THE FINITE ELEMENT ANALYSIS OF APEX THIN AND THICK WALLED HEXAGONAL DRIVE TOOL SOCKETS

A Thesis Presented to
The Faculty of the College of Engineering and Technology
Ohio University

In Partial Fulfillment of the Requirements For the Degree Masters of Science

By
James F. Dempsey
June, 1992
This thesis has been approved
for the Department of Mechanical Engineering
and the College of
Engineering and Technology

Moss Professor and Chairman of
Mechanical Engineering

Dean of the College of
Engineering and Technology
ACKNOWLEDGEMENTS

I wish to express my sincere thanks to my graduate advisor, Jay Gunasekera, Ph.D., P.E., for his insight and suggestions proved to be invaluable in producing this thesis. Secondly, I wish to thank Dr. Bhavin Mehta for his assistance in the utilizing the Ohio University computational resources and for serving as a member on my graduate committee. His advice and answers to many, many questions saved hours of fruitless efforts on the various computer systems.

I wish to express my gratitude to Mr. Mark Langout of the Cooper Industries Apex Division for his help in assembling the product data and specifications needed for the analysis. His insights into how to approach the analysis problem was greatly appreciated. I also wish to thank Cooper Industries for providing the financial support for this project.

Finally, I would like to thank Joseph A. Recktenwald, Ph.D., P.E., for serving as a member on my graduate committee. His constructive comments were very much appreciated. Thanks is also due to Mohammad Dehghani, Ph.D., P.E., for his guidance in the theory of elasticity.
TABLE OF CONTENTS

LIST OF FIGURES vi

CHAPTER 1 : Introduction 1

CHAPTER 2 : Review of Literature 7
    Elastic Theory Review 7
    Yielding Criteria 15
    Finite Element Method 16
    Review of Existing Literature 20

CHAPTER 3 : Analysis and Results 25
    I/EMS Socket Modeling 25
    I/FEM Solid Finite Element Models 31
    Analytical Verification of the Solid Models 77
    Two Dimensional Adaptive Analysis 78

CHAPTER 4 : Conclusions and Recommendations 101
    Conclusions 101
    Recommendations 103

REFERENCES 105
LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Engineering drawing of the Apex-3212 tool socket.</td>
<td>2</td>
</tr>
<tr>
<td>1.2</td>
<td>Engineering drawing of the Apex-3512 tool socket.</td>
<td>3</td>
</tr>
<tr>
<td>2.1</td>
<td>Elastic-Plastic Constitutive Relationship.</td>
<td>9</td>
</tr>
<tr>
<td>2.2</td>
<td>Generalized body subjected to external forces.</td>
<td>10</td>
</tr>
<tr>
<td>2.3</td>
<td>3-D stress element only showing x component of stress, assuming change across differential width of element.</td>
<td>13</td>
</tr>
<tr>
<td>3.1</td>
<td>I/EMS solid model of the Apex-3212 tool socket.</td>
<td>29</td>
</tr>
<tr>
<td>3.2</td>
<td>I/EMS solid model of the Apex-3512 tool socket.</td>
<td>30</td>
</tr>
<tr>
<td>3.3</td>
<td>I/FEM three dimensional finite element model of the Apex-3212 tool socket, created using solid hexahedral elements.</td>
<td>34</td>
</tr>
<tr>
<td>3.4</td>
<td>I/FEM three dimensional finite element model of the Apex-3512 tool socket, created using solid hexahedral elements.</td>
<td>35</td>
</tr>
<tr>
<td>3.5</td>
<td>Isometric view of the I/FEM three dimensional finite element model of the Apex-3212 tool socket.</td>
<td>36</td>
</tr>
<tr>
<td>3.6</td>
<td>Isometric view of the I/FEM three dimensional finite element model of the Apex-3512 tool socket.</td>
<td>37</td>
</tr>
<tr>
<td>3.7</td>
<td>Two dimensional representations of the load and constraint conditions applied to the solid finite element models.</td>
<td>40</td>
</tr>
<tr>
<td>3.8a</td>
<td>Von Mises stress contour plot of the Apex-3212 socket for Load Condition I. Side cutout view shown.</td>
<td>47</td>
</tr>
</tbody>
</table>
3.8b Von Mises stress contour plot of the Apex-3512 socket for Load Condition I. Side cutout view shown.

3.9a Von Mises stress contour plot of the Apex-3212 socket for Load Condition I. Bottom view (critical cross section) shown.

3.9b Von Mises stress contour plot of the Apex-3512 socket for Load Condition I. Bottom view (critical cross section) shown.

3.10a Maximum principal stress contour plot of the Apex-3212 socket for Load Condition I. Bottom view (critical cross section) shown.

3.10b Maximum principal stress contour plot of the Apex-3512 socket for Load Condition I. Bottom view (critical cross section) shown.

3.11a Von Mises stress contour plot of the Apex-3212 socket for Load Condition II. Side cutout view shown.

3.11b Von Mises stress contour plot of the Apex-3512 socket for Load Condition II. Side cutout view shown.

3.12a Von Mises stress contour plot of the Apex-3212 socket for Load Condition II. Bottom view (critical cross section) shown.

3.12b Von Mises stress contour plot of the Apex-3512 socket for Load Condition II. Bottom view (critical cross section) shown.

3.13a Maximum principal stress contour plot of the Apex-3212 socket for Load Condition II. Bottom view (critical cross
3.13b Maximum principal stress contour plot of the Apex-3512 socket for Load Condition II. Bottom view (critical cross section) shown.

3.14a Von Mises stress contour plot of the Apex-3212 socket for Load Condition III. Side cutout view shown.

3.14b Von Mises stress contour plot of the Apex-3512 socket for Load Condition III. Side cutout view shown.

3.15a Von Mises stress contour plot of the Apex-3212 socket for Load Condition III. Bottom view (critical cross section) shown.

3.15b Von Mises stress contour plot of the Apex-3512 socket for Load Condition III. Bottom view (critical cross section) shown.

3.16a Maximum principal stress contour plot of the Apex-3212 socket for Load Condition III. Bottom view (critical cross section) shown.

3.16b Maximum principal stress contour plot of the Apex-3512 socket for Load Condition III. Bottom view (critical cross section) shown.

3.17 Maximum Von Mises stress points A, B, and C, on the critical cross sections of the Apex-3212 and Apex-3512 sockets. This figure is used as the reference for Figures 3.18 through 3.20.

3.18a Maximum Von Mises stress points vs. applied torque for the Apex-3212 socket, Load Condition I.
3.18b Maximum Von Mises stress points vs. applied torque for the Apex-3512 socket, Load Condition I. 69

3.19a Maximum Von Mises stress points vs. applied torque for the Apex-3212 socket, Load Condition II. 70

3.19b Maximum Von Mises stress points vs. applied torque for the Apex-3512 socket, Load Condition II. 71

3.20a Maximum Von Mises stress points vs. applied torque for the Apex-3212 socket, Load Condition III. 72

3.20b Maximum Von Mises stress points vs. applied torque for the Apex-3512 socket, Load Condition III. 73

3.21 Deformed model of the Apex-3212 socket for Load Condition I and an applied torque of 500 in-lbs. Side view shown with an exaggeration factor of 50. 75

3.22 Deformed model of the Apex-3512 socket for Load Condition I and an applied torque of 500 in-lbs. Side view shown with an exaggeration factor of 50. 76

3.23 Apex-3212 socket critical cross section surface used for I/FEM two dimensional automeshing. 83

3.24 Apex-3212 socket critical cross section surface with geometry based features in place. 84

3.25 Automeshed Apex-3212 socket critical cross section surface with geometry based constraints and nodal loads. 87

3.26 Automeshed Apex-3212 socket critical cross section after h-adaptive mesh refinement for Load Condition I. Convergence criterion: HVM, refinement criterion: HVM. 89
3.27 Automeshed Apex-3212 socket critical cross section after h-adaptive mesh refinement for Load Condition II.
Convergence criterion : HVM, refinement criterion : HVM. 90

3.28 Automeshed Apex-3212 socket critical cross section after h-adaptive mesh refinement for Load Condition III.
Convergence criterion : HVM, refinement criterion : HVM. 91

3.29a Von Mises stress contour plot of the Apex-3212 two dimensional model of the critical cross section for Load Condition I. 95

3.29b Von Mises stress contour plot of the Apex-3512 two dimensional model of the critical cross section for Load Condition I. 96

3.30a Von Mises stress contour plot of the Apex-3212 two dimensional model of the critical cross section for Load Condition II. 97

3.30b Von Mises stress contour plot of the Apex-3512 two dimensional model of the critical cross section for Load Condition II. 98

3.31a Von Mises stress contour plot of the Apex-3212 two dimensional model of the critical cross section for Load Condition III. 99

3.31b Von Mises stress contour plot of the Apex-3512 two dimensional model of the critical cross section for Load Condition III. 100
CHAPTER 1

Introduction

The objective of the Cooper Industries Apex Tool project is to obtain the stress-strain characteristics of two similar hexagonal-driver tool sockets under a variety of load conditions. The finite element method (FEM) was chosen for this task for its ability to model the elastic characteristics of solid bodies and two dimensional representations and then display the stress and strain data in easy to read stress contour plots and deformed shape plots. This data obtained and recommendations made in this report could then be used by Apex tool designers to optimize the tools for increased strength, longer life, and more economical production. The two tool sockets selected for the analysis are the Apex-3212 and the Apex-3512 sockets. Both tool sockets are used for driving 3/8 inch hexagonal head bolts using a 3/8 inch square drive. The difference in the two sockets is that the Apex-3212 socket is a regular walled socket that tapers slightly near the hexagonal driver-end while the Apex-3512 is a thin walled socket that has a greatly reduced outer radius for the driver-end half of the socket. The engineering drawings of the Apex-3212 and the Apex-3512 sockets are given in Figures 1.1 and 1.2, respectively. The tool sockets are designed to be used in industry with impact drivers or hand drivers. High grade tool steel (4150 steel) is used to make these fastening tools, and they are designed to be operated in the elastic range so that no permanent plastic deformation of the socket bodies occur. Thus finite element methods is a logical choice for the elastic analysis of these tool sockets.
Figure 1.1 : Engineering drawing of the Apex-3212 tool socket.
Figure 1.2: Engineering drawing of the Apex-3512 tool socket.
Since the advent of the digital computer, the finite element method (FEM) has become standard practice in the analysis and optimization of mechanical designs. The finite element method is a widely accepted numerical procedure for obtaining solutions to differential equations of engineering and physics and is the computational basis of many computer-aided design systems. Originally when the finite element method was used in conjunction with digital computers, the FEM data was entered using bulk data decks. Thus the user was required to enter the data for each element numerically, which was quite time consuming. More recently the finite element method has been combined into computer-aided design systems allowing the user to enter the required FEM data graphically. This has made the finite element method much easier and faster to implement increasing its popularity among designers. The Intergraph Corporation is one such company that integrated CAD and FEM functions.

