Non-linear Plane Stress Analysis Using the Finite Element Method

1986

Habib Amirzadeh
University of Central Florida

Find similar works at: http://stars.library.ucf.edu/rtd

University of Central Florida Libraries http://library.ucf.edu

Part of the Engineering Commons

STARS Citation


This Masters Thesis (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Retrospective Theses and Dissertations by an authorized administrator of STARS. For more information, please contact lee.dotson@ucf.edu.
NON-LINEAR PLANE STRESS ANALYSIS
USING THE FINITE ELEMENT METHOD

BY

HABIB AMIRZADEH
B.S., University of Central Florida, 1982

RESEARCH REPORT

Submitted in partial fulfillment of the requirements for the degree of Master of Science in Engineering in the Graduate Studies Program of the College of Engineering University of Central Florida Orlando, Florida

Fall Term 1986
ABSTRACT

Purpose of the Study

The objective of this report is to check the Neuber and Linear rules, and to find the percent difference in comparison with the plane stress and plane strain solution using the non-linear finite element method. All the data are based on the maximum stress concentration where the crack forms and the material behaves non-linearly. This particular problem has been solved using ANSYS, which is a general purpose finite element program.

Life Prediction

Another purpose, which is not the main subject of this report, is to use the result from Neuber and Linear rules to predict the life of the structure in the crack initiation phase, using the Strain-Life curve (refer to chapter II for more detail).
# TABLE OF CONTENTS

LIST OF TABLES ........................................ iv

LIST OF FIGURES ....................................... v

INTRODUCTION TO THE NEUBER AND LINEAR RULES ........ 1
  Neuber's Rule ...................................... 1
  Linear Rule ....................................... 2

THEORY .................................................. 4
  General Stress and Strain Equations ................. 4
  Non-Linear Analysis ................................ 7
  Strain-Life Prediction .............................. 9

INTRODUCTION .......................................... 13
  Introduction to the Finite Element Method .......... 13
  Introduction to ANSYS .............................. 14
  Routines Descriptions .............................. 15
  Model Description ................................ 16
  Program Model ..................................... 16
  Linear Analysis ................................... 17
  Non-linear Analysis ............................... 18

RESULTS AND ANALYSIS .................................. 21
  Linear Analysis ................................... 21
  Non-Linear Analysis, Plane Stress ................. 21
  Non-Linear Analysis, Plane Strain ................. 22

CONCLUSION AND RECOMMENDATION ...................... 28
  Linear Analysis ................................... 28
  Non-Linear Analysis, Plane Stress ................. 28
  Non-Linear Analysis, Plane Strain ................. 29
  Further Research ................................ 30

APPENDIX ................................................. 31
  Appendix I ........................................ 31
  Appendix II ....................................... 33

REFERENCE .............................................. 36
LIST OF TABLES

1. The Maximum Equivalent Stress (psi) in the Structure, Using the Plane Stress Solution . . . . . . . . . . . . . . . . . . . 26

2. The Maximum Equivalent Stress (psi) in the Structure, Using the Plane Strain Solution . . . . . . . . . . . . . . . . . . . 27
LIST OF FIGURES

1. Stress and Strain Curve Showing the Linear and Neuber's Rule ........................................... 3
2. Rectangular Shape Elements ............................................. 10
3. Displacement Vector of the Element e .................. 10
5. Strain-Life Curve, Showing Total Elastic and Plastic Components ............................................ 12
6. ANSYS Routines Flow Chart ............................... 19
7. The Original Model .................................................. 20
8. The Finite Element Model ............................................ 20
9. Maximum Equivalent Stress vs Load Curve for Non-Linear Plane Stress Solution .................. 24
10. Maximum Equivalent Stress vs Load Curve for Non-Linear Plane Strain Solution ........... 25
11. The Complete Model .................................................. 31
12. Stress Concentration Region .................................. 32
INTRODUCTION TO THE NEUBER AND LINEAR RULES

Neuber's Rule

Neuber has derived an equation for longitudinally grooved shafts in torsion which agrees with measurements in the plane stress situation, such as thin sheets in torsion, and because of that, it was primarily used for solving the plane stress problem.

