

SS1D

A Finite Element Computer Program for One-dimensional Steady State Analyses

written by

V. N. Kaliakin
Department of Civil Engineering
University of Delaware

INTRODUCTION

The computer program SS1D described herein was developed to evaluate finite element analyses for a one-dimensional continuum under steady state conditions. Using the program, problems in elastostatics, in heat conduction, and in fluid flow through a porous medium can be analyzed.

SS1D is intended for instructional use. It is thus liberally commented and uses descriptive variable names. The program is based on a code originally developed by Professor L. R. Herrmann at the University of California, Davis to solve one-dimensional steady state heat conduction problems. SS1D represents a greatly expanded and enhanced (FORTRAN 77) version of this original code.

ELEMENT TOPOLOGY AND GLOBAL COORDINATE SYSTEM

The finite element analysis utilizes a linear approximation for the primary dependent variable within each element. As such, the discretization of the solution domain is achieved using two-noded elements. Depending on the type of analysis performed using this program, the nodes and elements should be sequentially numbered in the following manner:

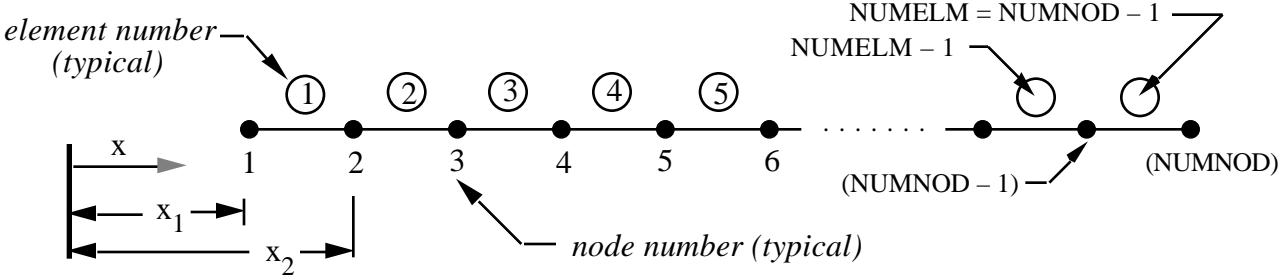


Figure 1 : Element and Node Point Numbering for a Horizontal Domain

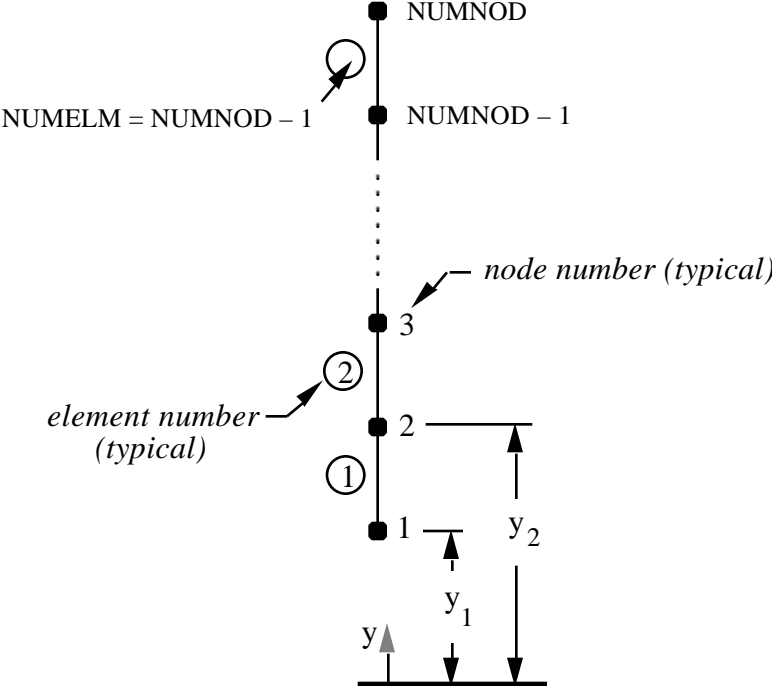


Figure 2 : Element and Node Point Numbering for a Vertical Domain

UNITS

The units used for various input quantities must be consistent, and determine the units of the output. In the subsequent description of the program input, the units associated with input quantities are described using the following conventions:

E : denotes units of energy, heat, or work (e.g., Joules, calories, Btu, etc.)

