STRESS CONCENTRATION INVESTIGATIONS USING NASTRAN †

M. C. Gillcrist and L. A. Parnell
Naval Ocean Systems Center, San Diego, CA

ABSTRACT

Parametric investigations are performed using several two-dimensional finite element formulations to determine their suitability for use in predicting extremum stresses in marine propellers. Comparisons are made of two NASTRAN elements (CTRIM6 and CTRIA2) wherein elasticity properties have been modified to yield plane strain results. The accuracy of the elements is investigated by comparing finite element stress predictions with experimentally determined stresses in two classical cases: (1) tension in a flat plate with a circular hole; and (2) a filleted flat bar subjected to in-plane bending. The CTRIA2 element is found to provide good results. The displacement field from a three-dimensional finite element model of a representative marine propeller is used as the boundary condition for the two-dimensional plane strain investigations of stresses in the propeller blade and fillet. Stress predictions from the three-dimensional analysis are compared with those from the two-dimensional models. The validity of the plane strain modifications to the NASTRAN element is checked by comparing the modified CTRIA2 element stress predictions with those of the ABAQUS plane strain element, CPE4.

INTRODUCTION

It is common practice in stress analyses of marine propellers to create a three-dimensional (3-D) finite element model of the blade without hub or fillet and apply a rigid boundary condition at the blade-hub interface. The stresses nearest the hub are normally the largest. For this reason it is important to know what influence the absence of the hub and fillet have on the predicted stresses.

One approach to this problem is to perform a full 3-D finite element analysis of the blade, hub and fillet, a costly and time consuming computational effort. An alternate approach, used at the David Taylor Naval Ship Research and Development Center (DTNSRDC), is to perform a 3-D finite element analysis of the clamped blade model (i.e., without hub or fillet) and apply the computed displacements as a boundary condition for a two-dimensional (2-D) plane strain model of the blade with hub and fillet. This paper takes the DTNSRDC approach to investigate stress concentrations in the fillet region of a marine propeller blade.

ELEMENT SELECTION

The first step in the finite element analysis is to choose appropriate elements from NASTRAN's element library (ref. 1,2). The 20-node hexahedron, CIHEX2, is the obvious choice for the 3-D portion of the analysis. For the 2-D plane strain analysis, the 6-noded and 3-noded triangular plate elements, CTRIM6 and CTRIA2 respectively, were selected as probable candidates because of the ease with which triangular elements can be meshed to irregular geometries.

---

† This work was supported by the Naval Sea Systems Command (NAVSEA 63R5).

The formulation of the triangular plate elements in NASTRAN is based on the assumption of 'plane stress'. A 'plane strain' solution can be obtained however, by modifying the elasticity coefficients as described by Schaeffer (ref. 3). The validity of the plane strain modifications to NASTRAN is checked by comparing the stress predictions of the modified NASTRAN element with those of the ABAQUS plane strain element, CPE4 (ref. 4).

VERIFICATION OF ELEMENT ACCURACY

Success in performing numerical stress calculations by the finite element method depends on the choice of element and the layout of the element mesh. When properly formulated, the finite element solution should converge to the exact analytical solution if progressively finer meshes are used. For the circular hole and fillet geometries investigated in this paper, mesh size can be represented by the dimensionless ratio, l/r, where l is the length of an element (in the region of interest) and r is the radius of the circular hole or fillet. Laura, Reyes and Rossi (ref. 5) tried to assess the accuracy of the constant strain triangular element in regions of high stress concentration. Finite element predicted stress concentration factors were compared with photoelastically determined values at the boundary of a slot in a plate subjected to uniaxial stress. For element meshes with l/r equal to 0.13, the difference between numerical and experimental results was on the order of 10%.

In order to gauge the accuracy of the NASTRAN triangular elements for a given mesh size, comparisons are made between stress results obtained by the finite element method and those obtained by experimental or other analytical techniques. Two stress concentration cases are selected for the comparisons: (1) a flat rectangular plate with a small circular hole subjected to a uniform tension, σ , in the x-direction; and (2) a filleted bar subjected to in-plane bending. For the case of an infinite plate with a circular hole, Timoshenko (ref. 6) found that the stress in the x-direction is 3σ at the edge of the hole and quickly drops to σ away from the hole. For a finite plate with width no less than 4 times the hole diameter, Timoshenko found that the stresses produced should be within 6% of those produced in the infinite plate, that is, between 2.82σ and 3σ.

