EVALUATION OF MSC'S PARABOLIC TETRAHEDRON FINITE ELEMENT

Kevin E. Arden

Newport News Shipbuilding
SEAWOLF Hull Engineering
4101 Washington Avenue
Newport News, VA 23607

ABSTRACT

Automatic mesh generators, based on the solid model geometry, will only produce tetrahedron elements. Solids based modeling techniques allow for efficient and easy creation of solid element models. This paper evaluates the accuracy of the element by comparing closed form solutions and actual test data to solutions calculated using MSC/NASTRAN.

The test data resulted from a recent design experience where solid modeling capabilities were beneficial. This design experience is presented along with the test validation of the finite element results.

The modeler used herein is I-DEAS VI.1 provided by SDRC.

The opinions of this paper are the author's and do not necessarily represent the opinions of Newport News Shipbuilding.

Table of Contents

1.0 Introduction

The purpose of this paper is to investigate the accuracy of the parabolic tetrahedron element. Closed form solutions will be used to test different aspects of the element. A patch test will be used to test membrane stresses on irregularly shaped elements and a cantilever beam will be used to test their bending and shear stress calculations.

A complex model comprised of tetrahedron elements and solved in MSC/NASTRAN will be compared to a physical test as a final validation of the element's performance.

2.0 Background

A stress evaluation of the box casting shown in Figure 1 was required as part of a design of a gyrofin stabilizer.

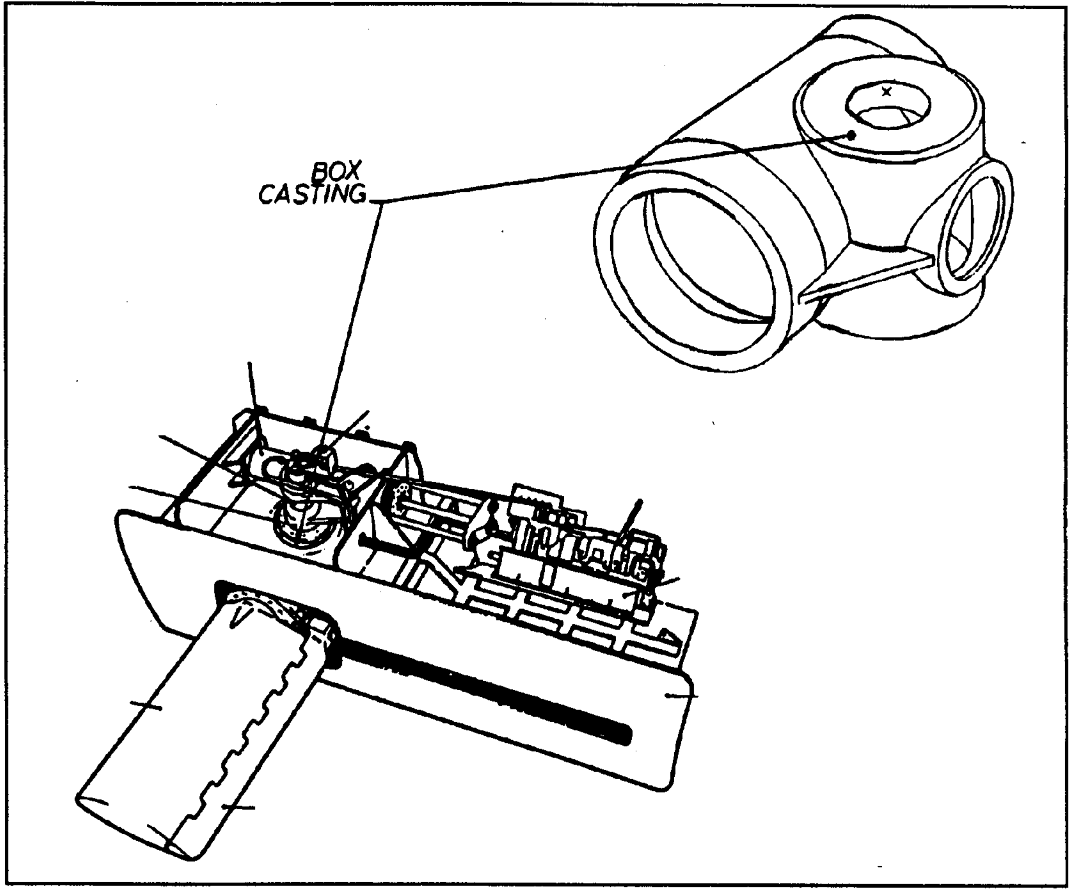


Figure 1
Gyrofin Stabilizer

Several proposed box casting configurations were to be evaluated. Due to the complex geometry and the different configurations required to analyze, automatic mesh generation on solid model geometries proved to be a beneficial time saver. Figures 2 through 5 show finite element models of four different configurations that were evaluated.

