The SUPERMESH:

Run-Time Meshing from Coarse CAD Models Using MSGMESH

Charles Lund
Structural Methods Specialist
NWL Control Systems
Kalamazoo, MI

The generation of MSC/NASTRAN plate and solid element models is greatly speeded by combining any CAD mesh generator with MSGMESH (MSC's batch mode mesh generator). CAD models are developed only fine enough to sufficiently define the geometry; then MSGMESH is used to generate the elements and nodes at run time.

Introduction

One of the more time-consuming tasks in finite element analysis is the construction of the model. CAD-based automatic meshing systems can become bogged down by very large models, and when it becomes necessary to modify the geometry to optimize the shape of the part, or to decrease mesh spacing to verify the accuracy of the stresses, the incremental time involved can approach the original modeling time. The SUPERMESH system transfers most of the mesh generation effort to MSC/NASTRAN, leaving the user with a much smaller and more manageable original model.

Since the SUPERMESH method requires only enough nodes to define the geometry, moving them in the original model becomes relatively effortless. The number of elements (mesh fineness) is easily manipulated by variables in the data file, through the use of a text editor.

Discussion

MSGMESH is an automatic data generation option in MSC/NASTRAN which enables the analyst to describe a model using a relatively small number of special data records which describe one-, two- and/or three-dimensional regions. These special records are processed in the preface of MSC/NASTRAN, and the generated records are automatically passed on to MSC/NASTRAN. The regions, known as MSGMESH grid point "fields," may be bounded by curved lines and surfaces. In the SUPERMESH application, only two- and three-dimensional regions are used and only a single straight line or circular arc appears on each field edge.

Geometric grid points (defined on "EGRID" records) which define the corners of the grid point fields are called "grid field vertices." The number of elements on field edges are defined by "grid field dimensions" or "indices." Refer to the MSGMESH ANALYST'S GUIDE [1] for details on the operation of MSGMESH.

Models generated by CAD systems, by non-graphics interactive programs, or even by hand can be reformulated so that the individual elements become MSGMESH fields. The corner nodes of each element will ultimately become field vertices, and mid-edge nodes will become control grid points. In the SUPERMESH system, the mid-edge control grid point is only used when the particular edge consists of a circular arc, which passes through the control point, terminating at the two adjoining field vertices.

The first step in employing the SUPERMESH technique is to write a "Level 1" file representing the CAD model. This is done automatically by most CAD modelers which have MSC/NASTRAN output capability. It is also possible to construct the file by the use of a text editor. The file looks like ordinary MSC/NASTRAN bulk data, but it need only contain grid and element definitions, as well as pressure loads when necessary.

Next, the Level 1 file is processed by a special FORTRAN program, which converts it into MSGMESH data, the "Level 2" file. GRIDs will be renamed "EGRIDs" and CHEXA or CQUAD8 records will be converted into "GRIDG" and "CGEN" records. The GRIDG records cause MSGMESH to generate grids on each field and the CGENs cause the elements to be created. Whenever the position of a mid-edge node indicates an arc, an "EDGER" record is written to describe the arc to MSGMESH. If there are any PLOADs, they are rewritten as "PLOADG" data. The level 1 file may contain other bulk data and even executive and case control, since these lines are simply echoed in the Level 2 file.

Third, case control and other necessary bulk data are added by the user to create the "Level 3" file. This is the file which is used in the job submission (note that MSGMESH runs require the "MESH" keyword on the NASTRAN definition). The user defines the grid field dimensions on MDIM records, which have variables already in place that occur throughout the file on GRIDG records.

If desired, the generated mesh can be written to a "Level 4" file by using the "ECHO=PUNCH" case control statement. This file will have no MSGMESH records at all, and should be used without the "MESH" option on the NASTRAN definition.

Changes in mesh density are made in the Level 3 file, so resubmission is immediate. Simple geometry changes can also be made in the Level 3 file, but major changes may require rewriting the Level 1 file, converting it to Level 2, then merging it into the Level 3 file in place of the obsolete MSGMESH data.

The elements used to create the original model may be 20-node hexahedral or 8-node quadrilateral elements. Hybrid elements could also be used, if the CAD modeler supports them. Hexahedral elements can be automatically degenerated into 15-node pentahedral elements, and 8-node quadrilaterals can be degenerated into 6-node triangular elements by using the same grid number on two adjacent corners.

The output of the FORTRAN intermediate-processor program consists of the Level 2 file, plus a file containing tables to guide the analyst through the model, indicating which field indices are used where. By locating the nodes on a plot of the model, the particular edges employing a given field dimension can be determined. Then values for the field dimensions can be selected and entered on the MDIM record in the Level 2 file.

Field vertices are referred to as "L, M, and N" in MSGMESH. The L direction runs from the first corner of the field to the second. The M direction runs from the first corner to the fourth. In the case of solid fields, N runs normal to the surface containing L and M.

Field dimensions are forced to be consistent between adjoining field edges by the program. All fields are scanned to determine their connectivities, and a letter variable is assigned to each unique field dimension and included in the MDIM record.

All loads in the SUPERMESH method are assumed to be pressures. They can be specified in the Level 1 file, using PLOAD records. They are converted into PLOADG records in the Level 2 file. Additional loads of any type can be applied in the Level 3 file, if the user is knowledgeable about MSGMESH or has run the model previously to obtain the node and element numbering.

Since SPCs on MSGMESH fields appear much the same as pressures, they are defined with PLOAD records in the Level 1 file, with the "pressure" value actually being the degrees-of-freedom to be suppressed. Their set ID is given a value of 9000 or greater to signal the FORTRAN program to convert them into SPCG records. As with loads, additional SPCs can be applied in the Level 3 file, if the user is knowledgeable about MSGMESH or has run the model previously to obtain the node and element numbering.

When changing the mesh refinement, the user need only refer to the original model and the FORTRAN program's output table to determine which field indices apply to the edges of concern. The entries on the MDIM record of the Level 3 file can then be altered with a text editor.

Geometry alterations can take place in the original CAD database, or, if they are simple enough, in the Level 1 or Level 3 file. If the CAD database is updated, then a new Level 1 file can be written, converted to a Level 2 file, and merged into the Level 3 file, replacing the obsolete MSGMESH data.

MSGMESH will assign numbers to the generated nodes using six digits. The leftmost two identify the field number, and the rightmost four indicate the topological position of the node within the field. Thus, the smallest grid number will be 10000, while the largest can be 999999.
SUPERMESH elements are numbered with five digits, the first two being the field ID, and the last three being the sequential order in which the elements were created.

