INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

**Bell & Howell Information and Learning
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA**

UMI[®]
800-521-0600

**NONLINEAR ELASTIC-PLASTIC ANALYSIS
USING THE MODIFIED FINITE QUASI-PRISM ELEMENT**

DISSERTATION

**Presented in Partial Fulfillment of the Requirements for
the Degree Doctor of Philosophy in the Graduate
School of The Ohio State University**

By

Rebecca M. Hoffman, B.S.M.E., M.S.

**The Ohio State University
2000**

Dissertation Committee:

Gary L. Kinzel, Adviser

Ramana V. Grandhi, Co-Adviser

June K. Lee

Donald R. Houser

Approved by

Adviser

Department of Mechanical Engineering

Co-Adviser

UMI Number: 9962400

UMI[®]

UMI Microform 9962400

Copyright 2000 by Bell & Howell Information and Learning Company.

**All rights reserved. This microform edition is protected against
unauthorized copying under Title 17, United States Code.**

**Bell & Howell Information and Learning Company
300 North Zeeb Road
P.O. Box 1346
Ann Arbor, MI 48106-1346**

ABSTRACT

In increasingly constrained product development cycles, physical product testing is being replaced with virtual product testing, typically in the form of finite element analysis. To accurately model the service environment of the product, nonlinear analyses are often required. A nonlinear finite element model, however, is generally very time consuming to solve. To be a useful design tool, these solution times must be reduced while maintaining the accuracy of the method's results.

A modified version of the Finite Quasi-Prism (FQP) element is a very accurate and efficient alternative to conventional nonlinear elastic-plastic finite element analysis. The three-dimensional FQP element can be used in the nonlinear analysis of bodies of nearly prismatic geometry. This element can replace several standard eight-noded hexahedral elements by using higher order interpolation functions in the direction of the nearly constant cross-section.

The reduction in the number of elements translates into fewer finite element equations to solve, leading to a significant decrease in CPU time for nonlinear problems, which often require many solution iterations to achieve a final converged solution. Further, the analysis results using this element are smoother and very near theoretical solutions. The element has also proven to be very effective in regions of loading singularities such as

concentrated loads. Thus, by speeding the analysis time, the FQP element significantly reduces the design cycle cost, which demands repetitive simulations.

This dissertation addresses the formulation and validation testing of the Modified FQP element and its performance in nonlinear finite element analysis. The remainder of the dissertation concentrates on the effect of element orientation and interpolation order on solution accuracy and CPU times, the type of cases for which this element is most appropriate, and the results of several numerical examples.

For Ron

ACKNOWLEDGMENTS

I wish to express my sincere gratitude to my adviser Professor Gary Kinzel for his time and invaluable assistance over the course of this research. I also wish to thank my co-adviser Professor Ramana Grandhi of Wright State University as well as Professor J.K. Lee, and Professor Donald Houser for their assistance and careful review of this dissertation.

In addition to my committee, I would like to take this opportunity to thank a few of the many people who helped me develop and implement the finite element code, especially Professor Henry Busby, Sandeep Vijayakar, Kaiping Li, Jong-Tae Jinn and Olaf Storaasli from NASA Langley. Additional thanks goes to the Dayton Area Graduate Studies Institute (DAGSI), which funded a large portion of this research and to NASA for the use of their GPS4 sparse matrix solver.

Finally, I offer a special thanks to my husband, Ron, for his patience and support throughout this course of study.

VITA

November 20, 1969 Born – Sidney, Ohio

December 1992 B.S.M.E., University
of Dayton, Dayton, Ohio

January 1995 – March 1996 Graduate Research Assistant,
The Ohio State University,
Columbus, OH

August 1995 MS, The Ohio State University,
Columbus, OH

March 1996 – January 1997 Engineer, Product Design Center,
Westerville, OH

January 1997 – December 1999 Graduate Research Assistant,
The Ohio State University,
Columbus , OH

January 2000 – present Product Design Engineer, Ford
Motor Company, Dearborn, MI

PUBLICATION

Hoffman, R.M. and Busby, H.R., “Stress based convergence analysis for p -adaptive hierarchical finite element analysis,” *Finite Elements in Analysis and Design*, v. 34, 25-35 (2000).

FIELDS OF STUDY

Major Field: Mechanical Engineering

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	ii
DEDICATION	iv
ACKNOWLEDGMENTS	v
VITA	vi
TABLE OF CONTENTS	vii
LIST OF TABLES	x
LIST OF FIGURES	xi
 CHAPTERS:	
1. INTRODUCTION	1
1.1 Introduction and Research Objectives	1
1.2 Scope of the Dissertation	4
1.3 Dissertation Organization	5
2. NONLINEAR FINITE ELEMENT ANALYSIS BACKGROUND INFORMATION	6
2.1 Nonlinear Finite Element Analysis	6
2.2 Nonlinear Material Modeling	6
2.2.1 Yield Criteria	7
2.2.2 Constitutive Equations	8
2.2.3 Anisotropy	10

	<u>Page</u>
2.3 Solution Algorithms	11
2.3.1 Direct Solution Methods	12
2.3.1.1 Symmetric Banded Solutions	12
2.3.1.2 Skyline Solutions	13
2.3.1.3 Frontal Solutions	14
2.3.1.4 Sparse Matrix Solutions	15
2.3.2 Iterative Solution Methods	16
2.3.3 Solution Method Utilized	17
2.4 Analyses Using Higher Order Elements	18
2.4.1 The Finite Prism Method	19
2.4.2 The Finite Quasi-Prism Element	20
 3. NONLINEAR FINITE ELEMENT FORMULATION	 23
3.1 Introduction	23
3.2 Principle of Virtual Work	23
3.3 General Matrix Equations of Displacement-Based Continuum Elements	27
3.4 Newton-Raphson Iteration Technique	30
3.5 Determination of Strain	33
3.6 Determination of Stress	36
3.7 The Backward-Euler Return	40
 4. THE MODIFIED FINITE QUASI-PRISM ELEMENT	 44
4.1 Introduction	44
4.2 Element Formulation	45
4.3 Element Testing	49
4.3.1 Element Convergence	50
4.3.1.1 Rigid Body Displacement	50
4.3.1.2 Constant Strain State	55
4.3.1.3 Mesh Compatibility	56
4.3.2 Element Performance in Shear	56
4.3.3 Allowable Element Distortion	59
4.4 Summary	63
 5. ELEMENT PERFORMANCE IN NONLINEAR ANALYSIS	 65
5.1 Introduction	65
5.2 Pure Bending of an Elastic-Plastic Beam	66
5.2.1 Effect of Variation of the Order of Interpolation	67
5.2.2 Comparison to the Theoretical Solution	69
5.3 Elastic-Plastic Thick Cylinder with Internal Pressure	70
5.3.1 Accuracy and Solution Time Comparisons	71
5.3.2 Variation of Element Order	85

	<u>Page</u>
5.3 Elastic-Plastic Thick Cylinder with Internal Pressure and a Concentrated Radial Load	. 89
5.4 Square Beam Loaded in Torsion and Bending	. 98
5.5 Summary	.121
6. FINAL REMARKS	.122
6.1 Conclusions	.122
6.2 Research Contributions	.124
6.3 Recommendations for Future Research	.125
BIBLIOGRAPHY	.127

LIST OF TABLES

<u>Table</u>	<u>Page</u>
4.1 Displacement results of the shear-locking test (one third-order element).	. 58
4.2 The effect of nonprismatic element geometries for Case 1 (pure bending).	. 59
4.3 The effect of nonprismatic element geometries for Case 2 (cantilever beam loaded to yielding).	. 62
4.4 The effect of nonprismatic element geometries for Case 3 (cantilever beam, plastic deformation).	. 62
5.1 The solution accuracy comparison for the $y_I=0.40$ in case as the order of interpolation is varied from $\zeta=3$ to $\zeta=5$	. 68
5.2 The solution accuracy comparison for the $y_I=0.20$ in case as the order of interpolation is varied from $\zeta=3$ to $\zeta=5$	. 69
5.3 Comparison of solution accuracy for the five beam pure bending cases. .	. 70
5.4 CPU time comparison for the radial axode model.	. 76
5.5 CPU time comparison for the circumferential axode model.	. 81
5.6 Comparison of CPU time and total number of iterations to convergence for varying orders of the circumferentially oriented axode model. . .	. 84
5.7 Comparison of CPU time for varying orders of the radially oriented axode model.	. 89
5.8 CPU time comparison for the models with internal pressure and a concentrated radial load.	. 98

LIST OF FIGURES

<u>Figure</u>	<u>Page</u>
1.1 Schematic illustrating the effect of design change timing on product cost. (Taken from Ryan, 1992)	2
3.1 The yield surface for the backward-Euler return method.	42
4.1 The Finite Quasi-Prism element. ζ is the direction of nearly constant cross-section requiring the higher order interpolation.	45
4.2 Two-dimensional quadratic elements: a) Serendipity element, b) Lagrangian element.	47
4.3 Model for the rigid body displacement test.	51
4.4 Shear force and bending moment diagrams for the plastically loaded cantilever beam of the rigid body displacement test.	52
4.5 Normal stress results near the top surface of the beam for the rigid body displacement test.	53
4.6 Vertical displacement results for the rigid body displacement test.	54
4.7 Two-element model used for the patch tests.	55
4.8 The third order one element cantilever beam model used for the shear- locking test.	57
4.9 Normal stress results of the shear-locking test (one third-order element).	58
4.10 Three-element model used to test the effect of nonprismatic element geometry. a) Case 1: pure bending of the beam, b) Case 2: cantilever beam loaded to yielding, and c) Case 3: plastically deformed cantilever beam.	60

<u>Figure</u>	<u>Page</u>
4.1.1 Deformed element cantilever beam models.	. 61
5.1 Schematic of the beam used for the pure elastic-plastic bending example. .	. 66
5.2 Material properties and the theoretical stress distribution of the beam. .	. 67
5.3 The two-element model used for the variation of the order of Interpolation study.	. 68
5.4 The Modified FQP model of the thick cylinder with radial axodes. . .	. 72
5.5 Variation of the equivalent stress along the radius of the cylinder for the model given in Figure 5.1.	. 73
5.6 Variation of the radial stress along the radius of the cylinder. . . .	. 74
5.7 Variation of the circumferential stress along the radius of the cylinder. .	. 75
5.8 The Modified FQP model of the thick cylinder with circumferential axodes.	. 77
5.9 Variation of the equivalent stress along the radius of the cylinder for the model given in Figure 5.5.	. 78
5.10 Variation of the radial stress along the radius of the cylinder. . . .	. 79
5.11 Variation of the circumferential stress along the radius of the cylinder. .	. 80
5.12 Variation of the radial stress along the radius of the cylinder for varying orders of modified FQP elements with circumferentially oriented axodes. .	. 82
5.13 Variation of the circumferential stress along the radius of the cylinder cylinder for varying orders of modified FQP elements with circumferentially oriented axodes.	. 83
5.14 Variation of the equivalent stress along the radius of the cylinder for varying orders of modified FQP elements with radially oriented axodes..	. 86
5.15 Variation of the radial stress along the radius of the cylinder for varying orders of modified FQP elements with radially oriented axodes. . . .	. 87
5.16 Variation of the circumferential stress along the radius of the cylinder for varying orders of modified FQP elements with radially oriented axodes.	. 88

<u>Figure</u>		<u>Page</u>
5.17	Modified FQP models of the thick cylinder with internal pressure and a concentrated radial load: a) circumferentially oriented elements, b) radially oriented elements.	. 90
5.18	Variation of the equivalent stress along the cylinder radius at an angle of 0° from the horizontal.	. 92
5.19	Variation of the equivalent stress along the cylinder radius at an angle of 54° from the horizontal.. . . .	. 93
5.20	Variation of the equivalent stress along the cylinder radius at an angle of 90° from the horizontal.	. 94
5.21	Variation of the equivalent stress along the inner circumference of the cylinder, radius $r = 0.15$ in.	. 95
5.22	Variation of the equivalent stress along the outer circumference of the cylinder, radius $r = 0.30$ in.	. 96
5.23	The four-element Modified FQP model of the beam with a square cross-section loaded in torsion, bending and shear.	. 99
5.24	The uniform finite element mesh used for the first ANSYS analysis.	.101
5.25	Deformation of the loaded corner elements of the mesh in Figure 5.21.	.102
5.26	The second iteration of the ANSYS finite element mesh. The elongated end elements are three times longer than the remainder of the mesh.	.105
5.27	Deformation of the loaded corner elements of the mesh in Figure 5.23.	.106
5.28	The third iteration of the ANSYS finite element mesh. The elongated end elements are two times longer than the adjacent elements and six times longer than the remainder of the mesh.	.108
5.29	Deformation of the loaded corner elements of the mesh in Figure 5.25.	.109
5.30	Equivalent stress results for the beam model illustrated in Figure 5.25.	.110
5.31	The theoretical equivalent stress distribution for a square rod loaded in pure torsion (taken from Annin and Cherepanov, 1988).	.111
5.32	The theoretical shear stress distribution for a square rod loaded in pure torsion. Note that the shear stresses τ_{xz} and τ_{yz} are both included on the same plot. (Taken from Annin and Cherepanov, 1988.)	.112

<u>Figure</u>		<u>Page</u>
5.33	The mesh of Figure 5.24 with a uniformly distributed torsional load.	.114
5.34	The Modified FQP model equivalent stress contour plot for the beam cross-section located 8.8 in from the fixed boundary.	.115
5.35	The ANSYS model equivalent stress contour plot for the beam cross-section located 8.8 in from the fixed boundary.	.116
5.36	The Modified FQP model shear stress (τ_{xz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.	.117
5.37	The ANSYS model shear stress (τ_{xz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.	.118
5.38	The Modified FQP model shear stress (τ_{yz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.	.119
5.39	The ANSYS model shear stress (τ_{yz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.	.120

CHAPTER 1

INTRODUCTION

1.1 Introduction and Research Objectives

Product design involves three fundamental steps: designing the part, building a prototype of the part based on this design, and testing the prototype. Given increasingly constrained product development cycles, how well these steps are integrated strongly affects a manufacturer's ability to compete in the marketplace. At one time, these three steps were distinct; a designer drew the part, handed the drawings to the model shop technician who made the part, who in turn handed the part to the test engineer. If a problem was encountered during testing, the failed part was returned to the designer who would revise the design. This process was repeated until a part meeting the design specifications resulted.

Designing parts by this method is expensive and time consuming. The high cost and long product development lead time associated with this method resulted in manufacturers who were unable to respond quickly to market changes. Through procedures such as Concurrent Engineering, Design for Assembly and Design for Manufacturing, design engineers were teamed with testing, manufacturing and reliability engineers to pool expertise and avoid expensive, and often foreseeable, product design

flaws. These procedures, when properly instituted, worked well to reduce the number of design-build-test iterations required to develop a product, thus saving development costs and significantly reducing product time to market.

As illustrated is illustrated in Figure 1.1 below, Ryan (1992) found that every \$100 spent to correct a design flaw at the production phase is equivalent to \$10 spent correcting that flaw at the prototype phase and \$1 spent correcting it at the design phase. With this in mind, manufacturers looked to the design phase of the product development cycle to further reduce their costs and accelerate their time to market. The idea is that if the product is designed well, it is necessary to build only one prototype, which is tested for design verification purposes only.

With this “one prototype” goal in mind, design emphasis has shifted from physical part testing to computer modeling and simulation. That is, the part is tested for fit,

Figure 1.1: Schematic illustrating the effect of design change timing on product cost.
(Taken from Ryan, 1992)

function, manufacturability, structural worthiness and durability on the computer long before the first prototype part is ever built. The same iterative design procedure is occurring, but now with a virtual part. Only when the design passes this virtual testing is a physical prototype built and tested, production tooling made and so forth. Thus, because of the growing importance of simulation in the design process, it is important that the analysis method be efficient, accurate and robust. Further, it is important that the analysis accurately model the operating conditions of the finished part. In most cases, this requires nonlinear three dimensional finite element analysis.

Three dimensional nonlinear finite element analyses, however, are time consuming to solve. Oftentimes, even after long solution times and strict incremental convergence criteria, there is no guarantee that the solution obtained through the analysis is the true solution. The reasons for inaccurate solutions are many and among other reasons may include:

- an improper time step may have been chosen
- improper convergence criteria may have been input
- the mesh may have deformed substantially in the course of the analysis resulting in a poor mesh and poorly shaped elements by the final load increments
- an improper material model may have been chosen
- the model in question may contain concentrated loads or other singularities.

For these and other reasons, the finite element solution may not match theoretical solutions or experimental data, and the time and resources devoted to the problem are essentially wasted.

It is the objective of this research to develop an accurate and efficient method to perform nonlinear three dimensional structural finite element analyses. This was achieved through the development and implementation of a modification to the Finite Quasi-Prism element. This three dimensional element was originally developed by Vijayakar in 1987 for use in linear elastic gear analysis and was shown to be highly effective for models where the geometry is nearly constant in one direction. The element varies from other elements in the literature in that it uses arbitrary order polynomial basis functions within the element to approximate the displacement field. This dissertation details the modifications required to use this element in large-strain, large-displacement, anisotropic, elastic-plastic analysis.