For the sake of comparison, a finite element analysis was performed on a flat square plate with the ratio of plate width to hole diameter of approximately 12 to 1. The applied stress is σ and the theoretical value of σ_x at the hole edge should be between 2.82σ and 3σ . Results for the NASTRAN CTRIA2 element are presented in table 1. Mesh fineness in the region of interest is represented in terms of the nondimensional ratio, l/r, where l is the length of an element and r is the radius of the circular hole. Table 1 shows that predicted stresses using the CTRIA2 element are within 6% of the theoretical value when mesh size values, l/r, of less than 40% are used. The CTRIM6 element yields surprisingly poor results. Nodal stresses at the edge of the hole range from 1.3σ to 4.3σ for very fine meshes (l/r less than 30%). Averaged nodal stresses yield better results; however, due to the disparity in nodal stresses from contributing elements, there is little confidence in the CTRIM6 predictions. This element is given no further consideration.

Hartman and Leven (ref. 7) used photoelastic techniques to determine stress concentrations in filleted bars subjected to in-plane bending as illustrated in figure 1. Stress concentrations of 1.2 to 3.0 were obtained for r/d values ranging from 0.03 to 0.50, where r is the fillet radius and d is the depth of the bar. The stress concentration factor, k, is defined as the ratio of the maximum stress at the fillet to the nominal stress computed by the flexure formula: $\sigma_{nom} = Mc/I$, where $c = d/2$, I = area moment of inertia and M = applied bending moment. Table 2 presents NASTRAN CTRIA2 stress predictions in the fillet region of a flat bar identical to one tested by Hartman and Leven. For this particular case ($r/d = 0.2$), Hartman and Leven determined the stress concentration to be k = 1.53. The NASTRAN predicted k value of 1.58 presented in table 2 is in excellent agreement with this, differing by only 3.3% from the experimentally determined value.

THE 3-D DISPLACEMENT FIELD AS A BOUNDARY CONDITION

The 2-D plane strain finite element analysis of the blade with hub and fillet is predicated on the hypothesis that the displacement field predicted by the 3-D finite element model can be applied as a boundary condition to the 2-D model. This technique, which is supported by unpublished numerical experiments, is illustrated by the case of a thick plate in bending. A thick rectangular plate lying in the x-z plane is subjected to out-of-plane bending by the application of a moment along one edge while the opposite edge is fixed; that is, the plate is cantilevered and a moment applied to the free end as illustrated in figure 2. Using sixteen CIHEX2 isoparametric brick elements, a 3-D finite element model is constructed and solved for displacements and stresses. A 2-D model in the x-y plane is created using CTRIA2 elements. Displacements computed by the 3-D model at nodes 5 through 7 are applied as a displacement boundary condition to the corresponding nodes in the 2-D model. Stresses are then computed using the 2-D model. Table 3 presents stress predictions from both models at nodes 1, 2, 3 and 4 along the top surface of the plate. With the exception of node 1 located on the rigid boundary, stress predictions from the two models differ by less than 2%. Similar results are obtained for the bottom surface of the plate. Since the 3-D model composed of isoparametric brick elements can be expected to yield reasonable stress predictions at locations away from the boundary, these results support the assertion that the 3-D displacement predictions can be applied as a boundary condition to the 2-D model in order to obtain reasonable stress predictions.

