THE USE OF COMPUTER GRAPHICS IN THE STIFFNESS APPROACH TO STRESS ANALYSIS

by

C. R. Carman, Universidad Industrial de Santander
Bucaramanga, Columbia

R. Hangartner, Warner and Swasey Co.
Solon, Ohio

R. J. Pick, University of Waterloo
Waterloo, Ontario

1.0 INTRODUCTION

In recent years computer programs utilizing the stiffness method have been developed to analyze previously intractable stress and structural analysis problems (1). The technique consists of dividing a body into small elements of defined or known mechanical behaviour. In structures, these would be beams, columns, tension members, etc., while for a continuum the body would be divided into finite elements (2). The force-displacement behaviour of each element can be described by one or more stiffness equations which when solved simultaneously with the equations resulting from other elements, describe the mechanical behaviour of the total body.

The correct representation of the body is highly dependent on the number, type, size and position of the elements or structural members chosen. As the number of elements or members is increased, the accuracy of the analysis improves but the solution requires more memory, computing time and input data. Thus a judious choice of elements is required to properly represent the body and remain within the machine capacity.

For simple or regular shaped bodies, mesh generation programs can be written to produce a grid of finite elements or structural members (3). However for more complex bodies it is usually necessary to prepare the mesh by hand using the experience gained from previous analyses and tailoring the mesh to fit the machine capacity. In this case the preparation of the input data; drawing of the mesh, tabulating, card punching and checking, is usually tedious and time consuming. The use of interactive computer graphics can aid in the preparation of this data with a considerable saving in time and effort.

From the point of view of the designer of structural components, the graphical preparation of data for computer programs is much more amenable to current drafting practices than preparation of input data through numerical tabulation and card punching. Thus the use of computer graphics should promote the use of some of the more sophisticated computerized techniques by making the preparation of the input data similar to the presentation of designs by drawing. The graphics terminal, however, cannot completely represent an engineering drawing and, therefore, it is necessary to determine what should be displayed and how this can support and supply data to an analysis program.

The optimizing of designs is often done through successive design iterations, each one being an improvement on the last. In many cases an experienced designer will be able to approach an optimum quickly unless each design change requires a difficult or long analysis for strengths, weights, cost,etc. The interactive nature of computer graphs can aid this procedure considerably if the design can be created and altered on the graphical display. Concurrently, the computer can be used to calculate design data such as strengths, deflections, weights, etc. utilizing analyses such as the stiffness method. In this way the designer can gain a great deal of experience with respect to optimizing the design without becoming involved in detailed calculations for each design iteration.

The authors have developed two programs to investigate computer graphics as an aid to the stiffness method of stress analysis and to determine the features necessary for practical applications.

2.0 ANALYSIS OF PIN JOINTED TRUSSES

A program utilizing the stiffness method and graphical input and output of data has been developed to investigate the optimisation of designs of pin jointed trusses. The geometry, loading and supports of a two dimensional pin jointed truss can be entered into the program graphically and numerical values such as member areas, etc. can be entered with a teletype. Using this data the analysis program will determine and display deformations and stresses. The design can then be altered and the modified design analyzed. Figures 1(a) to 1(e) illustrate this procedure with the design of a simple bridge truss and Figure 1(f) illustrates the results for a cantilever truss.

The program was used to investigate methods of inputing and displaying data and to study the optimisation of designs by trial and error.

It was found to be relatively easy to represent a two dimensional pin jointed structure by a simple line drawing (Figure 1(a)). Supports and loadings can be indicated simply (Figure 1(b)). The drawing of the structure is accomplished with a tracking cross and light pen. This is not satisfactory since it is difficult to draw symmetrical structures or structures with specified dimensions. The tracking cross should be independently controlled along each of the two axis and provision made for displaying cross coordinates so that structures can be created to specified dimensions. The necessity of typing in member characteristics such as the cross-sectional area and modulus of elasticity is not convenient although preferable to entering numerical data graphically. However, practical designs are often limited to standard components, e.g. structural steel sizes. If this could be provided as a menu then the designer could choose the desired element properties from the menu.

Figure 1(b) illustrates the prepared data prior to the analysis of the truss. The stiffness method determines the displacement of each joint and the loads and stresses (Figure 1(c)) in each member. These displacements are scaled up and displayed as in Figure 1(e). The displacements do not appear particularly useful in suggesting design improvements unless deflection is a design restraint. In general the stresses in each member appear to be the most useful in optimising the design with respect to strength.

