

**NOTICE WARNING CONCERNING COPYRIGHT RESTRICTIONS:**  
The copyright law of the United States (title 17, U.S. Code) governs the making of photocopies or other reproductions of copyrighted material. Any copying of this document without permission of its author may be prohibited by law.

**The Mechanical Strength Critic in CASE**

by

**Moshe Eisenberger**

**EDRC 12-36-90\***

# The Mechanical Strength Critic in CASE

Moshe Eisenberger<sup>1</sup>

Department of Civil Engineering  
Carnegie Mellon University  
Pittsburgh, PA

**Abstract:** This paper describes the mechanical strength critic in CASE (Computer Aided Simultaneous Engineering) (Rehg *et al.*, 1988a; Rehg *et al.*, 1988b; Sapossnek *et al.*, 1989). The mechanical strength critic is composed of two expert systems and several other supporting programs. The first expert system, the Model Generation (MG), generates a complete model for finite element analysis based on geometry, loading, boundary conditions, and performance criteria that are input by the designer. The second expert system draws the relevant results from the finite element analysis and checks for compliance with the performance specifications. If there are violations of the criteria, a set of recommendations for changes in the design are given by the Strength Critic expert system (SC) based on the assumptions of the behavior of the model made in MG and the results calculated in the analysis. The implementation of these changes are left to the designer, as he should also consider simultaneously other recommendations from other critics in the CASE project.

This work was supported in part by the Engineering Design Research Center, a National Science Foundation Engineering Research Center.

---

<sup>1</sup>On leave from Technion - Israel Institute of Technology, Technion City 32000, Israel

# 1 Introduction

For design purposes it is desirable that the analysis tools, in our case the commercial finite element analysis packages, will be incorporated into a more general framework that will allow automatic data transfer and result interpretation. This was attempted as part of the CASE project (Rehg *et al.*, 1988a; Sapossnek *et al.*, 1989). This work is the first step toward the creation of an intelligent CAD system, that will be able to automatically address several concerns in the design process, one of which is the mechanical strength aspect of the design.

Finite element analysis is carried out in three phases: preparation of the model for solution, solution, and interpretation of the results. Of these phases the solution phase is currently well established: many commercial finite element systems are available for the designer.

The first stage, model building, is the focus of many researchers and commercial products, that try to derive procedures that will assist in this task. Most of these algorithms and systems address only one aspect of the task, the mesh generation. This is a geometric problem which is constrained by several limitations of the analysis tools that are used for the solution. Current mesh generation techniques can be classified in several categories (following (Cheng *et al.*, 1989)):

1. Interpolation mesh generation (Cook, 1974; Gordon and Hall, 1973; Haber *et al.*, 1981; Zienkiewicz and Phillips, 1971) - partitioning of the structure into simpler subregions and meshing each individually. The resulting meshes are well-conditioned, but local refinements are harder to implement.
2. Automatic Triangulation (Bykat, 1976; Cavendish, 1974; Fredrick *et al.*, 1970; Rivara, 1987; Sadek, 1980) - mostly used for 2D mesh generation. Some recent developments (Cavendish *et al.*, 1988; Rivara, 1987) help overcome the limitations of ill-conditioned meshes that resulted occasionally using this method.
3. Quadtree-Octree approach (Baehmann *et al.*, 1987) - based on quadtree/octree encoding of the objects.
4. Constructive solid geometry (Lee *et al.*, 1984) - will yield good meshes for cases with constant mesh density, and less efficient for variable refinement meshing.

