GRID POINT STRESS CALCULATION, ERROR PREDICTION AND AUTOMATIC REMESHING PROCEDURE

by

J. C. Cheng, G. C. Campbell and L. I. Nagy

Ford Motor Co., Light Truck CAE & PVD

Dearborn, MI

ABSTRACT

This paper describes the following three post-processing methods as applied to MSC/NASTRAN static analysis:

1. Grid point stress computation
2. Error prediction due to mesh discrepancy
3. Automatic remeshing procedure

INTRODUCTION

The accuracy of finite element static analysis is a function of several criteria:

1. Element formulation
2. Numerical errors due to computer round off
3. Element discretization
4. Element stress recovery point

The user has no control over the element formulation or round off errors. These can be controlled by the code developers choice of element theory, and numerical methods. Therefore we will only address the last two items, discretization and stress recovery.

In MSC/NASTRAN[1] CQUAD4 and CTRIA3 elements, the stresses are recovered at the centroid of the elements. Although this stress recovery procedure may be acceptable for many problems, for others, the predicted stress values are unacceptable. An example of the latter case was demonstrated by Nagy[2] using a cantilever plate model, where the calculated maximum stress values varied between zero and two-thirds of the actual value as a function of the adopted mesh size. This error is clearly unacceptable, especially in automobile thin-walled structures, where the maximum stresses usually occur at the extreme edges. It is necessary therefore, to provide for a grid point stress recovery option, which would more accurately predict the stresses at the grid points.

However, a good grid point stress calculation routine alone does not guarantee the "goodness" of the mesh size or the "accuracy" of the analysis. Thus some error estimation capability will provide a good engineering tool, especially for new engineers.

We have accomplished the above tasks in the post-processor phase. First, the grid point stresses are calculated using the grid point displacement, then a posterior error estimator can be calculated by the program. At this point the analyst has the opportunity to evaluate the results, and if necessary redefine the mesh. If the answer is positive, the initial mesh will be subdivided based on the error estimator. The entire procedure is illustrated in Fig. 1.

Each step mentioned above will be described in detail in the following sections, examples will follow.

GRID POINT STRESS CALCULATION

As stated before, the stress output of MSC/NASTRAN CQUAD4 and CTRIA3 elements are recovered at the element centroid by default. Sometimes this will underestimate the true stress values. Although it is possible to obtain the grid point stresses in MSC/NASTRAN, it can be shown that in some cases, the result will still be inaccurate as will be illustrated in Example 1. One further drawback is that the currently available grid point stresses can not be output in a punch file format but only by OUTPUT2 file. Since OUTPUT2 file is a binary file and machine dependent, to recover this file from a mainframe to a work station is rather cumbersome.

Since the accuracy of the grid point displacements are one order higher than the stresses, even for the course mesh, we use the strain gage analogy to calculate the grid point stresses. The procedures are described as follows:

First, the corner displacements of the top and bottom surface of the element are calculated using the grid point displacement and rotations. These corner displacements are then used to calculate the stretching along the element edges and toward the element centroid. These are then converted to strains. This can be easily illustrated in Fig. 2(a). One can establish the strain Rosette equation[3] as:

$$\xi_A = \xi_X \cos^2\theta_1 + \xi_Y \sin^2\theta_1 + \gamma_{XY}\cos\theta_1 \sin\theta_1$$

$$\xi_B = \xi_X \cos^2\theta_2 + \xi_Y \sin^2\theta_2 + \gamma_{XY}\cos\theta_2 \sin\theta_2$$

$$\xi_C = \xi_X \cos^2\theta_3 + \xi_Y \sin^2\theta_3 + \gamma_{XY}\cos\theta_3 \sin\theta_3$$

Where the strain components ξ_A, ξ_B and ξ_C are calculated for element corner points at both top and bottom surfaces, resulting in eight sets of strain vectors for each CQUAD4 element and six sets of strain vectors for each CTRIA3 element.

The stress vectors for all corner points can be obtained from the material constitutive law as:

$$\{\sigma\} = [D]\,\{\xi\}$$

The displacement finite element formulation only guarantees continuation of displacement fields but not the stress fields. Under the assumption that the stress fields are also continuous for homogenous material, an averaging process must be enforced in the program. The effect of element size and performance (i.e., CQUAD4 is more efficient than CTRIA3) are also taken into account by some weighting factors. The Von Mises stresses are then calculated based on the averaged stresses. These stresses are output to be post-processed by PATRAN or some other graphic packages.

LOCAL AND GLOBAL ERROR CALCULATION

Since the finite element method is an approximation to the true solution of the mathematical problem posed, it is important for the user to know the magnitude of errors at each stage of subdivision, i.e., devising a posterior error estimator to estimate such errors. A survey was performed on the current literature[4–7]. Two different approaches were adopted in the program.

Approach 1: Grid point stress deviation

Here we assumed that the averaged grid point stresses are a good approximation of the "exact" solution. Since the grid point stresses of different elements sharing the same node are different before averaging, the deviations between the averaged stresses and unaveraged stresses are a good estimation of the solution error. Grid point stress deviations are calculated and output as an option in the current program.

i.e., for each grid point:

$$\Delta\sigma = \frac{\sqrt{\sum_{i=1}^{N} (\sigma_{fe}^{i} - \sigma_{ex})^{2}}}{N}$$

where σ_{fe} is the unaveraged stress vectors
σ_{ex} is the averaged stress vector
N is the number of elements connected to that grid point

The grid point stress deviation does provide physically useful information. But for some situations such as a grid point connected to a single element, there is no deviation because σ_{fe}=σ_e. Thus, a more elegant approach may be required.

