

INELASTIC STRAIN ESTIMATION BASED ON
LINE SEARCH AND DIRECT SECANT METHODS

CENTRE FOR NEWFOUNDLAND STUDIES

**TOTAL OF 10 PAGES ONLY
MAY BE XEROXED**

(Without Author's Permission)

DULAL CHANDRA LAHA



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.

ProQuest Information and Learning
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
800-521-0600

UMI[®]

INELASTIC STRAIN ESTIMATION BASED ON LINE SEARCH AND DIRECT SECANT METHODS

By

^D **Dulal Chandra Laha, B. Sc.**

**A thesis submitted to the
School of Graduate Studies
in partial fulfillment of the
requirements for the degree of
Master of Engineering**

**Faculty of Engineering and Applied Science
Memorial University of Newfoundland**

May 2002

St. John's

Newfoundland

Abstract

Approximate inelastic strain estimation is of great use in several types of applications. Besides classical nonlinear FEA, robust techniques such as Neuber's rule, EGLOSS, etc., are available for this purpose. These robust techniques are applicable for small loads just above the initial yield. These methods find secant modulus based on unbalanced local element energy. They do not account for change in the yield boundary while computing secant modulus. Several traditional secant techniques were developed to update the secant stiffness directly in FEA based on nonlinear schemes. The present study explores simple and systematic methods for determining inelastic effects based on line search and direct secant modulus. The main concept of these methods is the minimization of the total residual energy after first linear FEA. A line search with the displacements due to the unbalanced forces spreads the yield zone considerably closer to the actual state.

The present study summarizes important categories of available techniques based on the Newton-Raphson and secant schemes (traditional as well as robust). Seven different possible alternatives for robust estimation of inelastic strain based on line search are examined. Two schemes based on Neuber's rule are examined. These are compared with full nonlinear analysis and EGLOSS, etc. The schemes are applied to study the problems of simply supported beam, propped cantilever, fixed beam (all with UDL), bending of rectangular plate with irregular boundary, simple truss, stretching of a plate with a hole, thick cylinder with internal pressure, thick cylinder with a circumferential

notch, and torispherical shell. The problems were studied for load ranging from just above initial yield to nearly limit loads. The studies indicate that the line search techniques significantly improve the predictions as compared to those made by existing robust techniques. Recommendations based on these results have been made. Two alternatives have been found to be good for general bending and stretching type problems. Another alternative has been found to be good for strain concentration problems.

Acknowledgements

The author wishes to express his sincere appreciation to Dr. Seshu Madhava Rao Adluri under whose supervision this study has been carried out. His technical support, motivation and constructive guidance for the research discussions and personal care throughout the course of this master program are invaluable. The financial support provided by Dr. Adluri's research grant is gratefully acknowledged. The partial funding provided by the Faculty of Engineering and Applied Science, and the School of Graduate Studies, Memorial University of Newfoundland, is also thankfully acknowledged.

Appreciation is extended to Dr. R. Seshadri, Dean of Faculty of Engineering and Applied Science, and Dr. M. Haddara, Associate Dean of Engineering and Applied Science for their support and encouragement during the course of the program.

Many thanks are directed to all his friends at Memorial University and elsewhere especially Aman, Sajjad, Imtiaz, Rabiul, Shamim and Dipa for helping to cope with the problems and frustrations encountered during the program undertaking. The author also wishes to acknowledge the staff at C-CAE for their timely co-operation during difficulties faced with computers and software.

The author would like to acknowledge the role played by his wife, Papri, his parents, B. Bhousion Laha and Anjali Laha. Finally, special thanks are directed to his elder brother, G. C. Laha for his inspiration and encouragement.

Table of Contents

Abstract	ii
Acknowledgements	iv
Table of Contents	v
List of Tables	x
List of Figures	xi
Nomenclature	xiii
Chapter 1 Introduction	1
1.1 General Background	1
1.2 Necessity for the Robust Techniques	2
1.3 Objectives	4
1.4 The Scope of the Present Study	5
Chapter 2 Literature Review	8
2.1 Introduction	8
2.2 Need for Nonlinear Analysis	9
2.3 Causes of Non-linearity	10
2.4 Behavior of Materials: Conservative and Non-Conservative	11
2.5 Material Stress-Strain Curve	11
2.6 Theories of Failure	13
2.7 Basic Stress-Strain Relationships	15
2.8 Solution Algorithms for Nonlinear Problems	17
2.8.1 The Newton-Raphson Method	19
2.8.2 The Initial Stress Method	22
2.8.3 The MNR Method	24

2.8.4	Alpha-Constant Method	26
2.8.5	Self-Correcting Incremental Procedure	28
2.8.6	Variable Step Incremental Procedure	30
2.8.7	Load-Displacement-Constraint Methods	32
2.8.8	Classical Gauss-Seidel Method	36
2.8.9	Conjugate Gradient Method	37
2.8.10	Improved Iteration Strategy	38
2.9	Convergence Criteria	40
2.10	Summary	42
Chapter 3	The Secant Method	45
3.1	Introduction	45
3.2	The Direct Secant Techniques	46
3.2.1	Basic Secant Analysis	46
3.3	Robust Methods	50
3.3.1	Neuber's Rule	50
3.3.2	Reduced Modulus Technique	53
3.3.3	The MARS Method	55
3.3.4	The EGLOSS Method	58
3.4	Traditional Incremental Secant Methods	61
3.4.1	The Arc-Length Method	62
3.5	Incremental (or Path Following) Secant Methods	64
3.5.1	The BFGS Method	64
3.5.2	The SN Method	69
3.5.3	The Modified SN Method	70
3.5.4	Crisfield's MSN Method	72
3.5.5	Barne's Secant Method	75
3.5.6	Chen's Method	78
3.5.7	Tangent-Secant Approach	82

3.6	Summary	89
Chapter 4	Line Search Techniques for Secant Analyses	91
4.1	Introduction	91
4.2	Line Search	94
4.2.1	Basic Line Search Technique	95
4.2.2	LSM1	102
4.2.3	LSM2	102
4.2.4	LSM3	104
4.2.5	LSM4	104
4.2.6	LSM5	104
4.2.7	LSM6	106
4.2.8	LSM7	108
4.3	Procedures Based on Extension to Neuber's Rule	108*
4.3.1	N1 Approach	109
4.3.2	N2 Approach	109
Chapter 5	Numerical Examples	112
5.1	Introduction	112
5.2	Beams with Uniformly Distributed Load	114
5.2.1	Simply Supported Beam	115
5.2.2	Propped Cantilever Beam	121
5.2.3	Fixed-Fixed Beam	126
5.3	Simple Multi-Bar Truss	129
5.4	Plate with a Central Hole under Uniform Tension	134
5.5	Thick Cylinder with a Circumferential Notch Subjected to Internal Pressure	139
5.6	Bending of Rectangular Plate with Partial Fixity	145
5.7	Thick-Walled Cylinder under Internal Pressure	151

5.8	Torispherical Head under Uniform Internal Pressure	156
5.9	Summary	160
Chapter 6	Conclusions and Recommendations	162
6.1	Summary	162
6.2	Conclusions	168
6.3	Recommendations	170
References		172
Appendices		180
A	ANSYS Files for Proposed Techniques	180
A.1	First Linear Elastic Analysis	180
A.2	Unbalanced Nodal Forces Calculation	185 [*]
A.3	Nodal Displacements Calculation	189
A.4	Calculation of Line Search Parameter β	192
A.5	Balanced Nodal Forces Calculation	195
A.6	Modification of Young's Modulus	198
A.7	Second Linear Elastic Analysis	201
B	ANSYS Files for Linear and Nonlinear Analyses	204
B.1	Simply Supported Beam with UDL	204
B.1.1	Linear Elastic Analysis	204
B.1.2	Nonlinear Analysis	206
B.2	Propped Cantilever Beam with UDL	210
B.2.1	Linear Elastic Analysis	210
B.2.2	Nonlinear Analysis	212
B.3	Fixed-Fixed Beam with UDL	215
B.3.1	Linear Elastic Analysis	215
B.3.2	Nonlinear Analysis	217

B.4	Simple Multi-Bar Truss	221
B.4.1	Nonlinear Analysis	221
B.5	Plate with a Central Hole under Uniform Tension	224
B.5.1	Linear Elastic Analysis	224
B.5.2	Nonlinear Analysis	226
B.6	Thick Cylinder with a Circumferential Notch Subjected to Internal Pressure	230
B.6.1	Linear Elastic Analysis	230
B.6.2	Nonlinear Analysis	232
B.7	Bending of Rectangular Plate with Partial Fixity	236
B.7.1	Linear Elastic Analysis	236
B.7.2	Nonlinear Analysis	238
B.8	Thick-Walled Cylinder under Uniform Internal Pressure	242
B.8.1	Linear Elastic Analysis	242
B.8.2	Nonlinear Analysis	244
B.9	Torispherical Head under Uniform Internal Pressure	247
B.9.1	Linear Elastic Analysis	247
B.9.2	Nonlinear Analysis	251
C	Elastic Moduli Softening Macros for EGLOSS, N1 and N2 Methods	256
C.1	Elastic Modulus Softening Macro for EGLOSS Analysis	256
C.2	Elastic Modulus Softening Macro for N2 Analysis	260
D	Equivalent Strain Calculation in ANSYS 5.5	264
E	Exact Analysis of Simply Supported Beam	266

List of Tables

5.1	Simply Supported Beam - Inelastic Strain Prediction	119
5.2	Propped Cantilever Beam - Inelastic Strain Prediction	124
5.3	Fixed-Fixed Beam - Inelastic Strain Prediction	127
5.4	Simple Multi-Bar Truss - Inelastic Strain Prediction	131
5.5	Plate with a Central Hole - Inelastic Strain Prediction	137
5.6	Cylinder with Circumferential Notch - Inelastic Strain Prediction	143
5.7	Bending of Rectangular Plate with Partial Fixity - Inelastic Strain Prediction	149
5.8	Thick-Walled Cylinder - Inelastic Strain Prediction	154
5.9	Torispherical Head - Inelastic Strain Prediction	158

List of Figures

2.1	Material Stress-Strain Curve	12
2.2	Basic Newton-Raphson Scheme - Single DOF System	21
2.3	Initial Stiffness Method - Single DOF System	23
2.4	MNR Scheme - Single DOF System	25
2.5	Alpha-Constant Method - Accelerated Iteration	27
2.6	Basic Steps - Bulirsch and Stoer Method	31
2.7	Spherical Constant Arc-Length Criterion	34
2.8	Constant Increment of External Work Criterion	34
3.1	Basic Secant Analysis	49
3.2	Neuber's Rule - Iteration Procedure	52
3.3	MARS Method - Inelastic Response	56
3.4	EGLOSS Diagram - Inelastic Strain Estimation	60
3.5	Arc-Length Method - Iteration Process	63
3.6	BFGS Method - Single DOF System	68
4.1	LSM1 and LSM2 - Inelastic Strain Estimations	103
4.2	LSM3 and LSM4 - Inelastic Strain Estimations	105
4.3	LSM6 - Inelastic Strain Estimations	107
4.4	Inelastic Strain Estimations Based on Extensions of Neuber's Rule	110
5.1a	Simply Supported Beam - FE Model	118
5.1b	Boundary Conditions at Left Support	118
5.1c	Inelastic Strain Prediction - Simply Supported Beam	120
5.2a	Propped Cantilever Beam - FE Model	123
5.2b	Boundary Conditions at Left Support	123
5.2c	Inelastic Strain Prediction - Propped Cantilever Beam	125

5.3a	Inelastic Strain Prediction - Fixed-Fixed Beam	128
5.4a	Simple Multi-Bar Truss	130
5.4b	Inelastic Strain Prediction - Simple Multi-Bar Truss	132
5.5a	Plate with a Central Hole	135
5.5b	Plate with a Central Hole - FE Model	136
5.5c	Details Near the Hole	136
5.5d	Inelastic Strain Prediction - Plate with a Central Hole	138
5.6a	Cylinder with a Circumferential Notch	140
5.6b	Cylinder with a Circumferential Notch - FE Model	142
5.6c	Details at Notch	142
5.6d	Inelastic Strain Prediction - Cylinder with a Circumferential Notch	144
5.7a	Bending of Rectangular Plate with Partial Fixity	146
5.7b	Bending of Rectangular Plate with Partial Fixity -FE Model	148
5.7c	Inelastic Strain Prediction - Rectangular Plate	150
5.8a	Thick-Walled Cylinder	151
5.8b	Thick-Walled Cylinder (without Boundary Conditions) - FE Model	153
5.8c	Thick-Walled Cylinder (with Boundary Conditions) - FE Model	153
5.8d	Details at Left Side	153
5.8e	Inelastic Strain Prediction - Thick-Walled Cylinder	155
5.9a	Torispherical Head - FE Model	157
5.9b	Detail at Section A	157
5.9c	Detail at Section B	157
5.9d	Inelastic Strain Prediction - Torispherical Head	159
E1a	Simply Supported Beam Subjected to UDL	267
E1b	Yielded Zones over the Beam Length	267
E1c	Elastic Plastic Rectangular Section	267

Nomenclature

All notations are defined in the text when used first. The following is only a list of some frequently used symbols.