1.2 Scope of the Dissertation

The scope of the work comprises the following tasks:

1. To develop a finite element that can be used to solve nonlinear problems efficiently and accurately.
2. To implement this element into a large strain, large displacement materially nonlinear finite element code.
3. To perform accuracy and run time analyses of elastic-plastic problems and to compare these results with existing theoretical solutions and with a commercially available code.
4. To understand the behavior of this element in nonlinear analyses and to determine the effect of higher interpolation orders on accuracy and solution times.
5. To determine the types of analyses where this element is most effective.

1.3 Dissertation Organization

This dissertation is divided into six chapters. Chapter 1 outlines briefly the research objective and defines the scope of the work. Chapter 2 contains a review of the literature in nonlinear finite element analysis, solving algorithms, and various analysis approaches. Chapter 3 details the large strain, large deflection elastic-plastic finite element formulation used for this work while Chapter 4 describes in detail the element used in this analysis, the special considerations required in its formulation and the tests performed to validate the element. Chapter 5 discusses the performance of this element as compared to the commercial code ANSYS and theoretical results for several elastic-plastic examples. The effects of element order and size are also examined in Chapter 5. Finally, Chapter 6 gives the conclusions of this work, the author's research contributions and recommendations for future work.

CHAPTER 2

NONLINEAR FINITE ELEMENT ANALYSIS BACKGROUND INFORMATION

2.1 Nonlinear Finite Element Analysis

The earliest paper on nonlinear finite elements appears to be that by Turner, Dill, Martin, and Melosh in 1960. As with most early finite element work, this paper was the product of work being done in the aircraft industry. Most of the other early nonlinear finite element work related primarily to the linear buckling problem. See, for example, the work by Gallagher and Padlog, or Gallagher, Gellatly, Padlog and Mallet.

As was stated in Chapter 1, the goal of this work was to solve a subset of nonlinear finite element models both accurately and efficiently. For the purposes of this work, nonlinear analysis refers to large-strain, large-displacement, elastic-plastic analysis.

2.2 Nonlinear Material Modeling

Elastic-plastic analyses must include a yield criterion, a constitutive equation including material hardening, and, for greater generality, anisotropy. A brief review of the above stated physical phenomenon is given in the following sections.

2.2.1 Yield Criteria

For any elastic-plastic analysis, it is critical to know when the material has yielded. At this yield point the material's behavior can no longer be considered elastic but now obeys the flow laws of plasticity. Many yield criteria have been developed. Below is a brief outline of three common isotropic yield criteria used for metals.

In 1864 Tresca defined a yield criterion also known as the maximum shear stress theory. In terms of the ordered principal stresses ($\sigma_1 > \sigma_2 > \sigma_3$)

$$f = (\sigma_1 - \sigma_3) - \sigma_0 = 0 \quad (2.1)$$

where

f = the yield function

and

σ_0 = the yield stress.

In 1913 von Mises introduced a yield criterion also referred to as the distortion energy criterion, or the J_2 criterion, which is given in terms of the principal stresses by

$$f = (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 - 2\sigma_0^2 = 0 \quad (2.2)$$

Both the Tresca and the von Mises yield criteria are for isotropic plasticity. They assume that yielding is independent of the hydrostatic pressure; hence, there is no volume change during plastic deformation. These criteria also assume that the material is isotropic and that yielding is independent of the sign of the applied stresses. The Tresca and the von Mises criteria are a good first approximation for the yield surface of metals and have been used often.

To improve the approximation of the yield surface, Hershey (1954) and Hosford (1972) introduced a generalization of the von Mises yield criterion given by

$$f = |\sigma_1 - \sigma_2|^M + |\sigma_2 - \sigma_3|^M + |\sigma_1 - \sigma_3|^M - 2\sigma_0^M = 0 \quad (2.3)$$

For $M=6$ and $M=8$, this function closely fits the yield surfaces of materials with Body-Centered Cubic (BCC) and Face-Centered Cubic crystal structures, respectively. Note that the criterion reduces to the von Mises criterion for $M=2$ and to the Tresca criterion for $M=1$.

2.2.2 Constitutive Equations

In order for a formulation of an element to be applicable to a specific response prediction, it is necessary to use the appropriate constitutive description. The constitutive equations relate the stress to a number of variables such as strain, strain rate, and temperature. For linear elastic analysis, the stress is typically related to the strain using Hooke's Law. However, once the material yields, an equation relating the plastic flow stress to the strain, strain rate, deformation history, etc. is needed.

Due to the tensor character of stress, strain and strain rate, the constitutive equations in their most general form are not manageable within the framework of solving a highly nonlinear boundary value problem. The number of variables must be reduced to a practical level while assuring that the material behavior is modeled properly. This reduction in the number of variables is usually based on the flow theory of plasticity where the yield surface in stress space defines the allowable plastic stress states. (Stress states within the yield surface are elastic in nature and Hooke's Law can be used.) The yield functions discussed in the previous section can be simplified to functions of the

effective stress, $\bar{\sigma}$ (a scalar) and the instantaneous yield stress, σ_0 . Similarly, the strain tensor can be replaced by the effective strain, $\bar{\epsilon}$. Some constitutive equations for isotropic strain hardening materials in terms of the effective stress and the effective strain are given below.

Strain hardening laws are commonly derived by performing uniaxial tension tests or bulge tests and fitting a curve to the measured effective stress – effective strain curves.

In 1909 Ludwik proposed the constitutive equation

$$\bar{\sigma} = \sigma_0 + K\bar{\epsilon}^n \quad (2.4)$$

where

K = the strain hardening coefficient

and

n = the strain hardening exponent.

In 1945 Hollomon simplified this equation to

$$\bar{\sigma} = K\bar{\epsilon}^n \quad (2.5)$$

An initial effective strain term, $\bar{\epsilon}_0$, was added to Hollomon's equation first by

Krupkowski (1945) and then by Swift (1952) to give

$$\bar{\sigma} = K(\bar{\epsilon} + \bar{\epsilon}_0)^n \quad (2.6)$$

In 1948 Voce proposed a hardening law using the exponential function

$$\bar{\sigma} = A - Be^{-C\bar{\epsilon}} \quad (2.7)$$

Ratke and Welch reviewed these four constitutive equations in 1983, comparing the corresponding rates of strain hardening and the instantaneous hardening curves. More stress-strain relations can be found in a review by Osgood (1946).

2.2.3 Anisotropy

Materials such as sheet metal have different material properties in different directions. These differing properties are due, for the case of sheet metals, to the process of rolling the material to sheet thickness. It is generally assumed for this case that the resulting material anisotropy is orthotropic in nature, that is, there exist three mutually orthogonal planes of symmetry at each material point. The intersection of these planes is the principal axes of anisotropy. In 1948, Hill proposed his first generalization of the von Mises yield criterion for orthotropic materials given by

$$f = F(\sigma_y - \sigma_z)^2 + G(\sigma_z - \sigma_x)^2 + H(\sigma_x - \sigma_y)^2 + 2L\tau_{yz}^2 + 2M\tau_{xz}^2 + 2N\tau_{xy}^2 - \sigma_0^2 = 0 \quad (2.8)$$

where the coefficients in terms of the directional normal yield stresses (σ_{11} , σ_{22} , σ_{33})

and the directional shear stresses (σ_{12} , σ_{23} , σ_{31}) are given by

$$\begin{aligned} 2F &= \frac{1}{\sigma_{22}^2} + \frac{1}{\sigma_{33}^2} - \frac{1}{\sigma_{11}^2} \\ 2G &= \frac{1}{\sigma_{33}^2} + \frac{1}{\sigma_{11}^2} - \frac{1}{\sigma_{22}^2} \\ 2H &= \frac{1}{\sigma_{11}^2} + \frac{1}{\sigma_{22}^2} - \frac{1}{\sigma_{33}^2} \\ 2L &= \left(\frac{1}{\sigma_{23}} \right)^2 \\ 2M &= \left(\frac{1}{\sigma_{31}} \right)^2 \\ 2N &= \left(\frac{1}{\sigma_{12}} \right)^2 \end{aligned} \quad (2.9)$$

the x - y - z axes relate to the principal axes of anisotropy. Similar to the Tresca and the von Mises criteria, this criterion assumes no Bauschinger effect in the material and that hydrostatic pressure does not affect yielding. Hill's yield criterion was used extensively for the modeling of sheet metal forming and is often referred to in the literature as Hill's old yield criterion. It was found to give good results for steel sheet metal applications. (Hill, 1979)

However, Woodthorpe and Pearce (1970) found problems with Hill's 1948 theory in testing done on aluminum with an average normal anisotropy factor (strain in the width direction divided by the strain in the thickness direction) of less than one observed a biaxial-uniaxial yield ratio greater than one. To accommodate this phenomenon termed "anomalous behavior" by Woodthorpe and Pearce, Hill (1979) introduced a modification of his 1948 yield criterion. The modified criterion in terms of the principal stresses is given by

$$f = F|\sigma_2 - \sigma_3|^M + G|\sigma_3 - \sigma_1|^M + H|\sigma_1 - \sigma_2|^M + A|2\sigma_1 - \sigma_2 - \sigma_3|^M + B|2\sigma_2 - \sigma_3 - \sigma_1|^M + C|2\sigma_3 - \sigma_1 - \sigma_2|^M - \sigma_0^M = 0 \quad (2.10)$$

2.3 Solution Algorithms

As will be seen in greater detail in Chapter 3, obtaining a finite element solution requires the solution of a large system of equations of the form

$$[K]\{x\} = \{F\} \quad (2.11)$$

where, for the types of problems discussed in this dissertation, the stiffness matrix $[K]$ is sparse, banded, symmetric and positive definite. The choice of the method or algorithm

used to solve this system of equations has a significant impact on the CPU time required to obtain a finite element solution.

There are primarily two ways to obtain a solution to Equation (2.11) above. The first is to solve directly using Gaussian elimination; the second is by using iterative methods.

2.3.1 Direct Solution Methods

The more commonly used method is direct solution by Gaussian elimination. Several different Gaussian elimination algorithms exist to reduce the $[K]$ matrix; many of them were developed to take advantage of one or more of the matrix properties just mentioned.

2.3.1.1 Symmetric Banded Solutions

Symmetric banded solution algorithms improve the speed of traditional Gaussian elimination algorithms by reducing the size of the stiffness matrix, and thereby increasing the amount of memory available to carry out the actual row reduction and solution. A banded matrix is a matrix in which all of the nonzero elements are contained within a diagonal band of the matrix. Outside of this band, all matrix entries are zero. Further, because the stiffness matrix is symmetric, only the upper triangular portion of this matrix needs be stored. Symmetric banded solutions store only this non-zero upper triangular band. Row reduction and back substitution are done on this reduced matrix.

Note that the size of the bandwidth is directly linked to the nodal numbering scheme used in the finite element model. Thus, to get the maximum efficiency from a banded solver, it is important to order the nodes such that the node numbers for a given element are close numerically. That is, a one-dimensional element bounded by nodes two and

five will have fewer zeroes in its bandwidth (hence a smaller bandwidth and smaller stored stiffness matrix) than that same element bounded instead by nodes two and ten.

For three-dimensional problems, the size of the stiffness matrix increases rapidly even for banded methods of handling due to the increased number of degrees of freedom per node. In general, the more nodes per element and the more degrees of freedom per node that an element requires, the larger the bandwidth will be. For this reason, symmetric banded solvers are seldom used in three-dimensional problems.

2.3.1.2 Skyline Solutions

Skyline solutions differ from other Gaussian elimination solution algorithms in two fundamental ways. First, with skyline solvers, Gaussian elimination is carried out via column reduction. Column reduction works because the reduction of any column $j > 1$ can be done using only the already reduced columns one through $j-1$ and the already reduced elements in column j .

The second fundamental difference in the skyline solution is the way in which the stiffness matrix is stored in memory. Skyline algorithms use sequentially stored columns of the symmetric upper triangular stiffness matrix. The storage of the column begins with the first nonzero entry, and the location of the stiffness matrix diagonals is stored in an accompanying array. Simply put, the stiffness matrix is now stored in two one-dimensional arrays instead of the reduced two-dimensional array of the banded solutions. These one-dimensional arrays require less storage space in memory, thereby increasing the efficiency of the algorithm. Note that, as with the banded solvers, careful node numbering can reduce the size of the stored stiffness matrix by reducing the number of

zeroes between nonzero column entries. Also like the banded solvers, increasing the problem to three dimensions rapidly increases the size of the stiffness matrices, thereby reducing the algorithm's gains in efficiency.

2.3.1.3 Frontal Solutions

The frontal solution method is an alternative direct solution method that performs the assembly and solution steps simultaneously as the solver processes an element. In a finite element problem, the stiffness matrix is determined first for an individual element. This elemental matrix is then added to the global stiffness matrix $[K]$ of Equation (2.11). The frontal method, developed by Irons (1970), is based on the fact that a degree of freedom can be eliminated from the global system of equations as soon as all stiffness values in the rows and the columns for that degree of freedom are complete. This being the case then, all of the degrees of freedom for a node can be eliminated from the system of equations when that node appears for the last time as the global stiffness matrix is assembled in the ascending order of elements. Once a degree of freedom is eliminated, the corresponding equation is not needed until the back substitution phase of the solution. This equation is typically written to an external disk for storage until needed.

As the global stiffness matrix is assembled, the active matrix size, or wavefront (the number of "incomplete" degrees of freedom contained in the matrix), grows, and when some degrees of freedom are eliminated, the wavefront shrinks. Once all of the elements have been assembled and all degrees of freedom have been eliminated, the nodal solution is calculated by back substitution.

Because the full global stiffness matrix is never calculated, the frontal solution method requires low memory usage. Further, because of the compact nature of the front (only active degrees of freedom are included) and because variables are eliminated as soon as possible, the operations on coefficients equal to zero are minimized, and the total number of arithmetic operations is lower than with the other two methods. However, because individual equations, once eliminated, need to be stored for back substitution, the disk use is relatively high. Also, extensive bookkeeping is required to keep track of the eliminated degrees of freedom.

The performance of this algorithm and the memory it requires are directly affected by the size of the wavefront. Thus, in direct contrast to the banded and skyline solvers, the ordering of the elements is crucial to this algorithm's performance; the node numbering is irrelevant. This solution method is robust and works best for smaller problems and block-shaped structures.

2.3.1.4 Sparse Matrix Solutions

As the size and complexity of finite element models to be analyzed increase, banded and frontal solution methods have been found to be slow, and iterative methods often are also slow or fail to converge. (Storaasli, 1996) To remedy this, sparse matrix solvers were developed. Most sparse solvers contain three major components: matrix reordering, matrix factoring, and back substitution.

Similar to the banded and the skyline solution methods, the full stiffness matrix is reordered and solved in a significantly more compact form. However, unlike the banded and the skyline solution methods, coefficients equal to zero, which are sandwiched

between nonzero stiffness matrix entries, are not included in the reduced matrices.

Sparse matrix solvers, by using several matrices containing “pointers”, store only the nonzero stiffness matrix entries. The benefits of this are twofold. First, storage requirements are significantly reduced, and second, mathematical operations performed on zero coefficients are eliminated.

Like the frontal solution method, the sparse matrix solver works only on active equations, but unlike the frontal solver, it performs operations on several blocks at a time using a multi-front technique. Further, these blocks are accessed directly during the matrix factoring portion of the method, which reduces the costly bookkeeping associated with the indirect addressing used in the frontal solution method. This solution method works best for large problems and systems with a sparse structure, such as a spoked wheel or an engine block.

2.3.2 Iterative Solution Methods

The objective of iterative solution algorithms is to iteratively calculate the solution to Equation (2.11). This method includes algorithms such as the Gauss-Seidel method, which was used in early finite element applications, (Clough, 1960 and Wilson, 1965) and the conjugate gradient method developed by Hestenes and Stiefel (1952). The prime benefit of the iterative method is that significant storage space can be saved due to the way in which the $[K]$ matrix is stored. In fact, the ANSYS Basic Analysis Procedure Guide states that a preconditioned conjugate gradient solver usually requires less than one quarter of the disk file space used by a frontal solver, and it usually runs faster than the

frontal solver for models with wavefronts greater than about 1000. Hence, iterative methods are good for very large finite element systems.

However, the time required for a solution, which is largely a function of the number of iterations required to achieve convergence, can be estimated only approximately. The key to iterative schemes is a method to accelerate convergence when slow convergence is observed. This slow convergence was the reason that iterative methods were largely abandoned in the 1960's and 1970's. Further, indirect iterative solution methods depend on the solution convergence criteria supplied by the user. If a tolerance is too high, the solution may take unnecessarily long to converge. But tolerances that are too low may significantly affect the accuracy of the solution; especially the quantities derived from the solution values, that is, the stress and the strain. Work has been done to accelerate convergence, and effective acceleration procedures have become available for many applications, thus making the iterative solution method more attractive.