Approach 2: Error of energy norm

Since we have no access to the finite element source code, the error estimator proposed by Zienkiewicz and Zhu[4] was used because of its simplicity and the fact that it does not require the modification of the main source code.
This error estimator has been shown to be effective and convergent. For linear elastic problem, the energy norm ||U|| can be written as:

$$\|U\| = \left[\int_{\Omega} \left(\sigma_{ex}^{T} D^{-1} \sigma_{ex}\right) d\Omega\right]^{\frac{1}{2}}$$

where D is the constitutive matrix

The error of energy norm ||e|| can be written in a consistent fashion:

$$\|e\| = \left[\int_{\Omega} (\sigma_{ex} - \sigma_{fe})^{T} D^{-1} (\sigma_{ex} - \sigma_{fe})\, d\Omega\right]^{\frac{1}{2}}$$

The error estimator is not only quite accurate globally but it can be decomposed into element subdomain and used as local error indicator. i.e.,

$$\|e\|^2 = \sum_{i=1}^{m} \|e\|_i^2$$

where $\|e\|_i$ is similar to $\|e\|$ but only integrate through element subdomain.
m is the total number of elements in the analysis.

By using the grid point stress deviation or the element error of energy norm as local error indicators, an effective adaptivity process can be developed with the objective of achieving an overall percentage accuracy in the energy norm. The percentage error is defined as:

$$\eta = \frac{\|e\|}{\|U\|} \times 100$$

Now we have all of the ingredients of an adaptivity process. From the percentage error, the user can decide whether the solution quality is acceptable. If he wants to refine the mesh, he can use either the grid point stress deviation or the error of energy norm to decide which area needs a finer mesh. The new model can be reanalysis until the percentage error lower than a threshold value. The refining procedures can be performed either interactively or by computer program. Two different remeshing capabilities are currently available in the program to provide the least amount of user interaction. Detail description is in the following section.

AUTOMESH

In both approaches, the elements are subdivided based on the original mesh. Each element of the old mesh is treated as a patch or surface. Subdivision of those surfaces are performed based on the local error estimators. Two approaches are currently available.

Approach 1: refinement based on grid point stress deviation

In this approach, each edge of the element will be subdivided. The number of subdivisions are determined by the relative grid point stress deviation of the two nodes connecting that edge. The advantage of this approach is that each element is handled without information about the other element, and full compatibility can be expected. The disadvantage is that the different subdivisions of the element edges can sometimes create distorted elements.

The program can generate PATRAN session files[8] based on the user's request. New PATCHes are generated for each element. MESH commands are also generated to subdivide the elements. The user needs to run the session file through PATRAN to get a new MSC/NASTRAN bulk data. The quality of the new mesh depends on the automeshing capability of the graphic program; it can be further improved if better automeshing capability is available.

Approach 2: refinement based on element error of energy norm

In this approach, the elements with high error of energy norm are subdivided. The program can generate new MSC/NASTRAN bulk data based on the user's request. Each CQUAD4 element is subdivided into four new CQUAD4 elements while one CTRIA3 element is subdivided into one CQUAD4 and two CTRIA3 elements. A typical example of remeshing is illustrated in Fig. 2(b). The disadvantage of this approach is that in order to assure element compatibility, MPC cards are required. Elements can not be subdivided without the information of the surrounding elements. The advantage of this approach is that the majority of the resulting element shapes will be as good as the original shapes, i.e. no degradation.

Several observations were made from the two approaches above. First, in approach one, grid point stress deviation only considers the absolute error at each node, therefore the refinement usually happen locally. If the user is more interested in the peak stress, this approach may be more desirable. In approach two, the refinement is based on the error of energy norm of the entire element considering the element size. If the user is more interested in the overall stress distribution, this approach may be better. Second, at this time, the program only subdivides the elements based on their original geometry. For curved boundary conditions, the newly generated grid points do not move to their curved location. This drawback can be fixed with some modification of the program. Third, since different load cases and boundary conditions will create different stress distributions and error indicators, different meshes will be generated to optimize the results for different cases. This capability will be extremely powerful since the user can have the option to design different meshes based on different load cases.

EXAMPLES

1. Cantilever Beam under Moment at Free End

A cantilever beam fixed at the left end was analyzed. Total length is 200 mm and depth is 10 mm. The moment applied at the free end is 100 N-mm(Fig. 3(a)). Only one layer of CQUAD4 element was modeled through the depth. Even though the displacements were quite accurate from the MSC/NASTRAN results, MSC/NASTRAN will predict a stress free condition because the default stress output is at the element centroid. Also, MSC/NASTRAN grid point stresses are extrapolated from the element stresses, the grid point stresses in this case will be also zero (Fig. 3(b)). However, using the strain Rosette equation, the stress calculated are very accurate, even for this coarse mesh (Fig 3(c)).

2. Plate With Circular Hole under Edge Tension

Due to symmetry, only one quarter of a plate was modeled. A distributed force of 10 N/mm was applied at the remote edge. To estimate the "exact solution" of this problem, a very fine mesh was analyzed first. The Von Mises stress contour of this mesh revealed a peak stress of 33.5 MPa.

The same problem was next analyzed by using a very coarse mesh (Fig. 4(a)). The Von Mises stress results, shown in Fig 4(b), revealed a peak stress of 28.7 MPa. The global error index predicted a 6.46 % error, and the maximum grid point stress deviation, shown in Fig. 4(c), was 3.6 MPa. The error of energy norm distribution is shown in Fig. 4(d).

The coarse mesh was automatically remeshed using one of the two approaches discussed. In the first approach, the elements were subdivided based on the grid point stress deviation (Fig. 5(a)). Using this method, the Von Mises stress contour in Fig. 5(b) showed a peak stress of 29 MPa in the remeshed model. The global error in this case was 5.1 % which was less than that of the coarse model. The maximum stress deviation in Fig. 5(c) also decreased , to 2.76 MPa. Fig 5(d) shows the error of energy norm plot for this case.