The Neuber's rule is:

\[ \varepsilon \sigma = (K_t)^2 \varepsilon S = \sigma_y \text{ if } S < \sigma_y \text{ (yield strength)} \]
\[ \varepsilon \sigma = K_\sigma S K_e \varepsilon = K_t S K_t \frac{S}{E} = \frac{\sigma_y^2}{E} \]
\[ \varepsilon \sigma = \frac{\sigma_y^2}{E} \]  
(Equation 1)

where:

\[ S = \text{Nominal Engineering Stress} \]
\[ \sigma_y = \text{True Elastic Stress} \]
\[ \varepsilon = \text{Nominal Engineering Strain} = \frac{\sigma}{E} \]
\[ \sigma = \text{True Stress} \]
\[ \varepsilon = \text{True Strain} \]
\[ \sigma_y = \text{Elastic Strain} \]
\[ \varepsilon = \text{Elastic Strain} \]
\[ \varepsilon_p = \text{Plastic Strain} \]
\[ K_t = \text{Elastic Stress Concentration Factor} = \frac{\sigma_y}{S} \]
\[ K_\sigma = \text{Stress Concentration Factor} = \frac{\sigma}{S} \]
\[ K_e = \text{Strain Concentration Factor} = \frac{\varepsilon}{\varepsilon_y} \]
\[ E = \text{Young's modulus} \]

The non-linear material response equation is:

\[ \varepsilon = \varepsilon_e + \varepsilon_p = \frac{\sigma}{E} + \left( \frac{\sigma}{K} \right)^{1/n} \]  
(Equation 2)

where K and n are constant, that vary with the type of
material. Equation 1, and 2 have to be solved simultaneously to result in the Neuber's solution.

When $K_t * S$ is greater than the yield stress $S_y$, the material yields, and then the stress is not proportional to the strain, and $K_t$ is not equal to $K_\sigma$, and $K_\varepsilon$ anymore.

Neuber's rule is mainly constructed on one equation which says that the square of the elastic stress concentration factor ($K_t$) is equal to stress concentration factor ($K_\sigma$) multiplied by strain concentration factor ($K_\varepsilon$), or $K_t = (K_\sigma)^2 * K_\varepsilon$, which he derived for longitudinally grooved shafts in torsion.

Linear Rule

The Linear rule agrees with measurements in plane strain situations, such as circumferential grooves in shafts in tension or bending and it is used for solving the problems which consider to be the plane strain. It primarily says that the strain in the plastic region is the same for the elastic and plane strain equations, at the same load or $K_t = K_\varepsilon$.

Figure 1 shows both rules. Line BC represents the Linear rule with constant strain (plane strain), line AB represents the fictitious elastic stresses and strains, and curve BD represents the Neuber's rule. Point B is the linear elastic solution, point C is the linear rule solution, and point D is the Neuber's solution.
Figure 1. Stress and strain curve showing the Linear and Neuber's rule. Material Aluminum 2024-T4 with $E = 10600$ KSI, $K = 117$ KSI, and $n = 0.2$ (reference 1).
THEORY

General Stress and Strain Equations

For uniaxial loading, stress is related to strain by equation:

\[ E = \frac{\sigma}{\varepsilon} \]

and this is applicable within the linear-elastic range.

For the X-direction loading \( \sigma_x = E \varepsilon_x \), and this is known as Hooke’s law.

For a plane stress problem of an isotropic material, the stress and strain relationships are:

\[ \varepsilon_x = \frac{\sigma_x}{E} - v(\sigma_y/E) \]
\[ \varepsilon_y = -v(\sigma_x/E) + \sigma_y/E \]
\[ \gamma_{xy} = \tau_{xy}/G \]

which are linearly dependent upon \( \sigma_x \) and \( \sigma_y \).

For plane stress the assumptions are \( \sigma_z, \tau_{xz}, \) and \( \tau_{yz} \) are zero, and the stress is a two dimensional state of stress in the XY plane (except \( \varepsilon_z \neq 0 \)) in matrix form:

\[
\begin{pmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{pmatrix} = \frac{1}{E}
\begin{pmatrix}
1 & -v & 0 \\
-v & 1 & 0 \\
0 & 0 & 2(1+v)
\end{pmatrix}
\begin{pmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{pmatrix}
\]
solving for stresses:

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}
= \frac{E}{(1-\nu)^2}
\begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & (1-\nu)/2
\end{bmatrix}
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\]

This equation is valid for the case of plane stress.