Ohio University has recently purchased one of the more advanced Intergraph systems. This system is much faster and easier to use than the other Intergraph system that Ohio University purchased a few years earlier. The new system is based on the Clipper Unix (CLIX) operating system and workstations and runs the Intergraph/Engineering Modeling System (I/EMS) and the Intergraph/Finite Element Modeling/Solver (I/FEM) software. I/EMS combines the processing power of the Clipper workstations with advanced programming techniques and data structures to provide an efficient tool for generating drawings and solid models of mechanical parts and assemblies. I/FEM is a computer-aided engineering software package that works in conjunction with I/EMS for general purpose finite element analysis. I/FEMRASNA is software that is almost
identical to I/FEM except that it allows p-adaptive meshing studies, which will be explained later in Chapter 3.

The Cooper Industries Apex tool project utilized both I/EMS and I/FEM extensively to determine the stress/strain characteristics of the selected Apex tool sockets. Also, this software was used to determine the interaction between the selected Apex tool sockets and the bolts that are driven by them. The basic procedure utilized was to obtain the geometric and material specifications from the engineering drawings of the tool sockets and then create the respective finite element meshes. Boundary conditions and load vectors were then applied to the model along with the material properties and the model was verified for analysis.

Initially, three dimensional models of the two sockets were created using solid brick elements and then analyzed for various load cases. Stress distributions were obtained for the entire socket along with deformed shape plots. The next step was to analyze the "critical cross-section" of the sockets using two dimensional modeling techniques. The "critical cross-section" is the cross-section that was determined to have the maximum stresses in it from the three dimensional model analysis. The two dimensional modeling allows the use of such techniques as h-adaptive and p-adaptive refinement analysis, allowing a more accurate stress distribution throughout the cross-section. These techniques of model refinement are not available for three dimensional modeling, and will be discussed in Chapter 3.

Chapter 2 of this thesis presents review of existing applicable literature, including a basic review of the theory of elasticity and the related finite element method. Chapter 3 provides the analysis procedures
undertaken along with analysis results. Stress contour plots and strain deformation plots of the different models are given along with graphs of octahedral shear stress versus applied torque at important nodes. Chapter 4 presents conclusions formulated from the analysis of the data obtained on the finite element analysis of the sockets and recommendations for product improvement. Also, suggestions are provided here for continued research into different areas of the tool socket development.
CHAPTER 2

Review of Literature

The tool socket can be broadly defined as a tool that is driven by an applied torque to tighten or loosen a threaded fastener. The two sockets selected for the analysis are both used for driving 3/8 inch hexagonal head bolts using a 3/8 inch square drive. The difference in the two sockets is that the Apex-3212 socket is a regular walled socket while the Apex-3512 is a thin walled socket. The sockets are designed to be used in industry with impact drivers or hand drivers. High grade tool steel (4150 steel) is used to make these fastening tools, and they are designed to be operated in the elastic range so that no permanent plastic deformation of the socket bodies occur.

In order to understand the analysis of these sockets and the consequential results, a basic review of the theory of elasticity and the finite element method are required. The finite element procedure used for the analysis is directly related to the theory of elasticity in that the finite elements used are linearly elastic and the yield stress of the material is used as the limiting failure criterion for the sockets.

Elastic Theory Review

Elasticity is the behavior of solid bodies that when placed under external loads and strained, return to their original shape when the external loads are removed. This is in contrast to plasticity, where the behavior of solid bodies is to deform permanently under the action of external loads. Actually, however, an elastic body is an idealization, because all bodies
exhibit more or less plastic behavior even for the smallest loads (1). For the basic elastic body, though, this permanent deformation is so small as to be practically not measurable, if the loads are sufficiently low. A material that behaves elastically and also exhibits a linear relationship between stress and strain is said to be linearly elastic. The tool sockets under investigation were designed to be operated in the linear elastic range to avoid permanent deformations from plastic flow. The linear relationship between stress and strain for a cylinder in simple tension or compression can be expressed by the equation

\[ \sigma = E\varepsilon \]  

where \( \sigma \) designates stress, \( E \) is the modulus of elasticity, and \( \varepsilon \) represents strain. The specific modulus of elasticity depends on the material being used. The modulus of elasticity is the slope of the stress-strain diagram in the linearly elastic region, as seen in Figure 2.1 and is definitely a property of the material. Here the relationship between stress and strain is essentially linear. This linear part of the curve extends up to the proportional limit, sometimes taken as the yield point, and it is in this range that the linear theory of elasticity, using Hook's law, is valid. For some materials the yield point is so poorly defined that a line is constructed parallel to the initial linear portion of the stress-strain curve with a 0.2% offset of strain. The offset line intersects the curve and this point is known as the offset yield strength. Upon further increase in load, the strain no longer varies linearly with stress and plastic deformation occurs. As the load is increased from the yield point, the strain increases at a greater rate. The part continues its plastic flow under increasing load, necking down rapidly and then fracturing. This is of course a simple representation of
Stress ~ Fracture point

Yield point

slope = E (Young's Modulus)

Elastic region  Plastic region

Figure 2.1: Elastic - Plastic Constitutive Relationship
stress-strain relation in one dimension.

Stress and strain in a real system actually act in three dimensions and can be represented as tensors (2). The stress tensor stands for the nine components of stress that occur in a three dimensional system. Likewise the strain tensor stands for the nine components of strain in a three dimensional system. The stress tensor is given by:

\[
\sigma_{ij} = \begin{pmatrix}
\sigma_{11} & \sigma_{12} & \sigma_{13} \\
\sigma_{12} & \sigma_{22} & \sigma_{23} \\
\sigma_{13} & \sigma_{23} & \sigma_{33}
\end{pmatrix} = \begin{pmatrix}
\tau_{xy} & \tau_{xz} \\
\tau_{yx} & \tau_{yz} \\
\tau_{zx} & \tau_{zy}
\end{pmatrix}
\]  

(2.2)

Similarly, the nine components of strain are represented by \( \varepsilon_{ij} \).

Now consider a generalized body subjected to a system of forces \( F_1 \) through \( F_6 \) as shown in Figure 2.2.

Figure 2.2 : Generalized body subjected to external forces.
Consider a plane AB that passes through the generalized body dividing it into parts 1 and 2. If part 1 is considered, it is observed that it is in equilibrium under the actions of $F_4$, $F_5$, $F_6$, and the force $F_{12}$ that part 2 exerts on part 1, where $F_{12}$ is the resultant of the continuous distribution of forces on the plane AB that part 2 exerts on part 1. If a small area $\Delta A$ is taken on this plane with a force $\Delta F$ acting on it, then the unit stress acting at this point is defined as:

$$\sigma = \lim_{\Delta A \to 0} \frac{\Delta F}{\Delta A}$$  \hspace{1cm} (2.3)\

It is important to note that the unit stress, $\sigma$, must be referred to a particular plane. For any other plane passing through the same point it can be seen from Figure 2.2 that the force distribution on this plane, and thus the unit stress, will be different.

The unit stress, $\sigma$, need not be perpendicular to the plane AB. Generally, the unit stress, $\sigma$, is decomposed into two distinct components, one normal to the plane of reference, called the normal stress, and one parallel to the reference plane, called the shear stress. The normal stress is considered positive when it is tensile in nature and negative when it is compressive in nature.

From previous consideration, it is seen that to completely specify the stress at a point it is necessary to specify the stresses at that point on three mutually perpendicular planes passing through that point. The stress on any arbitrary plane through that point can then be represented in terms of the stresses on the three perpendicular planes (3). The three mutually
perpendicular planes are generally designated to be planes perpendicular to the x, y, and z coordinate axes. The stresses acting on these planes at their point of intersection are as designated in Figure 2.3. The stresses as shown consider only the x direction, and the subscripts denote the direction of the stress. The first subscript designates the normal to the plane under consideration, and the second subscript represents the direction of the stress. Therefore $\tau_{yz}$ designates a shearing stress acting on the face of the element that is perpendicular to the y axis, the stress acting in the direction of the z axis. The positive directions of the components of shearing stress on any side of the cubic element are taken as the positive directions of the coordinate axes. If one considers an infinitesimal rectangular parallelepiped surrounding a given point in a body, the static equilibrium of forces requires that the stresses at this point satisfy the following equations:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = -F_x$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} = -F_y \quad (2.4)$$

$$\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} = -F_z$$

where $F_j$ are the components of the external body forces per unit volume.
Figure 2.3: 3-D stress element only showing x component of stress, assuming change across differential width of element.
It should be noted that for an isotropic material:

\[ \tau_{yx} = \tau_{xy} \quad \tau_{yz} = \tau_{zy} \quad \tau_{xz} = \tau_{zx} \]  \hspace{1cm} (2.5)

These two sets of equations, 2.4 and 2.5, can be expressed in tensor notation:

\[ \sigma_{ij,i} = -F_j \]
\[ \sigma_{ij} = \sigma_{ji} \]  \hspace{1cm} (2.6)

The second expression of (2.6) shows the fact that the stress tensor is symmetric and that there generally are only six independent components of stress at a point instead of nine.

The generalization of Hooke's law in three dimensions relating strains to stresses for an isotropic material (4) is given by the relations:

\[ \varepsilon_x = \frac{1}{E} [\sigma_x - \mu(\sigma_y + \sigma_z)] + \alpha T \]
\[ \varepsilon_y = \frac{1}{E} [\sigma_y - \mu(\sigma_x + \sigma_z)] + \alpha T \]  \hspace{1cm} (2.7)
\[ \varepsilon_z = \frac{1}{E} [\sigma_z - \mu(\sigma_x + \sigma_y)] + \alpha T \]

where \( \mu \) is Poisson's ratio relating lateral strain to axial strain, \( \alpha \) is the coefficient of linear thermal expansion, and \( T \) is the temperature above some arbitrary reference temperature. For the case of the sockets, the \( \alpha T \)
terms can be neglected since the sockets are generally used in surroundings at ambient temperature. Also, as stated earlier, it is assumed that the material is isotropic, in that it has the same elastic properties in all directions.

**Yielding Criteria**

When a part is loaded so that the state of stress is uniaxial, then the stress level and the material strength can be compared directly to determine if the part will fail, or begin to deform. This method is rather simple since there is only one value of stress and one value of strength, generally the yield strength or the ultimate strength. The problem becomes more complicated when the stress state is biaxial or triaxial, as is consistent with the analysis of the sockets. In these cases there are a variety of stresses, but still only one significant strength. To determine whether the part will fail or not, a number of yield criteria have been proposed (4). Generally the two most accepted yielding criteria are the maximum shear stress theory and the Von Mises distortion energy theory.

