard. After the user becomes familiar with the interactive nature of this method, subsequent solutions may be accomplished through the use of input data files read from disk, as discussed briefly in the previous section. The remainder of this chapter is devoted to defining all possible input data. Even if the user chooses to create a data file via an ASCII text editor, the data definitions given in this chapter must be followed in

creating a data file. Further, this chapter discusses the output of the keyboard input data into the user specified input data file. Thus, when creating a data file for direct input into the program, simply follow the keyboard input definitions and place the data in the same order as the keyboard data is output to the user specified input data file, see PRINT #1 statements. Also, the user is prompted for correctness of the input data for each screen of data. If the data is correct, simply hit any key. However, *if you made a mistake in typing the input data*, hit “N” or “n” *to repeat the input of your current screen of data*.

A. System Data

After inputting the input and output data file names, the user is prompted to input:

```
SELECT UNIT SYSTEM (E = English or M = Metric) = US
INPUT THE PROBLEM TITLE DESCRIPTION: Limit to 72 alphanumeric characters
ITLES$
```

Once the system of units is specified, with English units being the default, KYFrame assumes the units specified in Table 1 are used for all input. Support translational displacements are given in terms of the appropriate length unit (in or mm), whereas rotational displacements are specified in terms of radians (rad). When using the keyboard to enter the structure data, the user is reminded of the units with the various input data screen prompts.

Table 1 – KYFrame Units

Parameter	English	Metric
Coordinates	ft	m
Elastic Modulus	ksi	kN/mm ²
Area	in ²	mm ²
Moment of Inertia	in ⁴	mm ⁴
Force	kips	kN
Moment	ft-k	m-kN
Distributed Load	k/ft	kN/m
Temperature	°F	°C
Depth	in	mm
Support Displacements	in or rad	mm or rad

Following the above input, the user is prompted to input:

```
Number of ELeMents (max = 50)           = NEL
Number of Node Points (max = 25)        = NNP
Number of Support NoDes (max = 10)      = NSND
Number of material Property Sets (max = 5) = NPS
Number of Load Sets (max = 15)         = NLS
```

Note that this input screen defines the system level parameters and utilizes terminology consistent with the finite element analysis of framework structural systems. For students not familiar with finite element terminology; an element can be interpreted as a member and a node as a joint. The number of support nodes defines the number of nodes for which an external support is attached. **NPS** defines the number of members made of different materials or with different cross sectional properties, i.e., areas and moments of inertia. **NLS** defines the number of different load sets for which a given structure is to be analyzed. ***Note the program assumes that the structure geometry and boundary conditions are the same for each load set (or load case).*** With the load set, the user has the option of defining a single load case with individual or multiple loads. For example, load set 1 may have a joint load and temperature load acting in combination while load set 2 may only have a member load. The user should also be aware of the fact that the program requires all load data for a particular load set to be entered before the next load set is considered. If two load sets are chosen, the user must first input the node, element, temperature, and support settlement displacements for load set 1 before proceeding to define load set 2. The user should enter this data carefully so that each loading is properly assigned to the particular load set desired for analysis. Additional details on the input load data are given in section D.

After confirmation of the system data; the unit data, problem title description, and system data are written as three lines to the input data file as

```
PRINT #1, U$
PRINT #1, ITLE$
PRINT #1, NEL; NNP; NSND; NPS; NLS
```

which is the same order that must be followed in creating the input file via an ASCII text editor.

B. Node Data

Coordinates for each node are input in the global coordinate system based upon the measurement system selected by the user (see Table 1). A detailed sketch, as previously recommended, is useful for entering the correct coordinate data. The user can locate the origin of coordinates at any point on the structure. All node point locations are measured relative to the selected origin. Obviously, selecting one of the nodes as the origin makes the determination of its coordinates trivial since both X and Y are zero.

The node data input is:

Node number	= NDNO
Number of generated nodes	= NGND
Node increment number	= NDIN
X-Coordinate (units)	= X
Y-Coordinate (units)	= Y
X-Coordinate increment	= DELX
Y-Coordinate increment	= DELY

where the variable following the equal sign is the variable used to temporarily store the input data. The nodal input data is output to the input data file in the exact same order as the input, upon confirmation that the input is correct, but on two lines, i.e.

```
PRINT #1, NDNO; NGND; NDIN
PRINT #1, X; Y; DELX; DELY
```

As indicated in the node data input list, it is possible to eliminate some node data input by generating the node data. KYFrame uses a linear interpolation scheme to generate the nodal coordinates, i.e.

```
XND(NDNO) = X
YND(NDNO) = Y
FOR IGND = 1 TO NGND
    NDNM = NDNO
    NDNO = NDNM + NDIN
    XND(NDNO) = XND(NDNM) + DELX
    YND(NDNO) = YND(NDNM) + DELY
NEXT IGND
```

This generation algorithm requires that the node number and coordinate increments remain constant. An example is shown in Fig. 2 where the node increment is 2, $DELX = DX$, and $DELY = DY$. Some of the Appendix example problems also demonstrate the use of node data generation. Node data input continues until all NNP nodes have been defined.

