

INELASTIC BENDING MOMENTS, END LOADS, STRESSES AND STRAINS FOR BEAM SECTIONS OF GENERAL POLYGONAL SHAPE

1. APPLICATION AND DERIVATION OF PROGRAM

1.1 Introduction

This Data Item is arranged in three sections. Section 1 gives information on the application of the program of ESDUpac A9026, and a summary of the theory from which the program is derived. Section 2 gives the instructions necessary for operating the program, guidance in interpreting the output and examples of input to and output from the program. Section 3 provides notes on the program structure and details of the tests that have been performed on the program.

The Data Item, together with ESDUpac A9026, presents a Fortran computer program for calculating the combination of bending moments and end load in a beam that is associated with a defined maximum allowable tensile or compressive extreme fibre stress or strain which is taken to be the same in both tension and compression. The program also provides a stress and strain analysis of the beam under such calculated loading or for a specified loading combination. The program offers the following choice of five categories of analysis for a beam cross section loaded by two orthogonal moments and an (offset) end load.

- (a) The two limiting values of one unknown load for two specified loads.
- (b) The two limiting values of two unknown loads, defined in ratio to each other, for one specified load.
- (c) The two limiting values of one unknown load for one specified load.
- (d) The stress and strain distribution for three specified loads.
- (e) The stress and strain distribution for two specified loads.

In categories (a), (b) and (d) the beam is assumed to be free to deflect in any direction. In categories (c) and (e) the beam is assumed to be constrained so that deflection in the x-direction is prevented. In categories (a), (b) and (c) the distribution of stress and strain can be presented for intermediate values of bending moment or end load at chosen increments between the limiting values. Also in these categories an elastic solution is given by assuming that stress is directly related to strain by the material modulus, E . In all categories the location of the neutral axis, the beam cross-sectional area, the position of the centroid and the other geometrical cross-section properties are given.

The beam is of general solid or hollow polygonal section and the allowable stress or strain may be above the limit of proportionality of the beam material. It is assumed that the behaviour of the beam material above the limit of proportionality can be represented by the generalised form of stress-strain curve given in Data Item No. 76016*. This curve is assumed to be applicable throughout the loading process. It follows that the program is only valid if the loading sequence is such that there is no unloading at any point in the cross section from a stress that is above the elastic limit since the elastic recovery would imply a different stress-strain curve. For example this condition will be satisfied: (a) if all the loads in any calculated combination are applied simultaneously or (b) if the known (input) loads produce only elastic stresses or (c) the stresses due to the unknown (found) loads numerically increase the stresses due to the known (input) loads.

* Data Item No 76016 "Generalisation of smooth continuous stress-strain curves for metallic materials".

The analysis assumes that the beam material is isotropic and homogeneous. The strain is considered to vary linearly across the section with plane sections remaining plane, but the beam may be stressed beyond the limit of proportionality to an extent that may be defined by a maximum strain or stress. The stress-strain curve of the material, assumed identical in tension and compression, is defined by the modulus together with a choice of two proof stresses, or by the material coefficient m and a reference stress f_n or by two values of total inelastic strain and their corresponding stresses. A summary of the theoretical basis of the program is given in Section 1.3.

The beam cross section may be subject to two orthogonal moments and an end load. The moments and end load may be entered (known) or left to be determined by the program (unknown) through a choice of six forms of load input to give analyses of the categories (a) and (b) above (see options 1 to 6 in Table 2.1). These analyses take account of the movement of the neutral axis from the elastic position resulting from the influence of the inelastic stress-strain properties for the particular loading condition. Two further input options (options 7 and 8 in Table 2.1) provide analyses of category (c) above which allow the beam to be constrained so that deflection in the x -direction is prevented (the neutral axis as a result remaining parallel to the x -axis). Finally two input options are given that perform a stress and strain analysis for specified values of end load and moment with the beam free (category (d), option 9) or constrained in the x -direction (category (e), option 10). Notes on the application of the program are given in Section 1.4.