The maximum shear stress theory (Tresca Criterion) states that yielding begins whenever the maximum shear stress in any element becomes equal to the maximum shear stress occurring in a tension test specimen of the same material when that specimen begins to yield. The maximum shear stress is equal to half the difference between the maximum and minimum principal stresses. For simple tension, since \( \sigma_2 = \sigma_3 = 0 \), the maximum shear stress at yield is \( 1/2\sigma_0 \), where \( \sigma_0 \) is the yield stress in simple tension. This criterion is in fair agreement with experimental results (1) but it suffers from one major difficulty, it is necessary to know in advance which are the maximum and minimum principal stresses.
The distortion energy theory (Von Mises-Hencky Criterion) for isotropic material states that yielding will occur whenever the distortion energy in a unit volume equals the distortion energy the same volume when stressed uniaxially to the yield strength. The yielding condition is given by:

\[ \sigma' = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}} \]

(2.8)

\[ \sigma' \geq \sigma_0. \]

(2.9)

The stress \( \sigma' \) represents the entire stress state \( \sigma_1, \sigma_2, \) and \( \sigma_3, \) and is referred to as the effective stress or the Von Mises stress. It can be shown that the yield stress in pure shear is \( 1/3 \) times the yield stress in simple tension, thus the Von Mises criterion predicts a pure shear yield stress which is about 15% higher than that predicted by the Tresca criterion. The Von Mises yield criterion usually fits experimental data better than other yield theories, and is generally easier to apply than the Tresca criterion because no knowledge is needed regarding the relative magnitudes of the principal stresses. Also, the Von Mises criterion takes into account the effect of the intermediate principal stress, thus representing the actual stress state more effectively. The Von Mises criterion is thus the yield criterion used for predicting the effective stress state of the Apex sockets.
**Finite Element Method**

The finite element method is a widely accepted numerical procedure for obtaining solutions to differential equations of engineering problems and is the computational basis of many computer aided design systems (5).

The basic philosophy of the finite element method is a piecewise approximation of a continuum. The method assumes that the solution to a complicated problem can be obtained by subdividing the region of interest and by representing the solution in each subdivision with a relatively simple function. The method produces a system of algebraic equations, a finite number of elements, and a finite number of degrees of freedom. The approach can be broadly applied to a variety of classes of problems such as elastic and plastic deformation, vibrations analysis, heat transfer, fluid flow, etc. In elastic analysis, different classes of structures can be analyzed such as frames, shells, and solids.

Most modern day finite element modeling packages break up the modeling and analysis into three parts: preprocessing, analysis, and postprocessing.

Preprocessing is the actual construction of the model. In the early days of finite element analysis, models were constructed by the engineer calculating the elemental and nodal data "by hand". This was very time consuming and inefficient. But since those days, preprocessors have been introduced to speed the construction of the model by linking computer aided design to finite element modeling. The preprocessor allows the user to utilize geometry and mapped meshing capabilities to place nodes and elements to represent the physical model. Meshing is the process of placing nodes and elements in the required fashion to represent the model.
Generally FEM packages provide the user two methods of constructing the finite element model, fully automatic meshing or semiautomatic mapped meshing. These meshing options will be described later. The preprocessor also allows the user to define boundary conditions, load cases, and material properties easily.

Once the model has been created, the boundary and loading conditions defined, and the material properties defined using the preprocessor, the model can be analyzed. This is the fundamental basis of the finite element package. In general, an approximation equation is specified. The order of the approximation, usually linear or quadratic, must be specified and the equations written in terms of the unknown nodal values. An equation is created for each element. The method proceeds to develop a system of equations which gives one equation for each unknown nodal quantity. The system of equations is constructed by computing the element's contribution and placing the values in the correct position within the final system of equations. The basic formulation for the resulting set of equations for the static analysis can be given by:

\[
[K]{u} - \{F\} = 0
\]

or

\[
[K]{u} = \{F\}
\]  

(2.10)

where \([K]\) is the global stiffness matrix, \(\{F\}\) is the global force vector, and \(\{u\}\) is the displacement vector being solved for. The global \([K]\) and \(\{F\}\) matrices are created by adding the elemental \([k^{(e)}]\) and \(\{f^{(e)}\}\) contributions.
into their correct position within the final system of equations. The direct stiffness method is the name given to the procedure for incorporating the element matrices into the final system of equations. The elemental contributions are calculated from the approximating equation derived from the differential equation describing the behavior of the element. The global stiffness matrix, \([K]\), is always symmetric for structural problems resulting from the mathematical formulation. The diagonal coefficients, \(K_{ii}\), are always positive and relatively large when compared with their off diagonal counterparts.

The final system of equations are solved for the unknown nodal quantities using matrix manipulations such as Gaussian elimination. The quantities of interest are then calculated from the displacement vector \(\{u\}\), such as stress and strain components, displacements, and reactions.

Once the analysis is complete, postprocessing comes into play. Postprocessing organizes the large amount of data that is produced from a finite element analysis run and allows it to be displayed in a variety of graphical and numerical manners. Most modern FEM software packages let the user display analysis results with color coded stress contours explained by a color legend denoting the stress intensities. The contours, which make use of nodal data can be plotted as line contours in wireframe windows or as filled and blended contours in shading windows, if available. The stress contours option allows the user to easily determine where the stress concentrations occur in a design. A variety of stresses can be displayed, including principal stresses and Von Mises stresses. Deformed shapes and animation of the deformed shapes is generally also available to show strain deformation in the static analysis. The nodal displacements
multiplied by an exaggeration factor produces the deformed shape, and the animation command allows the user to observe how the deformation is taking place. Model statistics and results reports can be produced using the postprocessor.

**Review of Existing Literature**

In a 1990 paper authored by Ray W. O'Clough, the original formulation of the finite element method and the circumstances that led to its' creation is discussed (6). He notes that the central feature of the procedure that was developed is the evaluation of the stiffness properties of structural elements on assumed sets of displacement interpolation functions. The method used direct stiffness assembly to establish the structure stiffness. O'Clough brings out two different views on the utilization of the finite element method. One view considers the finite element method to be merely an extension of basic methods of structural analysis in which a structure is treated as an assemblage of discrete structural elements (7). The other view is that the finite element method is a procedure for obtaining approximate solutions to problems in continuum mechanics based on applying assumed strain patterns independently in discrete regions of the system (8). Both views are valid because the concept that guided the development of the method was that it was an extension of standard analysis procedures introduced to solve difficult problems. O'Clough also emphasized that the finite element analysis cannot be better than the data and judgement used when formulating the mathematical model. The limitations of the approximation procedure must be understood and good engineering judgement is required in interpreting the results of an analysis.
J. Tinsley Oden presents in the paper "The Best FEM", considerations of which type of finite element method can produce the most accurate results in the least amount of time (9). He discusses how element sizes and spectral orders (polynomial orders) can be redistributed or changed throughout the finite element mesh to produce the best improvement of the solution with the least number of additional unknowns. There are two parameters available to improve the quality of FEM solutions: change the FEM mesh size (h-adaptive meshing) or the polynomial order of the approximating equations (p-adaptive meshing). It is expected that as h gets arbitrarily small and/or p becomes large, the finite element solution should approach the exact solution to the problem. Oden presents the analyses of a model of the deflection of a membrane and applies h-adaptive meshing, p-adaptive meshing, and a combination h-p-adaptive meshing to see how fast the quality of the model can be improved. For the h-adaptive analysis, Oden kept p=1 and allowed the mesh to refine itself until the localized error was below a preset tolerance throughout the mesh. In the p-adaptive analysis, a fixed number of elements were used to model the mesh and the local spectral order p of the elements were allowed to vary until the error was below the preset tolerance. For the combination h-p procedure, Oden used an optimization plan to allow general distribution of h and p mesh changes to achieve the preset tolerance. According to his results, the h-p adaptive analysis converged at the fastest rate and was thus deemed the superior refinement choice.

Arielle Emmett presents in a related article "Optimized Designs" new cutting edge optimization software by the Rasna Corporation (10).
The Rasna Corporation software uses p-adaptive meshing that automatically recalculates and converges to an accurate mathematical solution. But the new software goes a step further by taking the solution set geometry to optimize the product's shape subject to user applied constraints such as stress limits. The p-adaptive geometric elements allow significant shape changes to be made without changing the mesh. Therefore the finite element models can be made with somewhat simpler geometry, and it can show that the model has converged to a correct solution. It is noted though that the p-element solvers take rather long time to produce solutions, even for simple problems. This software is available as Rasna Applied Structure and works in conjunction with the Intergraph Corporation I/FEM software.

In the paper "Finite Element Analysis Convergence Curves", R. J. Melosh studies the use of element testing for assessing convergence characteristics which depend on the choice of element model; examines the effects of changes in boundary conditions; and demonstrates how remodeling decisions affect analysis accuracy and convergence (11).

In the publication "Applied Finite Element Analysis", Larry J. Segerlind presents a finite element analysis of the torsion of noncircular sections, with an illustrative example of the twisting of a square bar (5). Segerlind introduces the two general theories for calculating the shear stresses in solid shafts subjected to torsion: the theory developed by St. Venant and the other developed by Prantl. He uses Prantl's theory for the formulation of the governing differential equation. Segerlind analyzes the stresses introduced into a steel bar that is twisted one degree using finite element methods. Due to symmetry of the simple cross section, only one-eighth of the cross section needs to be analyzed. The section is initially
modeled with only three two-dimensional elements: two three nodded elements and one quadrilateral element. Only three elements are used initially so that the calculations can be illustrated easily. The solution is obtained by applying the direct stiffness procedure to obtain the desired nodal quantities. Segerlind goes on to show that more accurate values for the stress function and shear stresses can be obtained using grids with larger number of elements or by using elements with a higher level of interpolation, and he compares his results to theoretical findings. He also notes that a rather coarse mesh gives good results if nodal values are only of interest, but a refined mesh is required if derivative-related quantities are desired.

In the paper "The Torsional Analysis of Bars with Hollow Square Cross-sections" J. S. Lamancusa and D. A. Saravanos explore the analysis of square hollow tube sections with thin and thick walls (12). They explain that if the sections have thin walls, closed form solutions for torsional stiffness and shear stress can be found in various handbooks, but if the sections have thick walls, the solutions must be obtained using more complex methods, usually a three-dimensional finite element analysis. J. S. Lamancusa and D. A. Saravanos present an alternative to this method using a two-dimensional FEM thermal analogy, and the dependence of torsional properties on wall thickness is investigated. The authors modeled the hollow square shafts using two-dimensional isoparametric solid thermal elements. They also present an alternative method of modeling the system using three dimensional structural elements, noting that the three-dimensional modeling is more straightforward, but more elements are required for comparable accuracy increasing the CPU time substantially.
Due to symmetry only a quarter model of the cross-section was considered. The authors modeled the hollow square cross-section with two different shear moduli: one large shear moduli for the solid segment of the cross-section and one very small shear moduli representing the hollow segment of the cross-section. This representation allows the entire cross-section to be treated as a simply connected one of nonhomogeneous material, overcoming complexities introduced by boundary condition considerations (13). Solutions were obtained for various thickness/width ratios and stress contours were plotted. Closed-form algebraic equations were then developed representing the shear stress and torsional stiffness data using polynomial approximations.

The literature reviewed represents general to specific information that can be considered and applied when constructing and analyzing the finite element models of the tool sockets. Chapter 3 will discuss the methods used in the analysis of these bodies.
CHAPTER 3

Analysis and Results

The Cooper Industries Apex tool project utilized both the Intergraph Engineering Modeling System (I/EMS) and the Intergraph Finite Element Modeling system (I/FEM) extensively to determine the stress and strain characteristics of the Apex-3212 and Apex-3512 tool sockets. The first step in the analysis was to generate solid models of the two sockets using the I/EMS software.