Each mesh field is limited in the number of elements it may contain by the MSGMESH rule that, for hexahedral fields, one edge may have up to 99 elements and the remaining edges must have no more than 9 elements, and, for quadrilateral fields, both edges are limited to 99 elements. In any case, the maximum number of nodes in a field is 1000. The number of mesh fields is limited to 99. However, level 4 files can be renumbered in such a fashion that will allow ten or more SUPERMESH models to be combined into a single model. A FORTRAN program can accomplish this, but node numbers will loose their strict MSGMESH-style numbering. They can retain their field ID, and can include a model ID.

Other considerations need to be kept in mind when drawing up the original model fields and selecting the mesh sizes. Aspect ratios can vary widely with different field dimensions. Distortions tend to be less than the field element distortions, but can be worse if the field dimensions are too low. MSGMESH can recognize only 26 variable names (the upper case letters, A through Z) on the MDIM record. The FORTRAN program will continue to use subsequent ASCII characters ([,\,],^,a-z, etc.), but the user must replace them with the upper case letters. Fields with faces on cylindrical surfaces may have grids which do not lie exactly on those surfaces if the field edges do not fall on generator lines, that is, lines parallel to the axis of the cylinder or in a plane perpendicular to the axis.

Example 1

In lieu of the usual "frabbis" example, the first structure modeled here is the MSC logo (Figure 1). The letters S and C are connected to one another at one location, but the M is separate. The letters are modeled with only eleven hexahedral elements (Figures 2 and 3). Node replication features of the CAD modeling system are used to create 3 layers of nodes. Note that some mid-face nodes are created (Figure 4), but they are not used in the element definitions.

Pressures are added inside the C (Figure 5), and SPCs are applied to the lowest horizontal surface of each letter (Figure 6). The SPC "pressures" are given an ID of 9000, and a value of 123; hence, all translations will be SPCed.

The Level 1 file written by the CAD modeler (GRAFEM, in this case) contains only GRID, CHEXA, and PLOAD4 records (Figure 7), in addition to PSOLID and MAT1 records, which will be rewritten later.

The FORTRAN program execution (Figure 8) requires only the naming of the input and output files, along with coordinate system IDs to be used for the nodal displacement system and for MSGMESH to use when interpolating the positions of grids internal to the fields.

Output from the program consists of the Level 2 file (Figure 9) and tables describing the field dimensions (Figure 10).

After adding the necessary executive control, case control, and remaining bulk data, as well as entering the field dimensions on the MDIM records, the Level 3 file (Figure 11) is ready for submission to MSC/NASTRAN. Note that some field dimension variables have been combined with each other to force extra consistency throughout the model. The EQUIV record is added to tell MSGMESH to merge nodes on the faces or edges of fields that share the same EGRIDS.

The resulting model has 58 elements (Figure 12), and their displacements (Figure 13) demonstrate that the connection between the S and C was made, and that the SPCs were enforced. The stress contours (Figure 14) show irregularities resulting from too few elements, so the entries on the MDIM record are simply increased and the model regenerated by MSGMESH with 400 elements (Figure 15).

Example 2

A simple 2-D half-model of a spring clip (Figure 16) was easily made much finer (Figure 17), with quick alterations to the MDIM record.

Example 3

A quite detailed half model of a spherical bearing race was generated from 92 fields (Figure 18), resulting in about 2000 elements (Figure 19).

Example 4

99 fields were required to model a symmetric half of a complex hydraulic actuator end housing with 3300 elements (Figure 20). Mesh refinements in high stress areas were accomplished quickly, and altering the diameter of one of the bores required much less time and effort than if the fine model had been altered directly.

Conclusions

With the aid of MSGMESH and an intermediate program, very coarse models can be refined at run time to provide accurate analysis without correspondingly increasing the time and effort of building the original model. Likewise, changes can be introduced to the model geometry and mesh fineness with much less exertion.

References

[1] "MSC/NASTRAN MSGMESH Analyst's Guide," Version 63, The MacNeal-Schwendler Corporation, Los Angeles, CA, December, 1983.