PROPELLER MODELS

A 3-D and 2-D finite element propeller blade model are constructed for the purpose of analyzing the stresses in the blade root region. The 3-D blade model without hub or fillet consists of 40 CIHEX2 brick elements and is presented in figure 3. Figure 4 shows a 2-D model of a blade, hub and fillet cross section composed of 243 CTRIA2 elements. Maximum stress is usually developed at the base of the blade in the midchord region. For this reason, the 2-D analysis is based on a planar slice through the blade, hub and fillet in the vicinity of the midchord. The plane of interest is illustrated in figure 3 by superposition of a 2-D model on the 3-D model. R is the propeller radius measured from the center of the hub. Nodes 1, 2 and 3, located at approximately 0.36R, correspond to nodal points on both the 3-D and 2-D finite element models. Hydrodynamic and centrifugal loads are applied to the 3-D finite element model in order to predict the displacements at nodes 1, 2 and 3. These displacements are then applied to corresponding nodes on the 2-D planar model as a displacement boundary condition, ensuring that all out-of-plane degrees-of-freedom are constrained.

Several 2-D plane strain propeller models are analyzed in order to investigate the sensitivity of root stresses to fillet radius. The propeller under consideration has a fillet of constant radius, r, which is approximately one half the blade thickness, d. Table 4 presents stress predictions from both NASTRAN and ABAQUS 2-D analyses of the blade with hub and fillet. The extremum stresses (those of greatest absolute value) are the compression stresses developed in the fillet on the low-pressure face of the blade (see figure 4). The stress values presented in the table vary less than 4% over a range in r/d values from 0.37 to 0.56. The 3-D NASTRAN analysis of the clamped blade without hub or fillet predicts a maximum stress value within 3% of the 2-D NASTRAN predictions and within 4% of the 2-D ABAQUS prediction. This technique of combining the 2-D and 3-D finite element analyses to predict fillet stresses is predicated on the assumption that the plane strain condition exists in the region of the 2-D planar slice. It is reassuring to note that the 3-D analysis predicts out-of-plane displacements an order of magnitude smaller than the in-plane displacements. This result lends credence to the assumption of plane strain.

SUMMARY AND CONCLUSIONS

Parametric investigations were performed using several two-dimensional finite element formulations to determine their suitability for use in predicting root stresses in marine propellers. Comparisons were made of two NASTRAN elements, CTRIM6 and CTRIA2, wherein elasticity properties were modified to yield plane strain results. The accuracy of the elements was investigated by comparing stress results obtained by the finite element method with those obtained by experimental or analytical techniques for two stress concentration cases: (1) a flat rectangular plate with a small circular hole subjected to uniform tension, and (2) a filleted bar subjected to in-plane bending. For both cases, the CTRIA2 element was found to provide stress predictions within about 5% of the expected value (experimental or theoretical) so long as the mesh size parameter, l/r, did not exceed about 15%. Furthermore, it was discovered that when the CTRIM6 element was used, the solution did not appear to converge. The plane strain modification to the NASTRAN element was checked by comparison with the ABAQUS plane strain element, CPE4. NASTRAN and ABAQUS 2-D plane strain analyses were performed on a propeller model yielding stress predictions which differed by only a few percent.

A combined 2-D and 3-D analysis of a thick plate in bending demonstrated the validity of applying predicted displacements from the 3-D analysis as a displacement boundary condition to the 2-D model in order to predict stresses in the plate. This technique was then applied to the analysis of root stresses in a marine propeller blade. It was found that extremum stresses develop in the fillet on the compression face of the blade and that these stresses are rather insensitive to small changes in the fillet radius. Furthermore, it was demonstrated that there is no significant difference in the extremum stresses predicted by the 3-D clamped blade analysis and the 2-D blade-hub-fillet analysis. Although the close proximity of the fixed displacement boundary condition in the 3-D model may distort the stress field in the root region, it does not appear to adversely affect the prediction of extremum stress.

REFERENCES

1. The NASTRAN Users Manual, Vol. 1, NASA Scientific and Technical Information Office, Washington, D.C., June 1985.

2. The NASTRAN Theoretical Manual, NASA Scientific and Technical Information Office, Washington, D.C., Jan 1981.

3. Schaeffer, H.G., MSC/NASTRAN Primer: Static and Normal Modes Analysis, Wallace Press, Inc., Milford, New Hampshire, Jan. 1982, pp 271-272.

4. ABAQUS User's Manual, Hibbit, Karlsson and Sorensen, Inc., May 1984.

5. Laura P.A., Reyes J.A. and Rossi R.E., Numerical Experiments on the Determination of Stress Concentration Factors, Strain, vol. 10, no. 2, Apr. 1974, pp 58-63.