F : denotes units of force (e.g., Newtons, dynes, pounds, etc.)

L : denotes units of length (e.g., meters, centimeters, inches, etc.)

t : denotes units of time (e.g., seconds, minutes, etc.)

T : denotes units of temperature (e.g., degrees Kelvin, degrees Celsius, etc.)

PROGRAM INPUT

Data input to SS1D is quite simple. With the exception of alphanumeric strings, all input is performed using list-directed READ statements (i.e., free-format). Since input is terminated by exhaustion of the input list rather than by line boundaries, list-directed input records may span several lines. Multiple spaces between input fields are ignored and “null” fields in the input record may leave the value of a particular variable unchanged (as opposed to setting it to zero). *The safest way to avoid erroneous input is to include explicit values for all input variables that are being read*; this ensures that the correct mathematical model will be analyzed. When alphanumeric strings are required in the input, each of them should appear on a separate line and *must not be preceded by a blank record*. Users not familiar with list-directed input are advised to consult appropriate documentation for their computer.

In the subsequent description of the input quantities required by the program the variable types (i.e., alphanumeric, integer, or real) are given in brackets []. Numbers in braces { } signify that a note explaining the particular input quantity is provided in the following section. Since the safest way to avoid erroneous list-directed input is to include explicit values for all variables that are being read, no default values are set within the program. *Note also that only very rudimentary error checking is performed within the SS1D computer program.*

The data required by the program consists of the following quantities:

1. Identification Title Record (A72)

TITLE [alphanumeric] : any title describing the analysis.

2. Control Record

KIND [integer] = $\begin{cases} 1 & \text{: perform elastostatic analysis} \\ 2 & \text{: perform heat conduction analysis} \\ 3 & \text{: perform analysis of fluid flow through porous medium} \end{cases}$

NUMNOD [integer] = total number of node points used in the analysis (2).

NUMMAT [integer] = total number of material types used in modeling the body (1).

NUMLOD [integer] = total number of different load sets used in the analysis (0).

NUMSPE [integer] = total number of records containing nodal specifications (1).

3. Material Data

Supply NUMMAT records to describe all the different material types found in the mathematical model. These records may be supplied in *any order*.

- for KIND = 1 :

NUMBER [integer] = material number

EMOD [real] = value of the elastic modulus (units : FL⁻²).

For a given element, the elastic modulus is assumed *constant*.

ALPHA [real] = value of the coefficient of thermal expansion (units : T⁻¹).

For a given element the coefficient of thermal expansion is assumed *constant*.

- for KIND = 2 :

NUMBER [integer] = material number

CONDUCT [real] = value of the thermal conductivity (units : $\text{Et}^{-1}\text{L}^{-1}\text{T}^{-1}$).
For a given element, the thermal conductivity is assumed *constant*.

• for KIND = 3 :

NUMBER [integer] = material number

CONDUCT [real] = value of the hydraulic conductivity (permeability) (units : LT^{-1}).
For a given element, the permeability is assumed *constant*.

4. Load Set Data

Supply NUMLOD records to describe all the different load sets applied to the mathematical model (if NUMLOD = 0, skip this portion of the input entirely). These records may be supplied in *any order*.

• for KIND = 1 :

NUMBER [integer] = load set number {1}

BFORCE [real] = value of the body force acting in the axial direction;
i.e., force per unit volume (units : FL^{-3}). For a given element, the body force is assumed *constant*.

TRACT [real] = value of the surface traction acting in the axial direction;
i.e., force per unit area (units : FL). For a given element, the surface traction is assumed *constant*.

DELTAT [real] = value of the temperature change (units : T).
For a given element, the temperature change is assumed *constant*.

• for KIND = 2 :

NUMBER [integer] = load set number {2}

SOURC1 [real] = value of the source term S_1 (units : $\text{Et}^{-1}\text{L}^{-3}$).
Over a given element, the source term is assumed *constant*.

SOURC2 [real] = value of the source term S_2 (units : $\text{Et}^{-1}\text{L}^{-3}\text{T}^{-1}$).
Over a given element, the source term is assumed *constant*.