Symbols

A_k	representative area over which the equivalent stress acts for any element k
B	strain-displacement matrix
C_n	condition number for updating stiffness matrix
E_0	original Young's modulus
E_I	initial Young's modulus (same as E_0)
E_{II}	second Young's modulus
E_{eqm}	modified equilibrium Young's modulus
E_i	modulus at iteration i
E_r	reduced Young's modulus
E_s	secant modulus
E_e^{i-1}	error quantity defined as the inner product of the residual force vector after the accelerating and the corresponding displacement increment along the secant direction
E_f^{i+1}	error quantity defined as the self inner product of the residual force vector after the accelerating process
F	restoring force
f_i	function value at point u_i
J_i	jacobian at iteration i
I	unit matrix
Δl	arc-length
K_e	an approximate stiffness matrix
K_D	diagonal triangular matrix
K_L	lower triangular matrix
tK	tangent stiffness
K_n'	global tangent stiffness matrix
rK	tangent stiffness at equilibrium configurations
0K	original stiffness matrix

K_n^0	global constant stiffness matrix
\bar{K}_n'	global implicit secant stiffness matrix
R	applied load
R_i'	global residual force vector at iteration i
\bar{R}_i'	global residual force vector
R_{ref}	reference force value
ΔR	residual force
ΔR_e	external load increment
r_i	residual or unbalanced load difference between two successive iterations such as i and $i-1$
SI	element stress intensity
S_m	code allowable stress
u_1	displacements after first linear elastic analysis
u_i	displacements at iteration i
u_b	displacements after line search
Δu	incremental displacements
$\Delta u_i'$	improved incremental displacements
u_{ref}	reference displacement value
V	volume of the structure
W	work done
Z	scalar quantity
$\ \cdot \ $	Euclidean norm
α_i	unknown diagonal matrix of co-efficient at iteration i
β	line search parameter
β^{*+1}	accelerating parameter obtained from minimizing E_e^{i+1}
β_f^{*+1}	accelerating parameter obtained from minimizing E_f^{i+1}
δ_i	displacement changes between two successive iterations, e., g., i and $i-1$
λ	load factor
ν	Poisson's ratio
μ	strain in micron units (10^{-6})
θ	degree of multi-axiality and follow up
Π_i	total potential at iteration i
\in_i	equivalent strains after first linear elastic analysis
\in_e	equivalent element strain

ϵ_{e1}	equivalent element strain after first linear elastic analysis
ϵ_{e2}	equivalent element strain after second linear elastic analysis
ϵ_n	equivalent strain for iteration i
ϵ_{ls}	equivalent total strain after line search
ϵ_p	plastic strain
ϵ_R	force tolerance
ϵ_u	displacement tolerance
ϵ_i	principal strains ($i = 1, 2, 3$)
σ_1	equivalent stresses after first linear elastic analysis
σ_{arbi}	arbitrary stress in the modulus softening process
σ_e	equivalent element stress
σ_{e1}	equivalent element stress after first linear elastic analysis
σ_{e2}	equivalent element stress after second linear elastic analysis
σ_n	equivalent stress for iteration i
$(\Delta\sigma_e)_i$	additional equivalent stress
σ_i	principal stresses ($i = 1, 2, 3$)
σ_{ls}	stress after line search
σ_y	yield stress
σ'_y	modified yield stress
$\bar{\sigma}$	hydrostatic stress
σ_i	deviatoric stress ($i = 1, 2, 3$)

Subscripts

I	initial
II	second
$1,2,3$	indicate the principal directions
$arbi$	arbitrary
e	equivalent
i	iteration number corresponding to a set of analyses
ls	line search
max	maximum
r	reduced
y	yield
D	diagonal
L	lower

Superscripts

0	original
<i>t</i>	tangent
<i>T</i>	transpose

Abbreviations and Acronyms

ADPL	ANSYS Design Parametric Language
BFGS	Broyden-Fletcher-Goldfarb-Shanno
DOF	Degree of Freedom
EGLOSS	Extended Generalized Local Stress Strain
FE	Finite Element
FEA	Finite Element Analysis
FNR	Full Newton-Raphson
GLOSS	Generalized Local Stress Strain
MNR	Modified Newton-Raphson
MSN	Modified Secant-Newton
NR	Newton-Raphson
NFEA-2itr	Nonlinear Finite Element Analysis - within two iterations
QN	Quasi-Newton
SN	Secant-Newton
UDL	Uniformly Distributed Load

Chapter 1

Introduction

1.1 General Background

Inelastic strain determination in structures and structural components has been a topic of considerable interest to researchers in engineering problems. The importance of investigating the inelastic effects of various types of structures for adequate design is being recognized to an increasing extent. In some cases, the behavior is significantly nonlinear at even relatively small loading and for other structures the influence of the geometry changes due to plastic strain on the response of the structure can not be neglected. Economy of operational costs, ensuring performance, safety and durability of structure are needed of a good design. In certain designs, extensive testing is carried out in order to assess accurately the response of the structure considered. However, reliable test data is often very expensive and hence the need for parametric studies has increased the emphasis on theoretical nonlinear analysis. If appropriate analysis techniques are available, expensive testing and accumulation of data can be reduced significantly and a better understanding of the structural behavior can be obtained.

A robust and simple method for estimating inelastic effects aids design procedures in becoming more rational and economical. Besides, inelastic analysis provides reserve strength estimations that are available beyond the elastic limit. This reserve strength is

significant for statically indeterminate structures with high redundancies. Nonlinear FEA to determine inelastic response is frequently used in engineering problems. This requires a significant computational effort. It produces a large amount of output data that has to be interpreted properly to make practical sense. Although cost of computation is coming down significantly, it must be noted that more and more problems are being analyzed for nonlinear effects and hence, these factors create a need for the development of simplified techniques for inelastic analysis. Simplified techniques also serve as checks to verify the effectiveness of full-scale nonlinear analyses. Besides, many simplified techniques possess a robustness that is not often present in full-scale nonlinear analyses.

1.2 Necessity for the Robust Techniques

The Finite Element Analysis (FEA) of structures has proven to be very effective in linear analysis. With regard to nonlinear analysis, FEA is based on the extensions of linear analyses. In nonlinear analysis, stability and accuracy are a great deal more difficult to obtain than in linear analysis and depend on various factors. An important aspect is the use of a consistent continuum mechanics formulation and an effective FE discretization. The most difficult and inescapable stage in FEA is to solve the resulting system of simultaneous equilibrium equations. This problem becomes very difficult and costly if these equations are nonlinear. The use of nonlinear elasto-plastic stress-strain relationships makes the analysis more complicated when compared to linear elastic analysis. A second aspect is the use of material models, which represent the actual materials under field conditions. Specific attention needs to be given to the

implementation of the material model such that it does not introduce instabilities into the solution.

The determination of the most effective approach to a general nonlinear analysis is at present largely a matter of experience on the part of the analyst. In numerical analysis, the accuracy of the results obtained for a system of nonlinear algebraic equations also depends very much on the type of the solution method employed. The most important aspects in FEA are the appropriate finite element model selection and the corresponding interpretation of the results. Generally, incremental step by step solutions where the variables are updated incrementally for each load step thus tracing the full solution path are preferred regardless of which method is used. If complete solution path is not determined, for the particular case of material unloading, it is assumed that the response of the system cannot be evaluated properly. It is also essential in each load step to get good accuracy satisfying all FE equations; otherwise, errors can be significant. An expected solution of nonlinear FE equations mostly depends on the number of incremental load steps. But for a large problem, small incremental steps can result in high costs of analysis. On the other hand, larger load steps might require more iterations since the convergence process might be too slow. Performing equilibrium iterations to obtain proper results is necessary if large or moderately large load steps are used. Thus it is important for engineers to understand the general behavior of nonlinear analytical procedures to control the cost and accuracy of analyses.

Inelastic FEA has become a versatile tool of carrying out elastic-plastic analysis after the advancement of high-speed computers. For performing FEA, many commercial packages are available. It is a general method and could be applied for most engineering problems. A variety of element types and modeling techniques allows good simulation of the problems. Nevertheless inelastic FEA has some inherent drawbacks as well. Applying a detailed inelastic analysis is often questionable due to convergence difficulties and the time requirements. Therefore, a detailed nonlinear analysis may not be always significant in situations where great accuracy is not important. Moreover, the accuracy of FEA is affected by the simplifying assumptions while modeling the problems. This clearly shows the need of developing robust techniques. In the present context, robustness means the ability to provide acceptable results on the basis of conceptual insight and economy of computational effort. Such robust techniques are simple, reliable, and could be based on linear elastic analyses. They are capable of predicting inelastic effects. They are relatively insensitive to errors in material models and other such data collection problems. For performing a preliminary analysis to assess the feasibility of a structure, robust approximates are ideally suited. It can be used to identify critical locations and to approximate the response. Besides, they provide a good alternative estimate to verify full nonlinear analyses results.

1.3 Objectives

The following are the objectives of the present study:

1. Survey the conventional numerical nonlinear solution techniques (Secant and Tangent methods such as Newton-Raphson) and approximate methods (robust methods based on direct secant modulus such as Neuber's rule, EGLOSS, etc.).
2. Develop simplified alternative algorithms combining the ideas of conventional and direct secant methods for inelastic strain estimations. These ideas can include line searches, relaxation locus, etc.
3. Apply the possible alternatives to a variety of problems at different loading situations and compare the results with those obtained by inelastic FEA and the other robust methods.
4. Investigate other possible approximate techniques such as those based on Neuber's rule.
5. Based on the examination of possible alternatives, recommend simplified strategies to obtain approximate and yet robust estimates of inelastic strains.

1.4 The Scope of the Present Study

Chapter I gives an introduction about the importance of nonlinear analysis. The cost and time posed by nonlinear FEA and the need for the robust methods of determining inelastic effects are very briefly explained. The objectives and the scope of present study are also presented.

In Chapter 2, material behaviors such as nonlinearity, conservative, non-conservative, uniaxial stress-strain curve, plastic stress-strain relationship, and different types of yielding criteria are described. The nonlinear solution algorithms pertaining to the elasto-plastic problems are also described. The concept of combining incremental and iterative methods has been studied.

A review of nonlinear solution methods based on secant methods is presented in Chapter 3. Always using conventional methods (e.g., nonlinear FEA) are not suitable for feasibility study as well as the preliminary stages of design. Although these solution methods give the best representation of structural plasticity, obtaining solution may not be easy. Some approximate methods but robust methods (EGLOSS, etc.) based on direct secant techniques (elastic moduli adjustment techniques) have been studied. The usefulness of Neuber's rule for nonlinear evaluations is also discussed.