I/EMS Socket Modeling

The Intergraph/Engineering Modeling System is an integrated software system for two dimensional and three dimensional wireframe modeling, along with more complex solid modeling and sculptured surfacing. I/EMS produces graphical representations using Non-Uniform Rational B-Spline (NURBS) technology. Geometric associativity and element grouping allow relationships among elements to be maintained during construction and modification of models. The element associativity maintains relationships such as tangency within a model for different elements. The system provides such display features as view dynamics, surface and solid shading, and hidden surface removal. As in most advanced CAD systems, a model can be viewed from several perspectives simultaneously. I/EMS is easily operated by choosing commands from icon-based screen menus or table-based menus. I/EMS provides on-line help menus that include command descriptions, glossary of terms, and a table of contents. The help menus are easily accessed during working
sessions providing a convenient alternative to referring to the operations training manuals.

I/EMS was utilized in the Apex tool project by producing detailed solid models of the two sockets being studied by using the data extracted from the engineering specifications. Hybrid solid models can be created by varying the design parameters. Two dimensional models are also created representing the critical cross sections of the sockets. These two and three dimensional models provide the foundations upon which the finite element models can be built.

In order to create a solid model of the sockets, a series of two dimensional drawings must be created in an I/EMS design file representing the various cross sections of the socket at boundary or transition regions. The boundary regions are the top and bottom surfaces of the sockets and the transition regions are the regions in the socket where one cross section begins to change to another cross section in the socket. One such transition region is where the hexagonal opening cross section begins to change to the circular opening cross section. These various cross sections must be aligned along a designated axis, such as the z-axis, and be separated by the proper distances. The cross sections of the sockets can easily be created using the I/EMS commands "Place Circle By Radius Key-In And Center Point" and "Place N-Sided Polygon by Vertex". The spatial coordinates are most accurately and efficiently designated by using Precision Inputs from the keyboard. Line weight, color, and style can be controlled from the screen menus.

Once the socket cross sections were completed, the solid models of the sockets could be created. This was accomplished in one of two ways:
1) creating a modified solid socket from primitive and derived solids and
2) creating a skinned solid socket using the skinning solid command.

The first method to create the solid model, using primitive and derived solids to create the modified solid, is more complex and time consuming than the second method for this case. Primitive solids are solids that do not require any other elements for their creation, such as cylinders and rectangles. Derived solids are generated from other elements and are generally solids of projection or solids of revolution. The primitive and derived solids can be combined in the design file using Global Modification commands to create the complete model. The basic global commands are union, difference, and intersection. The union operation combines two elements into one element. The difference operation removes overlapping volume from one element. This is useful for creating holes and cylinders. The intersection operation removes any non-overlapping volume from both elements. In order to construct the Apex sockets, the required primitive cylinders and conical sections were generated and joined using the union and intersection commands. Then derived solids were created from the inner hexagonal, circular, and square cross sections using the "Create Solid of Projection" command. These derived solids would be subtracted from the body of the socket already created using the difference operation. It must be noted that in order to use the global modifications union, difference, and intersection, the component primitive and derived solids must overlap each other, not just touch at one point or surface. If the components do not overlap, then those commands will not fail to generate the solid. The final operation in this process is to compress the state tree, the record of all the global modifications used to create the model.
This produced the final solid model of the desired socket. Once the state tree is compressed, though, the components of the entire model can no longer be modified.

The second method of creating a solid model of the socket is to use the command "Place Skinning Solid". This "Place Skinning Solid" command is used to project one or several cross sections along a curve projection or around an axis. In the case of the Apex sockets, the Place Skinning Solid command was used twice to project several cross sections along the neutral axis of the socket. The first time the "Place Skinning Solid" command was used is to create the body of the socket by skinning the outer socket cross sections. The second time the command was used was to create the solid representing the inner hexagonal, circular, and square openings in the socket by skinning the inner socket cross sections. The inner solid was then subtracted from the body of the socket using the difference operation from the global modifications menu and the solid model of the socket is thus complete.

The solid model of the Apex 3212 socket can be seen in Figure 3.1 and was created using the first method of combining primitive and derived solids to create the modified solid model. The Apex 3512 socket is given in Figure 3.2 and was created using the "Place Skinning Solid" command. It can be seen the either method of construction produces satisfactory results, but the second method is more time efficient than the first method.

I/EMS allows the operator to display the solid model in a wide variety of colors, shades, and lights. It also allows the user to display the model as a wireframe or a shaded solid.
Figure 3.1: i/EMS solid model of the Apex-3212 tool socket.
Figure 3.2: I/EMS solid model of the Apex-3512 tool socket.
I/FEM Solid Finite Element Models

The next phase of the analysis was to model the sockets using three dimensional finite element modeling techniques. Initially, the modeling began by using the Intergraph Rand Micas finite element software on the Ohio University VAX 751. But then the Ohio University Department of Mechanical Engineering purchased new Unix based workstations and the Intergraph Finite Element Modeling system software.

Intergraph's Finite Element Modeling system is a computer aided engineering package for general purpose finite element analysis. Finite element analysis is a numerical technique based on the discretization of regions and structures, as was discussed in Chapter 2. The technique represents a continuum in a piecewise manner. The approach produces a system of algebraic equations, a finite number of elements, and a finite number of degrees of freedom.

The process of creating a finite element model using I/FEM takes a real physical geometry and discretizes it into pieces called finite elements. The behavior of these finite elements can then be approximated for various loadings and boundary conditions, considering the material properties of the given physical model. Each individual element is bounded by points called nodes, and the nodal connectivity defines the interactions between elements.

As with many modern day finite element modeling systems, I/FEM breaks up the modeling and analysis into three parts: preprocessing, analysis, and postprocessing.

Preprocessing is the actual construction of the model and was introduced to speed the construction of the model by linking computer
aided design to finite element modeling. I/FEM's preprocessor allows the user to utilize geometry and mapped meshing capabilities to place nodes and elements to represent the physical model. I/FEM gives the user two methods of constructing the finite element model, fully automatic meshing or semiautomatic mapped meshing.

Fully automatic meshing is designed for meshing two dimensional surfaces and three dimensional solids. For two dimensional models, I/FEM recognizes any single or composite surface drawn using I/EMS commands and generates the finite element model using plate or shell elements. For three dimensional models, I/FEM recognizes valid I/EMS solid models and generates the finite element mesh using solid 4-noded tetrahedron elements. According to the Intergraph I/FEM operator training guide, the automatic mesher will recognize any valid I/EMS solid model and produce the proper mesh. But after extensive experimentation with the automatic mesher on the Apex sockets and other very basic objects, it was found that the automatic mesher was basically useless unless a very primitive object such as rectangular or square block was being meshed. The I/FEM automatic mesher could not create a mesh for either of the Apex sockets being studied. In fact, automatic mesher refused to mesh even a basic solid cylinder. Also, the automatic meshing software is limited to placing only low order solid tetrahedral elements. Solid hexahedral elements (brick elements) produce a much neater and accurate mesh for these cases being studied, and were thus chosen for the job. It is recommended that Intergraph Corporation research the area of automatic meshing a great deal further to produce a more usable automatic meshing system. Therefore,
until this happens, the method of semiautomatic mapped meshing should be utilized for producing the meshed domain.

The drawback to semiautomatic mapped meshing is that it requires more user interaction. Meshes can be constructed using beam, rod, spring, triangular, quadrilateral, tetrahedral, or solid brick elements. The mapped meshes can be created using I/EMS geometry, or without any geometry whatsoever. The advantages of using semiautomatic mapped meshing over fully automatic meshing is that any type of finite element can be used, and the mesh can be locally refined to take into consideration possible stress concentration points such as small holes or reductions in cross sectional areas. The Apex-3212 and Apex-3512 sockets were meshed using the I/FEM semiautomatic mapped mesher with solid brick elements. Figures 3.3 and 3.4 respectively show side views of the Apex-3212 and Apex-3512 socket finite element models, as created using I/FEM. Isometric views of the I/FEM socket finite element models are presented in Figures 3.5 and 3.6, respectively. The basic models of these sockets were originally created using Intergraph Rand Micas (IRM) on the VAX 751 system and then imported to I/FEM using a neutral file format. Using the preprocessor, material properties were defined, constraints were applied, and load vectors were placed on the models.

The material modeled was 4150 tool steel with a modulus of elasticity of 30E6 psi and a Poisson's ratio of 0.3.

A variety of boundary conditions were experimented with before the final boundary conditions were chosen. Although the sockets are symmetrical across various planes, it was decided that the entire sockets
Figure 3.3: I/FEM three dimensional finite element model of the Apex-3212 tool socket, created using solid hexahedral elements.
Figure 3.4: I/FEM three dimensional finite element model of the Apex-3512 tool socket, created using solid hexahedral elements.
Figure 3.3: Isometric view of the I/FEM three dimensional finite element model of the Apex-3212 tool socket.
Figure 3.6: Isometric view of the I/FEM three dimensional finite element model of the Apex-3512 tool socket.
should be modeled to simplify boundary conditions and present a more realistic representation of the stress states. Initially, opposing torques were applied at each end of the sockets by means of load vectors, but results could not be obtained for this type of loading since the models were unstable due to inadequate constraints on the models. It was determined that the best results were obtained by constraining the driven end of the socket and applying a torque via nodal load vectors at the driver end of the socket, as can be seen in Figures 3.3 and 3.4. This simulated the case where a socket is being torqued to remove a bolt that refuses to turn. Similar results were obtained by constraining the driver end and torquing the driven end of the socket, but for the results presented in this paper, only the previously stated boundary conditions were applied.

The load vectors were calculated to produce an applied torque to the Apex-3212 and Apex-3512 socket finite element solid models. Three different loading conditions at five different torques per condition were considered. The torques applied were 50, 100, 250, 500, and 750 in-lbs for each condition. The load vector calculations are presented in the Appendix. The load vectors were placed at the appropriate nodes in the proper orientation at the driver end of the socket while the opposite end was constrained. Condition I modeled contact between the socket and a bolt at each corner of the hexagonal opening in the socket using load vectors, modeling an ideal condition. Condition II modeled contact between the socket and a bolt only at three corners of the hexagonal opening in the socket using load vectors, and Condition III modeled contact between the socket and a bolt only at two opposite corners of the hexagonal
opening in the socket, representing the worst case. Conditions I through III are illustrated in Figure 3.7, along with a planar representation of the constraint condition.

Once the models were created, the boundary and loading conditions placed, and the material properties defined using the I/FEM preprocessor, the models were analyzed. I/FEM has a unique feature that allows the user to verify the model before running the analysis. The user simply selects the command "Verify Model" from the menu. If the model is complete and stable, a "Model OK" appears in the message window. If there is something wrong with the model, the operator is alerted and the nature of the error is written to a file. This command was very convenient and can save an enormous amount of time that is generally wasted in error tracking and corrupted analysis.

The I/FEM models of the Apex sockets were analyzed using the I/FEM linear static solver. I/FEM allows the user to translate the model into a neutral file that can be used by a third party solver such as PATRAN or NASTRAN. A software specific direct translator is also available for use with the powerful software package ABAQUS making I/FEM very versatile. The advantage to using the I/FEM solver directly is that it is workstation based allowing a rapid run time since that is the only job being processed. Other systems require that the model be submitted in batch mode to a mainframe computer where it competes with hundreds of other processes. For example, it would take anywhere from three to seven hours the analyze either the Apex-3212 or 3512 solid models in batch mode using IRM on the VAX 751, depending on the number of other processes currently being run. But an average analysis of either of those sockets
Figure 3.7: Two dimensional representations of the load and constraint conditions applied to the solid finite element models.
using the I/FEM solver needs only fifteen minutes to be completed. Thus, the advantage is obvious.