- for $KIND = 3$:

NUMBER [integer] = load set number

SOURC [real] = value of the fluid source term $e(x)$ (units : $L^2 T^{-1}$).

Over a given element, the source term is assumed *constant*.

5. Nodal Specification Data

Supply NUMSPE records to describe all the records containing nodal specifications. These records may be supplied in *any order*.

NUMBER [integer] = number of node point at which the specification is to be applied.

- for $KIND = 1$:

$$\mathbf{IFLAG} \text{ [integer]} = \begin{cases} 0 : \text{if a known value of concentrated axial force} & \{3\} \\ \quad \quad \quad \text{is applied at node NUMBER} & \\ \\ 1 : \text{if a known value of axial displacement} & \{4\} \\ \quad \quad \quad \text{is specified at node NUMBER} & \end{cases}$$

VALUE [real] = specified value of $\begin{cases} \text{concentrated axial force (units : F)} \\ \text{axial displacement (units : L)} \end{cases}$

- for $KIND = 2$:

$$\mathbf{IFLAG} \text{ [integer]} = \begin{cases} 0 : \text{if a known value of concentrated heat flux} & \{5\} \\ \quad \quad \quad \text{is applied at node NUMBER} & \\ \\ 1 : \text{if a known value of temperature} & \{6\} \\ \quad \quad \quad \text{is specified at node NUMBER} & \end{cases}$$

VALUE [real] = specified value of $\begin{cases} \text{heat flux per unit area (units : } Et^{-1}L^{-2}\text{)} \\ \text{temperature (units : T)} \end{cases}$

- for $KIND = 3$:

$$\mathbf{IFLAG} \text{ [integer]} = \begin{cases} 0 & \text{: if a known value of fluid flux} \\ & \text{is applied at node NUMBER} \end{cases} \quad \{7\}$$

$$\begin{cases} 1 & \text{: if a known value of potential (head)} \\ & \text{is specified at node NUMBER} \end{cases} \quad \{8\}$$

$$\mathbf{VALUE} \text{ [real]} = \text{specified value of } \begin{cases} \text{nodal fluid flux (units : t}^{-1}\text{)} \\ \text{potential (head) (units : L)} \end{cases}$$

6. Node Point Data

Supply records to describe all NUMNOD node points in the mathematical model. These records may be supplied in *any order*.

NUMBER [integer] = node point number {9}

XY [real] = x- or y coordinate of the node point (see Figs. 1 and 2)
(units : L)

NUMEND [integer] = number of the final node point to be generated¹.

If node point generation is *not* desired, set NUMEND equal to NUMBER.

XYEND [real] = x- or y coordinate of the final node point to be generated¹.

INCR [integer] = numbering increment for successive node points¹.

RATIO [real] = spacing ratio for successive node points¹.

¹ this quantity is associated with the straight line generation of node point locations; further details are given in program note {9}.

7. Element Information

Supply records to describe all (NUMNOD – 1) elements in the mathematical model. These records may be supplied in *any order*.

NUMBER [integer] = element number

NODE1 [integer] = number of the node point at one end of the element {10}

NODE2 [integer] = number of the node point at other end of the element

MATNUM [integer] = number of the material associated with the element

LODNUM [integer] = number of the load set associated with the element.

If NUMLOD = 0, set LODNUM equal to any integer 1.

AREA [real] = cross-sectional area of the element.

Over a given element, the source term is assumed *constant*.

NUMADD [integer] = number of additional elements to be generated².

If element generation is *not* desired, set NUMADD equal to zero.

INCR [integer] = numbering increment for corresponding nodes in successive elements².

This completes the description of input quantities. To include several analyses in a single data file and to subsequently solve them as “stacked jobs”, repeat the above sequence of records for each analysis. *NOTE*: blank lines cannot precede title records in the data file.

² this quantity is associated with the generation of element information; further details are given in program note {10}.

NOTES REGARDING PROGRAM INPUT

{1} If they act in the direction of the longitudinal coordinate axis, the body force and/or surface traction are *positive*.

{2} The total source term has the following general form:

$$S(x) = S_1 + S_2 T$$

where T represents the temperature.