2.3.3 Solution Method Utilized

The program uses the sparse matrix solution method. This method was chosen after tests comparing this method with the frontal solution method resulted in considerable CPU timesavings. For the five-element thick cylinder model of Section 5.2.1, Figure 5.5, the sparse matrix solver obtained a solution in 15% to 20% of the time required by the frontal solution method for models of varying element order.

These timesavings are expected to be even more dramatic for larger models with more degrees of freedom. In addition, for process and product design, multiple Modified FQP analyses are needed, and so, the timesavings are multiplicative. When using the frontal

solution method with many elements, the size of the stiffness matrix and the wavefront grow considerably, and significant memory is required to store the active equations. Even if the mesh is efficiently built, the reading and writing of the equations to the disk and the extensive bookkeeping required with the frontal method slow the solution significantly. For extremely large models, the time advantage of the sparse matrix solver over the frontal solver is on the order of 100. (Storaasli, 1996)

2.4 Analyses Using Higher Order Elements

To obtain efficiently accurate finite element solutions to two-and-a-half dimensional problems, higher order elements were investigated. It has been shown for linear elastic bending analyses that higher order brick elements give more accurate solutions and are more computationally efficient than linear brick and triangular elements. (Clough, 1969) This work was later extended to elastic-plastic problems where “even greater advantages arise.” (Nayak and Zienkiewicz, 1972, Owen, Nayak, Kfoury and Griffiths, 1973, and Zienkiewicz, 1989) It has also been shown (Babuska and Szabo, 1982) that the rate of convergence of the p -version of the finite element method cannot be slower than the h -version provided that a “quasi-uniform” mesh is used in the h -version. (Note that the p -version refers to a method of solution refinement where the order of the polynomial shape functions is increased while the given mesh remains constant. The h -version refers to a method of solution refinement where the mesh is refined while the order of the shape functions remains constant.) Szabo (1979) and Szabo and Katz (1981) have summarized early work in the use of the p -version finite element method.

However, due to the geometric nature of two-and-a-half dimensional problems (complex geometry in two dimensions only), it is often computationally more efficient and easier to use many lower order elements in the plane of the complicated geometry. Two different analysis methods, the finite prism method, and finite element analysis using the Finite Quasi-Prism (FQP) element use arbitrary order bases in a three-dimensional element. The foundation of each alternative and its applicability to nonlinear problems is detailed below.

2.4.1 The Finite Prism Method

The finite prism method is a three-dimensional extension of the finite strip method and is used primarily for plate and structure analyses. (Cheung and Tham, 1998, Cheung and Li, 1992, Wong and Vardy, 1985 and Azizian and Dawe, 1984) For this method, the continuum is divided into three-dimensional prisms in which one pair of opposite sides forms the boundary of the continuum. The cross-section of the prism must be constant along the length of the element, that is, analysis is limited to prismatic geometries only.

The prisms are connected to each other along a discrete number of nodal lines coinciding with the element boundaries. Interpolation functions are then chosen for the element. Typically, a higher order of interpolation is used for the longitudinal direction than is used for the cross-section. Many different functions can be used for the longitudinal direction including polynomials, B-splines, Fourier series and decaying exponential functions. These interpolation functions must satisfy the boundary conditions. Because this method uses only one element for the length of the plate instead of many as required by the finite element method, the resulting element matrices when

summed over the domain of the continuum are usually small and have a low bandwidth. Often orthogonal shape functions are chosen for even sparser matrices. Thus, the benefit of this method is faster solution times due to the small matrices and low bandwidth, and this is achieved primarily by using fewer higher order elements.

The analysis method works well for cases where there are no sharp bends, that is, for cases where the cross-sectional area rotates more than it deforms. Unfortunately, an assumption made in the formulation of the finite prism element that results in a diagonal stiffness matrix prevents the element's use in more general analyses. The method assumes the cross-section of the element remains constant or very nearly constant. Clearly, this is not the case for many nonlinear analyses. Additionally, opposite ends of the element must form the boundaries of the part to be analyzed, severely limiting the types of problems this method can solve. Therefore, the finite prism method, while typically faster than the finite element method in plate analyses, is too restrictive to be used successfully for more complicated two-and-a-half dimensional analysis.

2.4.2 The Finite Quasi-Prism Element

The FQP element was developed by Vijayakar (1987) for three-dimensional analyses in which the cross-section of the part remains nearly constant in one direction. Thus, unlike the finite prism element, the cross-section does not have to remain constant in the longitudinal direction. The FQP element is formulated such that one element can replace several standard finite elements while still yielding results comparable to those achieved using traditional finite element analysis. This is done by using a higher order of interpolation for the shape function in the longitudinal or constant cross-section direction.

The element maintains a lower order of interpolation in the remaining two directions. Additionally, the FQP elements may be stacked end to end, that is, there is no requirement that opposite ends of the element form the boundary of the part.

The FQP element is not limited by aspect ratios as are standard finite elements, and, so, several smaller standard finite elements with proper aspect ratios can be replaced by one FQP element with a higher order of interpolation. Because of this, equivalent solution accuracy can be obtained with fewer elements and fewer degrees of freedom. This reduces the size of the stiffness matrix while still maintaining the banded characteristic of the stiffness matrix found in the traditional finite element method. (Vijayakar, Busby and Houser, 1987) Nonlinear analyses require multiple passes through a solver, and the time for each of these passes is proportional to the size of the stiffness matrix which is proportional to the square of the number of degrees of freedom contained in the finite element model of the structure. For example, nine one-dimensional linear traditional finite elements (ten nodes, ten degrees of freedom) require $(10 \times 1)^2 = 100$ stiffness matrix entries. If these nine elements can be replaced by one fourth order FQP element (five nodes, five degrees of freedom) with $(1 \times 5)^2 = 25$ stiffness matrix entries, the size of the stiffness matrix can be reduced by a factor of four. This stiffness matrix reduction can have significant effects in incremental nonlinear analyses.

Many of the new elastic-plastic finite element analysis developments involve using reduced integration methodology to eliminate the accompanying shear locking problems and speed solution times. (See, for example the recent work by Liu, Guo, Tang and Belytschko, 1998.) The reasoning behind reduced integration is that the same number of elements can be used, but there is now a smaller computational cost per element. The

modified FQP element is in direct contrast to this trend. The modified FQP element uses the standard integration, but the formulation of the element is such that fewer elements are needed to obtain comparable results.

The element as formulated in the literature is linear elastic. To implement the FQP element in a nonlinear analysis, a nonlinear version of this element is required. The element used in this thesis is a variation of the FQP element developed by Vijayakar. It's formulation and implementation will be discussed in the remaining chapters.

CHAPTER 3

NONLINEAR FINITE ELEMENT FORMULATION

3.1 Introduction

An updated Lagrangian virtual work formulation was used for the main portion of the program with the Newton-Raphson iteration technique used to obtain a converged solution at each increment. Additionally, the backward-Euler return was incorporated into the program to reconcile the plastic stresses with the yield surface. In this chapter, outlines of the virtual work and the finite element formulations, the modified Newton-Raphson technique and the backward-Euler return will be given. These theories and formulations are well documented in the literature, and more details on these subjects can be found in, among others, Crisfield, 1991, Bathe, 1996 and Kleiber, 1989.

3.2 Principle of Virtual Work

The basis of displacement-based finite element analysis is the principle of virtual work. The principle of virtual work applied to a continuum results in an integral form of the equations of equilibrium and the static boundary conditions. The theory states that for any set of compatible small virtual displacements imposed on the body in its state of equilibrium, the total internal virtual work is equal to the total external virtual work.

In order to obtain a solution, it is generally necessary to employ an incremental formulation for a nonlinear analysis. This being the case, the principle of virtual work must be applied to the body in its current configuration at each time step. The updated Lagrangian formulation updates the reference configuration after each successful increment, and so the incremental stresses at time $t+\Delta t$ are determined relative to the reference configuration of the stresses and strains of the last converged solution of time t . Using the principle of virtual work for an updated Lagrangian formulation, the problem at time $t+\Delta t$ relative to the reference configuration at time t is given by

$$\int_{t+\Delta t V} {}^{t+\Delta t}S_{ij} \delta({}^{t+\Delta t}\epsilon_{ij}) d{}^{t+\Delta t}V = {}^{t+\Delta t}R \quad (3.1)$$

The right hand side of Equation (3.1) is the external virtual work at time $t+\Delta t$. At a given time the external virtual work is generally assumed to be independent of displacement and is given by

$${}^{t+\Delta t}R = \int_{t+\Delta t V} {}^{t+\Delta t}f_i^B \delta u_i d{}^{t+\Delta t}V + \int_{t+\Delta t S_f} {}^{t+\Delta t}f_i^S \delta u_i^S d{}^{t+\Delta t}S \quad (3.2)$$

where

${}^{t+\Delta t}f_i^B$ = the body force per unit volume at time $t+\Delta t$

${}^{t+\Delta t}f_i^S$ = the applied external tractions at time $t+\Delta t$

δu_i = virtual displacements representing a continuous virtual displacement field and equal to zero at the prescribed displacement boundaries.

${}^{t+\Delta t}S_f$ = the surface at time $t+\Delta t$ on which external tractions are applied

and

$\delta u_i^S = \delta u_i$ evaluated on the surface ${}^{t+\Delta t}S_f$

The left-hand side of Equation (3.1) is the internal virtual work at time $t+\Delta t$ with

${}^{t+\Delta t}{}_tS_{ij}$ = the second Piola-Kirchoff stress at time $t+\Delta t$ measured in the reference configuration of the body at time t

$\delta({}^{t+\Delta t}{}_t\varepsilon_{ij})$ = the virtual Green-Lagrange strain at time $t+\Delta t$ measured in the configuration of the body at time t

and

${}^{t+\Delta t}V$ = the volume of the body at time $t+\Delta t$.

The second Piola-Kirchoff stress at time $t+\Delta t$ in Equation (3.1) can be decomposed into increments giving

$${}^{t+\Delta t}{}_tS_{ij} = {}^t\tau_{ij} + {}^t\Delta S_{ij} \quad (3.3)$$

where

${}^t\tau_{ij}$ = the Cauchy, or true, stress at time t

and

${}^t\Delta S_{ij}$ = the increment of the second Piola-Kirchoff stress.

Likewise, the incremental strain can be decomposed yielding

$${}^{t+\Delta t}{}_t\varepsilon_{ij} = {}^t\Delta\varepsilon_{ij} = {}^t\Delta e_{ij} + {}^t\Delta\eta_{ij} \quad (3.4)$$

where

$${}^t\Delta e_{ij} = \frac{1}{2}({}^t\Delta u_{i,j} + {}^t\Delta u_{j,i}) \quad (3.5)$$

is the linear incremental strain, and

$${}^t\Delta\eta_{ij} = \frac{1}{2}({}^t\Delta u_{k,i} {}^t\Delta u_{k,j}) \quad (3.6)$$

is the nonlinear incremental strain. Note that in Equations (3.5) and (3.6) ${}^t\Delta u$ is the incremental displacement at time t and

$$u_{i,j} = \frac{\partial u_i}{\partial x_j} \quad (3.7)$$

Inserting these decompositions into the equilibrium equation [Equation (3.1)] yields

$$\int_{V'} {}^t\Delta S_{ij} \delta({}^t\Delta \varepsilon_{ij}) dV + \int_{V'} {}^t\tau_{ij} \delta({}^t\Delta \eta_{ij}) dV = {}^{t+\Delta t}R - \int_{V'} {}^t\tau_{ij} \delta({}^t\Delta e_{ij}) dV \quad (3.8)$$

For small time steps the incremental Piola-Kirchoff stress increment can be approximated by

$${}^t\Delta S_{ij} = {}^tC_{ijrs} {}^t\Delta e_{rs} \quad (3.9)$$

where

${}^tC_{ijrs}$ = the material specific incremental stress-strain tensor at time t referred to the configuration at time t .

In addition, because virtual displacements, and, thus, virtual strains are, by definition, very small, the virtual strain can be approximated as

$$\delta({}^t\Delta \varepsilon_{ij}) = \delta({}^t\Delta e_{ij}) \quad (3.10)$$

Applying the linearizing approximations of Equations (3.9) and (3.10) then to the incremental form of the equilibrium equation [Equation (3.8)] yields

$$\int_{V'} {}^tC_{ijrs} ({}^t\Delta e_{rs}) \delta({}^t\Delta e_{ij}) dV + \int_{V'} {}^t\tau_{ij} \delta({}^t\Delta \eta_{ij}) dV = {}^{t+\Delta t}R - \int_{V'} {}^t\tau_{ij} \delta({}^t\Delta e_{ij}) dV \quad (3.11)$$

Equation (3.11) is the linearized, incremental form of the equilibrium equation of the updated Lagrangian formulation. It is an incremental linearized continuum mechanics equation that includes the nonlinear effects due to large displacements, large strains and material nonlinearities.

3.3 General Matrix Equations of Displacement-Based Continuum Elements

As with any displacement-based finite element formulation, the first step in generating the governing matrix equations is to discretize the body of interest into elements. Within these elements, the incremental displacement field is approximated using shape functions and incremental nodal displacements such that

$${}^t\Delta u = \sum_{i=1}^{NN} N_i {}^t\Delta u_i \quad (3.12)$$

where

${}^t\Delta u$ = the incremental displacements

N_i = the interpolation or shape functions for the displacement

${}^t\Delta u_i$ = the incremental nodal displacements

and

NN = the number of nodes in the element.

The element geometry is interpolated in a similar manner.

This summation is taken over all the nodes in the element. Note that this interpolation is for a given increment only. The total displacement of a given node can be found by adding all the incremental displacements over time, that is

$${}^{t+\Delta t}u_i = \sum_{j=1}^n {}^j\Delta u_i \quad (3.13)$$

where

j = the index specifying the incremental number

and

n = the total number of increments.

The displacement interpolation is then substituted into Equations (3.5), (3.6) and (3.11) to obtain the incremental finite element matrix equation

$$({}^t\mathbf{K}_L + {}^t\mathbf{K}_{NL}) {}^t\Delta\mathbf{U} = {}^{t+\Delta t}\mathbf{R} - {}^t\mathbf{F}_{int} \quad (3.14)$$

where

${}^t\mathbf{K}_L$ = the linear strain incremental stiffness matrix

${}^t\mathbf{K}_{NL}$ = the nonlinear (geometric) strain incremental stiffness matrix

${}^t\Delta\mathbf{U}$ = the vector of increments in the nodal point displacements

${}^{t+\Delta t}\mathbf{R}$ = the vector of externally applied nodal loads at time $t + \Delta t$

and

${}^t\mathbf{F}_{int}$ = the vector of nodal forces equivalent to the element stresses at time t (also known as the internal force vector at time t).

Comparing Equations (3.11) and (3.14) one finds that ${}^t\mathbf{K}_L {}^t\Delta\mathbf{U}$ is the matrix

formulation of the integral $\int_V {}^tC_{ijrs} ({}^t\Delta e_{rs}) \delta({}^t\Delta e_{ij}) dV$ with the matrix ${}^t\mathbf{K}_L$ given by

$${}^t\mathbf{K}_L = \int_V {}^t\mathbf{B}_L^T {}^t\mathbf{C} {}^t\mathbf{B}_L dV \quad (3.15)$$

where

${}^t\mathbf{B}_L$ = the linear strain-displacement transformation matrix given by

$${}^t\mathbf{B}_L = \begin{bmatrix} N_{1,1} & 0 & 0 & N_{2,1} & \dots & 0 \\ 0 & N_{1,2} & 0 & 0 & \dots & 0 \\ 0 & 0 & N_{1,3} & 0 & \dots & N_{NN,3} \\ N_{1,2} & N_{1,1} & 0 & N_{2,2} & \dots & 0 \\ 0 & N_{1,3} & N_{1,2} & 0 & \dots & N_{NN,2} \\ N_{1,3} & 0 & N_{1,1} & N_{2,3} & \dots & N_{NN,1} \end{bmatrix} \quad (3.16)$$

and

${}^t\mathbf{C}$ = the incremental stress-strain material property matrix.

The matrix ${}^t\mathbf{C}$ varies depending on the material properties and the loading history of the location considered. More details on this matrix will be given in Section 3.6.

