A comparison of convergence and modeling times for Cathode Ray Tube stress analysis with the Finite Element and Boundary Element Methods.

C.Y. Cha*, S.V. Iyer*, J. Trevelyan**

* Philips Display Components, Inc., Ann Arbor, Michigan, USA
** Computational Mechanics, Inc., Billerica, Massachusetts, USA

Summary

In a stress analysis of a cathode ray tube (CRT), it is important to consider the performance of the design under two loading conditions; the vacuum loading and the load applied by a pretensioned metal band around the tube. The stress analysis may be done using either the Finite Element Method (FEM) or the Boundary Element Method (BEM). With both techniques it is important to examine the convergence properties of the solution when the mathematical model is improved This gives a better idea of the true solution as well as giving information about the required mesh density for future models.

This study describes a comparison of the convergence properties of the solutions obtained for CRT as both FEM and BEM models are refined, giving recommendations for required mesh densities.

The study also considers the modeling times which were found to be needed to undertake the convergence studies, as this is an important factor in an industrial design analysis environment. The BEM modeling times were shown to be dramatically faster, particularly for the remeshing stage in the convergence study.

Introduction

With the potentially lethal consequences of a rapid implosive failure of a CRT in a home television it is essential that a detailed stress analysis of the component be performed at the design stage. Such an analysis will need to consider two major contributors to the stress and displacement profiles experienced by the product under normal operating conditions. Firstly, because the interior of a CRT is a vacuum, the atmospheric pressure acting on the outside of the tube will tend to compress it and will induce a state of stress. Secondly, a pretensioned, metallic "shrink-band" is applied to the outside of the tube, inducing a state of stress which acts to prevent a potentially dangerous implosive failure under impact[1]. The shrink band also induces a state of stress and displacement which must be calculated accurately The suitability of a cathode ray tube design is largely determined by two parameters - the maximum principal stress and the 'doming' - which are obtained from a stress analysis considering these two types of loading condition. The maximum principal stress is compared against known maximum permissible stress values, with a substantial factor of safety of 3. The doming measures the displacement of the face panel of the CRT, and must fit within known guidelines to minimize distortion of the images displayed on the screen

Techniques for the static stress analysis of engineering components have developed rapidly over the last few years, and today the two major alternatives are the Finite Element Method (FEM)[2] and Boundary Element Method (BEM)[3,4]. Both techniques formulate the equations of elasticity theory in a discretized manner, and both use a combination of numerical integration of known functions and matrix algebra to solve the system and find the required stress and displacement results.

With such numerical methods, it is well known that the accuracy of the solution is dependent on the discretization chosen to represent the component. If more elements are used the results (generally) improve. Classical engineering practice requires that a numerical analysis be considered complete only after a convergence study has been performed. That is, progressively finer meshes should be analyzed until the results of one analysis are found to be effectively the same as the results of the previous analysis. At this point, the results are said to have converged and engineers can feel more confident that their numerical solutions for stress and displacement are correct.

This paper compares the convergence characteristics of the BEM and FEM for a 27-inch cathode ray tube design. Alongside the results of the convergence themselves, the paper also presents the modeling times required to carry out the convergence study, as these are of great importance in a competitive industry. If a convergence study takes a long time, then there will be a corresponding delay in the time to market for a product, which will have a negative effect on the competitive position of the company producing the product.

It has been well documented[5,6] that the modeling times for the BEM are very favorable compared with typical FEM modeling times. A factor which is perhaps less well known is that the modeling times for remeshing a component show much more dramatic a benefit for the BEM over the FEM, as will be shown in the results of this study. Both benefits (i.e. in the initial model creation and the remeshing to define a finer mesh density) stem from two factors in the BEM code BEASY[7], which was adopted for the boundary element analysis software in this study.

a) The elements are only defined on the surface area

b) The elements may be defined in a discontinuous manner, such that the comers of one element do not need to lie in the same place as the corners of adjacent elements.

