UK Atomic Energy Authority, Northern Division
Risley Nuclear Power Development Establishment
Risley, Warrington

Development validation and use of computer codes for inelastic analysis

by

D A JOBSON

Presented at the IWGFR Specialists Meeting
on Advances in Structural Analysis for
LMFBR Applications

Paris, 11-15 October 1982

List of Figures

An introduction to the UNCLE finite element scheme

A finite element scheme is a system which provides routines to carry out the operations which are common to all finite element programs. The list of items that can be provided as standard by the finite element scheme is surprisingly large and the list provided by the UNCLE finite element scheme is *unusually comprehensive.* They are:

- (i) finite element mesh generation;
- (ii) input of nodal and element properties;
- (iii) standard restraint conditions;
- (iv) standard loading (source etc.) conditions;
- (v) node ordering;
- (vi) generation of linear (or linearised) global equations from element equations;
- (vii) solution of the linearised equations;
- (viii) standard printed output and general output editing facilities;
- (ix) *graphical output facilities;*
- (x) eigenvalue analysis;
- (xi) general recursive calculational routes for the implementation of non-linear and time-dependent solution algorithms;
- (xii) table-handling routines to deal with the immensely varied data requirements encountered.

FIGURE 1A

A finite element program* is constructed by incorporating specific elements, boundary and loading conditions, specific forms of material data input, and the output of results specific to the field of application. Within this field, however, the program that results is very general. A characteristic of UNCLE is the ease with which new elements, boundary conditions and loads are incorporated.

** Three such programs in the UNCLE scheme are:*

(a) FAUN[2] which covers frameworks, pipeworks and shells

(b) CAUSE[3] which covers the elastic and inelastic stress-analysis of continua in 2 and 3 dimensions.

(c) TAU[4] which covers heat transfer in 2 and 3 dimensions.

FIGURE 1B CONSTRUCTION OF A FINITE ELEMENT PROGRAM

FIGURE 1C: SETTING UP A FINITE ELEMENT MESH

Arrays

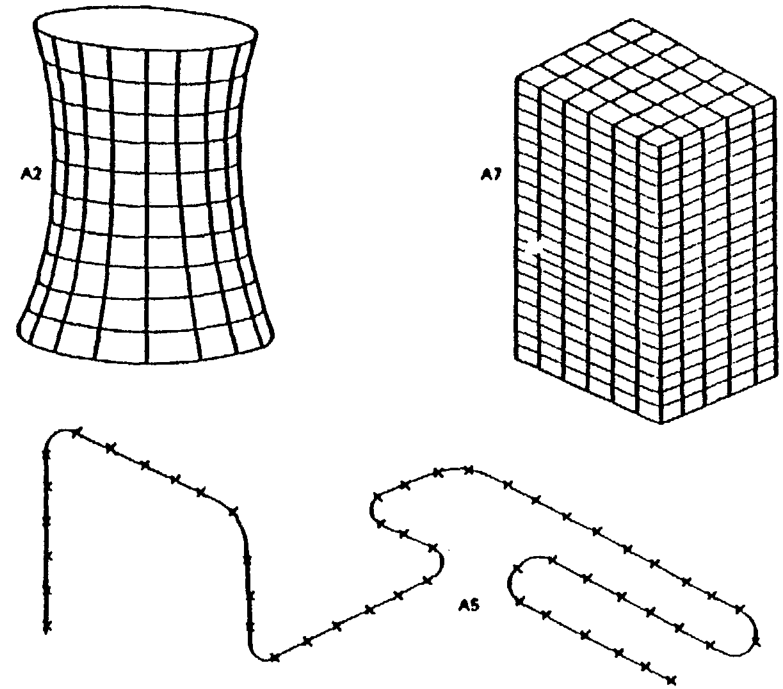
The breaking down of the structure into its elements is purely a matter of describing element connections (the topology of the structure).

The natural first step in analysing the structure is to divide it up into topologically simpler parts, known in UNCLE as ARRAYS. An array may be any one, two or three-dimensional shape, which lends itself to being covered by a reasonably uniform one, two or three-dimensional mesh.

An array may be joined to any other arrays or to itself in any manner which is natural and convenient.

Cells

The next step in the analysis is to break down the arrays into Cartesian meshes by specifying the number of cells to each dimension of the mesh.