Similarly, ${}^t\mathbf{K}_{NL} {}^t\Delta\mathbf{U}$ is the matrix formulation of $\int_V {}^t\tau_{ij} \delta({}^t\Delta\eta_{ij}) dV$ with the matrix ${}^t\mathbf{K}_{NL}$ given by

$${}^t\mathbf{K}_{NL} = \int_V {}^t\mathbf{B}_{NL}^T {}^t\boldsymbol{\tau} {}^t\mathbf{B}_{NL} dV \quad (3.17)$$

where

${}^t\mathbf{B}_{NL}$ = the nonlinear strain-displacement transformation matrix given by

$${}^t\mathbf{B}_{NL} = \begin{bmatrix} {}^t\tilde{\mathbf{B}}_{NL} & \tilde{\mathbf{0}} & \tilde{\mathbf{0}} \\ \tilde{\mathbf{0}} & {}^t\tilde{\mathbf{B}}_{NL} & \tilde{\mathbf{0}} \\ \tilde{\mathbf{0}} & \tilde{\mathbf{0}} & {}^t\tilde{\mathbf{B}}_{NL} \end{bmatrix} \quad (3.18)$$

with

$${}^t\tilde{\mathbf{B}}_{NL} = \begin{bmatrix} N_{1,1} & 0 & 0 & N_{2,1} & \dots & N_{NN,1} \\ N_{1,2} & 0 & 0 & N_{2,2} & \dots & N_{NN,2} \\ N_{1,3} & 0 & 0 & N_{2,3} & \dots & N_{NN,3} \end{bmatrix} \quad (3.19)$$

and

$$\tilde{\mathbf{0}} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \quad (3.20)$$

and

${}^t\boldsymbol{\tau}$ = the matrix of Cauchy stresses given by

$${}^t\boldsymbol{\tau} = \begin{bmatrix} {}^t\tilde{\boldsymbol{\tau}} & \bar{\mathbf{0}} & \bar{\mathbf{0}} \\ \bar{\mathbf{0}} & {}^t\tilde{\boldsymbol{\tau}} & \bar{\mathbf{0}} \\ \bar{\mathbf{0}} & \bar{\mathbf{0}} & {}^t\tilde{\boldsymbol{\tau}} \end{bmatrix} \quad (3.21)$$

with

$${}^t\tilde{\boldsymbol{\tau}} = \begin{bmatrix} {}^t\tau_{11} & {}^t\tau_{12} & {}^t\tau_{13} \\ {}^t\tau_{21} & {}^t\tau_{22} & {}^t\tau_{23} \\ {}^t\tau_{31} & {}^t\tau_{32} & {}^t\tau_{33} \end{bmatrix} \quad (3.22)$$

and

$$\bar{\mathbf{0}} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \quad (3.23)$$

Likewise, the internal force vector, ${}^t\mathbf{F}_{int}$ is the matrix formulation of $\int_{V'} {}^t\tau_{ij} \delta({}^t\Delta e_{ij}) dV$

with the vector ${}^t\mathbf{F}_{int}$ given by

$${}^t\mathbf{F}_{int} = \int_{V'} {}^t\mathbf{B}_L^T {}^t\boldsymbol{\tau}' dV \quad (3.24)$$

where

${}^t\boldsymbol{\tau}'$ = the vector of Cauchy stresses given by

$${}^t\boldsymbol{\tau}' = \begin{bmatrix} {}^t\tau_{11} \\ {}^t\tau_{22} \\ {}^t\tau_{33} \\ {}^t\tau_{12} \\ {}^t\tau_{23} \\ {}^t\tau_{31} \end{bmatrix} \quad (3.25)$$

Gauss quadrature is used to evaluate the integrals in Equations (3.15), (3.17) and (3.24).

3.4 Newton-Raphson Iteration Technique

The unknown to be solved for in Equation (3.14) is the incremental displacement vector, ${}^t\Delta\mathbf{U}$. However, because the stiffness matrix and the internal force vector are functions of the displacement field, it is necessary to perform solution iterations to obtain

a converged solution. The iteration method used in this work is the Newton-Raphson technique. Its formulation is detailed below.

Equation (3.14) can be rewritten as

$$f(\mathbf{U}^*) = \mathbf{0} \quad (3.26)$$

where

$\mathbf{f}(\mathbf{U}^*)$ = the out-of-balance force vector given by

$$\mathbf{f}(\mathbf{U}^*) = {}^{t+\Delta t}\mathbf{R} - {}^{t+\Delta t}\mathbf{F}_{int} \quad (3.27)$$

Taking a Taylor's series expansion of Equation (3.26), one obtains

$$\mathbf{f}(\mathbf{U}^*) = \mathbf{f}({}^{t+\Delta t}\mathbf{U}^{(i-1)}) + \left[\frac{\partial \mathbf{f}}{\partial \mathbf{U}} \right]_{{}^{t+\Delta t}\mathbf{U}^{(i-1)}} ({}^{t+\Delta t}\mathbf{U}^* - {}^{t+\Delta t}\mathbf{U}^{(i-1)}) + \text{higher order terms} \quad (3.28)$$

Note that the superscript $(i-1)$ on the right-hand side of the term ${}^{t+\Delta t}\mathbf{U}^{(i-1)}$ refers to the iteration number within the $t+\Delta t$ increment. This notation is continued for the remainder of the section.

Substituting Equation (3.27) into Equation (3.28) and using Equation (3.26) yields

$$\left[\frac{\partial \mathbf{f}}{\partial \mathbf{U}} \right]_{{}^{t+\Delta t}\mathbf{U}^{(i-1)}} ({}^{t+\Delta t}\mathbf{U}^* - {}^{t+\Delta t}\mathbf{U}^{(i-1)}) + \text{higher order terms} = {}^{t+\Delta t}\mathbf{R} - {}^{t+\Delta t}\mathbf{F}_{int}^{(i-1)} \quad (3.29)$$

Neglecting the higher order terms and recognizing that the current tangent stiffness matrix is given by

$${}^{t+\Delta t}\mathbf{K}^{(i-1)} = \left[\frac{\partial \mathbf{f}}{\partial \mathbf{U}} \right]_{{}^{t+\Delta t}\mathbf{U}^{(i-1)}} \quad (3.30)$$

the current displacement increment can be calculated using

$${}^{t+\Delta t}\mathbf{K}^{(i-1)} \Delta \mathbf{U}^{(i)} = {}^{t+\Delta t}\mathbf{R} - {}^{t+\Delta t}\mathbf{F}_{int}^{(i-1)} \quad (3.31)$$

This displacement solution is added to the solution from the previous iteration to obtain the improved displacement solution

$${}^{t+\Delta t}\mathbf{U}^{(i)} = {}^{t+\Delta t}\mathbf{U}^{(i-1)} + \Delta\mathbf{U}^{(i)} \quad (3.32)$$

Note that the tangent stiffness matrix, ${}^{t+\Delta t}\mathbf{K}^{(i-1)}$, in Equation (3.31) is the sum of the linear and nonlinear matrices given by Equations (3.15) and (3.17) and is a function of the nodal displacements. It was found through the course of this work that the full Newton-Raphson technique, which re-evaluates the stiffness matrix for each iteration, converged in significantly fewer iterations than the Modified Newton-Raphson technique. The difference in the required number of iterations was so large (ranging from a factor of three to a factor of 8) that despite the frequent time-consuming re-evaluations of the stiffness matrix, it was more efficient to use the full Newton-Raphson technique for this work.

Once the improved displacement vector has been determined for a given iteration, the stress at the integration points is calculated. Knowing the stress for iteration i , the internal force for this iteration is obtained using Equation (3.24). This value is then substituted into Equation (3.27). The Newton-Raphson process is repeated until the out-of-balance force vector and the incremental nodal displacement vector meet specified convergence criteria. For this work, two convergence criteria were used. The first is the force criterion given by

$$\frac{\|\mathbf{f}(\mathbf{U}^*)\|}{\|{}^{t+\Delta t}\mathbf{R} - {}^t\mathbf{R}\|} \leq \varepsilon_f \quad (3.33)$$

where

$\|\mathbf{f}(\mathbf{U}^*)\|$ = the norm of the out-of-balance force vector

$\|{}^{t+\Delta t}\mathbf{R}-{}^t\mathbf{R}\|$ = the norm of the external force vector for the current increment

and

ε_f = a user-specified out-of-balance force tolerance.

The second criterion used is the displacement criterion given by

$$\frac{\|\Delta\mathbf{U}^{(i)}\|}{\|{}^{t+\Delta t}\mathbf{U}^{(i)}\|} \leq \varepsilon_d \quad (3.34)$$

where

$\|\Delta\mathbf{U}^{(i)}\|$ = the norm of the iterative incremental displacement vector

$\|{}^{t+\Delta t}\mathbf{U}^{(i)}\|$ = the norm of the displacement solution

and

ε_d = a user-specified displacement tolerance.

Suggested ranges for values of ε_f and ε_d have been given by Nayak and Zienkiewicz (1972), among others.

3.5 Determination of Strain

Once the incremental nodal displacement vector is determined, the true strain is calculated. If the reference configuration of a node is given by the vector $\mathbf{X}$, then the new configuration, $\mathbf{x}$, of this same node is given by

$$\mathbf{x} = \mathbf{X} + {}^{t+\Delta t}\Delta\mathbf{u} \quad (3.35)$$

The deformation gradient for an updated Lagrangian formulation is given by

$${}^{t+\Delta t}\Delta\mathbf{F} = \frac{\partial \mathbf{x}}{\partial \mathbf{X}} = \frac{\partial (\mathbf{X} + {}^{t+\Delta t}\Delta\mathbf{u})}{\partial \mathbf{X}} = \mathbf{I} + \frac{\partial {}^{t+\Delta t}\Delta\mathbf{u}}{\partial \mathbf{X}} = \mathbf{I} + \frac{\partial \mathbf{N}}{\partial \mathbf{X}} {}^{t+\Delta t}\Delta\mathbf{u}_i \quad (3.36)$$

where

$\mathbf{I}$ = the (3x3) identity matrix.

Note that the deformation gradient given in Equation (3.36) above is the deformation gradient for the current increment only. The total deformation gradient at time $t+\Delta t$ is given by

$${}^{t+\Delta t}\mathbf{F} = {}^{t+\Delta t}\Delta\mathbf{F} {}^t\mathbf{F} \quad (3.37)$$

Using the Polar Decomposition theorem, the incremental deformation gradient matrix can be separated into an incremental rotation matrix and an incremental shape change matrix, or

$${}^{t+\Delta t}\Delta\mathbf{F} = {}^{t+\Delta t}\Delta\mathbf{Rot} {}^{t+\Delta t}\Delta\mathbf{U}_R \quad (3.38)$$

where

${}^{t+\Delta t}\Delta\mathbf{Rot}$ = the incremental rotation matrix of the integration point (a unitary matrix)

and

${}^{t+\Delta t}\Delta\mathbf{U}_R$ = the incremental right stretch, or shape change, matrix.

Once the polar decomposition of the incremental deformation gradient is performed and the right stretch matrix is obtained, the incremental logarithmic or true, strain is given by

$${}^{t+\Delta t}\Delta\boldsymbol{\varepsilon} = \ln({}^{t+\Delta t}\Delta\mathbf{U}_R) \quad (3.39)$$

Since ${}^{t+\Delta t}\Delta\mathbf{U}_R$ is a second-order tensor, the logarithmic strain is determined through the spectral decomposition of ${}^{t+\Delta t}\Delta\mathbf{U}_R$, or

$${}^{t+\Delta t}\Delta\boldsymbol{\varepsilon} = \sum_{i=1}^3 \ln({}^{t+\Delta t}\Delta\lambda_i) \mathbf{eig}_i \mathbf{eig}_i^T \quad (3.40)$$

where

${}^{t+\Delta t}\Delta\lambda_i$ = the incremental principal stretches at time $t+\Delta t$ (i.e. the eigenvalues of ${}^{t+\Delta t}\Delta\mathbf{U}_R$)

and

$\mathbf{eig}_i$ = the principal stretch directions (i.e. the eigenvector corresponding to the i^{th} eigenvalue of ${}^{t+\Delta t}\Delta\mathbf{U}_R$)

The incremental logarithmic strains can be summed to give the total logarithmic strain for time $t+\Delta t$, that is,

$${}^{t+\Delta t}\boldsymbol{\varepsilon}_{ij} = \sum_{k=1}^n {}^k\Delta\boldsymbol{\varepsilon}_{ij} \quad (3.41)$$

where

n = the total number of iterations.

The total rotation matrix at time $t+\Delta t$ for a point is found by taking the product of the incremental rotation matrices at that point

$${}^{t+\Delta t}\mathbf{Rot} = \prod_{k=1}^n {}^k\Delta\mathbf{Rot} \quad (3.42)$$

This total rotation matrix is necessary for the conversion of the Second Piola-Kirchoff stress values to the Cauchy stress values used in the calculation of the internal force vector and the total stress at an integration point.

Note that the procedure outlined in this section must be carried out at each integration point for each element.

3.6 Determination of Stress

Like strain, stress is also determined at the integration points of an element. If the material has not yet yielded at a particular integration point, then the incremental stress is determined using Hooke's Law, or

$$\Delta\sigma = {}^t\mathbf{C}\Delta\epsilon \quad (3.43)$$

where the stress-strain material matrix is given by

$${}^t\mathbf{C} = {}^t\mathbf{C}^E = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \quad (3.44)$$

where

${}^t\mathbf{C}^E$ = the linear elastic stress-strain material property matrix

E = Young's modulus

and

ν = Poisson's ratio.

If yielding has occurred, i.e. if

$$f(\sigma, \bar{\epsilon}_p) = 0 \quad (3.45)$$

where

f = the yield function

σ = the stress vector

and

$\bar{\epsilon}_p$ = the equivalent plastic strain

then the elastic-plastic response is assumed to be governed by the classical incremental plastic flow theory based on the Prandtl-Reuss equations. Using the additive decomposition of strains, that is

$${}^{t+\Delta t}\Delta\epsilon_{ij} = {}^{t+\Delta t}\Delta\epsilon_{ij}^E + {}^{t+\Delta t}\Delta\epsilon_{ij}^P \quad (3.46)$$

the stress increment for time $t+\Delta t$ is given by

$${}^{t+\Delta t}\Delta\sigma_{ij} = {}^tC_{ijrs}^E \left({}^{t+\Delta t}\Delta\epsilon_{rs} - {}^t\Delta\epsilon_{rs}^P \right) \quad (3.47)$$

where

${}^tC_{ijrs}^E$ = the components of the elastic stress-strain material property matrix

${}^{t+\Delta t}\Delta\epsilon_{rs}$ = the components of the total strain increment as established in Section 3.5

${}^{t+\Delta t}\Delta\epsilon_{rs}^E$ = the components of the elastic strain increment

and

${}^{t+\Delta t}\Delta\epsilon_{rs}^P$ = the components of the plastic strain increment.

This relationship holds throughout the inelastic material response. Note that for cases in which the material has yielded, Equation (3.47) gives an estimated stress increment because the plastic strain is a function of the as yet undetermined current stress state. Because the calculation of the plastic strain depends on the current stress state and

because Equation (3.47) is only an estimate, ${}^t\Delta\epsilon_{\pi}^P$ in the equation is the plastic strain increment from the previous iteration. This estimated stress will be corrected using the backward-Euler method described in the next section.

To calculate the plastic strains, a yield function, a hardening rule and a flow rule relating plastic strain increments to the current stresses and stress increments are required. During plastic deformation, the normality rule gives the incremental plastic strain as

$$d\epsilon_{ij}^P = d\lambda \frac{\partial f}{\partial \sigma_{ij}} \quad (3.48)$$

or, in vector form as

$$d\epsilon^P = d\lambda {}^t\mathbf{a} \quad (3.49)$$

where

$$d\lambda = \text{a constant}$$

and

${}^t\mathbf{a}$ = a vector defined for the stress state at time t as

$${}^t\mathbf{a} = \begin{bmatrix} \frac{\partial f}{\partial \sigma_{11}} \\ \frac{\partial f}{\partial \sigma_{22}} \\ \frac{\partial f}{\partial \sigma_{33}} \\ \frac{\partial f}{\partial \sigma_{12}} \\ \frac{\partial f}{\partial \sigma_{23}} \\ \frac{\partial f}{\partial \sigma_{31}} \end{bmatrix} \quad (3.50)$$

During plastic deformation, the yield function is equal to zero, thus

$$df = \left[\frac{\partial f}{\partial \boldsymbol{\sigma}} \right]^T d\boldsymbol{\sigma} + \frac{\partial f}{\partial \bar{\epsilon}_p} d\bar{\epsilon}_p = {}^t\mathbf{a}^T d\boldsymbol{\sigma} - d\lambda A = 0 \quad (3.51)$$

where

$$A = \frac{-1}{d\lambda} \frac{\partial f}{\partial \bar{\epsilon}_p} d\bar{\epsilon}_p \quad (3.52)$$

defines subsequent yield surfaces of an isotropic strain-hardening material.