For the plane stress case the real structure should be very thin. As a rule of thumbs, the structure is considered to be in plane stress, if the thickness is less than 10 percent of the length or width (which ever is less).

The following assumption is made for the plane strain solution:

\[
\varepsilon_z = \gamma_{xz} = \gamma_{yz} = 0
\]
\[
\sigma_z \neq 0 \quad , \quad \tau_{xz} = \tau_{yz} = 0
\]

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix}
= \frac{E}{(1+\nu)(1-2\nu)}
\begin{bmatrix}
(1-\nu) & \nu & 0 \\
\nu & (1-\nu) & 0 \\
0 & 0 & (1-2\nu)/2
\end{bmatrix}
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\]

\{ \sigma \} = [D] \{ \varepsilon \}

This equation is valid for the case of plane strain.

The theory of plane strain is for a structure having thickness relatively large compared to the length and width.

The typical element e of rectangular shape (Figure 2) shown is isolated in Figure 3. The strain and the stress are defined uniquely in terms of displacement functions. The strain matrix is of the form:
The relation between the strain and displacement is:

\[ ( \varepsilon ) = [ B ] ( d ) \]

and:

\[ ( \sigma ) = [ D ] [ B ] ( d ) \]

The force, displacement relation for an element is:

\[ ( F e ) = [ K e ] ( U e ) \]

where:

\[ K e = \int_{V} [ B ]^T [ D ] [ B ] dV \]

Leading to the force, displacement relation for structure under a given load:

\[ ( F ) = [ K ] ( U ) \]

where:

\[ [ K ] = \text{structural stiffness matrix} = \sum_{i=1}^{N.E.} [ K e ] \]

\[ ( U ) = \text{nodal point displacement vector} \]

\[ ( F ) = \text{Total load Vector} \]

This equation can be solved in a single iteration for a linear analysis, but when nonlinearities are present in the analysis, these same equations must be solved repetitively.

\[ ([ K ])(U) = (F) - [ S ](U) \quad (\text{for non-linear analysis}) \]
\[ n \text{ N.E. pl } (F) = (F) - \sum_{i=1}^{N} ([F_e]) \text{ plasticity} \]

where:
\[ \sum_{i=1}^{N} ([F_e]) = [S][U] \]

\( (N.E. = \text{ Number of Elements}) \)

The \([K]\) factors are stiffness coefficients relating the nodal deflections and forces, and calculated by the finite element program from material properties such as Young's modulus and Poisson's ratio, and from the element geometry.

**Non-Linear Analysis**

A non-linear analysis due to material characteristic of Figure 4 always requires an iterative solution to reach the optimum value.

Figure 4 is representing the equivalent stress vs. the equivalent strain which the equivalent stress for any three dimensional structure is:

\[ \sigma_{\text{equ.}} = \left( \sigma_1^2 + \sigma_2^2 + \sigma_3^2 - \sigma_1 \sigma_2 - \sigma_2 \sigma_3 - \sigma_1 \sigma_3 \right)^{1/2} \]  
(Equation 3)

In the absence of \(\sigma_2\) in the plane stress problem, only \(\sigma_1\) and \(\sigma_2\) are existed as principal stresses (\(\sigma_3 = 0\)).

The maximum distortion energy theory of failure finds considerable experimental support in situations involving ductile materials and plane stress. For this reason, it is in common use in design (reference 3).

The convergence criteria may cause an iterative solution to terminate before the specified maximum number of iterations.
within the load step is reached. If convergence occurs within the load step, the analysis then proceeds to the next load step (if any).

An example of two iteration procedure is described here:

First load step, and first iteration

\[
\{F_l\} = \text{input load} \\
\{F_l\} = [K]\{U_l\} \implies \{U_l\} = [K]^{-1}\{F_l\}
\]

by knowing \{U_l\}, \{\varepsilon_1\} is also known. And by knowing \{\varepsilon_1\}, \{\sigma_{el}\} is known (point A in Figure 4). Using equation 2, substituting for \{\varepsilon_1\}, \{\sigma_{1}\} can be found (point B in Figure 4). By knowing point B, point E and G can be found too. OE is equal to \(\varepsilon_{pl}\) and \(\Delta \varepsilon_{pl}\) and EG is equal to \(\varepsilon_{el}\). At point B if true stress \(\{\sigma_{1}\}\) is not equal to force divided by the area \((F/A)\) or \(\Delta \varepsilon_{pl}/ \varepsilon_{el}\) is not less than .01 (chosen by default by the program, and it can be changed at any time), then the program would automatically go to the next iteration.