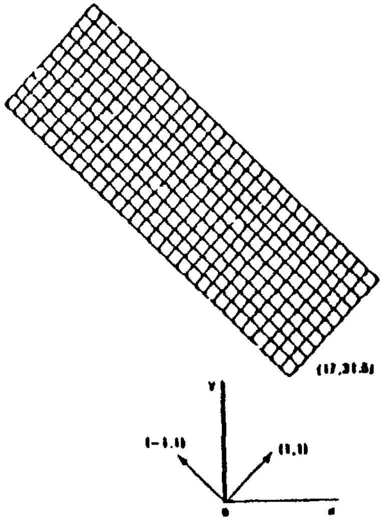


ARRAYS

* 2,10 & CL 12
* 5,50
* 7,5 & 6 & 20

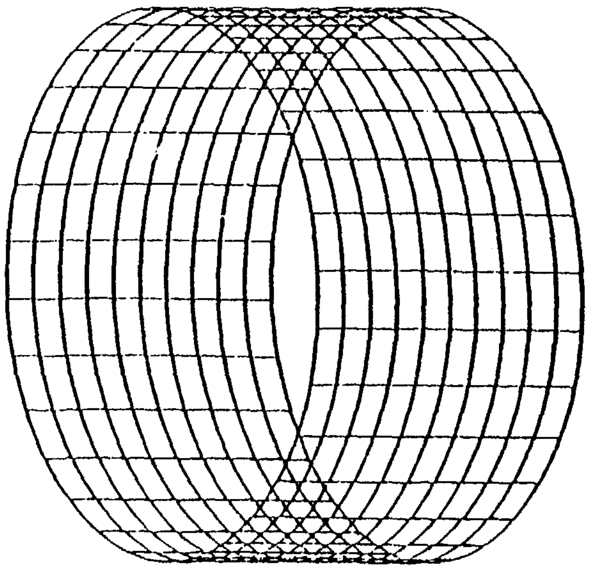
where array 2 is a 2-dimensional 10 by 12 mesh closed in the second direction; array 5 is a 1-dimensional mesh of 50 cells and array 7 is a 3-dimensional mesh

FIGURE 1D: GENERATION OF CO-ORDINATES




```
CELL = 31(11(0:3 & <0:9>))
```

To take the instance of a rectangular array wrapped around a cylinder parallel to the x-axis The y and z co-ordinates will be $r \cos \theta$ and $r \sin \theta$.

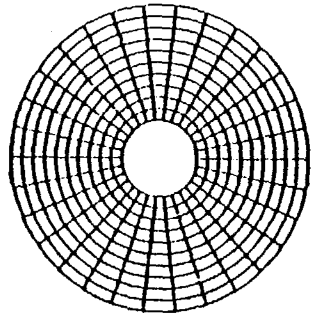


If the radius is 25, the prototype term is

```
0:3 & 25 * CS(0:12) & 25 * SN(0:12)

CELL = 31(11(0:3 & 25 * CS(<0:12>) & 25 * SN(<0:12>)))
```

A related problem is to distort the rectangular array to a pattern of concentric circles in a plane



The prototype term becomes

```
10:3 * CS(0:12) & 10:3 * SN(0:12)

CELL = 31(11(10:3 * CS(<0:12>) & 10:3 * SN(<0:12>)))
```

With these techniques mesh co-ordinates can be generated quite simply for really complicated structures