Substituting into Equation (3.46), then, gives

$$d\boldsymbol{\epsilon} = ({}^t\mathbf{C}^E)^{-1} d\boldsymbol{\sigma} + d\lambda {}^t\mathbf{a} \quad (3.53)$$

Premultiplying both sides of the above equation by ${}^t\mathbf{a}^T {}^t\mathbf{C}^E$ and using Equation (3.51) to eliminate $d\boldsymbol{\sigma}$ yields

$${}^t\mathbf{a}^T {}^t\mathbf{C}^E d\boldsymbol{\epsilon} = A d\lambda + d\lambda {}^t\mathbf{a}^T {}^t\mathbf{C}^E {}^t\mathbf{a} \quad (3.54)$$

Solving for $d\lambda$ gives

$$d\lambda = \frac{{}^t\mathbf{a}^T {}^t\mathbf{C}^E d\boldsymbol{\epsilon}}{A + {}^t\mathbf{a}^T {}^t\mathbf{C}^E {}^t\mathbf{a}} \quad (3.55)$$

and, thus, after using Equation (3.53) the incremental stress is given by

$$d\boldsymbol{\sigma} = ({}^t\mathbf{C}^E - {}^t\mathbf{C}^P) d\boldsymbol{\epsilon} = {}^t\mathbf{C}^{EP} d\boldsymbol{\epsilon} \quad (3.56)$$

where

${}^t\mathbf{C}^P$ = the plastic stress-strain material property matrix given by

$${}^t\mathbf{C}^P = \frac{{}^t\mathbf{C}^E {}^t\mathbf{a} {}^t\mathbf{a}^T {}^t\mathbf{C}^E}{A + {}^t\mathbf{a}^T {}^t\mathbf{C}^E {}^t\mathbf{a}} \quad (3.57)$$

Once the incremental stress has been determined, the total stress for time $t+\Delta t$ can be found using

$${}^{t+\Delta t}\boldsymbol{\sigma} = {}^t\boldsymbol{\sigma} + d\boldsymbol{\sigma} \quad (3.58)$$

The stress at time $t+\Delta t$ is next substituted into the yield function. If substitution of these stresses results in a yield function value of $f > 0$, then the yield surface has been crossed and the material has yielded. Once the material has yielded, it is necessary to adjust the stress values so that $f = 0$ while including the expanding yield surface due to isotropic hardening. In other words, the equation above is used to predict the stress at a point at time $t+\Delta t$. If the equivalent plastic stress obtained from this prediction exceeds the yield stress, then the predicted stress must be modified so that the stress values correspond to the yield surface. This was done using the backward-Euler return.

Finally, note that this program uses a simple isotropic hardening rule. While this rule does not accurately capture the Bauschinger effect associated with the bending and unbending of some metallic materials, this rule has been shown to be adequate for reversed loading in steel. (Gau, 1999) Further, a more realistic hardening rule can be inserted into the program at a later date.

3.7 The Backward-Euler Return

The backward-Euler return is based on the equation

$$\boldsymbol{\sigma}_C = \boldsymbol{\sigma}_B - \Delta\lambda' \mathbf{C}' \mathbf{a}_C \quad (3.59)$$

where

$\boldsymbol{\sigma}_C$ = the starting estimate for the stress, i.e. the first stress correction

and

σ_B = the stress as determined in Equation (3.58) of Section 3.6.

These points are illustrated in Figure 3.1 on the next page.

Typically the first guess for σ_C fails to satisfy the yield function, and, so, an iterative loop is established. The first step in the loop is to set up the vector representing the differences between the current stresses and the backward-Euler stresses, that is,

$$\mathbf{r} = \sigma_C - (\sigma_B - \Delta\lambda' \mathbf{C}' \mathbf{a}_C) \quad (3.60)$$

The iterations are continued until the norm of $\mathbf{r}$, $\|\mathbf{r}\|$, is below some user specified tolerance and the final stresses satisfy within a given tolerance the yield criterion $f = 0$ at time $t + \Delta t$.

With the trial stresses σ_B kept fixed throughout the iterations, a truncated Taylor's series expansion is applied to Equation (3.60) to produce a new residual $\mathbf{r}_n$ given by

$$\mathbf{r}_n = \mathbf{r} + \dot{\sigma} + \dot{\lambda}' \mathbf{C}' \mathbf{a}_C + \Delta\lambda \frac{\partial' \mathbf{a}_C}{\partial \sigma} \dot{\sigma} \quad (3.61)$$

where

$\dot{\sigma}$ = the change in the stress vector, σ

and

$\dot{\lambda}$ = the change in $\Delta\lambda$.

Setting $\mathbf{r}_n$ equal to zero gives

$$\dot{\sigma} = - \left(\mathbf{I} + \Delta\lambda' \mathbf{C} \frac{\partial' \mathbf{a}_C}{\partial \sigma} \right)^{-1} (\mathbf{r} + \dot{\lambda}' \mathbf{C}' \mathbf{a}_C) = -' \mathbf{Q}^{-1} (\mathbf{r} + \dot{\lambda}' \mathbf{C}' \mathbf{a}_C) \quad (3.62)$$

where

$\mathbf{I}$ = the (6x6) identity matrix.

Similarly, a truncated Taylor series on the yield function gives

$$f_n = f + \left[\frac{\partial f}{\partial \boldsymbol{\sigma}} \right]^T \dot{\boldsymbol{\sigma}} + \frac{\partial f}{\partial \bar{\epsilon}_p} \dot{\bar{\epsilon}}_p = f + {}^t\mathbf{a}_c^T \dot{\boldsymbol{\sigma}} + A_c \dot{\lambda} \quad (3.63)$$

where

A_c = the hardening term at point C.

Using this expansion to solve for $\dot{\lambda}$ gives

$$\dot{\lambda} = \frac{f - {}^t\mathbf{a}_c^T \mathbf{Q}^{-1} \mathbf{r}}{{}^t\mathbf{a}_c^T \mathbf{Q}^{-1} \mathbf{C}^t \mathbf{a}_c + A_c} \quad (3.64)$$

Figure 3.1: The yield surface for the backward-Euler return method.

The values of $\dot{\sigma}$ and $\dot{\lambda}$ are added to σ_c and $\Delta\lambda$, respectively. The residual, r , and the yield function, f , are then checked. This algorithm [Equations (3.59), (3.60), (3.62) and (3.64)] is repeated until the norm of the residual and the value of the yield function are both below some user-specified tolerance.

CHAPTER 4

THE MODIFIED FINITE QUASI-PRISM ELEMENT

4.1 Introduction

The accuracy and speed of a nonlinear finite element solution are functions of, among other things, the model used. Finite element models can vary from one another in the choice of element type, the density of the finite element mesh, the inputted material properties and the applied boundary conditions. It is the purpose of this research to obtain an accurate and efficient solution to a subset of nonlinear three-dimensional problems in which the geometric variation along one axis is significantly less than the remaining two axes. It is assumed throughout that both the material properties and the boundary conditions are entered correctly, thus, this research strives to obtain CPU time reduction and high solution accuracy through mesh optimization.

Many elements exist in the literature, but Vijayakar's (1987) Finite Quasi-Prism (FQP) element was formulated precisely for near prismatic elements. This element, as formulated, is subparametric with Chebyshev polynomial shape functions. Examples and uses of this element were in linear elastic analyses. The benefits of using this element were discussed in Chapter 2. This chapter will outline the development of an

isoparametric form of this element with Lagrangian shape functions to be used in three-dimensional, nonlinear analyses of near prismatic bodies.

4.2 Element Formulation

Vijayakar developed the Finite Quasi-Prism (FQP) element in 1987 for three-dimensional analyses in which the cross-section of the part remains constant or nearly constant in one direction for the length of the element. This element is formulated such that one FQP element can replace several standard finite elements while still yielding results comparable to those achieved using a traditional finite element analysis. This is done by using a higher order of interpolation for the shape function in the direction of the nearly constant cross-section referred to as the *axode*. The element maintains a lower order of interpolation in the remaining two directions. This element is illustrated in Figure 4.1 below.

Figure 4.1: The Finite Quasi-Prism element. ζ is the direction of nearly constant cross-section requiring the higher order interpolation.

The formulation of the modified FQP element is very similar to conventional finite elements. The main difference is in the shape functions that are used. For the FQP element, the geometric coordinates are approximated by

$$\begin{aligned}x(\xi, \eta, \zeta) &= \sum_{i=1}^{nc} \sum_{j=0}^{ord} x_{ij} N_i(\xi, \eta) T_j(\zeta) \\y(\xi, \eta, \zeta) &= \sum_{i=1}^{nc} \sum_{j=0}^{ord} y_{ij} N_i(\xi, \eta) T_j(\zeta) \\z(\xi, \eta, \zeta) &= \sum_{i=1}^{nc} \sum_{j=0}^{ord} z_{ij} N_i(\xi, \eta) T_j(\zeta)\end{aligned}\tag{4.1}$$

where

x_{ij}, y_{ij}, z_{ij} = the nodal point coordinates

nc = the number of axode curves per element (e.g. there are four axodes for an element linear in x and y),

ord = the order of interpolation in the ζ direction

$N_i(\xi, \eta)$ = the standard two-dimensional Lagrangian shape functions

and

$T_j(\zeta)$ = the higher order Lagrangian shape functions given by

$$T_j^{ord} = \frac{(\zeta - \zeta_0)(\zeta - \zeta_1) \cdots (\zeta - \zeta_{j-1})(\zeta - \zeta_{j+1}) \cdots (\zeta - \zeta_{ord})}{(\zeta_j - \zeta_0)(\zeta_j - \zeta_1) \cdots (\zeta_j - \zeta_{j-1})(\zeta_j - \zeta_{j+1}) \cdots (\zeta_j - \zeta_{ord})}\tag{4.2}$$

Lagrangian elements are not to be confused with the more common serendipity family of elements, both of which are illustrated in Figure 4.2 for a two-dimensional quadratic element. The primary difference between the two elements is that the nodes of serendipity elements are located only on the element edges; Lagrangian elements have what are known as face and bubble nodes. Face nodes are located in the middle of an element face (node 9 in Figure 4.2). Bubble nodes are found within the interior volume

of three-dimensional elements. These extra nodes require extra shape functions for the element. These extra shape functions give Lagrangian elements a more complete polynomial basis for the displacement field than serendipity elements. Because their bases are more complete, Lagrangian elements are less affected than serendipity elements by the severe angular distortions of an element that often accompany large displacement analyses. (Backlund, 1978, Gifford, 1979 and Lee and Bathe, 1993) In addition, this complete polynomial interpolation is required to maintain the monotonic convergence behavior of the element. For these reasons, Lagrangian shape functions were chosen for the Modified FQP element.

a) Serendipity element

b) Lagrangian element

Figure 4.2: Two-dimensional quadratic elements: a) Serendipity element, b) Lagrangian element.

In its original formulation, the element was subparametric, meaning that the order of interpolation for the geometry was less than the order of interpolation for the displacement. While the use of subparametric elements can somewhat decrease the computational effort, these elements also limit the generality of the finite element discretization. In addition, subparametric elements considerably complicate the solution procedures for geometrically nonlinear analysis where the new geometry of an element is obtained by adding the displacement solution to the previous geometry.

Therefore, to simplify the displacement and nodal location updating procedures at the end of each increment, an isoparametric version of the FQP element was developed. For isoparametric elements, the displacement interpolation uses the same shape functions as the coordinate interpolation. Thus, the displacement interpolation is given by

$$\begin{aligned} u(\xi, \eta, \zeta) &= \sum_{i=1}^{nc} \sum_{j=0}^{ord} u_{ij} N_i(\xi, \eta) T_j(\zeta) \\ v(\xi, \eta, \zeta) &= \sum_{i=1}^{nc} \sum_{j=0}^{ord} v_{ij} N_i(\xi, \eta) T_j(\zeta) \\ w(\xi, \eta, \zeta) &= \sum_{i=1}^{nc} \sum_{j=0}^{ord} w_{ij} N_i(\xi, \eta) T_j(\zeta) \end{aligned} \quad (4.3)$$

Equations (4.1) and (4.3) can now be substituted into the equations of Chapter 3, Section 3 to obtain the complete finite element formulation.

In addition, because the element uses the same shape functions for displacement as for geometry, the Modified FQP element is capable of modeling more complex geometries along the axode than standard linear or quadratic brick elements. This will be demonstrated in Section 5.3.1 where one element was used to model a 90° arc.

As mentioned in Chapter 2, the Modified FQP element does not use reduced integration. Indeed, the element uses more than the standard number of integration points

in practice. These extra integration points are used to more accurately capture the elastic-plastic boundary within an element. The nearer the integration points are to the elastic-plastic boundary, the more accurately the stress in the element is predicted, which results in a more accurate internal force vector, and, thus, faster convergence. So while more integration points require more calculations in the formation of the stiffness matrix, the element strain vector, the element stress vector and the internal force vector, the added accuracy provided by these extra points reduces the number of iterations required to achieve a converged solution. More computing time is saved by reducing the number of iterations required for convergence than is gained by using more than the standard number of integration points.

Finally, the order of the interpolation in the longitudinal direction is based on the expected longitudinal strain distribution. This order must be optimized to obtain the best solution in the least amount of time.

4.3 Element Testing

Once the element was formulated and inserted into the nonlinear finite element analysis code outlined in the previous chapter, several basic tests were performed to determine the element's convergence properties, its behavior in shear, and the amount of deformation along the axode that the element can sustain and still give accurate solutions.

4.3.1 Element Convergence

For monotonic convergence, an element must be complete and the mesh must be compatible. Completeness of an element means that the displacement functions of an element must be able to represent both the rigid body displacements and the constant strain states.

4.3.1.1 Rigid Body Displacement

Rigid body displacements are those displacement modes that an element must be able to undergo as a rigid body without stresses being developed in it. This requirement was tested using the cantilever beam model shown in Figure 4.3. Two separate cases were run; first the beam was loaded to yielding, next the load was increased to induce plastic deformation. The beam is 1"x1"x20" with a modulus of elasticity of 30 Mpsi, a Poisson ratio of 0.3, and a yield stress of 30 kpsi. A linear strain-hardening rule with a tangential modulus of 60 kpsi and the von Mises yield criterion were used for the analysis.

In the first test, the load $F=360\text{ lb}$ was applied to the six element, sixth-order mesh at $0.611L$. A simple shear and bending moment diagram (Figure 4.4) shows that there is no stress in the portion of the beam to the right of the load. Hence, elements five and six will undergo rigid body displacement. Results for this elastic loading and the plastic loading ($F=600\text{ lb}$) are given in Figures 4.5 and 4.6.

Figure 4.3: Model for the rigid body displacement test.

Figure 4.4: Shear force and bending moment diagrams for the plastically loaded cantilever beam of the rigid body displacement test.

As can be seen in Figure 4.5, the stress values for the portion of the beam to the right of the load are near zero for both the elastic and the plastic cases. Further, the elastic displacement plot of Figure 4.6 shows a good agreement with theoretical values, i.e., the Modified FQP element accurately models the rigid body movement of elements five and six. Thus, the element can represent rigid body motion.

Figure 4.5: Normal stress results near the top surface of the beam for the rigid body displacement test.

Figure 4.6: Vertical displacement results for the rigid body displacement test.

4.3.1.2 Constant Strain State

The ability of the element to model a constant strain state was verified using a simple patch test. In three separate tests, a 1"x1"x20" block of two elements (shown in Figure 4.7) was loaded uniaxially, one test per direction (x , y , z). Minimal boundary conditions were applied. This test was run for elements of orders ranging from three to eight. For each direction, a constant strain state existed throughout the block. And so, the modified FQP element meets the requirement of completeness.

Figure 4.7: Two-element model used for the patch tests.

4.3.1.3 Mesh Compatibility

The requirement of compatibility means that the displacements within the elements and across the element boundaries must be continuous, i.e., there are no gaps between the elements when loads are applied to the mesh. Since only translational degrees of freedom (u , v , and w) are defined at the element nodes, only continuity in the displacements need be preserved. This requirement is satisfied through the element shape functions and the element connectivity. For a properly formed mesh, nodes on the common face of two adjacent elements must be shared by both elements. This assures that the displacements are continuous across element boundaries. As a result, the compatibility requirement is also satisfied. The element converges monotonically.

4.3.2 Element Performance in Shear

Shear locking is a term used to refer to spurious shear stresses that are predicted in displacement-based elements like the Modified FQP element. These spurious shear stresses result in a strong artificial stiffening of the element as the thickness to length ratio decreases. Note that the thickness to length ratio for an FQP element is low by design. Thus, to avoid shear locking problems common in bending problems using standard linear hexahedral elements, the elements used for the remaining analyses will be quadratic in the transverse and the thickness direction, i.e., $nc=9$ in Equations (4.1) and (4.3). To verify that the element does not experience shear locking, a simple linear elastic cantilever beam analysis was performed where the beam was loaded to yielding. Several models with progressively fewer elements were run. The final model consisted of a single third order element of dimension 1"x1"x20" as shown in Figure 4.8. Young's

modulus for the beam was 30 Mpsi, Poisson's ratio was equal to 0.3, and the yield stress was 30 kpsi. The normal stress and displacement results for the final model are given in Figure 4.9 and Table 4.1, respectively. Note that the displacement solution included the cross-sectional distortion due to shearing stresses as given in the literature. (Timoshenko, 1970) This distortion is not typically included in similar traditional finite element solutions.

As can be seen in Figure 4.9 and Table 4.1, one modified FQP element quadratic in x and y is sufficient to accurately model shear and bending in a cantilever beam. That is, the quadratic element does not lock in shear, and it accurately captures the shear deformation of the beam found in the more complex Timoshenko beam theory equations.