Thus, in a remeshing procedure, the user need only remesh those surfaces of interest in the convergence analysis without undue concern for the transition of the more refined mesh to other parts of the model.

The convergence analysis

The finite element and boundary element models of the CRT are shown in figures 1 and 2 The ANSYS program[8] was used for the FEM study and the BEASY program used for the BEM study.

In the convergence analysis, the models were modified to increase the number of elements used to define the thickness of the glass. In the initial models only one element was used in the thickness direction, and this was increased to two and then to three elements. It should be noted that the BEM model did contain a number of elements in the thickness direction, passing through the glass material. These elements were defined for the purpose of dividing the model into subregions, or "zones". This has the effect of making a BEM analysis more efficient and improves accuracy for slender sections.

Table 1 shows the modeling times required to make the changes from 1 to 2 and to 3 elements in the glass thickness direction for the B-EM and FEM models.

Number of elements in thickness	Analysis Type	Total number of Elements	Modeling Time	Ratio FEM/BEM
1	BEM	542	6 hours	2.7
	FEM	1152	16 hours	
2	BEM	566	5 minutes	96
	FEM	1536	8 hours	
3	BEM	590	5 minutes	96
	FEM	1920	8 hours	

Table 1. Modeling times for convergence study

The reasons for such a dramatic difference in modeling times can be attributed to the simplicity of modeling only the boundary surface area in BEM instead of the entire volume in FEM. However, there were some more features of this model which accentuated the differences. In particular, the boundary conditions (i.e. the loading values) are applied to the FEM model on an individual element basis. This required that the element numbers be known. Much of the 8 hour remeshing time was taken in finding the new element numbers and applying loads to the new elements. With the BEM model, since the loads were applied to the surfaces and then automatically mapped onto the elements on that surface, any remeshing does not destroy the boundary conditions which were previously defined when the number of elements on a surface is changed

The BEM model consisted entirely of Q38 type, quadratic quadrilateral boundary elements The FEM model consisted of a mixture of STIF 95 20 node isoparametric solid elements and STIF93

8 node isoparametric shell elements.

The maximum principal stress results for the three BEM models and for the two loading condition types are shown in Table 2. Here the column labeled 'Deviation' is the percentage error assuming the exact solution is as given by the analysis model with three elements in the thickness direction (i.e., the most refined model).

Number of elements in Thickness	Loading condition	Max. principal stress (psi)	Deviation
1	Vacuum	1452.6	0.14%
	Shrink band	1195	0.084%
2	Vacuum	1451	0.028 ?/0
	Shrink band	1194	0%
3	Vacuum	1450.6	0%
	Shrink band	1194	0%

Table 2. Maximum principal stress results for convergence study (BEM)

Table 3 shows the results of the maximum principal stress as calculated during the FEM convergence analysis study.

Number of elements in Thickness	Loading condition	Max. principal stress (psi)	Deviation
1	Vacuum	1306	11%
	Shrink band	1059	11%
2	Vacuum	1458	0.81%
	Shrink band	1186	0.75%
3	Vacuum	1470	0%
	Shrink band	1195	0%

Table 3. Maximum principal stress results for convergence study (FEM)

The maximum principal stress profiles for the finite element and boundary element solutions for the vacuum stress condition are shown in figures 3 and 4, and show a similar profile for the two types of analysis.

Finally, table 4 shows the values of doming, the displacement of the center point of the face panel, as calculated by the Finite Element and Boundary Element techniques. As can be seen, the doming values are very similar for the two techniques, and are both within experimental error bounds of the experimentally obtained value of 0.097 mm. The fact that the FEM and BEM displacement results match so closely while there is some difference in the stress results is not unexpected, since the FEM is known to give generally better displacement results than stress results, since the stress computation involves taking a derivative.