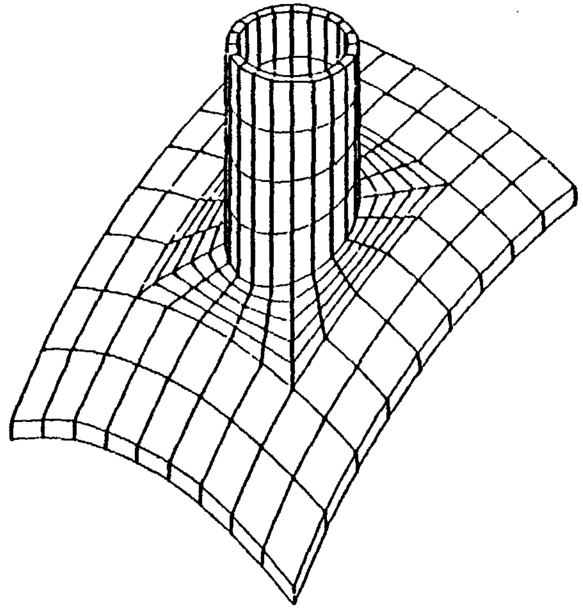


FIGURE 1E MORE COMPLICATED STRUCTURES

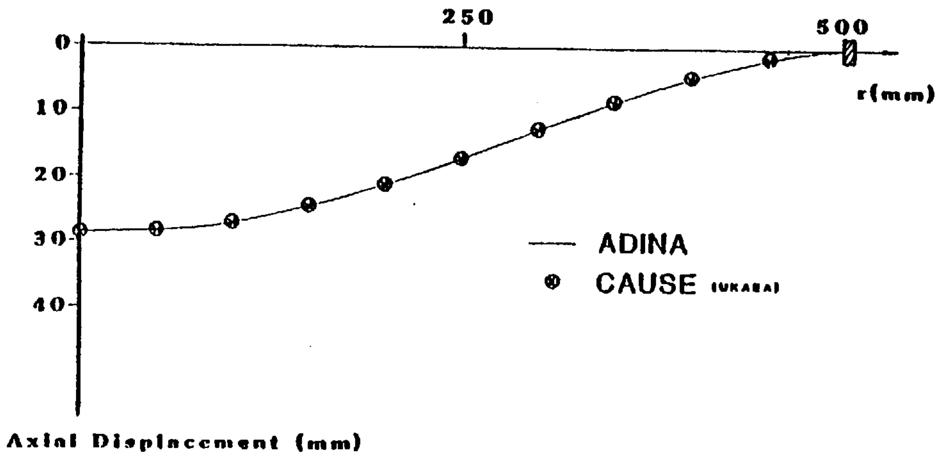
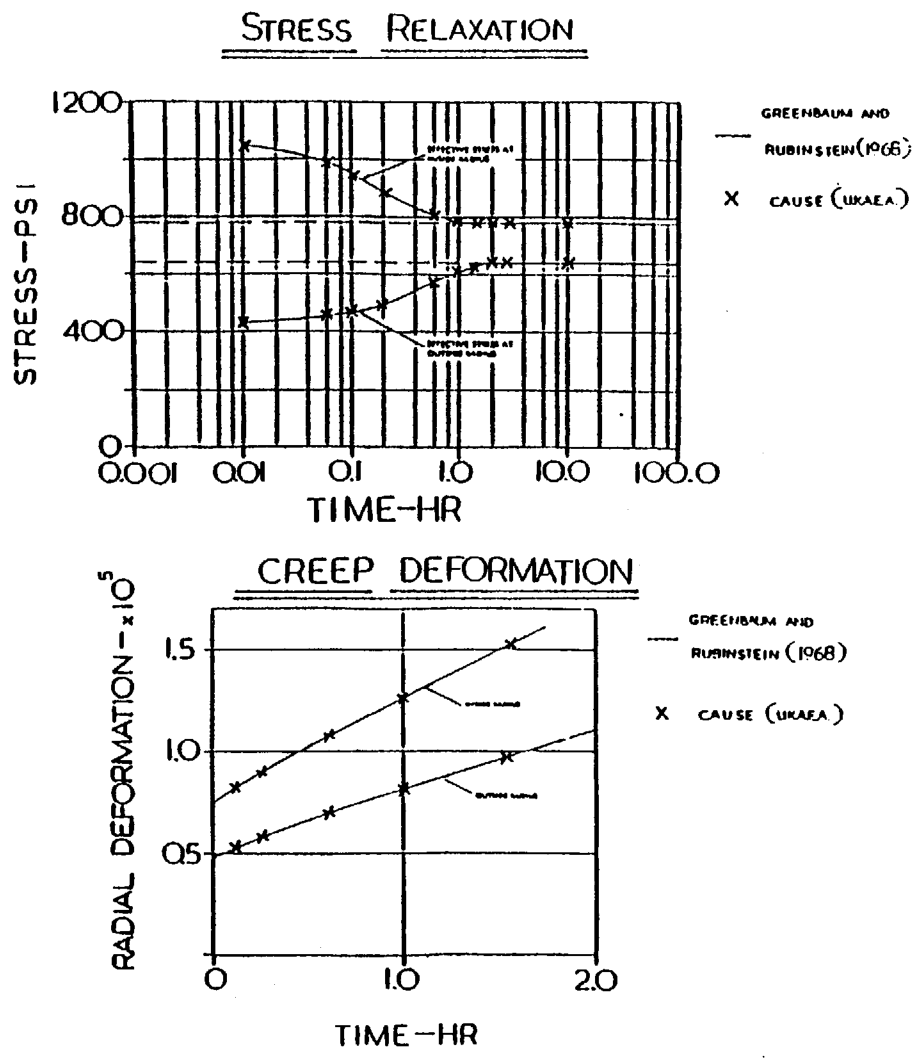