No account has been taken of any type of instability, the possibility of which must also be considered by separate calculation.

The analysis provided by this Data Item is complementary to that provided by Data Item No. 90004* which calculates the combination of bending moments and end load that are associated with a defined maximum allowable extreme fibre stress or strain but with the components of the loading in fixed ratios to each other as are those of the input loading.

1.2 NOTATION

Both S.I. and British Units are quoted but any coherent system of units may be used.

A	area of cross section of beam	m^2	in^2
E	modulus of elasticity of material	N/m^2	lbf/in^2
E_t	tangent modulus of material	N/m^2	lbf/in^2
f	stress	N/m^2	lbf/in^2
f_{max}	maximum allowable stress	N/m^2	lbf/in^2
f_n	stress at which $E_t = E/2$	N/m^2	lbf/in^2
f_t	specification ultimate tensile stress of beam material*	N/m^2	lbf/in^2
f_1, f_2	stresses provided at any two points on inelastic region of stress-strain curve	N/m^2	lbf/in^2

For footnotes to the Notation see end of Notation

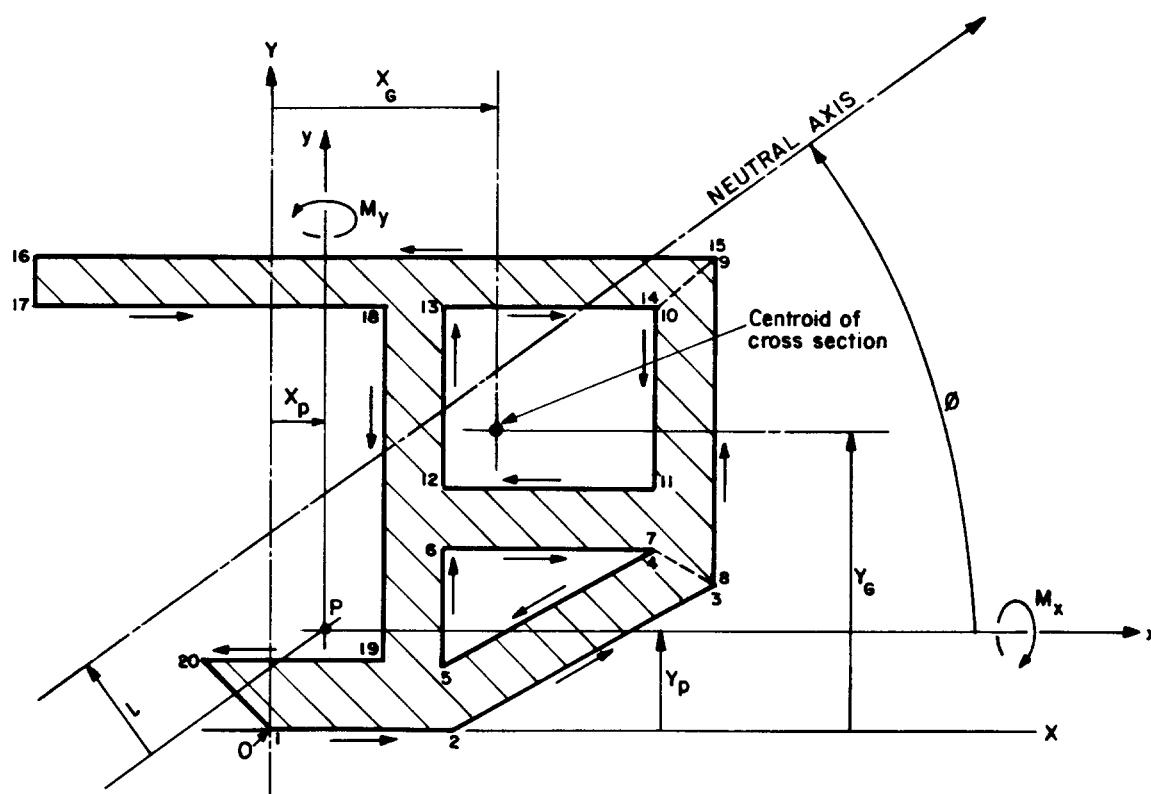
* Data Item No. 90004 "Maximum inelastic bending moments and end loads (in fixed ratio) for beam sections of general polygonal shape".

