ABSTRACT

A parametric study has been done to determine the best finite-element mesh, using QUAD8 elements, to predict the stress distribution adjacent to the notches in the plastic sabot used on a 25mm projectile. A sabot is a sleeve of low-density material used to support a subcaliber projectile in a gun barrel and then break up at muzzle exit. The objective of the study was to find the most efficient mesh to model the sharp stress gradient that exists at the apex of the notch when the projectile is subjected to a high angular velocity causing circumferential tensile stresses in the sabot. Ten different cases were analyzed and several were found to give good results.

Background

A parametric study has been done to determine the best finite-element mesh, using QUAD8 elements, to predict the stress distribution adjacent to the longitudinal notches in a plastic sabot used on a 25mm armor-piercing discarding-sabot type projectile. The projectile being studied is spin stabilized and is therefore fired from a rifled gun barrel. The helical form of the rifling grooves imparts the required spin to the projectile, giving it a very high angular velocity as it exits the muzzle of the barrel.

A sabot is a sleeve of low-density material surrounding a subcaliber projectile. The sabot under study here is made of plastic and fits tightly around the heavy metal penetrator. The sabot, shown in Figure 1, provides the lateral support required for projectile launch and also protects and restrains the penetrator during handling and feeding through the gun mechanism. A successful sabot design has low mass, is relatively rigid to minimize yaw in the barrel, and fractures cleanly and uniformly at the muzzle to minimize interference with the ballistic flight of the penetrator. This sabot is made with four longitudinal grooves or notches to provide weakened sections that will fracture readily at muzzle exit due to the circumferential stress imparted by the spin of the projectile.

Objective

The objective of the study was to find the most efficient mesh geometry to accurately model the sharp stress gradient that exists at the apex of the notch when the projectile is subjected to a high angular velocity causing circumferential tensile stresses in the sabot. A total of ten different mesh configurations were analyzed and several were found to give acceptable results for the stress distribution across the reduced section and for the maximum stress at the notch.

Technical Approach

It is recognized that there is a degree of artificiality in the optimization in that the actual part, which has a maximum radius of .005 inch at the groove, is being represented with a model having a 90^{o} notch with no radius. It was decided at the beginning of the study that it would be impractical to explicitly model the actual geometry of the radius at the base of the groove because it would require too many elements for the analysis to be cost effective.

The approach taken, therefore, was to use an iterative procedure trying different arrangements of nodes and elements, using elements both with and without mid-side node, using various sizes of elements and changing the node spacing at the reduced section in an attempt to converge to a mesh geometry giving a K factor close to the calculated theoretical value but using the fewest possible elements and having a configuration that would be practical for a 3-D model.

Discussion

MSC/NASTRAN, run on a Control Data Cyber 176 computer, was used for the finite-element modeling and stress analyses. MSC/NASTRAN was chosen for the analysis because the code offers both four-node and eight-node quadrilateral plate elements and the printed stress output tabulates stresses at the corner nodes of each element. Since the centerline of the notch through the reduced section forms one boundary of the model, as shown in Figure 5, the stresses at the restrained nodes along the boundary were used to plot the stress distributions for each of the ten cases investigated. The results of this study will be used to develop an optimized three-dimensional sabot model which will be analyzed for in-bore and muzzle exit loading conditions. The primary reasons for choosing MSC/NASTRAN for the 3-D analysis are based on keeping computer solution costs and model generation man-hours to a reasonable level. These reasons are as follows:

- o Linear, elastic material properties will be used.
- o MSC/NASTRAN has a 3-D mesh generating pre-processor, MSGMESH, which can handle complex intersecting points in a 3-D model.
- o Cyclic symmetry is available only with NASTRAN. This will allow a model of only one-half of one segment (1/8 of the total sabot) to be used for analyzing symmetric and unsymmetric loads.

A cross-section of one-quarter of the sabot with the groove is shown in Figure 2. Based on this cross-section a simplified configuration was developed, as shown in Figure 3, so that the stress concentration effect of the notch could be studied as a plane stress problem. For the configuration of Figure 3, the theoretical stress concentration factor was calculated as K=3.44, using the equations from <u>Formulas for Stress and Strain</u> by Roark and Young (the equations are from Table 37, case 2a: axial tension on a member of rectangular section with two V-notches.) Therefore, for an average tensile stress of 2,000. psi at the reduced section the maximum theoretical stress at the base of the 90^{0} V-notch is 6,880. psi. Also, the shape of the stress distribution curve for the stress across the reduced section should be similar to the curve shown in Figure 6 for a tension member with two notches.

Based on the simplified configuration developed in Figure 3 and utilizing the two planes of symmetry the basic geometry of the finite element model was chosen as shown in Figure 4. The boundary conditions for the model are as shown in Figure 5 with X-translations restrained for the nodes along the notch centerline, a sliding boundary for the nodes on the X-axis, X-translations coupled for nodes 1, 2 and 3, and the applied force F_x = -0.0548 lbs. giving an average stress of 2,000. psi across the reduced section.

CQUAD8 quadrilateral plate elements (both with and without mid-side nodes), with membrane stiffness only, were used to generate the ten different mesh geometries which were studied. The various meshes representing the ten cases are shown in Figures 7 through 9.

Results

The models representing the ten cases were analyzed for the in-plane tension load and the results were evaluated to determine the stress distribution and stress concentration factor for each case. The stress concentration factor was calculated by dividing the maximum principal stress at the notch apex node by the average stress of 2,000. psi. The results for the ten cases are summarized and presented in Table 1. Plots showing the stress distribution at the reduced section for each of the cases 1 to 10 are presented as Figures 10 through 19, respectively.

Case 1, with two elements at the reduced section, gives a stress concentration factor, K=1.55. Case 2 uses the same size elements as case 1 with the addition of mid-side nodes to the two elements at the notch and yields a value of 2.11 for the stress concentration factor. Again using two elements with mid-side nodes but decreasing the element size results in a K factor of 2.33 for case 3. Case 4 has a mesh with three four-node elements at the reduced section and gives a value of 2.92 for the stress concentration factor. For case 5, a mesh with 5 equally spaced nodes and 4 elements at the reduced section was analyzed and found to have a K factor of 2.93, just slightly higher than case 4. Seven equally spaced nodes at the reduced section with three eight-node elements gave a K value of 2.98 for the mesh of case 6. Case 7 features a slight modificaiton of the geometry of case 4 with the addition of one node and one element at the reduced section and the analysis shows an increase to 3.03 for the value of K.

Case 8 represents an attempt to simplify the mesh but still obtain good results for the stress distribution at the notch. The mesh has four nodes at the reduced section and only three elements and gives a K value of 3.03, the same as case 7.

Case 9 has seven equally spaced nodes at the reduced section and six four-node elements for plotting the stress distribution and gives fairly good results with a value of 3.07 for the stress concentration factor. However, this case is not very efficient as the mesh has a total of 22 elements versus 13 for case 8 and gives only about a one percent increase in the K value.

Case 10 has five nodes and four elements at the reduced section with two very small elements right at the base of the notch and gives a K value of 3.37, which is within two percent of the theoretical value of 3.44. The mesh of case 10 is complicated somewhat because it requires four RSPLINE elements for mesh refinement in the area at the base of the notch.

Conclusion

The mesh of case 10 does give the best results for the stress distribution and the stress concentration factor at the notch but for the 3-D model it was decided that avoiding the use of the RSPLINE elements would be beneficial in simplifying the model. The mesh of case 8 was judged to be the most efficient as it gave a K value that was twelve percent below the theoretical value and required only thirteen elements in total with three elements at the reduced section and no RSPLINE elements.

TABLE 1

Summary of Results

Case No.	Element Type at Notch	No. of Elements in Mesh	No. of Nodes at Reduced Section	No. of RSPLINE Elements	Stress Concentration Factor, K
1	4-Node	8	3	-	1.55
2	8-Node	8	5	2	2.11
3	8-Node	8	5	2	2.33
4	4-Node	13	4	2	2.92
5	4-Node	14	5	2	2.93
6	8-Node	16	7	4	2.98
7	4-Node	16	5	3	3.03
8	4-Node	13	4	-	3.03
9	4-Node	22	7	4	3.07
10	4-Node	14	5	4	3.37