It was important to validate the finite element results. First several smaller models were compared to closed form solutions in order to establish the parabolic tetrahedron's performance. Then, one of the configurations was tested to validate the finite element model results.

Automatic mesh generation of solid elements is a beneficial model creation tool, but useless for design if the element's accuracy is in question. To quote Reference (a):

> "Nothing is as important to the success of a finite element analysis as the accuracy of the elements. Indeed, in a linear static analysis, the finite elements embody all of the discretizing assumptions, the rest of the calculations are exact except for the lack of precision."

The work presented in this report, evaluates accuracy of the parabolic tetrahedron element in MSC/NASTRAN.

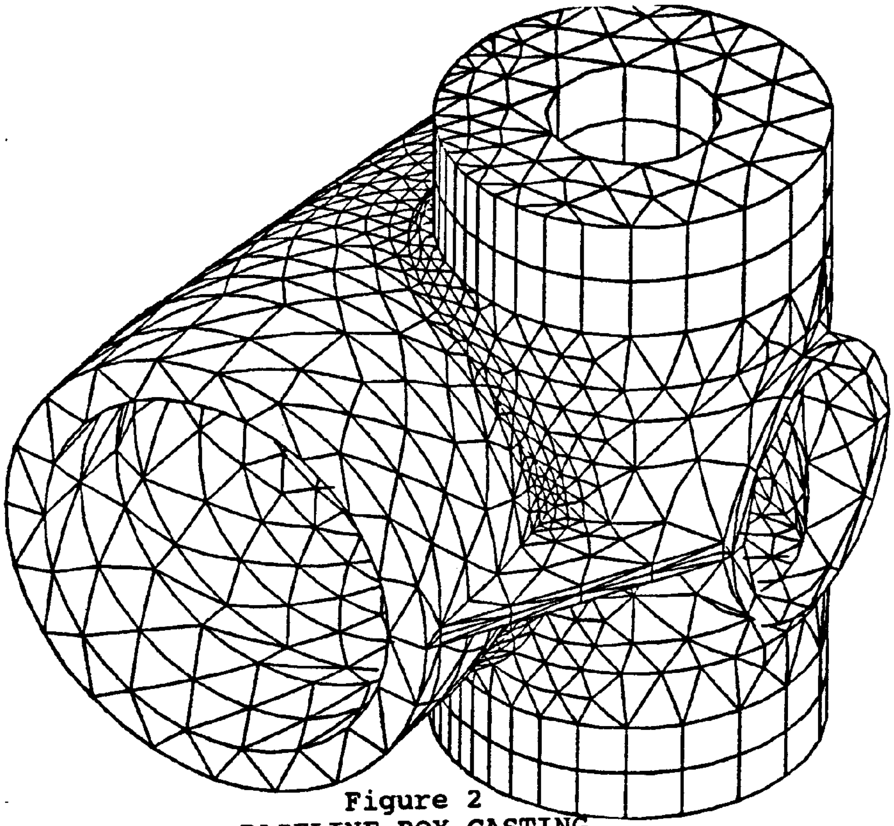


Figure 2
BASELINE BOX CASTING

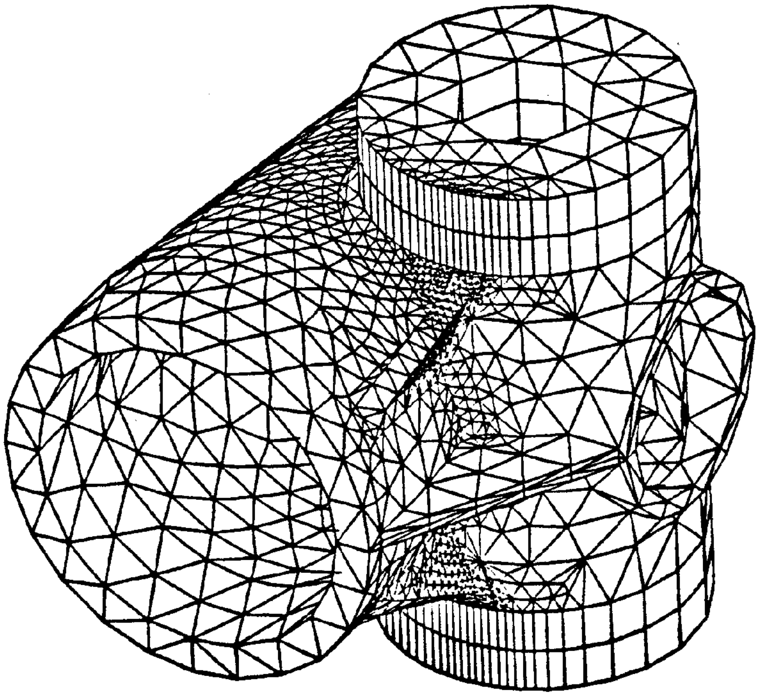


Figure 3
BOX CASTING WITH ONE CHOCK (TOP & BOTTOM)