Figure 1

Figure 2

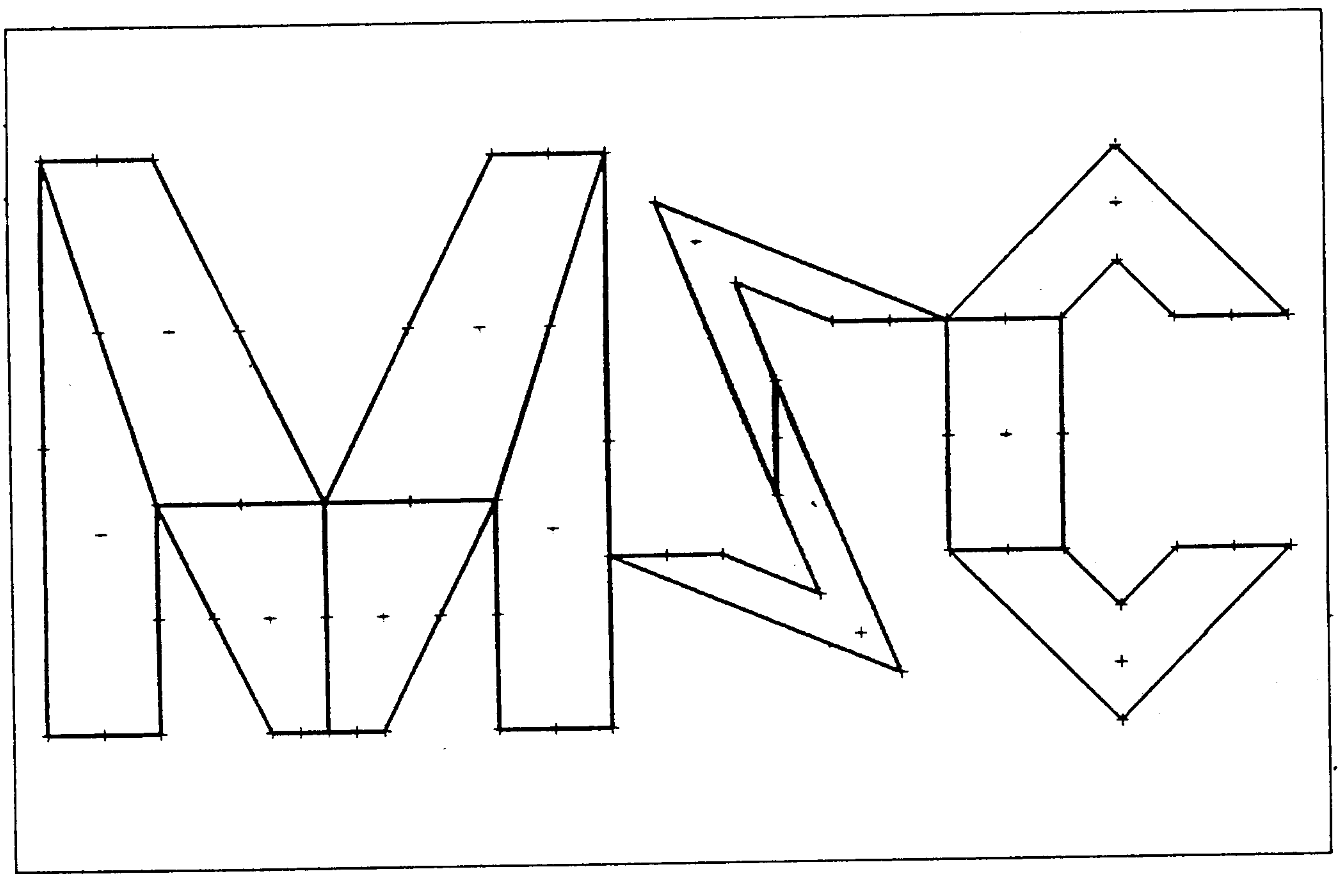


Figure 3

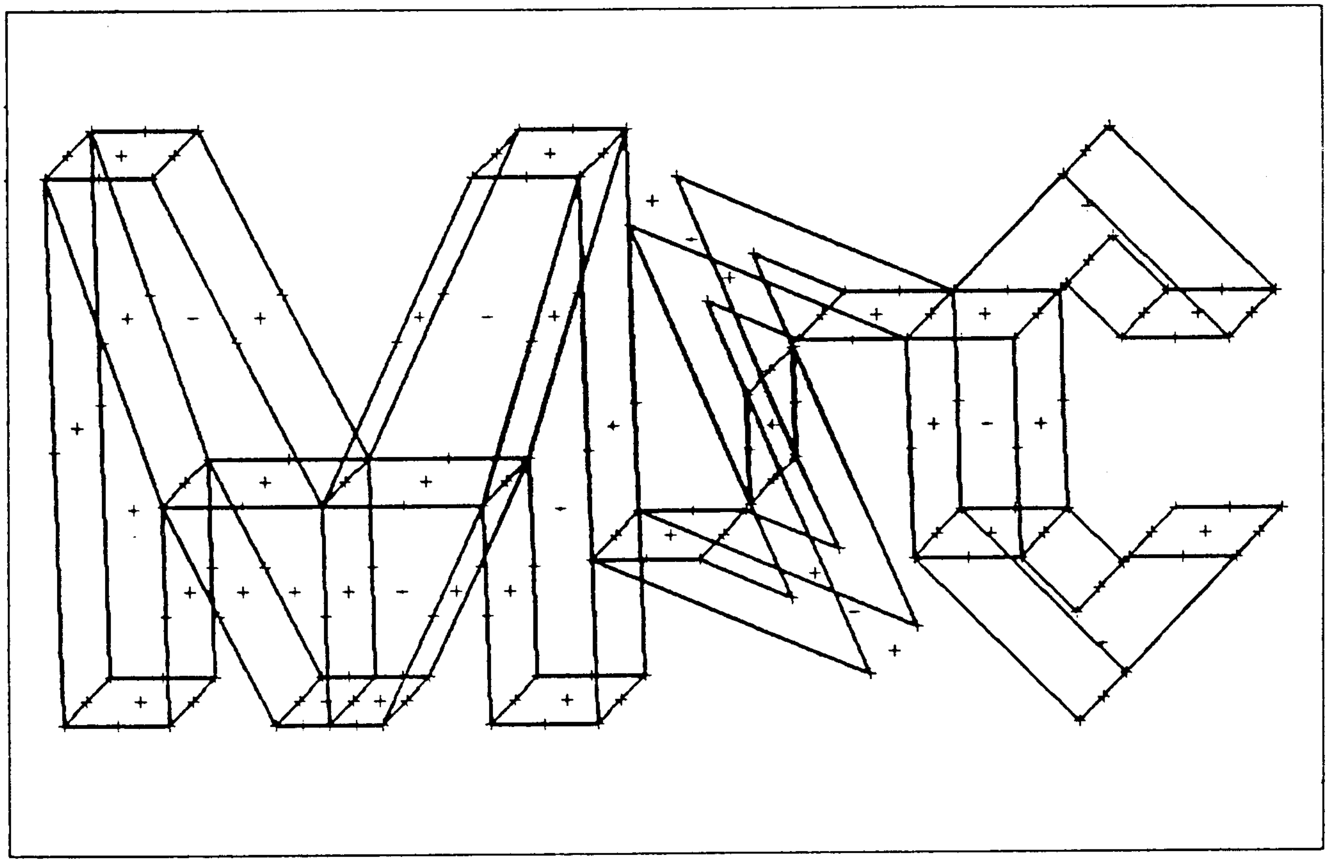


Figure 4

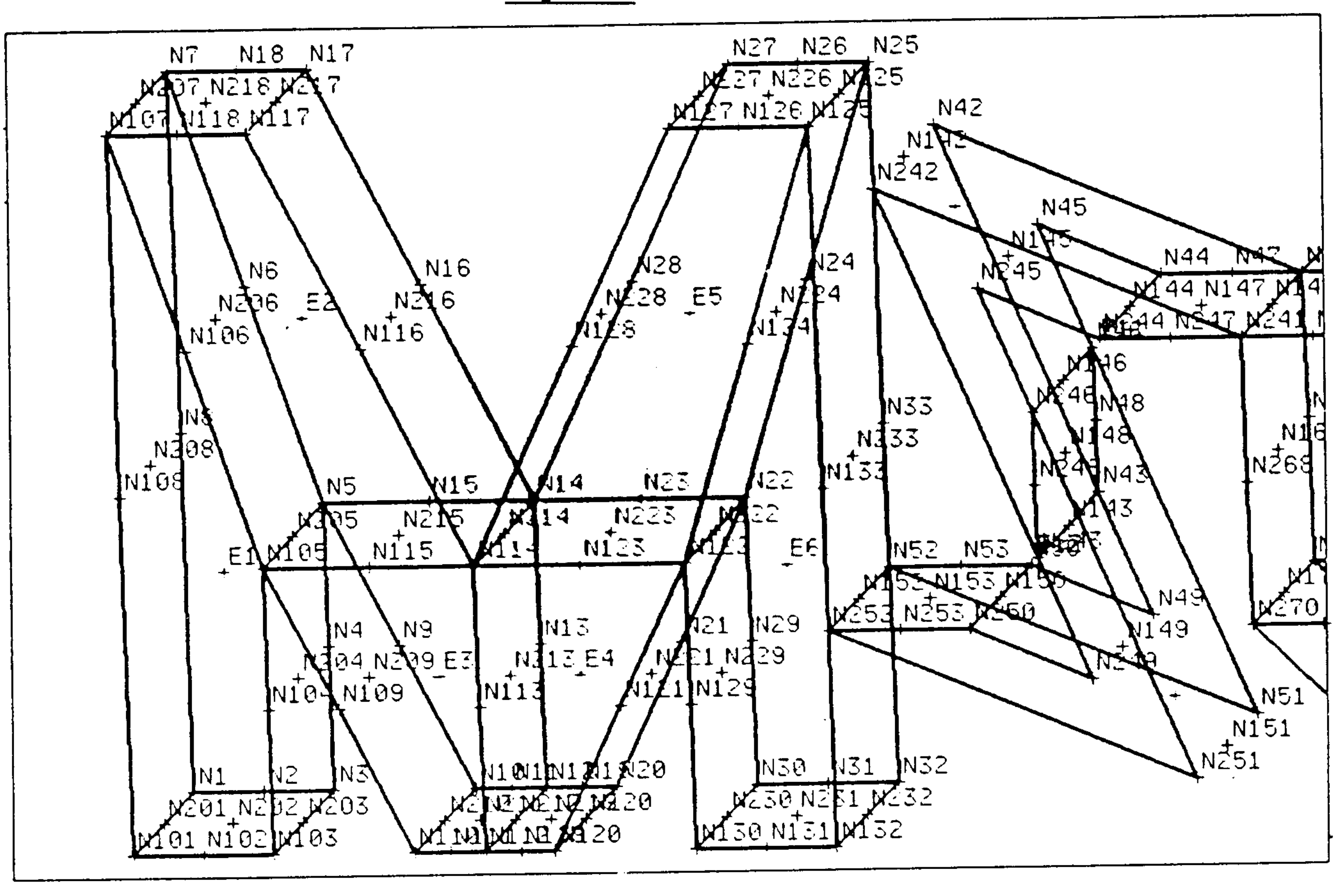


Figure 5

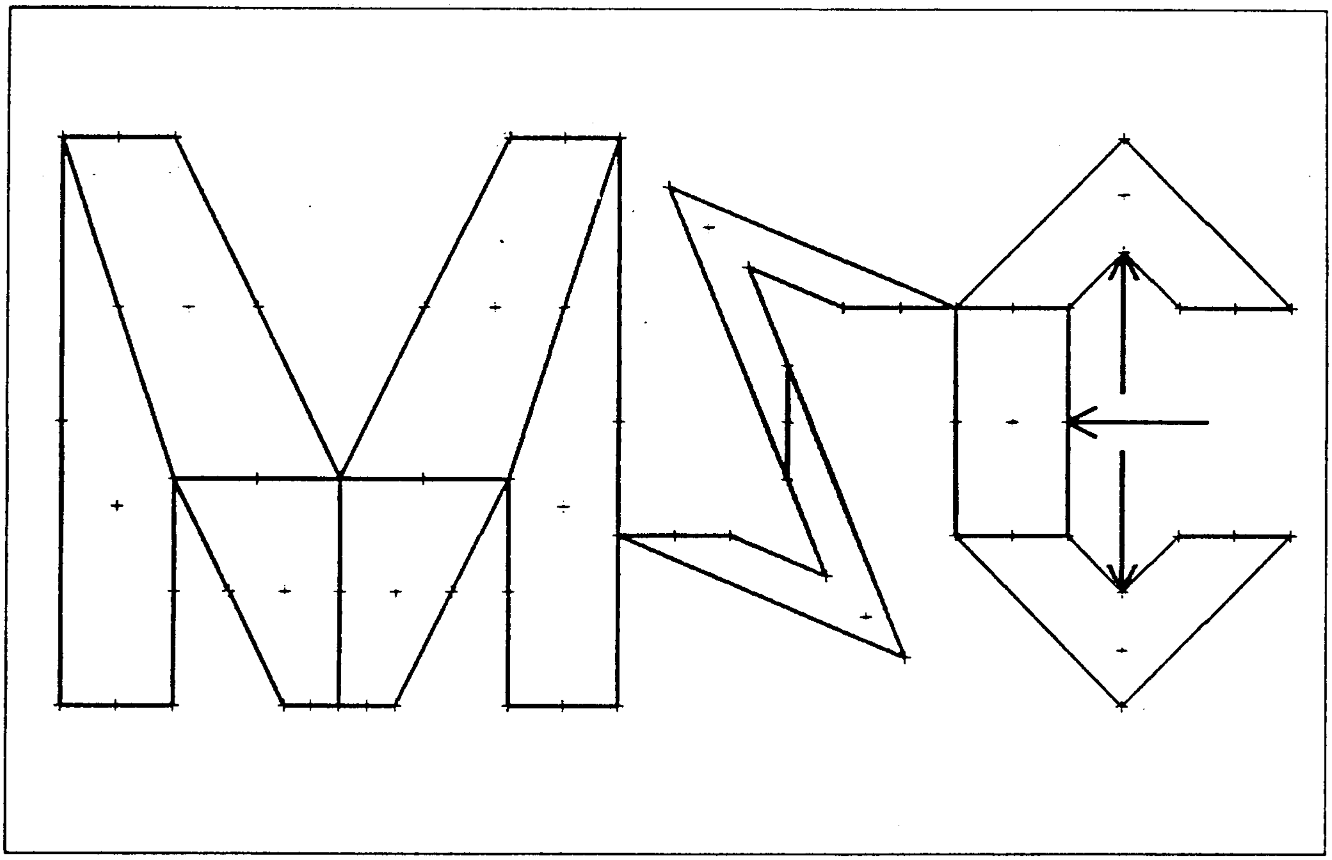
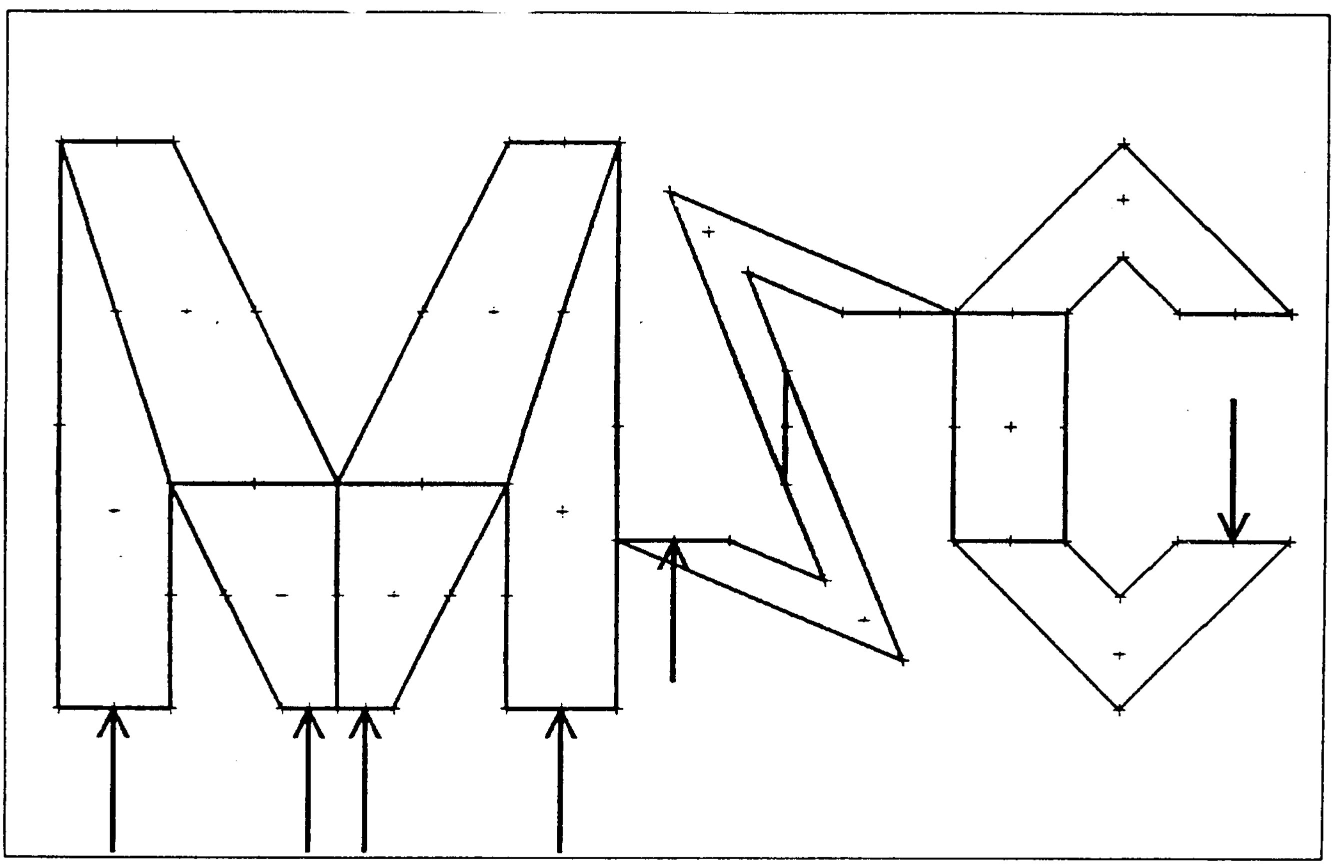


Figure 6



<u>Figure 7</u>

Level 1 file:

```
PSOLID,1,1
MAT1,1,1
GRID          1       0 0.00000 0.00000 0.00000       0
GRID          2       0 0.20000 0.00000 0.00000       0
GRID          3       0 0.40000 0.00000 0.00000       0
...
CHEXA         1       1       1       7       5       3     101     107+E100001
+E100001    105     103       8       6       4       2     201     207+E200001
+E200001    205     203     108     106     104     102
CHEXA         2       1       5       7      17      14     105     107+E100002
+E100002    117     114       6      18      16      15     205     207+E200002
+E200002    217     214     106     118     116     115
...
PLOAD4        1      21 -1000.0                                63     265
PLOAD4        1      22 -1000.0                                65     271
PLOAD4        1      23 -1000.0                                71     276
PLOAD4     9000       1  123.00                                 3     101