The mesh generation techniques that were reviewed, automate the process of mesh generation only partially: they still require the analyst to exercise his judgment with regards to the regions in the model where finer mesh is needed, and input this to the system. The analyst judgment, or in other words, experience and guess of the expected results of the analysis, is based mainly on a few rules of thumb that have been learned and compiled over the years, in several sources, such as finite element books and

state of the art reports (Meyer, 1987) The current mesh generation procedures fail to incorporate this common knowledge as part of them. Moreover, any attempt to produce a finite element model which does not consider the boundary conditions and loading on the artifact is bound to be incomplete. In practice, these should be input as part of the analyst judgment of the problem. Another aspect which is often overlooked, is the relation between the analysis model and the performance criteria for the artifact. If the main concern of the designer are the vibration frequencies or the buckling load of the artifact, a given mesh will be "good enough" for finite element analysis. This mesh will be very different from the one that we would like to use for stress analysis. This issue is not addressed at all in the available tools.

The third phase in finite element analysis, following the validation of the results (which is a "must" for all FE analysis) is the interpretation of the results. To this end, most commercial packages contain very good graphical tools for viewing the results. No tool, however, is available for interpretation of the results, and for making specific recommendations for changes in the design, so that the artifact will comply with performance requirements and material strength limitations.

In what follows, a knowledge-based approach to the complete task of finite element mechanical strength analysis is described. The Model Generation Expert System is presented in detail, and it's output is then used for the analysis and interpretation. Then, the Strength Critic Expert System is presented followed by several examples of the system capabilities.

## 2 The Model Generation and Strength Critic Architecture

The system is composed of several programs written in three computer languages (C, OPS83, and AWK). This was done in order to try to simplify the task of composing such a complex system that should also interact with commercial packages not available in source code, and mostly written in FORTRAN. A general flowchart of the system is given in Figure 1.

The modules of the system have the following functions:

- Feature Pre-Processor - Accepts the input to the system as presented in the next section and transforms several features into a simpler set of basic features (Written in Q).
- Model Generation Expert System - This module accepts the input and generates the entire model (written in OPS83). It's output contains all the required commands for the modeling of the problem, in the language of the analysis package (in the current version the data is for the ANSYS program). A second file is generated containing post processing commands for the SC expert system.
- FE Input, FE Analysis, FE Output - Input and output from the analysis stage (in ANSYS

command syntax). This stage can be modified for any analysis package.

- **Fe Post-Processing Input** - The file containing the post processing commands (in ANSYS command syntax).
- **Strength Critic Pre-Processor** - Modifies the FE output to a form which is sorted and organized so that the SC can process (includes programs in C and AWK to sort, remove system messages, and analysis package headings).
- **Strength Critic Expert System** - This module compiles the results of the analysis with the assumptions about the behavior and performance of the artifact and produces a summary file and recommendations for changes in the design (written in OPS83).
- **Report and Recommendations** - The report and recommendations of the SC are presented for the designer's action. the implementation of these changes is left to the designer in the current system. Future version will include automatic changes, which will be equivalent to shape optimization procedure.

### 3 Model generation

The Model Generation expert system builds an entire finite element analysis model, rather than just the mesh, as discussed previously. Thus, the input for the system is composed of the geometry, loading, boundary conditions, and performance specifications for the artifact. Each of these is input in terms of features that are known to the system. These features are the basic building blocks that one needs to start analysis of any design. Using them, one can describe to the system a wide range of parts for analysis and critic. The designs that the system can handle are those composed of plate like segments, planar or curved (i.e. shells), that are oriented in any direction in space. The plate characteristics are that its thickness is very small compared to its other two dimensions, and the assumptions of thin plate and shell theory hold.

The model generation program determines the type of analysis that is required, from the set of specifications of the desirable performance of the artifact. Therefore, the model that is generated using the system, is dependent on the specifications, as well as on the geometry, loads, and boundary conditions. As an example, the resulting model for stress analysis is very different from the stability analysis model that will be generated from the MG expert system for the same geometry, loading, and boundary conditions. This is illustrated in the two meshes in Figure 2.

The model generation program takes as input four families of input features for the following four types of data: (1) Geometry, (2) Loading, (3) Boundary Conditions, and (4) Performance specifications.