For the second iteration, program will recalculate the force and displacement.

\[
\{U_2\} = \{U_l\} - \{U_{p2}\}
\]

by knowing \{U_2\}, \{\varepsilon_2\} and \{\sigma_{e2}\} (point C) and \{\sigma_{2}\} (point D) are known. Point F and H can easily be found.

OF is \(\varepsilon_{p2}\), FH is \(\varepsilon_{e2}\), and EF is \(\Delta \varepsilon_{p2}\) (Figure 4).

If \(\Delta \varepsilon_{p2}/ \varepsilon_{e2}\) is less than .01 the program would converge and would go to the next load step, and if it is not, the third load step would begin.
Life Prediction

Strain-Life curves are more commonly used, because the strains can be calculated more easily than the stress, as long as an elastic constraint is surrounding a local plastic zone at the notch. The Strain-Life curves are often called low cycle fatigue data because much of the data are for less than about $10^5$ cycles.

$$\frac{\Delta \varepsilon}{2} = \frac{\Delta \varepsilon_e}{2} + \frac{\Delta \varepsilon_p}{2}$$

$$\frac{\Delta \varepsilon}{2} = \frac{\sigma'_{f}}{E} (2N) + \varepsilon'_{f} (2N)$$

where:

- $\Delta \varepsilon$ = Total Strain Amplitude
- $\Delta \varepsilon_e$ = Elastic Strain Amplitude
- $\Delta \varepsilon_p$ = Plastic Strain Amplitude
- $\varepsilon'_f$ = Fatigue Ductility Coefficient
- $\sigma'_f$ = Fatigue Strength Coefficient
- $c$ = Fatigue Ductility Exponent
- $b$ = Fatigue Strength exponent
- $E$ = Modulus of elasticity
- $N$ = The number of cycles that a given specimen sustains before any failure occurs.

Figure 5 taken from reference 1 shows an example of Strain-Life curve.
Figure 2. Rectangular shape elements.

Figure 3. Displacement vectors of the element e.
Figure 4. Non-linear analysis, iteration procedure.
Figure 5. Strain-Life curve, showing total elastic and plastic components.
INTRODUCTION

Introduction to the Finite Element Methods

For situations in which the cross section area is constant throughout the whole structure, reasonably accurate results can be predicted, by applying the equations derived on the basis of constant section. On the other hand, where some changes existed in the cross section, those equations cannot predict the maximum stress concentration in the structure. The changes are defined as holes, fillets, and notches, which are most likely to be the starting material failure points. Today, by using experimental techniques, such as strain gages, or photoelasticity methods, one can measure the high stresses in the structure, but sometimes due to high cost of testing, unavailability of any sample, and lack of time, these experiments are not possible. One alternative available for an engineer to predict the life of the structure, is using the finite element methods. The finite element method of stress analysis is no longer an engineering nightmare. Many of the tedious, time-consuming, and confusing steps have been replaced with automated computer operations. Models are easier to build, element arrays can be digitized more readily, and answers come in the form of easy to read stress plots.
instead of endless tables of numbers. All of these developments come together with the experimental methods, so that the behavior of a total machine or vehicle can be predicted before a prototype is built.

The power of today's computers has revolutionized structural design, with many general-purpose programs available to solve equations for stress and strain in solid structures. Accuracy of these programs varies with problem type, and the user must be aware of limitations. For example, linear models are well understood and computed results usually compare well with the actual conditions, but the non-linear analysis is a different matter.

An important point to remember with these programs, as with any tool, is the quality of the answers depend on the skill of the user. The user does not need to be a computer programmer or an expert in numerical methods, but a proficiency in solid mechanics and heat transfer is required.

**Introduction to ANSYS**

Finite element analysis (FEA) is a computer-based technique for determining stresses and deflections in a structure too complex for classical analysis.

The finite element method is based on arrays of large matrix equations that can only be realistically solved by computer. Most often, FEA is performed with commercial programs. One of these programs is ANSYS.
ANSYS is a widely used finite element analysis program created by Swanson Analysis System, Inc. Houston, PA.