The second approach to automatically remesh the coarse model was based on subdividing the elements according to the error of energy norm (Fig. 6(a)). For this approach, the Von Mises stress plot shown in Fig 6(b) indicated the peak stress was 31.1 MPa. The global error reduced to 5. 1% and the maximum stress deviation also reduced to 2.76 MPa as shown in Fig. 6(c). Fig. 6(d) shows the plot of the error of energy norm.

More iterations can be performed in the same fashion mentioned above if one chooses to do so.

3. Fuel Tank Strap Analysis

A simple fuel tank strap under static loading at the two ends was analyzed (See Fig. 7(a)). For the initial analysis, a very coarse mesh was used. The critical area of the strap (See Fig. 7(b)) was zoomed in on and a grid point stress contour plotted in Fig. 7(c). For this mesh, the global error index indicated 39 %. The peak stress was 54.4 MPa. The coarse model was remeshed using approach 1 (Fig. 7(d)). Notice how the critical area was heavily subdivided based on grid point stress deviation. The stress results for this case reveal a peak stress of 93.5 MPa which is greater than the coarse mesh indicates. The global error index also decreases to 20%. The same coarse mesh was also remeshed using approach 2, i.e., the error of energy norm. The stress results are shown in Fig. 7(e). The peak stress was increased to 83.0 MPa whereas the global error index reduced to 21% only.

CONCLUSION

The program described above serves as a good practical tool in helping the analyst to appraise and obtain a good overall quality analysis. Reanalysis, if necessary, can be done in a very efficient manner. Further work may include additional investigation of other error estimators; extension of this concept to solid elements; recovering geometric detail when remeshing and exploring the possibility of applying it to the normal mode analysis.

ACKNOWLEDGEMENT

The authors want to express their gratitude to Mr. Jim Brancheau for his initial work on the grid point stress calculation. Special thanks to Ms. Ann Kulik in executing the example problems. Mr. Nick Kazan and Mr. Steve Marulis's management support is also appreciated.

REFERENCES

1. "MSC/NASTRAN User's Mannual, Version 66, The MacNeal-Schwendler Corporation, Los Angeles, CA, November 1988."

2. L. I. Nagy., Introduction to Finite Element Analysis and NASTRAN Utilization, 1985

3. E. P. Popov., Introduction to Mechanics of Solid, Prentice-Hall Press, 1968

4. O.C. Zienkiewicz and J. Z. Zhu, "A simple error estimator and adaptivity procedure for practical engineering analysis", Int. J. Numer. Methods Eng., 24, 337-357, 1987

5. I. Babuska and W. C. Rheinboldt, "Error estimates for adaptive finite element computation". SIAM J. Numer. Anal. 15 (4), 1978

6. D.W. Kelly, J.P. de S. R. Gago, O.C. Zienkiewicz and I. Babuska, "A posterior error analysis and adaptive processes in the finite element method-Part 1: Error analysis", Int. J. Numer. Mthh. Eng. 19, 1593-1619, 1983

7. I. Babuska, O.C. Zienkiewicz, J. Gago, E. R. de A. Oliveria., Accuracy and Adaptive Refinement in Finite Element Computations, John Wiley and Sons, 1986