{3} Concentrated values of axial force may be specified at any number of locations along the body by placing nodes points at these locations. For each such node point **IFLAG** is set equal to zero (0) and **VALUE** is set equal to the known value of force. An axial force is considered positive if it acts in the positive coordinate direction.

{4} Known values of displacement may be specified at any number of locations along the body by placing node points at these locations. For each such node point **IFLAG** is set equal to one (1) and **VALUE** is set equal to the known value of displacement.

NOTE: At node points where the displacement is unknown and there is no axial force applied, **IFLAG** should be set equal to zero (0) and **VALUE** should set equal to 0.0. Since these values are pre-set upon initiation of the program, *there is no need to supply nodal specification records for such cases.*

{5} Concentrated (nodal) values of heat flux (input or output) may be specified at any number of locations along the body by placing nodes points at these locations. For each such node point **IFLAG** is set equal to zero (0) and **VALUE** is set equal to the known value of heat flux. A positive heat flux indicates that heat is being input to the body.

{6} Known values of temperature may be specified at any number of locations along the body by placing node points at these locations. For each such node point **IFLAG** is set equal to one (1) and **VALUE** is set equal to the known value of temperature.

NOTE: At node points where the temperature is unknown and there is no exchange of heat

with the surroundings, **IFLAG** should be set equal to zero (0) and **VALUE** should be set equal to 0.0 . Since these values are pre-set upon initiation of the program, *there is no need to supply nodal specification records for such cases.*

- {7} Concentrated values of fluid flux may be specified at any number of locations along the body by placing nodes points at these locations. For each such node point **IFLAG** is set equal to zero (0) and **VALUE** is set equal to the known value of flux. A positive value of flux indicates that fluid is flowing into the body.
- {8} Known values of hydraulic potential (head) may be specified at any number of locations along the body by placing node points at these locations. For each such node point **IFLAG** is set equal to one (1) and **VALUE** is set equal to the known value of potential.

NOTE: At node points where the potential is unknown and there is no fluid flux applied, **IFLAG** should be set equal to zero (0) and **VALUE** should be set equal to 0.0 . Since these values are pre-set upon initiation of the program, *there is no need to supply nodal specification records for such cases.*

- {9} To assist in defining the the locations of node points, a node generation scheme has been incorporated into SS1D. When generation of a sequence of node points is desired it is only necessary to input the following information (on a single record):
 - the numbers of the initial and final node points in the sequence (in the Program Input these points are denoted by **NUMBER** and **NUMEND**, respectively);
 - the coordinated of the initial and final node points in the sequence (denoted in the Program Input by **XY** and **XYEND**, respectively);
 - the value of **INCR**, which represents the difference between any two successive node numbers in the sequence; and,
 - the value of **RATIO**, which defines the ratio of the distance between any two adjacent pairs of nodes. A value of **RATIO** = 1.0 results in *equally spaced* node points. In selecting a value of **RATIO**, the following three formulas are of use (in the sequel denote the number of segments comprising a sequence of nodes by “m”):

1) If the total length of a line of nodes is equal to L and if **RATIO** = 1.0, then the length of the first segment will equal

$$\frac{L(1 - \mathbf{RATIO})}{(1 - \mathbf{RATIO}^m)}$$

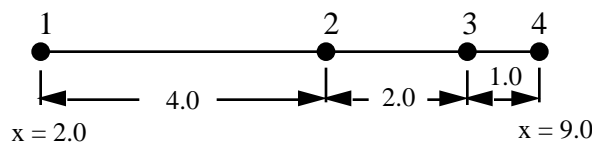
2) If the ratio of the lengths of the last and the first segments must equal a specified value “D” (i.e., $D = L_m / L_1$), then specify a value of **RATIO** equal to D^r , where $r = (m - 1)^{-1}$ and where $m > 1$.

3) Suppose a line of nodes has been generated with a particular value of **RATIO** and that this line is to now be regenerated with *twice* as many segments (i.e., with twice as many node points minus one). Furthermore suppose that it is desired to have the locations of the first set of nodes be retained in the second set. To achieve the above-mentioned goals use $\sqrt{\mathbf{RATIO}}$ as the new spacing ratio.