NOTE: $K_{theo} = 3.44$

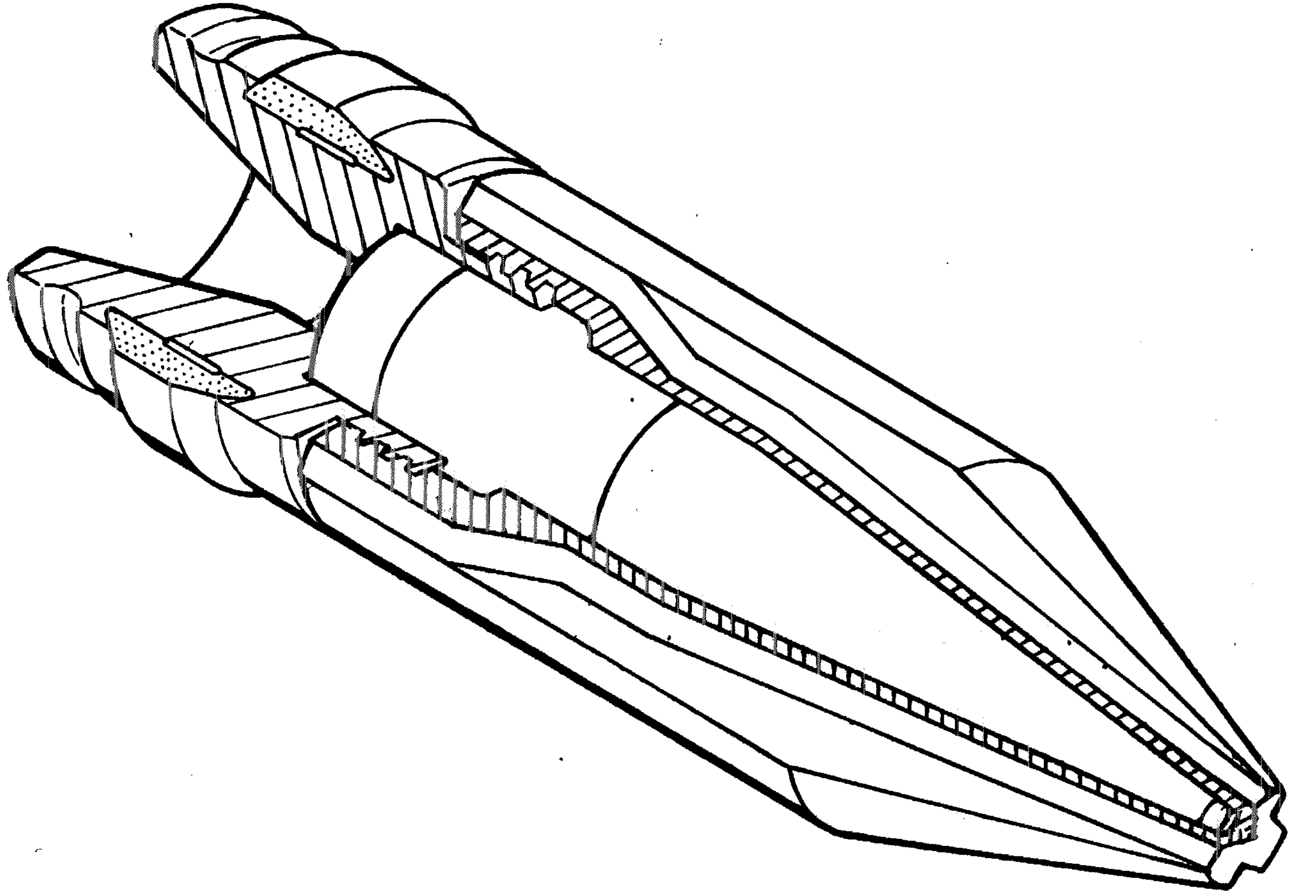


FIGURE 1. 25mm PROJECTILE

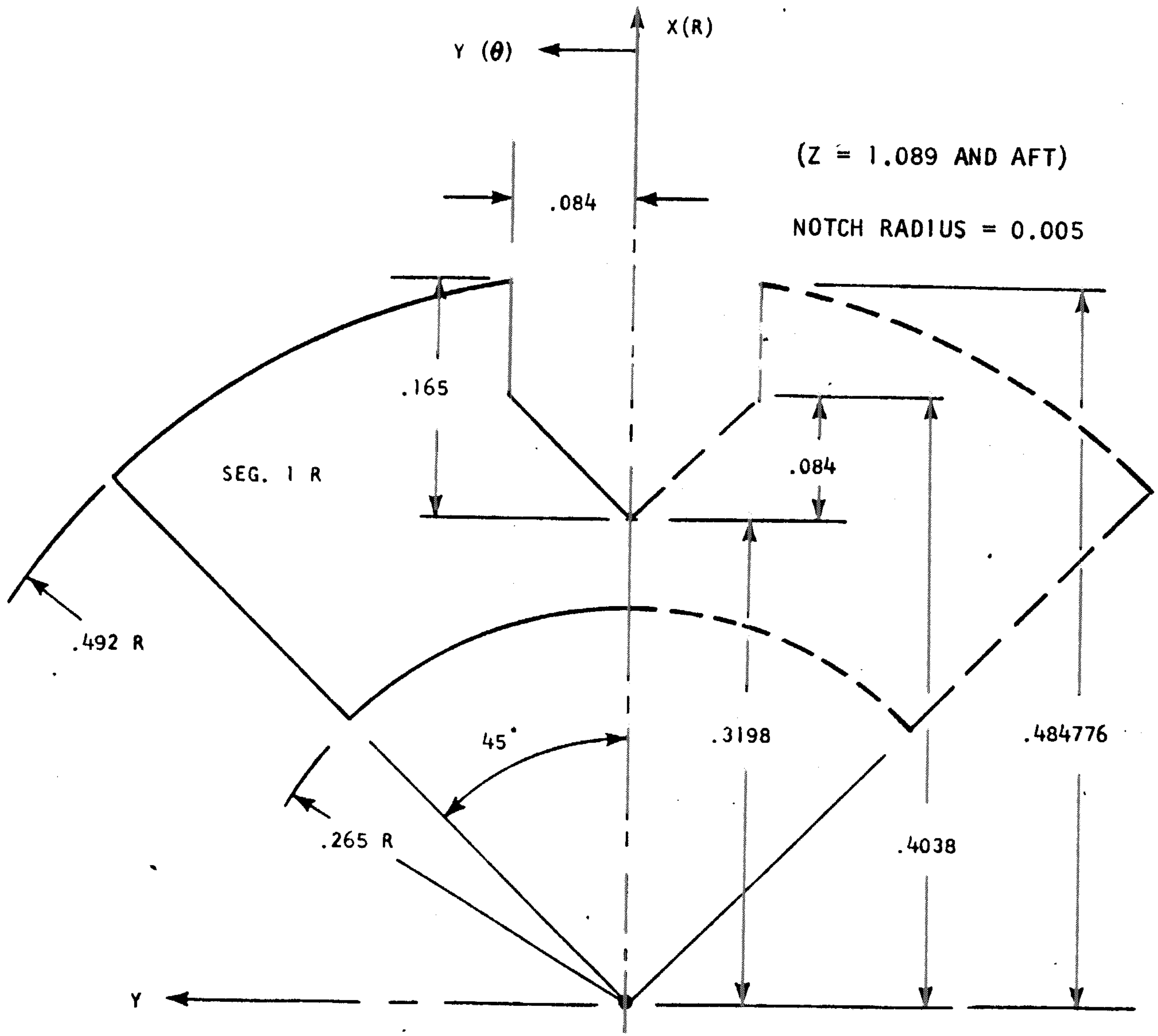


FIGURE 2. SABOT CROSS-SECTION AT NOTCH

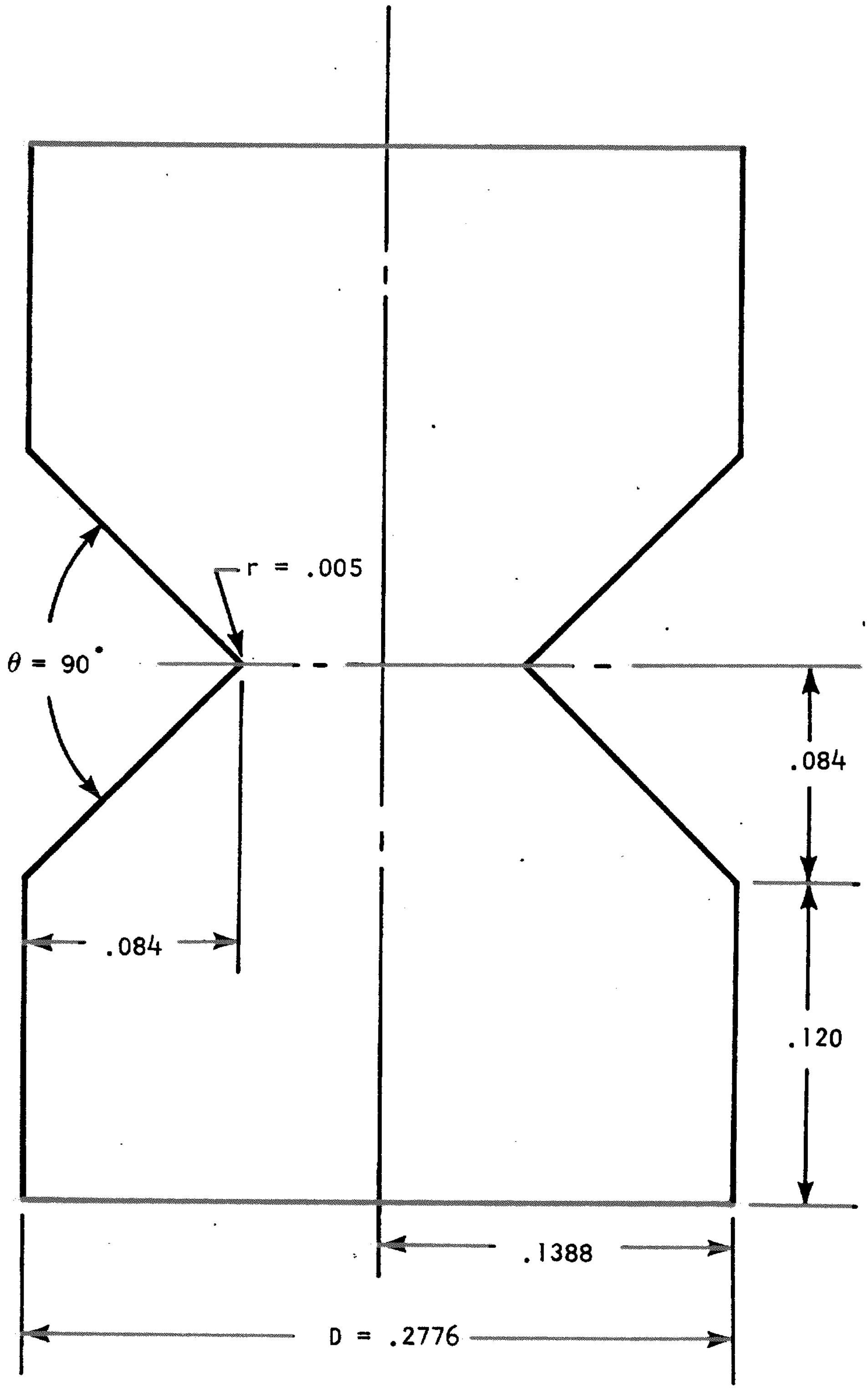


FIGURE 3. ASSUMED SECTION CONFIGURATION FOR CALCULATING STRESS CONCENTRATION FACTOR