6. Timoshenko, S.P. and Goodier, J.N., Theory of Elasticity, 3rd ed., McGraw-Hill, New York 1970, pp 90-92.

7. Hartman, J.B. and Leven, M.M., Factors of Stress Concentration for the Bending Cases of Fillets in Flat Bars and Shafts with Central Enlarged Section, Proceedings of the Society for Experimental Stress Analysis, vol. 9, no. 1, 1951, pp 53-62.

TABLE 1. STRESS AT THE EDGE OF A SMALL CIRCULAR HOLE IN A SQUARE FLAT PLATE SUBJECTED TO TENSION σ, IN THE x-DIRECTION.

Mesh size l/r	Stress† σ_x	Percent difference from infinite plate value
0.64	2.42σ	19.3
0.36	2.86σ	4.7
0.26	2.84σ	5.3
0.22	2.86σ	4.7

† NASTRAN predictions using triangular plate element CTRIA2.

l = element length
r = radius of hole
σ = applied stress
σ_x = stress in x-direction at the centroid of an element located at edge of hole

TABLE 2. STRESS CONCENTRATION IN A FILLETED BAR SUBJECTED TO IN-PLANE BENDING.

Mesh size l/r	Stress Concentration † k	Percent difference from experimental value*
0.140	1.46	4.9
0.025	1.58	3.3

† NASTRAN predictions using triangular plate element CTRIA2.

l = element length
r = fillet radius
* Experimental value = 1.53 (see reference 7).

TABLE 3. PLATE STRESSES PREDICTED BY NASTRAN 2-D AND 3-D MODELS

Node Number	Model	Dimensionless Stress*	Percent Difference
1	3D	1.221	14.0
	2D	1.392	
2	3D	1.106	0.8
	2D	1.096	
3	3D	1.011	0.4
	2D	1.015	
4	3D	1.000	1.2
	2D	1.012	

* The major principal stress at node 4 predicted by the 3D analysis has a dimensionless stress value of 1.000. The other stress values listed in the table are major principal stresses presented in terms of the unit stress at node 4.

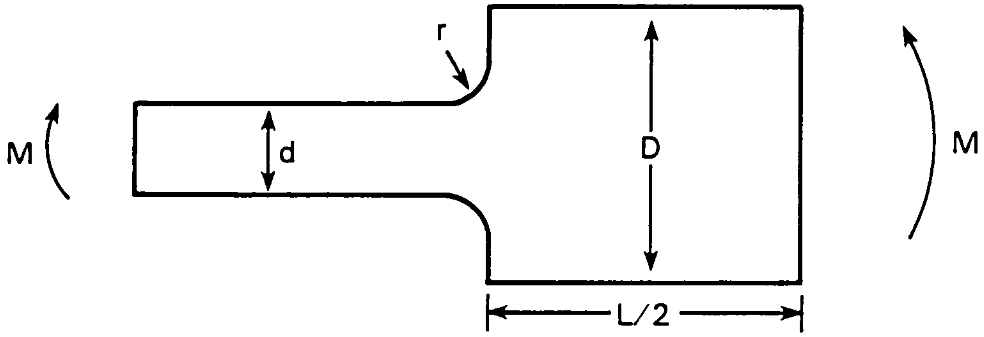
TABLE 4. EXTREMUM STRESS PREDICTIONS BASED ON 2-D PLANE STRAIN ANALYSES OF THE BLADE, HUB AND FILLET.

Finite Element Code	Element Type *	Number of Elements	Mesh Size l/r	Fillet Size r/d	Extremum Stress **
NASTRAN	CTRIA2	243	0.03	0.37	0.985
NASTRAN	CTRIA2	243	0.03	0.46	0.974
NASTRAN	CTRIA2	243	0.12	0.56	1.00
ABAQUS	CPE4	136	0.40	0.46	0.968

r = fillet radius
d = blade thickness at hub
l = element length in region of extremum stress

* CTRIA2 refers to the NASTRAN CTRIA2 element modified for plane strain. CPE4 refers to the ABAQUS plane strain quadrilateral element.