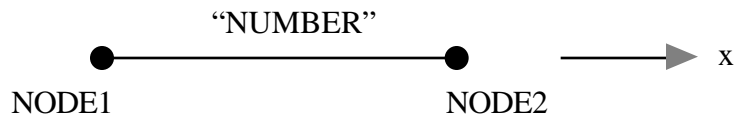
In generating a sequence of nodes, the end points may be entered in *either* order. For example, the sequence illustrated below can be defined by specifying either

1 2.0 4 9.0 1 0.5 or 4 9.0 1 2.0 -1 2.0

The spacing of the intermediate points (nodes 2 and 3) is controlled by the value selected for **RATIO**.

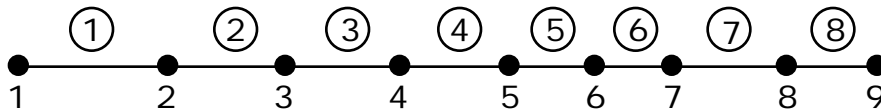


{10} Element number “NUMBER” has the following configuration:



If the material number (**MATNUM**), the load set number (**LODNUM**) and the cross-sectional area (**AREA**) are the same for a contiguous sequence of elements, the node numbers

associated with these elements can be simply established by means of the element generation option available in SS1D. To generate a sequence of elements, node points (**NODE1** and **NODE2**) are specified only for the first element, together with values for **MATNUM**, **LODNUM**, **AREA**, **NUMADD** and **INCR**. For example, in the grid shown below, elements 2 through 5 possess identical material numbers (equal to 2) and cross-sectional areas (equal to 24.0) and are subjected to a loading described by load set 1.



The record required to specify all information for element 2, and to generate all information for elements 3 through 5 is thus:

2 2 3 2 1 24.0 3 1

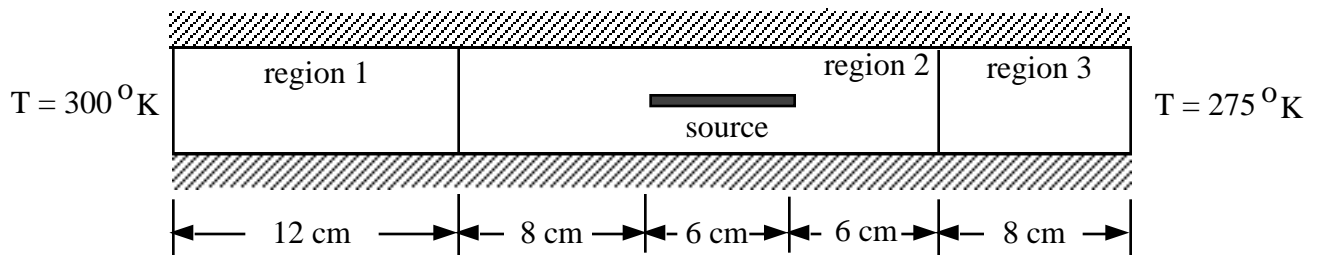
PROGRAM OUTPUT

The output generated by SS1D consists of:

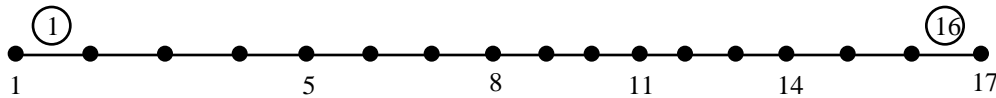
- an “echo” of the input data;
- the approximate values of the primary dependent variable (i.e., the displacement, temperature or hydraulic potential) at the nodes. Intermediate values can of course be found by returning to the linear approximation for temperature within each element; and,
- the approximate values of the secondary dependent variable (i.e., stress, heat flux or fluid velocity) at the element centers (the coordinates of these points are also printed).

SAMPLE ANALYSIS : one-dimensional steady state heat conduction

Consider the composite wall shown below with thermal conductivities of $k_1 = 18 \text{ W/m}^\circ\text{K}$, $k_2 = 10 \text{ W/m}^\circ\text{K}$, $k_3 = 40 \text{ W/m}^\circ\text{K}$, with a locally distributed heat source of 12.5 W/m^3 , and with a unit cross-sectional area.



The following finite element grid is used to model the above wall. Units of Watts, meters and degrees Kelvin are used.