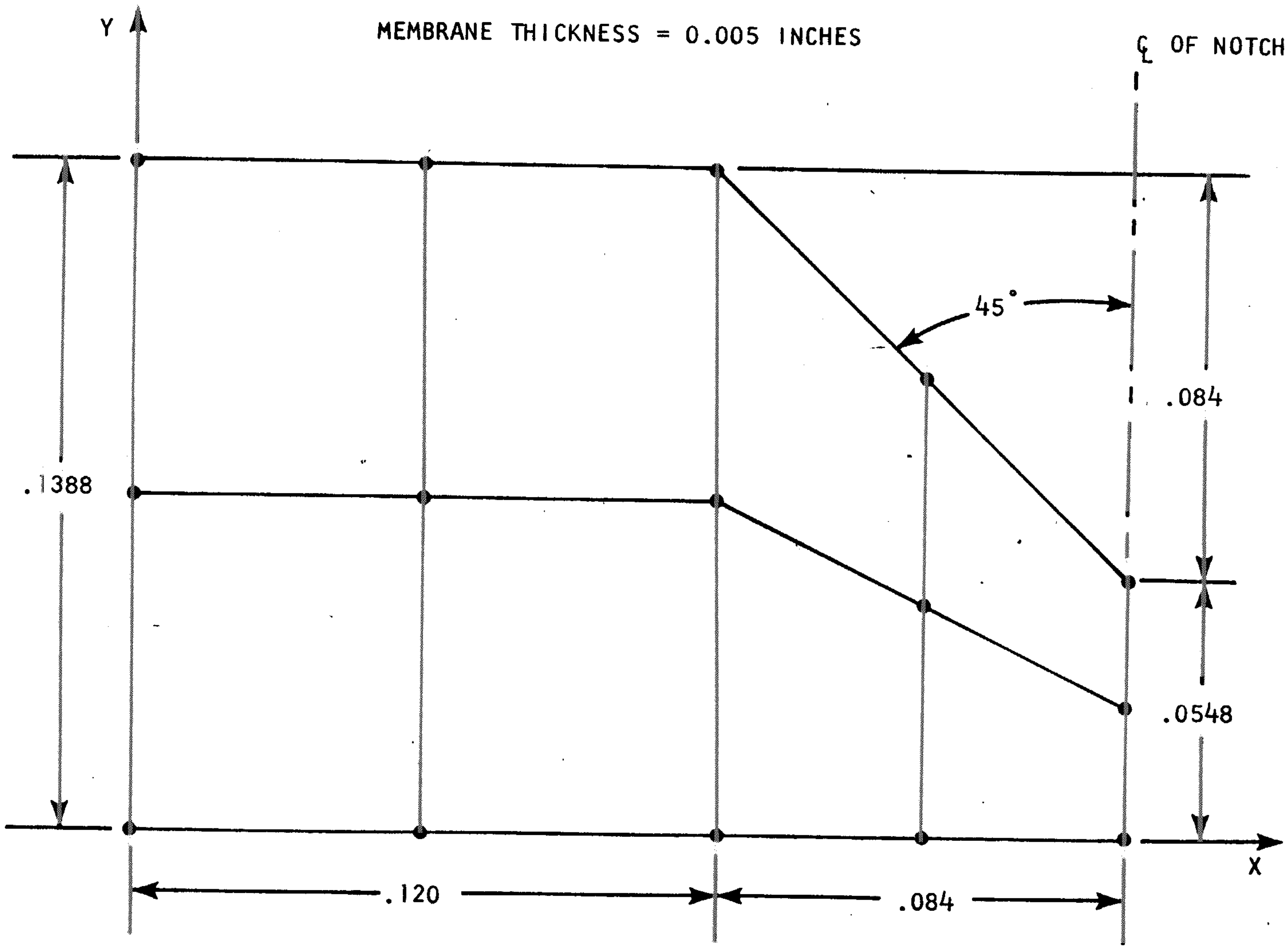


FIGURE 4. BASIC GEOMETRY AND DIMENSIONS

NOTES: 1. NODES 1, 2, AND 3 ARE COUPLED SUCH THAT $UX_1 = UX_2 = UX_3$

2. NOMINAL STRESS AT BASE OF NOTCH, σ_x = 2,000 PSI

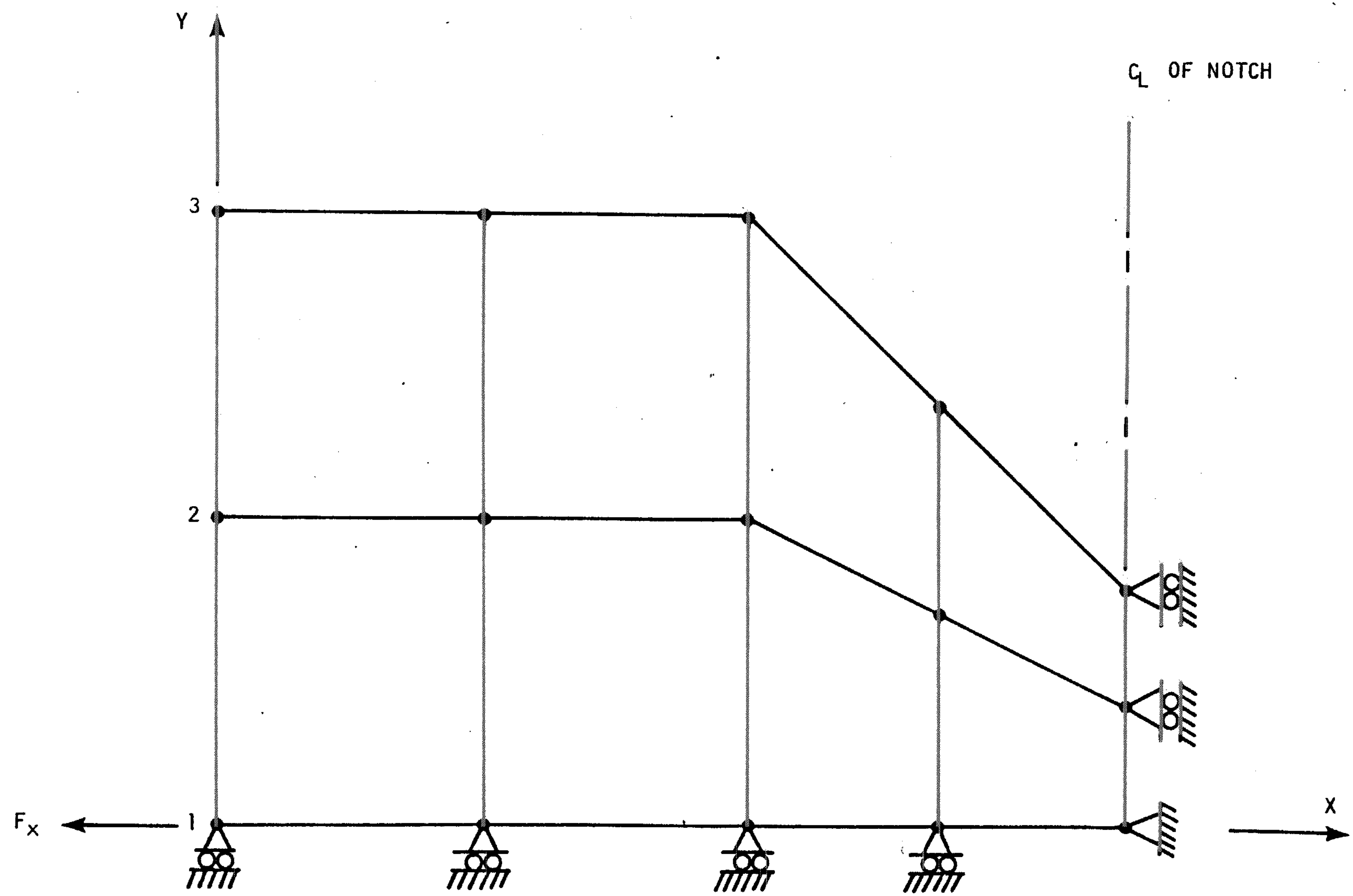
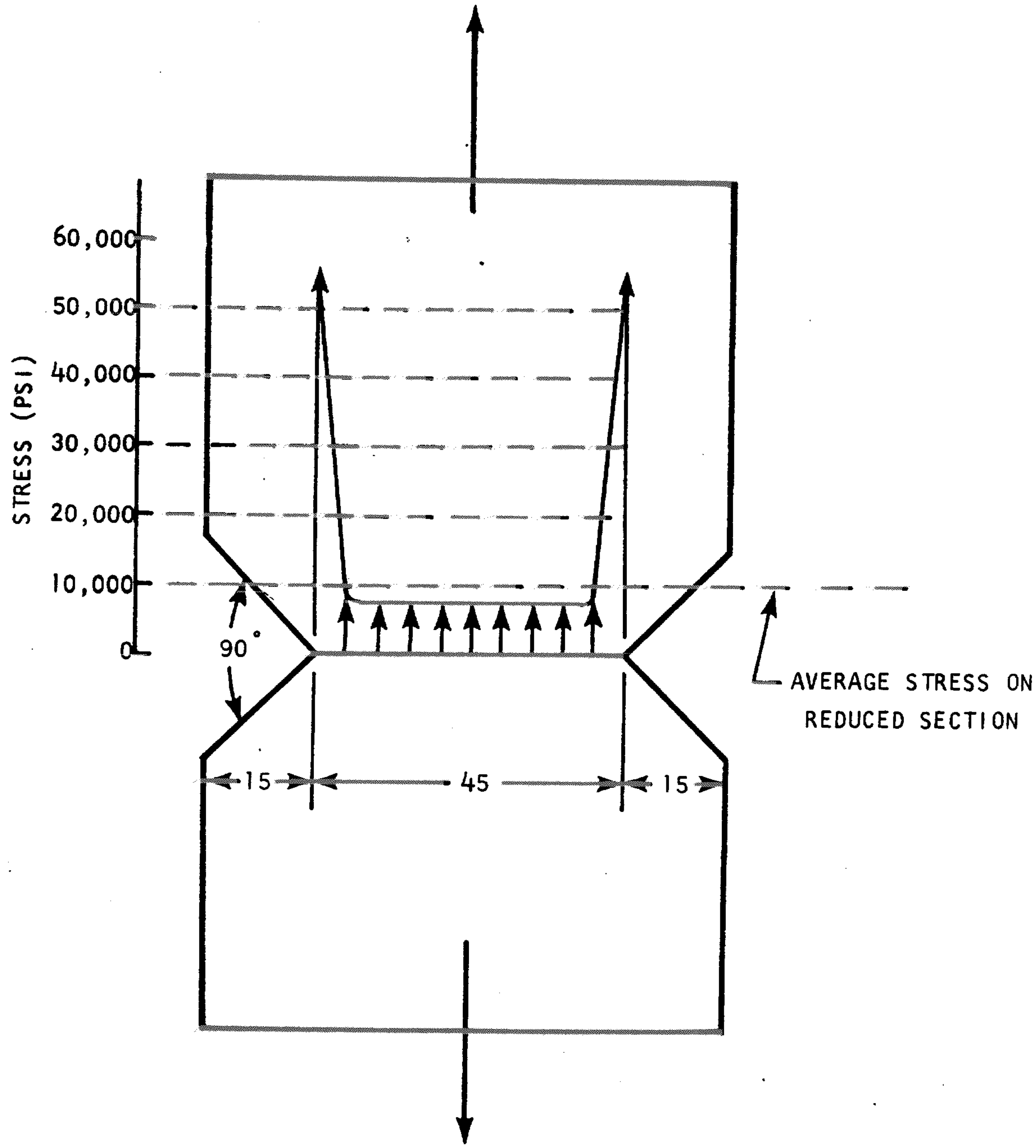
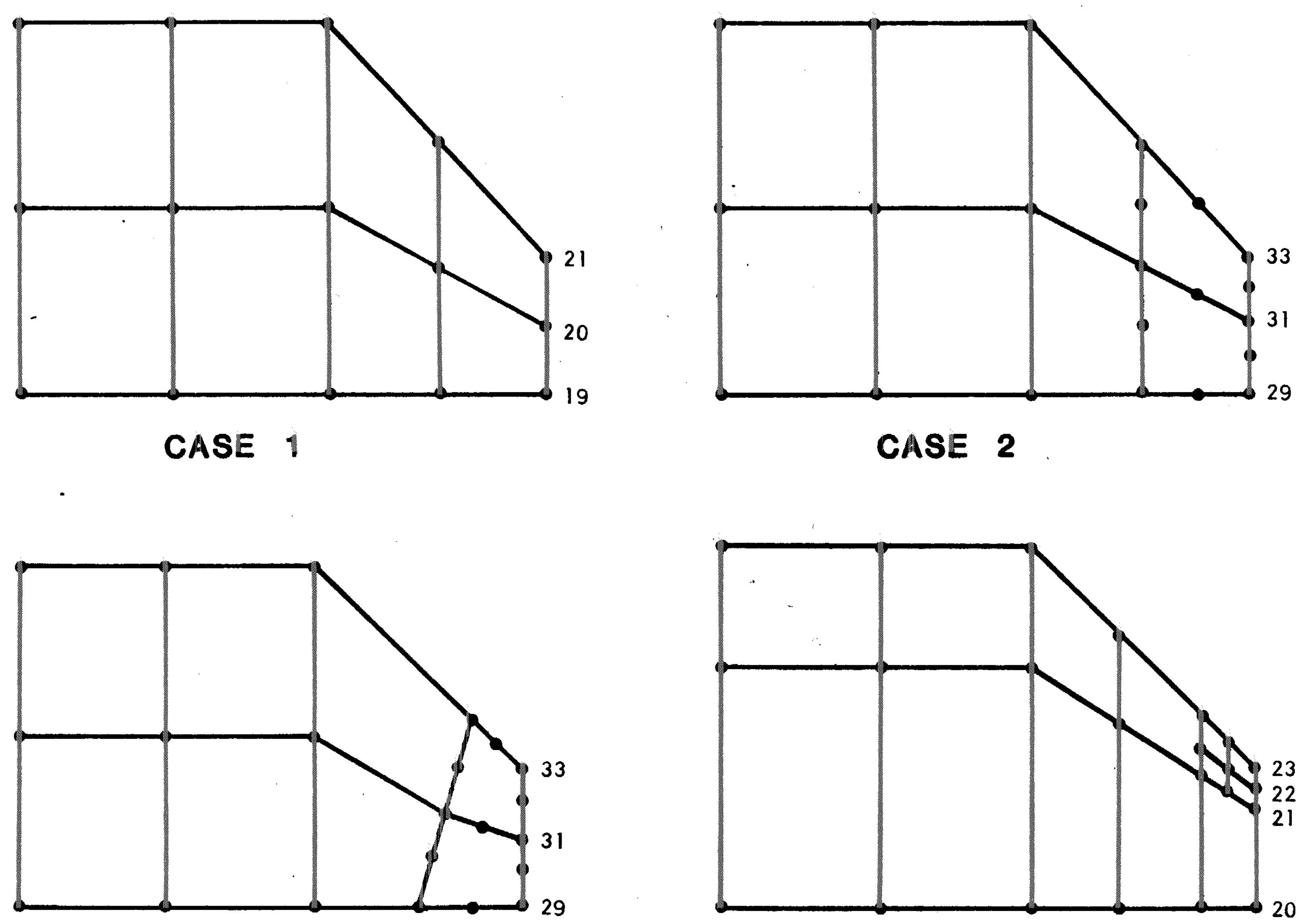