I_x	second moment of area of beam cross section about centroid parallel to x -axis	m^4	in^4	
I_y	second moment of area of beam cross section about centroid parallel to y -axis	m^4	in^4	
I_{xy}	product of inertia of beam cross section about centroid	m^4	in^4	
I_1, I_2, I_3	integrals used in beam analysis (see Section 1.3)	N	lbf	
m	material characteristic (see Data Item No. 76016)			
l	perpendicular distance from point of application of end load to neutral axis (see Sketch 1.1)	m	in	
M_x, M_y	applied moments about x - and y -axes, respectively	N m	lbf in	
M_{xlim}	moment about x -axis with specified stress or strain at extreme fibre	N m	lbf in	
M_{ylim}	moment about y -axis with specified stress or strain at extreme fibre	N m	lbf in	
N	total number of vertex coordinate sets given for polygonal beam cross section			
P	applied end load acting at any point on cross section	N	lbf	
P_{lim}	end load on section with specified stress or strain at extreme fibre	N	lbf	
R	radius of curvature of deflected beam	m	in	
t_A, t_B	proof stresses $t_A < t_B$	N/m ²	lbf/in ²	
t_1	0.1 per cent tensile proof stress	} related to stress-strain curve representing beam material	N/m ²	lbf/in ²
t_2	0.2 per cent tensile proof stress		N/m ²	lbf/in ²
t_{2spec}	specification value of 0.2 per cent proof stress of beam material*	N/m ²	lbf/in ²	
U_1, U_2, U_3	non-dimensional forms used in program, related to l , R and ϕ			
$O(X, Y)$	coordinate system for geometry	} parallel, see Sketch 1.1		
$P(x, y)$	coordinate system for loading			
α, β	coefficients in strain equation (see Section 1.3)	m^{-1}	in^{-1}	
γ	coefficients in strain equation (see Section 1.3)			
ϵ	strain			

For footnotes to the Notation see end of Notation

ϵ_{max}	maximum total strain in beam material		
$\epsilon_{pA}, \epsilon_{pB}$	permanent (proof) strains corresponding to t_A and t_B , respectively		
$\epsilon_{p1}, \epsilon_{p2}$	permanent (proof) strains corresponding to t_1 and t_2 , respectively		
ϵ_1, ϵ_2	total strains appropriate to stresses f_1 and f_2 , respectively (not necessarily t_1 and t_2)		
ϕ	angle between neutral axis and x -axis	degree	degree
X_G, Y_G	coordinates of position of centroid	m	in
X_P, Y_P	coordinates of point of application of end load	m	in

* Alternatively, a statistically derived value of design allowable stress such as a value from Reference 5 or 6 may be used.



Sketch 1.1 Coordinate system, loading sign convention and typical identification of vertices.
All quantities are shown positive.

Moments M_x and M_y are positive as shown. Both give tension stress in the positive xy quadrant. End load P is positive in tension, negative in compression.

The cross section is defined by the coordinates of continuously connected vertices numbered in sequence in the direction which keeps the material of the beam to the left. Holes are identified by departing from and rejoining the external geometry through the same adjacent vertices (see vertices 3 and 4, 7 and 8). Arrows show the direction of the numbering sequence.

The arrow on the neutral axis indicates the direction of viewing for the sign of l . If l is positive, point P (coordinates X_P, Y_P) lies on the right-hand side of the neutral axis when looking along the neutral axis in the direction of the arrow shown. The angle ϕ lies in the range -90 to $+90$ degrees. The value of R is defined as positive when the stress changes from positive to negative on passing from the left-hand to the right-hand side of the neutral axis.

