'FEM1' USER'S GUIDE *: version k*

The application ***'FEM1'*** carries out finite element static analyses of two-dimensional linearly elastic objects in a state of plane stress or strain, and enables the objects to be plotted. It is based on the FORTRAN code appearing in *'Introduction to the Finite Element Method'*, Desai & Abel (van Nostrand 1971), with major enhancements which include object plotting, subdivision of all elements (not just quadrilaterals) into constant strain triangles, use of a more efficient equation solver, nodal load reconstitution, support of tractions normal to element edges, and so on. The subdivision of an element into constant strain triangles - with a common centroidal node which is statically condensed before assembly of the element equations into the network - allows closed-form, as opposed to numerical, integration to be carried out.

Loading of the assembly is by prescribed displacements (zero in the case of supports) and concentrated forces at the nodes, by body forces such as weight, and by linearly varying inter-node tractions which the program replaces by equivalent concentrated nodal forces.

In order to optimise memory usage, the Pascal source organises data dynamically into handle and pointer lists rather than by the fixed vectors so characteristic of FORTRAN - however available memory is still limited. The program defines two constants - ***'maxnodes'*** the maximum number of nodes, and ***'maxsemibandwidth'*** the maximum semi-bandwidth of the assembly equations. These are set at 250 and 72 respectively in the complied application, but may be altered (within the memory's capabilities) in the Pascal code which must then be recompiled.

The program pays scant regard to execution time, preferring to use easily understandable coding - so if one tries to further subdivide a network in an attempt to improve the accuracy of the finite element approximation, then round-off errors and time constraints could be just as significant considerations as the node and bandwidth limitations mentioned above.

The program consists of three major suites of procedures :

INPUT This unit reads assembly-defining data from a user-prepared file and checks the data for consistency, whilst setting up pointer lists of material, node and loading records, together with a scratch file of element records. If desired, node and element input data may be echoed to assist data file debugging. Gross errors are picked up but more subtle mistakes like mis-assigning a certain material to a particular element, cannot be detected.

SOLVE Each element is subdivided into constant strain triangles, the centroidal node is statically condensed and the eliminated equations written to scratch for eventual determination of the centroidal stress; the element stiffness is then assembled into the system equilibrium equations which are stored in banded form via handles. When assembly is complete, the equations are reduced by the specified nodal loads and displacements, then solved for the nodal displacements. These results are output, together with the nodal loads which are reconstituted from them to indicate the ***numerical*** accuracy of the solution. In addition to these nodal results the stress components at the elements' centroids are output.

DRAW The two-dimensional network including supports and loads may be drawn in the undeformed state after INPUT has executed - drawing may be slowed down if desired to aid debugging. Following completion of SOLVE, the network may either be drawn in the deformed state with a user-selected displacement multiplier, or the elements shaded according to their centroidal equivalent stresses - an element whose centroidal stress is greater than the user-specified maximum contour is shaded black.

The program requires the user to first subdivide the two-dimensional object into an assembly of three- and/or four-noded elements, ***ensuring that the number of nodes and the semi-bandwidth of the assembly do not exceed the maxima defined in the program*** as mentioned above. The user then writes the resulting network data - material, geometric and loading - to a data file which is subsequently read by the program. File details are as follows.

Data File Preparation

An existing data file such as *FEX00* should be copied to the user's area, renamed and altered to describe the object using a text editor such as *Edit* or *SimpleText* **exactly** to the following layout :

Consistent units must be employed throughout the data file

first line - the problem title (including the units used for the data, for future information)
next line - either 'plane stress' or 'plane strain' (12 characters), as the case may be
one line for each material :
- material index, elastic modulus, Poisson's ratio, weight density, thickness

one line - '0' : this single zero terminates the lines of material properties
one line for each node, in ascending sequence 1,2,3 etc; or interpolate (see below) :
- node index, x-coordinate, y-coordinate

one line - '0' : this single zero terminates the lines of nodal parameters
one line for each element, in ascending sequence 1,2,3 etc; or generate (see below) :
- element index, material index, three or four nodal indices (no blank at end of line)

one line - '0' : this single zero terminates the lines of elemental definitions
one line for each specified nodal displacment, or load, or inter-node traction (no leading blank) :
- type identifier (see below), node index / indices, specified value / values

one line - '0' : this single zero signifies the end of transmissible data; further notes may follow this marker, but are ignored by the program.

Notes :

Coordinates and vectors described in a user-defined Cartesian system (counter-clockwise x-to-y).

Usually a separate line is required for each node, however if a group of sequential nodes is collinear and equi-spaced then only the first and last nodes of the group need be included in the file; the program interpolates intermediate nodes.

An element is defined by three or four nodal indices, counter-clockwise around it.