FIGURE 5. BOUNDARY CONDITIONS



(FROM PREUSS, IN SEELY AND SMITH, "ADVANCED MECHANICS OF MATERIALS", SECOND EDITION, p. 399)

FIGURE 6. STRESS AT NOTCHES FOUND BY LATERAL-STRAIN METHOD



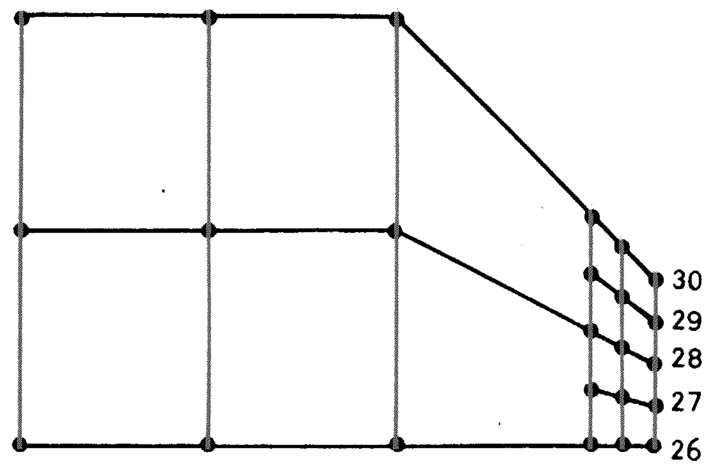
CASE 1

CASE 2

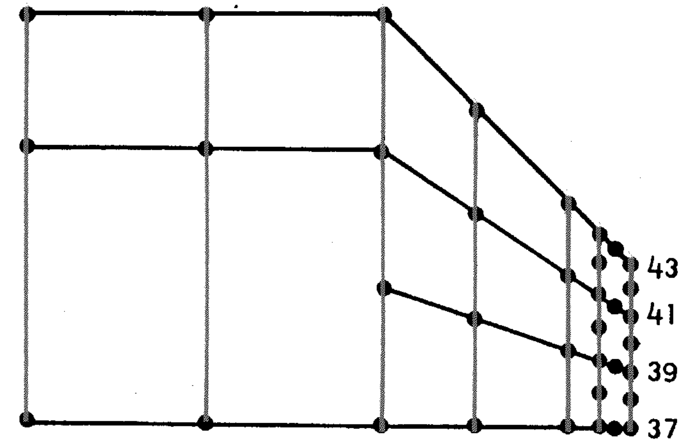
CASE 3

CASE 4

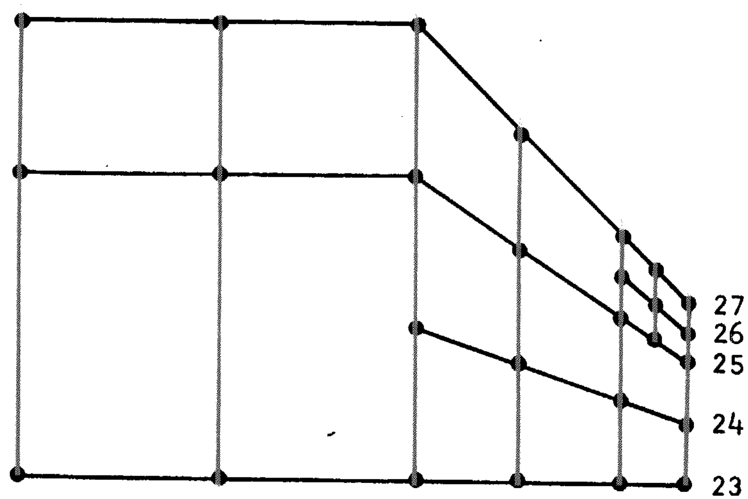
FIGURE 7. MESH GEOMETRIES FOR CASES 1 TO 4