The process consisted of analyzing both sockets for the three Load Conditions described above. Five different load intensities were analyzed per Load Condition, each load case being analyzed separately with the results loaded into the I/FEM post processor.

Once the analysis is complete, post processing comes into play. Post processing organizes the large amount of data that is produced from a finite element analysis run and allows it to be displayed in a variety of graphical and numerical manners. I/FEM's array of graphics tools allows the user display analysis results as stress contours with color coded elements explained by a color legend denoting the stress intensities. The contours, which make use of nodal data can be plotted as line contours in wireframe windows or as filled and blended contours in shading windows. It should be noted that in order to activate the color filled shaded contour plots, the obscure command "(esc) z t e e " must be keyed in. Otherwise the operator will become frustrated trying to get the color filled contours command to work.

Deformed shapes and animation of the deformed shapes is also available on I/FEM to show strain deformation in the static analysis. The deformation animation gives insight into how the model is reacting to that given set of loads and constraints. Hardcopies of post processed data are easily attained from one of four printers attached to the system: the line plotter, the laser printer, the static plotter, or the thermal color printer.

The following color plots are the shaded stress contours determined for the three separate load conditions from various perspectives. The plots
were created using the thermal color plotter and represent Von Mises stresses and maximum principal stresses. The plots of the Von Mises stress contours are very useful in determining where and at what loading the sockets will begin to yield according to the distortion energy yield criterion. The maximum principal stress contour plots show the user where the maximum principal stresses occur in the sockets and whether they are tensile or compressive. Tensile maximum principal stresses are represented by positive stress values indicated on the color coded legend and compressive maximum principal stresses are represented by negative stress values on the legend. The following plots are only given for an applied torque of 500 in-lbs because stress distribution was found to be very similar for different torques with only the magnitudes changing. This observation reduces the required number of plots that need to be presented to gain an understanding of the stress states created in the sockets under various load conditions. Instead, graphs of the Von Mises stress versus applied torque at critical nodes for the three different load conditions are presented for the two sockets.

Figures 3.8a and 3.8b are the color shaded Von Mises stress contour plots of the side views of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition I. These two plots actually show the sockets halved down their z-axis so that the inner and wall stress contours are visible. Load Condition I, as stated before, models the ideal case of the bolt contacting the socket at all six corners in the hexagonal opening. Both plots show that the maximum Von Mises stresses occur at the driver opening of the sockets, and are concentrated at the hexagonal corners. The maximum Von Mises stress for the Apex-3212 socket is 46,080 psi, and for
the Apex-3512 it is 50,710 psi, considering a torque of 500 in-lbs. The Von Mises stresses are lowest near the driven end of the socket due to the large amount of material surrounding the square opening. For this applied torque the yield stress of the socket material, which is 150,000 psi (1), is not approached, even in the areas of stress concentration. The Von Mises stress levels in the Apex-3512 thin walled socket of Figure 3.8b are approximately 3000 to 4500 psi higher than the levels found in the Apex-3212 socket near the driver end of the socket. Also, as seen in Figure 3.8b, the stress levels in the wall of the socket after the cross section reduces to the narrowest outer dimension are greatly elevated compared to the stress levels in the Apex-3212 socket.

Figures 3.9a and 3.9b are the color shaded Von Mises stress contour plots of the bottom views of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition I. These views present the critical cross section of the two sockets since the maximum Von Mises stresses occur at the bottom hexagonal opening of the sockets. These views show clearly that the maximum Von Mises stresses occur at the six inner hexagonal corners of the sockets in a symmetric fashion. The Von Mises stress levels in the Apex-3512 thin walled socket of Figure 3.9b are approximately 3000 to 4000 psi higher than the levels found in the Apex-3212 socket in Figure 3.9a near each inner corner of the socket. Yielding of the sockets would thus begin at one or several of the inner hexagonal corners, and the Apex-3512 socket would begin to yield before the Apex-3212 socket.

Figure 3.10a and Figure 3.10b show the maximum principal stress contour plots of the critical cross sections for Apex-3212 and Apex-3512 sockets, respectively, for Load Condition I. Maximum principal tensile
and compressive stresses are shown, with compressive stresses being
denoted by negative values. The compressive stresses occur in the socket
where the bolt would press against the wall of the socket, and the tensile
stresses occur on the adjacent wall that would be pulled by the action of the
bolt as it is being torqued. The plots show that the compressive stress
contour fields are larger than the tensile stress contour fields. The plots
tend to misrepresent the data slightly, since the maximum principal tensile
and compressive stress fields are actually similar in size. This error is due
to color variations in the thermal color plotter compared to the screen
colors.

As with Von Mises stresses, the maximum principal stresses in the Apex-3512 socket are consistently higher than in the Apex-3212 socket.

The next set of plots show the stress contours determined for load
condition II. Load Condition II is the most realistic condition where the
bolt contacts the socket only at three corners of the hexagonal opening, thus
being kinematically stable. For this case, the contact points were spaced
120 degrees apart at three corners in the socket.

Figures 3.11a and 3.11b are the color shaded Von Mises stress
contour plots of the side views of the Apex-3212 and Apex-3512 sockets,
respectively, for Load Condition II. These two plots actually show the
sockets halved down their z-axis so that the inner and wall stress contours
are visible, and the Von Mises stress distributions are presented. As found
with Load Condition I, both plots show that the maximum Von Mises
stresses occur at the driver opening of the sockets, and are concentrated at
the three hexagonal corners that are loaded. This can be seen easier in
Figures 3.12a and 3.12b which show the Von Mises stress contour plots of the bottom views of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition II. The maximum Von Mises stress is 76,750 psi for the Apex-3212 socket and it is 94,270 psi for the Apex-3512 considering a torque of 500 in\*lbs. These stresses are significantly higher that the stresses found in the analysis using Load Condition I. The stresses throughout the remainder of the socket are very similar to the stresses found in the results of Load Condition I, though, suggesting a localized effect at the inner loaded corners of the sockets. Figure 3.13a and Figure 3.13b show the maximum principal stress contour plots of the critical cross sections for Apex-3212 and Apex-3512 sockets, respectively, for Load Condition II. Maximum principal tensile and compressive stresses are shown, and as with the previous plots of the maximum principal stresses, these plots tend to misrepresent the data slightly, since the maximum principal tensile and compressive stress fields are actually similar in size. The contrasts in colors are slightly too harsh for adjacent color contour bands.

The next set of plots show the stress contours determined for Load Condition III. Load Condition III is the most worst possible condition where the bolt is deformed to the point that it only contacts the socket at two corners of the hexagonal opening. For this generally unlikely case, the contact points were spaced 180 degrees apart at two opposing corners in the socket. Various load vector angles were experimented with normal to the loaded corner, but it showed little affect on the stress magnitudes or contours obtained.
Figures 3.14a and 3.14b are the color shaded Von Mises stress contour plots of the side views of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition III. These two plots show also the sockets halved down their z-axis.

As found with Load Conditions I and II, both plots show that the maximum Von Mises stresses occur at the driver opening of the sockets, and are concentrated at the two hexagonal corners that are loaded. This can be more clearly seen in Figures 3.15a and 3.15b which show the Von Mises stress contour plots of the critical cross sections of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition III. The maximum Von Mises stress found at the loaded corners were 125,700 psi for the Apex-3212 socket and 196,400 psi for the Apex-3512, considering a torque of 500 in-lbs. These stresses are significantly higher than the stresses found in the analysis using Load Conditions I or II. The stresses closer to the driven end of the socket are very similar to the stresses found in the results of Load Conditions I and II, though, suggesting a very localized effect at the inner loaded corners of the sockets. Figure 3.16a and Figure 3.16b show the maximum principal stress contour plots of the critical cross sections for Apex-3212 and Apex-3512 sockets, respectively, for Load Condition III. Maximum principal tensile and compressive stresses are shown and, as expected, have greater magnitudes in the Apex-3512 thin walled socket than in the Apex-3212 regular walled socket.
Figure 3.8a: Von Mises stress contour plot of the Apex-3212 socket for Load Condition 1. Side cutout view shown.
Figure 3.8b: Von Mises stress contour plot of the Apex-3512 socket for Load Condition I. Side cutout view shown.
Figure 3.9a: Von Mises stress contour plot of the Apex-3212 socket for Load Condition I. Bottom view (critical cross section) shown.
Figure 3.9b: Von Mises stress contour plot of the Apex-3512 socket for Load Condition I. Bottom view (critical cross section) shown.
Figure 3.10a: Maximum principal stress contour plot of the Apex-3212 socket for Load Condition I. Bottom view (critical cross section) shown.
Figure 3.10b: Maximum principal stress contour plot of the Apex-3512 socket for Load Condition I. Bottom view (critical cross section) shown.
Figure 3.11a: Von Mises stress contour plot of the Apex-3212 socket for Load Condition II. Side cutout view shown.
Figure 3.11b: Von Mises stress contour plot of the Apex-3512 socket for Load Condition II. Side cutout view shown.
Figure 3.12a: Von Mises stress contour plot of the Apex-3212 socket for Load Condition II. Bottom view (critical cross section) shown.
Figure 3.12b: Von Mises stress contour plot of the Apex-3512 socket for Load Condition II. Bottom view (critical cross section) shown.
Figure 3.13a: Maximum principal stress contour plot of the Apex-3212 socket for Load Condition II.

Bottom view (critical cross section) shown.
Figure 3.13b: Maximum principal stress contour plot of the Apex-3512 socket for Load Condition II. Bottom view (critical cross section) shown.
Figure 3.14a: Von Mises stress contour plot of the Apex-3212 socket for Load Condition III. Side cutout view shown.
Figure 3.14b: Von Mises stress contour plot of the Apex-3512 socket for Load Condition III. Side cutout view shown.
Figure 3.15a: Von Mises stress contour plot of the Apex-3212 socket for Load Condition III. Bottom view (critical cross section) shown.
Figure 3.15b: Von Mises stress contour plot of the Apex-3512 socket for Load Condition III. Bottom view (critical cross section) shown.
Figure 3.16a: Maximum principal stress contour plot of the Apex-3212 socket for Load Condition III.
Bottom view (critical cross section) shown.
Figure 3.16b: Maximum principal stress contour plot of the Apex-3512 socket for Load Condition III. Bottom view (critical cross section) shown.
In order to compare how the different load conditions and torques affect the stress states in the two sockets, graphs of the Von Mises stresses at maximum stress points in the sockets versus applied torque were created. Figure 3.17 presents the critical cross section of the two sockets (the bottom face of the sockets) and shows the maximum stress points A, B, and C, that are referenced to the Von Mises stress versus applied torque graphs. The stress points were arbitrarily chosen surrounding one of the loaded inner corners, but they could have been taken surrounding any of the loaded corners since this is where the maximum stresses occur according to the color stress contour plots. Figures 3.18a and 3.18b present the Von Mises Stress vs. Applied Torque for the maximum stress points on the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition I. According to these graphs, the Apex-3512 socket will begin to yield before the Apex-3212 socket, but both will begin to yield at the same location which is the vertex of the inner corner of the hexagonal opening (Node B). The Apex-3512 socket will reach the yield stress state of 150,000 psi at an applied torque of approximately 1640 in-lbs while the Apex-3212 will not reach a yield stress state until an applied torque of 1820 in-lbs. Thus for the ideal conditions modeled by Load Condition I, the socket would not near yielding under general use. According to Mr. Mark Langhout, the manager of Cooper Industries Apex Tool Division, the torque applied to these sockets in industrial use ranges from 400 to 600 in-lbs.