ANSYS, as now written, revision 4.2A, can make static, dynamic, and field analysis. Static analysis cover linear, non-linear, and buckling variations. Dynamic analysis encompass natural frequencies, forced and random vibration, linear and non-linear transient behavior. Heat transfer and electromagnetic fields are included in field analysis. ANSYS also solves axisymmetry elements for both structural and heat transfer.

ANSYS has been developed to simplify and reduce the cost of computer-implemented finite element analysis, and may be applied to a variety of engineering problems. It is structured to reduce the computer programming knowledge and experience required for using it. The intent is to utilize the user's knowledge of finite element analysis and the problem area rather than expertise in programming methods.

**Routines Descriptions**

A typical ANSYS analysis is done through the different routines in three phases (shown in Figure 6):

1. Preprocessing: These routines are used to define the model, boundary conditions, and loading.

2. Solution: This part of the program takes the model data from previous routines, forms the necessary matrixes, and solves the equations for stresses, displacements, reaction forces, and etc.

3. Postprocessing: These routines are mostly used for sorting, printing, plotting, and combining the results.
Model Description

All the analysis have been done on the circular fillet in the axially loaded member shown in Figure 7. The width of the member changes from 8 inches to 4 inches while the thickness remains constant at .50 inch. The member is Aluminum 2024-T4 that has a Young's modules of 10600 ksi and Poisson's ratio of .25. The material is assumed to be isotropic, homogeneous, and initially it is entirely free of any residual stresses. The length on both sides of the member were changed several times to develop a uniform stress distribution. All the points far to the right were subjected to the zero displacement in the X direction, while the points to the left were subjected to a uniformly distributed load of 5000 to 15000 psi. Because the problem was symmetric about the X axis in the shape and loading, only half of the member was modeled for the analysis. All the points along the bottom edge were constraint to the zero displacement in the Y direction.

Figure 7 shows all the dimensions, Figure 8 is the model used for the analysis.

Since the constant K value was not available for cyclic loading, all the constants were selected for the monotonic loading. As shown in Figure 1, .2 percent off-set of the yield strength (about 35000 psi) was used for the analysis.

Program Model

The model consist of 374 nodes and 310 elements. Because stress accuracy varies as a function of mesh density, finer
meshes have been created around the critical areas of the structure. The reasons are, first a finer mesh allows for better definition of component geometry details, and second, a finer mesh permits more stress calculation points to define the stress gradient in a notch or fillet region.

Stresses are calculated at the element's centroid, since they are based on the nodal displacements which surround the centroid.

The 2-D Isoparametric solid was used as element type (Figure 3), because the elements are defined by four nodal points having two degrees of freedom at each node (displacement in the nodal X and Y directions), the non-linear option, plane stress and strain option, and also plane stress with thickness input option (for future use). Each element has a total of eight degrees of freedom.

**Linear Analysis**

The ANSYS static analysis is used to solve for the displacements, stresses and forces in the structure under the action of applied load. The material is generally assumed to be linearly elastic but special cases such as plastic deflection, creep, and large deflections can be handled in some instances.

First the program was run, using linear static analysis, to check the capability and accuracy of the system, by comparing the result with theoretical value and other programs previously run. The results are shown in Chapter IV.
Non-Linear Analysis

There are 11 different load steps in the non-linear program, because a small step increment load would give more accurate results. Loading starts from 5000 psi and finishes at 15000 psi with the increment of 1000 psi at each load step.

To represent the non-linear curve (Equation 2) as accurate as possible in the program, just 5 points were allowed by the program to be inputted, because of the program limitations. This representation of five straight lines, would cause a small error, and after some investigation and selecting different points, it was found that the error is too small which can be neglected. The 5 points are consisted of stresses, 5000, 15000, 25000, 40000, 55000 psi, and by solving equation 2 the strains become .000472, .00145, .0028, .0084, .0281 in/in.

The commands from /PREP7 to FINISH in the program are part of the preprocessing (refer to Appendix II for program printout), three lines after that are part of the solution phase, and the rest are the postprocessing part of the program.
Figure 6. ANSYS routines flowchart.
Figure 7. The original model.