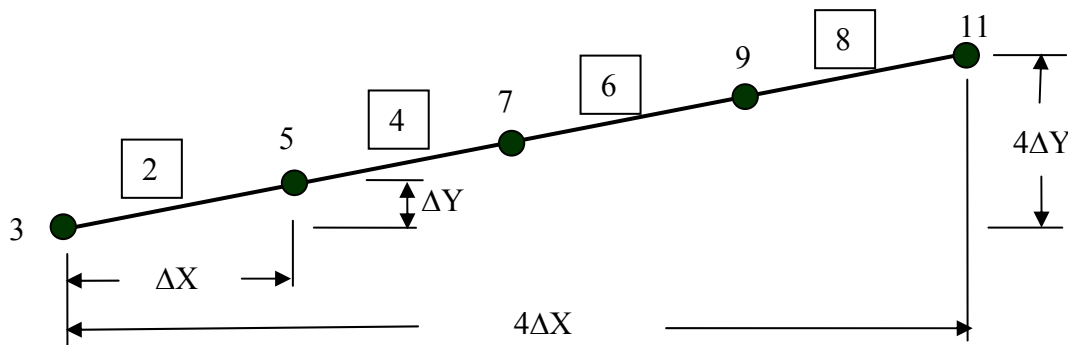


Figure 2. Node and Element Generation Schematic
(Node Increment = 2; Element Increment = 2)

C. Support Data

After defining the node data, the boundary conditions for the structural system are input. Figure 3 illustrates the support types available in the program. A list of these support types also appears on the prompt screen accompanied by a diagram showing each support type. For example, a horizontal roller (type 1) should be chosen if motion in the global X-direction is allowed while restraining the global Y-direction displacement. Conversely, a vertical roller (type 2) restricts motion in the global X-direction, but leaves the global Y-direction free. A

hinge (type 3) support restricts displacement in both the global X- and Y-axes directions. Lastly, a fixed support (type 4) restricts all displacement movement at a joint, including the rotational displacement.

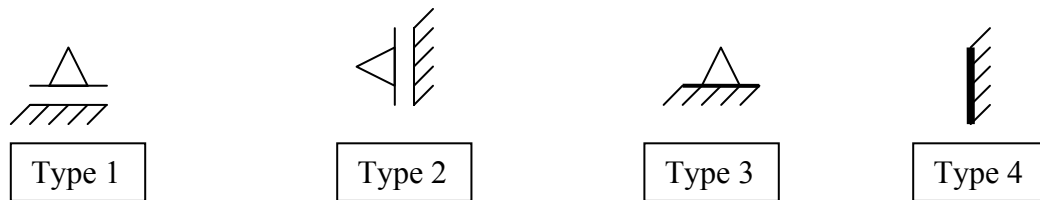
Support data input is:

Support Node Number = NSNP
Support Type Number = NSC

After confirmation that the support data has been input correctly, it is stored on the input data file name as

PRINT #1, NSNP; NSC

Support data input continues until all NSND nodes have been defined.



1 = Horizontal Roller; 2 = Vertical Roller; 3 = Hinge; 4 = Fixed

Figure 3 – Support Types

D. Element Data

Having defined the nodal coordinates and support information, the user must assign a number of attributes to the elements (members) in the structural system. Initial (J(i)) and end (J(e)) joints (nodes) must be assigned to each member (elements), see Fig. 1. The user should carefully consider which node is chosen for the beginning and end of the element - recall that the direction of the member ($x(\ell)$) is defined in this manner. Consistency is helpful since future data input will depend on the direction of a member. Figure 1 also shows how the local coordinate system is defined based upon the direction of the member. Members which run left to right, bottom to top are recommended. **NOTE: All nodes must be defined as one of the two nodes of an element.**

