ABSTRACT
The objective of the project is to introduce the students to the concept of plasticity in materials. In particular, the strain hardening occurring in the material when subjected to stresses beyond elastic limit is of interest. A brief explanation of the theory of plasticity has been provided along with a general methodology of numerical implementation, which is also employed in FEM packages. To illustrate the same, an example of an interference fit between a shaft and a bushing assembly has been considered. The plasticity behavior of the shaft has been simplified into a bilinear isotropic hardening behavior and the interference fit has been simulated in ANSYS.

INTRODUCTION
The importance of nonlinear material behavior effects numerous practical applications. While the assumption of linear behavior leads to a reasonable idealization of structural behavior, there are situations where nonlinear effects must be incorporated for a more realistic assessment of the structural response. Some of the reasons for nonlinear analysis include the use in assessment of existing structures whose integrity may be in doubt, the identification of the causes of structural failure, and the use in design for ultimate load and serviceability limit states [9]. The aim of this paper is to illustrate plasticity using the example of a finite element analysis of interference fit that undergoes bilinear plastic hardening behavior.

PLASTICITY MODEL THEORY
The term plasticity is used to describe the elasto-plastic behavior of a material that has been loaded beyond its yield strength. Such material would remain permanently deformed after unloading. Plasticity can further be described as rate independent or rate dependent. Rate independent means plastic strain is assumed to develop instantaneously (independent of time). There are several ways for characterizing material behaviors, such as bilinear or multi-linear. The bilinear stress-strain curve (Figure 2) is used as an approximation to the more realistic multi-linear true stress-true strain curve (Figure 1).

Three major parts of the plasticity model theory are discussed here, they are: yield criterion, flow rule, and hardening rule. In this paper, the bilinear stress-strain curve is used to study the nonlinear behavior of the interference fit, which is also characterized by rate independent plasticity.

YIELD CRITERION
The yield criterion determines the stress level at which yielding of a material will occur. The yield criteria can be expressed in terms of the stress state, strain state, and strain energy quantity. Yield criterion, also known as von Mises, is mathematically expressed in terms of yield function, $F(\sigma_{ij}, \sigma_Y)$, where

$$F = \sigma_Y - \sigma_Y$$

Note that $\sigma_{ij}$ is the state of stress and $\sigma_Y$ is the yield strength in uniaxial tension (or compression). The formulation of the yield function is achieved by combining the multi-axial stress components (stress tensor $\sigma_{ij}$). The material
behavior is elastic when \( F(\sigma_{ij}, \sigma_Y) < 0 \), the yield function is satisfied when \( F(\sigma_{ij}, \sigma_Y) = 0 \), and undefined for \( F(\sigma_{ij}, \sigma_Y) > 0 \) \(^7\). The concept of yield surface is used to demonstrate the nature of yield criterion \(^9\) in 3D principal stress states, shown in Figure 3. For ductile materials, the effective stress \( (\sigma_e) \), which is in terms of the distortion energy density criterion (von Mises), can be written as shown below. The effective stress can be written alternatively in terms of the second invariant of deviatoric stress as:

\[
\sigma_e = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 - (\sigma_2 - \sigma_3)^2 - (\sigma_1 - \sigma_3)^2]}
\]

\[
\sigma_e = \sqrt{3f_2}
\]

FLOW RULE
Flow rule, also known as plastic straining, states that the movement of the yield surface is directly dependent on the plastic strain increment \(^6\). The plastic strain increment

\[
d_\varepsilon^{PL} = \lambda \frac{\partial Q}{\partial \sigma}
\]

is obtained by multiplying the plastic multiplier, \( \lambda \), with the plastic potential gradient, \( \frac{\partial Q}{\partial \sigma} \). The plastic multiplier indicates the magnitude of plastic deformation and it changes with loading. The plastic potential gradient indicates the direction of plastic straining. In associative flow rule, which is the type of flow rule pertaining to bilinear isotropic hardening behavior, the plastic potential gradient is normal to the yield surface. Since plastic potential gradient is directional, thus the plastic strain increment is also normal to the yield surface.

HARDENING RULE
The hardening rule describes the changing of yield surface with progressive yielding, which means that yield stress grows in magnitude as plastic deformation continues (or increases). The two fundamental hardening types are isotropic and kinematic. Isotropic hardening is where the yield surface size changes, but the center axis and the general shape of the yield surface do not change\(^5\). The uniform expansion of the yield surface can also be seen on a stress-strain diagram of an isotropic hardening behavior; the compressive and tensile yield strengths are increasing together by the same amount (Figure 5).

\[
\{a\} = \int [D][d_\varepsilon^{PL}]
\]

The zero change in yield surface size can also be seen in the stress-strain diagram of a kinematic behavior (Figure 7). The sum of the compressive and tensile yield strengths is constant.
Isotropic hardening behavior is usually first assumed for small strain deformation, whereas kinematic hardening behavior is for large strain deformation.