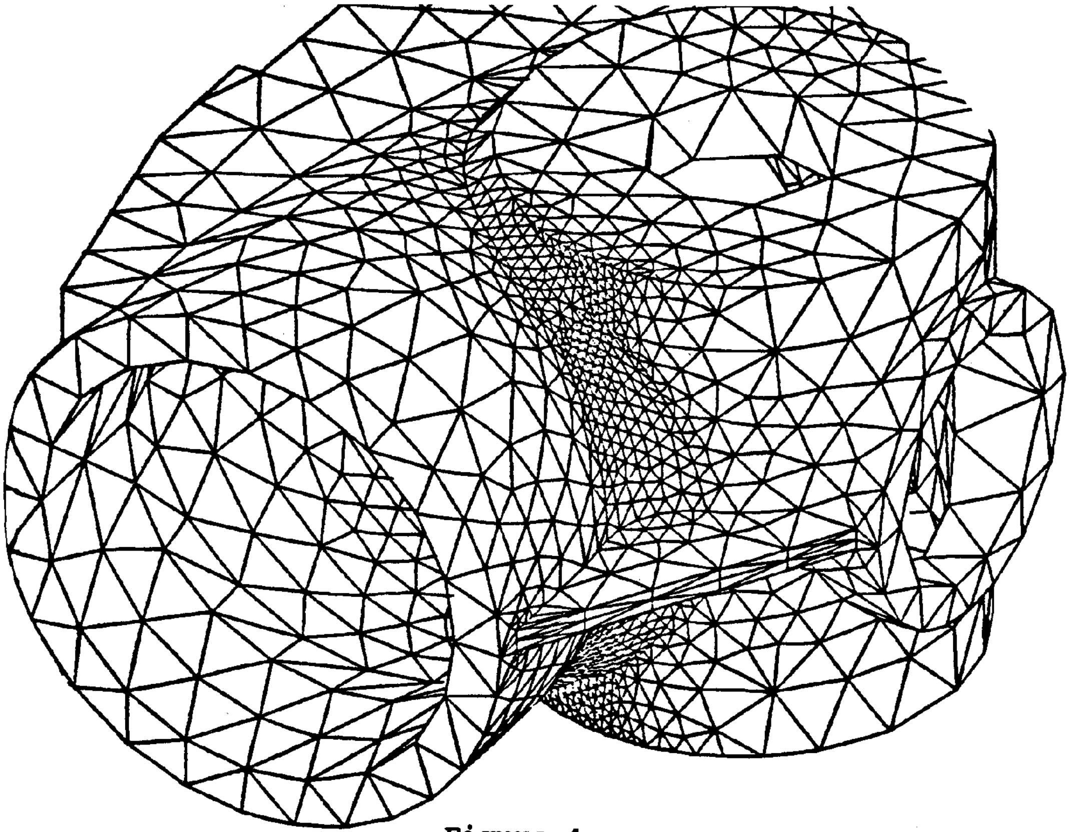


Figure 4
BOX CASTING WITH MODIFIED FILLETS

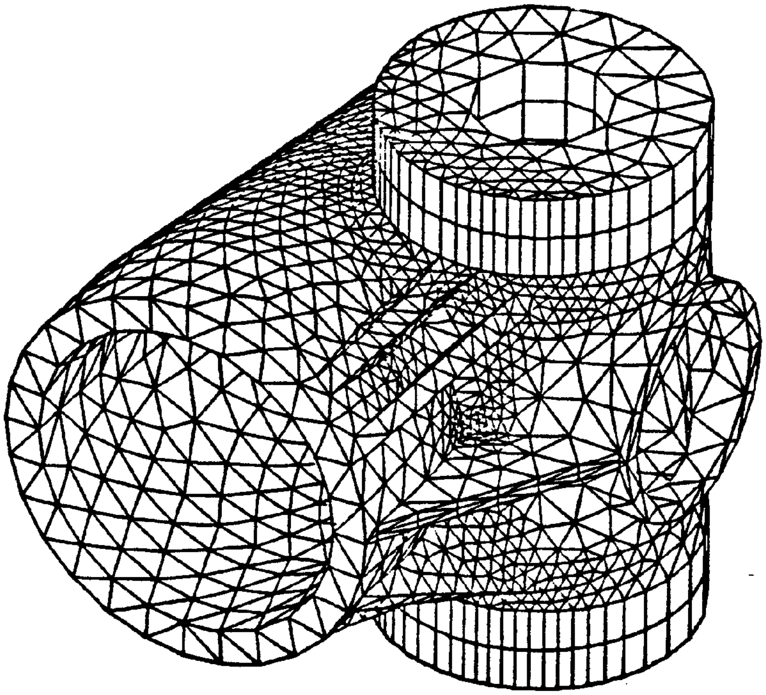


Figure 5
BOX CASTING WITH TWO CHOCKS (TOP & BOTTOM)