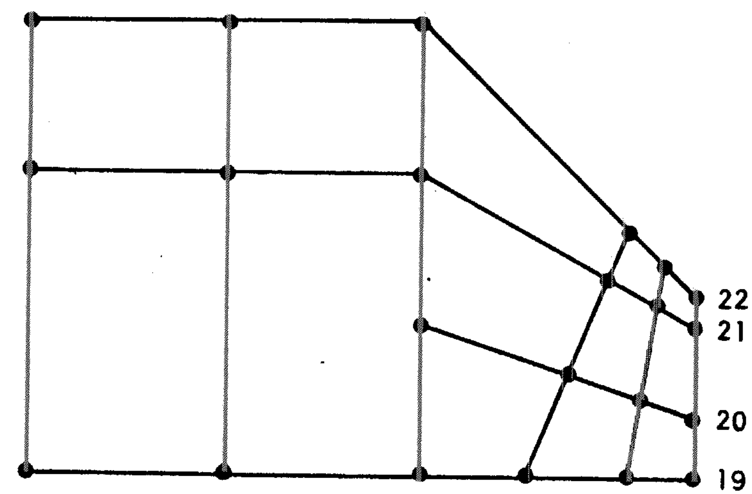
CASE 5



CASE 6

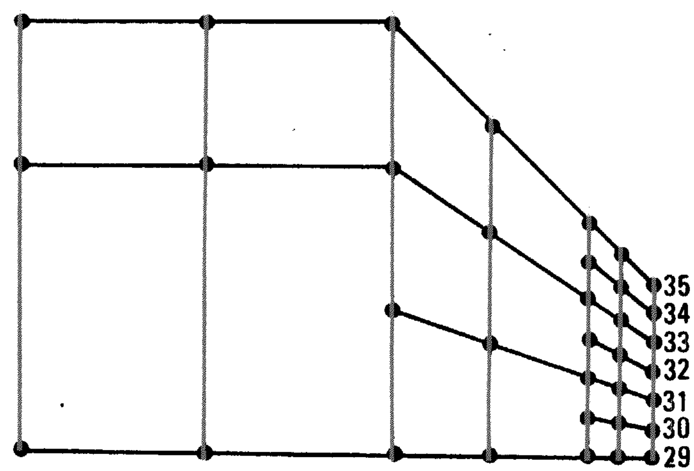


CASE 7

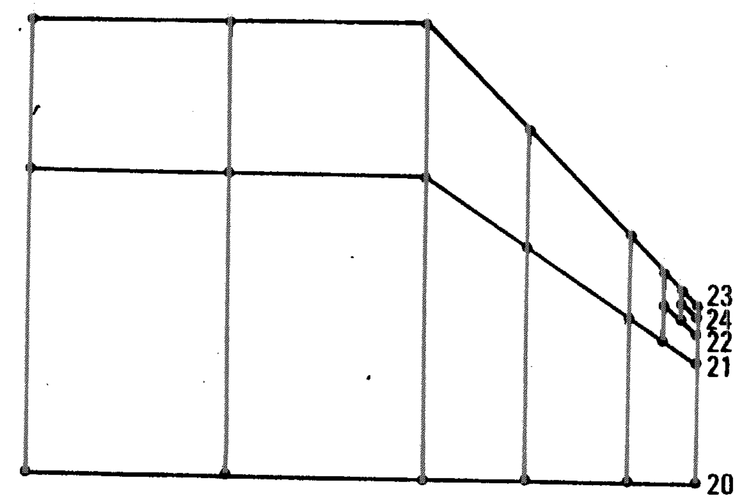


CASE 8

FIGURE 8. MESH GEOMETRIES FOR CASES 5 TO 8



CASE 9



CASE 10

FIGURE 9. MESH GEOMETRIES FOR CASES 9 AND 10