Figures 3.19a and 3.19b present the Von Mises Stress vs. Applied Torque for the maximum stress points on the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition II. Once again, the Apex-3512 socket will begin to yield before the Apex-3212 socket, but both will begin
to yield at the same place which is the point that is in tension from the action of the bolt acting on the hexagonal opening (Node A). The Apex-3512 socket will reach the yield stress state of 150,000 psi at an applied torque of approximately 900 in-lbs while the Apex-3212 will not reach a yield stress state until an applied torque of 1080 in-lbs. The maximum Von Mises stresses for Load Condition II are significantly higher than the the maximum Von Mises stresses for Load Condition I for the same given torques. According to these results, though, neither socket will begin to yield under normal industrial use for Load Condition II, which models the most realistic loading to be encountered in normal industrial use.

Figures 3.20a and 3.20b present the Von Mises Stress vs. Applied Torque for the maximum stress points on the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition III. Once again, the Apex-3512 socket will begin to yield before the Apex-3212 socket, but both will begin to yield at the same place which is the point that is in compression from the action of the bolt acting on the hexagonal opening (Node C). The Apex-3512 socket will reach the yield stress state of 150,000 psi at an applied torque of approximately 400 in-lbs while the Apex-3212 will not reach a yield stress state until an applied torque of 640 in-lbs. The maximum Von Mises stresses for Load Condition III are significantly higher than the the maximum Von Mises stresses for Load Condition I or II for the same given torques. According to these results these sockets will begin to yield under normal industrial use for Load Condition III, which models the worst case loading scenario to be encountered in industrial use. If this loading condition did arise in industrial use, the Apex-3212 socket should be used if
possible since it would not begin to yield until a significantly larger torque was applied as compared to the Apex-34512 socket.

Figure 3.17: Maximum Von Mises stress points A, B, and C, on the critical cross sections of the Apex-3212 and Apex-3512 sockets. This figure is used as the reference for Figures 3.18 through 3.20.
Von Mises Stress Vs. Applied Torque
Apex 3212 Hexagonal Socket
Load Condition I

Figure 3.18a: Maximum Von Mises stress points vs. applied torque for the Apex-3212 socket, Load Condition I.
Von Mises Stress Vs. Applied Torque
Apex 3512 Thin Walled Hexagonal Socket
Load Condition I

Figure 3.18b: Maximum Von Mises stress points vs. applied torque for the Apex-3512 socket, Load Condition I.
Von Mises Stress Vs. Applied Torque
Apex 3212 Hexagonal Socket
Load Condition II

Figure 3.19a: Maximum Von Mises stress points vs. applied torque for the Apex-3212 socket, Load Condition II.
Figure 3.19b: Maximum Von Mises stress points vs. applied torque for the Apex-3512 socket, Load Condition II.
Figure 3.20a: Maximum Von Mises stress points vs. applied torque for the Apex-3212 socket, Load Condition III.
Figure 3.20b: Maximum Von Mises stress points vs. applied torque for the Apex-3512 socket, Load Condition III.
The deformed shape models of the Apex-3212 and Apex-3512 solid models were created using analysis displacement data for Load Condition I and an applied torque of 500 in-lbs, and are shown in Figures 3.21 and 3.22 respectively. Both plots represent the side views of the respective deformed socket models, and have an exaggeration factor of 50. The exaggeration factor is required so that the type of deformation taking place can be seen, since the nodal displacements are very small. In order to demonstrate the differences in plotter performances, Figure 3.21 was created using the thermal color plotter while Figure 3.22 was created using the laser printer. For this application, it can be seen that either plotter produces sufficient results, but generally for contour plots, the thermal color is preferred. Both plots of the deformed sockets show that the sockets deform in a twisting fashion, with the greatest deformation occurring near the driver end of the sockets. The Apex-3512 socket expands more at the driver end of the socket than the Apex-3212 under the same loading conditions. Very little deformation occurs near the driven end of either socket since both sockets are fixed at this end, thus it can be taken as a reference point for the deformation occurring throughout the remainder of each socket model. The deformed shape models allow one to visualize the strain state occurring in each model.
Figure 5.21  Deformed model of the Apex-3212 socket for Load Condition 1 and an applied torque of 500 in-lbs. Side view shown with an exaggeration factor of 50.
Figure 3.22: Deformed model of the Apex-3512 socket for Load Condition I and an applied torque of 500 in*lbs. Side view shown with an exaggeration factor of 50.
Analytical Verification of the Solid Models

In order to verify if the finite element models were accurately predicting the actual Von Mises stress states in the tool socket, an analytical torsional analysis was performed for a section of each socket for a specified torque. The torque chosen for presentation purposes was 500 in-lbs. The calculations for each torsional analysis is found in Appendix B. The midsections of the sockets were considered where the inner wall is cylindrical and parallel to the outer wall. Localized stress concentration factors were not included and it was assumed that the socket section was in pure shear. The maximum and minimum outside diameters were considered along with an averaged diameter to show the approximate Von Mises stress ranges.

The results of the analytical torsional analysis on the two sockets were compared to the Von Mises stress levels found in the midsections of the two sockets from the finite element analysis cases. The Von Mises stress levels determined for the Apex-3212 was 6,466 psi for the maximum outside radius and 8,874 psi for the minimum outside radius possible for the midsection of that socket. The Von Mises stress level determined for the averaged outside radius was found to be 8,140 psi. For the Apex-3512 socket, the Von Mises stress levels determined were 6,466 psi for the maximum outside radius and 14,825 psi for the minimum outside radius possible for the midsection of that socket. The Von Mises stress level determined for the averaged outside radius was found to be 10,500 psi. Referring to Figures 3.8a&b for Load Condition I, it is seen that the Von Mises stress levels determined from the analytical torsional analysis are
within the ranges found in the finite element stress contour plots near the midsections of the two sockets.

As was stated before, stress concentrations near the ends of the sockets were not considered, but this torsional analysis supports the finite element model results since the determined and calculated stress levels fall into the same ranges for each tool socket.

**Two Dimensional Adaptive Analysis**

The next phase of the analysis process was to perform a two dimensional analysis on the critical cross sections that were determined from the three dimensional analysis of the two sockets. The sockets' hexagonal openings at the driver end of the sockets create stress concentrations at the vertices of the openings. This area requires a refined mesh in order to produce the most accurate results. Refining the mesh can basically be accomplished in one of two ways: manual remeshing or adaptive meshing. Manual remeshing means that the designer, utilizing a convergence criterion such as a maximum increase in stress between elements be limited to 10-15%, must inspect each element after an analysis and decide if remeshing of a region is required. If it is determined that remeshing is required, then the designer must manually delete the affected mesh region and replace it with a refined mesh. The manually refined mesh usually just has smaller elements, and thus more of them. This process was used in verifying and then remeshing the Apex-3212 and Apex-3512 solid models to an extent to obtain the most acceptably accurate results. But this method is very tedious and time consuming, and also limited in its ability to mesh transition regions. Thus, in applicable
circumstances the method of adaptive meshing and analysis is generally more efficient and desirable.

Adaptive mesh refinement is the technique for improving the finite element mesh through various means in order to cause successive sets of analysis results to converge within an acceptable error range. The increased accuracy with a new mesh can usually be obtained in one of two ways. The size of the elements can be decreased, thus using more elements and holding the polynomial order of the element fixed. This is called the h-adaptive technique.

The other method consists of increasing the order of the polynomial of the elements while holding the number of elements fixed. This is called the p-adaptive meshing technique. It is named this because the individual elements may be analyzed up to any given polynomial (P) order, generally up to 8 or 9. The simplest H finite elements are analyzed only to the linear order, but most h-FEM packages offer parabolic elements with midside nodes. When using a finite element package that offers only h-adaptive meshing, the user must initially specify whether each element in the model is linear or parabolic, and this assignment is fixed for the following analysis. But in FEM packages that offer p-adaptive meshing, the polynomial of each element may be varied by the software during the analysis. The polynomial order is iteratively increased until the results show convergence from one iteration to the next of a specified criterion such as Von Mises stresses. The geometric p-elements may be substantially larger than the h-elements to achieve the same accuracy in the results. Thus a geometric p-element model can typically have 10 to 100 times fewer elements than a corresponding traditional finite element model.
The I/FEM adaptive analysis is an automated process that uses h-adaptive technique to render a finite element model that results are accurate to a specified degree. The process involves determining the error in an analysis and then using this error indicator to selectively refine affected elements. This model is then reanalyzed until the selected criterion data converges to the required level of accuracy. Meshes created by using automeshing or mapped meshing can undergo I/FEM's h-adaptive refinement and analysis. In order to use the automated adaptive analysis the user meshes the model, selects geometry based boundary conditions, and then selects the command "Run Adaptive Structural Analysis". The adaptive analysis requires two pieces of user specified data, a convergence criterion and a refinement criterion.

The convergence criterion is the nodal or elemental data to be converged to the required level of accuracy. It is important to note that some resultant quantities converge faster than others, such as displacements which converge much faster than stresses. The maximum value of the data is recorded during the adaptive cycling, and an estimation is made of the error in this quantity by comparing its value to the value obtained during the preceding cycle.

The refinement criterion provides a basis for determining the accuracy of the analysis. This is employed as a criterion for selectively refining elements. A range of values for the refinement criterion is specified and this range is then used to choose the correct elements for refinement.

Unfortunately, at the present time the current version of I/FEM can perform h-adaptive analysis on two dimensional plate and shell elements
only. Therefore this method of adaptive analysis could not be used in the analysis of the Apex 3212 and 3512 solid finite element models, because they were constructed using solid hexahedral elements. But this method was used in the two dimensional analysis of the critical cross sections of the two Apex sockets to produce accurate results efficiently.

The p-type adaptive meshing technique that is available for use with the I/FEM system was developed by the Rasna Corporation, and is called I/FEMRASNA for this system. This software exploits the properties of higher order elements to create more accurately shaped elements. I/FEMRASNA has a general element library that consists of plate, shell, beam, spring and solid elements, and can produce 2-D and 3-D linear static or modal dynamic analyses. The p-adaptive code enacts an algorithm that selectively raises the polynomial order of the elemental shape functions only where it is determined to significantly improve the convergence of the results. The polynomial shape functions are associated with the edges of the elements. Since fewer elements are required, the total number of equations are greatly reduced, with considerable savings in computational time. The geometry is represented exactly without the need to facet curves or curved surfaces. It is important to note that loads and boundary conditions must be applied to the geometry and not to the individual nodes or elements. I/FEMRASNA will fail to operate properly if the loads and constraints are nodal or elemental.