3.0 Element Test Problems

According to Reference (a), a comprehensive set of element test problems should take into account the parameters which affect accuracy. These parameters are; loading, element geometry, problem geometry, and material properties.

With regarding to loading, this paper will test extension, in-plane shear, and bending of the element. Additionally, The element's sensitivity to distortion will be tested by using irregular shapes in the geometry. This paper will not attempt to produce a set of rules or boundaries for how much distortion is allowed. Most pre-processors have quality checks and tetrahedron fix routines that can achieve a reasonable mesh. Therefore, this paper will use irregular (i.e not perfect) shaped elements, but within a "reasonable" distortion limit (0.500 distortion using I-DEAS VI.1 modeler).

Geometry will be varied throughout the tests. Geometric parameters such as curvature can effect element accuracy not isolated to the shape of each individual element. Different material properties will not be tested herein. All tests were performed using ordinary steel.

4.0 <u>The Patch Test</u>

The patch test consisted of a unit block subjected to membrane stresses. Elements used to model the block were transitioned from larger to smaller elements, so as to produce irregularly shaped elements in the interior of the unit block.

The patch test was used in Reference (a) to test HEXA brick elements and is based on work performed in References (b) and (c). The arbitrarily irregular shapes were essential to the test. The rectangular exterior shape made it easy to provide boundary conditions corresponding to constant membrane stress, independent of element shape.

The patch test is important because it proves the element's accuracy for any problem if enough elements are used. The reason is that the stress within each element will be a uniform membrane stress as the element's size converges to zero. Reference (a) states that an element that can not pass the patch test should not be trusted. However, Reference (a) notes that passing the patch test does not guarantee satisfaction since the rate of convergence may be too slow for practical use.

The results of the CTETRA (10 noded) finite element model compared to the closed form solution (provide in Reference (a)) are shown in Table 4.1.

Table 4.1
The Patch Test on a Parabolic Tetrahedron
Finite Element used in MSC/NASTRAN (CTETRA)

Stress Type	Results	
	FEM	Closed Form
X-Normal	2000.9	2000
Y-Normal	2000.7	2000
Z-Normal	2001	2000
Shear	400.4	400

The results of the patch tests show that slightly irregular elements are acceptable for pure membrane load. Therefore, the element will converge to an accurate solution for all other cases.

5.0 The Cantilever Beam Test

The straight cantilever beam is a frequently used test problem. Reference (a) uses this beam to test the HEXA brick element. The CTETRA will be tested using a mesh density similar to the HEXA test model in Reference (a).

The simplicity of the closed form solution, and the fact that linearly varying strains can be imposed through the application of end loads, makes this a good element test to perform.

Four cases were analyzed: Regular shaped elements subjected to bending with shear, and a second load case of pure torsion. The second two cases are repeats of the same loadings on irregular shaped elements.

The results of the straight beam tests are shown in Table 5.1.

Table 5.1
The Straight Beam Test on a Parabolic Tetrahedron
Finite Element used in MSC/NASTRAN (CTETRA)

Element Shape	Load Case	Closed Form Deflection	FEM Deflection	HEXA(8) Results
Regular	Bending (In-plane shear)	0.1081	0.1040 (-3.8%)	0.1060 (-1.9%)
	Torsion (Twist)	0.003668	0.003303 (-9.5%)	0.003379 (-9.0%)
Irregular	Bending (In-plane shear)	0.1081	0.075634 (-30.0%)	0.007466 (-93.1%)
	Torsion (Twist)	0.003668	0.003667 (-0.03%)	0.003323 (-9.4%)

These results, when compared to the HEXA results in Reference (a), show that the parabolic CTETRA(10) element is comparable to the HEXA(8) brick element. The CTETRA(10) exhibits the same problem as the HEXA(8); irregularly shaped elements subjected to in-plane loads.

6.0 An Actual Physical Test

To validate the finite element approach for the actual design work being performed, it was decided to test an actual physical structure. On of the actual castings was subjected to a static load and strain gage measurements were made in the crease between the two cylinders. Figures 6 through 9 show a comparison of the measured stresses with the calculated stresses.

