Best Available Copy

(1)

BEAM: A FINITE ELEMENT PROGRAM FOR THE COLLAPSE ANALYSIS OF VEHICLE STRUCTURES

AR-008-636

S J CIMPOERU

MRL-TR-94-2

APPROVED

FOR PUBLIC RELEASE

MATERIALS RESEARCH LABORATORY

DSTO

BEAM: A Finite Element Program for the Collapse Analysis of Vehicle Structures

S.J. Cimpoeru

MRL Technical Report
MRL-TR-94-6

Abstract

BEAM (Beam Element Analysis with Mechanisms) is a finite element program for the analysis of the collapse of vehicle structures. BEAM can quickly estimate the collapse load and plastic hinge locations in three-dimensional thin-walled frames that would collapse in bending. Such information can be combined with separate rigid-plastic modelling to enable the large-deflection design of vehicle frameworks so that their collapse in roll-over accidents, for instance, can be prevented. One advantage of BEAM is that it allows a user to define his own experimentally or analytically determined loads and moments as failure criteria. The theory and operation of BEAM are presented and its application to the modelling of a collapsing two-dimensional frame is described.

DEPARTMENT OF DEFENCE
DSTO MATERIALS RESEARCH LABORATORY

Published by

DSTO Materials Research Laboratory
Cordite Avenue, Maribyrnong
Victoria, 3032 Australia

Telephone: (03) 246 8111
Fax: (03) 246 8999

AR No. 008-636

APPROVED FOR PUBLIC RELEASE

Author

S.J. Cimpoeru

Stephen Cimpoeru obtained a degree in Metallurgical Engineering from the University of Melbourne and later a Masters of Engineering Science. He joined MRL in 1987 and worked on high strain rate studies before commencing Ph.D. studies in 1989 as an MRL Cadet Research Scientist at the Department of Civil Engineering, Monash University. A thesis was completed entitled: "The Modelling of the Collapse during Roll-Over of Bus Frames Consisting of Square Thin-Walled Tubes", and is the basis for the current publication. He is currently working in the areas of light armour technology and impact dynamics.

Contents

1. INTRODUCTION **7**

2. FINITE ELEMENT MODELLING COLLAPSING THIN-WALLED VEHICLE STRUCTURES **8**

3. PROGRAM BEAM **9**
3.1 Step-By-Step Analysis and Program BEAM **9**
3.2 Operation of Program BEAM **13**
3.3 Verification of Program BEAM **18**

4. MODELLING THE COLLAPSE OF A TWO-DIMENSIONAL FRAME WITH PROGRAM BEAM **21**

5. CONCLUSION **26**

6. ACKNOWLEDGMENTS **26**

7. REFERENCES **26**

APPENDIX 1 – *Program BEAM: Input File Layout* **30**

APPENDIX 2 – *Program BEAM: Input File* **32**

APPENDIX 3 – *Program BEAM: Output File* **33**

BEAM: A Finite Element Program for the Collapse Analysis of Vehicle Structures

1. *Introduction*

Accurate large-deflection design techniques have been developed for vehicles whose superstructures are constructed of thin-walled frames which fail by collapsing in bending and exhibit drooping load-deflection behaviour (1),(2). The impetus for the development of these methods has come from the requirement to design passenger buses to meet new Australian Design Rules for occupant protection in roll-over accidents (3). These same techniques can also be applied to military vehicles which use their upper body-work as a Roll-Over Protective Structure (ROPS) or even for the design of add-on roll-bars.

The entire load-deflection behaviour of thin-walled vehicle frameworks can be determined by simply superimposing separate elastic and rigid-plastic models of a frame's behaviour (1),(2). While in many cases vehicle frames can be designed to withstand roll-over accidents with two-dimensional techniques, there are some instances where it is important to include the effects of structural members that are orientated in a third dimension.

A finite element technique was considered to be the most sensible method for determining the influence of three-dimensional effects on frame collapse behaviour. The three-dimensional finite element program 'BEAM' (Beam Element Analysis with Mechanisms) (2) was specifically written for this purpose because conventional finite element programs were unsuitable. One advantage of BEAM over other programs is that it allows a user to define their own experimentally or analytically determined loads and moments as failure criteria. BEAM will allow a quick determination of the collapse load of a vehicle frame as well as its deformation up to this load, in the process predicting the locations of the plastic hinges which form its collapse mechanism. The latter information allows a rigid-plastic modelling technique to be easily applied to determine the behaviour of the vehicle frame beyond the point of collapse.

This report outlines the theory and operation of BEAM and its application to modelling a collapsing thin-walled frame.

2. *Finite Element Modelling Collapsing Thin-Walled Vehicle Structures*

For various reasons it can be difficult to model collapsing vehicle frames with the finite element method. This is partly because of the gross plastic deformation and geometric distortion associated with the formation of plastic hinges within the thin-walled sections of the vehicle frame. Specialised large-deformation finite element programs are used to model this deformation because they have specific algorithms for accurate and time-efficient modelling of gross plastic deformation and distortion, e.g. (5). General purpose nonlinear finite element programs, that are used for common engineering structural analyses, for instance, are too inefficient for these purposes (6), (7).

Specialised large-deformation programs have major limitations. These programs may require access to expensive computer systems and require considerable specialised manpower and time to code and run a vehicle collapse analysis. In addition to these efficiency aspects there are also important technical problems which have not yet been overcome.

Even the largest and most specialised large-deformation programs have not yet been proven to accurately predict fundamental failure mechanisms (8). While such programs can still produce excellent representations of experimentally deformed components, quantitative comparisons of numerical and experimental crush force are generally not available (8). In any case, the accuracy of such comparisons, if available, can be greatly changed by the values assumed for various ill-defined quantities, for instance, the strain rate effect on flow stress (8). In some cases the manual intervention to the modelling process that is required to produce accurate failure mechanisms can have little relationship to reality, e.g. (9). To obtain realistic results from such numerical simulations, it appears that the deformed geometry of a structure is required *a priori*. This would require a vehicle prototype to be built and crash tested, an unsatisfactory situation.

Furthermore, the large computer systems required to efficiently run large-deformation programs may not provide the continuous access needed for design improvement and optimisation in the early stages of product development when the base structural design is being finalised. Precise modelling is unnecessary, as at this stage the exact design details are generally not specified. What is needed at the initial design stages is a quick first-order approximation of the structural strength of the major components. Such analyses can be used to optimise the base structural strength of a design well before vehicle prototypes have to be built. A vehicle model can therefore be simplified to just the major load bearing members, for instance, a framework consisting of beam elements.

A promising improvement to modelling gross plastic deformation, appears to be to incorporate heuristic or hybrid elements into a conventional finite element mesh (8). This approach has been adopted and developed by workers such as Kecman (10),(11), Mahmood and Paluszny (12) and Wierzbicki and Abramovich (13). Heuristic type elements can incorporate the mechanics of the plastic hinges found in the grossly deformed regions of thin-walled sections (8). The use of such elements can eliminate many thousands of conventional elements from an analysis as the plastic hinges are simply treated as 'black boxes'. In other words, the exact details of the deformation of the plastic hinges are not important to the operation of the program, only their external effects, i.e. their externally recognised resisting moments. The resulting coarser numerical models require less hardware and human resources, and allow an analysis of a crash to be performed on a PC or workstation.

There are various ways of incorporating heuristic elements into finite element programs. The 'CRASH-D' program (**CRA**nfield **S**tructural **H**igh **D**eformation) or as it is now known, 'CIC-STAT', is the best known example of this approach (14). This program uses beam elements to analyse the structural collapse of three-dimensional frames to large deflections. The resistance of the plastic hinges in a collapsing frame is simply obtained from analytical or experimentally determined quasi-static moment-rotation curves, allowing the specific details of the deformation at the hinges to be ignored. Consequently a crash analysis can be much more quickly performed than if conventional finite element modelling methods were used. Applications have included the crushing of various vehicles and components, viz, truck cabins (15), door intrusions and roof crushes of cars (16),(17),(18) as well as bus roll-overs (19),(20).

Programs like CRASH-D can be used to model the collapse of vehicle structures in crashes that are amenable to static analyses, with dynamic effects being ignored (21). Passenger bus roll-over accidents, for instance, fall into this category. This is in part because such accidents tend to occur laterally at low speeds (20),(22). Inertial effects can effectively be neglected because deformation is largely confined to the superstructure of the buses, with most of their mass being concentrated at floor level. In addition, it is often the case that the dynamic collapse modes in bending are the same as the static ones (11). At least this is the case for the square thin-walled tubes from which passenger buses have been constructed. More importantly, however, the dynamic enhancement of the material yield stress as well as flow stress at large plastic strains can be accounted for to some extent by multiplicative factors (23),(24).

Program BEAM can be applied to the same class of problems as CRASH-D and accounts for a plastic hinge in a member by modifying the element stiffness matrix to include a pin-joint which will represent the physical effect of a plastic hinge, i.e. zero stiffness. This operation is performed when an experimentally determined failure criteria is exceeded, the pin-joint being capable of modelling the behaviour of a plastic hinge from its point of formation up to when drooping moment-rotation behaviour is exhibited. Therefore BEAM, unlike CRASH-D, cannot determine the drooping load-deflection curve of a collapsing thin-walled frame. However, the load-deflection curve that BEAM calculates can be combined with a separate rigid-plastic analysis of the collapsing frame. This allows the behaviour of such a frame to be determined to large deflections where drooping load-deflection behaviour is exhibited.

3. *Program BEAM*

3.1 *Step-By-Step Analysis and Program BEAM*

Simple Plastic Analysis has been shown to be capable of accurately predicting the collapse load of simple thin-walled rectangular frames (25). This method requires an accurate assumption of the collapse mechanism of a structure, which is straightforward for simple structures. However, for more complex structures, trial and error procedures involving significant intuition are required to determine the correct collapse mechanism. It is difficult to program computers to make the necessary judgements for these procedures, but computer methods using Step-By-Step analysis, for instance, have been developed that can quickly calculate the collapse load and plastic hinge locations of statically loaded frames. Wang (26) first used this method to determine the collapse load of two-dimensional frames.

Step-By-Step analysis involves making an initial elastic analysis of a frame under a working load and then dividing the moments at the end of each element into their full-plastic values to obtain load-factors to form plastic hinges; the smallest load-factor identifying where the first plastic hinge would form. The relevant element stiffness matrix is then modified to include a pin-joint at the appropriate node by zeroing the relevant stiffnesses. The process is repeated and successive elastic analyses performed until the structure is converted into a mechanism.