Figure 4.8: The third-order, one-element cantilever beam model used for the shear-locking test.

Figure 4.9: Normal stress results of the shear locking test (one third-order element).

	$z=0.00$	$z=6.67$	$z=13.33$	$z=20.00$
Theoretical (elementary beam theory)	0.00000	-0.00988	-0.07901	-0.26667
Theoretical (Timoshenko beam theory)	0.00000	-0.03967	-0.13860	-0.26717
Program	0.00000	-0.03960	-0.13844	-0.26693
% difference from Timoshenko	--	0.18%	0.12%	0.09%

Table 4.1: Displacement results of the shear-locking test (one third-order element).

4.3.3 Allowable Element Distortion

In addition to the above tests, a test was conducted to determine the approximate effect of varying cross-sectional geometry along the axode on solution accuracy. Three simple tests, each illustrated in Figure 4.10, were run. The tests compared the results of the model with prismatic elements to that of progressively more geometrically nonuniform elements. The difference between the three tests was in the direction and magnitude of the applied force. The first case was the pure bending of a beam. The second and third cases were the bending of a cantilever beam. In the second case, the beam was loaded to yielding; in the third, the beam underwent significant plastic deformation. The beam used for this example was the same as the one previously described in the rigid body displacement test with the different meshes shown in Figure 4.11. Fourth order elements were used. The test results are given in Table 4.2 through Table 4.4.

Beam	max. deflection (in.)	% difference
Base	0.979125	Baseline
Beam 1	0.976498	-0.2683%
Beam 2	0.975716	-0.3481%
Beam 3	0.971554	-0.7732%
Beam 4	0.965672	-1.3739%
Beam 5	0.973413	-0.5833%

Table 4.2: The effect of nonprismatic element geometries for Case 1 (pure bending).

a)

b)

c)

Figure 4.10: Three-element model used to test the effect of nonprismatic element geometry. a) Case 1: pure bending of the beam, b) Case 2: cantilever beam loaded to yielding, and c) Case 3: plastically deformed cantilever beam.

Figure 4.11: Deformed element cantilever beam models.

Beam	max. deflection (in.)	% difference
Base	-0.267237	Baseline
Beam 1	-0.267237	-0.0001%
Beam 2	-0.267237	-0.0002%
Beam 3	-0.267226	-0.0043%
Beam 4	-0.267229	-0.0030%
Beam 5	-0.267224	-0.0048%

Table 4.3: The effect of nonprismatic element geometries for Case 2 (cantilever beam loaded to yielding).

Beam	max. deflection (in.)	% difference
Base	-1.015676	Baseline
Beam 1	-1.016642	0.0951%
Beam 2	-1.016870	0.1175%
Beam 3	-1.014300	-0.1355%
Beam 4	-1.007802	-0.7753%
Beam 5	-1.005430	-1.0088%
Beam 6	-1.018991	0.3264%
Beam 7	-1.019431	0.3697%
Beam 8	-1.017464	1.1969%
Beam 9	-0.982124	-3.3034%
Beam 10	-0.946229	-6.8375%

Table 4.4: The effect of nonprismatic element geometries for Case 3 (cantilever beam, plastic deformation).

As can be seen in the tables, the effect of geometrically nonuniform cross-sectional areas generally becomes more significant the more the element is distorted. However, the differences from the baseline results for Cases 1 and 2 are not very significant. The same, however, cannot be said of Case 3. For the models with extreme element distortion, depending on how the element is distorted, the change in the results is as great as 6.8% for the model with narrow cross-sectional areas in the region of highest stress (Beam 9 and Beam 10). Care must be taken, therefore, to guard against extreme element deformation in high stress regions.

4.4 Summary

In conclusion, the Modified FQP element differs from the linear elastic FQP element in the following aspects:

1. The Modified FQP element is isoparametric for ease of geometric and displacement updating.
2. The Modified FQP element uses a quadratic interpolation through the thickness to prevent shear locking of the element in bending. This allows for a reduction in the number of elements needed through the thickness for bending applications.
3. The Modified FQP element uses Lagrangian shape functions for better results as the element distorts and so that the polynomial basis for the shape functions is more complete.
4. The Modified FQP element uses more integration points through the thickness to more effectively capture the yield surface within the element.

The Modified FQP element has been shown to be both complete, namely that it is able to represent both rigid body displacement and constant strain states, and continuous. Hence, the element is monotonically convergent. The element was also shown to perform well in shear and for reasonably nonuniform cross-sectional areas.

CHAPTER 5

ELEMENT PERFORMANCE IN NONLINEAR ANALYSIS

5.1 Introduction

The Modified Finite Quasi-Prism element was placed in the large-displacement, large-strain, elastic-plastic finite element formulation outlined in Chapter 3. Several example cases were run. The following sections of this chapter discuss the accuracy of the program as compared to theoretical solutions, where available, and to the commercial code ANSYS 5.5. Additionally, the effects of different models and orders of interpolation on both the solution accuracy and the solution time were investigated. Finally, the time required to obtain the solution using the Modified FQP program is compared to the time required to obtain a solution of comparable accuracy in ANSYS. To avoid unfair skewing of the CPU time comparison data, the least refined mesh giving a converged solution in ANSYS was used for these comparisons.

The Modified FQP analyses used the NASA GPS4 (General Purpose Solver) sparse matrix solver. ANSYS 5.5 uses the Boeing sparse matrix solver, which is an optimized version of the GPS4 solver. All analyses contained in this chapter were performed on an SGI workstation with four 270 MHz IP27 processors and 1024MB of memory.

5.2 Pure Bending of an Elastic-Plastic Beam

A pure bending of an elastic-plastic steel beam example was run first to test the accuracy of the Modified FQP element in an elastic-plastic analysis. A linear isotropic strain-hardening rule and the von Mises yield criterion were used for the analysis. The beam is illustrated in Figure 5.1. Its material properties and its theoretical stress solution are given in Figure 5.2.

For the example, the location of the yield surface (the xz -plane located at a distance y_Y from the neutral axis in Figure 5.2) was varied. For different locations of y_Y the maximum stress was determined using the Modified FQP element and then compared to the theoretical stress value. In addition to modifying the yield plane, the order of interpolation along the length of the beam was varied.

Figure 5.1: Schematic of the beam used for the pure elastic-plastic bending example.

Figure 5.2: Material properties and the theoretical stress distribution of the beam.

5.2.1 Effect of Variation of the Order of Interpolation

The first study conducted on this beam was to determine the effect of increasing the order of interpolation of the element along the length of the beam. For this study, the simple two-element model pictured in Figure 5.3 was used. Second order interpolation in the x and the y directions was used for all runs; the order of interpolation in the z direction was varied from three to five. This was done for two cases. For the first case the yield plane was located 0.4 inches from the neutral axis ($y_Y \approx 0.4$ in Figure 5.2); for the second case the yield plane was located 0.2 inches from the neutral axis. The results for the two cases are given in Tables 5.1 and 5.2.

Figure 5.3: The two-element model used for the variation of the order of interpolation study.

	$\zeta = 3$	$\zeta = 4$	$\zeta = 5$
$\sigma_{\max}$	30009.7	30009.7	30009.6
% error	0.0170%	0.0170%	0.0167%
max. displacement	0.501947	0.502334	0.502580
% error	0.3894%	0.4668%	0.5160%
CPU time	40.92 sec.	70.10 sec.	116.68 sec.

Table 5.1: The solution accuracy comparison for the $y_T = 0.40$ in case as the order of interpolation is varied from $\zeta=3$ to $\zeta=5$. For this case the maximum theoretical stress at the integration point is 30,004.6 psi and the maximum theoretical deflection is 0.50 in.

	$\zeta = 3$	$\zeta = 4$	$\zeta = 5$
$\sigma_{\max}$	30077.6	30077.4	30077.0
% error	0.0280%	0.0274%	0.0260%
max. displacement	0.986998	0.990148	0.992273
% error	-1.3002%	-0.9852%	-0.7727%
CPU time	145.70 sec.	234.71 sec.	362.65 sec.

Table 5.2: The solution accuracy comparison for the $y_T = 0.20$ in case as the order of interpolation is varied from $\zeta=3$ to $\zeta=5$. For this case the maximum theoretical stress at the integration point is 30,069.2 psi and the maximum theoretical deflection is 1.00 in.

As can be seen from the two tables, there is very little change in the displacement or the stress solution as the order of interpolation along the length of the beam is increased. Indeed, the higher order solutions show that the first solution was, in fact, the converged solution. The amount of time, however, required to obtain a solution increased appreciably as the order increased. There appears to be an optimal value for the order of interpolation, beyond which the solution accuracy changes little, but the time required to obtain a solution changes appreciably. Solutions of orders higher than this optimal value serve only to verify the convergence of the previous solution.

5.2.2 Comparison to the Theoretical Solution

Next, the displacement and the maximum integration point stress value using the Modified FQP element was found for five separate locations of y_T . Based on the results of the previous section, cubic interpolation along the length was used. The accuracy of the five separate cases was verified and can be found in Table 5.3.

y_Y (in)	$\sigma_{\text{THEORETICAL}}$ (psi)	σ_{FQP} (psi)	% difference
0.40	30004.6	30009.7	0.0170%
0.30	30026.1	30029.5	0.0113%
0.25	30043.3	30055.8	0.0416%
0.20	30069.2	30077.6	0.0279%
0.10	30219.7	30223.1	0.0113%

Table 5.3: Comparison of solution accuracy for the five beam pure bending cases.

As can be seen in Table 5.3, the Modified FQP element gives highly accurate results with very few elements. Further, the Modified FQP element had a smooth stress distribution that was not affected by point loads or displacement boundary conditions. This was true for all five cases run.

5.3 Elastic-Plastic Thick Cylinder with Internal Pressure

Another relatively simple example case was run to test the accuracy and the speed of the Modified FQP element in an elastic-plastic analysis, and to determine the effect of different Modified FQP element orders and orientations on the solution accuracy and the solution time. The test case was a thick cylinder with an applied internal pressure. Due to symmetry, the whole cylinder need not be modeled. Because this model will be used later in a nonaxisymmetric example, a quarter of the cylinder was modeled. The internal pressure was applied to the inner diameter of the cylinder such that the elastic-plastic boundary would be located at half the cylinder thickness ($r = 0.225$). The results obtained by the Modified FQP program were then compared to the analytical solution given by

Phillips (1956) and to ANSYS results. A linear strain-hardening rule and the von Mises yield criterion were used for the analysis. The isotropic material properties used are: Young's Modulus, $E = 30$ Mpsi; Tangential Modulus, $E_T = 60$ kpsi; Poisson's Ratio, $\nu = 0.3$; and Yield Stress, $\sigma_Y = 30$ kpsi. The inner diameter of the cylinder is equal to 0.30 in; the outer diameter is equal to 0.60 in.

5.3.1 Accuracy and Solution Time Comparisons

Two different FQP element models were used. The first model, illustrated in Figure 5.4, consisted of four elements with the element axes aligned along the radius of the cylinder. The results of a model consisting of fifth-order elements were compared to the theoretical solution and to the ANSYS solution. The least refined converged ANSYS solution giving comparable results consisted of 60 linear brick elements along the circumference by 20 elements along the radius. This ANSYS solution gave an elastic-plastic boundary located within 2% of the theoretical solution. The results of both analyses are given in Figures 5.5 through 5.7. As can be seen in the three figures, the $\zeta=5$ model effectively captures the stress behavior of the cylinder.

Figure 5.4: The Modified FQP model of the thick cylinder with radial axodes.

Figure 5.5: Variation of the equivalent stress along the radius of the cylinder for the model given in Figure 5.4. Note that the elastic-plastic boundary occurs at $r = 0.225$ in.

Figure 5.6: Variation of the radial stress along the radius of the cylinder.

Figure 5.7: Variation of the circumferential stress along the radius of the cylinder.

The CPU time required to obtain these two solutions is given in Table 5.4 below. Note that the four element, fifth-order Modified FQP model obtained a solution of comparable accuracy in 22.9% of the time that ANSYS required.

While these results are promising, not all Modified FQP models perform equally well. Care must be taken in choosing the axode direction and the element order of interpolation. As mentioned in Chapter 4, because the Modified FQP element is isoparametric, the interpolation of the element geometry along the axode is also of higher order. The second cylinder model, illustrated in Figure 5.8, takes advantage of this fact. For this model, five elements, each traversing a full 90 degrees of the cylinder, were used. In other words, for this case the element axode was aligned with the cylinder circumference.

The results of a model with sixth-order elements are compared to the theoretical and the ANSYS solutions in Figures 5.9 through 5.11. A comparison of the solution time is given in Table 5.5.

Model	Mesh Size	CPU time	ANSYS/FQP CPU time
FQP ($\zeta=5$)	4 elements	49.16 sec	
ANSYS 5.5	1200 elements	214.44 sec	4.36

Table 5.4: CPU time comparison for the radial axode model.

Figure 5.8: The Modified FQP model of the thick cylinder with circumferential axodes.

Figure 5.9: Variation of the equivalent stress along the radius of the cylinder for the model given in Figure 5.8.

Figure 5.10: Variation of the radial stress along the radius of the cylinder.

Figure 5.11: Variation of the circumferential stress along the radius of the cylinder.

Model	Mesh Size	CPU time	ANSYS/FQP CPU time
FQP ($\zeta=6$)	5 elements	53.19 sec	
ANSYS 5.5	1200 elements	214.44 sec	4.03

Table 5.5: CPU time comparison for the circumferential axode model.

The Modified FQP equivalent stress results for this case are nearly the same as the ANSYS results (Figure 5.9), with both sets of results deviating equally from the theoretical solution in the elastic region of the cylinder. The FQP component stresses, however, do not correlate well with the theoretical solution. As can be seen in Figures 5.10 and 5.11, both the circumferential and the radial stresses oscillate in the plastic region of the cylinder. This is due in part to the fact that the problem to be solved is axisymmetric, meaning that the theoretical solution does not vary along the circumference. By placing the axode along the circumference, the model was effectively using a sixth-order polynomial to approximate a constant value. In other words, the Modified FQP element was misapplied in this example.

This misapplication becomes even more obvious upon examination of Figures 5.12 and 5.13. These figures depict the radial and circumferential stress components for the same circumferentially oriented element model with element orders ranging from $\zeta=4$ to $\zeta=8$ and for a model consisting of ten fourth-order elements, each spanning only 45 degrees. Note that the solution oscillation occurs for all element orders, but is least extreme for the lowest order cases ($\zeta=4$, five 90° elements and $\zeta=4$, ten 45° elements).

Figure 5.12: Variation of the radial stress along the radius of the cylinder for varying orders of modified FQP elements with circumferentially oriented axodes.

Figure 5.13: Variation of the circumferential stress along the radius of the cylinder for varying orders of modified FQP elements with circumferentially oriented axodes.

Notice especially the very poor results obtained with the odd ordered elements ($\zeta=5$ and $\zeta=7$). Again, this is due to poor element choice; the model is using an odd function to model a constant value, which is an even function. Not only were the results poor for these two cases, but the solution time and the total number of iterations required for these solutions are disproportionately high. See Table 5.6 for details.

Further, and more importantly, the model given in Figure 5.8 used only five second-order elements to model the behavior along the radius. Quadratic elements give linear stress and strain results. Clearly, so few elements of this order is inadequate to capture the circumferential stress behavior when the elastic-plastic boundary is contained within the element. (Recall that twenty linear elements were used in this direction to obtain the ANSYS results.) Since the load is applied incrementally in a nonlinear analysis, the first three elements (covering the cylinder up to $r = 0.24$ in) all have to model the yielding behavior. Notice that the solution improves considerably after $r = 0.24$ in for even ordered elements, that is, in the elastic range of the cylinder.

Model	CPU time	Total Iterations
$\zeta=4$, five elements	23.57 sec	35
$\zeta=5$	46.65 sec	46
$\zeta=6$	53.19 sec	36
$\zeta=7$	147.29 sec	72
$\zeta=8$	101.55 sec	35
$\zeta=4$, ten elements	99.45 sec	67

Table 5.6: Comparison of CPU time and total number of iterations to convergence for varying orders of the circumferentially oriented axode model.

5.3.2 Variation of Element Order

The order of interpolation was varied for the radially oriented axode model of Figure 5.4 to further study the effect of element order on the solution. Six variations of the model were run with the order of interpolation along the axode increased from $\zeta=3$ to $\zeta=8$. The results are given in Figures 5.14 through 5.16. The solution times are given in Table 5.7.

Note that except for the case where $\zeta=3$, the results change little among the remaining element orders. The similarity of these results indicates that a converged solution has been achieved. The relatively poor results of the $\zeta=3$ solution indicate that this order is too low to effectively capture the stress in the radial direction.