Seven alternatives (LSM1 to LSM7) based on line search, direct estimation of secant moduli and relaxation line projections are studied in Chapter 4. In addition, two combinations (N1 and N2) based on extensions of Neuber's rule for determining the inelastic strains have been studied.

All these alternatives are applied to a variety of numerical examples in Chapter 5. They include beams, truss, plate with hole, cylinder with notch, bending of plate, thick-walled cylinder and torispherical head etc. They include problems with general bending and stretching as well as problems having strain concentration. Results obtained either by the

robust techniques currently in use such as EGLOSS or the detailed inelastic FEA are compared with those obtained by possible selected alternatives.

In Chapter 6, conclusions and recommendations with a brief summary are discussed. The Appendices contain the input files and ANSYS 5.5 macros written using ADPL that are necessary for solving the numerical examples. An exact analysis for the bending of a simply supported beam with UDL is listed in Appendix E.

Chapter 2

Literature Review

2.1 Introduction

In numerical analysis, the accuracy of results obtained for the nonlinear simultaneous equations depends very much on the type of solution methods employed. The nonlinear solution techniques of simultaneous equilibrium equations that arise in the static analysis of structures and the overall effectiveness of an analysis of numerical procedures used for the solution depends on problem involved as well. In obtaining accuracy, the finite element model could be a significant factor. In the FEA, an accuracy of the analysis can, in general, be improved if a more refined mesh is used. However, this means the cost of the analysis. Therefore, in practice, an analyst tends to employ larger and larger finite element systems (i.e., load steps) to approximate the actual structure. But the fact is that considerable knowledge and judgement by the user might be required to assure a stable and accurate solution. In general, a nonlinear static FEA is most effectively performed using incremental formulation combined with iterative procedure where the variables are updated either incrementally (corresponding to successive load steps) or iteratively. In such a solution it is important that the governing finite element equations are satisfied in each load step. The equilibrium solution could result in many load steps that render the analysis of a large finite element prohibitively expensive. In fact, its practical feasibility

depends on the algorithms available for the solution of the resulting system of equations. Because of requirements that large system be solved, much research effort has gone into optimizing the equation solution algorithms. In the present Chapter, some of popular and related solution methods are outlined.

2.2 Need for Nonlinear Analysis

All phenomena in continuum mechanics are naturally nonlinear. Although using linear formulation is convenient in practice of solving many engineering problems, sometimes, nonlinear analysis is required in order to describe their behavior adequately [Elsawaf, 1979]. Attempts were made to solve such type of problems during the first half of the past century. Series approximations have been applied for solving problems with simple boundary conditions and idealized loading. Closed form solutions are seldom possible because of the complexity of the governing differential equations. Numerical techniques are being employed for such cases and many important problems can be solved for practical purposes using digital computers. Among the techniques used, the finite element method has proved to be the best in dealing with complicated problems, especially with complex boundaries and loading conditions.

The main difference between the mathematical theory of continuum mechanics and that of the finite element method lies in the fact that the former establishes the behavior of an infinitesimal element. By allowing the dimensions of this infinitesimal element to approach zero, partial differential equations can be derived to describe the behavior of the

whole domain. Such equations must be integrated over the domain to establish the solution. On the contrary, the finite element method studies the properties of an element of finite dimensions. Integration is replaced by a finite summation, resulting in a system of algebraic equations [Logan, 1992]. On solving these, the behavior of the whole domain is known.

2.3 Causes of Non-linearity

Non-linearity arises in problems from several sources, which can be grouped into three principal categories:

Changing Status

Some structural features exhibit status-dependent non-linearity. For instance, a tension cable is either slack or taut; a roller support is either in contact or not in contact. Status changes are directly related to load, or determined by some external causes. Situations where contact occurs are common to many different nonlinear applications, and are a distinctive and important subset to the category of changing-status non-linearity.

Geometric Non-linearity

When a structure experiences large deformations, its changing geometric configuration causes the structure to respond non-linearly. Basically, large deflections or rotations characterize geometric non-linearity.

Material Non-linearity

Nonlinear stress-strain relationships are the most common cause of nonlinear structural behavior. Many factors influence material stress-strain properties, including load history (as in elasto-plastic response), environmental conditions (such as temperature), and the amount of time that a load is applied (as in creep response).

2.4 Behavior of Materials: Conservative and Non-Conservative

When all energy put into a system by external loads is recovered if the loads are removed, the system can be said to be conservative. If some energy is dissipated by the system due to plastic deformation, the system is non-conservative. An analysis of a conservative system is path-independent. Loads usually can be applied in any order and in any number of increments without affecting the end results. Conversely, an analysis of a non-conservative system is path-dependent. The actual load response history of the system must be followed closely to obtain accurate results. Path dependent problems usually require that loads be applied slowly by using many sub-steps to the final load value. In the present work only proportional loading paths are considered. The work can be extended to non-proportional loads, if required.

2.5 Material Stress-Strain Curve

A typical stress-strain diagram for steel is shown in Fig. 2.1. The straight portion of the curve OA is the elastic range, and point A is the yield point that demarcates the linear and

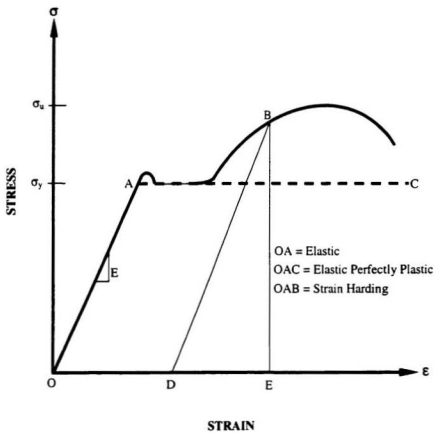


Fig. 2.1: Uniaxial Material Stress-Strain Curve

the nonlinear range of the behavior. On further increasing of load, the stress-strain curve follows the nonlinear path AB. Since the deformation continues the stress required also increases showing the resistance of the material to further plastic deformation.

The stress required to produce this further plastic deformation is usually referred to as the flow stress. Stress and strain are no longer proportional, therefore there is a need to characterize plastic behavior through more appropriate constitute equations. If the material unloads from stressed up to point B, the unloading path is considered to be linear and parallel to the loading OA. The total strain is comprised of two parts. The portion DE is the recoverable elastic and the portion OD is the irrecoverable plastic strains.

2.6 Theories of Failure

As the loading is increased, a point is eventually reached at which changes in geometry are no longer reversible. The beginning of nonlinear behavior is thus marked. The extent of the inelastic deformation preceding fracture is very much dependent upon the material involved. From the viewpoint of design, it is imperative that some practical guidelines be available to predict yielding under the condition of stress, as they are likely to exist in service. To meet this need and to understand the basis of material failure, a number of theories has been developed. Some of them are briefly outlined below [Ugural and Fenster, 1987; Shames and Cozzarelli, 1992].

According to the maximum principal stress theory (Rankine (1802-1872)), material fails by yielding when the maximum principal stress exceeds the tensile yield stress or the minimum principal stress exceeds the compressive yield stress.

According to the maximum shear stress theory (Tresca), yielding starts when maximum shear stress in the material equals the maximum shear stress at yielding in a simple tensile test.

In a multiaxial stress state (if $\sigma_1 > \sigma_2 > \sigma_3$), the maximum shearing stress τ_{max} is

$$\tau_{max} = \frac{1}{2} |\sigma_1 - \sigma_3| \quad (2.6-1)$$

Therefore, yielding begins when

$$\sigma_1 - \sigma_3 = \sigma_y \quad (2.6-2)$$

According to the maximum principal strain theory (St. Venant (1797-1866)), a material fails by yielding when the maximum principal strain exceeds the tensile yield strain or when the minimum principal strain exceeds the compressive yield strain. This theory has been applied with success in the design of thick-walled cylinders.

The von Mises Criterion was proposed by Huber (1904) and further developed by von Mises (1913) and Hencky (1925). According to this theory, failure by yielding occurs when the distortion energy per unit volume in a state of combined stress becomes equal to that associated with yielding in a simple tension test.

$$\sigma_v = \frac{1}{\sqrt{2}} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]^{\frac{1}{2}} \quad (2.6-3)$$

This theory finds considerable experimental support in situations involving ductile materials and plane stress. For this reason, it is commonly used in design.

2.7 Basic Stress-Strain Relationships

Consider an element subjected to a general state of stresses $\sigma_1 > \sigma_2 > \sigma_3$ where 1, 2, 3 indicate the principal directions. The principal stresses in any three-dimensional stress system can be written in the summation of $\bar{\sigma}$, the hydrostatic stress or the mean of the three principal stress values and σ' , the deviatoric stress. The hydrostatic or the mean stress for the uniaxial case is given by

$$\bar{\sigma} = \frac{\sigma_1 + \sigma_2 + \sigma_3}{3} = \frac{\sigma_1}{3} \quad (2.7-1)$$

Therefore, the deviatoric stresses are obtained as

$$\sigma'_1 = \sigma_1 - \bar{\sigma} = \frac{2}{3} \sigma_1, \quad \sigma'_2 = \sigma_2 - \bar{\sigma} = -\frac{1}{3} \sigma_1, \quad \sigma'_3 = \sigma_3 - \bar{\sigma} = -\frac{1}{3} \sigma_1 \quad (2.7-2)$$

or,
$$\frac{\sigma'_1}{\sigma'_2} = \frac{\sigma'_1}{\sigma'_3} = -2 \quad (2.7-3)$$

Similarly, the deviatoric strains can also be defined. For volume constancy, the sum of strains must be zero.

Symmetry in the uniaxial case leads to

$$\frac{\varepsilon_1}{\varepsilon_2} = \frac{\varepsilon_1}{\varepsilon_3} = -2 \quad (2.7-4)$$

and a comparison with Eq. 2.7-3 shows that

$$\frac{\varepsilon_1}{\sigma_1} = \frac{\varepsilon_2}{\sigma_2} = \frac{\varepsilon_3}{\sigma_3} = \text{constant} \quad (2.7-5)$$

Eq. 2.7-5 can be manipulated to give the following equations [Ugural and Fenster, 1987]

$$\varepsilon_i = \frac{1}{E_i} [\sigma_i - \nu(\sigma_j + \sigma_k)] \quad (2.7-6)$$

where, E_i is the secant modulus (a function of the state of stress) and evaluated by

$$E_i = \frac{\sigma_e}{\varepsilon_e} \quad (2.7-7)$$

in which σ_e and ε_e indicate the effective stress and strain, respectively.

According to von Mises theory, the effective stress connects the uniaxial yield stress to the general state of stress at a point given by

$$\sigma_e = \frac{1}{\sqrt{2}} \left\{ (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right\}^{\frac{1}{2}} \quad (2.7-8)$$

$$\epsilon_e = \frac{1}{\sqrt{2}(1+\nu)} \left\{ (\epsilon_1 - \epsilon_2)^2 + (\epsilon_2 - \epsilon_3)^2 + (\epsilon_3 - \epsilon_1)^2 \right\}^{\frac{1}{2}} \quad (2.7-9)$$

For loading situations in which the components of stress do not increase continuously, the incremental theory must be used. Upon these circumstances, the total strain theory or the deformational theory can not describe the complete plastic behavior of the material. The incremental theory offers another approach, treating not the total strain associated with a state of stress but rather the increment of strain.

2.8 Solution Algorithms for Nonlinear Problems

In the finite element idealization, the whole (continuous) body is divided into a finite number of sub regions called finite element, which interconnect at node. The solution function over each element is assumed in terms of the generalized nodal variables, which are usually the function itself and sometimes, its derivatives. Using this approximate function with an appropriate variational principle or the governing differential equations the element matrices are obtained. Finally, a system of algebraic equation is solved, in order to obtain the unknown nodal variables.