1.3 DERIVATION

1.3.1 Sources

The non-linear inelastic analysis on which the program of ESDUpac A9026 is based was developed by ESDU and is detailed in Derivation 1. A summary of the theory is given below.

1.3.2 Theory

The analysis is based on first order beam theory with non-linear material behaviour. Bending moments and end load are determined that give the maximum allowable extreme fibre stress or strain specified by the user. It is assumed that the strain distribution under the action of the bending moment is linear across the section and plane sections remain plane, but that the stress distribution may be non-linear.

The cross section is defined by the coordinates of continuously connected vertices numbered in sequence in the direction that keeps the material of the beam to the left. Holes should be numbered from a convenient vertex in the external geometry to which the sequence must return (see Sketch 1.1).

The end load P and moments M_x, M_y about axes P_x, P_y , are

$$P = \int_A f dA, \quad (1.1)$$

$$M_x = \int_A fy dA \quad (1.2)$$

and $M_y = \int_A fx dA, \quad (1.3)$

where f is the stress at (x, y) and integration is over the cross-sectional area, A .

The strain is expressed as

$$\varepsilon = \alpha x + \beta y + \gamma. \quad (1.4)$$

The stresses are obtained from the strains using the inelastic stress-strain relationship

$$\varepsilon = \frac{f}{E} \left[1 + \frac{1}{m} \left(\left[\frac{f}{f_n} \right]^2 \right)^{\frac{m-1}{2}} \right], \quad (1.5)$$

which is derived from Equation (3.1) of Data Item No. 76016.

The analytical method used replaces Equations (1.1) to (1.3) by the relations

$$P = I_1, \quad (1.6)$$

$$\alpha M_y + \beta M_x + \gamma P = I_2 \quad (1.7)$$

$$\text{and} \quad \beta M_y - \alpha M_x = I_3, \quad (1.8)$$

where I_1 , I_2 and I_3 are integrals that can be evaluated in closed form using Equation (1.5) and the cross section geometry (that is, the vertex coordinates), see Derivation 1. For a side along which there is relatively small change in stress, the closed form evaluation fails to provide sufficient numerical accuracy. In such cases this is automatically replaced by an alternative formulation that enables computing accuracies to be maintained but requires numerical integration by Simpson's rule.

The analyses used to determine the loading that satisfies the criterion of the maximum (inelastic) allowable stress at the extreme fibre (Section 1.1 analyses (a), (b) and (c)) vary slightly according to the load input option (options 1 to 8) chosen and the unknown loading quantities required. The program starts by using elastic theory to perform initial elastic analyses. The elastic neutral axis is then used as a starting point to perform inelastic analyses that use an iterative technique to determine the values of the unknown quantities appropriate to the known loading input, taking account of the movement of the neutral axis under the influence of the inelastic material properties. The analyses have been used to provide solutions to the above equations for eight loading options as detailed in Table 2.1. In the first six options the beam is assumed to be free to deflect in both the x- and y-directions, that is, the neutral axis is completely free to move under the influence of the loading and the stress-strain relationship. In options 7 and 8 the beam is assumed to be constrained so that deflection in the x-direction is prevented, that is, the neutral axis is parallel to the x-axis. The value of M_{ylim} (corresponding to P_{lim} and M_{xlim}) required to constrain the beam in this condition is given in the output.

For options 9 and 10 stress and strain distributions with the beam free or constrained are provided, as shown in Table 2.1.

Within the program, calculations are performed relative to the centroid rather than the point of application of load P .

1.3.3 Derivation and references

Derivation

This section lists selected sources that have assisted in the preparation of this Data Item.

1. ESDU Theoretical analysis of Data Item No. 90026. ESDU Memorandum No. 75, September 1990.

References

The references given are recommended sources of information supplementary to that in this Item.