There are several areas where the p-adaptive technique needs substantial development, one of which has already been identified. The strength of the high-order elements, its ability to yield accurate results in the presence of large stresses that vary from one side if the element to the
other, causes an important problem. The material could be in plastic deformation in a subregion of the element, such that the material properties would vary within a single element. Also, in a solid element, a finite element load applied to a node or edge results in a local stress singularity at the point of application. This singularity has a significant effect on the polynomial coefficients of elements touching this singularity. Since the stresses within the element rely on the same coefficients, results over the entire element are affected by the local singularity. To deal with this problem, it is customary to surround the singularity with smaller elements to obtain more accurate results.

The implementation of both methods of adaptive meshing proceeded in a trial and error fashion for the analysis of the two dimensional models of the socket critical cross-sections. A wide variety of load, constraint, mesh geometry, and material combinations were tested to determine which procedures would produce acceptable models and results. It was decided that for the initial experimentation with the methods that Load Condition II would be used so that mesh refinement could be easily observed due to the symmetry of the load vectors.

The first trial attempted automeshing the socket surface, as seen in Figures 3.23 and 3.24, utilizing completely geometry based constraints, loads, and material and property definitions. The geometry based features were placed on the model before automeshing the surface so that the automesher would recognize the loads and constraints and place nodes at those points. This trial and several others similar to it did not work because the automesher could not properly recognize the point loads on the inner edges of the model and created an invalid inner edge. The system
Figure 3.23: Apex-3212 socket critical cross section surface used for I/FEM two dimensional automeshing.
Figure 3.24: Apex-3212 socket critical cross section surface with geometry based features in place.
then could not mesh the doubly connected region and would freeze up, requiring that the process be killed.

Since the geometry point loads were determined to be the source of the trouble, the next logical step was to experiment with the application of the loads. The socket surface was created, the geometry based material and property definitions were applied, the geometry constraints were placed, and the surface was successfully automeshed. Then the geometry based point loads were placed on the model and the model was verified for validity. The analysis proceeded and was completed very quickly, but no stresses of strains were calculated upon review of the results. It was thus concluded that in order for the automesher to recognize geometry based features, the features must be applied to the model prior to automeshing the model.

The next trial attempted to use geometry based features with a manually placed mapped mesh. The geometry features were applied to the surface, including geometry point constraints and loads, the surface was mapped meshed by creating individual edges and using the doubly connected region command. The model was checked for validity, but an error message was returned stating that no element material or properties were defined. Thus it was deduced that the mapped mesher does not recognize geometry based features. The geometry features were then removed from the model and nodal forces and constraints were applied to the model along with elemental material and property definitions. This model was verified and a simple static analysis was run producing the expected stress strain results. A geometric analysis using FEMRASNA was attempted on this model but failed with the error code stating that there
was an Engine Module Failure. A call by Dr. Bhavin Mehta to the Intergraph software support group verified that this error was caused by not using geometry based features. An analysis was then attempted using h-adaptive meshing. The first iteration in the h-adaptive cycle proceeded properly, but when the system attempted to subdivide refinement-required elements, it could not, stating that there was no elemental geometry present for the refinement. Thus both the p-adaptive and h-adaptive processes failed for this type of model definition.

The next model creation trial attempt proved to be successful for the I/FEM h-adaptive meshing routine. The socket surface model was created, geometry material and property definitions were placed, and the surface and outer edge was properly constrained using geometry constraints. The surface was automeshed. This time nodal loads were applied to the automeshed surface instead of geometry point loads (Figure 3.25). The model was verified and an h-adaptive analysis was performed. The default convergence and refinement criterions were used with an acceptable convergence value being 10%. The default convergence criterion used was the strain energy density (SED) and the default refinement criterion used was the maximum principal stress (S1). The analysis took three iterations to come within the specified convergence value of 10%. The model went from 78 elements to 266 elements and was only refined in areas where stress concentrations would be suspected. It was thus concluded that this is a successful modeling approach when using h-adaptive meshing.

A total of ten different trials were completed using this type of model creation and h-adaptive meshing. Basically, the convergence criterion and refinement criterion were varied to see how they affected the
Figure 3.25: Automated Apex-3212 socket critical cross section surface with geometry based constraints and nodal loads.
h-adaptive analysis. The best results were obtained when using the Henke Von Mises Stress (HVM) for both the convergence criterion and the refinement criterion. The solution for this case converged in 12 seconds after only two iterations for Load Condition II. The model began with 16 elements and was finished with only 32 elements. The mesh refinement for the critical cross sections of the Apex-3212 and 3512 sockets were the same for like loading conditions because the geometry was very similar with only the outside radius being different for the cross sections. The resultant refined meshes for Load Conditions I through III can be seen in Figures 3.26 through 3.28, respectively. These meshes were refined in a logical and uniform manner from the initial mesh shown in Figure 3.25.

It was found that some combinations of convergence and refinement criterions took much longer to converge and produced poorly refined meshes. One example of this problem is using strain energy density as the convergence criterion and Henke Von Mises stress as the refinement criterion. This model, which also began with 16 elements, took several minutes to converge and produced a poorly refined mesh. The mesh was highly refined in one area, but was completely unrefined in areas that should have been refined due to the loading configuration. The final mesh contained 137 elements, more than necessary had a different convergence/refinement criterion combination been chosen. Thus it is important to define convergence and refinement criterions that will produce the most accurate results in the least amount of time.

I/FEMRASNA was then run on the same type of model that proved successful with the h-adaptive solver. Constraints were geometry based while loads were nodal based and placed after the mesh was created. This
Figure 3.26: Automeshed Apex-3212 socket critical cross section after H-adaptive mesh refinement for Load Condition I. Convergence criterion: HVM, refinement criterion: HVM.
Figure 3.27: Automeshed Apex-3212 socket critical cross section after H-adaptive mesh refinement for Load Condition II. Convergence criterion: HVM, refinement criterion: HVM.
Figure 3.28: Automeshed Apex-3212 socket critical cross section after H-adaptive mesh refinement for Load Condition III. Convergence criterion: HVM, refinement criterion: HVM.
model also was created using 16 low order elements, as was done in the previous h-adaptive trials. A convergence value was set at 15% and the response quantities were maximum principal stress and displacements in the x and y directions. The first p-adaptive trial failed immediately, and after review of the error file, it was determined that analysis setup should be set on '3-d analysis' and not 'plane stress analysis'. The model was over-constrained in the 'plane stress analysis' mode, causing the run to fail. The mode was changed accordingly, and the analysis began its p-loop passes. The analysis was run to polynomial order of 9, but did not converge to the designated convergence value of 15%. For the next run, the convergence value was changed from 15% to 25% and the other parameters were unchanged. This analysis converged in two minutes and ten seconds, with the final polynomial order being three. It should be noted that 25% is not a very good convergence value, 10% or 15% would produce more accurate results. Therefore it was deemed necessary to increase the number of elements in the total mesh and rerun the analysis for a convergence value of 10%. The evenly refined mesh contained twice the number of elements of the original mesh : 32. This run also proved to be successful, but it took over eight minutes to run and finished with a final p-order of 6 for the elements.

From these FEMRASNA runs, it can be seen that it is important to choose an acceptable convergence criterion and an adequately refined mesh to produce the most accurate results in the least amount of time.

The final experiment combined both h-adaptive and p-adaptive techniques to determine if they could be used together to make the analysis process more efficient. The h-analysis was run first for one iteration to
selectively refine the coarse mesh of 16 elements. Its convergence criterion and refinement criterion were maximum principal stress (S1) and the convergence value was set at 10%. After one iteration, the number of elements were 52 and the elapsed time was 20 seconds. Then the p-adaptive analysis was run on this model with a convergence value set at 10% and the response quantities being maximum principal stress and displacements in the x and y directions. The solution converged in 11 minutes with the final p-order being 4. Thus it was proved that these two methods can be used together, but not always with the most efficient results.

For the case of the modeling of the Apex socket cross-sections presented here, it was found that the FEM h-adaptive meshing was the most efficient method of analysis, providing accurate results in a short amount of time.

The FEMRASNA p-adaptive analysis gave accurate results very similar to those obtained from the h-adaptive runs, but it took much longer to converge and produce the solution. In other cases, though, p-adaptive meshing would be the logical choice as the analysis package. For example, h-adaptive meshing cannot handle solid three-dimensional meshes, whereas p-adaptive meshing can generally produce a solution for that type of mesh, as long as boundary conditions and material properties are geometry based.

The results of the two dimensional adaptive analysis are presented in Figures 3.29a &b through Figures 3.31a & b. Figures 3.29a and 3.29b present the Von Mises stress contour plots of the critical cross section of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition I.
Figures 3.30a and 3.30b are the color shaded Von Mises stress contour plots of the critical cross section of the Apex-3212 and Apex-3512 sockets, respectively, for Load Condition II. Figures 3.31a and 3.31b present the Von Mises stress contour plots for Load Condition III for the two socket critical cross sections. A comparison of these results to the results obtained from the three dimensional solid model analysis (Figures 3.8 - 3.16) show that the Von Mises stress ranges are very similar for both sockets for each Load Condition. Thus the results of the two dimensional adaptive analysis support the results of the three dimensional analysis. Differences in the stress contours between the two dimensional and three dimensional analyses can be attributed to the fact the two dimensional models had to be fully constrained on the outer edge of the cross sections in order for the models to be analyzed. Therefore, the results of the three dimensional analysis more accurately represent the stress state found in the socket bodies.
Figure 3.29a: Von Mises stress contour plot of the Apex-3212 two dimensional model of the critical cross section for Load Condition I
Figure 3.29b: Von Mises stress contour plot of the Apex-3512 two dimensional model of the critical cross section for Load Condition I.
Figure 3.30a: Von Mises stress contour plot of the Apex-3212 two dimensional model of the critical cross section for Load Condition II.
Figure 3.30b: Von Mises stress contour plot of the Apex-3512 two dimensional model of the critical cross section for Load Condition II.
Figure 3.31a: Von Mises stress contour plot of the Apex-3212 two-dimensional model of the critical cross section for Load Condition III.
Figure 3.34b: Von Mises stress contour plot of the Apex-3512 two dimensional model of the critical cross section for Load Condition III.
Conclusions and Recommendations

Conclusions

1. The finite element modeling techniques used in this study are acceptable for determining Von Mises and maximum principal stress states in the given tool sockets. The two dimensional adaptive finite element analysis and the analytical analysis support the results obtained from the three dimensional finite element analysis of the Apex tool sockets.

2. The Apex-3512 thin walled tool socket had a consistently higher stress state than the Apex-3212 tool socket for the same Load Condition, regardless of the Load Condition considered. This is due to the fact the load is distributed throughout a larger socket section in the Apex-3212 socket. Thus under similar load conditions, the Apex-3512 tool socket would begin to yield before the Apex-3212 tool socket.

3. The maximum Von Mises stress concentrations for both sockets occurred at the driver end of the sockets, with the highest stresses occurring in the inner loaded corners of the hexagonal openings in the sockets. This can be seen clearly in the stress contour plots presented in Chapter 3. The sockets would begin to yield at these points first when the yielding torque was achieved for each socket.
4. The results presented for Load Condition II represent the most probable case when removing or tightening bolts with the sockets, since it presents the kinematically stable case of three contact points between the socket and bolt head. The results presented for Load Condition I represent the ideal loading condition where all six faces of the bolt head comes in contact with the socket. The results given for Load Condition III represent the worst case scenario where the bolt head only contacts two points of the socket hexagonal driver opening. This case would probably be unlikely since it is not kinematically stable, unless the bolt was sufficiently deformed.