8. PATRAN 2.4 User Manual

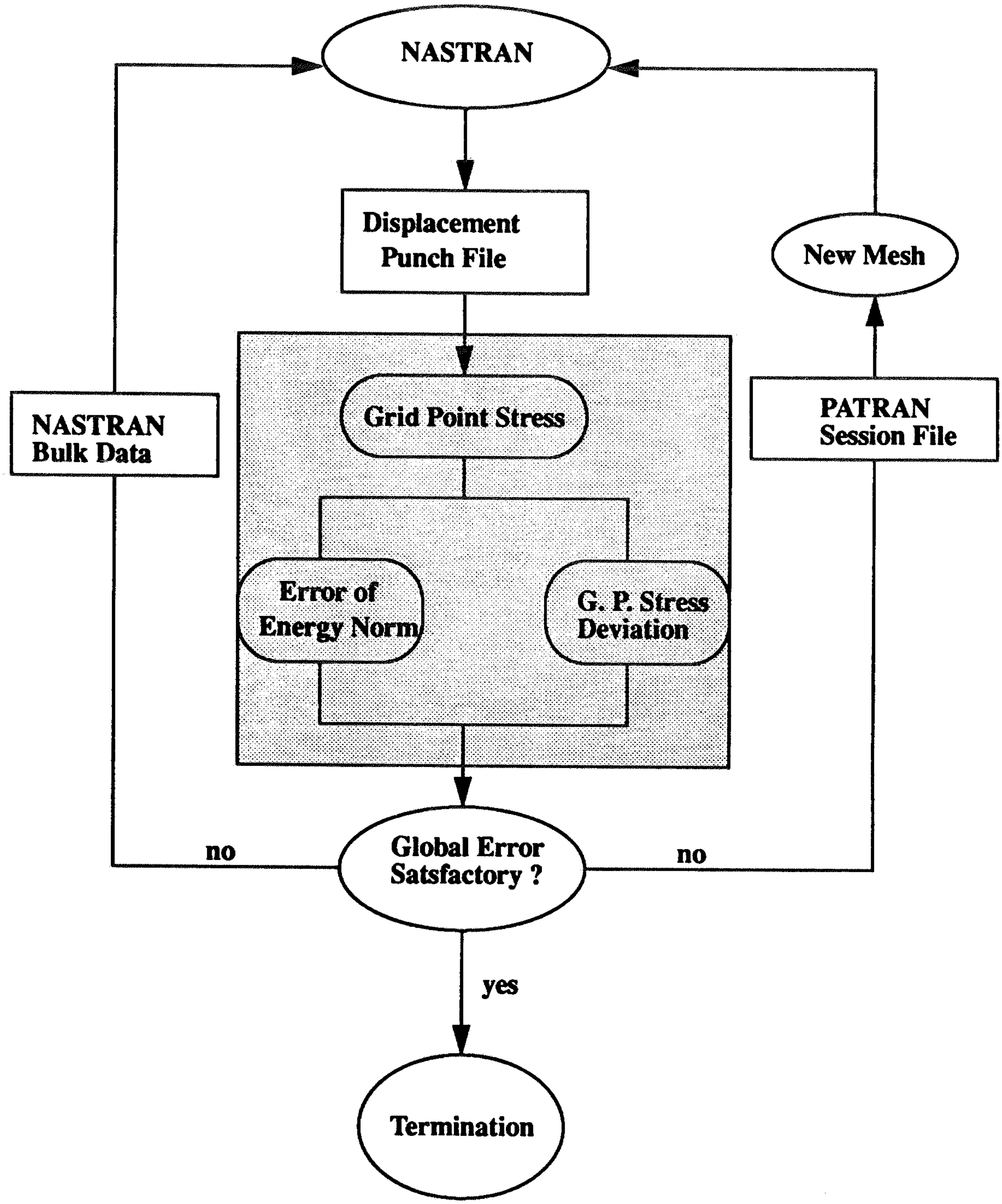

Fig. 1 Analysis Flow Chart At Ford Light Truck CAE Department
Shaded Area are Calculation Done by Program ERRGPS