**CONSISTENCY CONDITION**

Consistency condition is the requirement that the stress state has to coincide with the yield surface as long as loading continues, which agrees with what was previously stated that the yield criterion changes with hardening. Yield surface will expand when \( \sigma_e > \sigma_y \), and the two values will converge until \( \sigma_e \) cannot be greater than \( \sigma_y \) anymore. During hardening, stress should always lie on the yield surface, which is governed by

\[
F + dF = 0
\]

Since \( F = 0 \), \( dF \) must also always equal to zero.

\[
dF = \left\{ \frac{\partial F}{\partial \sigma} \right\} d\sigma + \left\{ \frac{\partial F}{\partial \varepsilon^{pl}} \right\} d\varepsilon^{pl} = 0
\]

**GOVERNING EQUATIONS**

The strong and weak forms for the general material nonlinear equations are similar\(^8\) to the linear equations\(^2\) that were taught in class.

**Strong form:**

\[
\mathbf{D}^{pl} \varepsilon = -\mathbf{B} d
\]

**Weak form:**

\[
\sum_{e=1}^{nel} \int_{\Omega} (\mathbf{w}^T) D^{EPL} \mathbf{w} u d\Omega = \sum_{e=1}^{nel} \int_{\Gamma} \mathbf{w}^T t d\Gamma + \sum_{e=1}^{nel} \int_{\Omega} \mathbf{w}^T b d\Omega
\]

**Matrix form:**

\[
\mathbf{w}^T \left( \sum_{e=1}^{nel} L^T \int_{\Omega} [\mathbf{B}]^T [D^{EPL}] [\mathbf{B}] d\Omega \right) L d
\]

\[
= \int_{\Gamma} [\mathbf{N}]^T t d\Gamma - \int_{\Omega} [\mathbf{N}]^T b d\Omega \]

\[
= 0
\]

However the D-matrix, material parameter, is no longer linear because it changes at each time step due to possible plastic strain increment\(^6\). In this report, material parameter matrix is denoted as \([D^{EPL}]\), where

\[
[D^{EPL}] = [D] - \frac{\left\{ \frac{\partial F}{\partial \sigma} \right\} \left\{ \frac{\partial F}{\partial \varepsilon^{pl}} \right\}^T}{A + \left\{ \frac{\partial F}{\partial \sigma} \right\}^T [D] \left\{ \frac{\partial F}{\partial \varepsilon^{pl}} \right\}} [D]
\]

Where \( A = -\left\{ \frac{\partial F}{\partial \varepsilon^{pl}} \right\}^T \left\{ \frac{\partial F}{\partial \sigma} \right\} \)

The \([D^{EPL}]\) is a function of the linear material parameter \([D]\) with partial of yield surface with respect to partial of stress and strain at each time step. The second term of the equation takes possible plastic strain increment into consideration. Because of \([D^{EPL}]\), the following relationships will also be non-linear:

\[
\varepsilon = \psi_s u = [B] d
\]

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

\[
[K] = \int_{\Omega} [\mathbf{B}]^T [D^{EPL}] [\mathbf{B}] d\Omega
\]

\[
\{f\} = \int_{\Gamma} [\mathbf{N}]^T t d\Gamma + \int_{\Omega} [\mathbf{N}]^T b d\Omega
\]

**IMPLEMENTATION**

The dependence of stress on strain is no longer linear, but rather a nonlinear behavior. As such, a direct solution is not possible and a numerical approach is required. Since the plasticity and stress states also depend on the load history, it is generally advised to divide the load into small incremental load steps.

The basic procedure of the numerical implementation (see APPENDIX, Figure 11) involves the following steps\(^6\):

1. \( \sigma_y \) for the current step is determined
2. A trial displacement \( \Delta u \) is assumed and strain \( (\varepsilon^{tr}) \) and stress \( (\sigma^{tr}) \) corresponding to that displacement is calculated. \( \sigma_e \) is calculated from \( \sigma^{tr} \)
3. If \( \sigma_e < \sigma_y \), the step is elastic and step 9 is implemented, else step 4 is implemented.
4. The plastic multiplier \( \lambda \) is calculated using Newton Raphson iterative method such that consistency (dF=0) is maintained.
5. Plastic strain \( \Delta \varepsilon^{pl} \) is computed using \( \Delta \varepsilon^{pl} = \lambda \left\{ \frac{\partial F}{\partial \sigma} \right\} \)
6. Total plastic strain \( \varepsilon^{pl} = \varepsilon^{pl}_{prev} - \Delta \varepsilon^{pl} \) where \( \varepsilon^{pl}_{prev} \) is the plastic strain from the previous step
7. The elastic strain is updated as \( \varepsilon^{el} = \varepsilon^{tr} - \Delta \varepsilon^{pl} \)
8. The stress is computed using \( \sigma = [D^{pl}] \varepsilon^{el} \), and step 9 is implemented
9. Internal forces from the resulting stress is computed and checked against the applied load. If the out of balance load (Residual) is below a
threshold, the load step had converged. The stresses and strains are updated