5. The angle of the load vectors with respect the the hexagonal opening walls on the two sockets did not affect the magnitude of the stress states in the sockets as it it was changed. The shape of the maximum stress contours near the driver end of the sockets were changed slightly as this angle was varied, but it was not appreciable.

6. For the case of the modeling of the Apex socket cross-sections presented in the two dimensional adaptive analysis, it was found that the I/FEM h-adaptive meshing was the most efficient method of analysis, providing accurate results in a relatively short amount of time. The FEMRASNA p-adaptive analysis gave accurate results similar to those obtained from the h-adaptive runs, but it took much longer to converge and produce a solution. As adaptive mesh technology grows in the future, solid h-adaptive meshing could become a reality and p-adaptive analysis should become more competitive with h-adaptive analysis.
Recommendations

This study does not conclude the necessary analysis work to be completed on Apex tool sockets. This study was limited to two similar hexagonal-opening tool sockets with the same size drive and driver openings. Other sizes and shapes of sockets should be studied, along with different types of boundary conditions.

Changing the material properties of the sockets will have an effect on resultant stress and strain states in the sockets. This could be achieved by changing the modulus of elasticity and Poisson's ratio to other potential tool socket materials.

Changing the shapes of the sockets would also influence the stress states obtained in the sockets. Adding more material near the driver end of socket where the maximum Von Mises stresses occur would lower the stress state and allow the socket to be utilized to a higher torque level before yielding begins. This would be accomplished by creating a larger wall thickness near the driver end of the socket.

Rounding the inner corners of the hexagonal opening in the driver end of the sockets should be studied to determine its effects in reducing the stress concentrations at these points.

The finite element models of the sockets considered the material to be isotropic, but in reality the material has been influenced by cold work done on it during the punching operations. Also fatigue conditions are not considered in the finite element analysis of the sockets. Thus further experimental tests should be conducted on the sockets to further prove the durability and reliability of these tool sockets under everyday usage.
The finite element analysis could also be extended to the forming of these sockets through the present procedure of punching. This would mean that large bulk plastic deformation would be considered thus complicating the analysis. The entire stress/strain curve for the tool material would be required and I/FEM could not be used since it cannot handle this kind of finite element analysis.

A forging process could also be examined for producing these tool sockets. Forging would probably produce a stronger socket since the grain boundaries in the tool material would not be disrupted as they would be in the punching operation.
REFERENCES


APPENDIX A

Load Vector Calculations

The load vectors are calculated to produce an applied torque to the Apex-3212 and Apex-3512 socket finite element solid models. Three different loading conditions at five different torques per condition were considered. The load vectors are placed at the appropriate nodes near the driver end of the socket while the opposite end was constrained. Condition I models contact between the socket and a bolt at each corner of the hexagonal opening in the socket using load vectors, modeling the ideal condition. Condition II models contact between the socket and a bolt only at three corners of the hexagonal opening in the socket using load vectors, and Condition III models contact between the socket and a bolt only at two opposite corners of the hexagonal opening in the socket, the worst case.

**Determination of the required load vectors:**

Perpendicular distance (Rh) from inner hexagon side to center:

\[ Rh = 0.1905 \text{ inches} \]

Total force (TF) required to produce the required torques (LC#):

- **LC50:** \( TF1 = 50 \text{ in} \times \text{lbs} / 0.1905 \text{ in} = 262.47 \text{ lbs} \)
- **LC100:** \( TF2 = 100 \text{ in} \times \text{lbs} / 0.1905 \text{ in} = 524.93 \text{ lbs} \)
- **LC250:** \( TF3 = 250 \text{ in} \times \text{lbs} / 0.1905 \text{ in} = 1312.33 \text{ lbs} \)
- **LC500:** \( TF4 = 500 \text{ in} \times \text{lbs} / 0.1905 \text{ in} = 2624.67 \text{ lbs} \)
- **LC750:** \( TF5 = 750 \text{ in} \times \text{lbs} / 0.1905 \text{ in} = 3937.01 \text{ lbs} \)
Condition I:

Individual load vectors \( F \) for applied at four nodal layers:

- \( \text{ILC50:} \quad F_1 = 262.47 \text{ lbs/24 nodes} = 10.94 \text{ lbs/node} \)
- \( \text{ILC100:} \quad F_2 = 524.93 \text{ lbs/24 nodes} = 21.87 \text{ lbs/node} \)
- \( \text{ILC250:} \quad F_3 = 1312.33 \text{ lbs/24 nodes} = 54.68 \text{ lbs/node} \)
- \( \text{ILC500:} \quad F_4 = 2624.67 \text{ lbs/24 nodes} = 109.36 \text{ lbs/node} \)
- \( \text{ILC750:} \quad F_5 = 3937.01 \text{ lbs/24 nodes} = 164.00 \text{ lbs/node} \)

Condition II:

Individual load vectors \( F \) for applied at four nodal layers:

- \( \text{IIILC50:} \quad F_1 = 262.47 \text{ lbs/12 nodes} = 21.87 \text{ lbs/node} \)
- \( \text{IIILC100:} \quad F_2 = 524.93 \text{ lbs/12 nodes} = 43.74 \text{ lbs/node} \)
- \( \text{IIILC250:} \quad F_3 = 1312.33 \text{ lbs/12 nodes} = 109.36 \text{ lbs/node} \)
- \( \text{IIILC500:} \quad F_4 = 2624.67 \text{ lbs/12 nodes} = 218.72 \text{ lbs/node} \)
- \( \text{IIILC750:} \quad F_5 = 3937.01 \text{ lbs/12 nodes} = 328.07 \text{ lbs/node} \)

Condition III:

Individual load vectors \( F \) for applied at four nodal layers:

- \( \text{IIILC50:} \quad F_1 = 262.47 \text{ lbs/8 nodes} = 32.81 \text{ lbs/node} \)
- \( \text{IIILC100:} \quad F_2 = 524.93 \text{ lbs/8 nodes} = 65.62 \text{ lbs/node} \)
- \( \text{IIILC250:} \quad F_3 = 1312.33 \text{ lbs/8 nodes} = 164.04 \text{ lbs/node} \)
- \( \text{IIILC500:} \quad F_4 = 2624.67 \text{ lbs/8 nodes} = 328.08 \text{ lbs/node} \)
- \( \text{IIILC750:} \quad F_5 = 3937.01 \text{ lbs/8 nodes} = 492.12 \text{ lbs/node} \)
**Condition I, 2-D analysis**:  
Individual load vectors \( (F) \) for applied at single nodal layer:

\[
\begin{align*}
\text{ILC50} : & \quad F_1 = 262.47 \text{ lbs/6 nodes} = 43.75 \text{ lbs/node} \\
\text{ILC100} : & \quad F_2 = 524.93 \text{ lbs/6 nodes} = 87.49 \text{ lbs/node} \\
\text{ILC250} : & \quad F_3 = 1312.33 \text{ lbs/6 nodes} = 218.72 \text{ lbs/node} \\
\text{ILC500} : & \quad F_4 = 2624.67 \text{ lbs/6 nodes} = 437.45 \text{ lbs/node} \\
\text{ILC750} : & \quad F_5 = 3937.01 \text{ lbs/6 nodes} = 656.17 \text{ lbs/node}
\end{align*}
\]

**Condition II, 2-D analysis**:  
Individual load vectors \( (F) \) for applied at single nodal layer:

\[
\begin{align*}
\text{IILC50} : & \quad F_1 = 262.47 \text{ lbs/3 nodes} = 87.49 \text{ lbs/node} \\
\text{IILC100} : & \quad F_2 = 524.93 \text{ lbs/3 nodes} = 174.98 \text{ lbs/node} \\
\text{IILC250} : & \quad F_3 = 1312.33 \text{ lbs/3 nodes} = 437.44 \text{ lbs/node} \\
\text{IILC500} : & \quad F_4 = 2624.67 \text{ lbs/3 nodes} = 874.89 \text{ lbs/node} \\
\text{IILC750} : & \quad F_5 = 3937.01 \text{ lbs/3 nodes} = 1312.33 \text{ lbs/node}
\end{align*}
\]

**Condition III, 2-D analysis**:  
Individual load vectors \( (F) \) for applied at single nodal layer:

\[
\begin{align*}
\text{IIILC50} : & \quad F_1 = 262.47 \text{ lbs/2 nodes} = 131.24 \text{ lbs/node} \\
\text{IIILC100} : & \quad F_2 = 524.93 \text{ lbs/2 nodes} = 262.47 \text{ lbs/node} \\
\text{IIILC250} : & \quad F_3 = 1312.33 \text{ lbs/2 nodes} = 656.16 \text{ lbs/node} \\
\text{IIILC500} : & \quad F_4 = 2624.67 \text{ lbs/2 nodes} = 1312.33 \text{ lbs/node} \\
\text{IIILC750} : & \quad F_5 = 3937.01 \text{ lbs/2 nodes} = 1968.50 \text{ lbs/node}
\end{align*}
\]
APPENDIX B

Analytical Torsional Analysis Calculations

Torsional analysis of the Apex-3212 and Apex-3512 sockets for verification of the average Von Mises stress levels found in the solid finite element models of the sockets. The midsections of the sockets were considered where the inner wall is cylindrical and parallel to the outer wall. Localized stress concentration factors were not included and it was assumed that the socket section was in pure shear. The maximum and minimum outside diameters were considered along with an averaged diameter to show the approximate Von Mises stress ranges.

Free Body Diagram:

\[ T \max = \frac{T \cdot r_o}{J} \quad \text{where} \quad J = \frac{\pi}{32} (d_o^4 - d_i^4) \]

Considering Von Mises Stresses for the plane stress condition of torsional loading on the shaft:

\[ \sigma' = \sigma_1^2 - \sigma_1 \sigma_2 + \sigma_2^2 \]

\[ \sigma' = T \max \]
Apex-3212 Socket:

\[ T = 500 \text{ in-lbs} \]
\[ \text{maximum } r_\circ : 0.375" \]
\[ \text{minimum } r_\circ : 0.312" \]
\[ \text{averaged } r_\circ : 0.342" \]
\[ r_i : 0.190" \]

Von Mises Stress level for maximum \( r_\circ \): \( \sigma' = 6,466 \text{ psi} \)
Von Mises Stress level for minimum \( r_\circ \): \( \sigma' = 8,874 \text{ psi} \)
Von Mises Stress level for averaged \( r_\circ \): \( \sigma' = 8,140 \text{ psi} \)

Apex-3512 Socket:

\[ T = 500 \text{ in-lbs} \]
\[ \text{maximum } r_\circ : 0.375" \]
\[ \text{minimum } r_\circ : 0.296" \]
\[ \text{averaged } r_\circ : 0.335" \]
\[ r_i : 0.190" \]

Von Mises Stress level for maximum \( r_\circ \): \( \sigma' = 6,466 \text{ psi} \)
Von Mises Stress level for minimum \( r_\circ \): \( \sigma' = 14,825 \text{ psi} \)
Von Mises Stress level for averaged \( r_\circ \): \( \sigma' = 10,500 \text{ psi} \)