Notice that while the results change little as the element order increases beyond $\zeta=4$, the time required to obtain a solution increases significantly, such that the $\zeta=8$ solution takes 4.56 times longer than the $\zeta=4$ solution. This result is not entirely unexpected. Recall from Chapter 2 that the solution time per iteration is proportional to the square of the number of degrees of freedom contained in the finite element model. One fourth-order element contains 45 nodes and 135 degrees of freedom; one eighth-order element contains 81 nodes and 243 degrees of freedom. Thus, for the same number of iterations, an eighth-order element with almost twice as many degrees of freedom as the fourth-order element will take approximately four times longer to solve. This example effectively illustrates this fact.

Figure 5.14: Variation of the equivalent stress along the radius of the cylinder for varying orders of modified FQP elements with radially oriented axodes.

Figure 5.15: Variation of the radial stress along the radius of the cylinder for varying orders of modified FQP elements with radially oriented axodes.

Figure 5.16: Variation of the circumferential stress along the radius of the cylinder for varying orders of modified FQP elements with radially oriented axodes.

Model	CPU time
$\zeta=3$	17.86 sec
$\zeta=4$	36.89 sec
$\zeta=5$	49.16 sec
$\zeta=6$	83.97 sec
$\zeta=7$	121.65 sec
$\zeta=8$	168.20 sec

Table 5.7: Comparison of CPU time for varying orders of the radially oriented axode model.

5.4 Elastic-Plastic Thick Cylinder with Internal Pressure and a Concentrated Radial Load

As mentioned before, the previous axisymmetric example was run to verify the accuracy of the model, to understand the behavior of the Modified FQP element, and to obtain an optimal ANSYS model. In this example, a concentrated radial load of $F = 300$ lb/in was applied to the quarter cylinder as shown in Figure 5.17. In addition, an internal pressure of $p = 13,000$ psi, the pressure required to begin yielding the cylinder, was applied. The model geometry and material properties used for this example are the same as those used in the previous section. Note that two Modified FQP models were run; the first consisting of five circumferentially oriented sixth-order elements, and the second consisting of four sixth-order radially oriented elements. Circumferentially oriented elements may be appropriately used for this example due to the behavior of the system. When the concentrated radial load is applied, the model is no longer axisymmetric, that is, the stresses in the cylinder now vary depending on the angular location of interest. With more complex stress patterns expected in the circumferential

Figure 5.17: Modified FQP models of the thick cylinder with internal pressure and a concentrated radial load: a) circumferentially oriented elements, b) radially oriented elements.

direction, it now makes sense to align the element axodes along this direction.

Additionally, with the internal pressure reduced, fewer elements are required along the cylinder radius to capture the circumferential stress field along the radius. The second model, consisting of four elements with radially oriented axodes, was run to capture the behavior of the cylinder near the concentrated load. Note that the same element meshes from the previous section were used for this example.

The results of the Modified FQP analyses were compared to ANSYS results. The data is given in Figures 5.18 through 5.22. Figures 5.18 through 5.20 depict the equivalent stresses in the cylinder along the radius for a specified constant angle. Figures 5.21 and 5.22 illustrate the equivalent stresses along the circumference of the cylinder at the inner and outer radii, respectively. A solution time comparison is given in Table 5.8.

Figure 5.18 shows the change in equivalent stress in the cylinder along the horizontal boundary of the model ($\theta = 0^\circ$). As can be seen in the figure, both Modified FQP models follow the ANSYS results up to about $r = 0.285$ in for the circumferential model and up to about $r = 0.24$ in for the radial model. This variation at the edge is believed to be due to the applied symmetry boundary conditions along this edge as the variation quickly disappears as the angle from the horizontal is increased. The similarity of the curves for an increased angle ($\theta = 54^\circ$) in Figure 5.19 supports this observation. Thus, the small discrepancy in Figure 5.18 near the outer radius between the Modified FQP and ANSYS results is attributed to St. Venant's effect.

Figure 5.18: Variation of the equivalent stress along the cylinder radius at an angle of 0° from the horizontal.

Figure 5.19: Variation of the equivalent stress along the cylinder radius at an angle of 54° from the horizontal.

Figure 5.20: Variation of the equivalent stress along the cylinder radius at an angle of 90° from the horizontal. Note that the concentrated load is applied to the cylinder at $r = 0.30$ in.

Figure 5.21: Variation of the equivalent stress along the inner circumference of the cylinder, radius $r = 0.15$ in.

Figure 5.22: Variation of the equivalent stress along the outer circumference of the cylinder, radius $r = 0.30$ in. Note that the radial load was applied at the 90° -angle position.

Notice also in Figure 5.18 the difference in the way the two Modified FQP models vary from the ANSYS solution. Both variations are consistent with the element formulation and the orientation of the axode. The radial model uses one sixth-order polynomial to model the radial displacement field along the entire horizontal boundary; the circumferential model uses five quadratic polynomials to model this same displacement field. For this reason, the radial model gives a smooth stress distribution along the horizontal axis, while the circumferential model has a more dramatic increase near the outer surface, i.e. for the last element of the cylinder.

A much larger difference between the Modified FQP and the ANSYS solutions is found at the vertical boundary of the cylinder ($\theta = 90^\circ$) as seen in Figures 5.20 and 5.22. As illustrated in Figure 5.20, all three models capture the plastic behavior and elastic-plastic boundary of the cylinder near the inner radius. After this boundary, however, the results begin to diverge, only slightly at first, then dramatically after about $r = 0.265$ in. The concentrated load applied at $r = 0.30$ in, $\theta = 90^\circ$ introduces a singularity into the model. This singularity, which will be discussed in greater detail in the following section, is responsible for the large increase in the ANSYS equivalent stress values along the outer surface of the cylinder seen in the two figures. Because the increase is so large, the stress field for a significant portion of the model (from $r = 0.265$ in to $r = 0.30$ in and from $\theta = 80^\circ$ to $\theta = 90^\circ$) was over predicted. Note that such a large increase did not occur in either Modified FQP solution. The stress localization behavior of linear brick elements near a concentrated load is not unexpected and has been well documented by Zienkiewicz (1989) and Bathe (1996) among others.

Finally, unlike the internal pressure only case, there is little difference in solution accuracy for the two Modified FQP models. However, because fewer iterations were required to achieve the solution for the circumferential element model, this model was solved in 61.5% of the time required by the radial element model. For this reason, the circumferential element appears to be the better model for this case.

Model	Mesh Size	Order	CPU time	ANSYS/FQP time
Circumferential	5 elements	$\zeta=6$	35.81 sec	5.43
Radial	4 elements	$\zeta=6$	58.23 sec	3.34
ANSYS 5.5	1200 elements	linear	194.38 sec	—

Table 5.8: CPU time comparison for the models with internal pressure and a concentrated radial load.

5.5 Torsion of a Square Beam

The final example is the torsional loading of a square steel beam rigidly fixed to a wall. A concentrated load of $F = 3600$ lb was applied to both the top left-hand and the bottom right-hand corners of the beam. Because the cross-section of the beam is noncircular, warping will occur in the z direction. Since the beam is rigidly fixed at $z = 0$, a normal stress distribution that prevents warping occurs at this end. Additionally, a shear stress distribution is developed at this end to balance the torsional load. Therefore, this is a true three-dimensional analysis.

As with the previous analyses, a linear strain-hardening rule and the von Mises yield criterion were used. The isotropic material properties used are: Young's Modulus, $E = 30$ Mpsi; Tangential Modulus, $E_T = 60$ kpsi; Poisson's Ratio, $\nu = 0.3$; and Yield Stress, $\sigma_Y = 30$ kpsi. The beam has a one inch by one inch cross-section and is fifteen inches long.

The Modified FQP model for this example consisted of four elements as illustrated in Figure 5.23 below. A sixth-order element was found to be the lowest order element to effectively capture the warping behavior of the beam in the z direction. The Modified FQP solution using this model was obtained in 102.84 CPU seconds.

Figure 5.23: The four-element Modified FQP model of the beam with a square cross-section loaded in torsion, bending and shear.

Several different ANSYS models were required to obtain a solution to this problem. Initially a mesh of evenly spaced linear elements, given in Figure 5.24, was used. The model failed to converge after 44% completion due to the extreme element deformation at the concentrated loads. To improve the finite element solution at a model singularity such as a concentrated load, it is an accepted practice (see, for example, the work by Zienkiewicz) to either refine the mesh (the h -adaptive method) or to increase the element order (the p -adaptive method) in the neighborhood of the singularity. Following this practice, the elements in the mesh of Figure 5.24 were changed from eight-noded brick elements to twenty-noded bricks, that is, the elements were changed from linear bricks to quadratic Serendipity bricks (the p -adaptive method). The solution using the quadratic elements failed to converge at about 38% completion due again to an element formulation error.

The reason for this error is illustrated in Figure 5.25. As can be seen in the figure, the two loaded elements at the free end of the beam experience extreme deformations at the loaded node. The maximum deformation at this node is 0.56 in. Compare this to the maximum element dimension of 0.33 in. Note also that this large deformation occurred only at the node to which the load was applied. This extreme localized deformation at the loaded node created a severely distorted element. Such a distorted element made it impossible for the software to iterate to a converged solution, and so the analysis failed.

Finer meshes in the region of the concentrated load (the h -adaptive method) led to failures at even lower solution completion percentages. This is because the element failure is due to the relative distortion of the element. The smaller the element, the less the total distortion has to be before convergence becomes impossible. In other words, a

Figure 5.24: The uniform finite element mesh used for the first ANSYS analysis.

Figure 5.25: Deformation of the loaded corner elements of the mesh in Figure 5.24.
 Maximum element deflection is 0.561 in; maximum element dimension is
 equal to 0.33 in.

smaller element fails at a lower load because deformations on the order of the size of the smaller element are achieved at lower loads.

A closer look at the region of the concentrated load gives some insight to the failure of the element after both the p -adaptive and the h -adaptive methods were applied. The elasticity solution for a concentrated force on a straight boundary (Timoshenko and Goodier, 1970) states that the stress is infinite at the point of load application and inversely proportional to the distance from the point of interest to the point of application over the elastic half-space. To proceed to the next increment in a nonlinear analysis, the external forces due to the applied loads must equal within a specified tolerance the forces equivalent to the element stresses at the end of each increment. Element stresses are evaluated at the integration points within an element. Higher order elements require more integration points. With more integration points in the same size element, it is much more likely that at least some integration points will be very close to the point of load application. Likewise, refined meshes near the singularity result in more integration points near the singularity. The stresses at the integration points near the singularity are very high and large deformations result as the solution tries to converge, causing the eventual failure of the model. Mesh refinement and element order increases are effective for linear elastic and explicit nonlinear analyses because these analyses do not require the solution to iterate to convergence at each increment. (A linear elastic analysis is typically solved in one step. An explicit nonlinear analysis avoids iterating to convergence by breaking the analysis into a large number of very small steps. Each step is solved only once, and the result of the current step is added to the sum of the previous results.)

In a final attempt to obtain a converged solution with this mesh and this loading condition, the initial time step for the quadratic element mesh in ANSYS was reduced from 0.15 seconds (the total solution time is one second) to 0.05 seconds. The idea was that if the time step is smaller, then the percentage of the total load applied at each increment is less, and applying the loads in smaller increments typically facilitates convergence. However, this was not the case for this example. The analysis with the reduced initial time step failed due to severely distorted elements at 0.37 seconds, or at about the same percentage of applied load as the larger time step model. The initial time step was then further reduced to 0.01 seconds. Again, the analysis failed due to distorted elements at 0.38 seconds.

Because traditional meshing and time step reduction methods to resolve the loading singularity do not work in implicit nonlinear analysis and because ANSYS does not currently have remeshing capabilities, an alternative mesh, given in Figure 5.23, was used. For this mesh, the element dimension along the length of the beam was tripled near the loaded end of the beam. The same concentrated loading was applied, and quadratic Serendipity brick elements were used. A converged final solution was obtained after 2880.20 CPU seconds. The results, however, were very poor in the region of the applied load. A maximum equivalent stress value of nearly 75,000 psi was obtained near the free end of the beam. Figure 5.24 shows the final shape of the loaded elements. As can be seen in the figure, the elements underwent extreme distortion, but because the element was larger, the magnitude of the distortion relative to the element size was not enough to prematurely end the analysis as it was for the previous mesh.

Figure 5.26: The second iteration of the ANSYS finite element mesh. The elongated end elements are three times longer than the remainder of the mesh.

Figure 5.27: Deformation of the loaded corner elements of the mesh in Figure 5.26.
 Maximum element deflection is 1.527 in; maximum element dimension is equal to 1.0 in.

To improve the results, a third mesh, illustrated in Figure 5.28, was tried. This mesh consisted of quadratic brick elements, and this time included sixteen elements, each two inches long, at the free end of the beam. For reference, the end elements for this third case were twice as long as the end elements of the mesh in Figure 5.26 and six times longer than the elements of the mesh of Figure 5.24. This analysis converged in 255.95 CPU seconds. (Note: this time is 2.49 times longer than the Modified FQP solution required.) The deformed loaded elements are illustrated in Figure 5.29. Notice that there is no excessive element deformation for this mesh. Hence, it appears that large elements are very beneficial near singularities to obtain a stable, converged solution in an implicit nonlinear analysis.

This converged solution comes at a price, however. The ANSYS equivalent stress solution for the complete beam is given in Figure 5.30. Despite the low element distortion for this case, a stress concentration is developed at the applied load and the stress solution in the region near the load is generally poor. This is due to the low order, high aspect ratio elements located here. Compare the results of the ANSYS analysis near the free end of the beam with the theoretical equivalent stress and shear stress distributions for an elastic-plastic square rod in pure torsion as given by Annin and Cherepanov given in Figures 5.31 and 5.32, respectively. The equivalent stress plot of Figure 5.30 begins to approximate the theoretical solution about one-third of the way from the loaded end of the beam. In other words, the ANSYS solution does not begin to approximate theoretical results until approximately one-third of the beam has been truncated. By comparison, the Modified FQP solution, which also experienced some stress localization near the applied load (384 psi overstress a small distance from the

Figure 5.28: The third iteration of the ANSYS finite element mesh. The elongated end elements are two times longer than the adjacent elements and six times longer than the remainder of the mesh.

Figure 5.29: Deformation of the loaded corner elements of the mesh in Figure 5.28.
Maximum element deflection is 0.595 in; maximum element dimension is equal to 2.0 in.

Figure 5.30: Equivalent stress results for the beam model illustrated in Figure 5.25.

Figure 5.31: The theoretical equivalent stress distribution for a square rod loaded in pure torsion (taken from Annin and Cherepanov, 1988). The lines in the figure above are lines of constant stress.

Figure 5.32: The theoretical shear stress distribution for a square rod loaded in pure torsion. Note that the shear stresses τ_{xz} and τ_{yz} are both included on the same plot. (Taken from Annin and Cherepanov, 1988.)

applied load compared to 3034 psi overstress for ANSYS Mesh 3) requires only one-thirtieth of the beam to be truncated before the results began to compare favorably to theoretical solutions. Thus, the long, higher order Modified FQP elements better capture the behavior of the beam and gives a stable converged solution near the concentrated load. This is by no means an insignificant result. Many more complicated example cases were aborted because there was no way to verify the accuracy of the Modified FQP results.

Because of the problems associated with modeling the concentrated load, the torsional load on the beam was reapplied uniformly across the free end of the beam as shown in Figure 5.33. The same loading was applied to the Modified FQP element mesh of Figure 5.23. After rerunning these models, the analysis results were compared at a cross-section located 8.8 in from the fixed end of the beam. The Modified FQP stress results at this cross-section are given in Figures 5.34, 5.36 and 5.38; ANSYS results are given in Figures 5.35, 5.37 and 5.39 on the following pages. The following six plots are organized first by stress type (equivalent or shear) and then by source (Modified FQP followed by ANSYS).

From these figures it is clear that at this distance from the boundary, both the Modified FQP program and ANSYS results qualitatively match the results given by Annin and Cherepanov. The equivalent stress results match the general shape of the theoretical stress isobar curves given in Figure 5.31. The shear stress plots of Figures 5.36 through 5.39 show isobars very similar in shape to those of Figure 5.32.

Figure 5.33: The mesh of Figure 5.24 with a uniformly distributed torsional load.

Figure 5.34: The Modified FQP model equivalent stress contour plot for the beam cross-section located 8.8 in from the fixed boundary.

Figure 5.35: The ANSYS model equivalent stress contour plot for the beam cross-section located 8.8 in from the fixed boundary.

Figure 5.36: The Modified FQP model shear stress (τ_{xz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.

Figure 5.37: The ANSYS model shear stress (τ_{xz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.

Figure 5.38: The Modified FQP model shear stress (τ_{yz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.

Figure 5.39: The ANSYS model shear stress (τ_{yz}) contour plot for the beam cross-section located 8.8 in from the fixed boundary.

5.6 Summary

This chapter has shown through four extensive examples the accuracy and efficiency of the Modified FQP element for nonlinear elastic-plastic problems. Further, the effect of axode orientation, mesh structure and element order were investigated. The element was shown to give results comparable to or better than a commercially available code and often in significantly less time. In addition, the Modified FQP element was shown to be particularly effective in quickly obtaining stable, converged solutions near concentrated loading singularities.