Input for defining the element data is

Element number	= IEL
Number of generated elements	= NGEL
Element increment number	= NEIN
Node increment number	= NDIN
Element property set (1-5)	= IELPS
Element type (1-4)	= IELTP

Element node J(i)	= IEN1
Element node J(e)	= IEN2

where the variable following the equal sign is used to temporarily store the input data. If IELPS is not in the range of 1 to 5, then it is set equal to one, i.e., IELPS = 1 by default. Similarly, if $1 \leq IELTP \leq 4$ is not true, then IELTP = 1. The element property set number refers to the material data set number defined in the next section, limited to five as mentioned in the system data input section. The four options for “element type” refer to the end node conditions defining the element. Type 1 signifies that the connections at both ends of the member are rigid or that the rotational displacements are continuous between connected members. Type 2 signifies an internal hinge is located at element node 1 (or joint J(i)), whereas type 3 signifies an internal hinge is located at element node 2 (or joint J(e)). Type 4, internal hinge at both ends, is reserved for truss elements, i.e., members which can only resist axial forces. If a moment resisting member has an internal hinge at both ends, then the member must be divided into two elements with only one internal hinge per element. Internal hinges are also frequently referred to as a moment release. Terminology for both indicates that the internal bending moment is zero at the location of the internal hinge or moment release.

The element input data is output to the input data file in the exact same order as the input, with appropriate modifications if necessary, upon confirmation that the input data is correct:

```
PRINT #1, IEL; NGEL; NEIN; NDIN; IELPS; IELTP; IEN1; IEN2
```

As indicated in the element data input list, it is possible to eliminate some element data input by generating the data. KYFrame assumes the generated element data is the same as the input data except for the element number and element nodes, i.e.

```
NELPS(IEL) = IELPS
NELTP(IEL) = IELTP
IEN(1, IEL) = IEN1
IEN(2, IEL) = IEN2
FOR IGEL = 1 TO NGEL
    IELM = IEL
    IEL = IELM + NEIN
    NELPS(IEL) = NELPS(IELM)
    NELTP(IEL) = NELTP(IELM)
    IEN(1, IEL) = IEL(1, IELM) + NDIN
    IEN(2, IEL) = IEL(2, IELM) + NDIN
NEXT IGEL
```

This generation algorithm requires that the element and node number increments remain constant and that the element type and material property set numbers remain constant. Considering the example of Fig. 2, NEIN = 2 and NDIN = 2.

Element data input continues until all NEL elements have been defined.

E. Material Property Data

The user is prompted to enter the unique material and cross sectional properties. Input data is prompted as

Elastic modulus (E, units) = EM(IPS)
Cross sectional area (A, units) = AREA(IPS)
Moment of inertia (I, units) = RMIZ(IPS)

where the material data units are given in Table 1. Upon confirmation that the data is correct, the input data is output in exactly the same sequence, i.e.

PRINT #1, EM(IPS); AREA(IPS); RMIZ(IPS)

The input/output loop is continued for IPS = 1 to NPS.

At this stage of the program, KYFrame prompts the user regarding the display of the input structure. Pressing any keys other than “N” or “n” will display the structure geometry for visual confirmation of the input data.

F. System Load Data

Once the structural system has been defined, the user must now assign loadings to be considered in the analysis. The load data must be defined for each load set in sequence from 1 to NLS. Only the data required to define the structure loading for the current load set is input. Load set limits are defined by the user input:

Number of Concentrated Force Nodes = NCFN
Number of Distributed Loaded Elements = NDLE
Number of Thermal Loaded Elements = NTLE
Number of Displacement Loaded Nodes = NDLN

Upon confirmation that the input data is correct, it is written to the input file name as

PRINT #1, NCFN; NDLE; NTLE; NDLN

G. Joint Load Data

If NCFN > 0, then the user is prompted to input the joint load data:

FOR ICN = 1 TO NCFN
Concentrated Force NoDe Number = NCFND
global X-axis force (F_X, units) = CF(1)
global Y-axis force (F_Y, units) = CF(2)
applied nodal moment (M_Z, units) = CF(3)
NEXT ICN

where the units are defined in Table 1. Positive joint force data coincides with the global coordinates selected, typically F_X is positive when it acts to the right, F_Y is positive when it acts in an upward vertical direction, and M_Z is positive when it acts in a counterclockwise direction, as shown in Fig. 4. Remember, joint loads are concentrated loads located at the intersection (or end) of members in a structural system.