Number of elements in thickness	Doming value from BEM	Doming value from FEM
1	0.087mm	0.088mm
2	0.087mm	0.088mm
3	0.087mm	0.089mm

Table 4. Finite element and Boundary element results for doming

Conclusions

The study described in this paper shows how the boundary element and finite element methods compare in a convergence study. The major conclusions to be drawn from the analysis runs performed are as follows:

1. The boundary element results have achieved a high degree of convergence even for a single element in the thickness direction. Although it is found to be unnecessary to use a more refined mesh, it is also found that the remeshing presents no difficulty and can be done in an extremely short space of time.
2. The finite element results appear to require a minimum of two elements in the thickness direction before the results approach convergence. The difficulty of modifying the model to consider this refinement is an obstacle to a rapid convergence analysis.
3. 'Doming' displacement results do not appear to be affected greatly by the mesh refinement in the thickness direction.
4. The simplicity of producing a refined analysis model in the BEM allows a more detailed convergence study, and therefore promotes confidence in the stress and doming values obtained.

References

1. Enstrom, R.E., Stepleman, R. S., Appert, J. R., 'Application of finite element methods to the analysis of stresses in television picture tubes', pp. 665-698, RCA Review, Vol. 39, 1978.

2. Zienkiewicz, O.C. 'The Finite Element Method', 3rd edition, McGraw Hill, 19??.

3. Brebbia, C.A., Dominguez, J. 'Boundary Elements: An Introductory Course', 2nd edition, Computational Mechanics Publications & McGraw Hill, 1992.

4. Trevelyan, J. 'Boundary Elements for Engineers: Theory and application', Computational Mechanics Publications, 1994.

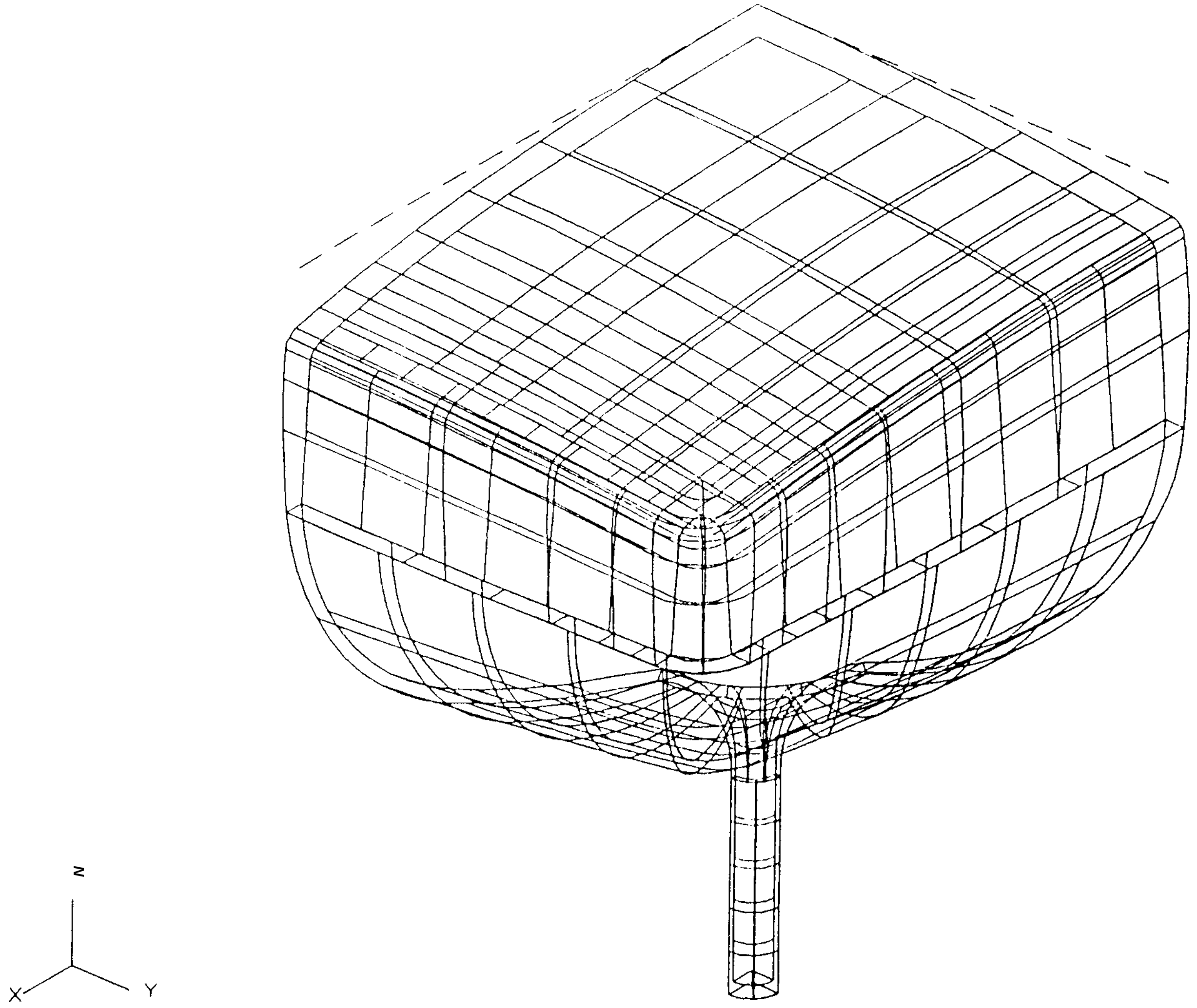
5. Wanderlingh, A. 'Impact of the boundary element method on ngineering quality and productivity', from Industrial Applications of the Boundary Element Method', eds. C. A. Brebbia and M. H. Aliabadi, Computational Mechanics Publications, 1993.