A finite element model of the same casing was developed and analyzed using MSC/NASTRAN CTETRA (10 noded) elements. This model was automatically generated from solid model geometry using a single free mesh volume.

These tests were important because they confirmed the mathematical model. The comparison of the tests and the model verified the following:

- Model Refinement
- Shapes of Elements and Element Types
- Convergence Accuracy
- Correct Interpretation of Results
- Confidence level

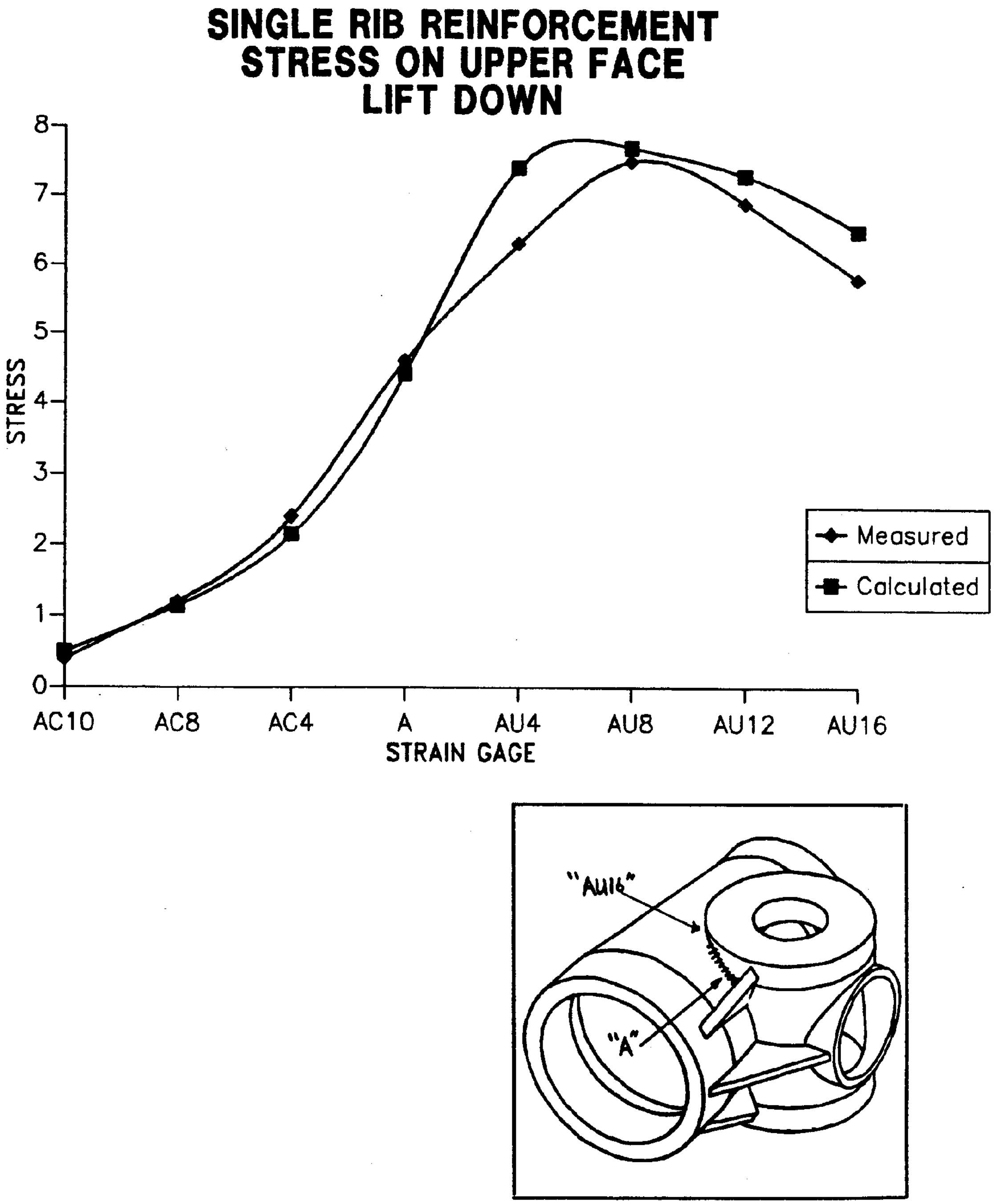

SINGLE RIB REINFORCEMENT
STRESS ON UPPER FACE
LIFT DOWN
STRESS
0
1
2
3
4
5
6
7
8
STRAIN GAGE
AC10
AC8
AC4
A
AU4
AU8
AU12
AU16
Measured
Calculated
"AU16"
"A"