Based on any of the well-known variational principles in solid mechanics, a variety of finite element models has been established. At present, the displacement model is most widely used because of simplicity and easy to program. The principle of minimum of potential energy is used to construct the load-nodal variable equations for the displacement model. The displacement functions are assumed over each element so that

compatibility within each element and across inter-element boundaries is assured. A displacement model of a nonlinear finite element problems demands the simultaneously satisfaction of the global stiffness equation. In this model, the equilibrium equations are required to solve, namely:

$$\Delta R = R - \int_V B^T \sigma dV = 0 \quad (2.8-1)$$

where, R and ΔR are the applied (external) and the generalized residual or unbalanced force vectors, respectively. B is the derivative matrix defining strain-displacement relationship and σ is the stress vector based on strain energy per unit undeformed volume.

These solution techniques are quite general and are entirely and directly applicable to all those analyses that lead to symmetric coefficient matrices. Two types of solution techniques such as direct and iterative are available. In direct (incremental) solution techniques, the equilibrium equations are solved using a number of load steps and operations are predetermined in an exact manner. In such techniques, the elasto-plastic problem is dealt as a series of linear analyses. The load vector is broken into a number of smaller but finite increments. The structure is assumed to respond linearly within each step and the response is obtained without iteration. The stiffness (in secant methods) or tangent matrix (in Newton-Raphson schemes) is evaluated at the beginning of each step and assumes constant for the whole increment. The final solution obtained by the

summation of the incremental displacements due to each load increment. Errors are likely to accumulate after several steps unless very fine steps are adopted.

Otherwise, the solution may diverge considerably from the true response. The accuracy can be improved by applying equilibrium corrections.

Iterations are used when an iterative technique is employed. In such technique, the whole load is applied on the structure and equilibrium is restored by iteration. Either the stiffness or tangent matrix is reformed at every iteration (e.g., Full Newton -Raphson method) or held constant for several iterations (e.g., modified Newton-Raphson). An iterative procedure can be assumed to have converged when the unbalanced load becomes acceptably small. The Euclidean norm or some other property of the vector judges the convergent tolerances. It may also be advantageous to devise mixed iteration schemes combining the features of both techniques. Currently, most of the finite element packages are based on a step-by-step load incrementation and a corresponding iteration procedure.

2.8.1 The Newton-Raphson Method

The most frequently used iterative schemes in the FEA are the Newton-Raphson (NR) schemes [Kao, 1974; Bathe, 1996]. Such Newton-Raphson schemes are based on the tangential stiffness matrix and can be applied as an incrementally or iteratively or both. After each load increment or iteration, the tangent stiffness matrix is reformed. At each iteration, the NR procedure evaluates the unbalanced load. The difference between the

restoring force and the applied load gives the unbalanced load. A linear solution is performed with the unbalanced load. The unbalanced load is re-evaluated until the convergence is satisfied.

The coefficient matrix is updated and a new solution is obtained. At each sub step, a number of equilibrium iterations may be performed to obtain a converged solution. This iteration continues until the problem converges.

In this method, the converged solution u_i for the particular load step is known. The solution for the next load step or iteration is required. The updated tangent matrix $'K_i$ and the restoring load F_i are computed corresponding to the configuration of known solution u_i . The incremental displacement Δu_i and the next approximation of u_{i+1} are evaluated by

$$'K_i \Delta u_i = R - F_i \quad (2.8-2)$$

$$u_{i+1} = u_i + \Delta u_i \quad (2.8-3)$$

Repeating Eqs. 2.8-2 to 2.8-3 and updating of tangent stiffness matrix at each iteration gives converged solution. The solution obtained at converged would correspond to applied load level. The final solution would be in equilibrium such that the restoring load F_i (computed from the current stress state) would equal the applied load R (or at least

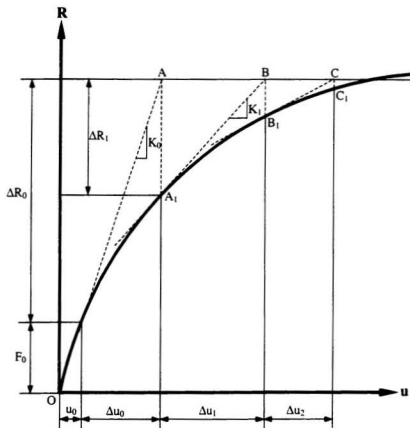


Fig. 2.2: Basic Newton-Raphson Scheme - Single DOF System

within some tolerance). This procedure is also known as Full Newton-Raphson (FNR) procedure. Iterative solution (for one DOF system) is depicted graphically in Fig. 2.2.

Usually, the Newton-Raphson methods are used according to their original definition in which the load increment is predetermined. The determination of load step size in advance requires a lot of intuition, nevertheless the final convergence failure in the neighborhood of critical points can not be avoided [Ramm, 1982]. Without additional modifications the solution procedure is not able to trace the structural response beyond critical points. Although post critical states are usually not tolerated in the design of a structure the knowledge of this range allows a much better judgement of the structure.

2.8.2 The Initial Stiffness Method

In the Full Newton-Raphson (FNR) schemes, re-calculating and factorizing tangent stiffness matrix at every iteration are expensive and laborious. The expense of these re-calculating and factorizing many times the coefficient matrix defined in Eq. 2.8-2 can be avoided. Only the stiffness matrix needs to be formulated corresponding to a linearization of the response about the initial configuration of the finite element system [Zienkiewicz, et al., 1969].

The initial stiffness matrix 0K is applied in Eq. 2.8-2 and operates on the equilibrium equations given below

$${}^0K\Delta u_i = R - F_i \quad (2.8-4)$$

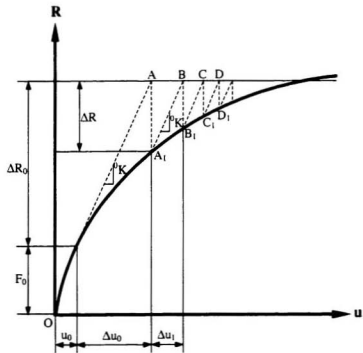


Fig. 2.3: Initial Stiffness Method - Single DOF System

This process may lead to a very slow convergence for significant non-linearity. Even the solution may diverge [Schmidt, 1977]. This occurs particularly when the system stiffens during the response.

To prevent divergence of the solution for slowly stiffening problems and meet the convergence early, it may be effective to use the modified Newton-Raphson procedure described below.

2.8.3 The Modified Newton-Raphson Method

In a modified form of Newton-Raphson method, the tangent stiffness is held constant for several iterations before updating is required. Evaluating and factoring a new tangent stiffness matrix at each iteration is expensive and time consuming. In practice, depending on the non-linearity present in the analysis, to evaluate a new tangent stiffness at a certain time, can be more efficient. The choice of load steps when the stiffness matrix should be updated depends on the degree of non-linearity in the system response and the effectiveness of the solution approaches also depends on the specific problem involved. For small load increment, matrix updating at every iteration appears unnecessary. Establishing new tangent stiffness ${}^{\tau}K$ (where, τ indicates an accepted equilibrium configurations for tangent stiffness tK) only at the beginning of each load step modifies the FNR procedure. This reduces the computing time considerably involving fewer tangent stiffness reformations than the FNR procedure. Of course, the modified Newton-Raphson method (MNR) procedure requires much more iterations in a

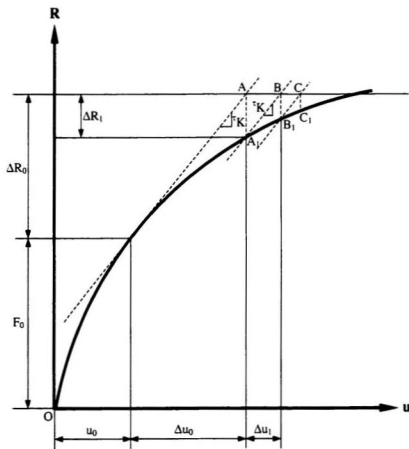


Fig. 2.4: Modified Newton-Raphson Scheme - Single DOF System

load step compared to the FNR procedure. The most powerful procedure for reaching convergence is definitely the FNR procedure but if the MNR procedure is employed, the solution cost may be reduced significantly. Therefore, practically, both solution options can also be very valuable.

The procedure followed in such method is exactly the same as in the case of the FNR procedure except no reformulation of tangent stiffness at each iteration.

In such cases, Eq. 2.8-2 can be replaced by

$${}^e K \Delta u_i = R - F_i \quad (2.8-5)$$

2.8.4 Alpha-Constant Stiffness Method

An improved value $\Delta u_i'$ of incremental displacement Δu_i obtained in the FNR schemes could be evaluated as [Nayak and Zienkiewicz, 1972]

$$\Delta u_i' = \alpha_i \Delta u_i \quad (2.8-6)$$

where, α_i is an unknown diagonal matrix of coefficients.

Defining $\Delta u_i'$ as the improved displacement change corresponding to force ΔR_{i-1} , the approximate relation is expressed as

$${}^e K \Delta u_i' = ({}^e K - {}^e K) \Delta u_i' = -\Delta R_{i-1} \quad (2.8-7)$$

where, ${}^{\alpha}K = {}^{\alpha}K - {}^{\alpha}K$ is a function of displacements and the degree of non-linearity of the problem at any stage.

Pre-multiplying the above equation by ${}^{\alpha}K^{-1}$ and inserting Eq. 2.8-2.

$${}^{\alpha}_i \Delta u_i - {}^{\alpha}K^{-1} ({}^{\alpha}K \Delta u_i) = -{}^{\alpha}K^{-1} \Delta R_{i-1} = \Delta u_i \quad (2.8-8)$$

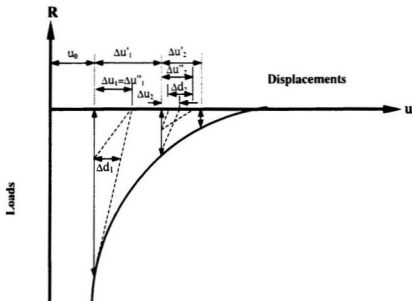


Fig. 2.5: Alpha - Constant Method-Accelerated Iteration

The approximation to the second term by taking the previous α_{i-1} is

$$\Delta u_i = \Delta u_i = \alpha_{i-1} \Delta u_i \quad (2.8-9)$$

Thus writing,

$$\Delta d_i = {}^o K^{-1} {}^c K \alpha_{i-1} \Delta u_i \quad (2.8-10)$$

or/ and

$$(\alpha_i - I) \Delta u_i = {}^o K^{-1} {}^c K \alpha_{i-1} \Delta u_i \quad (2.8-11)$$

which defines the matrix α_i for the k^{th} diagonal term results in

$$\alpha_i^k = \frac{\Delta d_i^k}{\Delta u_i^k} + 1 \quad (2.8-12)$$

with the restriction that when $\Delta u_i^k = 0$, $\alpha_i^k = 1$.

This allows an improved value to be used immediately and the new displacement correction is calculated from Eq. 2.8-6.

2.8.5 Self-Correcting Incremental Procedure

Stricklin, et al. [1971] first proposed this form and Stebbins [1971] studied it again. The procedure has proven itself to be stable and accurate upon proper selection of Z

[Masset, et al., 1971]. The advantages are best realized for problems of highly nonlinear behavior and for systems with many degrees of freedom.

Derivative of equilibrium equations of $Ku = \lambda R - F$ (where K = structural stiffness matrix, u = generalized displacements, λR = generalized forces due to applied external loads and F = column matrix of pseudo forces due to non-linearity) with respect to the scalar multiplier λ yields

$$K\dot{u} = R - \dot{F} \quad (2.8-13)$$

or,

$$(K + k)\dot{u} = R \quad (2.8-14)$$

where, $\dot{F}(u) = k(\dot{u})$, $k = dF_i/du_i$

A common solution procedure is by an Euler forward difference, which results in the incremental solution procedure. The solution procedure tends to drift away from the true solution unless very small load increments are taken where Z is a scalar quantity.

$$\Delta\dot{R} + Z\Delta R = 0 \quad (2.8-15)$$

It is notable that even if ΔR becomes nonzero the solution tends to reduce the unbalanced loads ΔR exponentially to zero and is thus called a self-correcting procedure.