## 1. Geometric features.

The current system has two basic geometric features that have several attributes that enhance their generality and ability to describe the geometry of the artifact. These two are:

- *tri* - triangular planar plate or curved shell.
- *quad* - quadrilateral planar plate or curved shell.

Any of these features can have one or more of the following attributes:

- *care* - circular arc as an edge.
- *chole* - circular hole.

The set of geometries that can be represented using these features and attributes is quite general as will be demonstrated in the examples.

## 2. Loading features.

The current system supports two types of loading features. These two are:

- *point load* - a force or moment applied at a point.
- *line load* - a distributed force or moment applied along a line.

## 3. Boundary condition features.

The current system supports two types of boundary condition features. These two are:

- *point fix* - a restraint applied at a point.
- *line fix* - a restraint applied along a line.

## 4. Performance specifications

The system supports four of performance specification features:

- *point disp* - allowable translation or rotation at a point.
- *general disp* - allowable translation or rotation in the artifact.
- **tension str** - allowable tension stress in the artifact
- *compression str* - allowable compression stress in the artifact.

### 3.1 Model generation expert system

The input for the mesh generation system is given in terms of the features listed above. This input is first processed by a geometric pre-processor (written in C) that generates the input file for the Model Generation (MG) expert system (written in OPS83).

The MG expert system uses the features that are passed into it to search for the first rule to fire. The model building procedure for finite element analysis is composed of several interconnected tasks:

1. Analysis type selection.
2. Geometric representation of the artifact in Finite Element terminology: Keypoints, Line segments, and Elements.
3. Loading representation in Finite Element terminology.
4. Boundary condition representation in Finite Element terminology.
5. Control and display commands for the finite element program.

The current version of MG prepares the input file for analysis using the ANSYS program. ANSYS is a general purpose finite element program that has enhanced pre and post processing capabilities, that reduce the number of calculations needed for the explicit generation of all nodes, lines, and elements. This saves the user from using its own numerical routines for the generation of the complete model. The MG program can easily be tied to other analysis packages by replacing the final output with another format.

### 3.1.1 Analysis **type** selection

The selection of the analysis type is controlled by the performance specification features. Limits on stresses and displacements will yield a static analysis procedure. Limits on buckling loads and/or vibration frequencies will lead to eigenvalue analysis for the model. The type of analysis will then determine several parameters that will influence the process of mesh generation for the model.

### 3.1.2 FE mesh generation

The mesh **that** is used for **the** analysis will determine the accuracy and reliability of the results. The main concern of the analyst is to device a mesh for analysis that will capture the characteristics of the response of the part to the applied loads. Over the last 30 years much experience has been gathered about "good" and "bad" finite element meshes for analysis. Most of these are in the form of rules of thumb, but several have mathematical and analytical support from research in FE convergence and accuracy (such as element aspect ratio, element internal angles, etc.). Some of these guidelines are summarized in the collection that was published by ASCE (Meyer, 1987).

The possible starting points in the model building process that are implemented in MG are:

1. Holes - the presence of a circular hole in a feature gives rise to possible stress concentration along the hole edge, and around it.



2. Loaded points - at which we expect stress concentrations and will require a finer element meshing.
3. Fixed points - where we expect the same phenomena as for the loaded point.
4. Re-entrant comers - where singularities might occur (these are defined as regions where the material edges make angles that are smaller than  $135^\circ$ ).

From the above we can get an initial state for the distribution and size of the elements in the model. The elements are generated in regions by defining the number of elements along the edges of the region. Thus, using the data that is generated from the C pre-processor, divisions along edges are determined based on the feature that is meshed. The special features that are pointed above, control the number of elements, the ratio of neighboring elements and the internal angles within the elements. The expert system fires rules that control that the mesh that is generated is in compliance with the practices of "good" mesh generation.