Program BEAM was written in FORTRAN 77 as a three-dimensional beam element finite element program. It is modular in nature and employs an upper-half-bandwidth solver that uses Gaussian Elimination and Back Substitution to obtain a solution (27). The Step-By-Step procedure of BEAM was originally derived from the two-dimensional EPFOSS program of Harrison (28). This latter program originally accounted for bending and axial deformation only and used a Triple Decomposition method for solution. The 12 x 12 linear elastic stiffness matrix of BEAM was able to account for bending, shear, axial and torsional deformation. Fig. 1 defines the degrees of freedom of this element. However, the current version of BEAM only checks for the possibility of bending collapse in the x and y local axes of a section. Furthermore, BEAM currently has no provision to detect plastic hinges formed under significant multi-axial loads. This limited capability is sufficient for the collapse of regular prismatic frames, for instance civilian passenger buses, but other failure modes can contribute to the structural collapse of frames of more general geometry. Axial crushing, e.g. (29) and torsional collapse should also be accounted for. Further experimental characterisation is also required to determine correctly the failure criteria for these modes of collapse as well as in multi-axial situations. BEAM has provision to include these failure criteria when appropriate data becomes available, e.g. (30).

BEAM employs a linear elastic stiffness matrix that is based on the original undeformed structure geometry. This is because the aim of BEAM is to provide only a quick first-order estimate of a frame's load-displacement curve up until it forms a mechanism. Consequently BEAM cannot account for effects such as the P-δ Effect (4), i.e. when a strut carrying an axial load P deflects a lateral distance δ, its bending stresses are increased. Nor can BEAM account for the reduction of plastic moment capacity due to axial loads (4). While non-linear elastic stiffness matrices can be included that can account for a structure's changing geometry, they would make the operation of BEAM cumbersome and detract from its inherent simplicity. The load-deflection response of a real structure will therefore not be as stiff as that predicted by BEAM. However, first-order analysis is sufficiently accurate for quick analyses of thin-walled structures up to collapse. The errors involved with such a philosophy are likely to be minimal considering the large deformations that such structures experience. Furthermore, it is assumed that a structure's joints and connections maintain their integrity to large deformations. Since it is not easy to predict joint strength to large deflections, this should ideally be verified experimentally prior to modelling by either individual joint tests (31) or full-sized frame tests (2).

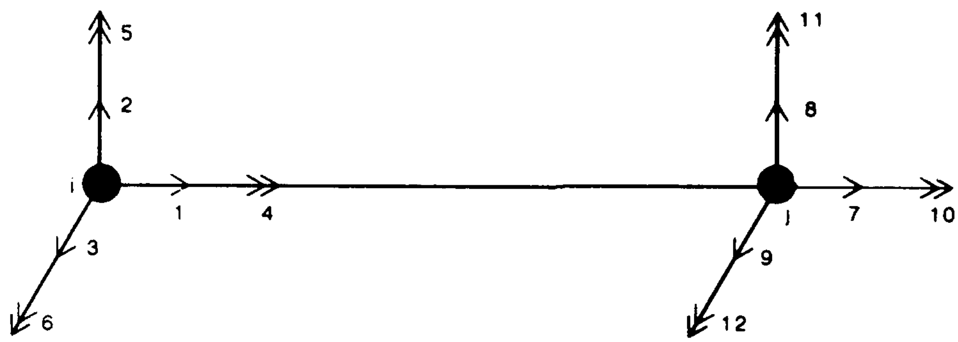


Figure 1 Degrees of Freedom associated with the three-dimensional beam element of BEAM that is defined by nodes i and j. Arrows with single heads denote loads while double-headed arrows denote moments.

The insertion of a plastic hinge (or pin-joint) can be achieved by modifying portions of the relevant element stiffness matrix. As an example, Fig. 2 shows possible element stiffness matrix modifications for a simple beam element with two bending degrees of freedom. The bending stiffness at the node of the plastic hinge and the cross-stiffnesses were zeroed, with the bending stiffness at the opposite end of the element being reduced to three-quarters of its usual stiffness. As has been the practice with other workers (14),(26),(28), the axial stiffnesses were not modified. However, the 12 x 12 stiffness matrix of BEAM also includes shear stiffness terms which must be modified to account for the presence of plastic hinges. The appropriate shear stiffnesses were modified according to established rules (32). However, it was assumed (for simplicity) that a plastic hinge forming on one local element axis would not influence the bending stiffnesses associated with the other local element axis. Moreover, the formation of a plastic hinge in bending was not considered to affect a member's torsional stiffness. While these assumptions may not be strictly correct, they are nevertheless reasonable in the context of the present sparse knowledge of the three-dimensional properties of plastic hinges in bending collapse.

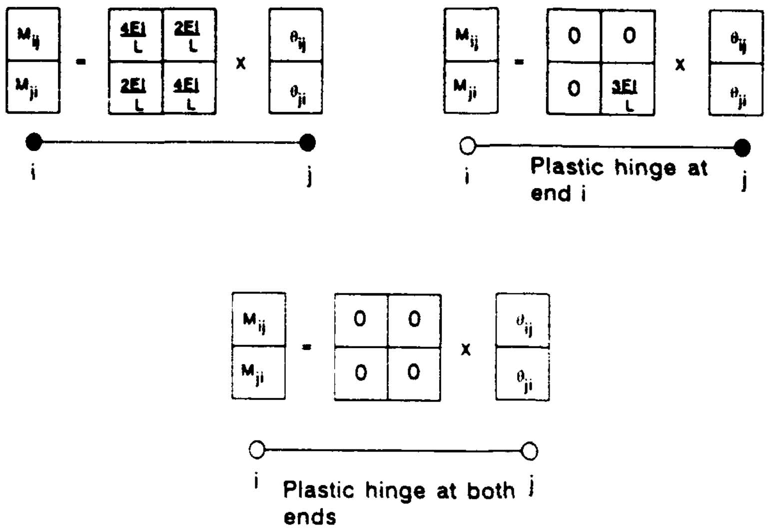


Figure 2: Element stiffness matrix modifications caused by various plastic hinge insertions for a simple beam element with two bending degrees of freedom. (After Harrison (28)).

There are other assumptions inherent in the Step-By-Step procedure of BEAM that are common to Simple Plastic Analysis (25). For instance, it is assumed that as the load-factor is increased until the structure reaches its ultimate load, each plastic hinge maintains a constant moment. It can also sometimes occur that a plastic hinge that forms early in an analysis may not be required for the final frame collapse mechanism. The moment at such a hinge can actually unload. While this phenomena cannot be accounted for in BEAM (28), the predicted load factor for collapse would nevertheless be conservative, i.e. a lower-bound estimate, because the equilibrium and yield conditions are satisfied but not the mechanism condition (4). This is an example of a partial collapse (33).

One special feature of BEAM is the FORMRT subroutine that calculates the three-dimensional rotation matrices that are used to transform element stiffness matrices into global co-ordinates. FORMRT was based on the vector method of Wilson (34), and allows BEAM to be easily applied to three-dimensional frames where the orientation of the principal axis (i.e. plane of principal bending) has to be specified. Three nodes are required to be specified, viz, nodes i and j defining an element's location and node k, that defines its principal axis, Fig. 3. The program requires node k to be specified in the positive quadrant of the local axes defined by the first and second degrees of freedom of the element.

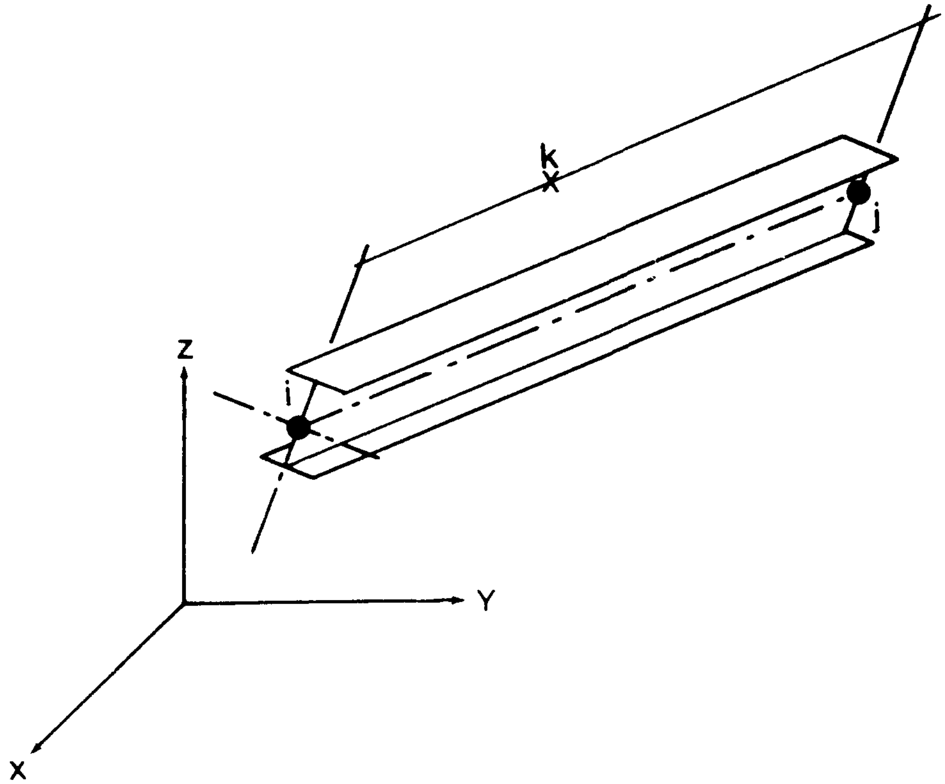


Figure 3: Node k defines the principal axis of the element defined by nodes i and j. Node k, denoted by (X), must be specified within the positive quadrant defined by the axes of the first and second degrees of freedom of the element (After Wilson (34)).