Displaying the stresses beside each member proved unsatisfactory as the notation often crossed the member or other members, making reading difficult. The final technique was to number each member and to tabulate the values (Figure 1(c)). Although not the most desirable technique, stresses from previous designs can also be tabulated to allow comparison.

After each analysis the area of any member can be altered and the analysis repeated. It was found that after 3 or 4 alterations a design could be produced with equal stresses in each member, i.e. an optimum design with respect to strength (Figure 1(e)).

This technique is very useful but becomes more difficult if two or more variables are considered such as member size and the structural geometry. However the ease of altering and analyzing designs suggests that this technique would be very attractive to designers. This is particularly true for the case of three dimensional structures which are difficult to analyze and represent on paper. Three dimensional graphical presentation, the use of perspective and rotation of the structure should enhance this use of computer graphics. The authors do not forsee any difficulty in expanding the program to three dimensions and to analyze more complex structures such as frames, etc. Expansion of the program may lead to problems of program size and computation time. The stiffness method of analysis generally requires memory core storage of large matrices. These matrices plus large graphical routines may increase the program size to greater than the machine capacity unless the analysis and graphics are run as separate programs. The DEC-PDP-9-340 system at the University of Waterloo can provide graphical support for the analysis of structures of up to 20 members using 32K words of memory.

3.0 FINITE ELEMENT MESH GENERATION

The stiffness method applied to the analysis of continua is generally termed finite element analysis, the continum being represented by elements of finite size.

The simplest finite element used in two dimensional analysis is the constant stress/strain triangle. If stress gradients are to be properly represented with this type of element many small elements are required. More sophisticated elements can be used (4) but often lead to increased computation and more difficulty in fitting the element to complex shapes. Two typical meshes of finite elements are shown in Figure 2 and represent sections of Buttress and Acme screw threads (5). In this case the elements are triangular rings considered to have constant stress and strain throughout.

The number of elements is usually limited by the memory size of the computer; the analysis of Figure 2 requires 450 K bytes of memory on an IBM 360/75. The preparation of input data of this type is often tedious but very critical in the accuracy of the analysis.

To support the finite element work being done at the University of Waterloo a graphics program has been developed (6) to prepare finite element analysis input data.

Since there are numerous other numerical techniques that require irregular meshes or grids of elements it is useful to describe the program features. The program was developed specifically for two dimensional triangular elements, however the techniques are applicable to almost any type of element or mesh.

After initial studies (7) it became obvious that drawing individual elements with a light pen was desirable but slow and tedious for large meshes (greater than 50 elements). Therefore the computer was used to create and display regular sets of grids (Figure 3(b)) and it was found convenient to join and distort these to fit the shape of the object (Figures 3(c)(d)). The program was arranged so that the grids could have different sizes of elements allowing smaller elements to be concentrated in areas of high stress/strain gradient. Apart from its speed this technique has advantages in the numbering of the nodes (the corners of the triangles). If the nodes are numbered correctly the matrices created in the stiffness analysis will be banded. Storage of only the band will give a considerable saving in the memory required. This is essential in all but the smallest analyses. The regular grids used can be numbered in this optimum way as they are produced. Distortion of the mesh and addition of joining elements between grids will not alter the optimum numbering system.

If in adding elements new nodes are created the optimum numbering system will be disrupted. A routine has been incorporated to renumber the grids into a system which minimizes the matrix bandwidth. This routine requires some operator assistance since there are situations in which the numbering system can only be partially optimised and a number of different numbering schemes may be tried to minimize the matrix bandwidth.

When preparing the mesh by distortion of the grids it is desirable to also display an accurate drawing of the object (Figure 3(a)). The elements should fit the outline of the object but need not be placed accurately in the component interior. The component shape can be entered from a sketch routine using the light pen, however a more accurate drawing can be made by entering points on the outline using punched cards. Curved surfaces will be represented by straight lines, as well as straight sided elements and therefore, the number of points on the curve should correspond to the number of elements for the most accurate representation of the curved surface.

To aid in the fitting of elements to the outline a routine to scale up the area of interest is incorporated. Using this technique the grids can be more accurately distorted to fit the outline of the component.