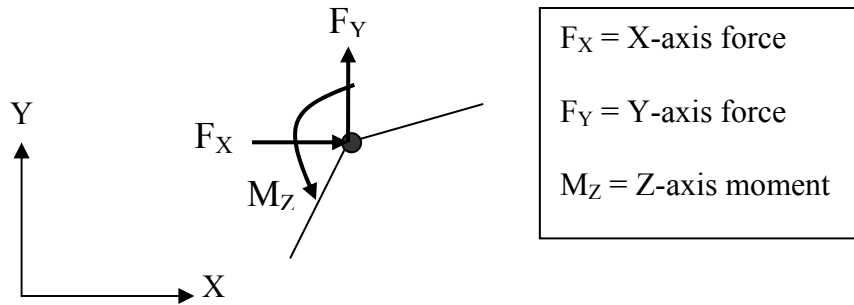


Figure 4 – Joint (Node) Load Data

Upon confirmation that the user is satisfied with a screen of joint load data, it is written to the input file name as

```
PRINT #1, NCFND; CF(1); CF(2); CF(3)
```

for each loaded joint.

H. Member Load Data

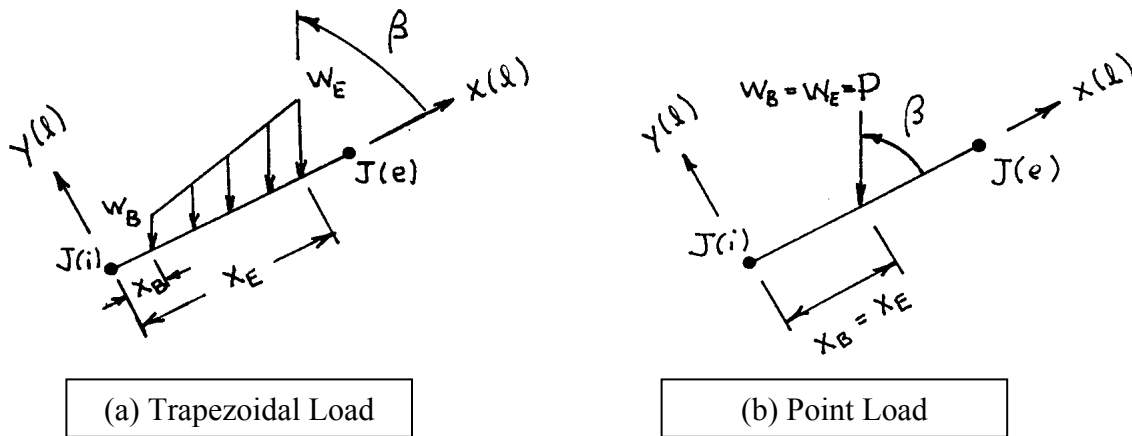


Figure 5 – Member (Element) Load Data

If $NDLE > 0$, the user is prompted to input:

```
FOR IDLE = 1 TO NDLE
  Loaded Element Number = LDEL
```

Coordinate x_B (units) = TXB
 Coordinate x_E (units) = TXE
 Distributed Load w_B (units) = TWB
 Distributed Load w_E (units) = TWE
 Load Angle $BETA$ (rad) = TB

NEXT IDLE

where the “T” in front of the variables is to distinguish between the input values and stored element values. Referring to Fig. 5, x_B defines the beginning distance on the member for the distributed trapezoidal load (**defaults to zero for no input**) and x_E represents the end distance (if $x_E \leq 0$ or $L \leq x_E$, then $x_E = L$; **defaults to L for no input**). *Note, both x_B and x_E are measured along the member length, they do not represent the global X-coordinates of points B and E.* The trapezoidal load magnitudes are defined by w_B at the beginning distance point and w_E at the ending distance point (a uniform load is defined by $w_B = w_E$). Angle Beta defines the load orientation with respect to the member local axis $x(\ell)$. Default values for the first loaded member are all zero except for the load angle which is 90° . Subsequent member defaults use the input values of the previously loaded member. All units are defined in Table 1.

If a point load is applied to a member, then input $TXB = TXE$ and $TWB = TWE$ (in terms of the force units of Table 1). The program recognizes $TXB = TXE$ as being a point load on the member or element.

Upon confirmation by the user that the input member load data is correct, the program writes the information out to the input file name as

PRINT #1, LDEL; TXB ; TXE ; TWB ; TWE ; TB

for each loaded element.

Once the joint and member loads (the mechanical loads) have been entered, the user may opt to assign non-mechanical loadings to the structural system (thermal and support displacements) under investigation.