3.2 Operation of Program BEAM

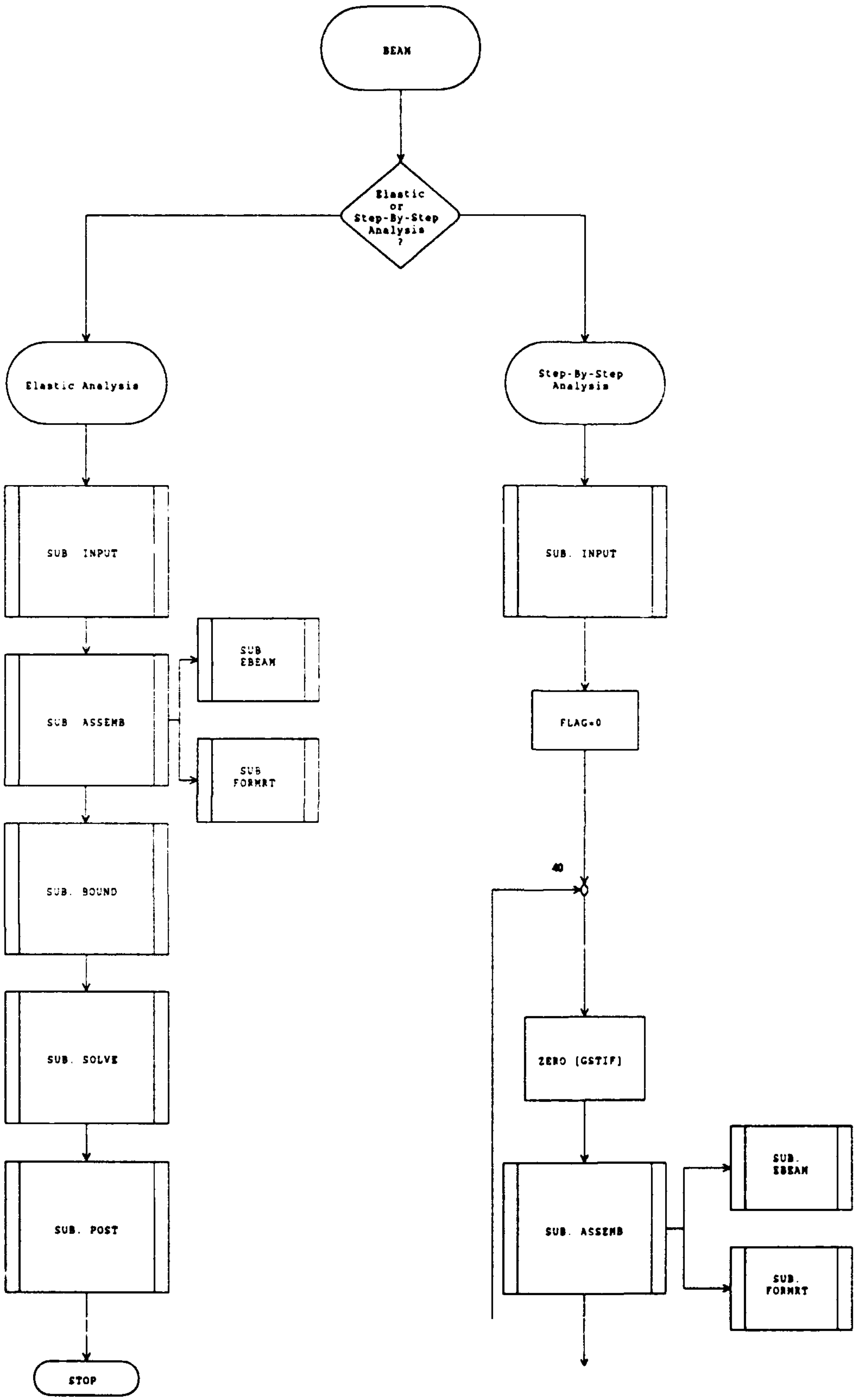
The operation of program BEAM is summarised in the flow chart in Fig. 4. A full listing of the source code for BEAM is given in a separate publication (35). Appendix 1 outlines the layout of a generic input file. BEAM gives an analyst the choice of either an Elastic analysis or a Step-By-Step analysis to collapse. If the latter option is chosen, the structural stiffness matrix [GSTIF] is assembled in subroutine ASSEMB which obtains the elemental stiffness matrices from subroutine EBEAM and the corresponding rotation matrices from subroutine FORMRT. Matrix [GSTIF] is then inspected for zero stiffnesses in its main diagonal, a singular matrix indicating that the structure has been transformed into a mechanism. This check is the primary means for terminating the program. If the analysis continues, a typical working load is assigned to the loading matrix [AJ]. Matrix [GSTIF] is then corrected for restrained boundary conditions in subroutine BOUND and solved by subroutine SOLVE to obtain the nodal displacements.

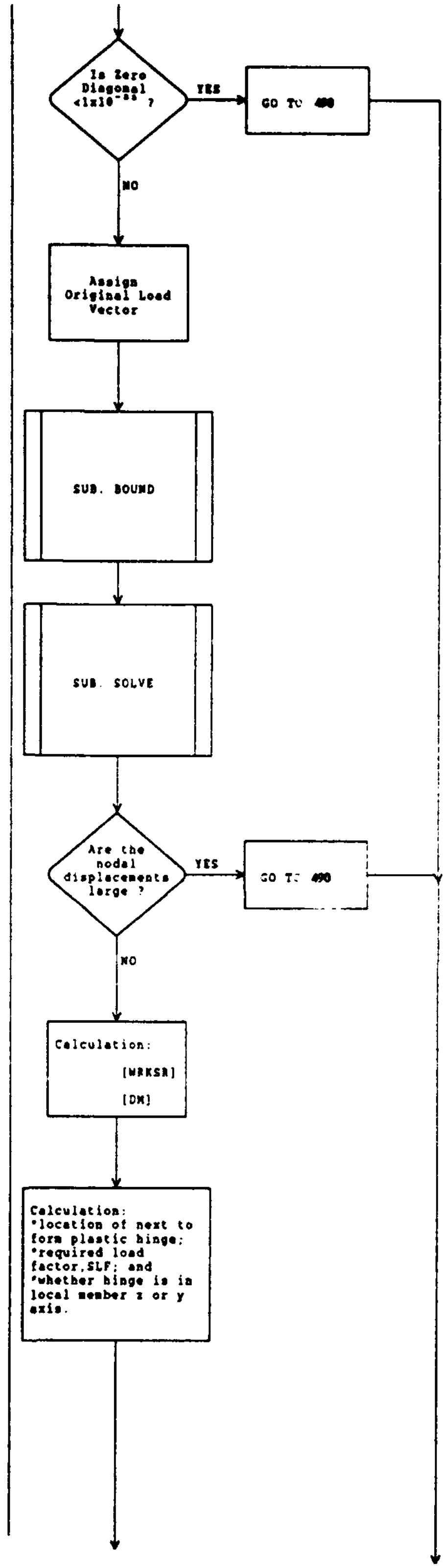
BEAM incorporates another test to check for collapse because round-off errors could delay program termination if a check was made only for singularity. An additional check is made for very large computed displacements. Large displacements indicate that the structure's load-displacement curve had effectively become horizontal, as would be the case in the early stages of structural collapse.

The member stress resultants matrix [WRKSR] and member displacements matrix [DM] are next calculated. The moments at the bending degrees of freedom are then divided into the specified failure moments to obtain load factors for possible plastic hinges. The first plastic hinge will form at the degree of freedom with the smallest load factor, SLF, BEAM determining whether this hinge forms in the local member y axis or z axis. This information is recorded along with which end of the element the plastic hinge is situated. Matrices [WRKSR], [DM] and the nodal displacements are therefore multiplied by SLF and added to the cumulative stress-resultants, matrix [CUMSR]; cumulative member displacements, matrix [CUMDM]; and the cumulative nodal displacements, matrix [CUMX]; respectively. SLF is also added to CUMLF, the cumulative load-factor. These quantities are printed together with the location and load-factor of the plastic hinge.

The program loop is repeated with the structure stiffness matrix [GSTIF] again being assembled. Once plastic hinges have begun to form, the element stiffness matrices that are generated within subroutine EBEAM are modified to allow for the presence of plastic hinges in either or both ends of the element on its y or z local member axes. These heuristically modified matrices are again assembled in ASSEMB and the loop continued with the original loading matrix [AJ] again applied and a further solution obtained.

Eventually, when sufficient plastic hinges have formed to initiate structural collapse, the program will detect either a zero diagonal in the structure stiffness matrix or excessive nodal displacements. The program then proceeds to line 490 and sets a flag so that a final output can be made of matrices [CUMX], [CUMSR] and [CUMDM] along with the unfactored displacements remaining after the last solution.


BEAM
Elastic or Step-By-Step Analysis ?
Elastic Analysis
Step-By-Step Analysis
SUB INPUT
SUB ASSEMB
SUB EBEAM
SUB FORMRT
SUB. BOUND
SUB. SOLVE
SUB. POST
STOP
SUB. INPUT
FLAG=0
40
ZERO [GSTIF]
SUB. ASSEMB
SUB. EBEAM
SUB. FORMRT



Is Zero Diagonal <1x10^-?? ?
YES
GO TO 490
NO
Assign Original Load Vector
SUB. BOUND
SUB. SOLVE
Are the nodal displacements large ?
YES
GO TO 490
NO
Calculation:
[WRKSR]
[DM]
Calculation:
*location of next to form plastic hinge;
*required load factor,SLF; and
*whether hinge is in local member z or y axis.