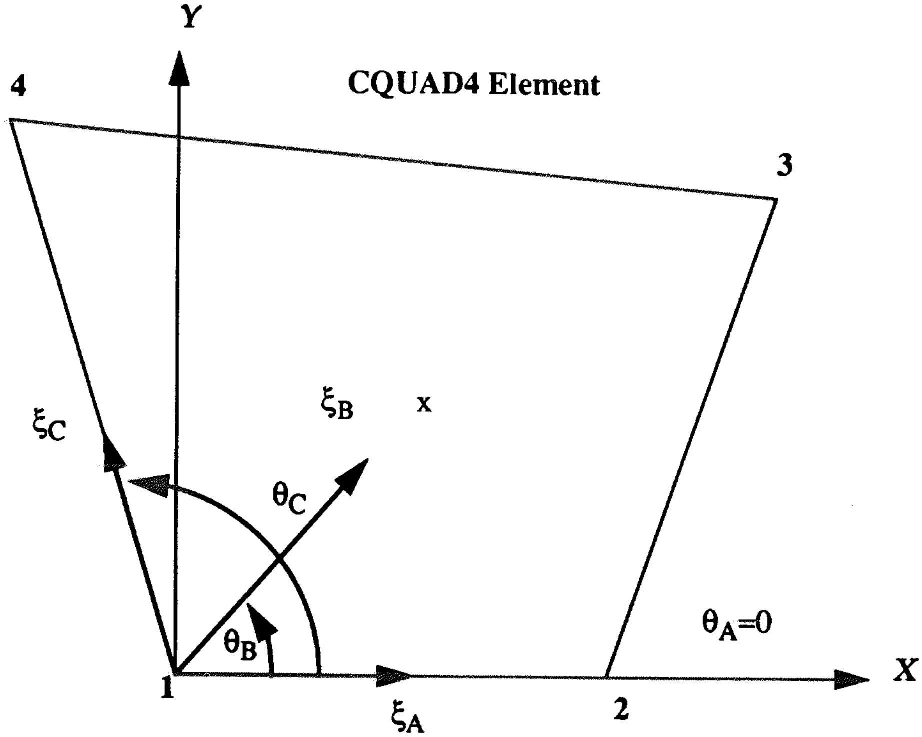

Fig. 2(a) Grid Point Stress Calculation

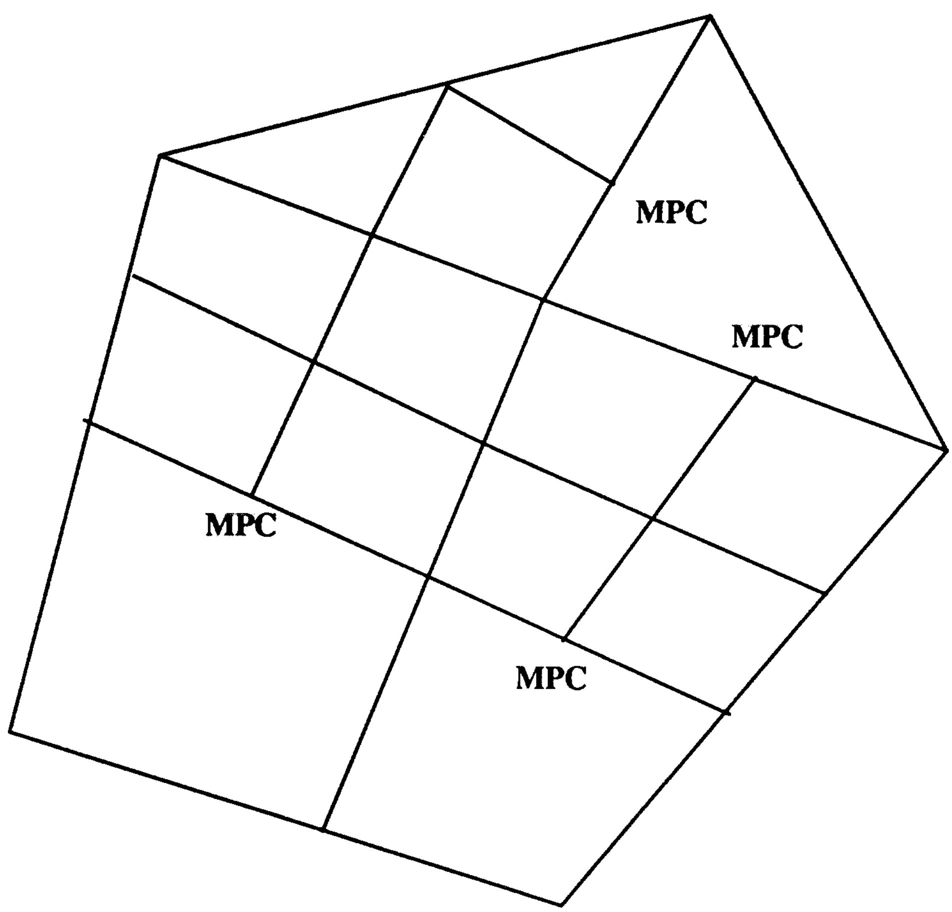

Fig. 2(b) Typical example of refinement based on Error of Energy Norm

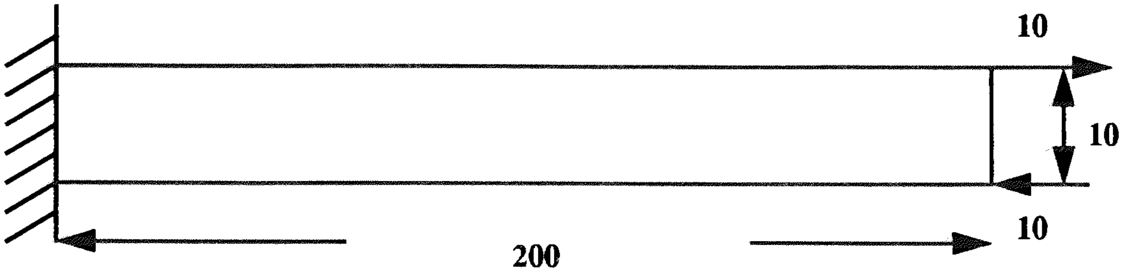

Fig. 3(a) Cantilever Beam Under Constant Moment

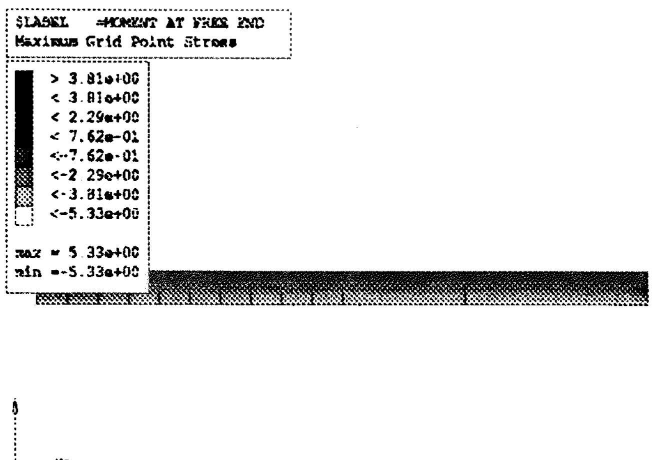

Fig. 3(b) Stress Distribution Based on Strain Rosette Equation

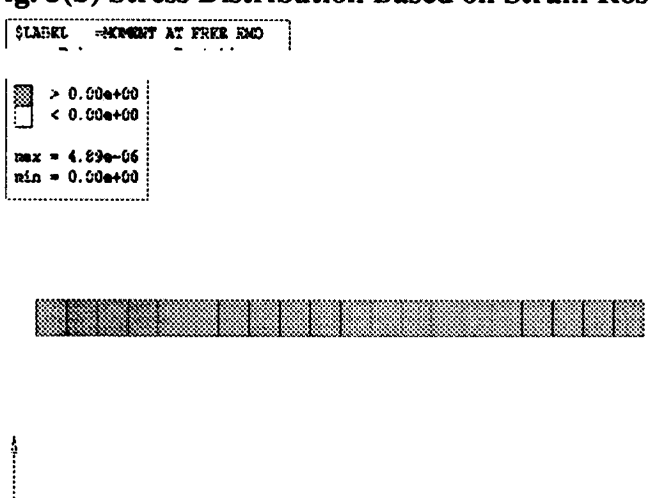

Fig. 3(b) Nastran Element Stress & Grid Point Stress