I. Temperature Load Data

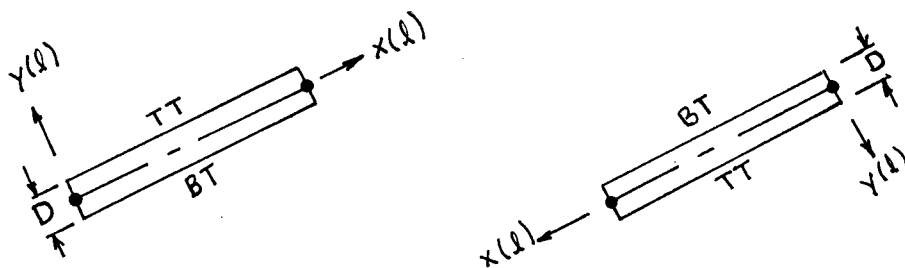


Figure 6 – Member (Element) Thermal Load Data

```

If NTLE > 0, then the user is prompted to input
  FOR ITLE = 1 TO NTLE
    Thermal Loaded Element = LDEL
    Temperature TT (units)      = TT
    Temperature BT (units)      = BT
    Beam depth Dph (units)      = DPH
    Temp expansion coefficient Alpha (units) = ALPHA
  NEXT ITLE

```

where TT is the Top Temperature on the beam, BT is the bottom temperature on the beam, and Alpha is the linear coefficient of thermal expansion, see Table 1 for units. Default values for the first loaded member are all zero. Subsequent member defaults use the input values of the previously loaded member.

To assist the user, KYFrame includes a graphic of the input data including a table of Alpha values for both English and Metric systems of units for standard structural materials (wood, concrete, aluminum, and steel). Beam “top” and “bottom” are defined as the surfaces with the maximum positive and negative local y-axis ($y(\ell)$) values, respectively. Figure 6 exhibits two possibilities for defining the “top” and “bottom” beam surfaces.

A linear temperature variation is assumed between the top and bottom beam surfaces, i.e., through the depth D, in KYFrame. If $TT = BT$, then a constant temperature is specified through the beam depth.

Upon confirmation of the input data for each thermally loaded element, it is output as

```

PRINT #1, LDEL; TT; BT; DPH; ALPHA

```

J. Nodal Displacement Load Data

The final loading option available to the user is the non-mechanical, nodal displacement load data or the specification of support displacements. Certain systems induce member forces based solely on the displacement of any of its supports. The user may define movement in terms of the global X- and Y-directions, as well as rotations about the global Z-axis direction.

```

If NDLN > 0, the user is prompted to input:
  FOR IDLN = 1 TO NDLN
    Nodal Displacement NoDe Number = NDLN
    X-axis Displacement (Dx, units) = DX
    Y-axis Displacement (Dy, units) = DY
    nodal Rotation (Dm, rad)      = DM
  NEXT IDLN

```

Displacement units are defined in Table 1. Upon confirmation of each displacement prescribed nodal values, KYFrame writes the input data as

```

PRINT #1, NDLN; DX; DY; DM

```

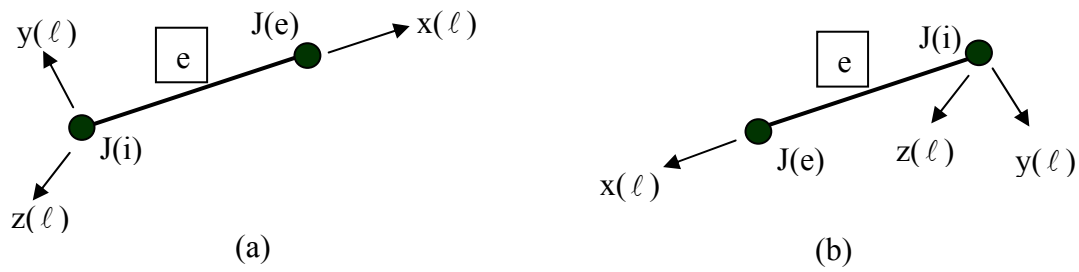
IV - Output File

KYFrame does not offer a list of the results directly on screen, but rather writes the analysis results, along with the input data, to a file whose name is specified by the user at the beginning of the analysis session. The output file may be viewed in a word processing program for review or printing. *Use of a fixed-width font is recommended for printing since the program does produce output based on a fixed width format and will line up in column wise with title blocks, which makes for a clearer output printing.* If subsequent analysis trials are conducted using the same file name, the output data will be overwritten with the new results. In order to guard against accidental erasures, assigning separate names to each file, regardless of the difference between it and a previous trial, is strongly recommended.