PLOAD4     9000       3  123.00                                12     110
PLOAD4     9000       4  123.00                                20     112
PLOAD4     9000       6  123.00                                32     130
```

<u>Figure 8</u>

Execution of FORTRAN program:

```
$ RUN SUPRMESH

               ***** SUPRMESH *****

FILE NAME FOR INPUT: MSC.GRAFEM OUTPUT
FILE NAME FOR OUTPUT (MSGMESH DATA), [DEFAULT= NAMSC.MSG]: MSC.SUPRMESH OUTPUT
ENTER COORD. SYS IDs FOR DISPLACEMENTS
AND INTERPOLATION [DEFAULT=SYS. ON GRID CARD]: 0,0

FILE FOR006.DAT WILL CONTAIN TABLES THAT SUMMARIZE
THE L, M, AND N DIMENSIONS OF EACH FIELD, ALONG WITH THE
NODES THAT DEFINE THE L, M, AND N DIRECTIONS.

FORTRAN STOP
$
```

Figure 9

Level 2 file:

```
BEGIN BULK
PSOLID,1,1
MAT1,1,1
MDIM           A       0       B       0       C       0       D       0
MDIM           E       0       F       0       G       0       H       0
MDIM           I       0       J       0       K       0       L       0
MDIM           M       0       N       0       0       0       P       0
EGRID          1       0 0.00000 0.00000 0.00000       0
EGRID          2       0 0.20000 0.00000 0.00000       0
EGRID          3       0 0.40000 0.00000 0.00000       0
...
GRIDG          1       0       0       A     ALL                        +GG 1A
+GG 1A         B               C                                       0+GG 1B
+GG 1B         1       7       5       3     101     107     105     103
CGEN    HEXA8       1001       1       1
...
$
EDGER         11       B      46       C      44      45
EDGER         11       D      41       A      43      42
EDGER         11       F     246       G     244     245
EDGER         11       H     241       E     243     242
GRIDG         11       0       0       I     ALL                        +GG11A
+GG11A         J               K                                       0+GG11B
+GG11B        43      46      44      41     243     246     244     241
CGEN    HEXA8      11001       1      11
...
$
PLOADG         1      21 -1000.0 -1000.0 -1000.0 -1000.0       C       H
PLOADG         1      22 -1000.0 -1000.0 -1000.0 -1000.0       C       H
PLOADG         1      23 -1000.0 -1000.0 -1000.0 -1000.0       C       H
SPCG        9000       1     123       D       E
SPCG        9000       3     123       D       E
SPCG        9000       4     123       D       E
SPCG        9000       6     123       D       E
ENDDATA
```