2. COZZONE, F.P. Bending strength in the plastic range. *Jl aero. Sci.*, Vol. 10, No. 5, pp. 137-151, May 1943.
3. BARRETT, A.J. The bending of some common beam sections into the plastic range. *Jl R. aeronaut. Soc.*, Vol. 57, pp. 110-115, February 1953.
4. DWIGHT, J.B. An investigation into the plastic bending of aluminium alloy beams. The Aluminium Development Association, RR. No. 16, London, May 1953,
5. ESDU Metallic materials data handbook. Defence Standard 00-932, ESDU International plc.
6. USA, DoD Military standardization handbook. Metallic materials and elements for aerospace vehicle structures. MIL-HDBK-5. Department of Defense, Washington, D.C., USA.

1.4 Application of the Program of ESDUpac A9026

The principal purpose of the program is to provide solutions for combinations of maximum moments and end load that may be applied to a beam of any polygonal cross section (see Table 2.1, options 1 to 8). The program will also fulfil the additional purpose of providing a stress and strain analysis for a specified combination of particular moments and end load (see Table 2.1, options 9 and 10).

The combinations of limiting moments, M_{xlim} and M_{ylim} , and end load, P_{lim} , determined by inelastic analysis, are those appropriate to a maximum stress (f_{max}) or strain (ϵ_{max}), tensile or compressive (identical) as defined by the user, at extreme fibres of the cross section. The program determines the limiting values of the unknown quantities chosen from M_{xlim} , M_{ylim} , and P_{lim} that combine with known (input) values of the remaining quantities to give the specified maximum stress or strain at the extreme fibre.

The results are always correct if the loads, known (input) and unknown (determined), are applied simultaneously and will be correct in certain circumstances when the unknown loads determined by the program are applied after the known (input) loads. These are circumstances in which the application of the unknown loads after the known loads does not result in the stresses anywhere in the cross section being unloaded from an inelastic stress. Such unloading would be down a stress-strain curve offset from the original by a degree of permanent strain and the program cannot account for this. The program does however indicate when no such stress unloading from inelastic stress occurs in the cross-section so that results are valid if the unknown (determined) loads are applied after the known (input) loads. In the program consecutive application of the loads will give valid results when all the vertices satisfy the conditions that the stress at any vertex is either (a) elastic under application of the known (input) loads alone or (b) numerically greater under the known (input) plus unknown (determined) loading than under the known (input) loading alone. In the program the judgement of "elastic", (a), is when the stresses are less than or equal to the 0.01 per cent proof stress.

In addition to the limiting load combination the program also gives the location of the neutral axis, the radius of curvature of the deflected beam and the stress and strain at each vertex under the limiting loads. The program also provides the corresponding data for an elastic solution appropriate to t_{2spec} at the extreme fibre, an inelastic stress-strain analysis with the beam subjected to the known (input) loads only and additional inelastic analyses for intermediate loading levels at chosen increments between the two limiting

values determined. Additionally, values are given for the area of the cross section, its second moments of area and the position of the centroid.

The maximum stress or strain at the extreme fibre may be defined directly by the user who may alternatively choose to use the default value equal to the 0.5 per cent proof strain at the extreme fibre. Values higher than this should be used with caution since there may be problems when designing against proof requirements; also high shear stresses can occur. The program is applicable to any beam cross section that can be defined by line-linked points and for which the material modulus of elasticity and any two inelastic points on, or characteristics of, the material stress-strain curve are known.

Table 2.1 gives ten loading options that fall into five categories of analysis, (a) to (e). In category (a) the program determines values for one unknown load chosen from P_{lim} , M_{xlim} and M_{ylim} when values of the other two are specified. This analysis is used for input options 1, 2 and 3. In category (b) the program determines the actual values for two unknowns chosen from P_{lim} , M_{xlim} or M_{ylim} when one is specified together with the ratio of the remaining two. This analysis is used for input options 4, 5 and 6. In the analyses of categories (a) and (b) the beam is completely free to move in the x - and y -directions. For the third category (c), input options 7 and 8, the beam is constrained so that deflection in the x -direction is prevented (the neutral axis as a result remaining parallel to the x -axis). The analyses of category (c) are the constrained equivalents of (a), and one of the unknown limiting loads, P_{lim} and M_{xlim} , is determined when the other is specified. A value is also given for the constraining moment M_{ylim} . The two further categories (d) and (e), input options 9 and 10, provide stress and strain distributions for specified values of P , M_x and M_y (option 9) when the beam is free and P and M_x (option 10) when the beam is constrained so that the deflection in the x -direction is prevented.