** The extremum stress predicted by the 3-D blade model has a dimensionless stress value of 1.0. The stresses listed in the table are presented in terms of this unit stress.



r/d = 0.2
L/D = 2
D/d = 3

K = 1.53 determined photoelastically by Hartman and Leven (ref. 7).

Figure 1. A filleted bar in a field of pure bending.

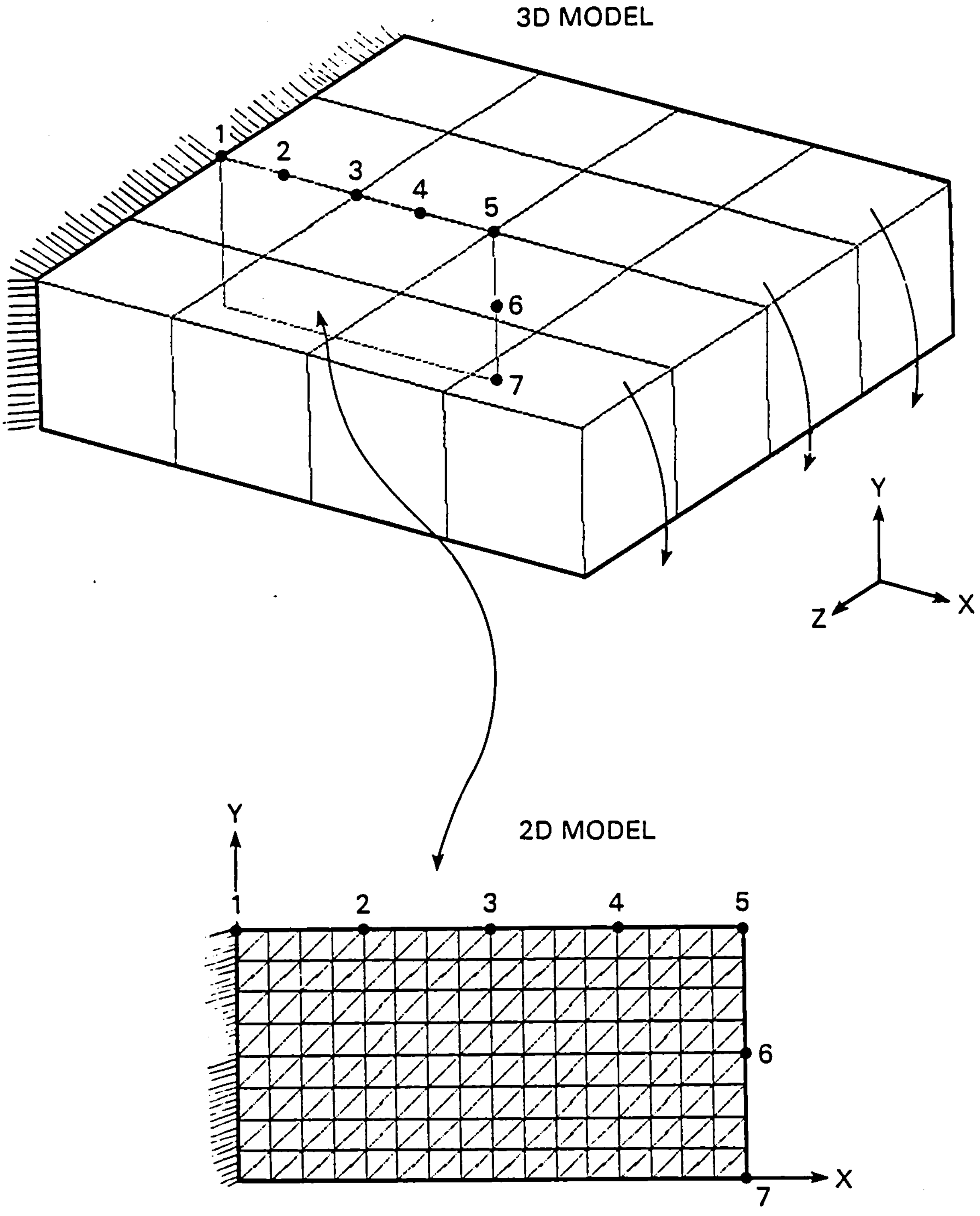


Figure 2. 3D and 2D thick-plate models.