To summarize it was found that two dimensional finite element grids can be created most efficiently with the following routines:-

1) A routine to read and display points on the component boundary.
2) A routine to read and display an existing mesh which is to be modified.
3) A routine to produce grids of regular elements with an optimum numbering scheme.
4) A routine to scale up the display.
5) A routine to distort the grids to fit the component outline.
6) A routine to add single elements and to join the elements in each grid.
7) A routine to renumber the nodes to minimize the stiffness matrix bandwidth.
8) An output routine to produce a data set or deck of punched cards and a permanent record (calcomp plot).

The calling of these subroutines is controlled from a push button box and any routine may be called at any time e.g. the component may be scaled up before adding grids to be able to place a very fine grid on the part.

Although this technique of creating finite element meshes is considerably different from the usual manual methods it was found to be easily learned and very convenient.

The current program is capable of designing grids of 200-300 elements being limited by the size of the computer memory. Expansion of the program to three dimensions will be restricted considerably by memory core size. In such cases only parts of the component need be displayed and the remaining mesh data stored out of core, however this may lead to difficulties in visualizing the component, particularly in three dimensions.

The size of the finite element analysis program is such that it cannot be run on the PDP-9 used for the preparation of the mesh data. The data set created is therefore transmitted to the IBM 360/75 in the university computing center for the analysis and the output stored as a data set to be returned to the PDP-9. Display of this data is the subject of future studies.

The output data consists of a set of displacements, strains and stresses for each node. Since there may be many nodes it will not be possible to display all of this data and editing will be required to present only highlights or points of interest. Contour plotting is a possibility but will require a sophisticated program due to the complexity of the element grid.

The possibility of interactively editing finite element data has considerable potential. Values of stresses, strains, etc. are at discrete points which often must be extrapolated to points of interest and surfaces. These are often the areas of maximum stresses and therefore much care is required in the extrapolation and in the treatment of the data. The presentation of this data in tabular form often makes this process tedious and difficult to visualize. The use of interactive computer graphics could simplify this problem considerably.

4.0 CONCLUSION

The use of interactive computer graphics can be applied to the stiffness approach of stress analysis to provide a design tool in a form useful to the designer. However, in the development of such a system, one must determine what is the most useful data to display to represent the component while providing sufficient information for analysis. Similarly, the usual techniques of engineering drawing are often not efficient or accurate in creating displays of engineering components or graphical preparation of input data for analysis programs. In such cases, modified drawing techniques are necessary such as in the finite element mesh generation program.

In many situations the graphical presentation of results (such as the deflected shape of a structure) will not be as useful as simply displaying tabulated values (stresses). However, when large amounts of data result from the analysis this is not practical and a form of editing will be necessary to only display highlights of the analysis.

Modifications of designs and repetition of design calculations can be easily accomplished through the use of interactive computer graphics. This can lead to the optimisation of a design by simple trial and error and with the experience gained from previous analyses.

The use of interactive computer graphics in the design and analysis of structural components appears to be advantageous, however, it requires considerable development in the graphical representation of structures and design data.

ACKNOWLEDGEMENTS

We would like to thank Dr. D. J. Burns for his encouragement and advice.

The present research has been supported by the National Research Council of Canada.

REFERENCES

1. Przemieniecki, J.S., Theory of Matrix Structural Analysis, McGraw Hill, 1968.

2. Zienkiewicz, O.C., The Finite Element Method in Structural and Continuum Mechanics, McGraw-Hill, 1967.

3. Frederick, C.O., Wong, Y.C., Edge, F.W., Two Dimensional Automatic Mesh Generation for Structural Analysis, International Journal for Numerical Methods in Engineering, Vol. 2, pp 133-144, 1970.

4. Ergatoudis, I., Irons, B., Zienkievicz, O., Curved Isoparametric Quadrilateral Elements for Finite Element Analysis, International Journal of Solids and Structures, Vol. 4(1), pp 31-42, January, 1968.

5. Pick, R. J., Computer-Aided Design of Thick-Walled High Pressure Vessels. Ph.D. Thesis, University of Waterloo, 1970.

6. Carman, C.R., An Interactive Mesh Generation Program for Triangular Finite Elements, M.A.Sc. Thesis, University of Waterloo, 1970.

7. Carman, C.R. and Pick, R. J., Interactive Graphical Generation of Finite Element Analysis Input Data, Decus Proceeding, Fall 1969 Symposium.