The information written to all output files is as follows:

- 1) problem description,
- 2) output file name,
- 3) problem system of units,
- 4) overall system data,
- 5) all node coordinates plus support X Y R constraint codes
(X: X-axis displacement, Y: Y-axis displacement, R: rotation;
if X, Y, or R = 0, free to displace, otherwise = 1, which signifies no
displacement; a word description is also attached based on the support type
specified),
- 6) all element data (property set, type, and nodes),
- 7) load data for each load set,
- 8) nodal displacements for each load set,
- 9) member end forces for each load set, and
- 10) support reactions for each load set.

The user should note that no forces or displacements are listed for incremental positions along a member. The member end forces are given with respect to the local coordinate system for the particular member, i.e., the axial force is in the direction of the member (local x-direction), not in the global x-direction for the system. Figure 7 shows the member-end force output sign convention.



Two Local Coordinate Systems of Figure 1

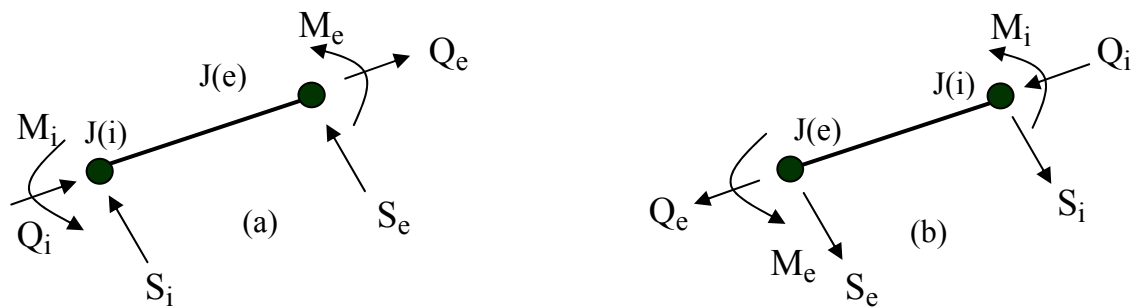


Figure 7 – Member (Element) End Force Sign Conventions

V - Original and Deformed Structure Plots

KYFrame includes the graphic capabilities to plot the original and deformed structure as part of the analysis results. However, the plots are not directly written to a file. Plots of the original and deformed structure are important since they offer a quick check on the response of a structural system to a load set in addition to being a safeguard for checking whether or not the input data was entered correctly.

When viewing the deformed structure, a scale factor may be used to enlarge the displacement experienced by the system. The minimum scale factor is one, the maximum is 20. Fractional scales are available between 1 and 20. Any attempt by the user to assign a scale value outside of this range will cause the program to set the scale factor value to one. The original structure may be omitted from view so that an unobstructed diagram of the deformed structure is displayed. The plot operation of KYFrame is terminated if the user answers “N” or “n” to the prompt to view the deformed structure for a specific load set. A prompt for displaying the original and deformed structure plot is given for each load set. A different scale factor can be chosen for each load set. In fact, the user continues to be prompted for the display of the deformed structure within a load set until a “N” or “n” is entered. This gives the user the option of changing the scale factor in order to increase or decrease the deformed structure plot.

VI - Member Axial Force Status Plots

After the user has completed the original and deformed structure plots, KYFrame prompts for the display of the element axial force status. Input of either “N” or “n” will prevent the display for the current load set. If the user desires to know the element axial force status, KYFrame plots the elements of the original structure in either **white** (zero force member), **red** (tension force element), or **blue** (compression force element).

VII - Shear and Bending Moment Plots

The next KYFrame prompt is for the display of the element shear and bending moment diagrams. Input of either “N” or “n” will prevent the display for the current load set. The shear and bending moment diagrams are drawn for one element at a time using the full screen. Beam sign convention is used to construct the plots and the maximum shear and moment values for the member are displayed with the plots to provide a scale of the individual element values, see example problems for sample plots. Plots begin with element node 1 (or J(i)) and progress across the element to element node 2 (or J(e)). All beam element shear and moment diagrams are plotted. Obviously, truss elements do not resist shears or bending moments, consequently they are not drawn. The shear and moment plots are not intended to be final diagrams. Rather, they simply give a scaled representation of the diagrams and provide assistance to students in constructing their detailed shear and moment diagrams.