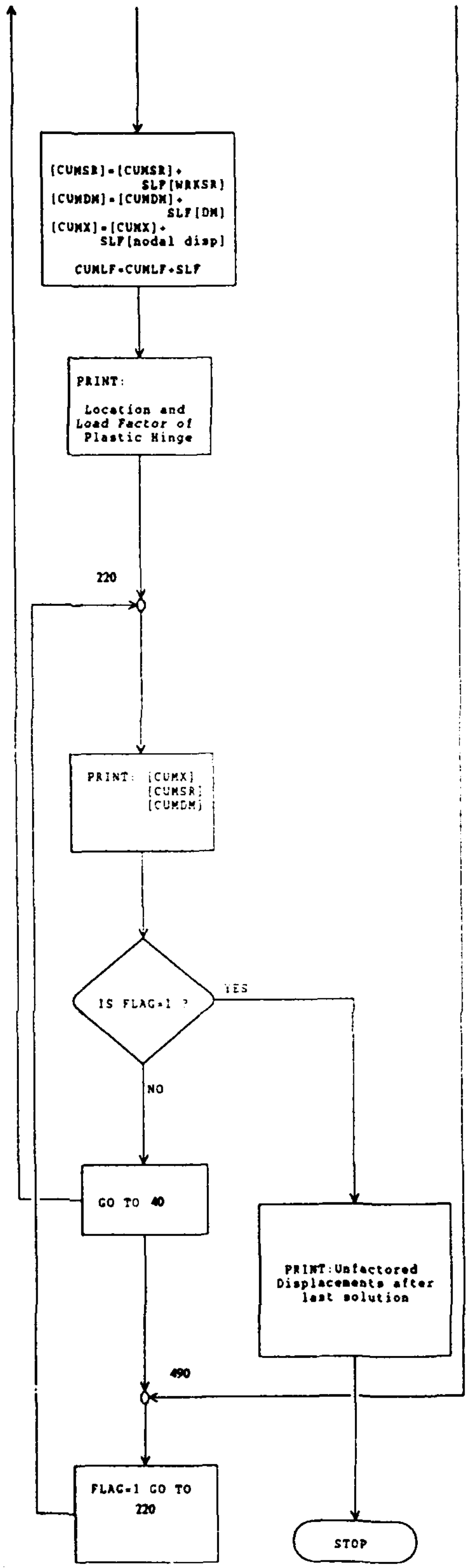


Figure 4: *Flow chart of the operation of program BEAM.*

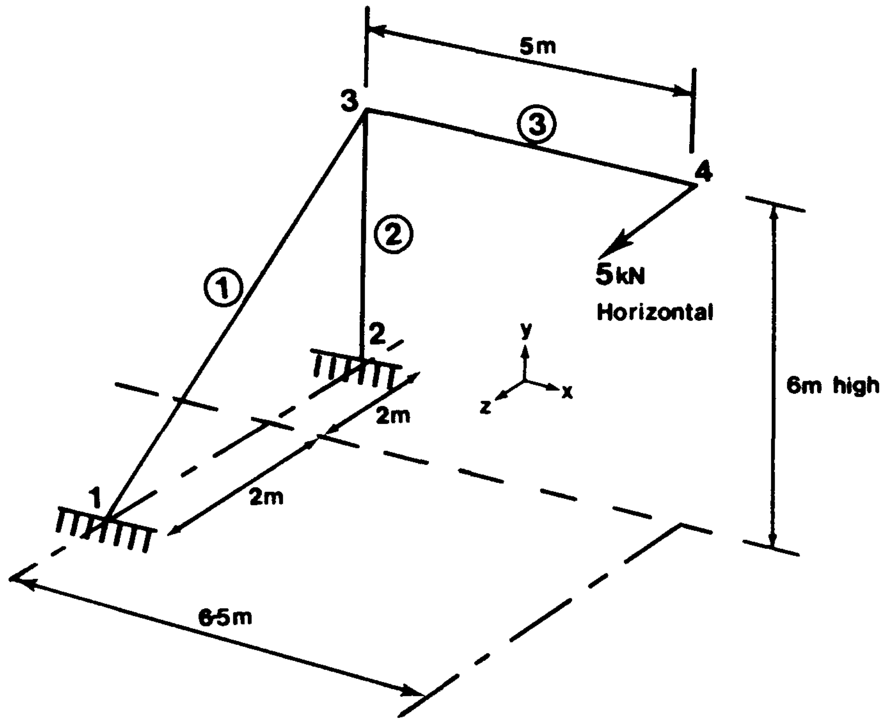
An important feature of BEAM is its ability to insert plastic hinges of zero stiffness into the structure stiffness matrix without creating a singular matrix. This is achieved because each node has some finite stiffness associated with it, that is if a plastic hinge forms at one end of an element, the adjacent element connecting to that element will still have some finite stiffness. This allows the lack of stiffness associated with a plastic hinge to be incorporated into an elastic analysis. One problem that was encountered during validation was the inability of BEAM to insert a plastic hinge into a degree of freedom associated with a fixed boundary condition that had zero displacement. BEAM overcame this problem by incorporating a test in subroutine EBEAM which checked whether any plastic hinges coincided with such fixed boundary conditions. If coincidence was detected, the appropriate rotation degree of freedom was set to 1×10^{-23}, a negligible value but sufficient to prevent the structure stiffness matrix from becoming singular.

In comparison to CRASH-D (14) BEAM does not appear to account for the resisting moments of existing plastic hinges in its solution procedure, the solution technique of BEAM being purely stiffness based. However, the load-displacement curve of BEAM is obtained from the cumulative stress-resultants matrix [CUMSR]. The moments in this matrix at the degrees of freedom associated with plastic hinges are equal to the failure moment because the results of each elastic analysis, matrix [WRKSR], are multiplied by the minimum load factor, SLF, before being added to matrix [CUMSR] (28),(36). One advantage of BEAM is its ability to quickly determine a structure's load-displacement curve up to the collapse load with a minimum number of solution steps. In comparison, the incremental solution procedure of CRASH-D requires more operator skill and an indefinite number of structure stiffness matrix inversions.

3.3 *Verification of Program BEAM*

BEAM was elastically verified in three dimensions by analysing the frame shown in Fig. 5.

The Step-By-Step procedure of BEAM was verified by analysing the frames detailed in Fig. 6 and Fig. 7 up to the point of collapse. These frames had 1 and 6 degrees of redundancy, respectively, and so required only 2 and 7 plastic hinges, respectively, to form for structural collapse. As well as predicting the correct locations and order of plastic hinge formation, BEAM obtained exactly the same displacements and loads as the two-dimensional EPFOSS program of Harrison (28). The two-dimensional SOAPS program (37), which used a similar Step-By-Step method, independently confirmed these results. The two-dimensional PCAP program (38), which used a linear programming technique for solution, again independently confirmed the collapse load of these frames. Overall, the operation of BEAM was sufficiently verified to justify its application to predict the plastic hinge locations and collapse load of an experimentally tested two-dimensional frame.

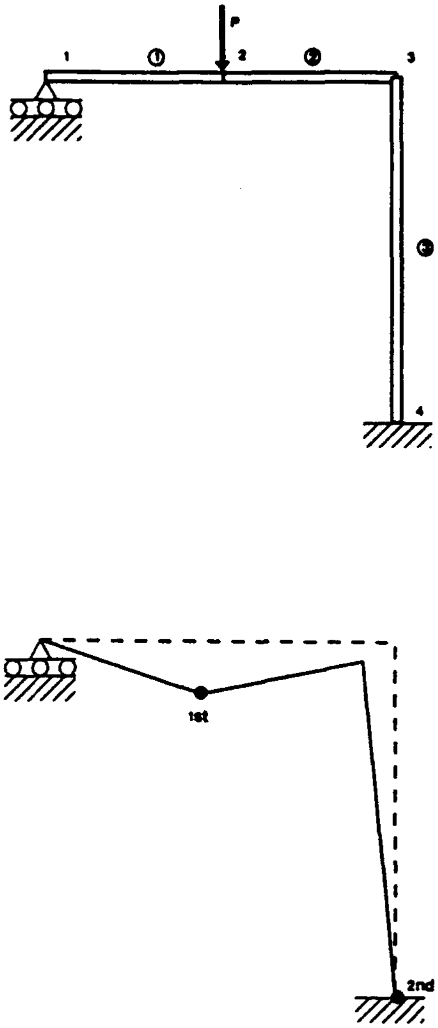


(a)

	Translations (mm)			Rotations (deg)		
	X	Y	Z	X	Y	Z
1	0	0	0	0	0	0
2	0	0	0	0	0	0
3	0	0	0.147	-0.0267	0.1274	0
4	0	0	9.117	-0.0267	0	0

(b)

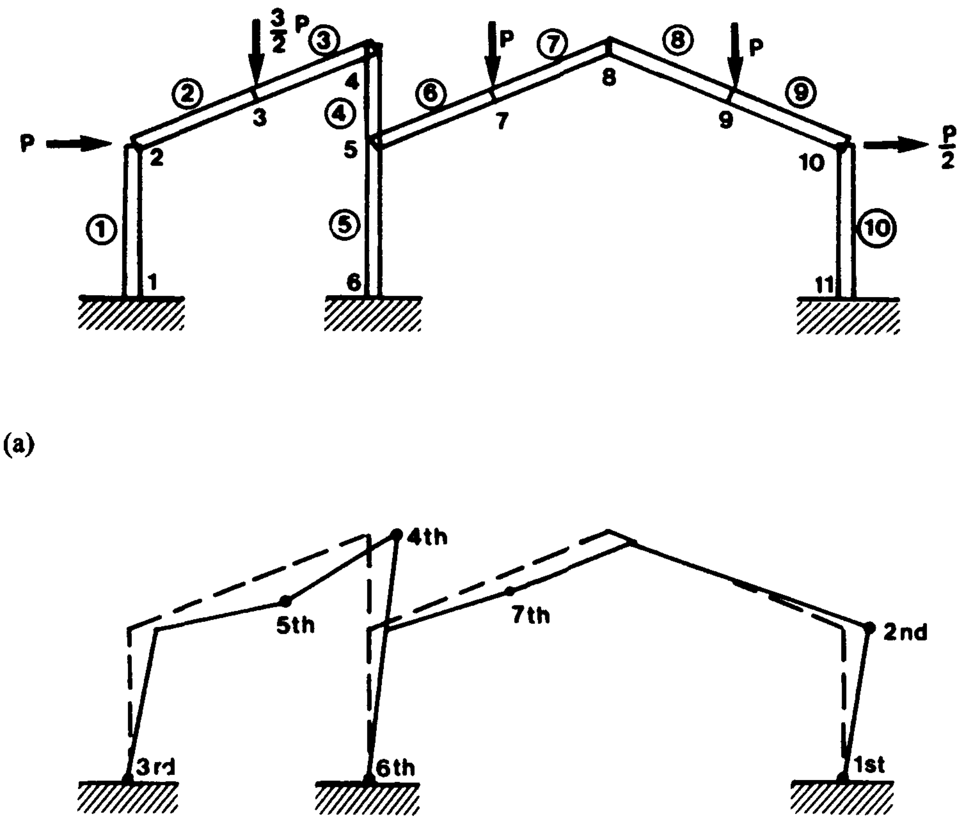
***Figure 5**: Frame 1, (a) initial loading and (b) results (global nodal displacements). Member 3 is 310UC97 with a vertical web. Members 1 and 2 are 310UB46 with the webs lying in the plane containing the relevant member itself and Member 3. Young's Modulus, E, and the Shear Modulus, G, are 206×10^6 kNm^{-2}, and 90×10^6 kNm^{-2}, respectively.*



(a)

(b)

Figure 6: *Frame 2: (a) initial geometry, showing node numbers with element designations circled; and (b) deformed geometry with sequence of plastic hinge formation. (After Harrison (28)).*



(a)

(b)

Figure 7: Frame 3: (a) initial geometry, showing node numbers with element designations circled; and (b) deformed geometry with sequence of plastic hinge formation. (After Harrison (28)).

4. Modelling the Collapse of a Two-Dimensional Frame with Program BEAM

A two dimensional frame made from 50x50 mm square thin-walled tubes of 2 mm wall thickness, Frame 4, was tested to confirm the validity of the model predictions (2). Fig. 8 depicts the geometry of the frame, (a), and its idealised finite element model, (b). The experimental load-displacement response of the frame was obtained by simply displacing the frame by means of a hydraulic tension jack that pulled from an initial angle of 15° (2). The applied load and the movement of the point of load application were measured as deformation proceeded.