For $Z = 1/\Delta\lambda$, the procedure reduces to the incremental approach with a one step NR method.

2.8.6 Variable Step Incremental Procedure

Simplicity is the main attraction of the incremental approach and easy to incorporate in a computer program. But in basic form of the incremental approach, it gives rise to an appreciable drift error. Methods that have been presented in this section to reduce this error tend. The one-step NR corrector of Hommeister, et al. (1970) and the mid-increment procedure [Roberts and Ashwell, 1971] among of others are very effective at improving the accuracy of the basic incremental technique. However, these improved methods do not possess an accurate estimate of the discretization errors. Thomas [1973] presented an algorithm that does have such an estimate and suggested for use where the varying step size might prove profitable.

In this algorithm, the nonlinear problem of structural analysis is transformed into a first order ordinary differential equation such as

$${}^tK(u)du = d\lambda R \quad (2.8-16)$$

In the limit as $d\lambda$ approaches zero, the first order ordinary equation can be written as

$$\dot{u} = {}^tK^{-1}(u)R \quad (2.8-17)$$

The extrapolation method such as Bulirsch and Stoer method (1966) can be used to solve the above equations. Two separate estimates of displacements are given at every other load increment. These values are averaged before the algorithm is repeated with a new starting point as shown in Fig. 2.6.

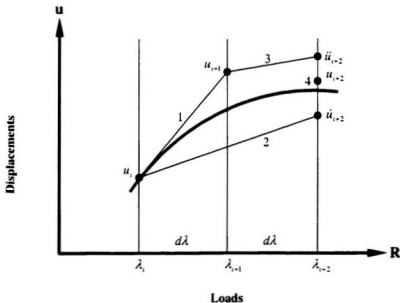


Fig. 2.6: Basic Steps - Bulirsch and Stoer Method

The basic steps of Bulirsch and Stoer method to be followed are listed below.

$$u_{i+1} = u_i + d\lambda \left\{ {}^t K(u_i) \right\}^{-1} R$$

$$\dot{u}_{i+2} = u_i + 2d\lambda \left\{ {}^t K(u_{i+1}) \right\}^{-1} R$$

$$\ddot{u}_{i+2} = u_{i+1} + d\dot{\lambda} \left\{ {}^t K(\ddot{u}_{i+2}) \right\}^{-1} R$$

$$u_{i+2} = \frac{1}{2} \dot{u}_{i+2} + \frac{1}{2} \ddot{u}_{i+2} \quad (2.8-18)$$

2.8.7 Load-Displacement-Constraint Methods

Structural collapse load calculation is an important requirement in a nonlinear FEA. The structural response becomes increasingly nonlinear as the load increases. At certain point the collapse load is reached. In order to calculate the response for collapse mechanism, initially relatively larger increments are employed. But at the approaching of collapse mechanism, the load increment needs to be small. However, there is a difficulty of traversing the collapse point. At that point the stiffness becomes singular (i.e., the slope of the load-displacement response curve is zero) and beyond that point a special solution procedure that allows for a decrease in load and an increase in displacement must be used to calculate the ensuing response. In this approach, when iteration in the load displacement space is performed the incremental equation becomes

$${}^t K \Delta u_i = (\lambda_{i-1} + \Delta \lambda_i) R - F_{i-1} \quad (2.8-19)$$

Both incremental displacement Δu_i and load multiplier $\Delta \lambda_i$ are the unknowns. The additional equation required for solution is

$$f(\Delta \lambda_i, \Delta u_i) = 0 \quad (2.8-20)$$

Several constraint equations of this form have been proposed e.g., the tangent constant arc-length [Risks, 1979] and the spherical constant arc-length [Crisfield, 1981 and Ramm, 1981-82]. Bathe and Dvorkin [1983] considered two different constraints depending on the response and load level: the spherical constraint arc-length and a constant increment of external work.

The spherical constant arc-length is in general used in the response of regions far from critical points and incremental equation mentioned above becomes

$$(\lambda_i)^2 + \frac{u_i^T u_i}{\beta} = (\Delta l)^2 \quad (2.8-21)$$

where, Δl is the arc length for the step and β is a normalizing factor.

The scheme of constant increment of external work W based on the history of iterations in the previous incremental steps is used near the critical points. In this case, the incremental equation becomes

$$\left(\lambda_i + \frac{1}{2} \Delta \lambda_i \right) R^T \Delta u_i = W \quad (2.8-22a)$$

and for $i = 2, 3, 4, \dots$

$$\left(\lambda_{i-1} + \frac{1}{2} \Delta \lambda_i \right) R^T \Delta u_i = 0 \quad (2.8-22b)$$

To solve the governing equations, this incremental equation can be rewritten as

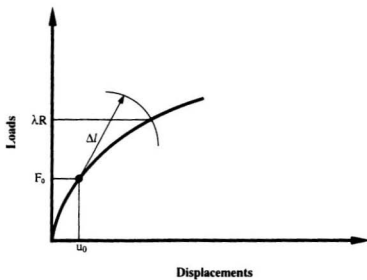


Fig. 2.7: Spherical Constant Arc-Length Criterion

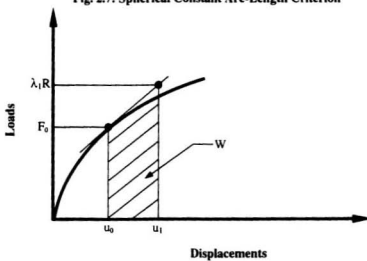


Fig. 2.8: Constant Increment of External Work Criterion

$${}^r K \Delta \bar{u}_i = \lambda_{i-1} R - F_{i-1} \quad (2.8-23)$$

$${}^r K \Delta \bar{u} = R \quad (2.8-24)$$

Therefore,

$$\Delta u_i = \Delta \bar{u}_i + \Delta \lambda_i \Delta \bar{u} \quad (2.8-25)$$

Employing the spherical constant arc length criterion, the next load multiplier and displacements are evaluated. The load multiplier $\Delta \lambda_i$ is determined from the quadratic equation given by the combination of Eqs. 2.8-21 and 2.8-26 to 2.8-27.

$$\lambda_i = \lambda_{i-1} + \Delta \lambda_i \quad (2.8-26)$$

$$u_i = u_{i-1} + \Delta \bar{u}_i + \Delta \lambda_i \Delta \bar{u} \quad (2.8-27)$$

The load multiplier $\Delta \lambda_i$ is directly calculated using external work criterion from Eq. 2.8-22 and the values $\Delta \lambda_i$ for $i = 2, 3, 4, \dots$ are obtained from

$$\Delta \lambda_i = -\frac{R^T \Delta \bar{u}_i}{R^T \Delta \bar{u}} \quad (2.8-28)$$

A complete solution algorithm based on the above method must of course also contain a special scheme to start the incremental solution and iterating when divergence is imminent and then restart itself with new iterative parameters. Complete solution

methods with these ingredients are very valuable and are in common use for the structural collapse analysis.

In some nonlinear static analyses in the case of Newton-Raphson method, the tangent stiffness matrix may become singular (or non-unique), causing severe convergence difficulties. Such occurrences include nonlinear buckling analyses in which the structure either collapses completely or 'snaps through' to another stable configuration. The arc-length method causes the Newton-Raphson equilibrium iterations to converge along an arc, thereby often preventing divergence.

2.8.8 Classical Gauss-Seidel Method

Clough and Wilson applied the Gauss-Seidel method in the early applications of the FEA. An initial estimate u_1 for the displacement is assumed. The number of iterations required depends on the quality of the starting estimate u_1 and on the conditioning of the stiffness matrix. After an initial estimate, the iterative equation to evaluate the solution is

$$u_{i+1} = K_D^{-1} (R - K_L u_{i+1} - K_L^T u_i) \quad (2.8-29)$$

where, K_D and K_L are the diagonal and lower triangular matrices, respectively.

To increase the convergence rate, an over relaxation β can be used. In such cases, the equation mentioned above becomes

$$u_{i+1} = u_i + \beta K_D^{-1} (R - K_L u_{i+1} - K_D u_i - K_L^T u_i) \quad (2.8-30)$$

2.8.9 Conjugate Gradient Method

The conjugate gradient algorithm of Hestenes and Stiefel (1952) is one of the most effective and simple iterative techniques for solving equilibrium equations [Bathe, 1996]

The algorithm based on the idea of minimizing total potential given as

$$\Pi_i = \frac{1}{2} u_i^T K u_i - u_i^T R \quad (2.8-31)$$

The aim is to improve displacement u_{i+1} efficiently corresponding to decrease total potential (i.e., for $\Pi_{i+1} \leq \Pi_i$) in each iteration. Like the classical Gauss-Seidal method, the starting displacement u_1 is chosen and calculated the residual or unbalanced force ΔR_1 based on the formula

$$\Delta R_1 = R - K u_1 \quad (2.8-32)$$

If the residual force ΔR_1 equals to zero, quit that evaluation for next independent load otherwise, the values of u_{i+1} , ΔR_{i+1} and p_{i+1} are calculated based on the equations given by

$$u_{i+1} = u_i + \alpha_i p_i \quad (2.8-33)$$

$$\Delta R_{i+1} = \Delta R_i - \alpha_i K p_i \quad (2.8-34)$$

$$p_{i+1} = \Delta R_{i+1} + \beta_i p_i \quad (2.8-35)$$

$$\alpha_i = \frac{\Delta R_i^T \Delta R_i}{p_i^T K p_i} \quad (2.8-36)$$

$$\beta_i = \frac{\Delta R_{i+1}^T \Delta R_{i+1}}{\Delta R_i^T \Delta R_i} \quad (2.8-37)$$

The iteration continues until the convergence achieves.

2.8.10 Improved Iteration Strategy

The computational self-correcting method proposed by Haisler and Stricklin [1971] was rather cumbersome. Batoz and Dhatt [1979] modified the self-correcting method and proposed a simpler procedure discussed below.

In this method, incremental displacements Δu_e and Δu_r due to ΔR , unbalanced load and ΔR_r , an external load increment, respectively, are calculated as

$$\Delta u_e = {}^t K^{-1} \Delta R \quad (2.8-38)$$

$$\Delta u_r = {}^t K^{-1} \Delta R_r \quad (2.8-39)$$

The actual external load increment to be applied is $\alpha \Delta R_r$ such that that displacement increment satisfies the specified displacement limit. The value of α is easy to calculate from

$$\Delta u = \Delta u_n + \alpha \Delta u_r \quad (2.8-40)$$

Powell and Simons [1981] modified this strategy by choosing a linear combination of the two separate increments for iteration such as

$$\Delta u = \alpha_n \Delta u_n + \alpha_r \Delta u_r \quad (2.8-41)$$

where, α_n and α_r are determined according to certain criteria.

This procedure has a great deal of flexibility because of using varieties of criteria. The simplest iteration scheme is obtained by requiring (a) that the entire unbalanced load be applied in each iteration ($\alpha_n = 1$) and (b) that a specified displacement component u_n remains constant (i.e., $\Delta u_n = 0$). The increment Δu_n is expressed as

$$\Delta u_n = b_n^T \Delta u \quad (2.8-42)$$

in which b_n contains zero except for unity at location n . Therefore, the requirement for the iteration (with $\alpha_n = 1$) is

$$\Delta u_n = 0 = b_n^T \Delta u_n + \alpha_r b_n^T \Delta u_r \quad (2.8-43)$$

Therefore,

$$\alpha_r = -\frac{\Delta R_r^T \Delta u_n}{\Delta R_r^T \Delta u_r} \quad (2.8-44)$$

where, b_s is any convenient vector (i.e., $b_s = \Delta R_s$).