All other features that are not meshed in the first cycle based on the special cases outlined above, are then meshed based on their partial division information that is generated for the edges that are common to the special features. The same rules, that control the element sizes, are applied again to divide all the remaining unmeshed regions.

The current MG system includes another phase that is temporary, and will be removed in future version due to the limitations of the current version of the finite element analysis program (ANSYS 4.3). The meshing commands for the current version of ANSYS were not producing in many cases acceptable results and thus another phase was added to impose equal number of divisions on opposite edges in a feature. This will be dropped when version 4.4 of ANSYS will be used since it overcomes these limitations.

In Figure 3 an example is given for the nodes, line segments, and elements that are generated using MG for the lift arm part in CASE.

### 3.13 Loading

The loads are input in a general way and are transformed to nodal, edge, or face form based on the mesh that was generated for the model. The presence of a load at a point will cause the meshing procedure to reduce the element size around that point, to reduce modeling errors.

### 3.1.4 Boundary conditions

As for the loads there is a transfer of the general input features into the appropriate model entities that were generated for the geometric representation. As for loaded points, support will also case mesh refinement in their vicinity.

### 3.13 Control and display

The commands that are needed to perform the desired analysis are generated and added into the model. Graphic commands that enable the user to view the generated model during the execution of the analysis are added too. Additional commands are given for dumping the finite element analysis package internal representation into files that will be needed for the Strength Critic.

A second file for ANSYS post processing is generated as the final phase of the model generation procedure. It contains commands for plotting deflected shapes and stresses for interpretation by an expert, and for the general knowledge of the novice analyst (Fig. 4).

## 4 Result interpretation and recommendations

The Strength Critic (SC) expert system looks into the results of the final element analysis in two stages: In the first it prepares a report of all the violations in the model of the performance specifications set by the analyst\* and in the second it tries to reason about the characteristics of the model response under the loads\* and gives recommendations for changes in the design.

The report is summarizing all violations in the performance specifications (*point disp, general disp, tension str, compression str*). **All** the results are given in a sorted manner, for ease of reasoning about the cause of problems, **and** comparisons with the recommendations for changes.

The strength critic matches assumed modes of behavior with the actual output from the analysis. If these match, than the cause of violations is known, and the known set of recommendations for change is printed. If there is no known reason, in the special features that can cause the observed behavior, a check is made to determine if the overall dimensions in the region of violation of specifications are too small. Confirmation of this hypothesis, will yield the appropriate recommendation.

In the current state of checking of the SC module, all the possible cases of behavior characteristics were identified by the system. It is almost certain that in future tests problems that were not anticipated will arise, and additional rules will be added. In Fig. 5 two examples of recommendations that were

made by the SC expert system are given.

## 5 Summary

This paper presents the mechanical strength critic in *CASE*. Two expert systems are used as front and back ends for the analysis. The first - *MGES* - Model Generation Expert System, has the task of preparing the input commands for pre processing and analysis by ANSYS. The second - *SCES* - Strength Critic Expert System, has the tasks of summarizing the results, interpreting them, communicating them to the designer, and making recommendations with respect to improvements in the design.

*MGES* draws information from several sources for the task of finite element model building. Most of the information will come from the synthesis process in *CASE*, where features are assembled to design the part, and from a generic elements library that contains knowledge about features and their finite element modeling. The relationship among features, in addition to loading, boundary conditions, and performance requirements for the part, are then transformed, utilizing knowledge about finite element model construction (such as specific element type deficiencies, element aspect ratio limitations, and distortion), into input for the ANSYS program pre processor. Analysis type, and to some extent, mesh refinement demands, are extracted from a set of performance specifications for the designed artifact. The performance specifications give a list of items that are of interest to the designer such as the maximum allowed deflection in a given direction, the highest stress values allowed, Buckling load safety factor, maximum out of plane deflections , etc.