Figure 10

FORTRAN program output:

```
                    ***** SUPRMESH *****

   SUMMARY OF FIELD DIMENSIONS -

   FIELD  L    L-NODES     M    M-NODES      N    N-NODES

     1    A     1-   7     B     1-   3      C     1- 101
     2    B     5-   7     D     5-  14      C     5- 105
     3    E    10-   5     D    10-  12      C    10- 110
     4    E    12-  14     F    12-  20      C    12- 112
     5    G    14-  27     F    14-  22      C    14- 114
     6    H    30-  22     G    30-  32      C    30- 130
    11    I    43-  46     J    43-  41      K    43- 243
    12    I    46-  43     L    46-  52      K    46- 246
    21    M    41-  61     N    41-  65      K    41- 241
    22    O    70-  41     N    70-  71      K    70- 270
    23    P    74-  70     N    74-  76      K    74- 274


   FIELDS ASSOCIATED WITH EACH FIELD DIMENSION -

   FIELD DIMENSION A USED IN FIELD(S):   1
   FIELD DIMENSION B USED IN FIELD(S):   1  2
   FIELD DIMENSION C USED IN FIELD(S):   1  2  3  4  5  6
   FIELD DIMENSION D USED IN FIELD(S):   2  3
   FIELD DIMENSION E USED IN FIELD(S):   3  4
   FIELD DIMENSION F USED IN FIELD(S):   4  5
   FIELD DIMENSION G USED IN FIELD(S):   5  6
   FIELD DIMENSION H USED IN FIELD(S):   6
   FIELD DIMENSION I USED IN FIELD(S):  11 12
   FIELD DIMENSION J USED IN FIELD(S):  11
   FIELD DIMENSION K USED IN FIELD(S):  11 12 21 22 23
   FIELD DIMENSION L USED IN FIELD(S):  12
   FIELD DIMENSION M USED IN FIELD(S):  21
   FIELD DIMENSION N USED IN FIELD(S):  21 22 23
   FIELD DIMENSION O USED IN FIELD(S):  22
   FIELD DIMENSION P USED IN FIELD(S):  23
```

Figure 11

Level 3 file:

```
NASTRAN MESH
TIME 3
SOL 24
CEND
TITLE=                          SUPERMESH EXAMPLE
SUBTITLE=C. LUND
ECHO=UNSORT
SPC=9000
SUBCASE 1
 LABEL= PRESSURE LOAD IN LETTER C
 LOAD=1
 DISP=ALL
 OLOAD=ALL
STRESS(PLOT)=ALL
SPCFORCES=ALL
BEGIN BULK
MDIM           A       8       B       4       C       1       D       3
MDIM           E       6       J      16
MDIM           M      12       0       8
EGRID          1       0 0.00000 0.00000 0.00000       0
EGRID          2       0 0.20000 0.00000 0.00000       0
...
$
EQUIV   AUTO    YES     YES     YES
$
GRIDG          1       0       0       A     ALL                        +GG 1A
+GG 1A         B               C                                        +GG 1B
+GG 1B         1       7       5       3     101     107     105     103
CGEN    HEXA8       1001       1       1
...
$
EDGER         11       B      46       C      44      45
EDGER         11       D      41       A      43      42
EDGER         11       F     246       G     244     245
EDGER         11       H     241       E     243     242
GRIDG         11       0       0       B     ALL                        +GG11A
+GG11A         J               C                                        +GG11B
+GG11B        43      46      44      41     243     246     244     241
CGEN    HEXA8      11001       1      11
$
PLOADG         1      21 -1000.0 -1000.0 -1000.0 -1000.0       C       H
...
SPCG        9000       1     123       D       E
...
$ END OF MSGMESH DATA $$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
SPC1    9000    456     1       THRU    240000
MAT1    1       10.+6           .33     .10     13.-6   80.     .0001
PARAM   AUTOSPC NO
PSOLID  1       1
ENDDATA
```