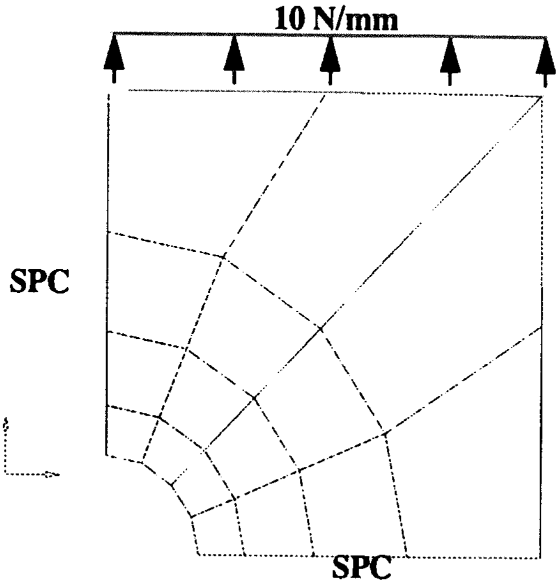

Fig. 4(a) Iniital Geometry and Mesh

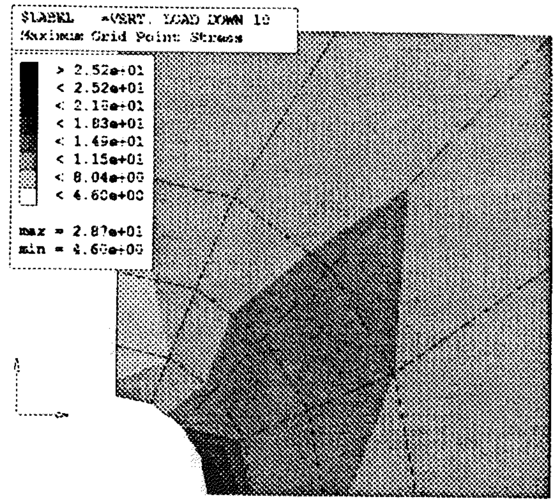

Fig. 4(b) Grid Point Stress

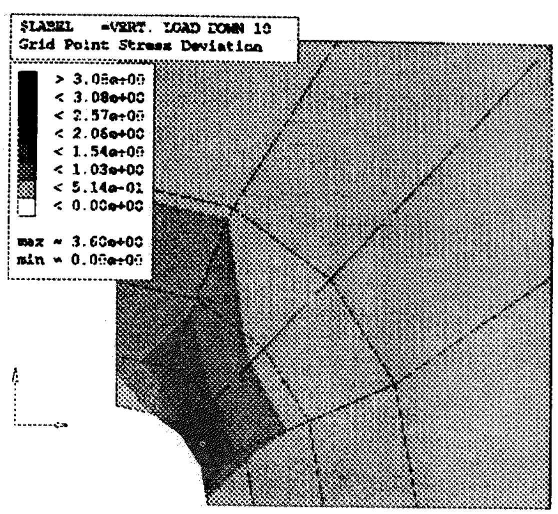

Fig. 4(c) Grid Point Stress Deviation

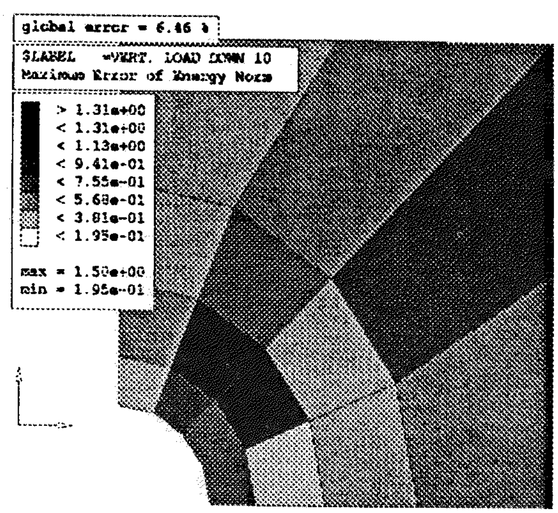

Fig. 4(d) Error of Energy Norm
Global Error = 6.46 %

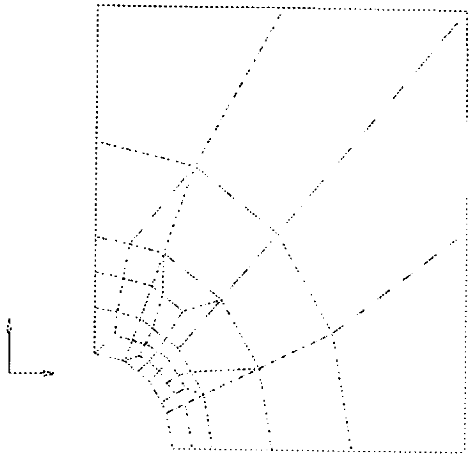

Fig. 5(a) New Mesh Based on G.P. Stress Deviation

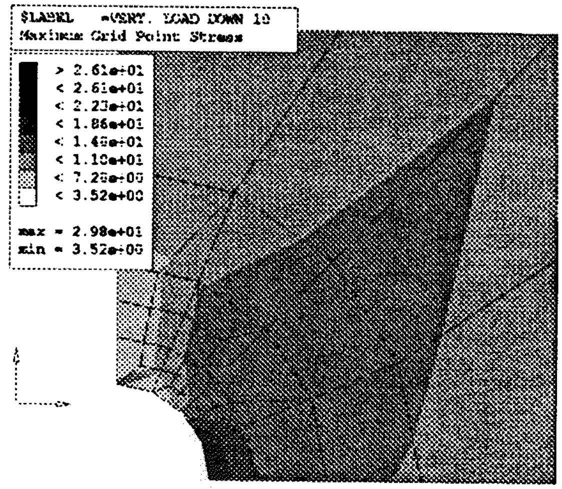

Fig.5(b) Grid Point Stress