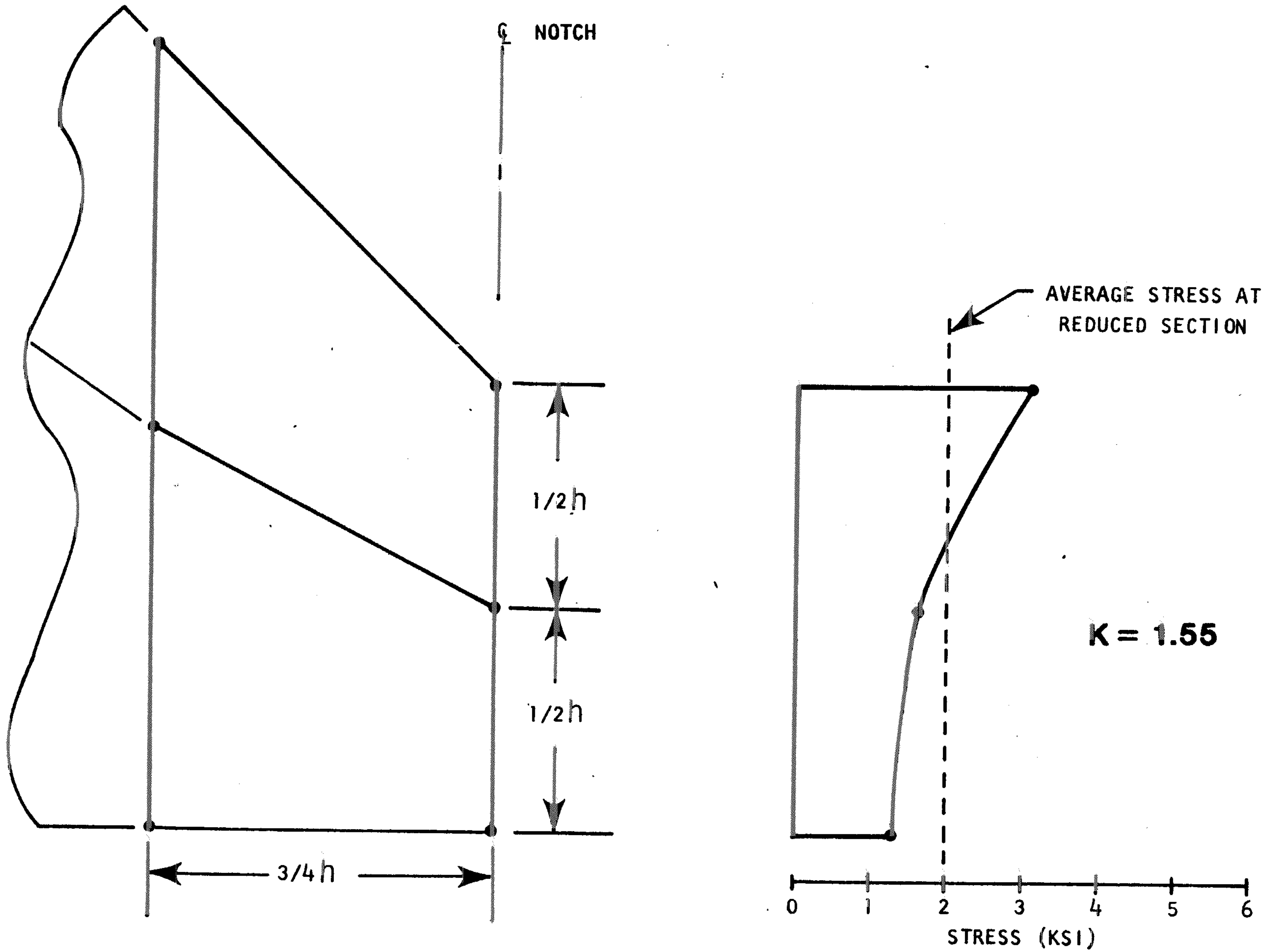


FIGURE 10. STRESS DISTRIBUTION FOR CASE 1

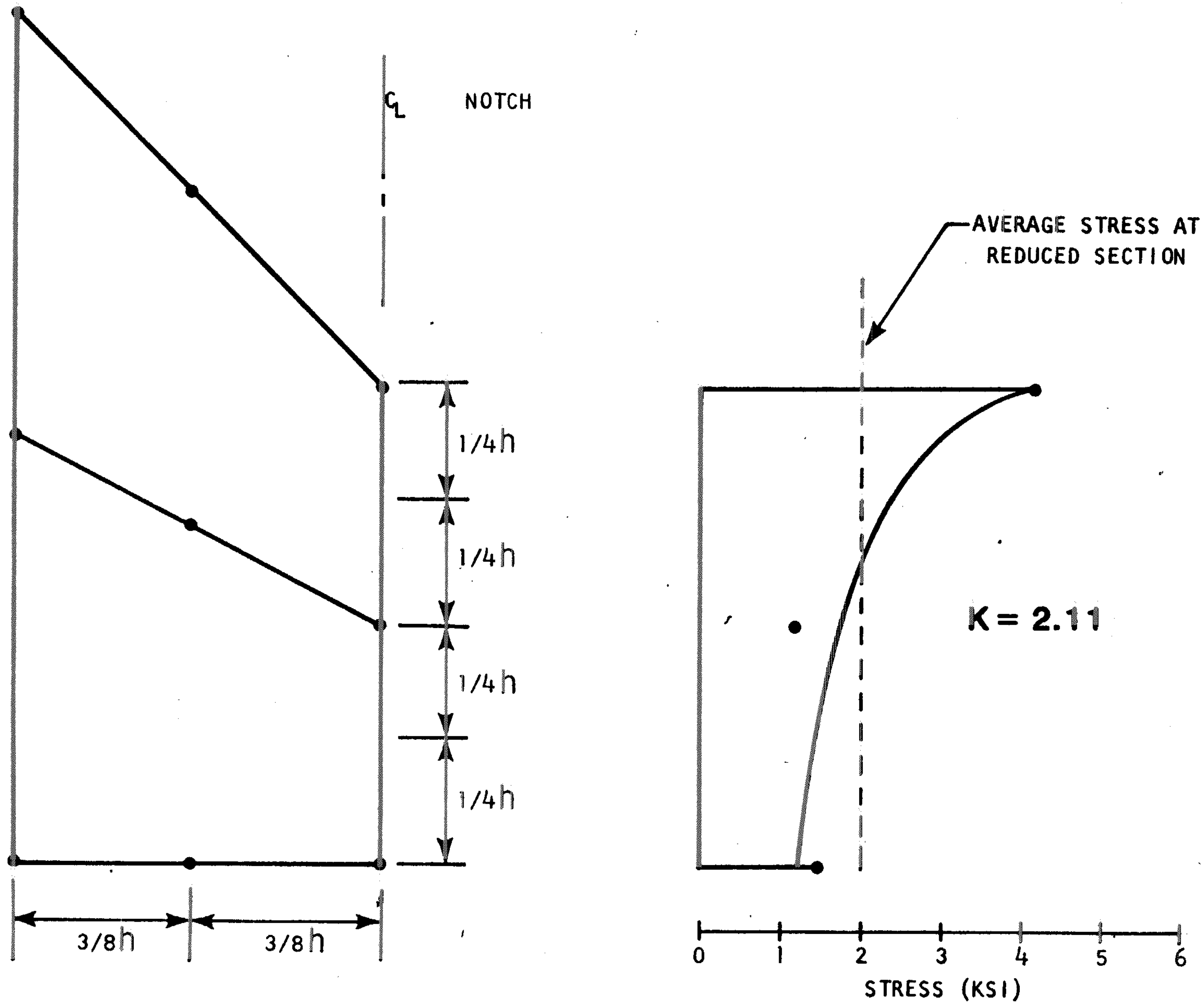


FIGURE 11. STRESS DISTRIBUTION FOR CASE 2

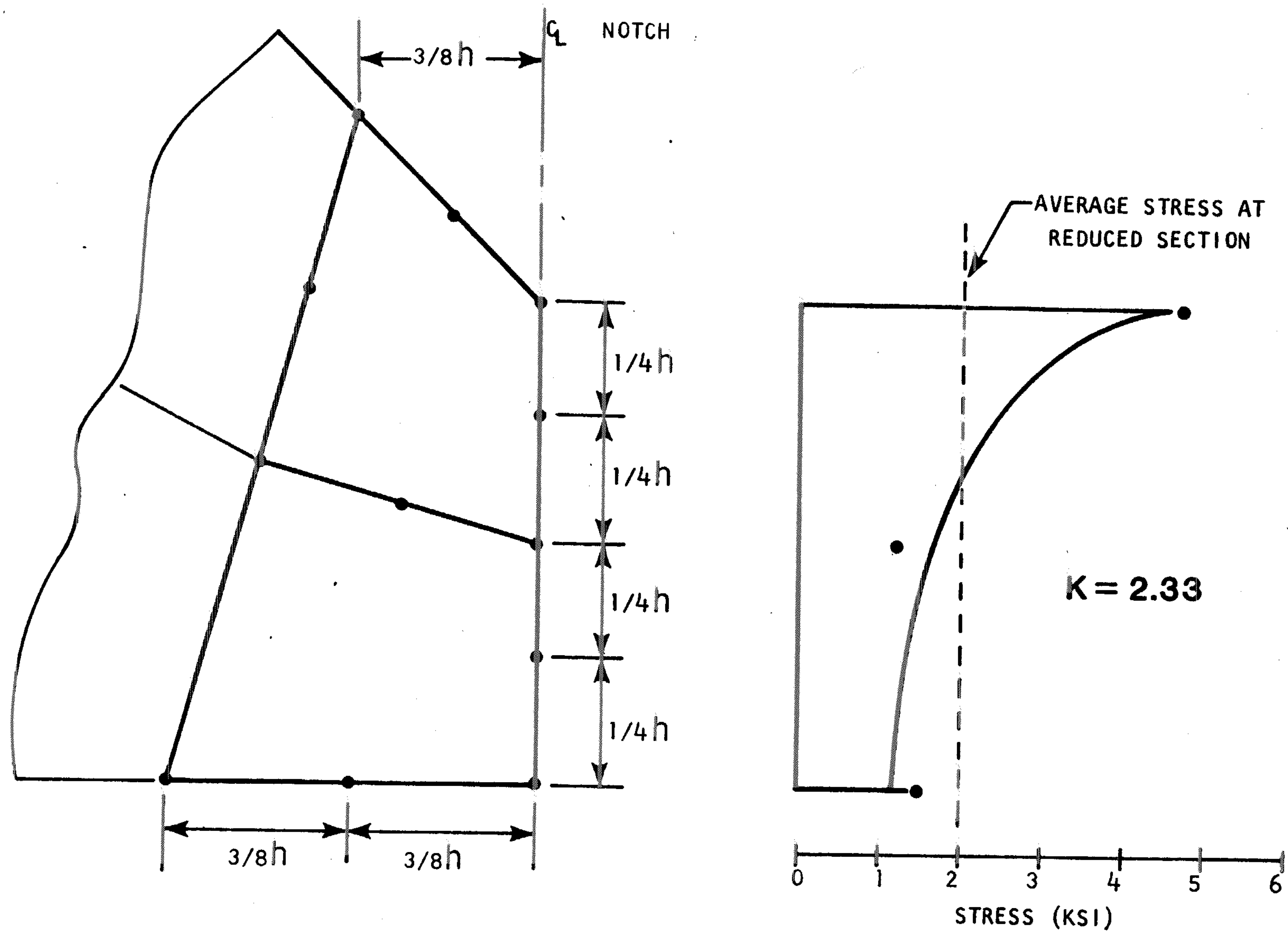


FIGURE 12. STRESS DISTRIBUTION FOR CASE 3

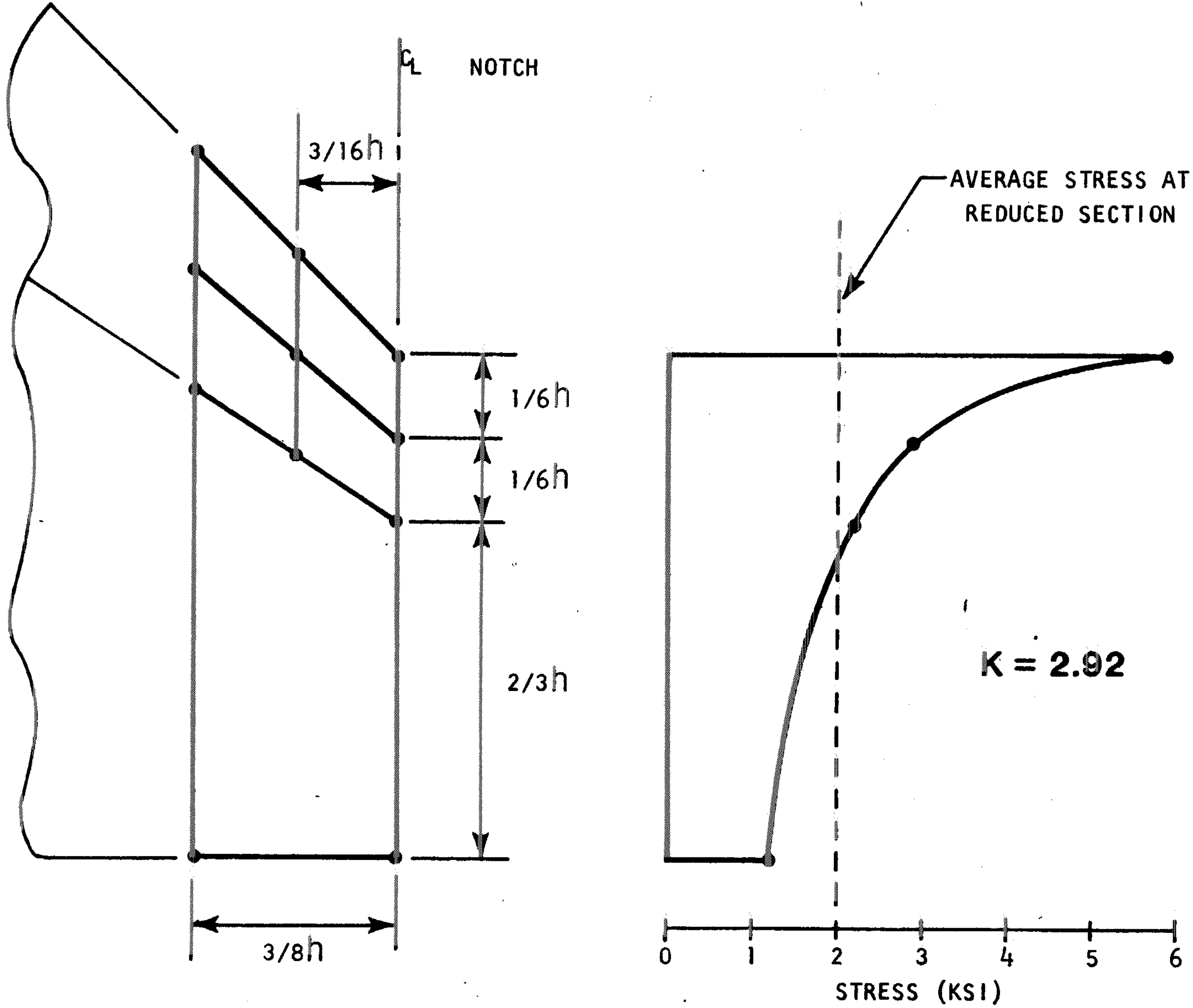