Figure 8. The finite element model. (the elements are shown in Appendix I)
RESULT AND ANALYSIS

Linear Analysis

The elastic stress concentration factor \((K_t)\) for this particular problem is \(K_t=1.42\) given by Singer, and the theoretical value for maximum stress becomes:

\[
\sigma_{\text{MAX}} = K_t \times S = 1.42 \times 30000 = 42600 \text{ psi}
\]

Other finite element programs, by using the triangular elements, have reached the maximum value of 44024 psi, with a difference of about 3.3% from theoretical. The maximum value reached using linear analysis option (plane stress) of ANSYS was 44203 psi at element 198 (the element numbers are shown in Appendix I), which is very close to the other programs previously ran, and within 3.8% of theoretical value of 42600. The maximum equivalent stress for the structure was 43910 psi at element 198 (using the plane stress solution).

The static linear analysis was run to check the accuracy of the model created, the input loading, and the boundary conditions chosen. The result was compared well.

Non-Linear Analysis, Plane Stress

The same model was used for running the nonlinear program. The following formula has been used to represent the
non-linear curve for the Aluminum 2024-T4 (assuming 2D model):

\[ \sigma_{\text{equ.}} = \sigma_{\text{equ.}}/E + \left( \sigma_{\text{equ.}}/K \right)^{1/n} \]  
(Equation 2)

\[ \sigma_{\text{equ.}} = 44203 \text{ psi} \] (will be different for variable loads)

\[ E = 10600000 \text{ psi} \]

\[ n = 0.2 \]

\[ K = 117000 \text{ psi} \]

\[ E, n, \text{ and } K \] are the constants for Aluminum 2024-T4 with monotonic strain properties taken from reference 1, since the \( K \) value was not available for cyclic loading.

The result is shown on Table 1 and Figure 9. Figure 9 shows the plot of equivalent stress vs. load of result from Neuber's rule and ANSYS analysis. It appears that up to the yield strength of the material (35000 psi) the two curves (ANSYS and Neuber's) are very close, maximum difference of about 3 percent.

Neuber's values were calculated by solving Equations 1 and 2 simultaneously for different loads. The separation starts getting wider and wider as the part passes the yield strength (the results are shown up to 15000 psi input load).

Non-Linear Analysis, Plane Strain

The program was also run to check the linear rule (plane strain). The results are shown on Table 2 and Figure 9. As we can see similar to the plane stress, the results are very close between the Linear rule and ANSYS up to the yield strength (Table 2), and after that the difference would go to zero
and starts rising as it gets more into the plasticity range. (Figure 10).
Figure 9. The maximum equivalent stress vs load curve for non-linear plane stress solution.
Figure 10. The maximum equivalent stress vs load curve for non-linear plane strain solution.
TABLE 1. THE MAXIMUM EQUIVALENT STRESS (psi) IN THE STRUCTURE, USING THE PLANE STRESS SOLUTION.