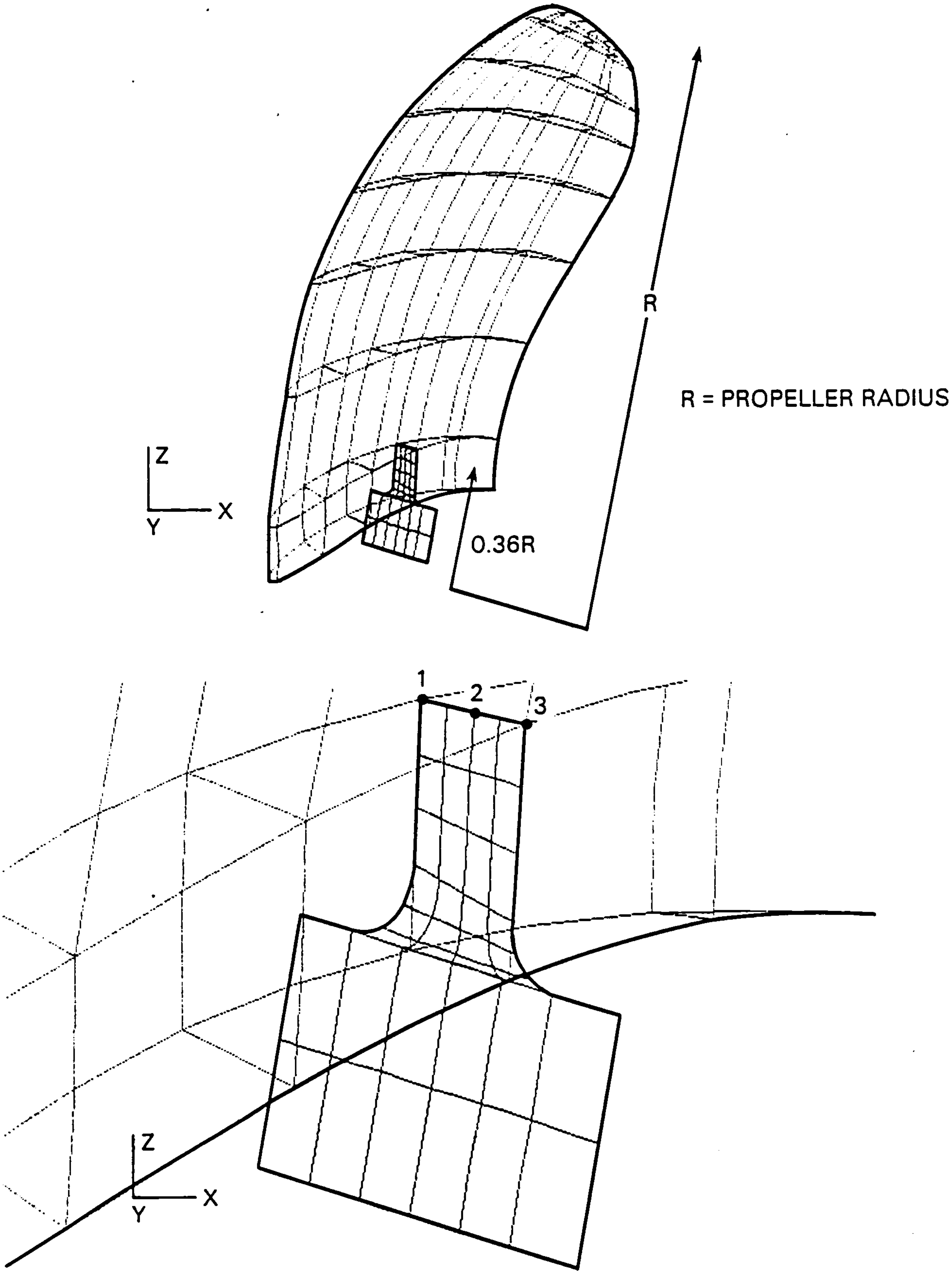


Figure 3. 2D blade, hub and fillet model superimposed on a 3D propeller blade model composed of 40 CIHEX2 elements.