Figure 12

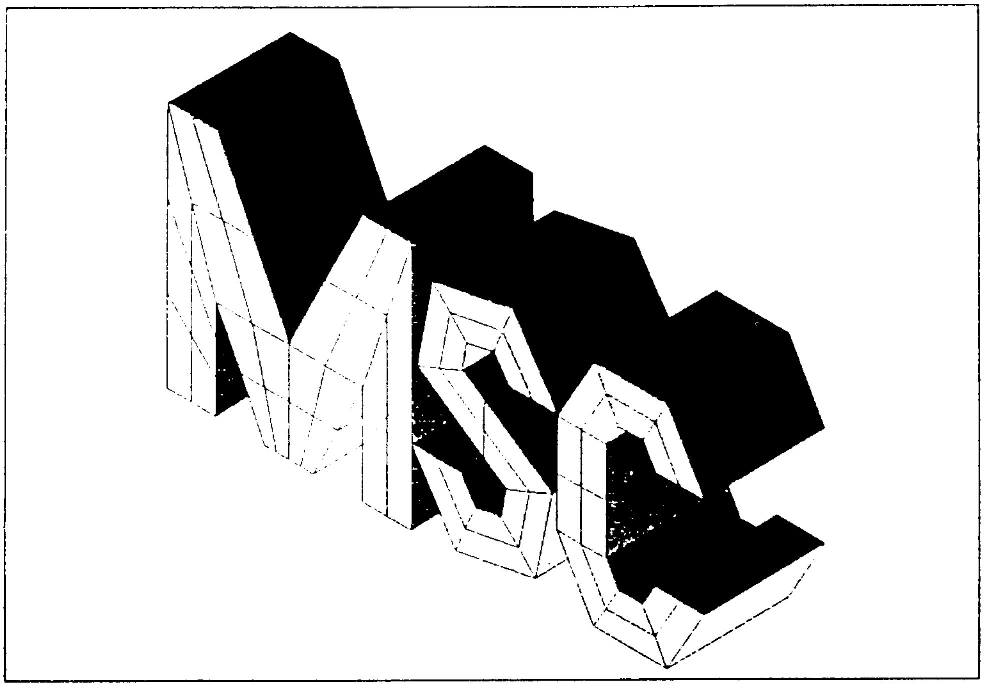


Figure 13

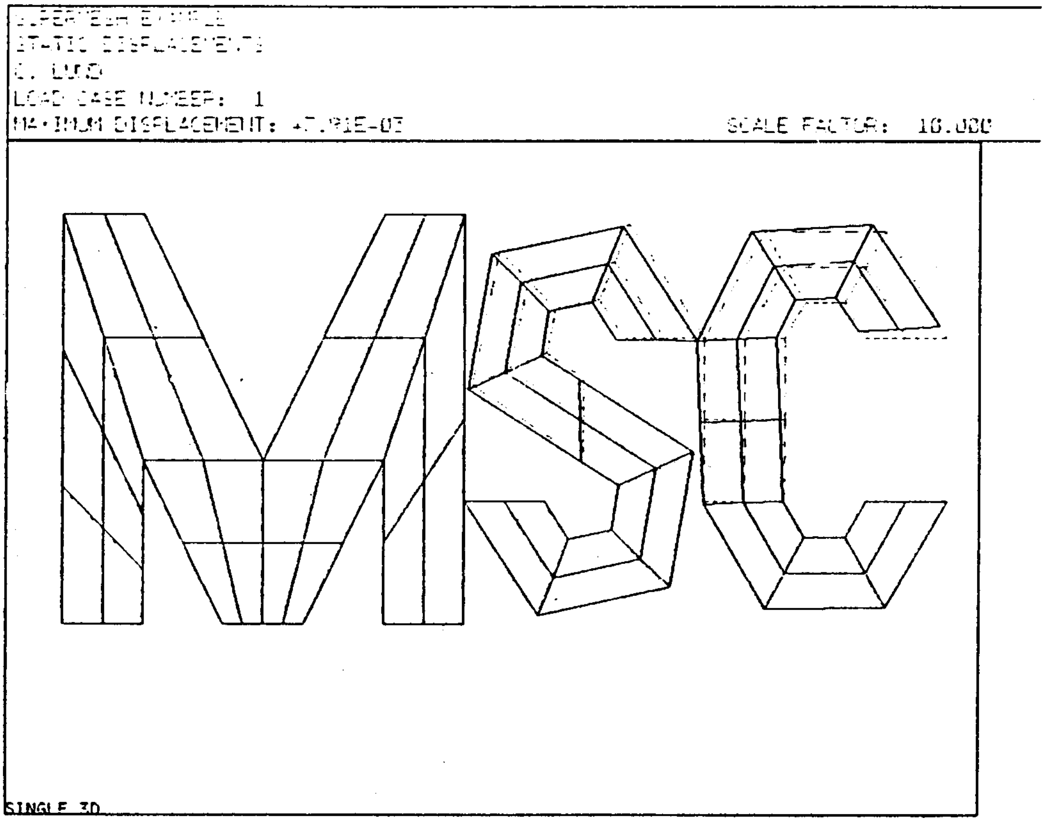


Figure 14

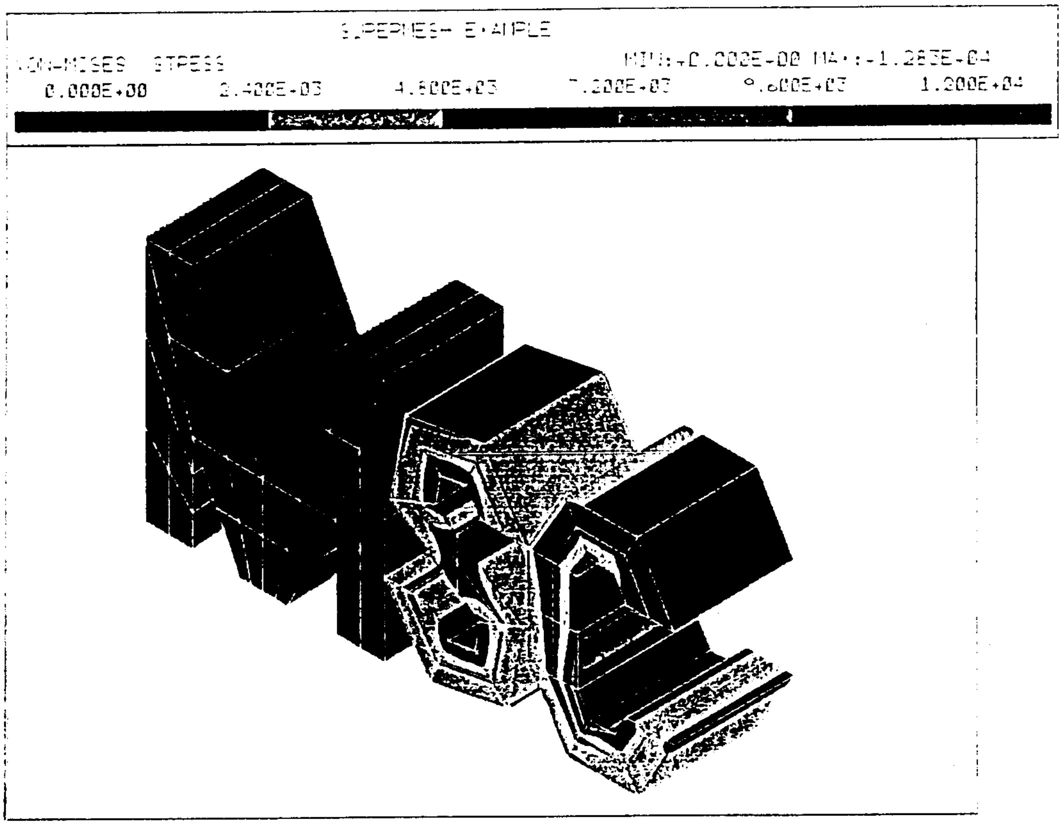


Figure 15

Figure 16