Other solution strategies could also be considered e.g., $\alpha_s=1, \alpha_r=0$ and $\alpha_s=0, \alpha_r=1$ corresponds to constant load iteration and step by step analysis without an equilibrium criterion.

2.9 Convergence Criteria

Often the numerical solution techniques are based either on iterative process alone or involving the combination of an incremental and iterative procedures. A problem associated with iterative techniques is the decision as to whether the current solution is sufficiently close to the true solution without knowing itself. An incremental solution strategy based on iterative process is effective and rational when realistic criteria are used for the termination of iteration. The solution obtained at the each iteration must be checked to see whether it has converged or diverged. If the convergence tolerance is too high inaccurate results are obtained and if the tolerance is too small much more computational effort is spent to obtain needless accuracy. Similarly, an ineffective divergence check can terminate the iteration when the solution is not actually diverging or force the iteration to search for an unattainable solution.

The convergence criteria used for nonlinear structural problems that are solved by iteration can in general be grouped into four categories described below [Bergan and Clough, 1972].

Force Criterion

This criterion is based on a comparison between the unbalanced forces or residual forces ΔR within the structure and external loads. This requires that the norm (a scalar measure of the magnitude) of the unbalanced load vector within a preset tolerance, e.g.,

$$\|\Delta R\| \leq \epsilon_R R_{ref} \quad (2.9-1)$$

where, ϵ_R and R_{ref} are the tolerance and reference values, respectively.

When the force quantity requires to be compared with completely different order or even of different dimensions, such comparison does not make sense. For instance, the inconsistencies in units can appear in the force vector e.g., forces and moments in beam elements and the displacement solution does not enter the termination criterion. However, this in fact corresponds to working with displacements so that direct use of a displacement criterion would appear to be preferable.

Displacement Criterion

This criterion is based on the displacements instead of unbalanced forces. In this criterion, the displacement at the end of each iteration is within a certain tolerance of the true displacement solution, e.g.,

$$\|\Delta u\| \leq \epsilon_u u_{ref} \quad (2.9-2)$$

where, ϵ_u and u_{ref} are the tolerance and reference values, respectively.

Incremental Internal Energy Criterion

There are some difficulties (e.g., an elasto-plastic truss with a very small strain-hardening modulus entering the plastic region) where the unbalanced loads may be very small but the displacements may still be much in error. In this situations, there is another convergence criterion based on internal energy increment would be much more effective. In this approach, the increment in internal energy at each iteration (i.e., the amount of work done by the unbalanced loads on the displacement increments) can be compared to the initial internal energy increment.

Stress Criteria

A stress criterion involves a check on changes in stress values during an iteration cycle. These changes can be compared with prescribed stress level. This type of criterion is well suited for truss, cable and membrane structures.

2.10 Summary

Material behaviors such as non-linearity, uniaxial stress-strain curve, plastic stress-strain relationship, and different types of yielding criteria have been studied [Ugural and Fenster, 1987; Shames and Cozzarelli, 1992]. The nonlinear solution algorithms (regular NR, MNR, initial stiffness methods, etc.) pertaining to the elasto-plastic problems have

been also reviewed [Zienkiewicz, et al., 1969; Stricklin, et al., 1971; Kao, 1974; Bathe, 1996; Kowalczyk and Bojczuk, 1996]. Most of them are based on updating co-ordinates of tangent stiffness matrix and initial displacements iteratively. The concept of combining incremental and iterative methods [Thomas, 1973; Batoz and Dhatt, 1979; Wellford and Sen, 1981] with line search could also be adopted. In incremental procedure, operations are predetermined in an exact manner as a series of linear problems. Errors are likely to accumulate after several steps unless very fine steps are adopted. Therefore, the solution may diverge considerably from the true response. To prevent this, iteration could be used and assumed to have converged when the unbalanced load becomes acceptably small judged by the Euclidean norm. The modified Newton-Raphson method is the same as the regular Newton-Raphson method except for updating of stiffness matrix is performed after a given number of iterations. The conventional incremental procedure is a single iteration of its modified version wherein the unbalanced forces in the previous load increment are neglected. Always using conventional methods (e.g., nonlinear FEA) is not suitable for feasibility study as well as the preliminary stages of design. Although full nonlinear analysis gives the best representation of structural plasticity, obtaining solution may be difficult. The nonlinear process could lead to numerical convergence and undetectable errors and even to solution instabilities [Risks, 1979; Crisfield, 1981-84; Ramm 1981-82; Bathe and Dvorkin, 1983]. Thus, it requires the analysis be restarted with necessary modifications made to the geometry, applied loading conditions or the predefined convergence criteria [Bergan and Clough, 1972]. Besides, the accuracy of the solution obtained depends on the size of load

increments taken and the degree of non-linearity of the problem involved. Consequently, there is no guarantee of a numerical solution being reasonably accurate. In the absence of a method to easily verify the full numerical solution, sometimes it is difficult to judge the effectiveness of the solution.

Chapter 3

The Secant Method

3.1 Introduction

Knowledge of the behavior of the materials in the plastic range is essential in order to understand structural behavior and to have reliable estimate of the usefulness and the life of the structure. The nonlinear stress-strain relationship and the loading path dependency in the plastic range make the analysis tedious. Over the years, FEA has been successfully employed in analyzing the material behaviors in elastic and elasto-plastic range. The nonlinear FEA techniques can be broadly classified into Newton-Raphson based and Secant based methods. The first category is described in the previous Chapter. The secant methods can be further classified as direct secant methods or incremental secant methods. Amongst the direct methods are the 'robust' methods. Various robust methods have been developed to evaluate local inelastic effects, based on elastic modulus adjustment procedures [Neuber, 1961; Molski and Glinka, 1981 and Seshadri, 1991]. They are based on direct secant method for determining inelastic effects where linear elastic FEA is carried out for solving elasto-plastic problems using material parameters as field variables. Most of these methods currently in use are described below.

3.2 The Direct Secant Techniques

The slope obtained by joining any two points located on the uniaxial stress-strain curve is known as secant modulus. Usually in robust methods, one of the two points is taken to be the origin. Any method taking the secant line from the origin can be called as the direct secant method. Regular Newton-Raphson technique if applied without considering load increments results in a form of direct secant analysis. Several other 'incremental' methods can also be extended to obtain 'secant' results. Usually, this is not adopted in traditional nonlinear FEA since the convergence and other numerical difficulties are seen to be difficult to overcome. However, in a simplified method where a good estimate of the nonlinear strain at a critical point is all that is required, approximate but highly effective 'secant techniques' come in handy. They achieve the required objective very efficiently with considerably less labour. Besides this, these methods are 'robust' in the sense that they can absorb defects in data collection and are not very susceptible to minor changes in material and geometric parameters. Full nonlinear FEA in an 'incremental' manner is seen to be susceptible to such problems.

3.2.1 Basic Secant Analysis

Secant analysis can be carried out in a simple iterative manner using basic principles borrowing from classical (Newton based) secant methods for the solution of single nonlinear algebraic equations. The general methodology in secant analysis can be briefly described as below.

Carry out a linear elastic FEA for the given loading and geometry of structure. Use the results to compute equivalent stress and strain as per the applicable yield criteria (e.g., von Mises). If the loading is more than the yield load for the structure, at least some elements will exhibit equivalent stresses above the yield stress, e.g., point D in Fig. 3.1. The excess stress at point D is the result of assigning a larger stiffness to the corresponding element than should have been the case. This excess can be removed by assigning a more 'appropriate' stiffness to all such elements. Methods that attempt to reduce the stiffness by finding the 'tangent' to the load-deformation curve are described in Chapter 2. The appropriate stiffness can also be assigned in a 'secant' manner. 'Secant' approaches are well known in several branches of engineering including structural and soil mechanics. In the present context, a line joining the origin to the point representing the 'correct' state of stress and strain is considered to be the correct secant line. Adluri [2001] showed that the total strain energy represented by the total area under all such 'secant' lines for all the elements of the model will be exactly equal to the total work represented by a similar secant line on the load-deformation curve. Thus, for proportional loading, a single linear analysis using the correct secant stiffness will yield the correct state of stress and strain even if the material or geometry shows nonlinear properties. The aim of all the secant methods is to obtain this 'correct secant stiffness' as accurately as possible. In the basic secant analysis, an approximate secant stiffness is obtained by reducing the moduli of all the elements (such as those represented by point D in Fig. 3.1) with stresses above the yield limit. This new modulus is used to carry out a second analysis. The second analysis will indicate that

- a. the stress at the previously yielded elements is reduced (not necessarily to yield stress level), and
- b. the yield zone has expanded beyond that indicated by the first analysis.

Further improvements to the analysis can be made by iteratively adjusting the moduli. The simplest method of evaluating the new modulus after each analysis is to assume strain control. Assuming that the strain of the element remains the same while the stress drops to the yield surface

$$E_{i+1} = \left(\frac{\sigma_y}{\epsilon_e} \right) = \frac{\sigma_y}{\sigma_e} E_i \quad (3.2-1)$$

where, σ_e, ϵ_e are the equivalent stress and strain for iteration i ,

E_i, E_{i+1} are the Young's moduli for iteration i and $i+1$, and

σ_y is the yield stress.

The modulus can be adjusted by several other means. All such modulus adjustment techniques result in re-computing the full stiffness matrix after each analysis. Techniques based on this process are being called as 'robust' methods, examples of which can be found in the application of Neuber's rule, EGLOSS [Seshadri and Babu, 2000], etc.

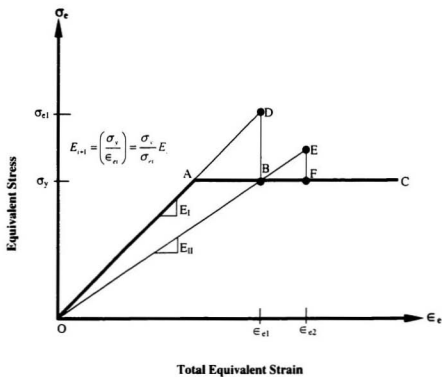


Fig. 3.1: Basic Secant Analysis

Instead of adjusting the modulus, the overall stiffness can be directly adjusted based on the results of the current analysis. This new stiffness matrix is used for the next iteration. Traditional nonlinear FEA theory produced several studies that adjust the stiffness matrix directly (Quasi-Newton, BFGS, etc.). These secant methods are incremental just like the 'tangent' techniques described in Chapter 2. Although they have been developed as 'incremental' techniques, they can easily be applied without the incremental option, i.e., each iteration starts by applying the entire load on the structure and the structure starts from zero deformation. These techniques are not classified as 'robust.' However, they present many efficient and mathematically sound ideas that can be used for improving the 'robust' techniques. Some of the representative methods of the two kinds are described below.

3.3 Robust Methods

In this section, different 'robust' methods for the determination of inelastic response and limit load estimation are described. Many such methods and their variations have been developed in recent years. All of them have the idea of elastic modulus adjustment scheme as common.

3.3.1 The Neuber's Rule

Strain concentration problems are often subjected to load that causes localized yielding. The resulting inelastic strains are of interest in determining the fatigue life of structures.

Neuber's rule has been traditionally used extensively for such inelastic strain estimations. It states that [Neuber, 1961], the geometric mean of stress and strain concentration factors during plastic deformation remains invariant and is equal to the elastic concentration factor. Using this approach, the strain in an inelastic body can be estimated using an elastic analysis. Let point D in Fig. 3.2 represent a yielded element as observed in an elastic analysis. It can be seen that the elastic modulus (and hence stiffness) needs to be modified to get a better estimate of the strain. This is achieved by projecting point D onto point H (on the uniaxial stress-strain curve) such that the elastic strain energy is conserved (i.e., the area ODB is the same as OHB' or OGDB is the same as OG'HB'). If an elastic-perfectly-plastic material is considered, the modulus of yielded elements is:

$$E_H = \frac{\sigma_i^2}{\sigma_{ei} \epsilon_{ei}} \quad (3.3-1)$$

This new effective modulus (represented by the slope of OH) is used to perform the next linear elastic analysis. This procedure can be iteratively repeated until all the effective material parameters converge and equivalent stress falls on the experimental uniaxial stress-strain curve.