FIG. 2A: CREEP OF A CIRCULAR PLATE UNDER PRESSURE



THICK WALLED CYLINDER SUBJECT TO INTERNAL PRESSURE

FIG. 2B

Effective Stress Contours For A Pressure Vessel Subject To Internal Pressure

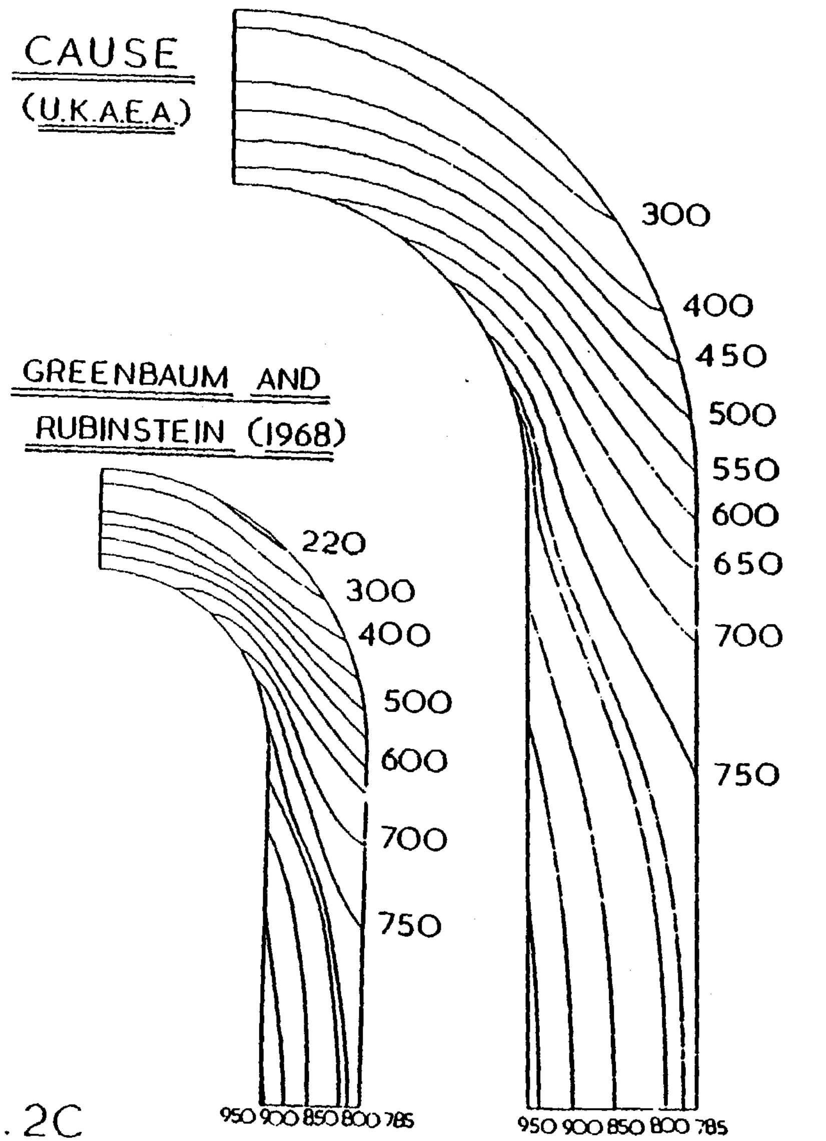


FIG. 2C

GENERAL ARRANGEMENT OF PFR INTERMEDIATE HEAT EXCHANGER

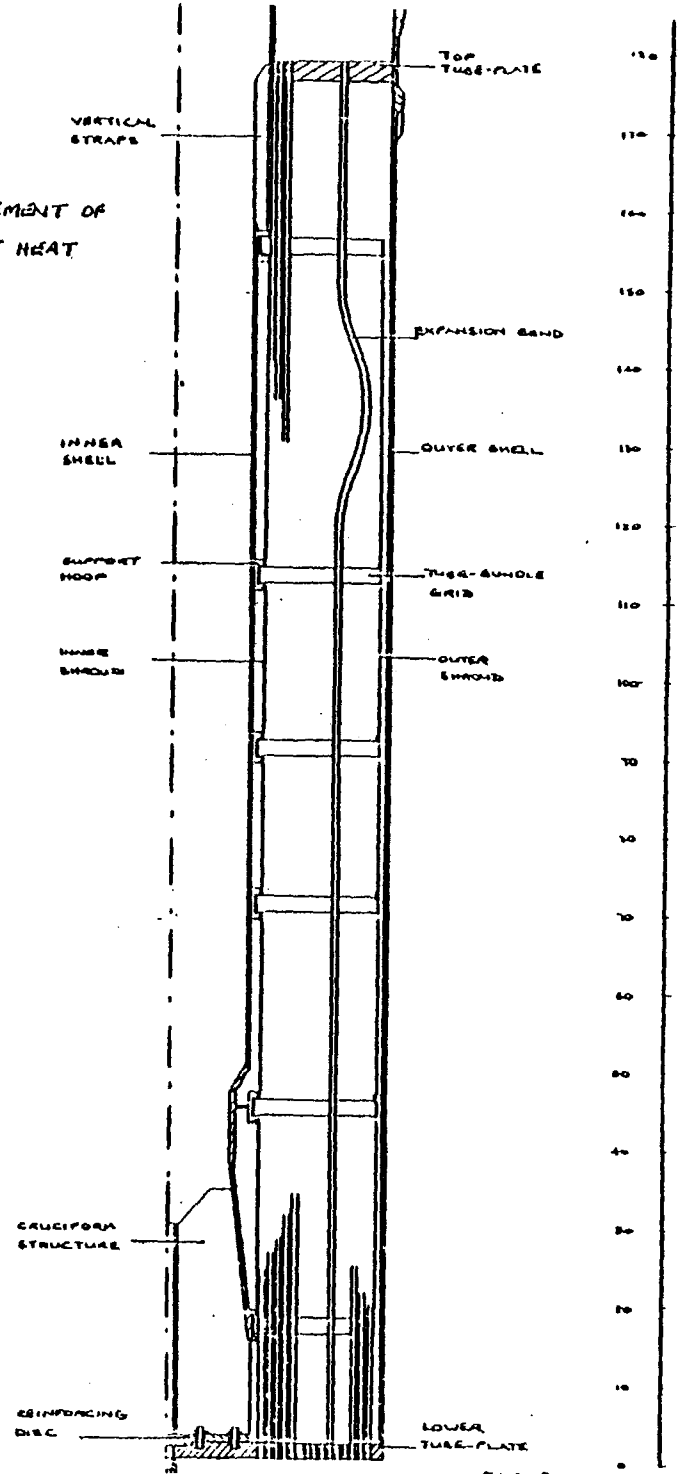
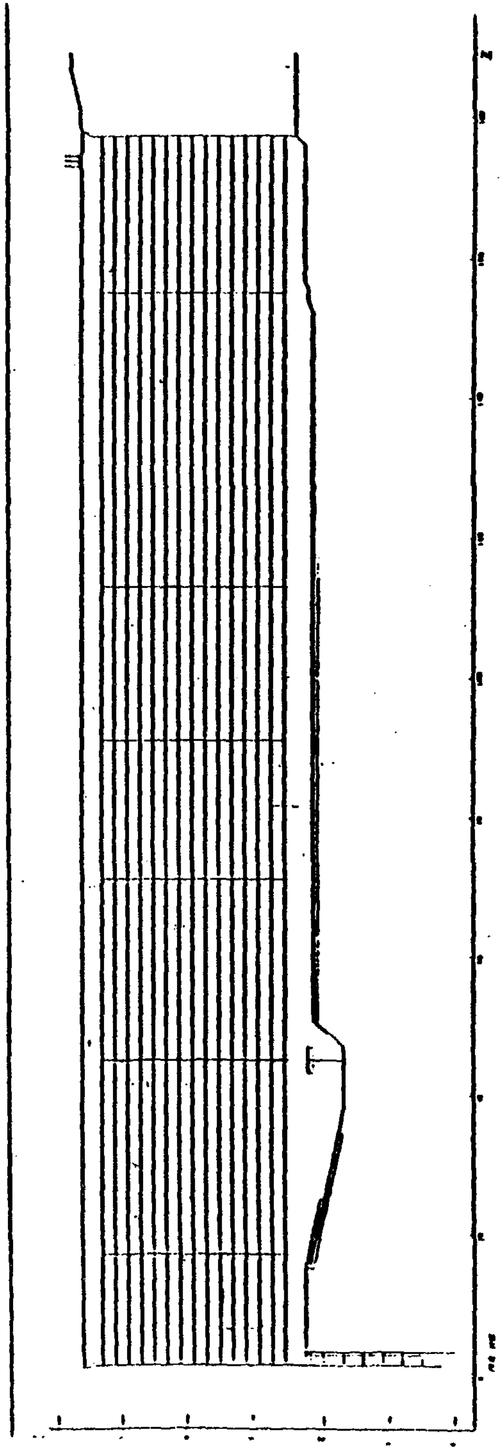