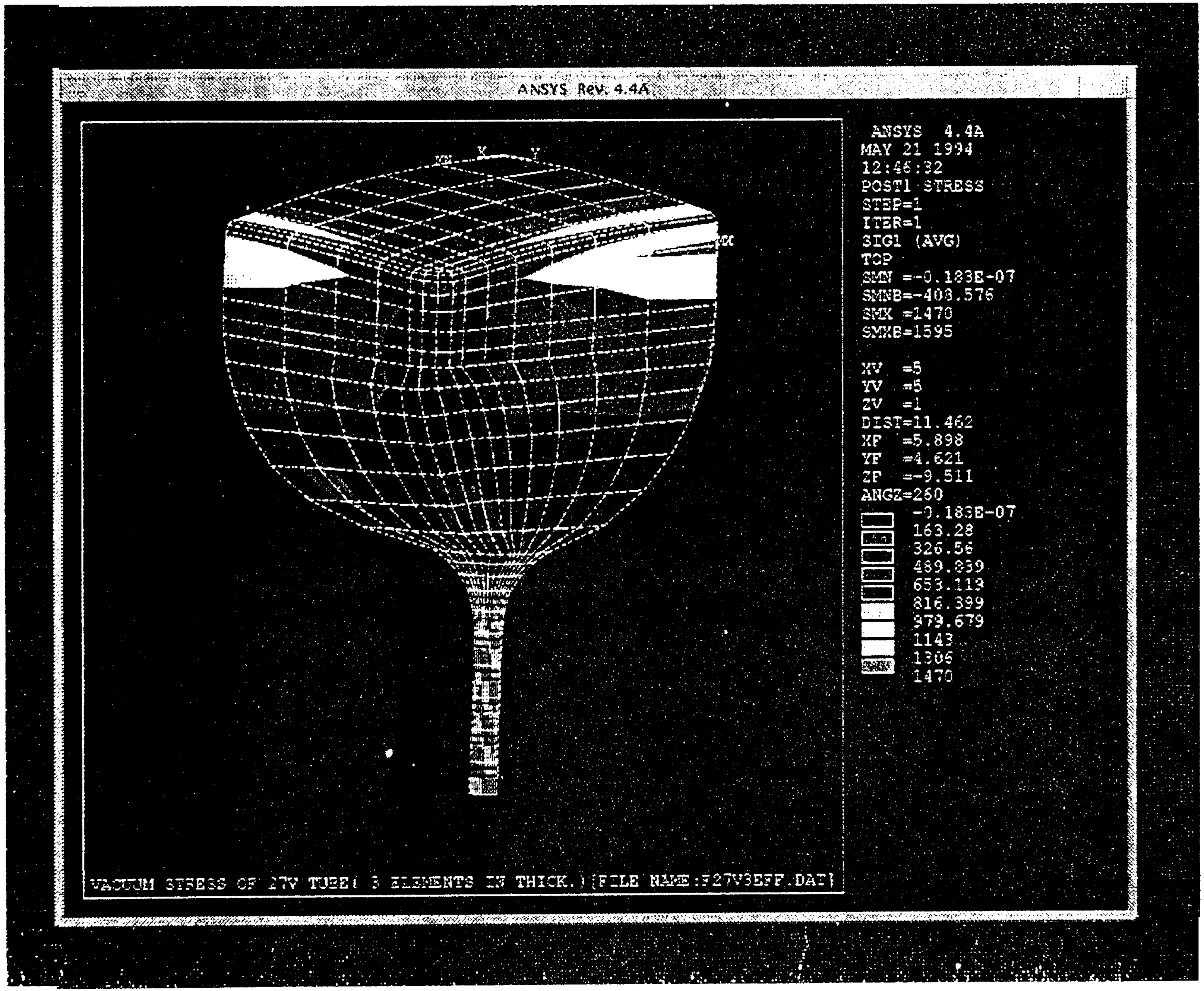
6. Cha, C. Y., Vaidyanathan, S., Trevelyan, J. 'Elastic stress analysis techniques for assessing the doming of a cathode ray tube', Proc. 15th Int. Conf on Boundary Element Methods in Engineering, Vol. 2, pp. 19-42, Computational Mechanics Publications & Elsevier Applied Science, 1993