Given the ANSYS analysis results, *SCES* is executed. The analysis results contain a lot of information that is not very important for the designer. Only relevant values, about extreme stresses, maximum displacements, buckling loads, and vibration frequencies are summarized and reported. These values are compared with the performance specifications for compliance, and warning messages are issued. The failure mode is identified, and recommendations are made as for changes in the part design. The system is designed to provide the designer with detailed analysis results and plots on demand, through a graphic interface.

Future work on the system will concentrate on the enhancement of the modeling and diagnosing phases. A new input feature that will include just the contour of the artifact will be introduced, and then there will be the option to accept input from commercial CAD systems. As for the automatic implementation of the recommended changes, this will be included in a separate version for designs that have the mechanical strength as their main concern.

## References

- Baehmann, P., Wittchen, S., Shephard, M., Grice, K., and Yeny, M., 1987.** Robust, geometrically based, automatic two-dimensional mesh generation. *International Journal for Numerical Methods in Engineering*, **24**,1043-1078.
- Bykat, A., 1976. Automatic generation of triangular grids: i-subdivision of general polygon into convex subregions, ii-triangulation of convex polygons. *International Journal for Numerical Methods in Engineering*, **10**,1329-1342.
- Cavendish, J., 1974. Automatic triangulation of arbitrary planar domains for the finite element method. *International Journal for Numerical Methods in Engineering*, **5**, 679-696.
- Cavendish, J., Field, D., and Frey, W., 1985. An approach to automatic three-dimensional finite element mesh generation. *International Journal for Numerical Methods in Engineering*, **21**,329-347.
- Cheng, R., Jaromczyk, J., Lin, J., Chang, S., and Lu, J., 1989. A parallel mesh generation algorithm based on the vertex label assignment scheme. *International Journal for Numerical Methods in Engineering*, **14**,1429-1448.
- Cook, W., 1974. Body oriented (natural) coordinates for generating three-dimensional meshes. *International Journal for Numerical Methods in Engineering*, **8**,27-43.
- Fredrick, C., Wong, Y., and Edge, F., 1970. Two dimensional automatic mesh generation for structural analysis. *International Journal for Numerical Methods in Engineering*, **2**,133-144.
- Gordon, W. and Hall, C, 1973. Construction of curvilinear coordinate systems and applications to mesh generation. *International Journal for Numerical Methods in Engineering*, **7**,461-477.
- Haber, R., Shephard, M, Abel, J., Gallagher, R., and Greenberg, D., 1981. A general two-dimensional finite element preprocessor utilizing discrete transfinite mapping. *International Journal for Numerical Methods in Engineering*, **17**, 1015-1044.
- Lee, Y, De-Fenington, A., and Shaw, N., 1984. Automatic finite element mesh generation from geometric models - a point based approach. *ACM Trans, Graphics*, **3**,287-311.
- Meyer, C, 1987. *Finite Elements Idealization*. ASCE, New-York.
- Rchg, J., Elfes, A., Ifelukdar, S., Woodbury, R., Eisenbeiger, M, and Edahl, R., 1988a. Case: computer aided simultaneous engineering. In *Artificial Intelligence in Engineering: Design*, pages 339-360, Computational Mechanics Publications, Southampton.
- Rehg, J., Elfes, A., Talukdar, S., Woodbury, R., Eisenberger, M., and Edahl, R., 1988b. Design systems integration in case. In *Expert Systems for Engineering Design*, pages 279-301, Academic Press.
- Rivara, M., 1987. A grid generator based on 4-triangles conforming mesh refinement algorithms. *International Journal for Numerical Methods in Engineering*, **24**,1343-1354.
- Sadek, E., 1980. A scheme for the automatic generation of triangular finite elements. *International*

*Journal for Numerical Methods in Engineering*, 15, 1813–1822.

Sapossnek, M., Talukdar, S., Elfes, A., Sedas, S., Eisenberger, M., and Hou, L., 1989. Design critics in the computer-aided simultaneous engineering (case) project. In *Symposium on Concurrent Product and Process Design, Winter Annual Meeting*, ASME.