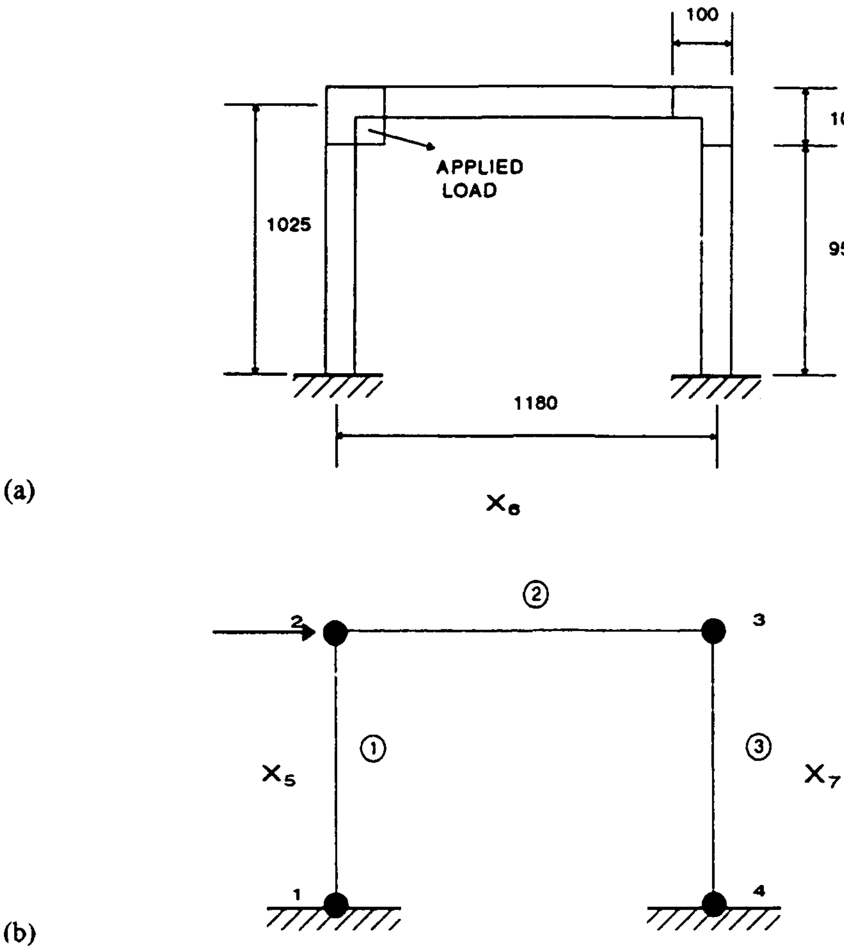


Figure 8: The initial geometry, (a), and finite element model, (b), of the tested two-dimensional thin-walled frame, Frame 4. Structural nodes in (b) are depicted by (●), k nodes (refer to Fig. 3) by (X), and the element numbers circled.

An input file was prepared with a working load of 5 kN. The input file and the output file are included in Appendix 2 and Appendix 3, respectively. Pure bending experiments were used to determine the maximum moment of the tubing (2), (39). A maximum moment of 3.12 kNm was obtained and used for the failure moment in the input file. Note that in the output file negative signs denote the locations of the plastic hinges. The plastic hinges formed first at the base and then, with further deformation, at the roof joints as expected. The plastic hinges formed in the following order:

plastic hinge 1 at node 1 at a load of 10.466 kN

plastic hinge 2 at node 4 at a load of 10.484 kN

plastic hinge 3 at node 2 at a load of 12.173 kN

plastic hinge 4 at node 3 at a load of 12.176 kN

These Step-By-Step analysis results were confirmed independently using the Step-By-Step SOAPS (37) and PCAP (38) finite element programs and are plotted as a function of the horizontal displacement of the loading point in Fig. 9. Separate elastic and rigid-plastic models are also superimposed. It has already been shown that the complete load-displacement curve of this frame can be modelled accurately by superimposing separate elastic and rigid-plastic models of its behaviour (1),(2). This was for a first-order elastic model which assumed that the frame behaved elastically up to its collapse load and for a rigid-plastic model which assumed that the plastic hinges necessary for the collapse of the frame began rotating at zero displacement.

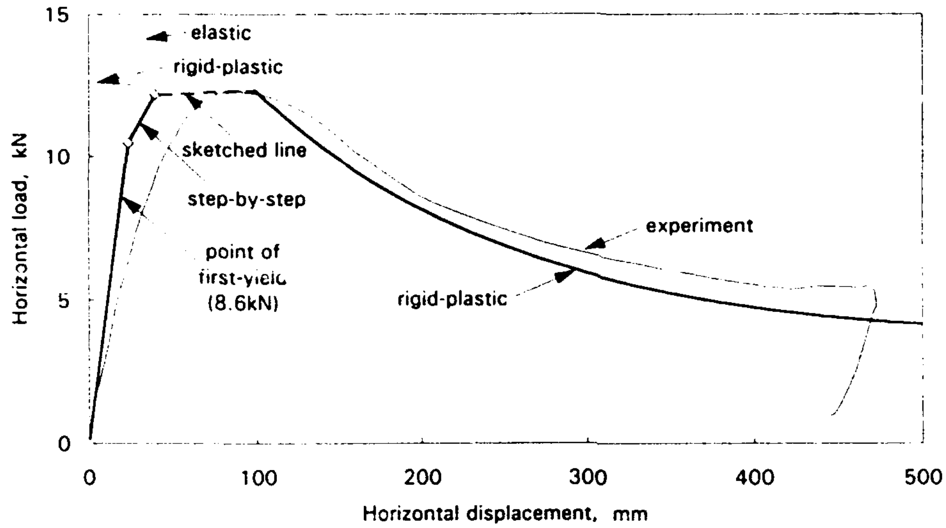


Figure 9: The load-displacement curve for Frame 4, comparing elastic, rigid plastic and Step By-Step analyses with experimental results. The combined model of the load-displacement curve is shown in bold. The rigid-plastic model assumes that the plastic hinges form at the corners of the frame and its boundaries with the fixed supports.

Figure 9 shows that if the results of the Step-By-Step analysis are combined with the separate rigid-plastic analysis then a better model of the complete load-displacement curve of the frame can be obtained, as shown by the bold lines. Fig. 10 has an expanded displacement scale to show more clearly the detail of the first 120 mm displacement. A line between the Step-By-Step analysis and the collapsing portion of the rigid-plastic curve can be sketched by hand to complete the model (1),(2) (shown by the bold dashed lines in Figures 9 and 10).

It was found in the experimental frame test that a maximum load of 12.30 kN was reached at 88 mm deflection, with collapse occurring at a load of 12.20 kN at 101 mm deflection. BEAM actually determines the point at which the frame first forms a mechanism. At this point BEAM predicted a maximum load of 12.18 kN for 39 mm deflection. Once plastic hinges form in the particular thin-walled members of the frame, they deform at a relatively constant moment before exhibiting a drooping moment-rotation curve (39). As a consequence, a collapsing thin-walled frame will also deform at a relatively constant load from the point of mechanism formation to the point of collapse (2). The maximum load predicted by BEAM will therefore approximate the collapse load of the frame.

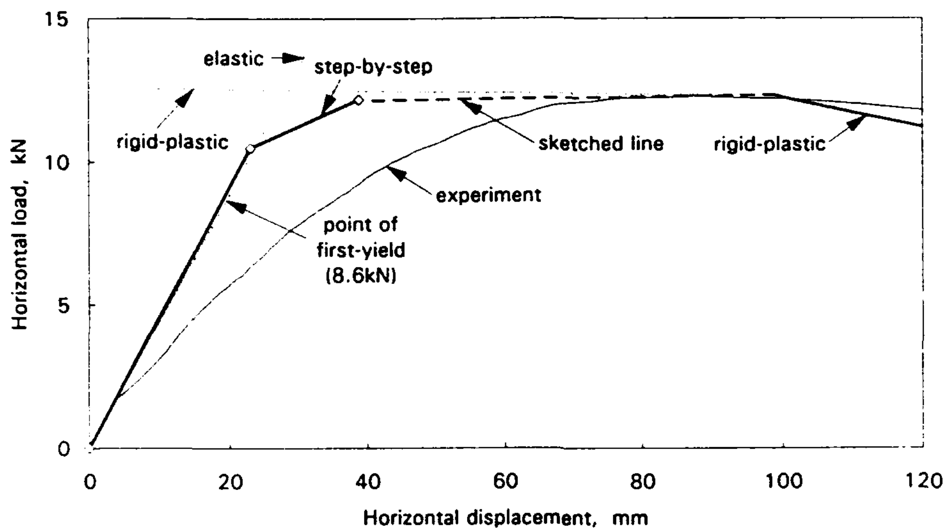


Figure 10: An enlargement of the initial portion of the load-displacement curve for Frame 4, comparing elastic, rigid-plastic and Step-By-Step analyses with experimental results. The combined model of the load-displacement curve is shown by the bold line. The rigid-plastic model assumes that the plastic hinges form at the corners of the frame and its fixed supports.

A compatibility check was also performed to check the reliability of the joint and member rotations predicted by BEAM. These rotations were obtained from the output file in Section 3 of the Appendix, converted into degrees, and compared with the axis rotations which were the rotations of the element centrelines from their original positions, Fig. 11. These comparisons were made at the point of mechanism formation. The following compatibility check shows that all these rotations are consistent, where AR, JR, and MR, refer to axis, joint and member rotations, respectively:

element 1, node 2 AR =JR+MR $-2.16^\circ \sim -1.17^\circ + (-1.17^\circ)$

element 2, node 2 AR=MR-JR $0^\circ = -1.17^\circ - (-1.17^\circ)$

element 2, node 3 AR=MR-JR $0^\circ = -1.16^\circ - (-1.16^\circ)$

element 3, node 3 AR=JR+MR $-2.16^\circ \sim -1.16^\circ + (-1.16^\circ)$

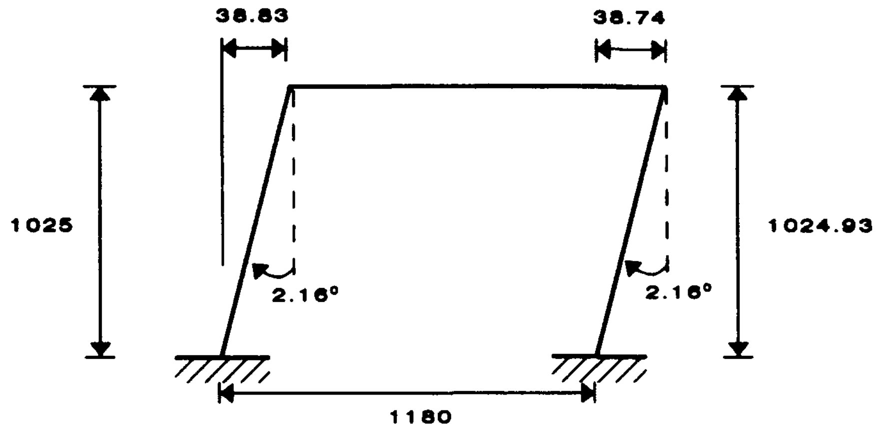


Figure 11: Axis rotations at the point of mechanism formation. Dimensions in mm.