7. BEASY. Computational Mechanics, Southampton, UK & Billerica, Massachusetts, USA

8. ANSYS. Swanson Analysis Systems, Inc., Houston, Pennsylvania, USA

Load set 1

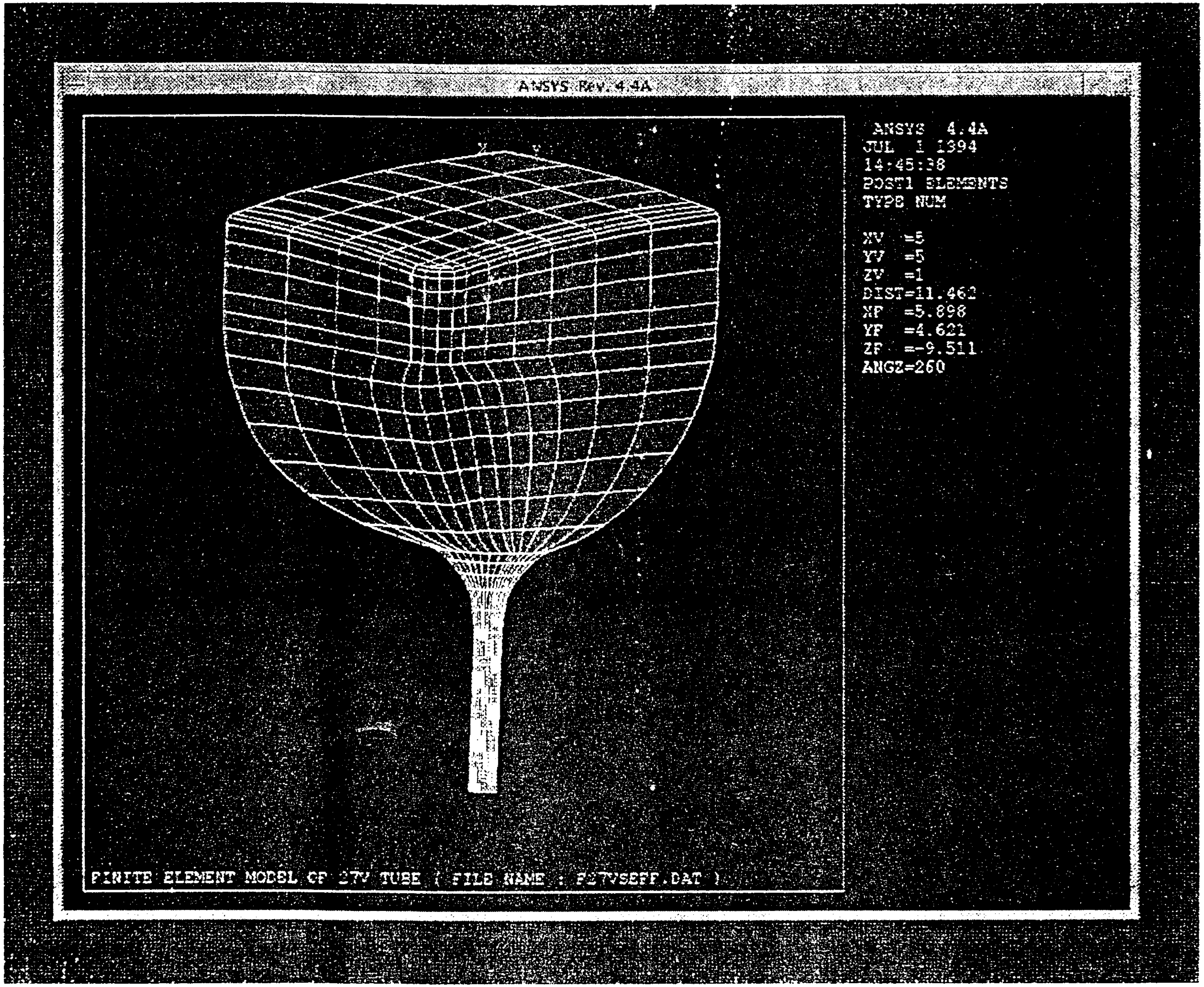




```
ANSYS  4.4A
MAY 21 1994
12:46:32
POST1 STRESS
STEP=1
ITER=1
SIG1 (AVG)
TOP
SMN =-0.183E-07
SMNB=-403.576
SMX =1470
SMXB=1595

XV  =5
YV  =5
ZV  =1
DIST=11.462
XF  =5.898
YF  =4.621
ZF  =-9.511
ANGZ=260
      -0.183E-07
▨      163.28
▨      326.56
▨      489.839
▨      653.119
▨      816.399
▨      979.679
▨      1143
▨      1306
▨      1470
```