10. If convergence is not achieved, the trial displacement is updated using Newton Raphson iterative method and the entire process from step 2 is repeated. This is done till convergence is achieved.

The above is one of the many possible techniques employed in several FEM packages.

**ANSYS**

To illustrate the behavior of plasticity, a simplified FEA was performed considering the example of an interference fit between a shaft and a bushing. The following assumptions have been made for simplification of the analysis

1. A plane stress behavior exists
2. The bushing is considered to be rigid
3. An elastic bilinear isotropic hardening behavior is observed in the shaft

Due to the symmetry of the system, a quarter model has been analyzed. Furthermore, in order to show a distinct comparison between elastic and plastic behavior in the contour plots of stresses, the shaft geometry has been assumed as hollow. The geometry and material inputs \(^{11}\) are shown in Table 1 and Table 2 below.

**Table 1: Geometric Inputs**

<table>
<thead>
<tr>
<th>Component</th>
<th>OD</th>
<th>ID</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shaft</td>
<td>10 in</td>
<td>6 in</td>
</tr>
<tr>
<td>Bushing</td>
<td>--</td>
<td>9.9 in</td>
</tr>
</tbody>
</table>

*OD - Outer Diameter; ID - Inner Diameter*

**Table 2: Material properties of the shaft**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>30 x 10^6 psi</td>
</tr>
<tr>
<td>ν</td>
<td>0.3</td>
</tr>
<tr>
<td>E\text{\textsuperscript{T}}</td>
<td>75000 psi</td>
</tr>
<tr>
<td>σ\text{\textsuperscript{Y}}</td>
<td>36500 psi</td>
</tr>
</tbody>
</table>

*E - Young’s Modulus; ν – Poisson’s Ratio; E\text{\textsuperscript{T}} – Tangent Modulus; σ\text{\textsuperscript{Y}} – Yield Strength*

As shown in Table 1, a radial interference of 0.05 led to plasticity in the shaft. Figure 8 shows a scaled version of the geometry used in the analysis. Figure 9 and Figure 11 show the von Mises stress result in the deformed configuration for the plastic analysis as well as a purely elastic analysis of the interference.

The von Mises stresses for the elastic analysis shows unrealistic stresses (Figure 10) with the maximum going up to 500,000 psi. The elastic plastic analysis, on the other hand, shows more realistic stress behavior (Figure 9) with stresses in the ball park of 37,000 psi.
MATLAB

An attempt has been made to implement bilinear isotropic hardening plasticity in MATLAB for 2D problems, in a simplified form \[^{[1][3][4][6][9][10]}\]. The codes provided for 2DBVP and 2D stress analysis have been used as basis. The code is running into convergence issues with the plasticity algorithm and subsequent Newton Raphson equilibrium iteration. As such the results have not been accurate and correlative of the ANSYS results (see APPENDIX, Figure 18 through Figure 21). A more robust formulation seems to be required for better convergence and accuracy.

CONCLUSIONS

Through this project study, the students obtained understanding of plasticity model theory in the parts of yield criterion, flow rule, and hardening rule. The importance and accuracy of material non-linear analysis were also realized. The governing equations for linear analysis and non-linear analysis were compared and distinguished. Following the learned numerical implementation, ANSYS model analysis was explained but MATLAB results were not able to converge.

REFERENCES

11. http://www.matweb.com
APPENDIX

Figure 11: Brief representation of numerical implementation of plasticity

Figure 12: Elastic-Plastic Analysis - Radial Stress

Figure 13: Elastic-Plastic Analysis - Tangential Stress

Figure 14: Elastic Analysis - Radial Stress

Figure 15: Elastic Analysis - Tangential Stress
Figure 16: Elastic-Plastic Analysis - Radial Displacement

Figure 17: Elastic Analysis - Radial Displacement

Figure 18: MATLAB Result: von Mises Stress

Figure 19: MATLAB Result - Radial Stress

Figure 20: MATLAB Result - Tangential Stress

Figure 21: MATLAB Result - Radial Displacement