The output of the program for options 1 to 8 comprises:

- (i) an input data check
- (ii) an elastic analysis (elastic solutions 1 and 2)
- (iii) an inelastic analysis with (a) the input loads only applied (inelastic solution 1), (b) input plus the unknown loads applied (inelastic solutions 2 and 3) and (c) intermediate loads applied (inelastic solutions 4 onwards).

The input data check includes a print out of all the quantities in Table 2.2. The elastic analyses will only be given if it is possible to obtain combinations of the known (input) loads and unknown (determined) loads that give stresses less than t_{2spec} in the cross section. It is provided for comparison with the inelastic solutions.

The first inelastic solution (solution 1) provides one part of the basis for judging whether the results are valid with loads applied consecutively rather than simultaneously. A message is printed out according to the judgement made on this as detailed in the third paragraph of this section.

The principal inelastic solutions provide two limiting values of the unknown (determined) quantities that, in combination with the known (input) loading, give the maximum allowable stress, f_{max} , at the extreme fibres. Both limiting values may be of the same sign or of opposite sign according to the loading, geometry or option chosen. The sign of f_{max} associated with the solutions may vary similarly. The facility to obtain inelastic analyses at specified intervals between the limiting values is provided so that the rate of variation of the inelastic stress distribution may be studied between those values. If the loading levels input prevent an inelastic solution being obtained a message will be given. Data Item No. 90004 used with the same known (input) loads and f_{max} may give an indication of the level to which the loading should be reduced to obtain a solution.

The stress and strain analyses provided by input options 9 and 10 are not limited by the input values of

ϵ_{max} , f_{max} or the 0.5 per cent proof strain (see Table 2.2). However, when the maximum stress or strain in the cross section due to the specified loading exceeds ϵ_{max} or f_{max} a warning will be printed out with the results.

2. ESDUpac A9026

2.1 Introduction

This section of the Data Item gives the instructions necessary to operate the Fortran program of ESDUpac A9026. Section 2.2 details the form of input required for the program. Section 2.3 describes the form of output to be expected and the way in which it should be interpreted. Examples of both input and output are given in Section 2.4.

The program is available in two formats: (a) a Fortran source code with example input and output files, and (b) an executable program for PC's. Both versions can be found on the Compact Disc and on disk in the Sub-series Software Volume.

2.2 Data Input Information

2.2.1 Data input form

To perform an analysis the program requires data to be input defining the form and extent of the material stress-strain curve, the geometry of the cross section and the form and values of loading applied. Table 2.1 lists the loading input options available for the various categories of analysis and Table 2.2 details the input data and the form in which they are to be entered.

The first three lines of the input data files are reserved for the title of the user's analysis. Any lines that are not used here should be left as blank lines in the input data file.

To define the material stress-strain curve it is first necessary to specify two points on the curve or two characteristic quantities. These may be specified in one of the three following forms.

- (1) Any two proof stresses, which are entered by quoting the permanent strain (proof strain) followed by the related proof stress in each case. Note that the permanent strains are not total strains but the offset values (see Data Item No. 76016).
- (2) The generalised stress-strain characteristic values m and f_n as defined in Data Item No. 76016.
- (3) Any two values of total strain with their related stresses.

The modulus of elasticity is required next, followed by values for t_{2spec} and f_t .

The extent of the stress-strain curve is then defined by choosing one of the following entries:

- (1) The maximum strain.
- (2) The maximum stress.
- (3) A default value of 0.5 per cent proof strain.

When (3) is selected, 0 is entered and the program will work to the true strain equivalent of the 0.5 per cent proof strain. Note that positive values should be entered but that the analysis will be performed using

identical positive (tensile) and negative (compressive) stress-strain curves.