Zienkiewicz, O. and Phillips, D., 1971. An automatic mesh generation scheme for plane and curved surfaces by isoparametric coordinates. *International Journal for Numerical Methods in Engineering*, 3, 519–528.

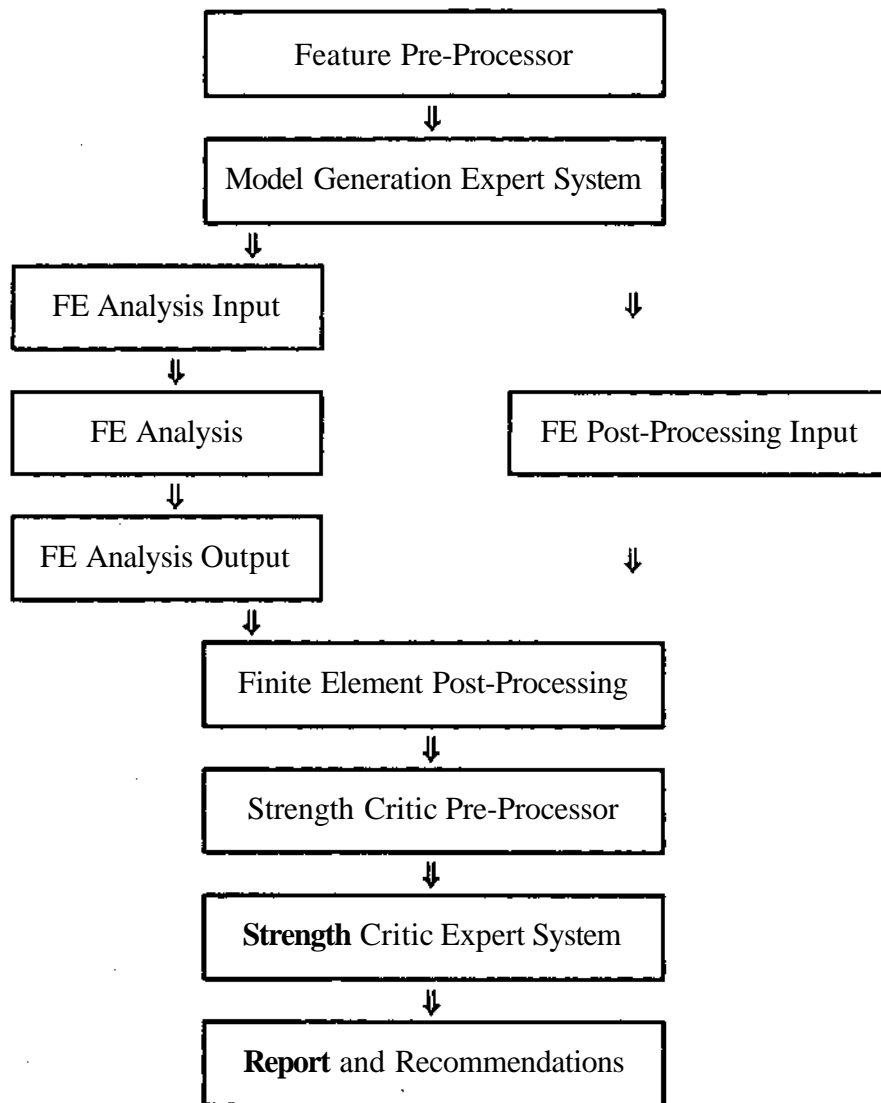


Figure 1: Architecture of the system.