FIG. 3A



COMPUTER MODEL OF PFR INTERMEDIATE HEAT EXCHANGER

FIG 3B

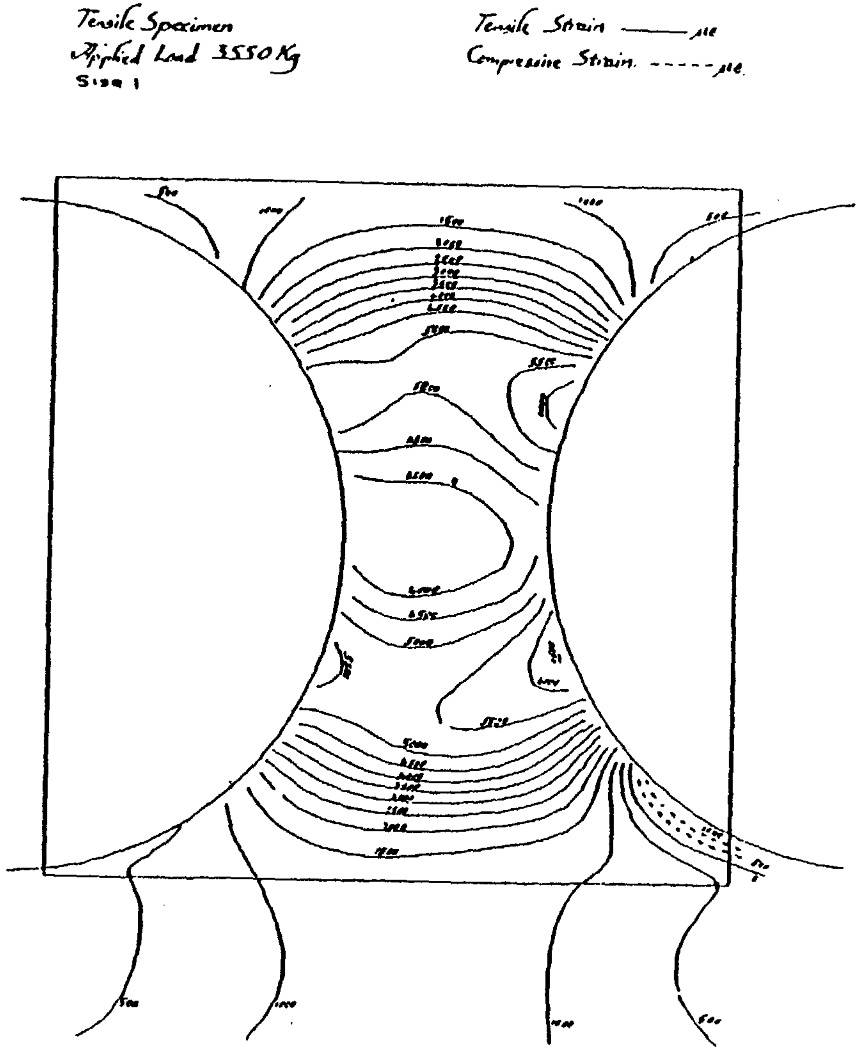


FIG 3C EXAMPLE OF STRAIN CONTOURS (TENSILE)