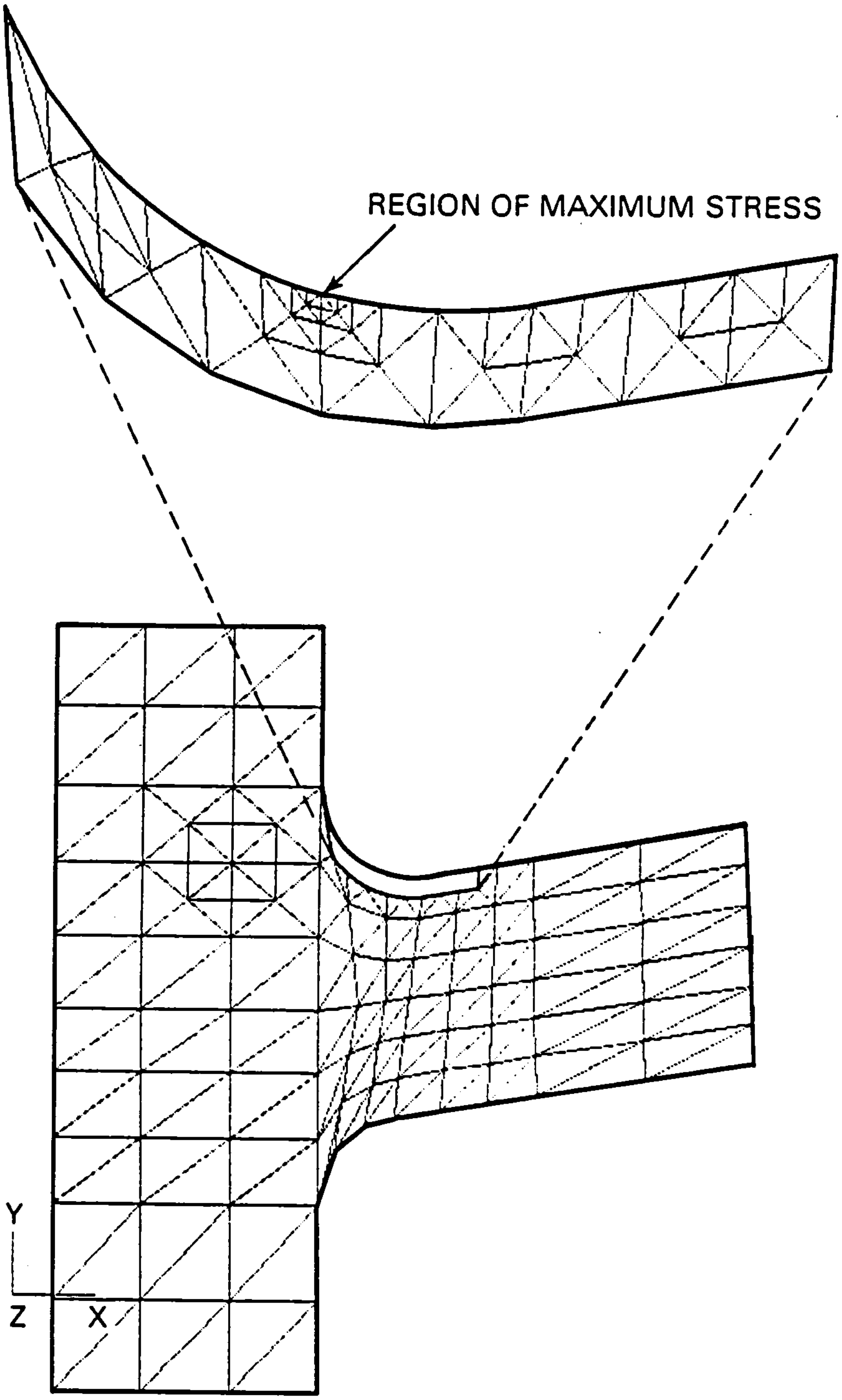


Figure 4. 2D Model of blade, hub and fillet showing detail of upper fillet. Model is composed of 243 CTRIA2 elements.