The rigid-plastic model that was depicted in Fig. 9, predicted a plastic hinge rotation of 2.34° for a 39 mm frame deflection at a load of 12.48 kN. It is interesting to note that this rotation is equivalent to the axis rotation obtained by adding the joint and member rotations at element 1, node 2. The rigid-plastic model predicted collapse to occur at a plastic hinge rotation of 5.95°, which corresponds to a 98 mm deflection of the frame at a load of 12.33 kN.

In general, there is good correspondence between the predictions of the combined model and the experimental results. While the stiffness of the frame is overpredicted prior to collapse, this is because its joints and, especially the fixity at its base, are considered to be perfectly rigid. Furthermore the elastic and Step-By-Step analyses are only first-order models and do not therefore account for the changing frame geometry. The differences such as there are between the predicted and experimental load-displacement curves beyond collapse are mainly due to the design of the rigid joints of the frame which caused the plastic hinges to develop in slightly different locations to those predicted by BEAM, i.e. at the edges of the rigid joints. This was to prevent joint failure such as had occurred in earlier frame tests (1),(2).

Overall it can be seen that BEAM provides a valuable adjunct to separate rigid-plastic modelling. As well as predicting the locations of the plastic hinges, BEAM also provides an accurate estimate of the collapse load of the frame, i.e. 12.18 kN, compared to 12.20 kN found experimentally. This is because BEAM should usually satisfy the mechanism, yield and equilibrium conditions. Moreover, if the load-deflection curve produced by BEAM is combined with a rigid-plastic model of the collapsing frame then the definition of the elasto-plastic transition prior to collapse is improved compared to simpler modelling methods (1),(2). Figures 9 and 10 show that simply sketching a curve by hand from the point of mechanism formation (predicted by BEAM) to the actual point of collapse (predicted by the separate rigid-plastic analysis) completes the prediction of the frame's load-displacement curve (1),(2).

5. *Conclusion*

Program BEAM will determine the deformation of a thin-walled frame up to the point at which sufficient plastic hinges form to transform the frame into a mechanism, i.e. the point of mechanism formation. This allows the locations of the plastic hinges composing the collapse mechanism of the frame to be predicted along with an estimate of its maximum load. When BEAM is used in conjunction with a large-deflection rigid-plastic modelling technique, the load-deflection curve of a thin-walled frame can be determined beyond the point of mechanism formation to where drooping load-displacement behaviour is exhibited at large deflections.

6. *Acknowledgments*

BEAM was written as part of a Ph.D. thesis at the Department of Civil Engineering at Monash University. It could not have been written without the tuition of Dr Raphael Grzebieta, Norm Sneath, Dr Mario Attard and Prof. Noel Murray in the finer aspects of the theory of finite elements. The author also appreciates the Research Scientist Cadetship awarded by MRL which allowed him to undertake this work.

7. *References*

1. Cimpoeru, S.J. and Murray, N.W. "A Large-Deflection Design Technique for Modelling the Collapse of Bus Frames Constructed from Thin-Walled Tubes in Roll-Over Accidents", Dynamic Loading in Manufacturing and Service Conference, Melbourne, 9-11 Feb. 1993, IE(Aust.), pp. 169-176.

2. Cimpoeru, S.J. "The Modelling of the Collapse During Roll-Over of Bus Frames Consisting of Square Thin-Walled Tubes", Ph.D. Thesis, Department of Civil Engineering, Monash University, Australia, 1992.

3. Australian Design Rule 59/00, "Omnibus Roll-Over Strength", Federal Dept. of Transport and Communications, Australia, 1988.

4. Murray, N.W. "Introduction to the Theory of Thin-Walled Structures", Clarendon Press, Oxford, 1986.

5. Haug, E., Arnaudeau, F., Dubois, J., de Rouvray, A. and Chedmail, J.F. "Static and Dynamic Finite Element Analysis of Structural Crashworthiness in the Automotive and Aerospace Industries", Structural Crashworthiness (eds N. Jones and T. Wierzbicki), Butterworths, London, 1983, p. 175.

6. Grzebieta, R.H., and Dayawansa, P.H. "Roll-Over Analysis of a Truck Cabin Frame", Structural Crashworthiness and Property Damage Seminar, Monash University, Melbourne, 1987, pp. 111-133.

7. Belytschko, T. "On Computational Methods for Crashworthiness", Proceedings of the 7th International Conference on Vehicle Structural Mechanics, Detroit, SAE Paper No. 880893, SAE, 1988, pp. 83-92.

8. Khalil, T.B. and Bennett, J.A. "Nonlinear Finite Element Analysis in Crashworthiness of Automotive Shell Structures - An Overview", Proc. Winter Annual Meeting ASME, Analytical and Computational Models of Shells (eds A.K. Noor, T. Belytschko and J.C. Simo), San Francisco, Dec. 1989, ASME, New York, 1989, p. 569.

9. Flockhart, C., Woodward, R.L., Lam, Y.C. and O'Donnell, R.G. "The Use of Velocity Discontinuities to Define Shear Failure Trajectories in Dynamic Plastic Deformation", Int. J. Impact Eng., Vol. 11, 1991, pp. 93-106.

10. Kecman, D. "Bending Collapse of Rectangular Section Tubes in Relation to the Bus Roll-Over Problem", Ph.D. Thesis, Cranfield Institute of Technology, Great Britain, 1979.

11. Kecman, D. "Analysis of Framework Type Safety Structures in Road Vehicles", Structural Crashworthiness (eds N. Jones and T. Wierzbicki), Butterworths, London, 1983, p. 371.

12. Mahmood, H.F., Paluszny, A. and Tang, X. "Crash Analysis of Thin-Walled Beam-Type Structures", Proceedings of the 7th International Conference on Vehicle Structural Mechanics, Detroit, SAE Paper No. 880894, SAE, 1988, pp. 103-112.

13. Wierzbicki, T. and Abramovich, W. "CRASH-CAD a Computer Program for the Design of Columns for Optimum Crush", Emerging Technologies in Vehicle CAE and Structural Mechanics, SAE Paper No. 900462, SAE, 1990, pp. 23-32.

14. Miles, J.C. "The Determination of Collapse Load and Energy Absorbing Properties of Thin-Walled Beam Structures Using Matrix Method of Analysis", Int. J. of Mech. Sci., Vol. 18, 1976, pp. 399-405.

15. Miles, J.C. and Wardhill, G.A. "The Analysis and Design of a Fire Engine Safety Cab Using Finite Element Methods", Inst. Mech. Eng., Vol. 191, 29/77, 1977, pp. 187-193.

16. Kecman, D. and Miles, J. "Application of the Finite Element Method to the Door Intrusion and Roof Crush Analysis of a Passenger Car", Int. Conf. on Vehicle Structural Mechanics, SAE Paper No. 790990, SAE, 1979.

17. Tidbury, G.H. and Miles, J.C. "Collapse Analysis of Automotive Structures Using Finite Element Techniques", XVI FISITA Congress, Tokyo, 1976, pp. 8-17 - 8-22.

18. Wardhill, G.A. and Suthurst, G.D. "The Prediction of the Deformation and Movement of the Target Car in Vehicle to Vehicle Side Impact", Inst. Mech. Eng., C314/80, 1980, pp. 113-119.

19. Wardhill, G.A. and Kecman, D. "Theoretical Prediction of the Overall Collapse Mode and Maximum Strength of a Bus Structure in a Roll-Over Situation", XVIII FISITA Congress, Hamburg, Vol. 1, 1980, pp. 281-290.

20. Kecman, D. and Tidbury, G.H. "Theoretical Prediction of the Complete Collapse Behaviour of a Coach Subject to the Proposed Roll-Over Test", XIX FISITA Congress, Melbourne, 1982, pp. 121.1-121.7.

21. Kecman, D., Miles, J.C., Sadeghi, M.M., and Tidbury, G.H. "Investigation of PSV Roll-Over Safety", Inst. Mech. Eng., 1977, C139/77, pp. 67-76.

22. Bruns, H. "Safety of High Speed Luxury Coaches", The Bus '86, Inst. of Mech. Eng., 1986, p. 13.

23. Symonds, P.S. "Viscoelastic Behaviour in Response of Structures to Dynamic Loading", Behaviour of Materials under Dynamic Loading (ed N.J. Huffington, Jr), ASME, 1965, pp. 106-124.

24. Cimpoeru, S.J. "The Flow Stress of a Mild Steel for Dynamic Calculations Involving Large Plastic Deformations", Dynamic Loading in Manufacturing and Service Conference, Melbourne, 9-11 Feb. 1993, IE(Aust.), pp. 17-23.

25. Horne, M.R. "The Plastic Theory of Structures", Thomas Nelson and Sons, 1971.

26. Wang, C.K. "General Computer Program for Limit Analysis", Proc. J. Struct. Div., ASCE, Vol. 89, No ST6, 1963, pp. 101-117.

27. Reddy, J.N. "An Introduction to the Finite Element Method", McGraw-Hill, New York, 1984, p.164.

28. Harrison, H.B. "Structural Analysis and Design, Some Microcomputer Applications, Part 1 and 2", Pergamon, Oxford, 1990, p. 125.

29. Abramowicz, W. and Jones, N. "Dynamic Progressive Buckling of Circular and Square Tubes", Int. J. Impact Eng., Vol. 4, 1986, pp. 243-270.

30. White, G.J., Grzebieta, R.H. and Murray, N.W. "Maximum Strength of Square Thin-walled Sections Subjected to Combined Loading of Torsion and Bending", Int. J. Impact Eng., Vol. 13, 1993, pp. 203-214.

31. Kecman, D. and Djokic, M. "The Effect and Modelling of 'Finite-stiffness Hinges' in the Collapse Analysis of Roll-Over Safety Rings in Buses and Coaches", Int. J. of Vehicle Design, Vol. 11, 1990, pp. 374-384.

32. Vanderbilt, M. "Matrix Structural Analysis", Quantum Publishers, New York, 1974, p. 275.

33. Neal, B.G. "The Plastic Methods of Structural Analysis", Chapman and Hall, London, 1977.

34. Wilson, E.L. "SOLID SAP - A Static Analysis Program for Three-Dimensional Solid Structures", University of California, Berkeley, Report SESM 71-19, Sept. 1971.