A separate line is usually necessary for each element, however if a group of elements is such that the node indices each increase by one within the group (eg. 5-2-13-9, 6-3-14-10, 7-4-15-11) then only the first element of the group needs be included in the file; the program will automatically generate the other elements of the group, assigning the material of the first member to them. The last element of the assemblage must always be supplied however.

Specified nodal displacements and loads are identified by a two-character 'type' code :

First character, one of	Second character, one of . . . . .
'd' specified nodal displacement	'x' in the x-direction
'f' specified nodal concentrated force	'y' in the y-direction
't' specified inter-node traction (force per unit edge length, ie thickness∗pressure or ∗stress)	'n' normal to the edge, towards the body (this code is relevant only to an inter-node traction)

A fixed support is thus represented by two lines, type 'dx' and 'dy', at the same node and of zero value. Tractions are specified only between neighbouring nodes.

Jim's Example

This example is presented solely as an illustration of data file preparation - its mesh is far too coarse for accurate representation of the prototype.

The geometry of the cantilevered flat plate, Figure 1 below, is known. The plate is made in two thicknesses from a material with elastic modulus 50 GPa and Poisson's ratio 0.25. The density is

unspecified, so presumably weight forces are negligible.
The plate is discretised into the 11-noded, 6-element mesh of Figure 2, in which the edges 1-2 and 2-3 are of equal length, and the four elements 2,3,5,6 are identical parallelograms. The non-uniform pressures of the prototype are transformed into the equivalent edge tractions shown in Figure 2. Node 10 suffers a prescibed displacement, and node 11 is loaded by an inclined force whose components are sketched. The sliding supports at nodes 5 and 9 are justified by knowledge of the support of the prototype.

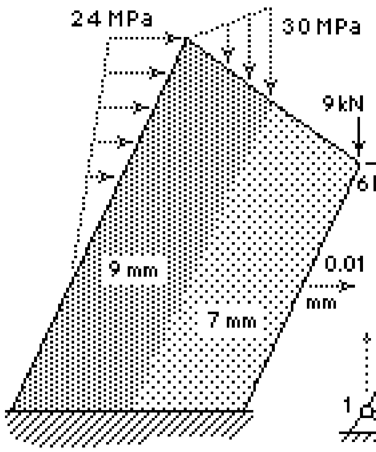

Figure 1

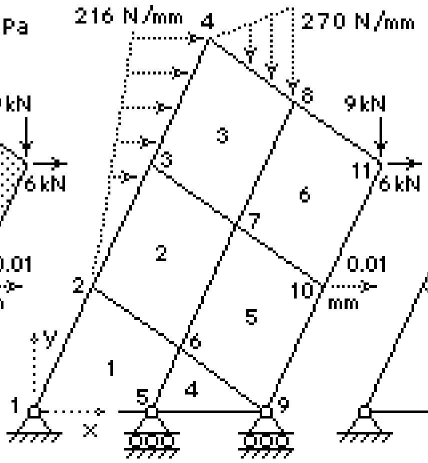

Figure 2

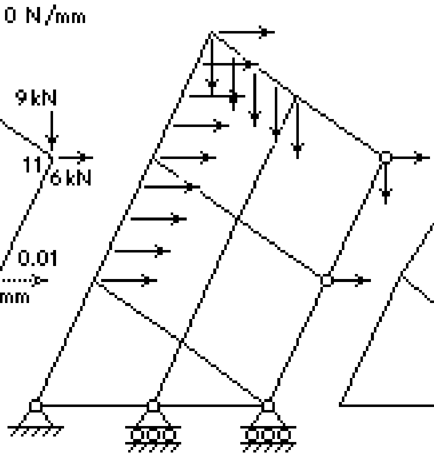

Figure 3

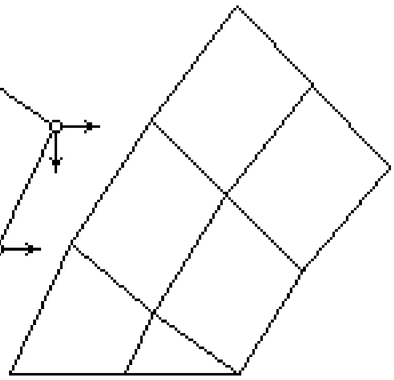

Figure 4

The data file for this problem, *FEX00,* appears as follows with italicised comments added:

Jim's Example (N, mm)						*Units stated for later ease of interpreting stress results*
plane stress						
1	50000	0.25	0.0	9		*Notice the use of consistent units*
2	50000	0.25	0.0	7		
0						*Zero material-list-terminator*
1	0	0				*Nodal index and coordinates*
4	48	102				*Nodes 2 and 3 will be interpolated . . . .*
5	32	0				
6	40	17				*. . . . as will node 7 . . . .*
8	72	85				
9	64	0				*. . . . and node 10*
11	96	68				
0						*Zero node-list-terminator*
1	1	1	5	6	2	*Element index, material and counterclockwise node indices*
4	2	5	9	6		*Elements 2 and 3 will be generated - see output*
5	2	6	9	10	7	*Ensure that there is no trailing blank at end of an element-line*
6	2	7	10	11	8	*The last element of the assembly must be included*
0						*Zero element-list-terminator*
dx	1	0				*Specified nodal value : type, node and value - the first two lines refer to the fixed support*
dy	1	0				
tx	2	3	0	108		*Specified inter-nodal edge traction : type, indices of the neighbouring node pair and the corresponding values of the traction at the two nodes (no leading blanks)*
tx	3	4	108	216		
ty	4	8	0	-270		
dy	5	0				
dy	9	0				
dx	10	0.01				*Specified nodal displacement . . . .*
fx	11	6000				*. . . . and concentrated force components*
fy	11	-9000				
0						*Zero end-of-data indicator*

Care should be exercised to ensure that the data file is open only to one application at a time.

Program Operation

Initiating the program leads to a call for selection of the input data file. The user is then asked to define an output filename on a write-enabled disc - this name becomes the basis for :

- a text echo file, which saves the contents of the text window,
- two trashable scratch files, which are empty on leaving the application, and
- three files for saving drawings of the undeformed, deformed and patterned mesh.

The input data is checked for consistency by INPUT, after which the DRAW or SOLVE options may be selected; a typical dialogue runs as follows, user-responses being underlined :

```
********                                                        17: 3:97
FEM1k              Finite Element Analysis in Two-dimensional Linear Elasticity
********                               with 250*72 maximum nodes*semibandwidth
```

FEX00 : Jim's Example (N,mm) — *Stress units will subsequently be N/mm^2, ie. MPa*

Echo nodes & elements ? (y/n) : n — *'y' enables data verification, stepped through with button*

Plane stress - with 11 nodes, 6 elements, and a semi-bandwidth of 12

op ? : d — *Draws the undeformed net, Figure 3, and saves to file*

Enter drawing retardation 0..4 : 2 — *Slowing down drawing execution aids checking for data errors*

op ? : s — *Solves the network, and writes results to text echo file*

Node	Location x	Location y	Displacement x	Displacement y	Force x	Force y	
1	0.0	0.0	0.0000	0.0000	3239.97	-9576.94	
2	16.0	34.0	0.0484	0.0657	676.38	-0.00	
3	32.0	68.0	0.2771	0.0616	4058.26	0.00	*Depress button*
4	48.0	102.0	0.5832	-0.0459	3381.90	-1323.49	*to suppress*
5	32.0	0.0	-0.0198	0.0000	0.00	-4277.10	*screen output*
6	40.0	17.0	-0.0192	-0.0054	-0.00	0.00	*of results though*
7	56.0	51.0	0.1453	-0.0861	0.00	-0.00	*they are still*
8	72.0	85.0	0.4413	-0.2410	-0.01	-2646.98	*written to text*
9	64.0	0.0	-0.0455	0.0000	-0.00	26824.51	*echo file*
10	80.0	34.0	0.0100	-0.2318	-17356.52	-0.01	
11	96.0	68.0	0.3267	-0.4787	6000.00	-9000.00	

Element & mat'l	Centroid x	Centroid y	Stress - components x	y	shear	- and principals max	min	deg	equivalent
1 :1	22.00	12.75	-39.17	60.49	9.66	61.42	-40.10	84.5	88.6
2 :1	36.00	42.50	6.66	35.34	34.41	58.27	-16.27	56.3	67.9
3 :1	52.00	76.50	17.88	-14.15	24.98	31.53	-27.81	28.7	51.4
4 :2	45.33	5.67	-35.22	-20.78	6.22	-18.47	-37.53	69.6	32.5
5 :2	60.00	25.50	-75.18	-139.55	-20.36	-69.28	-145.45	-16.2	126.0
6 :2	76.00	59.50	25.58	-74.20	-10.66	26.70	-75.33	-6.0	91.6

op ? : d — *Draws and saves the deformed network, Figure 4, nodal*

Enter displacement multiplier : 25 — *displacements being magnified by the multiplier.*

op ? : p — *Patterns the undeformed elements according to centroidal von*

Maximum equivalent stress is : 126 — *Mises equivalent stress (not shown here) - black if greater than*

Enter desired maximum contour : 100 — *the maximum contour, then dark grey, etc. The plot is saved.*

op ? : q — *Quit (Option 'h' lists the options available).*

Clicking the mouse button enables exit from the drawing window to the text window.