A copy of the complete input data file for this analysis is shown below.

sample analysis - one-dimensional heat conduction

2 17 3 2 2

1 18.0

2 10.0

3 40.0

1 0.0 0.0

2 12.5 0.0

1 1 300.0

17 1 275.0

1 0.0 5 0.12 1 1.0

5 0.12 8 0.20 1 1.0

8 0.20 11 0.26 1 1.0

11 0.26 14 0.32 1 1.0

14 0.32 17 0.40 1 1.0

1 1 2 1 1 1.0 3 1

```

5  5  6  2  1  1.0  2  1
11 11 12  2  1  1.0  2  1
8  8  9  2  2  1.0  2  1
14 14 15  3  1  1.0  2  1

```

The results obtained using the above data in conjunction with the SS1D computer program are shown below.

sample analysis - one-dimensional heat conduction

Input data for one-dimensional steady state heat conduction analysis

```

Total number of NODES    = 17
Total number of ELEMENTS = 16
Total number of MATERIALS = 3
Total number of LOAD SETS = 2
Total number of NODAL SPECIFICATIONS = 2

```

MATERIAL DATA:

```

-----
material      thermal
number        conductivity
-----
1             1.800E+01
2             1.000E+01
3             4.000E+01

```

THERMAL LOAD SET DATA:

```

-----
load set      source      source
number        S1(x)        S2(x)
-----
1             0.000E+00    0.000E+00
2             1.250E+01    0.000E+00

```

NODAL SPECIFICATIONS:

```

-----
node

```

number	specification
-----	-----
1	specified temperature = 3.000E+02
17	specified temperature = 2.750E+02

NODE POINT DATA:

node number	nodal coordinate
-----	-----
1	0.000E+00
2	3.000E-02
3	6.000E-02
4	9.000E-02
5	1.200E-01
6	1.467E-01
7	1.733E-01
8	2.000E-01
9	2.200E-01
10	2.400E-01
11	2.600E-01
12	2.800E-01
13	3.000E-01
14	3.200E-01
15	3.467E-01
16	3.733E-01
17	4.000E-01

ELEMENT DATA:

element number	node 1	node 2	material number	load set number	area
-----	-----	-----	-----	-----	-----
1	1	2	1	1	1.000E+00
2	2	3	1	1	1.000E+00
3	3	4	1	1	1.000E+00
4	4	5	1	1	1.000E+00
5	5	6	2	1	1.000E+00
6	6	7	2	1	1.000E+00
7	7	8	2	1	1.000E+00
8	8	9	2	2	1.000E+00
9	9	10	2	2	1.000E+00

10	10	11	2	2	1. 000E+00
11	11	12	2	1	1. 000E+00
12	12	13	2	1	1. 000E+00
13	13	14	2	1	1. 000E+00
14	14	15	3	1	1. 000E+00
15	15	16	3	1	1. 000E+00
16	16	17	3	1	1. 000E+00

OUTPUT DATA:

node number	nodal coordi nate	nodal temperature
-----	-----	-----
1	0. 000E+00	3. 000E+02
2	3. 000E- 02	2. 985E+02
3	6. 000E- 02	2. 971E+02
4	9. 000E- 02	2. 956E+02
5	1. 200E- 01	2. 942E+02
6	1. 467E- 01	2. 919E+02
7	1. 733E- 01	2. 895E+02
8	2. 000E- 01	2. 872E+02
9	2. 200E- 01	2. 855E+02
10	2. 400E- 01	2. 837E+02
11	2. 600E- 01	2. 820E+02
12	2. 800E- 01	2. 802E+02
13	3. 000E- 01	2. 785E+02
14	3. 200E- 01	2. 767E+02
15	3. 467E- 01	2. 762E+02
16	3. 733E- 01	2. 756E+02
17	4. 000E- 01	2. 750E+02