Figure 6

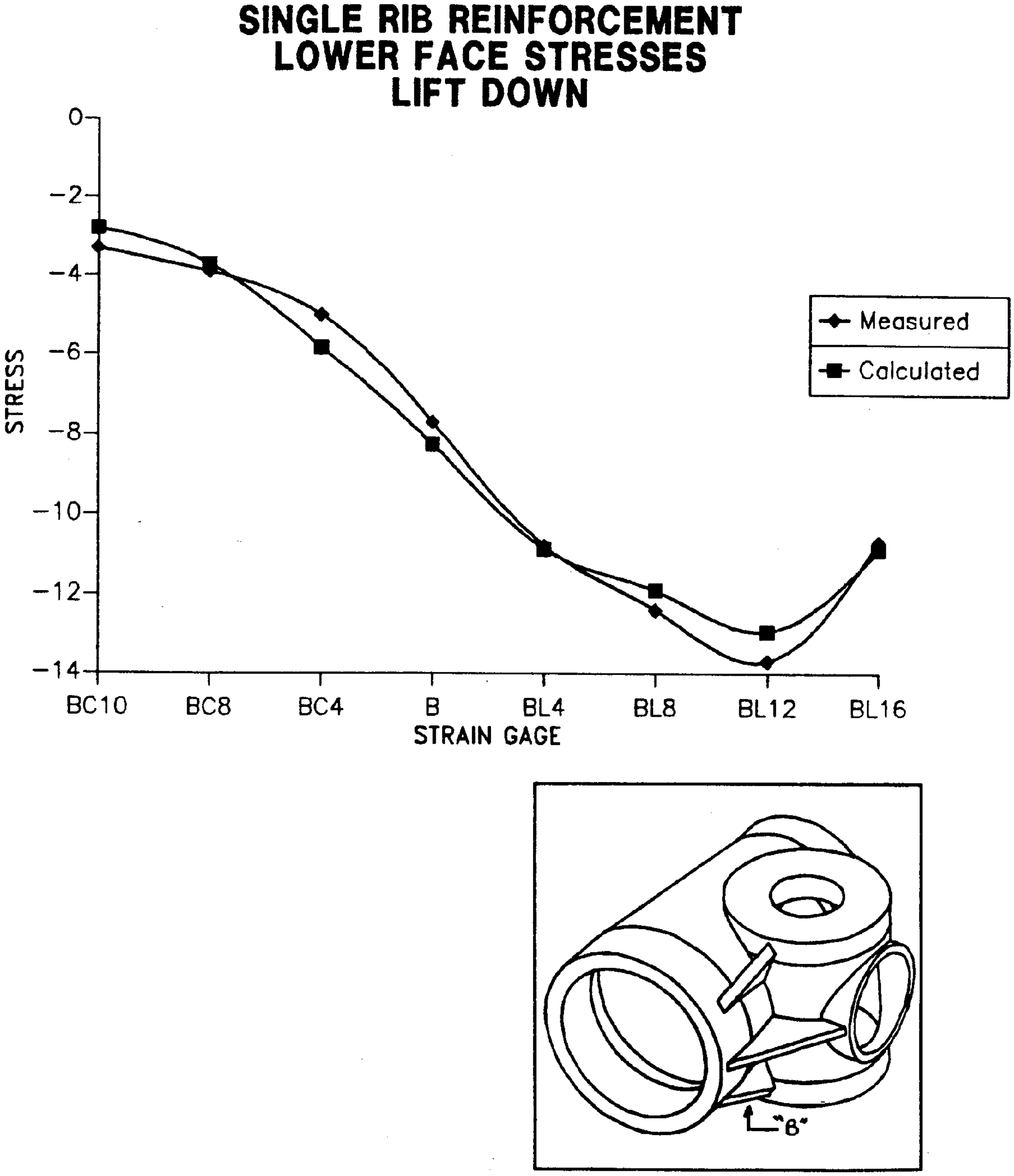