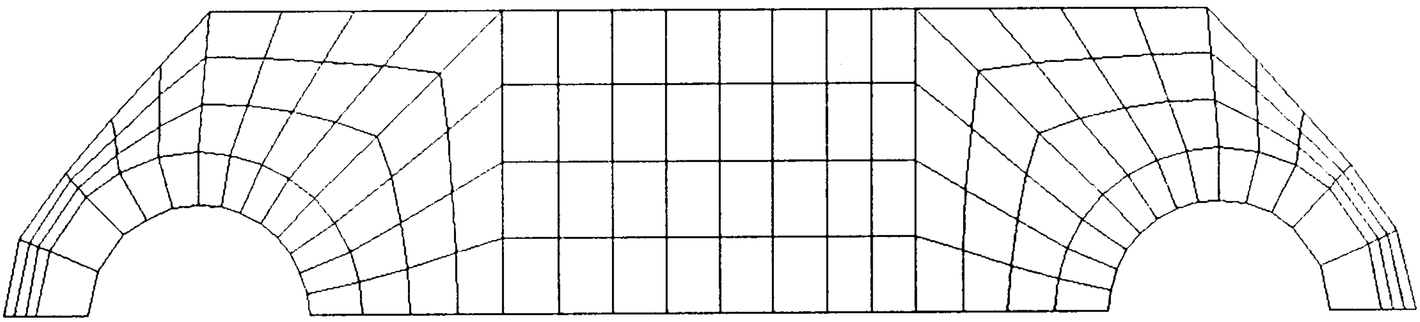


Figure 17

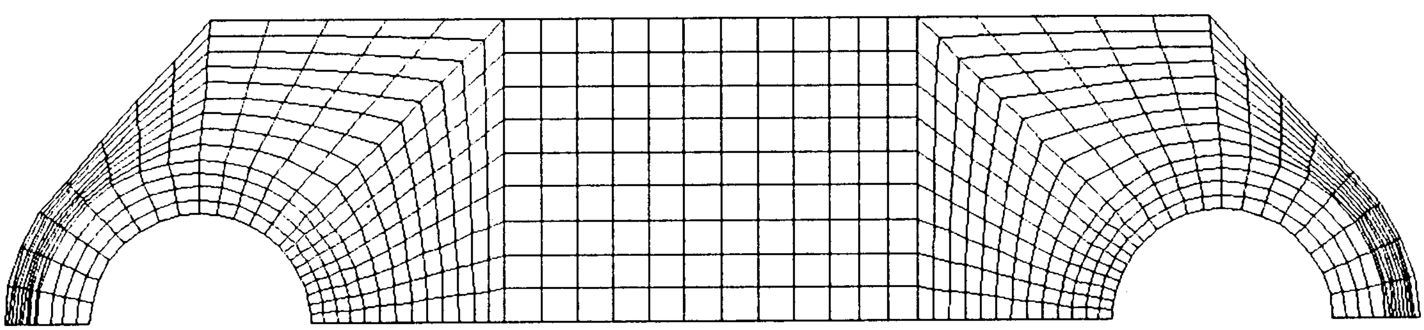


Figure 18

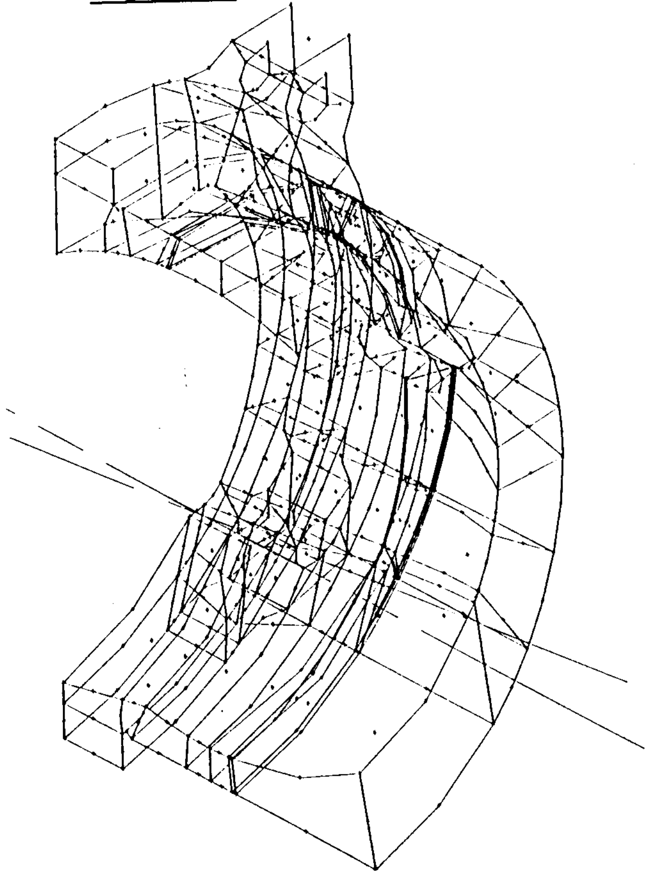


Figure 19

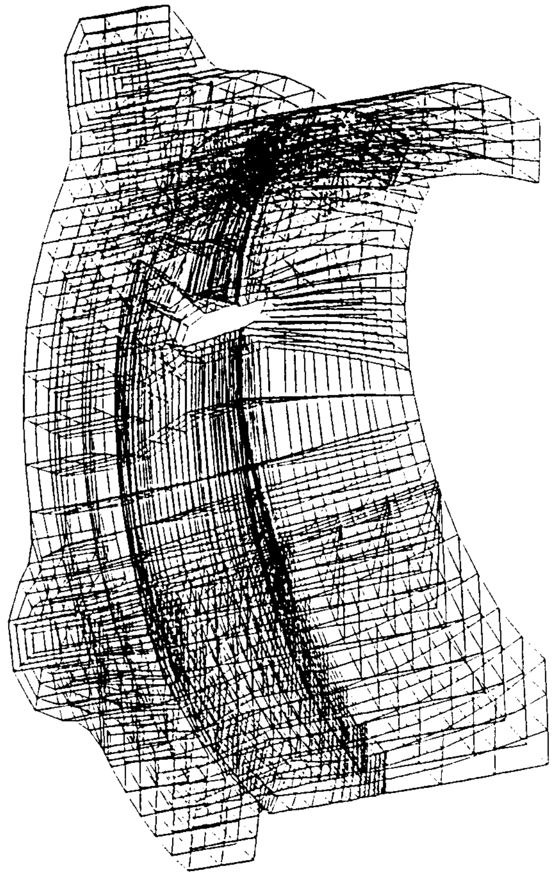


Figure 20