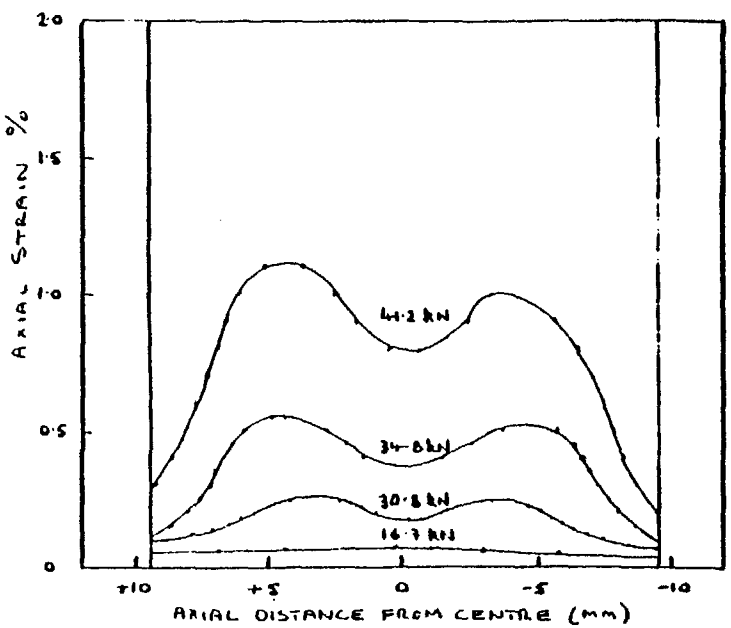


FIG 3D TENSILE TEST : CENTRE-LINE AXIAL STRAIN DISTRIBUTION

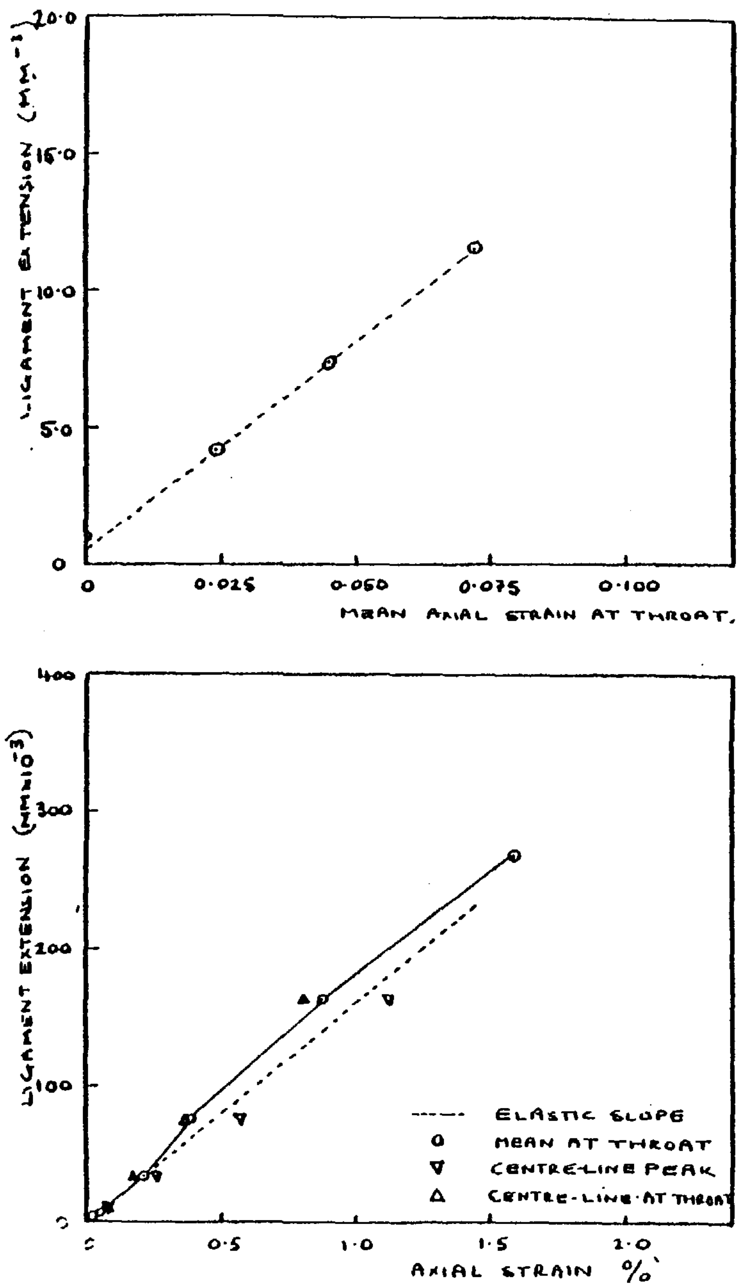


FIG 3E TENSILE TEST: AXIAL STRAIN WITH DISPLACEMENT