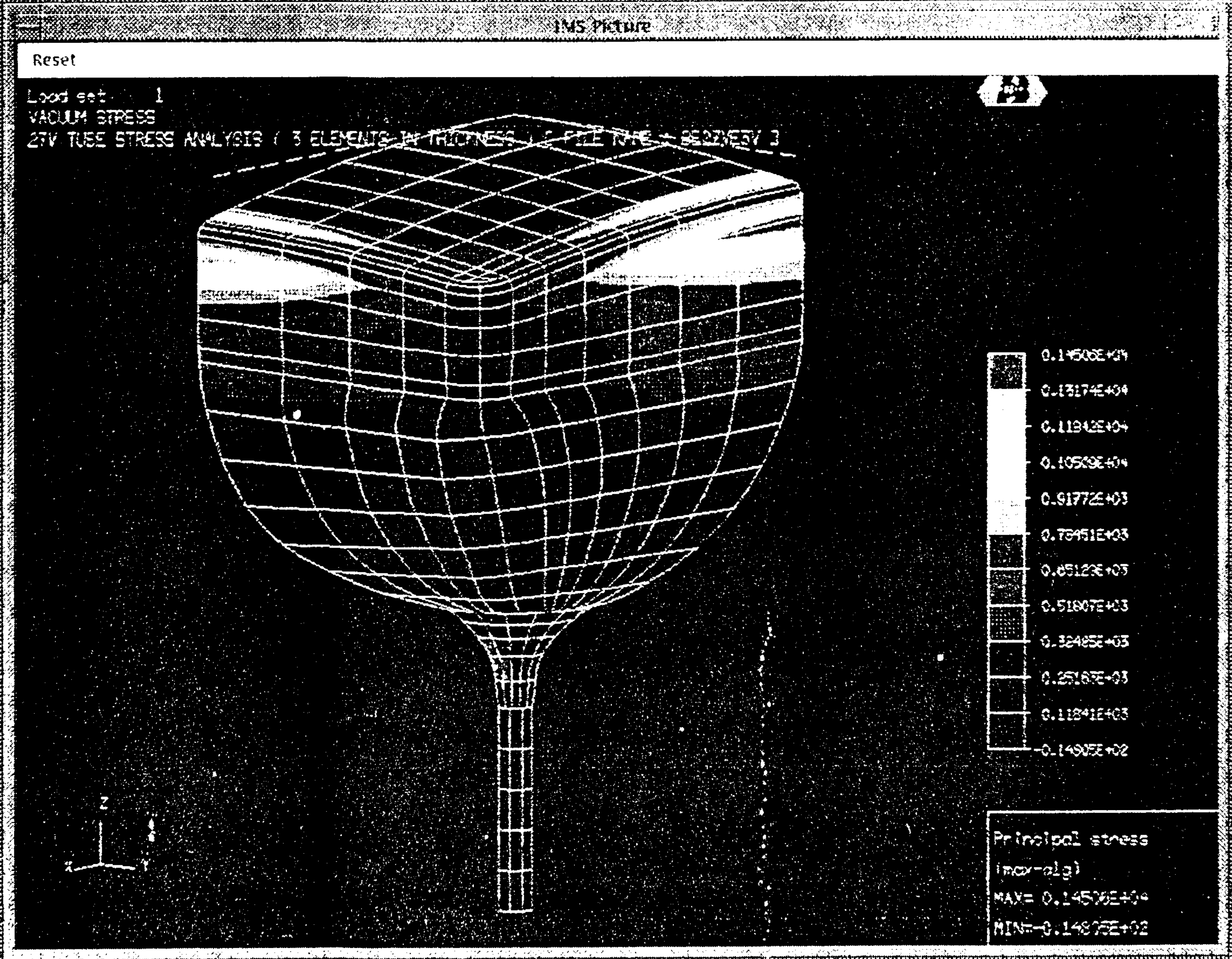
```
VACUUM STRESS OF 27V TUBE( 3 ELEMENTS IN THICK.)(FILE NAME:F27V3EFF.DAT)
```



```
 ANSYS  4.4A
JUL  1 1394
14:45:38
POST1 ELEMENTS
TYPE NUM

XV  =5
YV  =5
ZV  =1
DIST=11.462
XF  =5.898
YF  =4.621
ZF  =-9.511
ANGZ=260
```

FINITE ELEMENT MODEL OF 27V TUBE (FILE NAME : F27VSEFF.DAT)



```
Load set   1
VACUUM STRESS
27V TUBE STRESS ANALYSIS ( 3 ELEMENTS IN THICKNESS ) [illegible]
```

```
Principal stress
(max-alg)
MAX= 0.14506E+04
MIN=-0.14805E+02
```