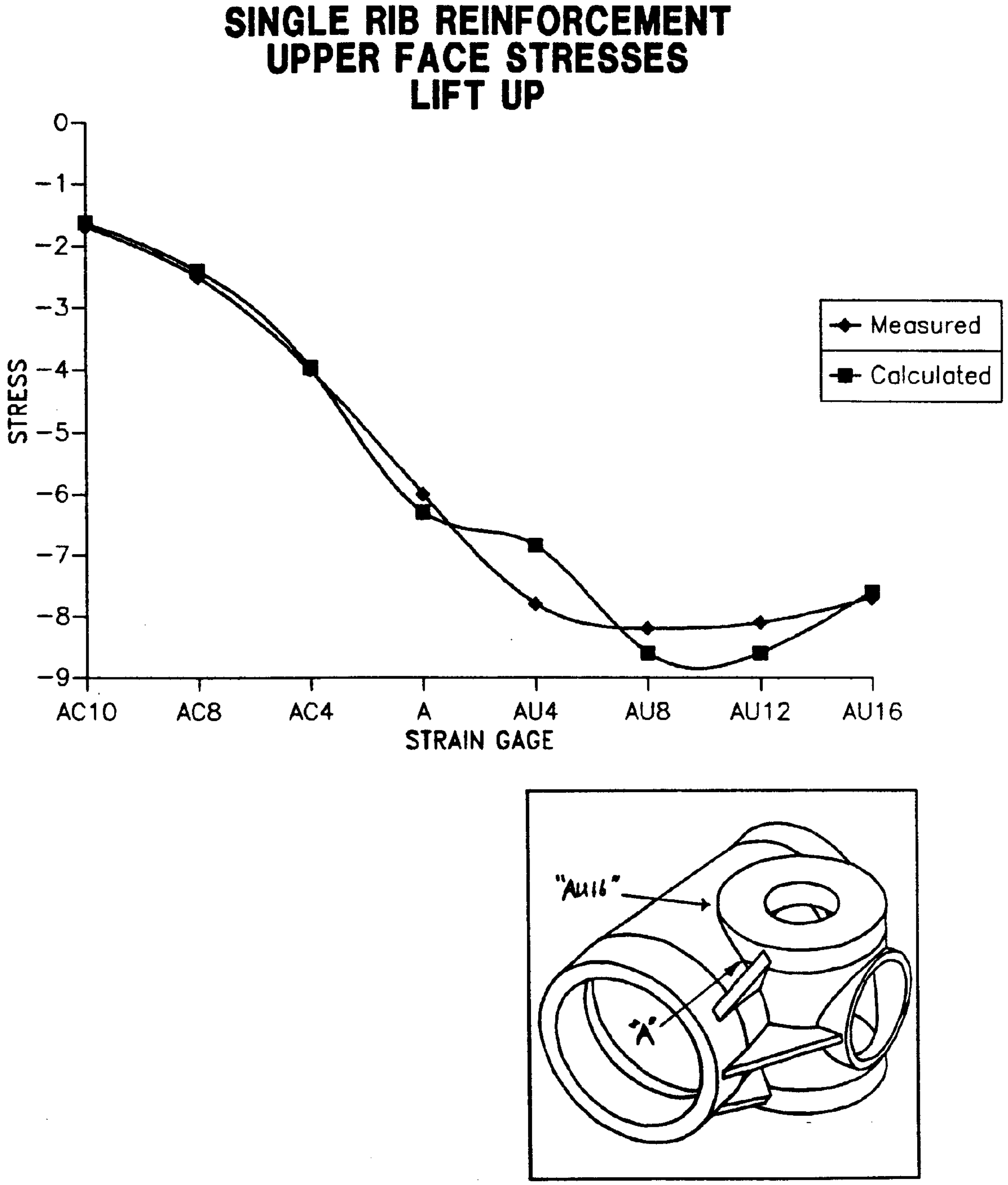