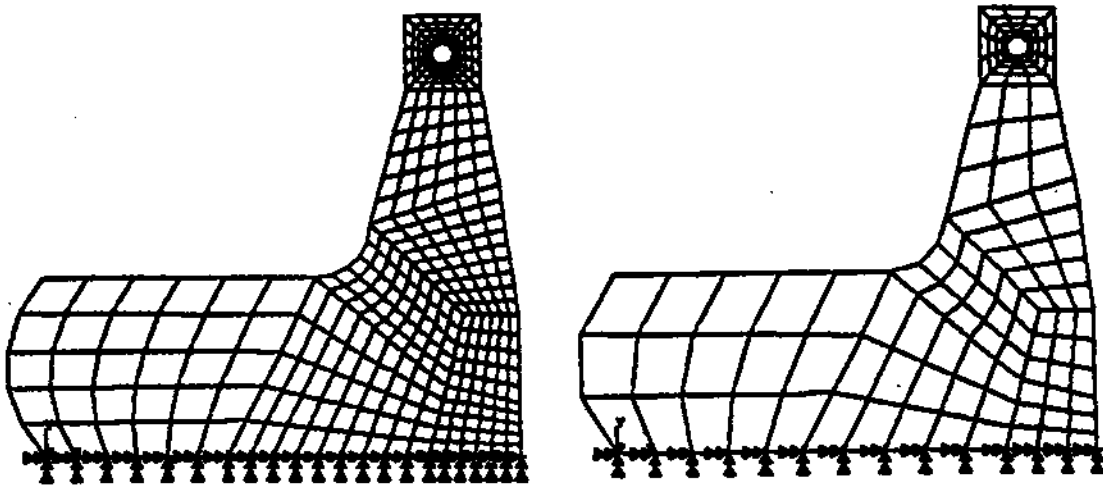


Figure 2: Meshes for stress and stability analyses.

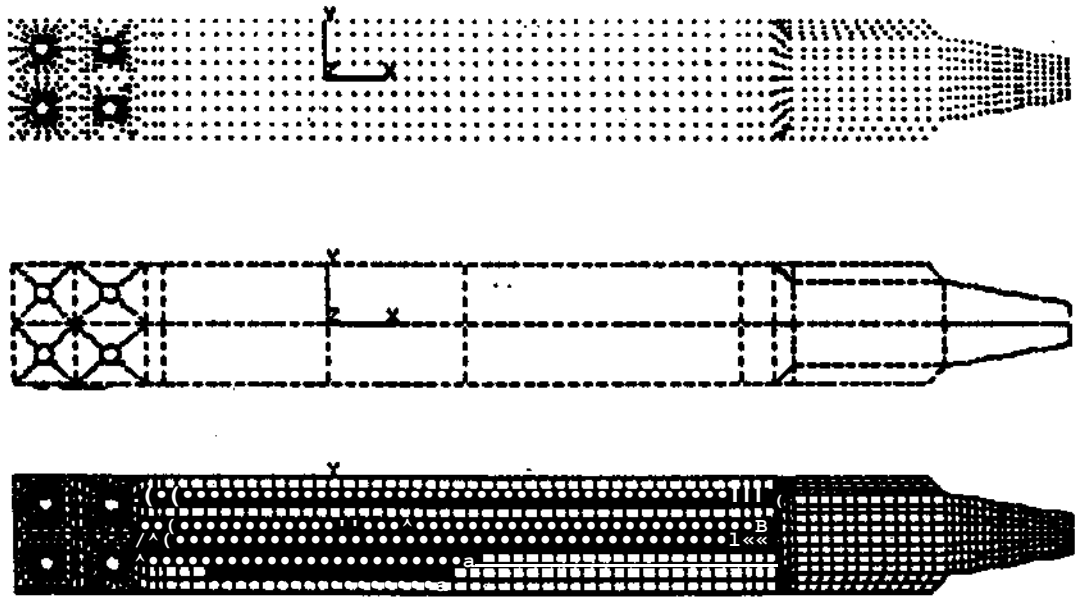


Figure 3: Finite element model for the lift aim.