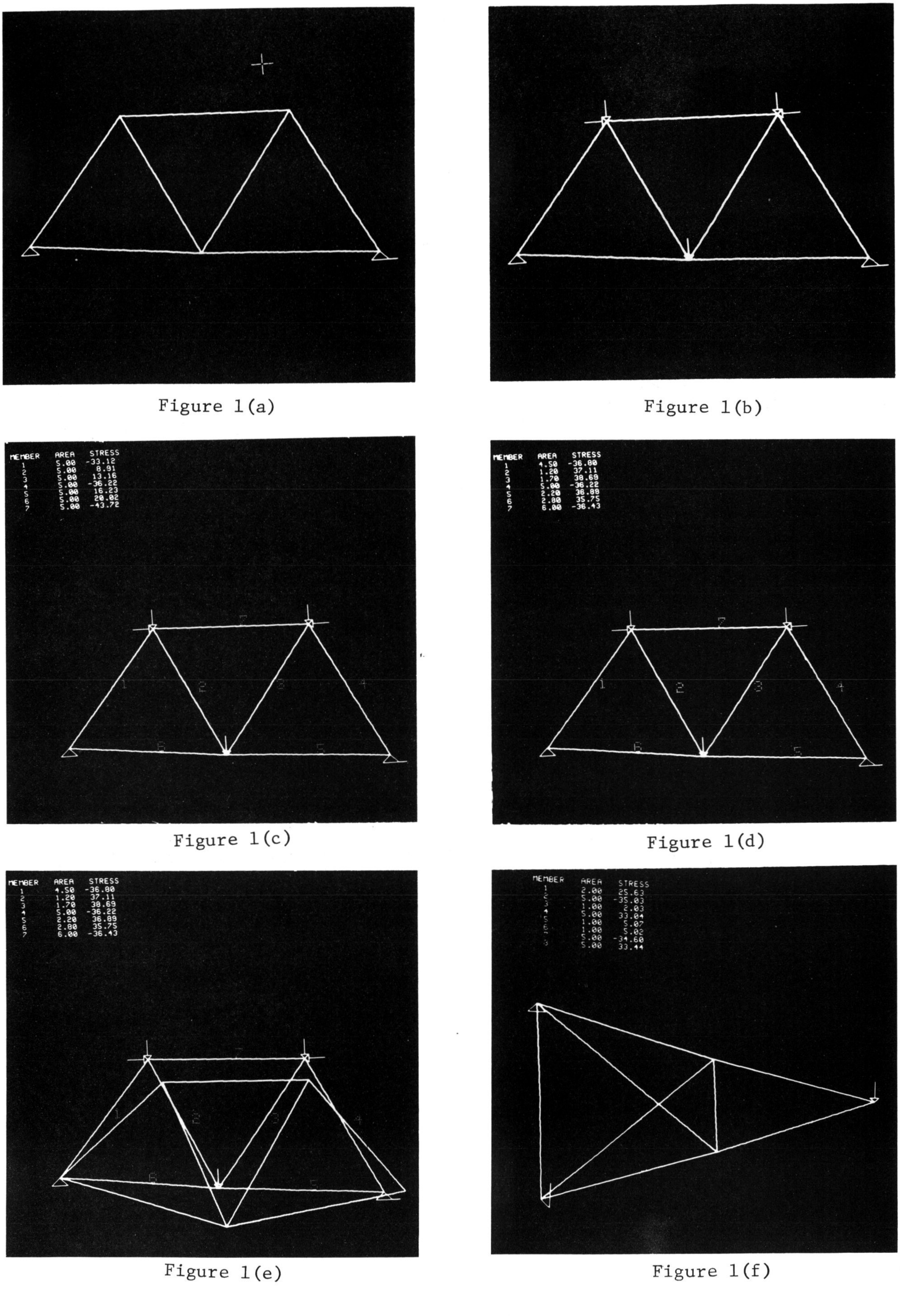

Figure 1(a)

Figure 1(b)

Figure 1(c)

```
MEMBER    AREA   STRESS
   1      5.00  -33.12
   2      5.00    8.91
   3      5.00   13.16
   4      5.00  -36.22
   5      5.00   16.23
   6      5.00   20.02
   7      5.00  -43.72
```

Figure 1(d)

```
MEMBER    AREA   STRESS
   1      4.50  -36.80
   2      1.20   37.11
   3      1.70   38.69
   4      5.00  -36.22
   5      2.20   36.89
   6      2.80   35.75
   7      6.00  -36.43
```