CHAPTER 6

FINAL REMARKS

6.1 Conclusions

As stated in Chapter 1, the purpose of this research was to develop an efficient and accurate method to analyze nonlinear structural problems. To achieve this objective, a modified form of the Finite Quasi-Prism element was developed and then implemented in a large-strain, large-displacement materially nonlinear finite element code. Based on the result of element testing and the numerical testing outlined in Chapters 4 and 5, the following conclusions can be drawn.

First, the Modified FQP element is a monotonically convergent element. The element is free of shear-locking concerns and can be substantially deformed normal to its axode before the solution begins to be affected adversely. Further, the element is effective in large-strain, large-displacement materially nonlinear finite element analysis by generating solutions of comparable or better accuracy in a fraction of the time required by a commercially available code. The displacement field and resulting stress field solutions are smooth and closely match theoretical solutions available in the literature.

The Modified FQP element is especially effective in regions of concentrated loads. Due to the iterative nature of implicit nonlinear analysis, refined meshes of traditional

elements distort drastically in the region of the loading singularity. This distortion is often large enough to halt the solution. It was determined through the course of this research that in order to obtain a converged solution for models with loading of this type, it is necessary to use a coarser rather than a refined mesh in the region of the singularity. This coarser mesh, however, gives poor results in the loaded region, necessitating the truncation of a significant portion of the model. The Modified FQP element mesh is coarse by design, but, due to the higher order displacement interpolation of the element, it still gives solutions near theoretical values in the region of the singularity.

The Modified FQP element is best applied in two-and-a-half-dimensional problems such as gears and structural elements like beams and plates where one Modified FQP element can replace several traditional three-dimensional finite elements. Care must be exercised in model creation, however, as axode orientation and element order can have significant and occasionally adverse effects on the final results and convergence behavior of the solution.

Finally, the solving algorithm used in a nonlinear program has a profound effect of the efficiency of the finite element code. In its original form the program utilized a frontal solver. In order to be competitive with ANSYS 5.5, which now uses the Boeing sparse matrix solver, the frontal solver was replaced with the NASA GPS4 sparse matrix solver. Solution times for identical models dropped on average by a factor of six for the small models used in this work. Now that the Modified FQP element has been verified, it can be used in larger, more complex models. The time saving benefits of the sparse matrix solver are expected to be even greater for these larger models as the frontal solver slows significantly as the wavefront size grows.

6.2 Contributions to Research

This dissertation details a novel method to efficiently perform nonlinear finite element analysis on two-and-a-half-dimensional problems, and marks the first time that the FQP element was used in large-strain, large-deformation, materially nonlinear applications. The specific research contributions from this work include the development and implementation of a modified form of the Finite Quasi-Prism element for use in nonlinear analyses. This work also further defined the behavior of the FQP element by investigating

- the effect of different polynomial orders on solution accuracy and solution time
- the effect of element orientation on solution accuracy and solution time
- the degree to which the FQP element shape can vary from a prismatic shape
- the monotonic convergence behavior of the element

Additionally, the use of the Modified FQP element in nonlinear analysis resulted in smooth, accurate solutions achieved in 20% to 40% of the time required by the commercial code ANSYS.

Finally, and most significantly, the Modified FQP element exhibits stable convergence behavior in regions of loading singularities. As shown in Chapter 5, improper meshing in traditional nonlinear implicit finite element analysis can result in failed, unconverged solutions due to highly distorted elements or in extreme stress concentrations near the singularity that in most cases extends to significant portions of the model. Application of the Modified FQP element in such regions can greatly improve the results and speed the solution time by reducing the number of iterations required for convergence. Note also that the benefit of this element near concentrated loads is not limited to two-and-a-half-

dimensional problems. Recall from Chapter 2 that one of the advantages of using the FQP element over the Finite Prism method is that the FQP element can be placed next to traditional elements in a finite element mesh. It is possible then, to strategically place the FQP element within traditional finite element meshes of more complex three-dimensional problems.

6.3 Recommendations for Future Work

Now that the effectiveness of the Modified Finite Quasi-Prism element has been demonstrated, the following recommendations are suggested to expand the types of problems for which this element can be used.

First, in order for the element to be used in manufacturing analyses such as roll forming, a contact algorithm with friction needs to be included in the program. This algorithm needs to account for the relatively large spacing between nodes on the contact surface. Further, due to the inclusion of friction, the stiffness matrix will no longer be symmetric, which will require program changes within the stiffness matrix formulation and storage subroutines.

Applying the Modified FQP element to roll forming analysis is particularly attractive due to the near prismatic geometry of the roll-formed sheet and the prohibitively long finite element solution times reported in the literature (up to 250 CPU hours as reported by Heislitz, Livatyali, Ahmetoglu, Kinzel and Altan in 1996). Further, the roll forming industry currently relies heavily on empirical rules and acquired experience when designing roll patterns and setting up the roll schedule. Oftentimes this roll schedule is determined by trial and error, and significant amounts of sheet metal may have to be

scrapped while the process is being perfected. Additionally, much of this roll forming experience base has been developed for low carbon steels. Increasingly these steels are being replaced with high strength steels in roll forming applications such as automotive bumper beams. Clearly, these new materials behave differently, and so a new experience base needs to be built. Roll forming is a very competitive industry. As such, manufacturers cannot afford to lose valuable time and material learning how to form these new materials and would benefit from the incorporation of analysis into the design process.

Second, variation of the shape functions along the axode may lead to improved finite element solutions. Although Lagrangian polynomial shape functions were used for this research, other basis functions such as Chebyshev or Legendre polynomials or the trigonometric sine and cosine functions can be used instead. Future research should investigate the effects of these different shape function bases on solution accuracy. For example, trigonometric shape functions may be especially effective in modeling the buckling behavior of structural members under cyclic plastic loading. Experimental research done on square beams and T-beams loaded in completely reversed pure bending (Corona and Vaze, 1996 and 1997 and Corona and Ellison, 1997) show a sinusoidal displacement field that develops prior to the buckling of the beam. It is believed that with appropriately chosen shape functions, the Modified FQP element could accurately model deformation behavior of this type.

BIBLIOGRAPHY

1. ANSYS Basic Analysis Procedures Guide, 3rd Edition, SAS IP, Inc.
2. ANSYS Theory Reference, Ninth Edition, SAS IP, Inc.
3. Annin, B.D. and Cherepanov, G.P., Elastic-Plastic Problems, ASME Press, New York, 1988.
4. Azizian, Z.G. and Dawe, D.J. "Geometrically nonlinear analysis of rectangular Mindlin plates using the finite strip method," *Computers and Structures*, v. 21, no. 3, 423-436 (1985).
5. Babuska, Ivo and Szabo, Barna A., "On the rates of convergence of the finite element method," *International Journal for Numerical Methods in Engineering*, v. 21, 65-88 (1982).
6. Backlund, Jan, "On isoparametric elements," *International Journal for Numerical Methods in Engineering*, v. 12, 731-732 (1978).
7. Bathe, Klaus-Jurgen, Finite Element Procedures, Prentice Hall, Inc., Upper Saddle River, NJ, 1996.
8. Bettess, P., "Operation counts for the boundary integral and finite element methods," *International Journal for Numerical Methods in Engineering*, v. 17, 306-308 (1981).
9. Chandrupatla, Tirupathi R. and Belegundu, Ashok D., Introduction to Finite Elements in Engineering, Prentice Hall, Upper Saddle River, NJ, 1997.
10. Cheung, M.S. and Li, Wenchang, "Finite strip method combined with other numerical methods for the analysis of plates," *Computers and Structures*, v. 45, no. 1, 1-7 (1992).
11. Cheung, Y.K. and Tham, L.G., Finite Strip Method, CRC Press, Boca Raton, FL, 1998.

12. Clough, R.W., "The Finite Element Method in Plane Stress Analysis," *Proceedings, Second ASCE Conference on Electronic Computation*, Pittsburgh, PA, 345-378 (1960).
13. Clough, Ray W., "Comparison of three dimensional finite elements," *Proceedings of the Symposium on Application of Finite Element Methods in Civil Engineering*, Vanderbilt University, Nashville, TN, 1-26 (1969).
14. Corona, E. and Vaze, Suhas, "Response and stability of square tubes under bending," *Journal of Applied Mechanics*, v. 64, pp. 649-657 (1997).
15. Corona, E. and Ellison, M.S., "Plastic buckling of T-beams under pure bending," *Journal of Engineering Mechanics*, v. 123, pp. 466-474 (1997).
16. Corona, E and Vaze, S., "Buckling of elastic-plastic square tubes under bending," *International Journal of Mechanical Science*, v. 38, No. 7, pp. 753-775 (1996).
17. Crisfield, M.A., Non-linear Finite Element Analysis of Solids and Structures, Volume 1: Essentials, John Wiley and Sons, Inc., Chichester, 1991.
18. Crisfield, M.A., Non-linear Finite Element Analysis of Solids and Structures, Volume 2: Advanced Topics, John Wiley and Sons, Inc., Chichester, 1997.
19. Esche, Sven K., Gary L. Kinzel, and Taylan Altan, "Issues in Convergence Improvements for Non-Linear Finite Element Programs," *International Journal for Numerical Methods in Engineering*, v. 40, pp. 4577-4594 (1997).
20. Gallagher, R.J., and Padlog, J., "Discrete element approach to structural stability," *AIAA Journal*, v. 1, No. 6, 1437-1439 (1963).
21. Gallagher, R.J., Gellatly, R.A., Padlog, J., and Mallet, R.H., "A discrete element procedure for thin shell instability analysis," *AIAA Journal*, v. 5, No. 1, 138-145 (1967).
22. Gau, Jenn-Terng, A Study of the Influence of the Bauschinger Effect on Springback in Two-Dimensional Sheet Metal Forming, Ph.D. dissertation, The Ohio State University (1999).
23. Gifford, L. Nash, "More on distorted isoparametric elements," *International Journal for Numerical Methods in Engineering*, v. 14, 290-291 (1979).
24. Heislitz, Frank, Livatyali, Haydar, Ahmetoglu, Mustafa, Kinzel, G. and Altan, Taylan, "Simulation of Roll Forming Process with the 3-D FEM Code PAM-STAMP", *J. of Mat. Proc. Tech.*, Vol. 59, pp. 59-67 (1996).

25. Hestenes, M.R., and Stiefel, E., "Methods of Conjugate Gradients for Solving Linear Systems," *Journal of Research of the National Bureau of Standards*, v. 49, 409-436 (1952).
26. Hill, R., "A theory of the yielding and plastic flow of anisotropic metals," *Proceedings of the Royal Society London, Series A*, v. 193, 281-297 (1948).
27. Hill, R., The Mathematical Theory of Plasticity, Clarendon Press, Oxford, 1950.
28. Hill, R., "Theoretical plasticity of textured aggregates," *Mathematical Proceedings of the Cambridge Philosophical Society*, v. 85, 179-191 (1979).
29. Hinton, E. and Owen, D.R.J., Finite Element Programming, Academic Press, London (1977).
30. Hollomon, J.H., "Tensile deformation," *Transactions of the American Institute of Mining and Metallurgical Engineers*, v. 162, 268-290 (1945).
31. Hosford, W.F., "A generalized isotropic yield criterion," *Journal of Applied Mechanics*, v. 39, 607-609 (1972).
32. Irons, Bruce M., "A frontal solution program for finite element analysis," *International Journal for Numerical Methods in Engineering*, v. 2, 5-32 (1970).
33. Khan, Akhtar S. and Huang, Sujian, Continuum Theory of Plasticity, John Wiley and Sons, Inc., New York, 1995.
34. Kleiber, Michael, Incremental Finite Element Modelling in Non-linear Solid Mechanics, Ellis Horwood, Limited, Chichester, 1989.
35. Krupkowski, A., "The deformation of plastic metals by strain (copper and nickel)," *Annales de L'Academie Polonaise Scientifique et Technique*, v. 7, 113-120 (1945).
36. Lee, Nam-Sua and Bathe, Klaus-Jurgen, "Effects of element distortions on the performance of isoparametric elements," *International Journal for Numerical Methods in Engineering*, v. 36, 3553-3576 (1993).
37. Liu, C., Zhou, Y, and Lu, W., "Numerical simulation of roll-forming by B-spline finite strip method," *Journal of Materials Processing Technology*, v. 60, 215-218 (1996).
38. Liu, Wing Kam, Guo, Yong, Tang, Sing and Belytschko, Ted "A multiple-quadrature eight-node hexahedral finite element for large deformation elastoplastic analysis," *Computational Methods in Applied Mechanics and Engineering*, v. 154, 69-132 (1998).

39. Ludwik, P., Elemente der Technologischen Mechanik, Springer-Verlag, Berlin, 1909.
40. Malvern, Lawrence E., Introduction to the Mechanics of a Continuous Medium, Prentice-Hall, Inc., Englewood Cliffs, NJ, 1969.
41. Medeiros, S.R.P., Pimenta, P.M. and Goldenberg, P., "An algorithm for profile and wavefront reduction of sparse matrices with a symmetric structure," *Engineering Computations*, v. 10, 257-266 (1993).
42. Nayak, G.C. and Zienkiewicz, O.C., "Elasto-plastic stress analysis. A generalization for various constitutive relations including strain softening," *International Journal for Numerical Methods in Engineering*, v. 5, 113-135 (1972).
43. Osgood, W.R., "Stress-strain formulas," *Journal of the Aeronautical Sciences*, v. 13, 45-48 (1946).
44. Owen, D.R.J., Nayak, G.C., Kfoury, A.P. and Griffiths, J.R., "Stresses in a partly yielded notched bar – an assessment of three alternative programs," *International Journal for Numerical Methods in Engineering*, v. 6, 63-73 (1973).
45. Phillips, Aris, Introduction to Plasticity, The Ronald Press Company, New York, 1956.
46. Ratke, L. and Welch, P.I., "The questionability of empirical work-hardening laws," *Zeitschrift for Metallkunde*, v. 74, 226-232 (1983).
47. Ryan, Robert S., "The Role of Failure/Problems in Engineering: A Commentary on Failures Experienced – Lessons Learned," NASA technical Paper 3213, Washington, D.C., March 1992.
48. Storaasli, Olaf O., "Performance of NASA equation solvers on computational mechanics applications," AIAA Paper No. 96-1505, Salt Lake City, UT, April 15-17, 1996.
49. Swift, H.W., "Plastic instability under plane stress," *Journal of the Mechanics and Physics of Solids*, v. 1, 1-18 (1952).
50. Szabo, B.A. and Katz, I.N., "The p -version of the finite element method," *SIAM Journal of Numerical Analysis*, v. 18, 515-544 (1981).
51. Szabo, B.A., "Some recent developments in finite element analysis," *Computers and Mathematics with Applications*, v. 5, 99-115 (1979).
52. Timoshenko, S.P. and Goodier, J.N., Theory of Elasticity, McGraw-Hill Book Company, New York, 1970.

53. Tresca, H., "Sur l'écoulement des corps solides soumis a de fortes pressions," *C.R. Acad. Sci., Paris*, v. 59, 754 (1864).
54. Turner, M.J., Dill, E.H., Martin, H.C., and Melosh, R.J., "Large deflection of structures subject to heating and external load," *Journal of Aerospace Sciences*, v. 27, 97-106 (1960).
55. Vijayakar, S.M., Finite element methods for quasi-prismatic bodies with application to gears, Ph.D. dissertation, The Ohio State University (1987).
56. Vijayakar, Sandeep M., Busby, Henry R. and Houser, Donald R., "Finite element analysis of quasi-prismatic bodies using Chebyshev polynomials," *International Journal for Numerical Methods in Engineering*, v. 24, 1461-1477 (1987).
57. Voce, E., "The relationship between stress and strain for homogeneous deformation," *Journal of the Institute of Metals*, v. 74, 537-562 (1948).
58. von Mises, R., "Mechanik der festen Korper im plastisch deformablen Zustand," *Nachrichten von der Gesellschaft der Wissenschaften zu Gottingen, Mathematisch-Physikalische Klasse*, 582-592 (1913).
59. Wilson, E.L., "Structural Analysis of Axisymmetric Solids," *AIAA Journal*, v. 3, 2269-2274 (1965).
60. Wong, Carlos C.K. and Vardy, Alan E., "Finite prism analysis of plates and shells," *International Journal for Numerical Methods in Engineering*, v. 21, 529-541 (1985).
61. Woodthorpe, J. and Pearce, R., "The anomalous behavior of aluminum sheet under balanced biaxial tension," *International Journal of Mechanical Sciences*, v. 12, 341-347 (1970).
62. Yalamanchili, K.K. and Anand, S.C., "Use of sparse diagonal storage in finite element analysis," *Computers & Structures*, v. 46, No. 6, 963-967 (1993).
63. Zienkiewicz, O.C. and Taylor, R.L., The Finite Element Method, Volume 1, McGraw-Hill Book Company, London, 1989.