As mentioned above, Neuber's rule is well known in Mechanical design. It can be profitably used to carry out inelastic analysis as well. The procedure described above is not strictly Neuber's rule but is an extension of it for nonlinear analysis.

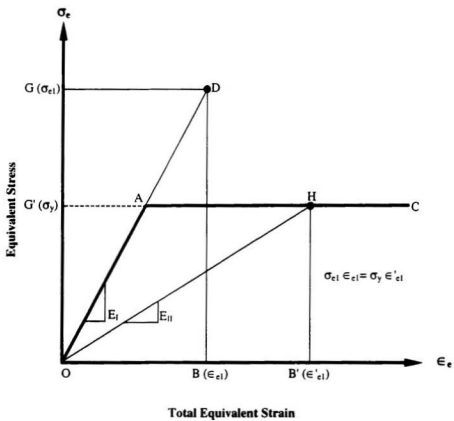


Fig. 3.2: Neuber's Rule - Iteration Procedure

3.3.2 Reduced Modulus Technique

A technique using a reduced modulus was developed by Jones and Dhalla, [1981] for stress categorization (particularly for piping stress). This procedure was extended to analyze the inelastic response and follow up characteristics of piping systems. The analysis involves progressively modifying the elastic modulus at each stage by performing repeated linear elastic analyses [Dhalla, 1984 and Severud, 1984]. Dhalla [1987] later directed his efforts toward developing a simple procedure of classifying stresses at highly loaded regions using linear elastic analysis.

This procedure is straightforward. In essence, it is the same as that described as 'basic secant technique' at the beginning of the present Chapter. Dhalla and co-workers applied it for stress classification in a detailed manner. An initial elastic analysis is performed to obtain the effective stress and strain at the highly loaded regions. The inelastic strain is then estimated roughly based on the calculated elastic stress. The minimum secant modulus is then calculated as the ratio of the elastic stress to the estimated inelastic strain defined as

$$E_s = \frac{\sigma_e}{\epsilon_p} \quad (3.3-2)$$

where, σ_e and ϵ_p are effective element stress and estimated inelastic strain, respectively.

A relaxation trend with repeated iterations is established by this procedure. Marriot [1988] proposed a reduced modulus method in a modified version for determining primary stresses in pressure vessel components and highlighted the possibility of determining limit load. The procedure identifies all elements having stresses above the code allowable stress. The elastic moduli of these elements are then reduced by using

$$E_r = \left(\frac{S_m}{SI} \right) E_0 \quad (3.3-3)$$

where, E_0 is the previous value of the modulus. S_m and SI are the code allowable stress and the element stress intensity, respectively.

A second analysis is carried out to evaluate a new stress distribution followed by a readjustment of the elastic moduli for critically stressed element. This procedure is repeated in an iterative manner until the maximum stress intensity is less than code allowable stress or some other selected convergence criteria. Reducing stresses in a structure so that the stresses are everywhere below the code allowable or yield stress of the material suggests that a statically admissible stress field exists. A lower bound limit load solution is one in which a statically admissible stress field exists in which the stress no where exceeds the yield stress of the material. Thus, the procedure of modulus reduction is one that yields a lower bound limit load, provided all stresses are everywhere below yield. If however, the procedure gives a converged solution in which the maximum stress is greater than the yield stress the applied load does not meet the lower bound limit load criterion. This method was later extended by Boyle and co-worker

[e.g., Mackenzie and Boyle, 1993] and Seshadri and co-worker [e.g., Seshadri and Fernando, 1992] to develop robust limit load estimation methods such as the elastic compensation method and the r-node method.

3.3.3 The MARS Method

In the Modulus Adjustment and Redistribution of Stress (MARS) method, the concept of Neuber's rule and the Generalized Local Stress Strain (GLOSS) method are used based on an iterative strategy combined with a modulus reduction technique [Babu and Iyer, 1998]. Statically admissible stress and kinematically admissible strain distributions are brought close to actual distributions. This ensures satisfaction of constitutive equation more closely.

For the first iteration, linear FEA is carried out for a given load, geometry and material properties to obtain point D defined by the equivalent stress σ_{e1} and equivalent total strain ϵ_{e1} (Fig. 3.3). The equilibrium point E is obtained by drawing a line whose slope is equal to the final equilibrium state E_{eqm} from point B. To evaluate E_{eqm} , an iterative strategy (which satisfies equilibrium conditions and yield condition alternately) combined with a modulus reduction technique is used.

At point D, equilibrium and compatibility conditions are satisfied, but constitutive equation is violated. For all points where equivalent stress exceeds the yield stress, it is brought back to yield level by scaling down the stress. Once the stress level is brought to yield level, equilibrium is lost.

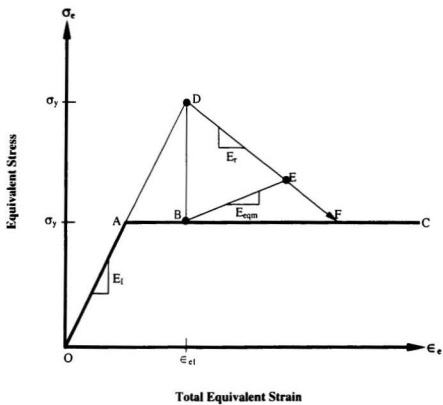


Fig. 3.3: MARS Method - Inelastic Response Estimation

However, constitutive and compatibility conditions are satisfied. Internal forces corresponding to this reduced stress level are calculated and compared with the given total load. The difference gives the unbalance force or residual force. In the next iteration, this residual force is applied as an external load.

The modulus of elasticity corresponding to those points, where the equivalent stress exceeds the yield stress before bringing it on to the yield surface, is reduced using modulus reduction technique based on Neuber's energy criterion.

$$E_i = \left\{ \frac{\sigma_y}{(S_{eq})_{i-1}} \right\}^2 E_i \quad (3.3-4)$$

where, i indicates the number of iterations and

$$E_i = \left\{ \frac{\sigma_y}{(\sigma_e)_{i-1}} \right\}^2 E_{i-1} \quad (3.3-5)$$

$$(S_{eq})_{i-1} = \frac{(\sigma_e)_{i-1} + (\sigma_e)_{i-2}}{2} \quad (3.3-6)$$

Modulus of elasticity at all other points is left unchanged. A linear elastic FEA with the residual force is carried out and the additional equivalent stress $(\Delta\sigma_e)_i$ is calculated.

The total equivalent stress at the end of the iteration is given by

$$(\sigma_e)_i = \sigma_y + (\Delta\sigma_e)_i \quad (3.3-7)$$

For all points with equivalent stress exceeding yield stress, the stress components are again linearly scaled down to the yield level and residual force is calculated. This residual force is applied again as an external load with the modulus of elasticity given by Eq. 3.3-4. The procedure is repeated until the convergence is achieved.

The modulus of elasticity corresponding to converged iteration is taken as the modulus of elasticity to equilibrium state E_{eqm} and used to locate E as shown in Fig. 3.3. The relaxation locus is obtained by joining points D and E by a straight line. The point of intersection F of relaxation locus and material stress-strain curve determines the final state of the structure.

3.3.4 The Extended GLOSS Method

It is a simple and systematic method for carrying out inelastic strain evaluations of pressure components and structures based on two linear elastic FEA. It is based in part on the reduced modulus method used by Dhalla. It was initially developed as the GLOSS method [Seshadri, 1991; Adinarayana and Seshadri, 1996]. In the GLOSS (Generalized Local Stress-Strain analysis) method, the structure or component to be analyzed is divided into two regions, viz., local region and the remainder. The local region is chosen as the highest stressed element and is determined on the basis of first linear elastic FEA. The largest inelastic effects are experienced in the local region and are often of interest from a design standpoint, while the remainder of the system typically operates at nominal levels of elastic stresses. The determination of the local region relaxation modulus as an

approximation to the relaxation locus is the key to GLOSS evaluations. The GLOSS diagram (Fig. 3.4) is essentially a plot of the equivalent stress and the total equivalent strain. For small plastic zone sizes, the relaxation locus is almost linear. Therefore, the local region relaxation modulus is determined assuming a linear relaxation locus. Thus local region response appears to be insensitive to the precise nature of the local region nonlinearly and the material constitutive relationship. The implication is quite significant in that the relaxation modulus can be determined using any convenient constitutive relationship that allows progressive relative softening of the local region. Therefore, GLOSS analysis can be used to predict inelastic response whether the inelasticity arises due to first stage creep, steady state creep, or even time-dependent plasticity. Local region softening by systematically reducing the elastic modulus, for instance, is therefore an attractive prospect.

In the Extended Generalized Local Stress Strain (EGLOSS) analysis, an initial FEA is carried out assuming that the entire structure is linearly elastic. Seshadri and Babu [2000] argued that for every individual element in which the equivalent stress exceeds the yield stress, pseudo-equilibrium can be postulated as

$$(\sigma_e - \sigma_y)_k A_k = (\sigma_y - \sigma_y^*)_k A_k \quad (3.3-8)$$

where, modified yield stress is $\sigma_y^* = 2\sigma_y - \sigma_e$ and A_k indicates representative area over which the equivalent stress σ_e acts.

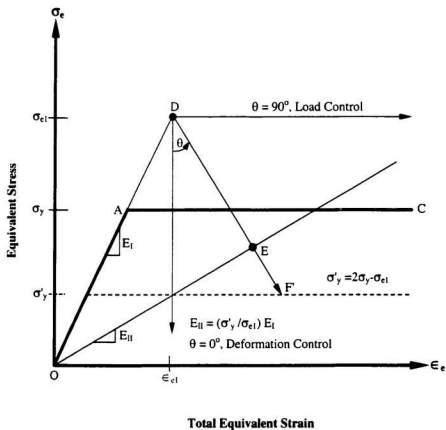


Fig. 3.4: EGLOSS Diagram - Inelastic Strain Prediction

Using this idea, the approximate secant moduli of all elements beyond yield stress (e.g., point D in Fig. 3.4) are estimated as

$$E_n = \left(\frac{\sigma_i}{\sigma_e} \right) E_t \quad (3.3-9)$$

A second linear analysis is carried out. The Poisson's ratio is initially left unchanged. The results of the first and second analyses (points D and E in Fig. 3.4) are joined to obtain an approximate relaxation locus.

The intersection of this locus and the modified yield curve (point F) is obtained. This point would satisfy the strain-displacement relationship, since displacements have been stipulated to be continuous and also satisfy the stress-strain relationship. The point F' gives the desired strain that we are looking for, despite the fact that the equilibrium requirement is not fully satisfied.

3.4 Traditional Incremental Secant Methods

As outlined earlier, there are several studies in traditional nonlinear FEA that use secant stiffness in carrying out incremental nonlinear analysis. All these can be reduced to 'direct' secant methods by suppressing the incremental option. Many of these are considered to be theoretically very sound. However, they are generally believed to converge somewhat more slowly than their 'tangent' counterparts described in the previous Chapter. They nevertheless are implemented in several major software

packages because of their numerical stability and other useful characteristics. They contain many useful techniques that can be used with other procedures. For example, the highly useful 'line search' has been first developed in conjunction with secant methods but later adopted by 'tangent' methods. Some of the secant methods of this kind (including the path following methods) are discussed briefly below.

3.4.1 The Arc-Length Method

The arc-length methods are intended to enable solution algorithms to pass limit points. Those methods were originally introduced by Riks and Wempner [Crisfield, 1991] with later modifications being made by several authors. They are suitable for nonlinear static equilibrium solutions of unstable problems. Applications of the arc-length method involve the tracing of a complex path in the load-displacement response into the buckling or post-buckling regime.