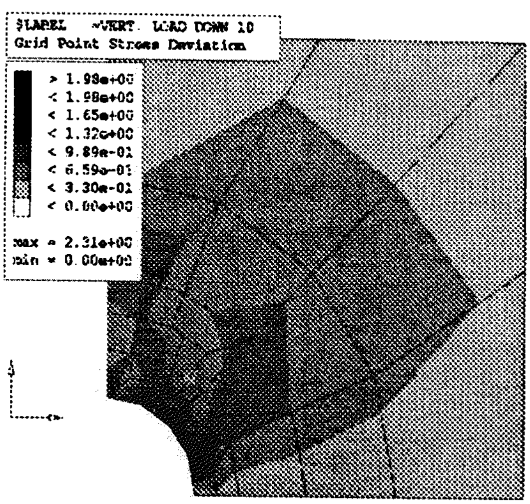

Fig. 5(c) Grid Point Stress Deviation

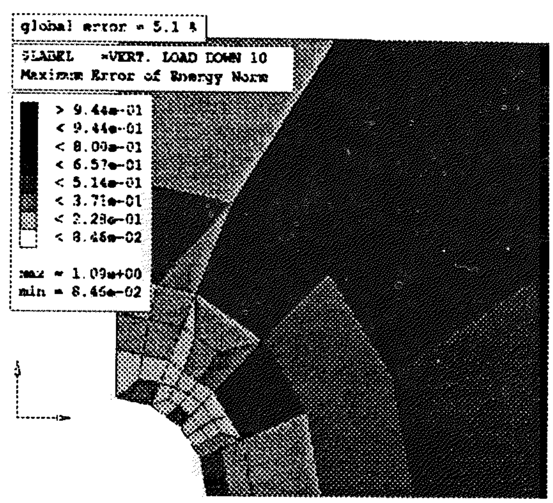

Fig. 5(d) Error of Energy Norm
Global Error = 5.1 %

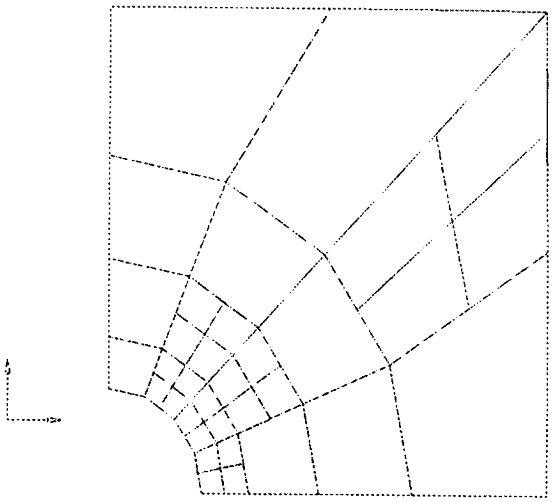

Fig. 6(a) New Mesh Based on Error of Energy Norm

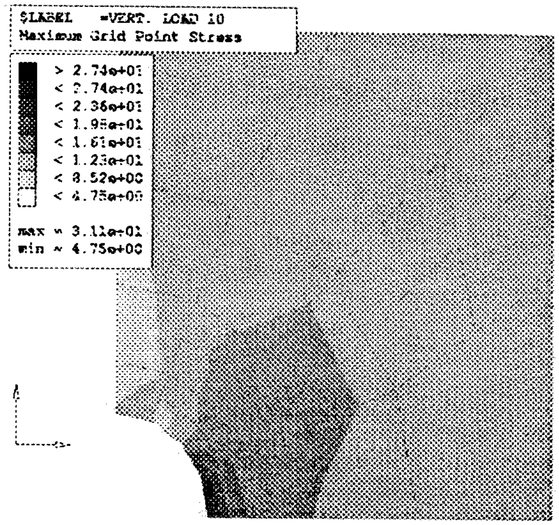

Fig.6(b) Grid Point Stress

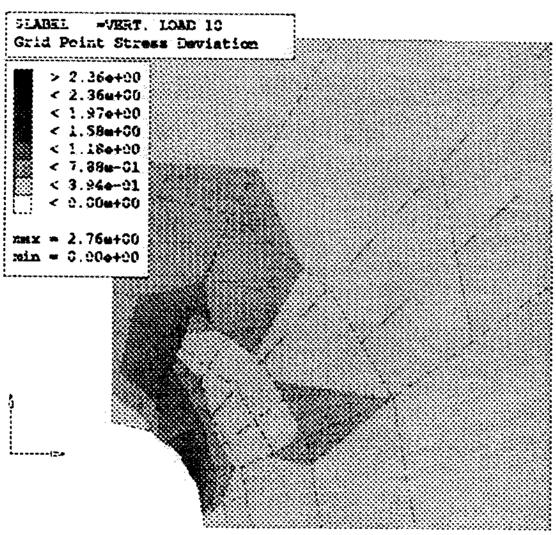

Fig.6(c) Grid Point Stress Deviation

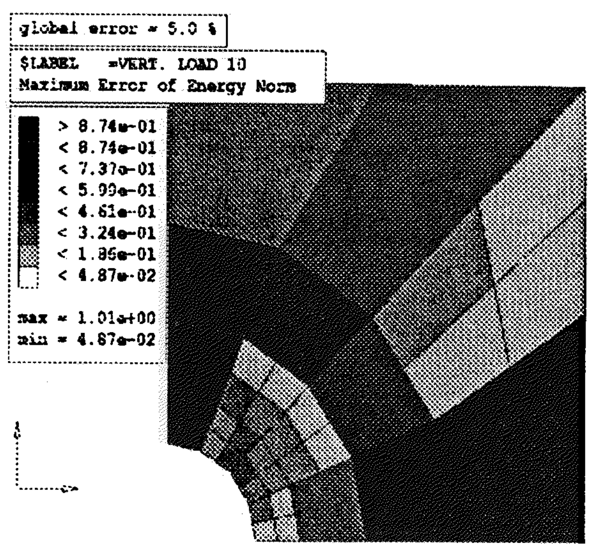

Fig. 6(d) Error Of Energy Norm