Figure 1(e)

```
MEMBER    AREA   STRESS
   1      4.50  -36.80
   2      1.20   37.11
   3      1.70   38.69
   4      5.00  -36.22
   5      2.20   36.89
   6      2.80   35.75
   7      6.00  -36.43
```

Figure 1(f)

```
MEMBER    AREA    STRESS
   1      2.00    25.63
   2      5.00   -35.03
   3      1.00     2.03
   4      5.00    33.04
   5      1.00     5.07
   6      1.00     5.02
   7      5.00   -34.60
   8      5.00    33.44
```

Figure 1 Graphical Input for the Analysis of Pin Jointed Trusses

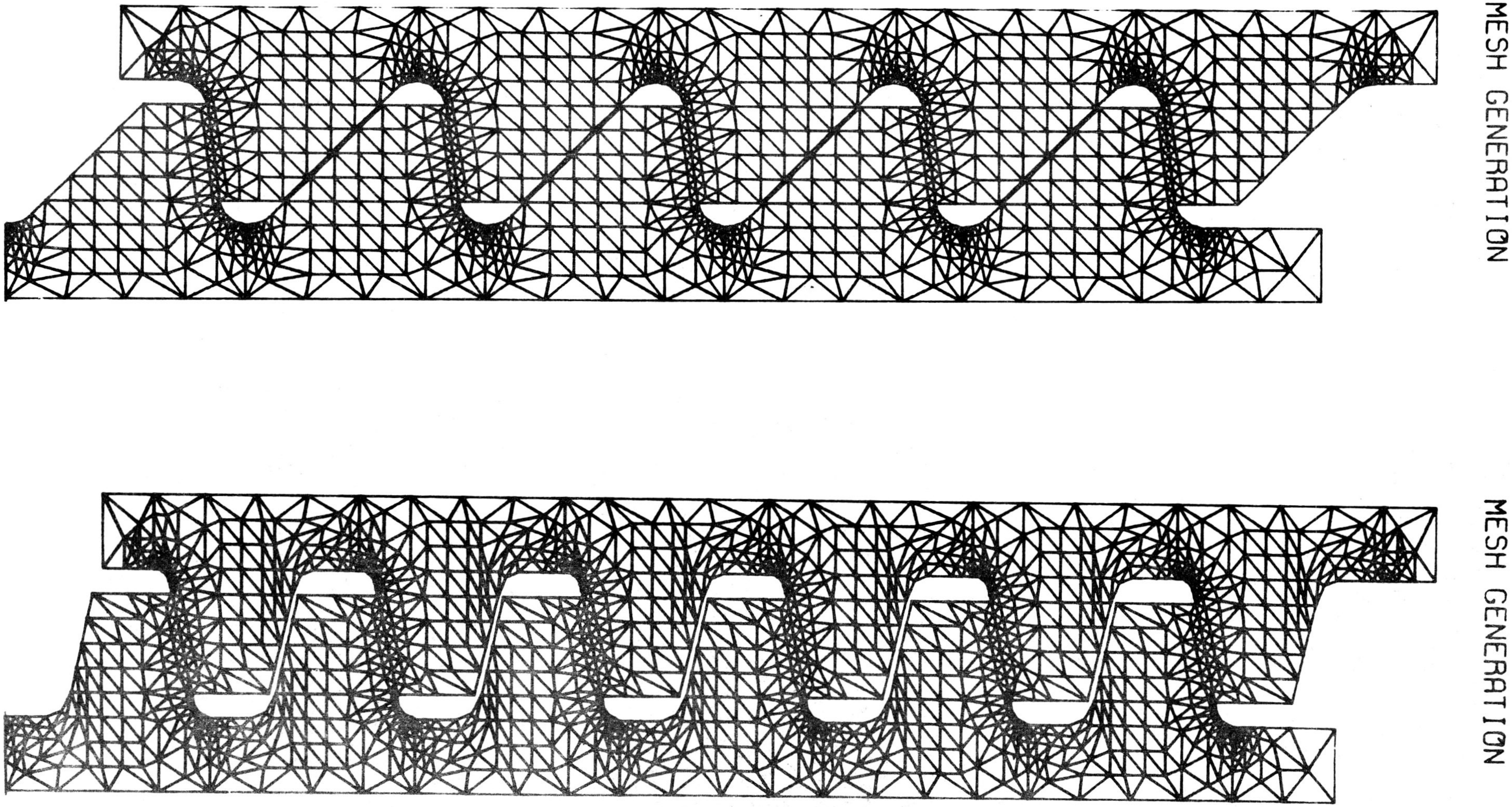

Figure 2 Finite element meshes of the Buttress and Acme thread

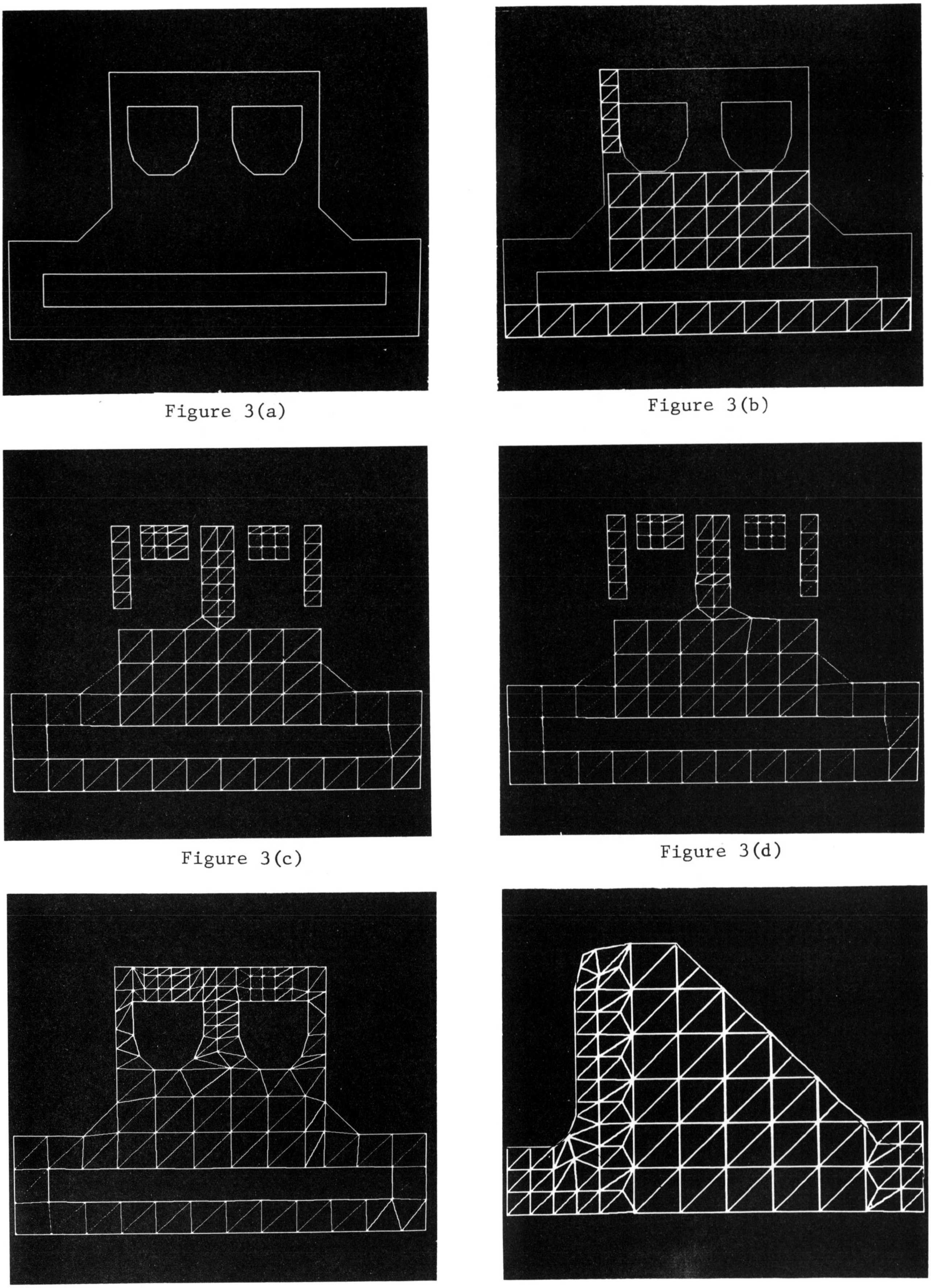

Figure 3(a)

Figure 3(b)

Figure 3(c)

Figure 3(d)

Figure 3(e)

Figure 3(f)

Figure 3 Graphical Generation of Finite Element Meshes for a Seat Belt Clamp and a Buttress Thread