When the stress-strain curve has been defined, the data for the beam cross section geometry are entered. The vertices of the polygonal cross section are identified and numbered in sequence in the direction that keeps the material to the left (see Sketch 1.1). Where the section has internal holes, the vertices of the holes (no matter how many) are numbered in a sequence following on from any vertex in the external geometry, to which the numbering must return (see Sketch 1.1). Note this means that the related points in the external and hole geometries where the change occurs will both appear twice in the numbering sequence. Further, the effect of this left hand rule is to move round the external geometry in an anticlockwise direction and to move round holes in a clockwise direction. The vertices are specified with respect to any conveniently chosen coordinates X and Y as shown in Sketch 1.1. The geometry is specified by entering first the total (N) of vertex numbers and then their X , Y coordinates in order. Examples given in Section 2.4.1 illustrate this system.

The form and magnitude of the loading must be entered next. This takes the form of specifying one of the ten options available as shown in Table 2.1. When one of options 1 to 8 are chosen the program will obtain the limiting values of the unknown loads that, in combination with the known (specified) loading, will give the defined maximum allowable stress or strain at the extreme fibre. If options 9 or 10 are chosen a stress/strain analysis will be given for the loads specified. For options 1 to 6 and 9, the beam is assumed to be free to deflect in both the x - and y -directions (the neutral axis being completely free to move). For options 7, 8 and 10, the beam is assumed to be constrained so that deflection in the x -direction is prevented (the neutral axis as a result being kept parallel to the x -axis). Values for the quantities shown in Table 2.1 for the chosen option are entered in the order shown. Finally, the number of intermediate levels of load output are specified (maximum of 10) followed by the value of each level compared to the two extreme values.

Values of the loading P , M_x and M_y entered should use the sign convention given in Sketch 1.1. Specification of units is not required for any input values. All values entered must be based on a coherent system of units. The output values will be compatible with the system chosen. Examples of the use of British Units and the S.I. system are given in Section 2.4.2.

For each option the values of the input loads or ratios, as shown, are entered. Values of the unknown loads determined by the analysis are given as P_{lim} , M_{xlim} and M_{ylim} . Options 9 and 10 give a stress-strain analysis for the loads input as shown.

TABLE 2.1 LOADING OPTIONS

Option	Input (known) loads			Loads Calculated		Condition (cross section not constrained or constrained parallel to x -axis)	Analysis category (see Sections 1.1 and 1.4)
	End load	Moments	Ratio	End load	Moments		
1	—	M_x M_y	—	P_{lim}	— —	} Not constrained	(a)
2	P	M_x —	—	—	— M_{ylim}		(a)
3	P	— M_y	—	—	M_{xlim} —		(a)
4	P	— —	M_y/M_x	—	M_{xlim} M_{ylim}		(b)
5	—	— M_y	P/M_x	P_{lim}	M_{xlim} —		(b)
6	—	M_x —	P/M_y	P_{lim}	— M_{ylim}		(b)
7	P	— —	—	—	M_{xlim} (M_{ylim})	Constrained	(c)
8	—	M_x —	—	P_{lim}	— (M_{ylim})	Constrained	(c)
9	P	M_x M_y	—	Stresses and strains only		Not constrained	(d)
10	P	M_x —	—			Constrained	(e)

2.2.2 Program limitations

The values of the generalised material characteristic m have been limited to the range $2 \leq m \leq 100$. This should embrace all practical values.

The highest vertex number, N , is limited to 50.

The maximum allowable stress may not exceed the ultimate tensile stress so that $f_{max} \leq f_t$. If $f_{max} > f_t$, either from input stress values or implied by ϵ_{max} , the analysis will be terminated. If options 9 or 10 are specified the resulting stresses may exceed f_{max} (and even f_t) but an advisory message will be printed to this effect (see Section 2.3.2, message 15).

The maximum total strain to which the program will work is 15 per cent, that is $\epsilon_{max} \leq 0.15$.