SINGLE RIB REINFORCEMENT
LOWER FACE STRESSES
LIFT DOWN
STRESS
0
-2
-4
-6
-8
-10
-12
-14
BC10
BC8
BC4
B
BL4
BL8
BL12
BL16
STRAIN GAGE
Measured
Calculated
"B"

Figure 7



SINGLE RIB REINFORCEMENT
UPPER FACE STRESSES
LIFT UP

Figure 8

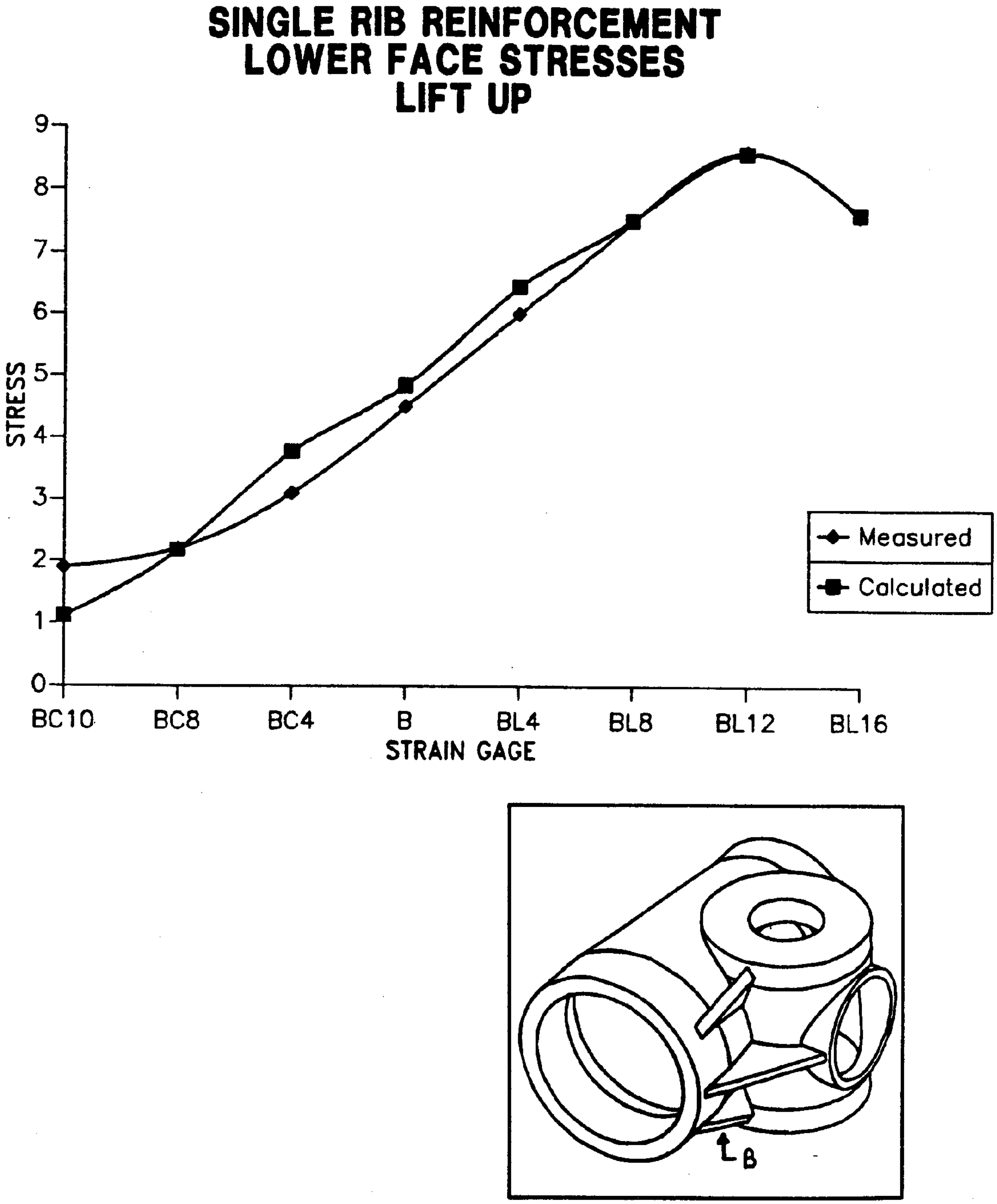

SINGLE RIB REINFORCEMENT
LOWER FACE STRESSES
LIFT UP
STRESS
0
1
2
3
4
5
6
7
8
9
BC10
BC8
BC4
B
BL4
BL8
BL12
BL16
STRAIN GAGE
Measured
Calculated
t_B

Figure 9

7.0 Conclusions and Recommendations

Based on the results of the patch test using irregular shaped elements, it can be concluded that it is possible to get convergence to acceptable answers with the CTETRA parabolic element. However, it appears that the CTETRA is comparable to the HEXA(8) brick element.

Results of the straight beam tests indicate that the element exhibits locking (gains stiffness) when irregular shapes are subjected to in-plane shear (-30% on deflection). The patch tests show that convergence is possible for all cases, and by comparing the beam results to the brick beam results in Reference (a), it can be seen that convergence for parabolic CTETRAs is similar to linear bricks.

Based on the comparison to actual tests it does appear that acceptable results can be obtained if the following factors are accounted for:

1) Regularly shaped elements are important in the area of concern. Densities similar to those used for linear brick elements are appropriate. A 0.50 limit on distortion is suggested as a lower bound.

2) Mid-side nodes should be kept centered between nodes.

The tests performed herein demonstrate that good answers are possible with the CTETRA parabolic element in MSC/NASTRAN. However, it is advisable to pay extra concern to element shapes and densities. Most free mesh generators provide element fixes intended to increase the quality of the generated element's shape.

The use of automatic free mesh generators save time when creating complex solid models, and MSC/NASTRAN's parabolic CTETRA provides a good element solution when used properly.

REFERENCES

(a) MacNeal, Richard H. and Robert L. Harder, A Proposed Standard Set of Problems to Test Finite Element Accuracy, March 1984.

(b) Robison J. and S. Blackham, "An Evaluation of Lower Order Membranes as Contained in MSC/NASTRAN, ASAS, and PAFEC FEM Systems," September 1979.

(c) Robison J. and S. Blackham, "An Evaluation of Plate Bending Elements: MSC/NASTRAN, ASAS, PAFEC, ANSYS, and SAP4," August 1981.