<table>
<thead>
<tr>
<th>LOAD</th>
<th>ELAS. STRESS</th>
<th>NEUBER'S</th>
<th>ANSYS</th>
<th>% DIFF.</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>5000</td>
<td>14734</td>
<td>14575</td>
<td>14553</td>
<td>-0.2%</td>
</tr>
<tr>
<td>6000</td>
<td>17681</td>
<td>17309</td>
<td>16938</td>
<td>-2.1%</td>
</tr>
<tr>
<td>7000</td>
<td>20628</td>
<td>19890</td>
<td>19314</td>
<td>-2.9%</td>
</tr>
<tr>
<td>8000</td>
<td>23575</td>
<td>22283</td>
<td>21753</td>
<td>-2.4%</td>
</tr>
<tr>
<td>9000</td>
<td>26522</td>
<td>24482</td>
<td>24483</td>
<td>0.0%</td>
</tr>
<tr>
<td>10000</td>
<td>29468</td>
<td>26486</td>
<td>26044</td>
<td>-1.7%</td>
</tr>
<tr>
<td>11000</td>
<td>32415</td>
<td>28314</td>
<td>27473</td>
<td>-3.0%</td>
</tr>
<tr>
<td>12000</td>
<td>35362</td>
<td>29985</td>
<td>29048</td>
<td>-3.1%</td>
</tr>
<tr>
<td>13000</td>
<td>38309</td>
<td>31520</td>
<td>30724</td>
<td>-2.5%</td>
</tr>
<tr>
<td>14000</td>
<td>41256</td>
<td>32937</td>
<td>32915</td>
<td>-0.1%</td>
</tr>
<tr>
<td>15000</td>
<td>44203</td>
<td>34253</td>
<td>35308</td>
<td>3.1%</td>
</tr>
<tr>
<td>LOAD</td>
<td>ELASTIC STRESS</td>
<td>LINEAR'S</td>
<td>ANSYS</td>
<td>% DIFF.</td>
</tr>
<tr>
<td>-------</td>
<td>----------------</td>
<td>----------</td>
<td>--------</td>
<td>---------</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>5000</td>
<td>14734</td>
<td>14430</td>
<td>13029</td>
<td>-9.7%</td>
</tr>
<tr>
<td>6000</td>
<td>17681</td>
<td>17000</td>
<td>15500</td>
<td>-8.8%</td>
</tr>
<tr>
<td>7000</td>
<td>20628</td>
<td>19330</td>
<td>17551</td>
<td>-9.2%</td>
</tr>
<tr>
<td>8000</td>
<td>23575</td>
<td>21390</td>
<td>19670</td>
<td>-8.0%</td>
</tr>
<tr>
<td>9000</td>
<td>26522</td>
<td>23250</td>
<td>21839</td>
<td>-6.1%</td>
</tr>
<tr>
<td>10000</td>
<td>29468</td>
<td>24860</td>
<td>24267</td>
<td>-2.4%</td>
</tr>
<tr>
<td>11000</td>
<td>32415</td>
<td>26320</td>
<td>25471</td>
<td>-3.2%</td>
</tr>
<tr>
<td>12000</td>
<td>35362</td>
<td>27600</td>
<td>26240</td>
<td>-4.9%</td>
</tr>
<tr>
<td>13000</td>
<td>38309</td>
<td>28770</td>
<td>27091</td>
<td>-5.8%</td>
</tr>
<tr>
<td>14000</td>
<td>41256</td>
<td>29830</td>
<td>27944</td>
<td>-6.3%</td>
</tr>
<tr>
<td>15000</td>
<td>44203</td>
<td>30760</td>
<td>29233</td>
<td>-5.0%</td>
</tr>
</tbody>
</table>
CONCLUSION

Linear Analysis

The result of the finite element program (ANSYS) was very close to the theoretical value (found by Singer) (Chapter IV) for perfectly elastic material. It is feasible to use the simplified solution for this type of problem rather than generating the linear finite element model, specially for the first approximation or for the larger models.

Non-Linear Analysis, Plane Stress

In conclusion, the Neuber's rule can be applied for plane stress problem with confidence up to the yield strength of the material. For this particular problem for Aluminum 2024-T4 with the yield strength of about 35000 psi, the Neuber's solution was compared well with the finite element result (ANSYS) as it was expected. The separation of the two curves is increasing as the load increases.

It is very unlikely to use any solution beyond the yield strength of the material by any engineer because of the permanent deformation. The failure by yielding is predicted by the distortion energy theory (or maximum normal stress theory) whenever:

\[ \sigma_{\text{equ.}} = \sigma_y \]
Similarly, safety is predicted by:

\[ \sigma_{\text{equ.}} = \frac{S_y}{n} \]

where \( n \) is the factor of safety.

Using the Neuber's rule will save a lot of time, consumed in modeling and mesh generating for the non-linear finite element solution. It is conservative and accurate enough for the life prediction (since by knowing the stress, the strain can be found for using the strain-life curve), and other analysis done by an engineer.

Fortunately, many of the problems encountered in practice are such that they can be considered plane stress problems.

**Non-Linear Analysis, Plane Strain**

The result from ANSYS for plane strain solution was very close to the Linear rule. As for the case of plane stress it would be wasteful, and time-consuming to use the non-linear finite element program to solve for plane strain problem rather than using the simple Linear rule solution.

we have to accept the fact that the result of the plane strain solution may not be lower for every single case than the plane stress solution, and it has not been clearly stated by any author. And because of that, it would be very helpful to run the finite element program (if it has not been done before) once and check the result before making any conclusion.
Further Research

More research can be done on this subject by investigating the axisymmetric problem and comparing the result with the plane stress and plane strain solution. This can be done easily by using the program in Appendix II (non-linear) and changing some commands.