When selecting the vertex with the maximum stress the program works to a maximum accuracy of 0.1 per cent difference in the strains at the compared vertices.

TABLE 2.2 PROGRAM INPUT DATA

ENTRY ORDER	ENTRIES (enter numerical values indicated by notation)
1	Analysis title Enter: Title for the analysis Note: Use first three lines of input file (78 characters per line). Leave blank lines where there is no title.
2	Stress-strain curve selector Enter: 1, 2 or 3 as required. 1 for proof stresses 2 for m and f_n 3 for actual strain and stress values
3	Stress-strain curve definition For selection Enter: (1) $\epsilon_{pA}, t_A, \epsilon_{pB}, t_B$ (2) m, f_n (3) $\epsilon_1, f_1, \epsilon_2, f_2$ Note: ϵ_{pA} and ϵ_{pB} are permanent (proof) strains, t_A and t_B are proof stresses ($t_A < t_B$). (For example if $t_A = t_1$ then $\epsilon_{pA} = 0.001$.) m and f_n are related to the generalised stress-strain curve given in Data Item No. 76016. ϵ_1 and ϵ_2 are total strains, and f_1 and f_2 are their corresponding stresses. Also: $2 \leq m \leq 100$.
4	Material modulus of elasticity Enter: E

TABLE 2.2 PROGRAM INPUT DATA (Continued)

ENTRY ORDER	ENTRIES (enter numerical values indicated by notation)
5	0.2 per cent proof stress and ultimate tensile stress Enter: t_{2spec} , f_t
6	Maximum stress-strain selector Enter: 1, 2 or 3 as required. 1 for maximum strain 2 for maximum stress 3 for default value (0.5 per cent proof strain).
7	Maximum stress-strain For selection Enter: (1) ϵ_{max} (actual strain) (2) f_{max} ($f_{max} \leq f_t$) (3) 0 (this gives 0.5 per cent proof strain) Note: The value of f_{max} entered or implied here must be less than f_t in input 5. The value of ϵ_{max} entered or implied here must be less than or equal to 0.15. It must also be significantly less than the permissible elongation of the material; note this is not tested by the program.
8	Cross-section geometry Number of vertex coordinate sets Enter: NE Note: $N \leq 50$
9	Coordinates of vertices Enter: X_1, Y_1 to X_N, Y_N Note: Vertices are numbered in sequence in accordance with Sketch 1.1.
10	Loading option Enter: 1 to 10 as required (see Table 2.1).
11	Loading Enter: $P, M_x, M_y, P/M_x, P/M_y$ or M_y/M_x as indicated in Table 2.1. Note: see Sketch 1.1 for sign convention.
12	Coordinates of end load Enter: X_P, Y_P
13	Intermediate load values Enter: number of values required (0 to 10).

TABLE 2.2 PROGRAM INPUT DATA (Continued)

ENTRY ORDER	ENTRIES (enter numerical values indicated by notation)
14	Fractions of load values Enter: Value of each intermediate load as a decimal fraction of the difference between the minimum and maximum limiting values Note: The number of entries here must be the same as the number entered at input 13

Note: Entries 13 and 14 are not required if loading options 9 or 10 have been specified.

2.3 Output Data Interpretation

2.3.1 General output form

The output of the program provides first the title of the analysis provided by the user and then a check on the input data. It then presents the results of the analysis. If limitations on the input have been infringed, a warning message will be given as detailed in Sections 2.3.2 or 2.3.3. No units are indicated as the units of the output depend on, and are compatible with, those of the input.

The program title is first given.

This is followed by the

INPUT DATA CHECK

This provides a check on the data that have been input and stress-strain curve properties derived from them under the four following headings:

MATERIAL DATA

Values are given of t_{2spec} , f_t , ϵ_{max} , f_{max} and E .

STRESS-STRAIN CURVE

Values are given for the quantities chosen to define two points on the stress-strain curve, together with the derived values for those options not chosen, in the form shown in Table 2.3.