el ement number	center	heat flux	heat flux per unit area
-----	-----	-----	-----
1	1. 500E- 02	8. 718E+02	8. 718E+02
2	4. 500E- 02	8. 718E+02	8. 718E+02
3	7. 500E- 02	8. 718E+02	8. 718E+02
4	1. 050E- 01	8. 718E+02	8. 718E+02
5	1. 333E- 01	8. 718E+02	8. 718E+02
6	1. 600E- 01	8. 718E+02	8. 718E+02
7	1. 867E- 01	8. 718E+02	8. 718E+02
8	2. 100E- 01	8. 719E+02	8. 719E+02
9	2. 300E- 01	8. 722E+02	8. 722E+02
10	2. 500E- 01	8. 724E+02	8. 724E+02
11	2. 700E- 01	8. 725E+02	8. 725E+02
12	2. 900E- 01	8. 726E+02	8. 726E+02
13	3. 100E- 01	8. 726E+02	8. 726E+02
14	3. 333E- 01	8. 726E+02	8. 726E+02

15	3. 600E- 01	8. 726E+02	8. 726E+02
16	3. 867E- 01	8. 726E+02	8. 726E+02

>>> END- OF- FILE encountered during read <<<

APPENDIX A

List of Significant Variables and Arrays Used in the Program

- *Variables controlling the maximum size of a problem:*

MAXBND = maximum size of the half-bandwidth of the global conductivity matrix.

MAXELM = maximum number of elements permitted in an analysis.

MAXLOD = maximum number of records describing load set data.

MAXMAT = maximum number of material types.

MAXNOD = maximum number of node points permitted in an analysis.

MAXSPE = maximum number of records describing nodal specifications.

- *Logical unit numbers (LUN's):*

NUMIN = LUN associated with input.

NUMOUT = LUN associated with output.

- *Variables controlling the maximum size of a problem:*

NUMELM = number of elements in the mesh being analyzed.

NUMLOD = number of records used to describe load sets associated with the present analysis.

NUMMAT = number of material associated with the present analysis.

NUMNOD = number of node points associated with the present analysis.

NUMSPE = number of records used to describe node point specifications.

- *Integer arrays:*

IFLAG : array defining the type of specification at a given node.

LODNUM : array storing the load set number associated with a given element.

KOUNT : keeps track of which nodes and/or elements have been considered in generation.

MATNUM: array storing the material number associated with a given element.

NOD : array containing the numbers of the two nodes associated with a given element.

NODSPE : array containing the number of the nodes at which specifications have been made.

- *Real arrays:*

ALPHA : array containing the values of the coefficient of thermal expansion associated with a material type.

AREA : array containing the cross-sectional area of each element.

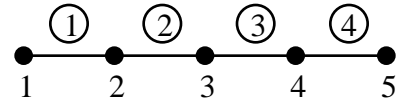
CONDOC : array containing the conductivity associated with a material type.

DELTAT : array storing the value of temperature change associated with a given load set.
ELEMF : element right hand side vector.
ELEMK : element coefficient matrix.
GLOBF : system (global) right hand side vector.
GLOBK : system (global) coefficient matrix.
SOLVEC : solution vector (i.e., the vector containing values of the primary dependent variable).
SOURC1 : array storing the value of a source term associated with a given load set.
SOURC2 : array storing the value of a source term associated with a given load set.
VALUE : array storing the value of a quantity specified at a node.
X : array containing the coordinates of the node points.

APPENDIX B

Scheme Used in Storing the System Arrays GLOBK and GLOBF

Consider the following simple finite element grid:



The “standard” way for storing the entries of the system arrays GLOBK and GLOBF is as follows:

$$\text{GLOBK} = \begin{bmatrix} K_{11} & K_{12} & 0 & 0 & 0 \\ K_{12} & K_{22} & K_{23} & 0 & 0 \\ 0 & K_{23} & K_{33} & K_{34} & 0 \\ 0 & 0 & K_{34} & K_{44} & K_{45} \\ 0 & 0 & 0 & K_{45} & K_{55} \end{bmatrix}; \quad \text{GLOBF} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{bmatrix}$$

NOTE: The half-bandwidth for the above example equals *two*.

In SS1D, the following symmetric, banded form is used to store elements of GLOBK in an array having dimensions (NUMNOD x NBW) :

$$\text{GLOBK} = \begin{bmatrix} 0 & K_{11} \\ K_{12} & K_{22} \\ K_{23} & K_{33} \\ K_{34} & K_{44} \\ K_{45} & K_{55} \end{bmatrix}$$

The system right hand side vector GLOBF remains unchanged.