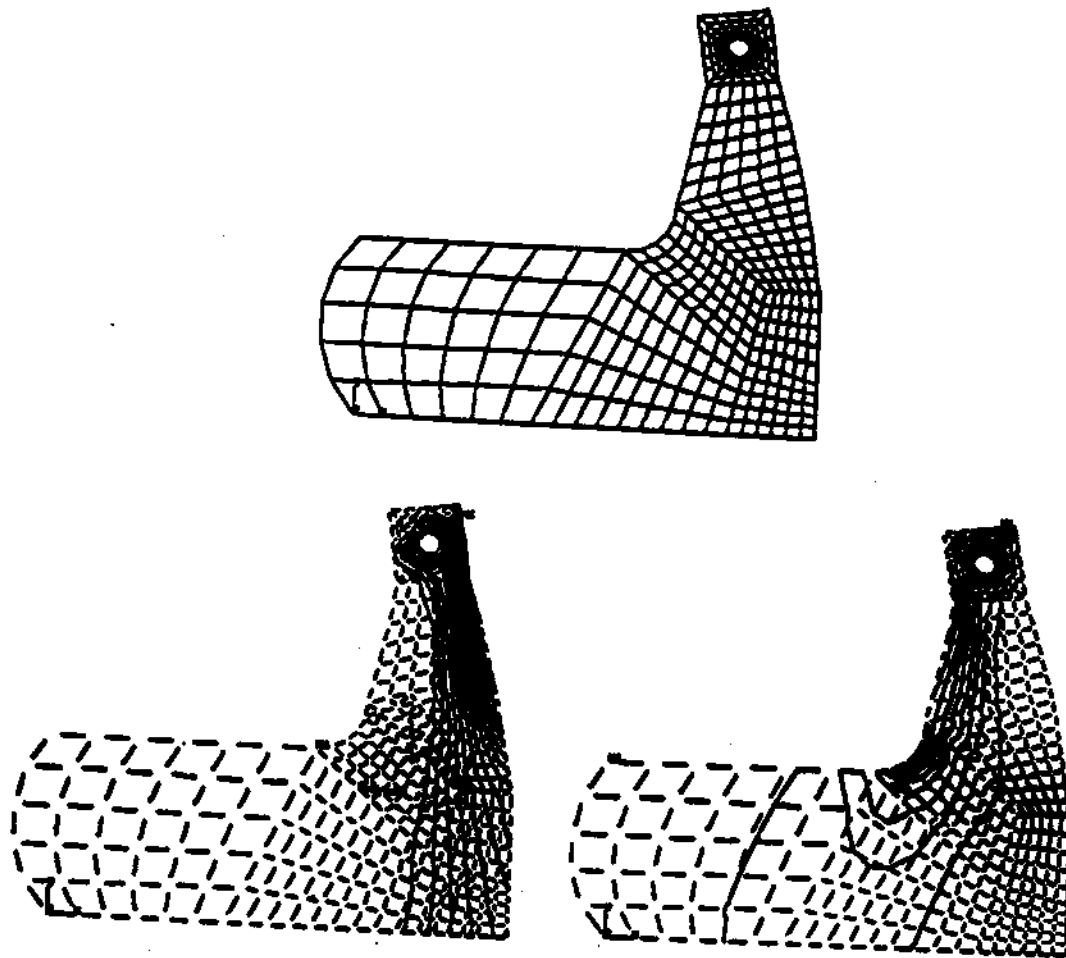


Figure 4: Deflected shape and stress plots.



*I compression stress is too high along edge connecting points ( 0,10,0) and (15,10,0)*

*The reason is that the overall dimension in this area is too small -*

*try increasing dimensions or thickness to lower stresses*

*II compression stress is too high at ( 26,20,0)*

*The reason is the concentrated load at this point*

*Check if load is really a point load -*

*if yes - increase thickness at this point,*

*if no - model load more accurately at this point*

**Figure 5: Sample recommendations from SC.**