Global Error = 5.1 %

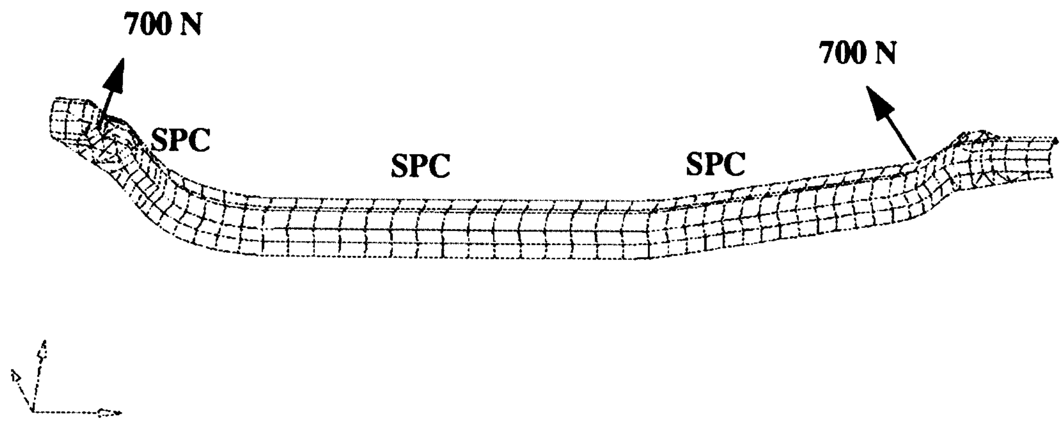

Fig. 7(a) Initial Mesh of Fuel Tank Strap

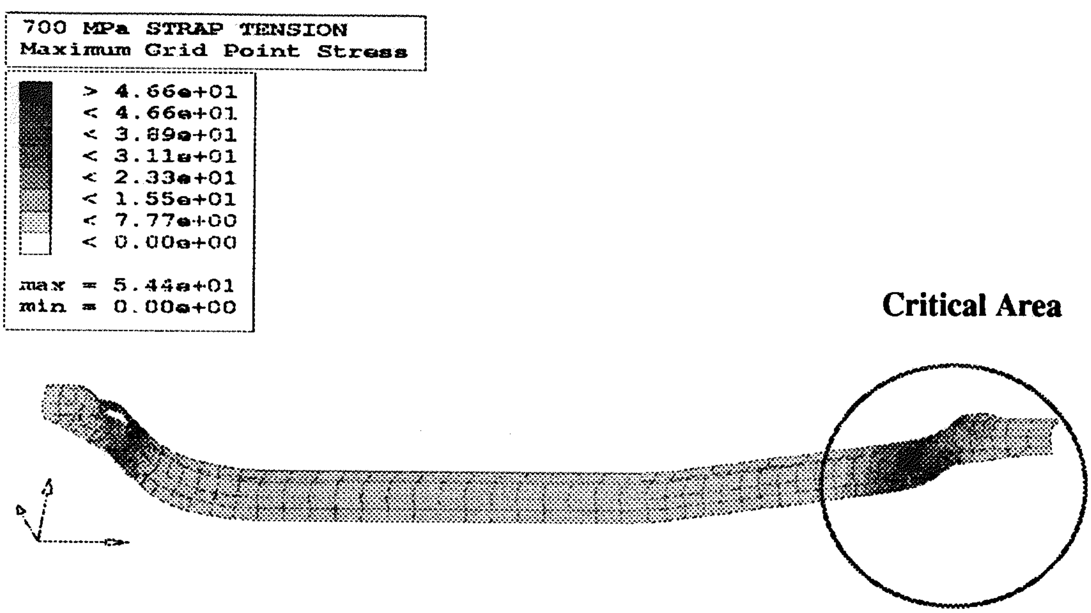

Fig. 7(b) Grid Point Stress

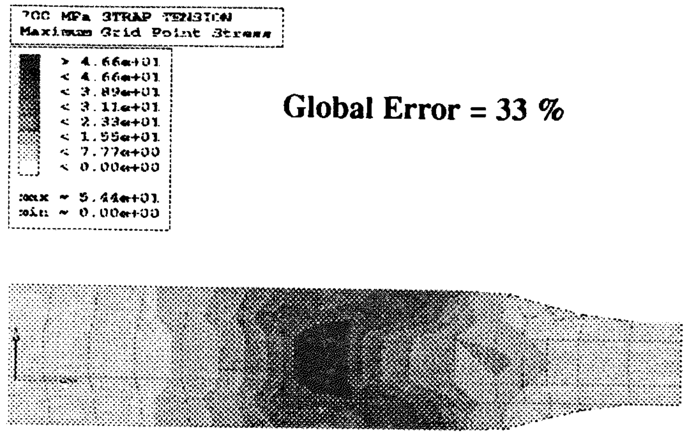

Fig. 7(c) Grid Point Stress: Initial Mesh of Critical Area

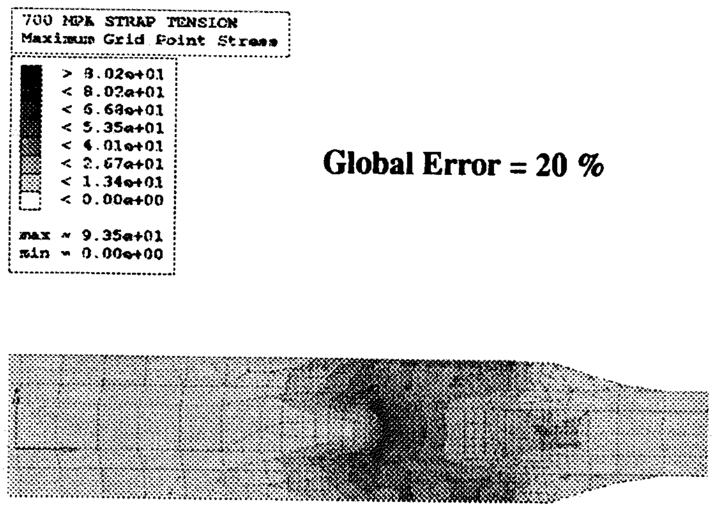

Fig. 7(d) Grid Point Stress: New Mesh Based on G. P. Stress Deviation

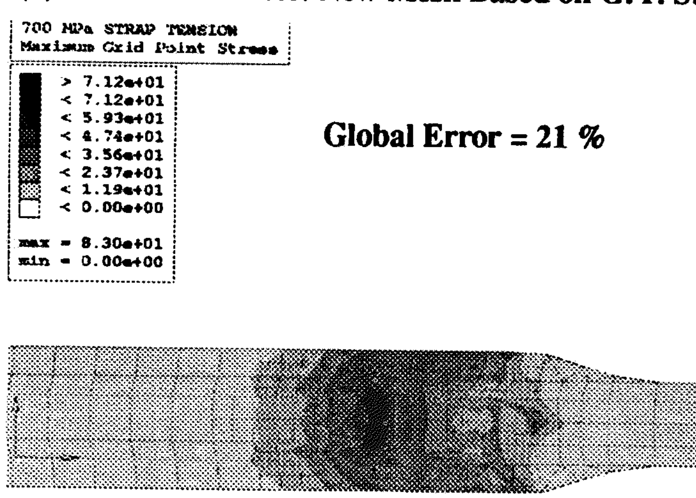

Fig. 7(e) Grid Point Stress: New Mesh Based on Error of Energy Norm