FIGURE 13. STRESS DISTRIBUTION FOR CASE 4

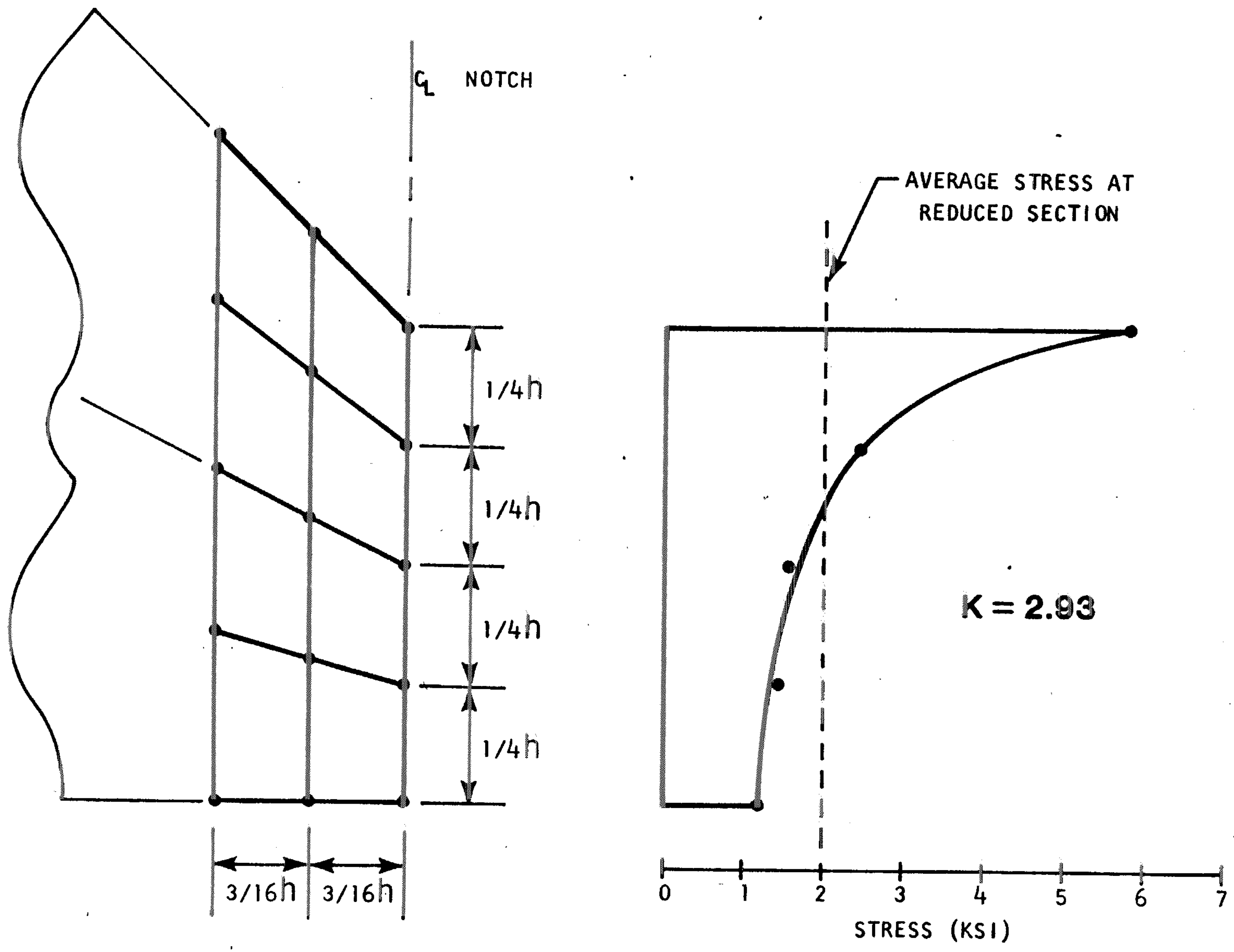


FIGURE 14. STRESS DISTRIBUTION FOR CASE 5

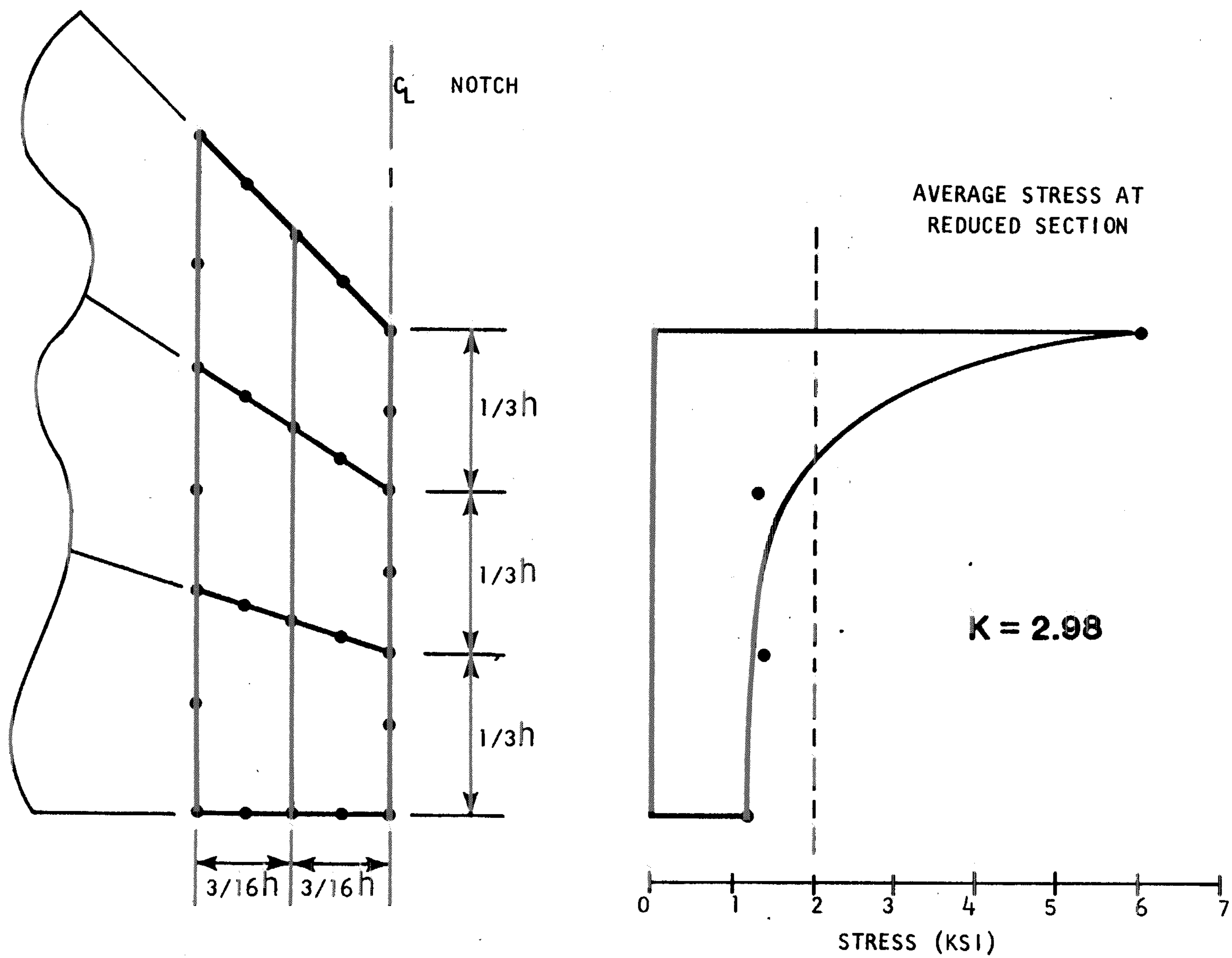


FIGURE 15. STRESS DISTRIBUTION FOR CASE 6

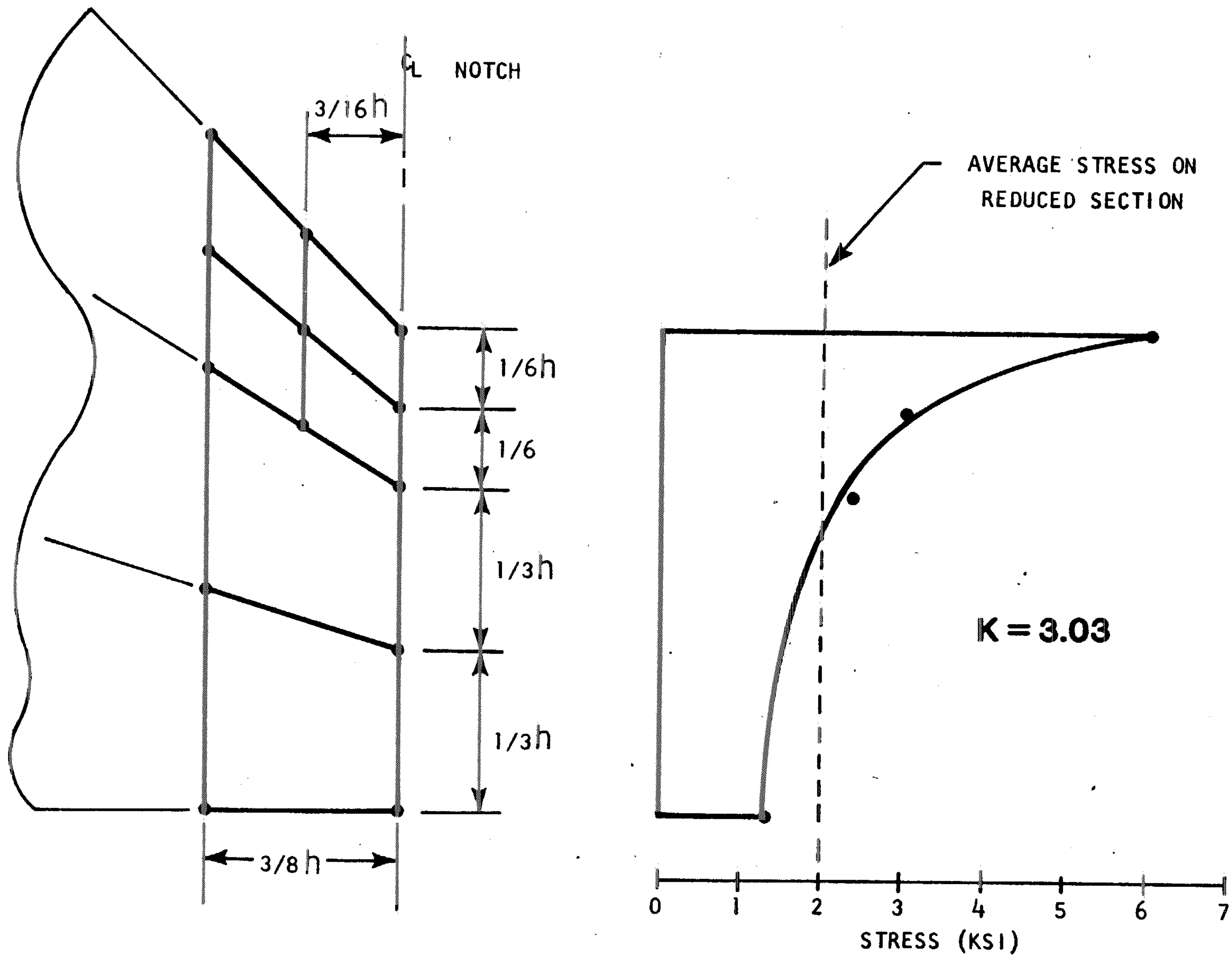