Another subject which is very important is to check the effect of a number of elements and nodes in the solution. This can be done on a 2D or 3D model by changing the number of elements and nodes and finding the maximum or minimum element size for which the result would compare well with the actual test data (simple bar in tension).
Figure 11. The complete model.
Figure 12. Stress concentration region.
Appendix II

Program Print Out

Linear Model

1 /PREP7
2 /TITLE, LINEAR ANALYSIS, PLANE STRESS
3 ET, 1, 42,,0
4 EX, 1, 1.06E7
5 NUXY, 1, .25
6 LOCAL, 11, 1, 09, 4
7 SF, 1, 11, 1, 2
8 CSYS, 0
9 PT, 1,,05, 4
10 PT, 2, 1,,07, 4
11 PT, 3, 1,,09, 2
12 PT, 4,,09, 0
13 PT, 8,,05, 0
14 REGS, 11, 21
15 CSCIR, 11, 1
16 LS, 1, 8, 10
17 LS, 6, 4, 10
18 LS, 2, 1, 10, 11
19 LS, 3, 4, 10, 11
20 AREA, 1, 2, 3, 4
21 PT, 5,,14, 2
22 PT, 6,,14, 0
23 PT, 7,,1, 0
24 PT, 9,,1, 4
25 REGS, 11, 9
26 LS, 5, 6, 10, 11
27 AREA, 6, 5, 3, 4
28 REGS, 4, 11
29 AREA, 9, 1, 6, 7
30 MERGE
31 ITER, 1,,
32 KRF, 2
33 D, 371, UY,,373, 1
34 D, 232, UX,,242, 1
35 D, 111, UY,,221, 11
36 D, 232, UY,,309, 11
37 CONV, 1
38 PSF, 0, 1, 1,-22000
39 LWRITE
40 AFWRITE
41 FINISH
42 /EXEC
43 /INPUT, 27
44 FINISH
45 /POST1
46 STRESS, STA, 42, 51
47 SET, 1, 1
48 ESORT, STA
49 PRSTRS
50 FINISH
Non-Linear Model

1 /PREP7
2 /TITLE,NON LINEAR ANALYSIS, PLANE STRESS
3 ET,1,42,,0
4 EX,1,1.06E7
5 NUXY,1,.35
6 KNL,1
7 LOCAL,11,1,09,4
8 SF,1,11,1,2
9 CSYS,0
10 PT,1,,,,05,4
11 PT,2,1,,,,07,4
12 PT,3,1,,,,09,2
13 PT,4,,,,09,0
14 PT,6,,,,05,0
15 REGS,11,21
16 CSCIR,11,1
17 LS,1,8,10
18 LS,8,4,10
19 LS,2,1,10,11
20 LS,3,4,10,11
21 AREA,1,2,3,4
22 PT,5,,,,14,2
23 PT,6,,,,14,0
24 PT,7,,,,1,0
25 PT,9,,,,1,4
26 REGS,11,9
27 LS,5,6,10,11
28 AREA,6,5,3,4
29 REGS,4,11
30 AREA,9,1,8,7
31 MERGE
32 ITER,100
33 KRF,2
34 D,371,UY,,,,373,1
35 D,232,UX,,,,242,1
36 D,111,UY,,,,221,11
37 D,232,UY,,,,309,11
38 NL,,14,.00047169,.00144973,.002804,.006444,.0281441
39 NL,,13,17
40 NL,,19,0,5000,15000,25000,40000,55000
41 NL,,25,100,5000,15000,25000,40000,55000
42 CONV,1
COM, LOAD STEP 1
PSF, 0, 1, 1, -5000
LWRITE
ITER, 100
CONV, 1
/COM, LOAD STEP 2
PSF, 0, 1, 1, -6000
LWRITE
ITER, 100
CONV, 1
/COM, LOAD STEP 3
PSF, 0, 1, 1, -7000
LWRITE
ITER, 100
CONV, 1
/COM, LOAD STEP 4
PSF, 0, 1, 1, -8000
LWRITE

AFWRITE
FINISH
/EXEC
/INPUT, 27
FINISH

POST1
STRESS, STA, 42, 47
SET, 01, 100
ESORT, STA
PRSTRS
FINISH
/POST1
STRESS, STB, 42, 47
SET, 02, 100
ESORT, STB
PRSTRS
FINISH
/POST1
STRESS, STC, 42, 47
SET, 03, 100
ESORT, STC
PRSTRS
FINISH
/POST1
STRESS, STD, 42, 47
SET, 04, 100
ESORT, STD
PRSTRS
FINISH