35. Cimpoeru, S.J. "BEAM: Source Code", MRL Intellectual Property in Confidence Report, IP No. 19, 1994. Maribyrnong, Vic.: Materials Research Laboratory.

36. Wang, C.K. "Limit Analysis of Continuous Beams and Rigid Frames", Structural Analysis on Microcomputers, Macmillian, New York, 1986, p. 275

37. Mendis, P.A. and Darvall, P. LeP. "Computer Program For Stability Analysis of Softening Frames", Research Report, Dept. of Civil Eng., Monash University, Australia, Report No. 3/1987.

38. PCAP - Plastic Analysis Program, Department of Engineering, Cambridge University.

39. Cimpoeru, S.J. and Murray, N.W. "The Large-Deflection Pure Bending Properties of a Square Thin-Walled Tube", Int. J. Mech. Sci., Vol 35, 1993, pp. 247-256.

Appendix 1

Program BEAM: Input File Layout

```
      PROGRAM BEAM
C ******************************************************************
C
C The following text describes the input data layout:
C
C CARD NO.-description         COMMAND SYNTAX
C
C  1-title              TITLE
C  2-control            NNODE,NNODEL,NMAT,NGEO,NELEM,NBD
C  3-coordinates        I,X,Y,Z
C  4-materials          I,E,PR,G
C  5-geometric prop.    I,AA,J,I2,I3,MF
C  6-connectivity       I,NI,NJ,NK,MTYP,STYP,ETYP
C  7-member actions     M
C  8-                   N,FEA1,FEA2........FEA12
C  9-nodal actions      M
C 10-                   N,F1,F2............F12
C 11-settlements        M
C 12-                   0,VAL1,VAL2,VAL3,VAL4,VAL5,VAL6
C 13-displacements      N,DOF1,DOF2,DOF3,DOF4,DOF5,DOF6
C
C
C  Where:  TITLE            - problem title (60 characters long)
C
C          NNODE            - number of structural nodes
C          NNODEL           - total number of nodes listed (incl. k nodes)
C          NMAT             - number of material types
C          NGEO             - number of geometric sections
C          NELEM            - number of elements
C          NBD              - number of nodal displacement conditions
C
C          I                - Node No. (-ve for k nodes)
C          X(I)             - X coordinate of node <e.g. metres>
C          Y(I)             - Y coordinate of node <e.g. metres>
C          Z(I)             - Z coordinate of node <e.g. metres>
C                             N.B. The nodes should be in sequential order
C
C
C          I                - Material No.
C          E  [MAT(I,1)]    - Young's modulus <e.g. kNm^-2>
C          PR [MAT(I,2)]    - Poisson's ratio
C          G  [MAT(I,3)]    - Shear Modulus <e.g. kNm^-2> [Optional! If set
C                             to zero, G calculated from E/2(1+PR)]
C
C
C          I                - Geometric Property No.
C          AA  [GEO(I,1)]   - Axial Area <e.g. m^2>
C          J   [GEO(I,2)]   - Moment of Inertia abt. 1 axis <e.g. m^4>
C          I2  [GEO(I,3)]   -          "          2 axis <e.g. m^4>
C          I3  [GEO(I,4)]   -          "          3 axis <e.g. m^4>
C          MF  [GEO(I,5)]   - Failure Moment <e.g. kNm>
C
C
C          I                 - Element No.
C          NI   [ICON(I,1)]  - Global node number   I
C          NJ   [ICON(I,2)]  -        "             J
C          NK   [ICON(I,3)]  -        "             K (+ve in +ve quad,
C          MTYP [ICON(I,4)]  - Material No.            -ve in -ve quad)
C          STYP [ICON(I,5)]  - Geometric Property No.
C          ETYP [ICON(I,6)]  - Element Type [Set to zero as not used]
C
C
C
C          M           - no. of elements subjected to Member Actions
C                                                     <e.g. kN or kNm>
C          N           - element no.
```

```
c         FEA1  [A(1)]        - Fixed End Action at element DOF 1
c              |                (local coord)    |
c              |                                 |
c         FEA12 [A(12)]       - Fixed End Action at element DOF 12
c                               (local coord)
c
c
c
c         M                   - no. of nodes subjected to Nodal Actions
c                                                     <e.g. kN or kNm>
c         N                   - node no.
c         F1     [A(1)]       - Nodal Action at node DOF 1
c                |                              |
c                |                              |
c         F6     [A(6)]       - Nodal Action at node DOF 6
c
c
c
c         M                   - no. nodes subjected to settlements/
c                                           specified displacements
c
c         N,VAL1,VAL2,VAL3,VAL4,VAL5,VAL6  node no., [IDOF1-IDOF6]
c                             If VALn='val', this DOF is set to 'val',
c ONLY FOR                    the settlement ,<e.g. metres>
c ELASTIC                     If VALn='0', no value is set for this DOF
c ANALYSIS !                  eg. y settlement of 0.05m : 0,0,0.05,0,0,0,0
c
c
c
c         N,DOF1,DOF2,DOF3,DOF4,DOF5,DOF6  node no., [IDOF1-IDOF6]
c                              If DOFn='0', this DOF is set to
c                              zero displacement
c                              If DOFn='1', the movement of this DOF
c                              is possible and will be determined
c                              eg. 3D fixed end : N,0,0,0,0,0,0
c                              eg. 3D simple support : N,0,0,0,1,1,1
c
c
c*******************************************************************
```

Appendix 2

Program BEAM: Input File

```
2D FRAME
4,7,1,1,3,2
1,0,0,0
2,0,1.025,0
3,1.180,1.025,0
4,1.180,0,0
5,-0.2,0.5,0
6,0 5,1.5,0
7,1.5,0.5,0
1,206E6,0.3,0
1,384E-6,0.2212E-3,147712E-12,147712E-12,3.12
1,1,2,5,1,1,0
2,2,3,6,1,1,0
3,3,4,7,1,1,0
0
1
2,5,0,0,0,0,0
0
1,0,0,0,0,0,0
4,0,0,0,0,0,0
```

Appendix 3

Program BEAM: Output File

```
***********************************************************************************
2D FRAME
***********************************************************************************


CONTROL DATA
************

NUMBER OF NODES IN STRUCTURE              =    4
TOTAL NUMBER OF NODES (INCL. K NODES)     =    7
NUMBER OF MATERIALS                       =    1
NUMBER OF GEOMETRIC SECTIONS              =    1
NUMBER OF BEAM ELEMENTS                   =    3
NUMBER OF BOUNDARY CONDITIONS             =    2


NODAL COORDINATES TABLE
***********************

NODE            X              Y              Z

   1         0.0000         0.0000         0.0000
   2         0.0000         1.0250         0.0000
   3         1.1800         1.0250         0.0000
   4         1.1800         0.0000         0.0000
   5        -0.20[illegible]         0.5000         0.0000
   6         0.5000         1.5000         0.0000
   7         1.5000         0.5000         0.0000


MATERIAL PROPERTIES
*******************

MATERIAL    YOUNG S        POISSON S      SHEAR
            MODULUS        RATIO          MODULUS

    1     0.2060E+09       0.3000       0.7923E+08


GEOMETRIC PROPERTIES
********************

SECTION  AX.AR.       J            I2           I3           MF

  1  0.3840E-03  0.2212E-03  0.1477E-06  0.1477E-06  0.3120E+01


ELEMENT CONNECTIVITY TABLE
**************************

ELEMENT    NODE1    NODE2     K    MATERIAL  SECTION  ELEMENT TYPE

   1         1        2       5       1         1         0
   2         2        3       6       1         1         0
   3         3        4       7       1         1         0


APPLIED NODAL ACTIONS IN TERMS OF GLOBAL AXES
*********************************************

JOINT NO. X FORCE  Y FORCE  Z FORCE  X MOMENT  Y MOMENT  Z MOMENT

    2       5.00     0.00     0.00     0.00      0.00      0.00
```

KINEMATIC BOUNDARY CONDITIONS
****

NODE	DOF1	DOF2	DOF3	DOF4	DOF5	DOF6
1	0.	0.	0.	0.	0.	0.
4	0.	0.	0.	0.	0.	0.

****
********* END OF INPUT DATA *********
****

PLASTIC HINGE NO. 1 IS FORMED IN ELEMENT 1
AT NODE 1 WHEN LOAD FACTOR= 2.09310

CUMULATIVE DISPLACEMENTS
****

TRANSLATIONS				ROTATIONS		
JOINT	X	Y	Z	X	Y	Z
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	0.2296E-01	0.4937E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.1465E-01
3	0.2288E-01	-0.4937E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.1457E-01
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

CUMULATIVE STRESS-RESULTANTS
****

MEM	NODE	AXIAL	X MOMENT	Y MOMENT	Z MOMENT	FAILURE MOMENT
1	1	-0.3810E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
1	2	0.3810E+01	0.0000E+00	0.0000E+00	0.2250E+01	0.3120E+01
2	2	0.5226E+01	0.0000E+00	0.0000E+00	-0.2250E+01	0.3120E+01
2	3	-0.5226E+01	0.0000E+00	0.0000E+00	-0.2246E+01	0.3120E+01
3	3	0.3810E+01	0.0000E+00	0.0000E+00	0.2246E+01	0.3120E+01
3	4	-0.3810E+01	0.0000E+00	0.0000E+00	0.3111E+01	0.3120E+01

CUMULATIVE MEMBER DISPLACEMENTS
****

MEM	NODE	AXIAL	X ROT	Y ROT	Z ROT
1	1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	2	0.4937E-04	0.0000E+00	0.0000E+00	-0.1465E-01
2	2	0.2296E-01	0.0000E+00	0.0000E+00	-0.1465E-01
2	3	0.2288E-01	0.0000E+00	0.0000E+00	-0.1457E-01
3	3	0.4937E-04	0.0000E+00	0.0000E+00	-0.1457E-01
3	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

****

PLASTIC HINGE NO. 2 IS FORMED IN ELEMENT 3
AT NODE 4 WHEN LOAD FACTOR= 2.09682

CUMULATIVE DISPLACEMENTS
****

TRANSLATIONS				ROTATIONS		
JOINT	X	Y	Z	X	Y	Z
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	0.2303E-01	0.4948E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.1467E-01
3	0.2295E-01	-0.4948E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.1462E-01
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