FIGURE 16. STRESS DISTRIBUTION FOR CASE 7

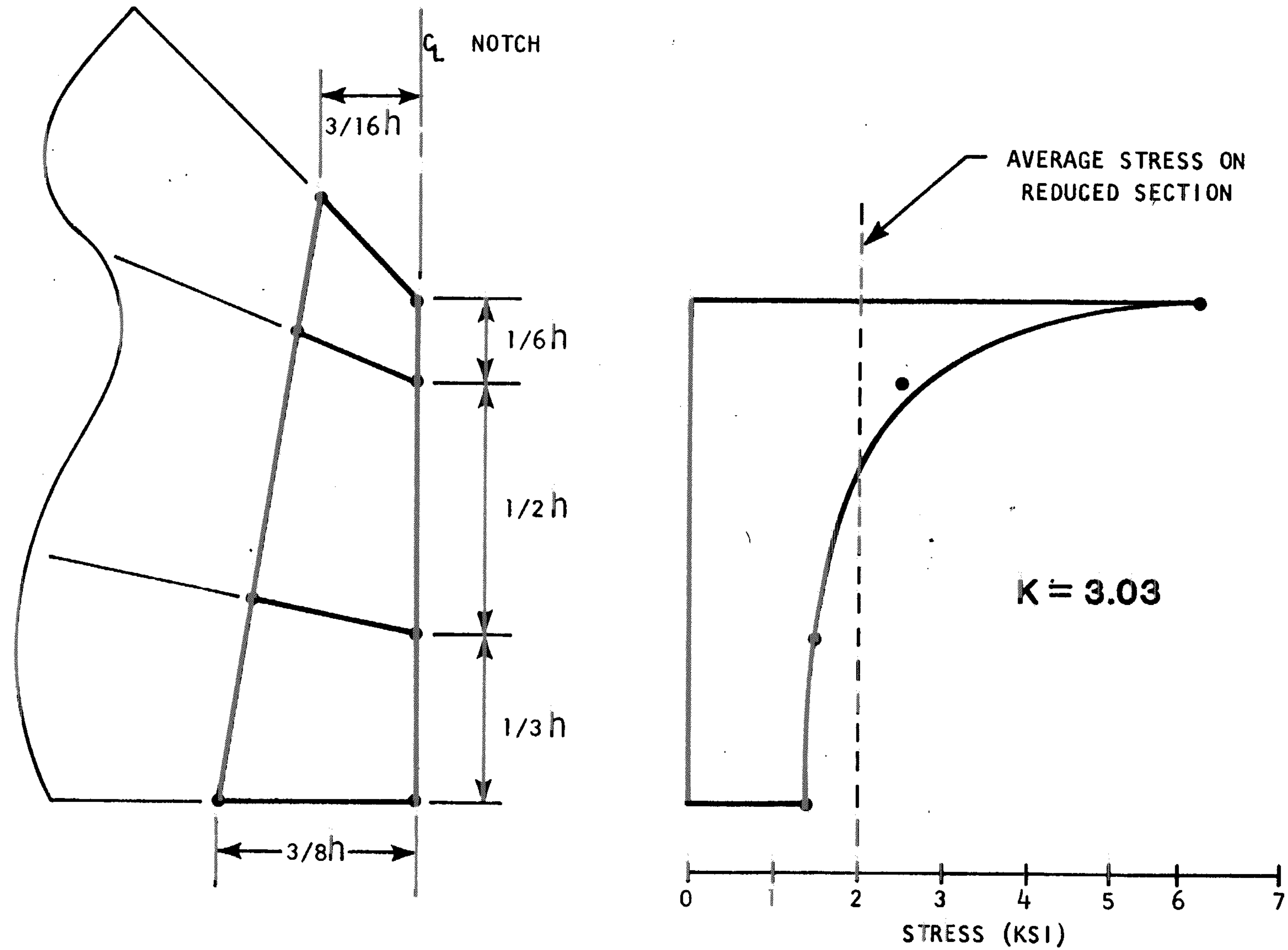


FIGURE 17 STRESS DISTRIBUTION FOR CASE 8

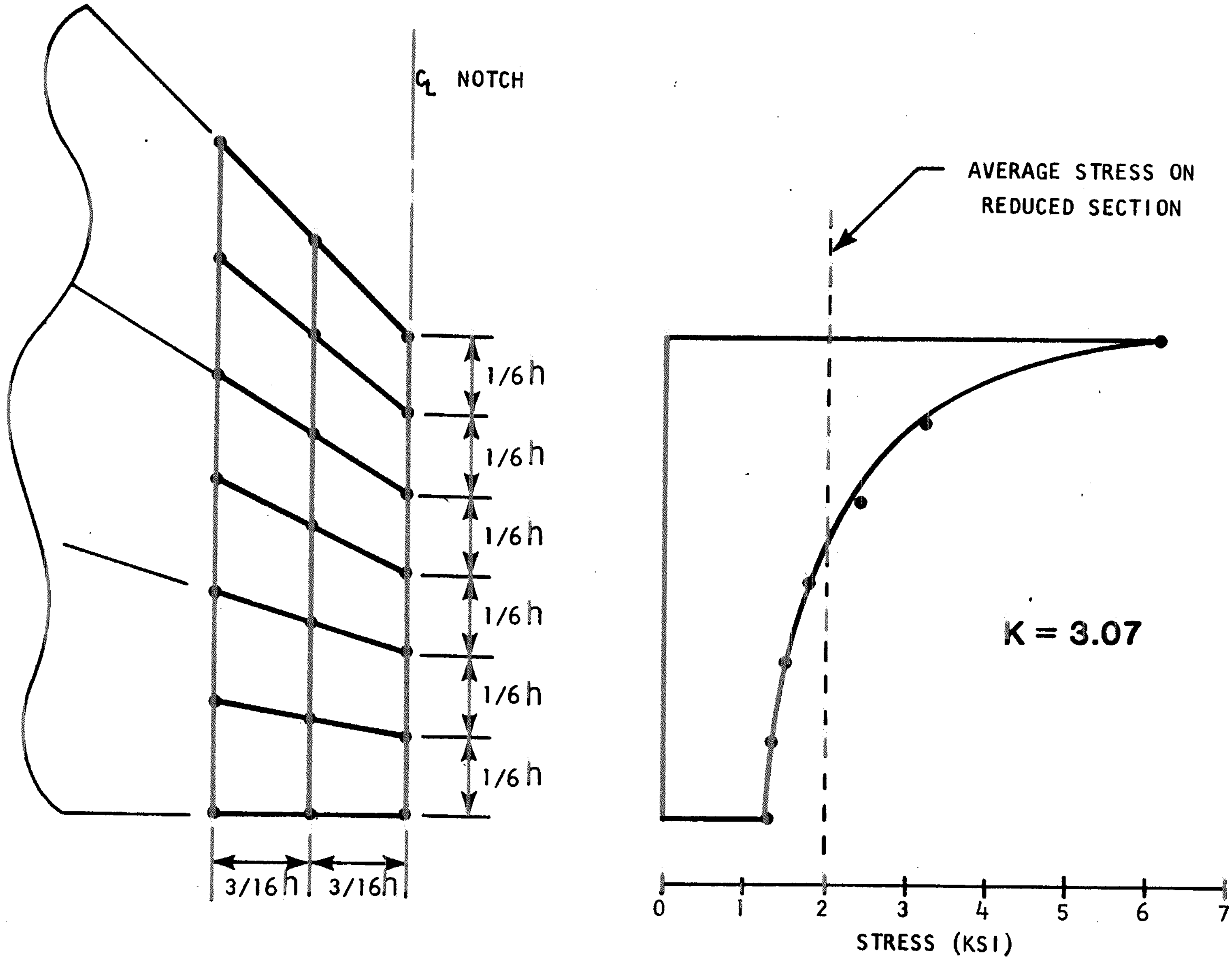


FIGURE 18. STRESS DISTRIBUTION FOR CASE 9

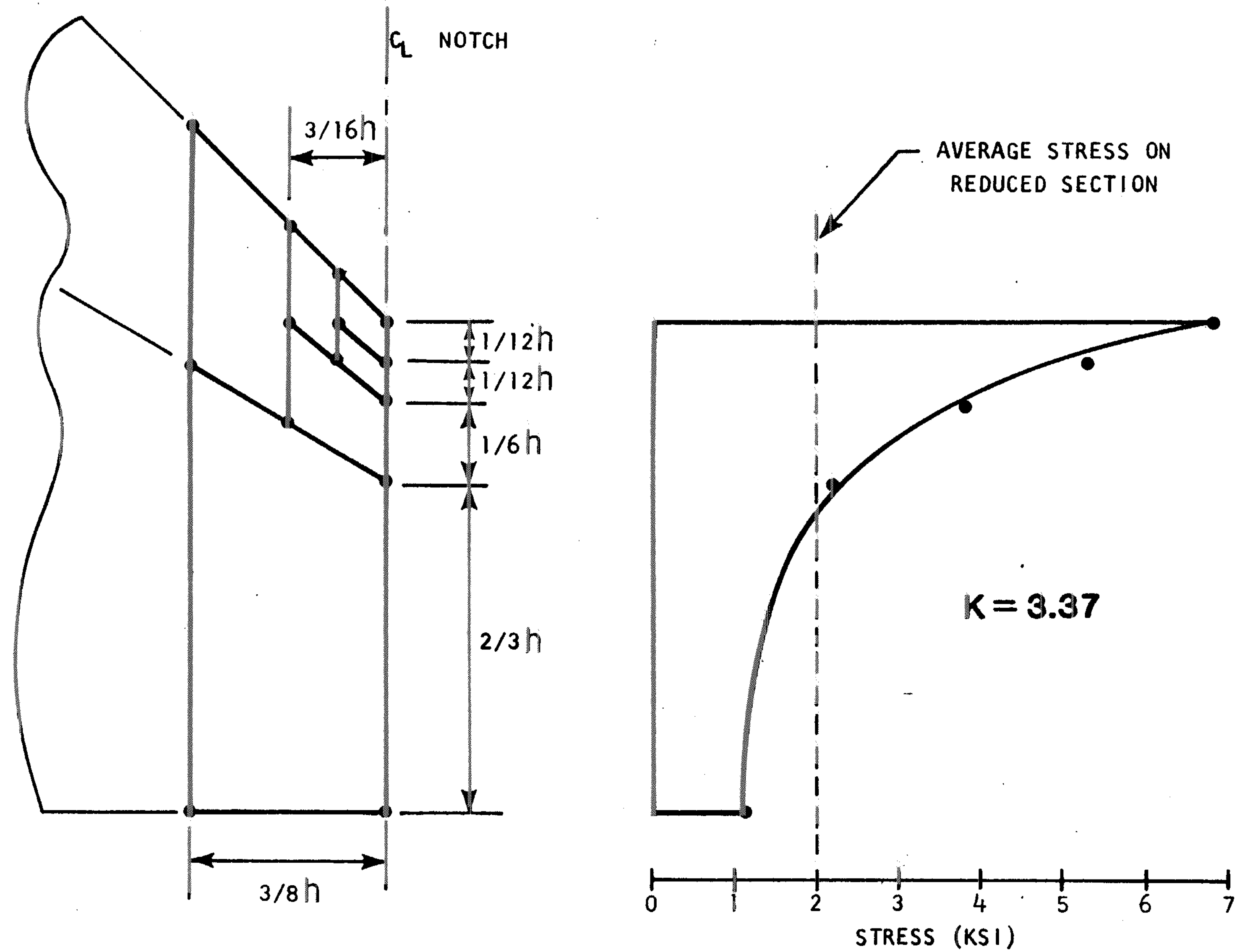


FIGURE 19. STRESS DISTRIBUTION FOR CASE 10