CUMULATIVE STRESS-RESULTANTS
****

MEM	NODE	AXIAL	X MOMENT	Y MOMENT	Z MOMENT	FAILURE MOMENT
1	-1	-0.3819E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
1	2	0.3819E+01	0.0000E+00	0.0000E+00	0.2254E+01	0.3120E+01
2	2	0.5241E+01	0.0000E+00	0.0000E+00	-0.2254E+01	0.3120E+01
2	3	-0.5241E+01	0.0000E+00	0.0000E+00	-0.2252E+01	0.3120E+01
3	3	0.3819E+01	0.0000E+00	0.0000E+00	0.2252E+01	0.3120E+01
3	4	-0.3819E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01

CUMULATIVE MEMBER DISPLACEMENTS
****

MEM	NODE	AXIAL	X ROT	Y ROT	Z ROT
1	-1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	2	0.4948E-04	0.0000E+00	0.0000E+00	-0.1467E-01
2	2	0.2303E-01	0.0000E+00	0.0000E+00	-0.1467E-01
2	3	0.2295E-01	0.0000E+00	0.0000E+00	-0.1462E [illegible]
3	3	0.4948E-04	0.0000E+00	0.0000E+00	-0.1462E-01
3	4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

****

PLASTIC HINGE NO. 3 IS FORMED IN ELEMENT 2
AT NODE 2 WHEN LOAD FACTOR= 2.43456

CUMULATIVE DISPLACEMENTS
****

TRANSLATIONS				ROTATIONS		
JOINT	X	Y	Z	X	Y	Z
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	0.3876E-01	0.6849E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.2030E-01
3	0.3867E-01	-0.6849E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.2024E-01
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

CUMULATIVE STRESS-RESULTANTS
****

MEM	NODE	AXIAL	X MOMENT	Y MOMENT	Z MOMENT	FAILURE MOMENT
1	-1	-0.5286E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
1	2	0.5286E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
2	2	0.6085E+01	0.0000E+00	0.0000E+00	-0.3120E+01	0.3120E+01
2	3	-0.6085E+01	0.0000E+00	0.0000E+00	-0.3117E+01	0.3120E+01
3	3	0.5286E+01	0.0000E+00	0.0000E+00	0.3117E+01	0.3120E+01
3	-4	-0.5286E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01

CUMULATIVE MEMBER DISPLACEMENTS
****

MEM	NODE	AXIAL	X ROT	Y ROT	Z ROT
1	-1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	2	0.6849E-04	0.0000E+00	0.0000E+00	-0.2030E-01
2	2	0.3876E-01	0.0000E+00	0.0000E+00	-0.2030E-01
2	3	0.3867E-01	0.0000E+00	0.0000E+00	-0.2024E-01
3	3	0.6849E-04	0.0000E+00	0.0000E+00	-0.2024E-01
3	-4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

****

PLASTIC HINGE NO. 4 IS FORMED IN ELEMENT 2
AT NODE 3 WHEN LOAD FACTOR= 2.43512

CUMULATIVE DISPLACEMENTS
****

TRANSLATIONS				ROTATIONS		
JOINT	X	Y	Z	X	Y	Z
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	0.3883E-01	0.6852E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.2037E-01
3	0.3874E-01	-0.6852E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.2028E-01
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

CUMULATIVE STRESS-RESULTANTS
****

MEM	NODE	AXIAL	X MOMENT	Y MOMENT	Z MOMENT	FAILURE MOMENT
1	-1	-0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
1	2	0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
2	-2	0.6088E+01	0.0000E+00	0.0000E+00	-0.3120E+01	0.3120E+01
2	3	-0.6088E+01	0.0000E+00	0.0000E+00	-0.3120E+01	0.3120E+01
3	3	0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
3	-4	-0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01

CUMULATIVE MEMBER DISPLACEMENTS
****

MEM	NODE	AXIAL	X ROT	Y ROT	Z ROT
1	-1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	2	0.6852E-04	0.0000E+00	0.0000E+00	-0.2037E-01
2	-2	0.3883E-01	0.0000E+00	0.0000E+00	-0.2037E-01
2	3	0.3874E-01	0.0000E+00	0.0000E+00	-0.2028E-01
3	3	0.6852E-04	0.0000E+00	0.0000E+00	-0.2028E-01
3	-4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

****

DEFORMATIONS LARGER THAN 0.110E+02 IN CYCLE NO. 5

CUMULATIVE DISPLACEMENTS
****

	TRANSLATIONS			ROTATIONS		
JOINT	X	Y	Z	X	Y	Z
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	0.3883E-01	0.6852E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.2037E-01
3	0.3874E-01	-0.6852E-04	0.0000E+00	0.0000E+00	0.0000E+00	-0.2028E-01
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

CUMULATIVE STRESS-RESULTANTS
****

MEM	NODE	AXIAL	X MOMENT	Y MOMENT	Z MOMENT	FAILURE MOMENT
1	-1	-0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
1	2	0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
2	-2	0.6088E+01	0.0000E+00	0.0000E+00	-0.3120E+01	0.3120E+01
2	-3	-0.6088E+01	0.0000E+00	0.0000E+00	-0.3120E+01	0.3120E+01
3	3	0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01
3	-4	-0.5288E+01	0.0000E+00	0.0000E+00	0.3120E+01	0.3120E+01

CUMULATIVE MEMBER DISPLACEMENTS
****

MEM	NODE	AXIAL	X ROT	Y ROT	Z ROT
1	-1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
1	2	0.6852E-04	0.0000E+00	0.0000E+00	-0.2037E-01
2	-2	0.3883E-01	0.0000E+00	0.0000E+00	-0.2037E-01
2	-3	0.3874E-01	0.0000E+00	0.0000E+00	-0.2028E-01
3	3	0.6852E-04	0.0000E+00	0.0000E+00	-0.2028E-01
3	-4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

****

UNFACTORED DISPLACEMENTS AFTER LAST SOLUTION
****

	TRANSLATIONS			ROTATIONS		
JOINT	X	Y	Z	X	Y	Z
1	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
2	-0.4993E+12	-0.2788E-22	0.0000E+00	0.0000E+00	0.0000E+00	0.4871E+12
3	-0.4993E+12	-0.5473E-23	0.0000E+00	0.0000E+00	0.0000E+00	0.4871E+12
4	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

REPORT NO.	AR NO.	REPORT SECURITY CLASSIFICATION
MRL-TR-94-6	AR-008-636	Unclassified

TITLE

BEAM: A finite element program for the collapse analysis of vehicle structures

AUTHOR(S)	CORPORATE AUTHOR
S.J. Cimpoeru	DSTO Materials Research Laboratory PO Box 50 Ascot Vale Victoria 3032

REPORT DATE	TASK NO.	SPONSOR
June, 1994	ARM 92/216	Army

FILE NO.	REFERENCES	PAGES
G6/4/8-4596	39	38

CLASSIFICATION/LIMITATION REVIEW DATE	CLASSIFICATION/RELEASE AUTHORITY
	Chief, Ship Structures & Materials Division

SECONDARY DISTRIBUTION

Approved for public release

ANNOUNCEMENT

Announcement of this report is unlimited

KEYWORDS

Finite Element Modelling Crash Analysis	Load-Deflection	Thin-Walled Vehicle Structures

ABSTRACT

BEAM (Beam Element Analysis with Mechanisms) is a finite element program for the analysis of the collapse of vehicle structures. BEAM can quickly estimate the collapse load and plastic hinge locations in three-dimensional thin-walled frames that would collapse in bending. Such information can be combined with separate rigid-plastic modelling to enable the large-deflection design of vehicle frameworks so that their collapse in roll-over accidents, for instance, can be prevented. One advantage of BEAM is that it allows a user to define his own experimentally or analytically determined loads and moments as failure criteria. The theory and operation of BEAM are presented and its application to the modelling of a collapsing two-dimensional frame is described.

BEAM: A Finite Element Program for the Collapse Analysis of Vehicle Structures

S.J. Cimpoeru

(MRL-TR-94-6)

DISTRIBUTION LIST

Director, MRL
Chief, Ship Structures and Materials Division
Dr J.C. Ritter
Dr S.J. Cimpoeru
Mr G.M. Weston
DWVP-A, Russell Offices, Canberra
MRL Information Services

Chief Defence Scientist (for CDS, FASSP, ASSCM)	1 copy only
Director (for Library), Aeronautical Research Laboratory	
Head, Information Centre, Defence Intelligence Organisation	
OIC Technical Reports Centre, Defence Central Library	
Officer in Charge, Document Exchange Centre	8 copies
Army Scientific Adviser, Russell Offices	
Air Force Scientific Adviser, Russell Offices	
Scientific Adviser - Policy and Command	
Director General Force Development (Land)	
Senior Librarian, Main Library DSTOS	
Librarian, DSD, Kingston ACT	
Serials Section (M List), Deakin University Library, Deakin University, Geelong 3217	
NAPOC QWG Engineer NBCD c/- DENGRS-A, HQ Engineer Centre, Liverpool Military Area, NSW 2174	
ABCA, Russell Offices, Canberra ACT 2600	4 copies
Librarian, Australian Defence Force Academy	
Head of Staff, British Defence Research and Supply Staff (Australia)	
NASA Senior Scientific Representative in Australia	
INSPEC: Acquisitions Section Institution of Electrical Engineers	
Head Librarian, Australian Nuclear Science and Technology Organisation	
Senior Librarian, Hargrave Library, Monash University	
Library - Exchange Desk, National Institute of Standards and Technology, US	
Exchange Section, British Library Document Supply Centre	
Periodicals Recording Section, Science Reference and Information Service, UK	
Library, Chemical Abstracts Reference Service	
Engineering Societies Library, US	
Documents Librarian, The Center for Research Libraries, US	
Librarian - MRL Sydney - data sheet only	
Navy Scientific Adviser - data sheet only	
DASD, APW2-1-OA2, Anzac Park West, Canberra ACT - data sheet only	
SO (Science), HQ 1 Division, Milpo, Enoggera, Qld 4057 - data sheet only	
Counsellor, Defence Science, Embassy of Australia - data sheet only	
Counsellor, Defence Science, Australian High Commission - data sheet only	
Scientific Adviser to DSTC Malaysia, c/- Defence Adviser - data sheet only	
Scientific Adviser to MRDC Thailand, c/- Defence Attache - data sheet only	

Dr M.M. Attard, Dept. of Structural Engineering, UNSW, PO Box 1, Kensington 2033	
Dr R.H. Grzebieta, Dept. of Civil Engineering, Monash University, Clayton 3168	4 copies
Mr N. Sneath, Dept. of Civil Engineering, Monash University, Clayton 3168	
Prof N.W. Murray, Dept. of Civil Engineering, Monash University, Clayton 3168	
Mr T. Creedy, Vehicles Group, EDE, Raleigh Road, Maribyrnong 3032	
Mr D. Ludowici, Vehicles Group, EDE, Raleigh Road, Maribyrnong 3032	