Ramm [1982] has shown that the constant arc-length method of Riks/Wempner seems to be the most versatile technique, being advantageous in the entire load range. The arc-length method uses explicit spherical iterations to maintain the orthogonality between the arc-length radius and the orthogonal direction. It is assumed that all load magnitudes are controlled by a single scalar parameter (i.e., the total load factor). Unsmooth or discontinuous load-displacement response in the cases often seen in contact analyses and elastic-perfectly plastic analyses can not be traced effectively by the arc-length procedure. Mathematically, the arc-length method can be viewed as the trace of a single

equilibrium curve in a space spanned by the displacement variables and the total load factor [ANSYS, 1998]. During the solution, the arc-length method will vary the arc-length radius at each arc-length substep according to the degree of non-linearity that is involved.

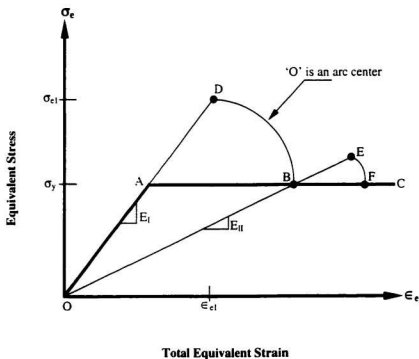


Fig. 3.5: Arc-Length Method - Iteration Process

In this method, the yielded point D as shown in Fig. 3.5 is projected onto the uniaxial stress-strain curve considering OD as the arc radius and O as the center. The intersecting point B on the experimental uniaxial stress-strain curve is used to obtain the new effective modulus for the next iteration.

3.5 Incremental (or Path Following) Secant Techniques

Most nonlinear solution algorithms of the traditional kind (including the secant based ones) use the path following or incremental techniques. Only at the beginning of the solution, the origin is used. After that, the origin is shifted to another point evaluated after first iteration. In all subsequent iterations, the point evaluated in the previous iteration is considered as the origin. The origin is always shifted and moved following the path influenced by the loading and the experimental uniaxial stress-strain curve. Some of the techniques of this kind are described below.

3.5.1 The BFGS Method

It is a generalization of one-dimensional secant method to multi-dimensional problems to solve nonlinear simultaneous equations. In secant methods, an approximation to the tangent matrix ' K ' is used at each iteration. In the Quasi-Newton (QN) methods, a simple updating is deduced to compute a new secant matrix from the previous one [Dennis and More, 1977]. The QN methods are basically a compromise between the full reformation of the stiffness matrix performed in the Full Newton-Raphson (FNR) method

and the use of a stiffness matrix from a previous configuration as is done in the Modified Newton-Raphson (MNR) method. Among the QN methods, the BFGS method appears to be the most effective. The method has been developed based on the contributions of Broyden, Fletcher, Goldfarb and Shanno and has been summarized by several authors, e.g., Matthies and Strang [1979].

In this method, for each load increment, the stiffness matrix is formed using the results of the previous load step by updating the stiffness matrix in a 'secant' manner. Let K_{i-1} , R_{i-1} and F_{i-1} be the stiffness matrix, the applied external load and the resistance of force (by the structure) in the iteration $i-1$, respectively. Then a displacement vector increment (Δu_i) is evaluate as

$$\Delta u_i = K_{i-1}^{-1} \Delta R_{i-1} \quad (3.5-1)$$

where, $\Delta R_{i-1} = R_{i-1} - F_{i-1}$ indicates the residual or unbalanced loads. A line search in the direction Δu_i at iteration i satisfying equilibrium is performed. Using this line search the displacement vector u_i at iteration i is determined as follows:

$$u_i = u_{i-1} + \beta \Delta u_i \quad (3.5-2)$$

where, β is a scalar multiplier. Calculation of $\Delta R_i = R_i - F_i$ gives the unbalanced loads corresponding to these displacements. The parameter β is varied until the component of

the unbalanced loads as defined by the inner product $\Delta u^T \Delta R_i$ is approximately zero. In other words

$$\Delta u^T \Delta R_i \leq Tol \Delta u^T \Delta R_{i-1} \quad (3.5-3)$$

where, Tol is the convergence tolerance.

It is noted that the initial displacement vector u_{i-1} at iteration $i-1$, for a given load vector increment R_{i-1} is obtained by

$$u_{i-1} = K_{i-1}^{-1} R_{i-1} \quad (3.5-4)$$

The updated matrix is evaluated depending on the new displacements u_i and expressed in product form of

$$K_i^{-1} = B_i^T K_{i-1}^{-1} B_i \quad (3.5-5)$$

where, $B_i = I + p_i q_i^T$, and the updated matrix (K_i) must satisfy the Quasi-Newton (QN) relation defined as $K_i \delta_i = r_i$.

The vectors p_i and q_i are calculated from the known displacements and forces (that are equivalent to element stresses).

$$p_i = - \left[\frac{\delta_i^T r_i}{\delta_i^T K_{i-1} \delta_i} \right]^{-1/2} K_{i-1} \delta_i - r_i \quad (3.5-6)$$

$$q_i = \frac{\delta_i}{\delta_i^T r_i} \quad (3.5-7)$$

where, $\delta_i = u_i - u_{i-1}$ and $r_i = \Delta R_{i-1} - \Delta R_i$ are the displacement changes and the unbalanced load difference between two successive iterations (for a given load increment), respectively.

The product defined in Eq. 3.5-6 is positive definite and symmetric. To avoid numerically dangerous updates, the condition number C_n defined in Eq. 3.5-6 of the updating matrix B_i will be compared with some prescribed tolerance limit. The updating is not performed if the condition number exceeds the prescribed tolerance.

$$C_n = \left[\frac{\delta_i^T r_i}{\delta_i^T K_{i-1} \delta_i} \right]^{1/2} \quad (3.5-8)$$

This method and its variations are quite regular. Computational costs for the solution of large nonlinear system of equations can be reduced drastically by using convenient Quasi-Newton updates or by adequate combined Newton/Quasi-Newton strategies [Geradin, et al., 1981]. They are implemented in several software packages. For example, ANSYS uses a method similar to this based on the work of Schweizerhof and Wriggers, [1986]. ABAQUS FE software uses these techniques (Quasi-Newton methods, e.g., BFGS method) as well.

3.5.2 The SN Method

An improved technique for the solution of implicit equations is the Newton Secant method (SN) which uses a numerical approximation of the function derivatives [Garcia, 1998]. This method simplifies the solution procedure, by addressing only two equations. The first equation is the implicit function and the second equation represents the numerical approximation of the derivative. The traditional Newton-Raphson (NR) method takes a seed value of u (the unknown being solved) and finds a derivative of the implicit function evaluated at u . A new value u_{i+1} is found by correcting the initial value u_i by the ratio of the function to the function derivative. The iterative process for structural equilibrium equations in the traditional NR method is as follows:

$$u_{i+1} = u_i - \frac{f(u_i)}{f'(u_i)} \quad (3.5-9)$$

The iterative process is continued until the absolute value of the function is within a specified error limit. It requires the computation of the function derivative with respect to the unknown being solved. Each different unknown requires the formulation of a new derivative. The secant modification as per the Secant Newton method consists of replacing the derivatives by a different quotient such as

$$f'(u_i) = \frac{f(u_i) - f(u_{i-1})}{u_i - u_{i-1}} \quad (3.5-10)$$

where, u_{i-1} and $f(u_{i-1})$ are the previous values of the unknown displacements and implicit function, respectively.

The SN method is a more simplified iterative procedure where the function derivative is numerically approximated. In this manner, only two functions are evaluated regardless of the unknown. However, care must be exercised in supplying the correct seed value for the iterations.

3.5.3 The Modified SN Method

In the Modified SN (MSN) method of Zhang and Owen [1982], the iterative displacement change involves only a scalar multiple. Both the MSN and SN methods are based on a secant approach but the MSN method leads to a reduction in the required number of iterations compared to the SN method. In the SN iteration scheme, the iterative deflection change is a scalar times the previous iterative change plus a further scalar times the usual unaccelerated change. These scalars are automatically recalculated at each iteration. These are related to inner products involving the iterative deflection and the present and past out of the balance force vectors.

Before introducing the MSN method, the complete iteration process for the SN method is summarized by the following expressions.

$$\Delta u_i = e_i \Delta u_{i-1} + h_i \Delta \bar{u}_i \quad (3.5-11)$$

$$\Delta \bar{u}_i = K_0^{-1} \Delta R_i \quad (3.5-12)$$

$$u_{i+1} = u_i + \Delta u_i \quad (3.5-13)$$

$$h_i = -\frac{d_i}{b_i}, \quad e_i = h_i \left(1 - \frac{c_i}{b_i} \right) - 1 \quad (3.5-14)$$

$$b_i = u_{i-1}^T r_i, \quad c_i = \Delta \bar{u}_i^T r_i, \quad d_i = \Delta u_{i-1}^T \Delta R_{i-1} \quad (3.5-15)$$

where, u_i , u_{i+1} are the displacements on the i^{th} and $(i+1)^{\text{th}}$ iterations and Δu_i , Δu_{i-1} are displacement changes on the i^{th} and $(i-1)^{\text{th}}$ iterations. $\Delta \bar{u}_i$ is the usual unaccelerated change. K_0^{-1} , ΔR_i and ΔR_{i-1} are the stiffness matrix at the beginning of the increment, residual forces on the i^{th} and $(i-1)^{\text{th}}$ iterations, respectively. $r_i = \Delta R_i - \Delta R_{i-1}$ gives the difference between the two residual or unbalanced forces.

In the Modified Secant Newton (MSN) method, Eq. 3.5-11 is replaced by the following expressions.

$$\Delta u_i = \bar{h}_i \Delta \bar{u}_i \quad (3.5-16)$$

$$\bar{h}_i = (h_i - 1) \frac{b_i}{c_i} \quad (3.5-17)$$

It is evident that iterative deflection change is now only a scalar times the usual unaccelerated change. Zhang and Owen [1982] compared the SN and MSN methods and

concluded that the latter one is more economic with regard to computational time, due to the reduced number of iterations. It is noted that the MSN method can also be derived from the variable metric method.

3.5.4 Crisfield's MSN Method

Crisfield [1980, 1981] has argued that line search with the MNR method is a simple and effective method for increasing the robustness of the iterative technique. The line search is an integral part of the solution method. With the line search performed within an iteration, the expense of iteration increases but fewer iterations may be need for convergence. Also the line search may prevent the divergence of the iterations and in practice this increased robustness is the major reason why a line search can in general be effective.

While the SN method can be derived [Crisfield, 1980-81] from any QN techniques, Crisfield has shown that it is also possible to update inverse stiffness matrix K_i^{-1} from the previous inverse stiffness matrix K_{i-1}^{-1} using the BFGS procedure of Broyden [1970], Fletcher (1970) and Wolfe (1975), and given by

$$K_i^{-1} = K_{i-1}^{-1} - \frac{\Delta u_{i-1}^T r_i K_{i-1}^{-1}}{\Delta u_{i-1}^T r_i} - \frac{K_{i-1}^{-1} r_i \Delta u_{i-1}^T}{\Delta u_{i-1}^T r_i} + \left(1 + \frac{r_i^T K_{i-1}^{-1} r_i}{\beta_{i-1} \Delta u_{i-1}^T r_i} \right) \frac{\beta_{i-1} \Delta u_{i-1} \Delta u_{i-1}^T}{\Delta u_{i-1}^T r_i} \quad (3.5-18)$$

Crisfield's SN method [1980] can be considered as single-cycle restarted versions of these up-dates and the procedure to be taken is as follows:

$$\Delta \bar{u}_i = -K_{i-1}^{-1} \Delta R_i \quad (3.5-19)$$

$$\Delta \bar{u}_{i-1} = -K_{i-1}^{-1} \Delta R_{i-1} \quad (3.5-20)$$

$$\Delta u_i = -K_i^{-1} \Delta R_i = A_i \Delta \bar{u}_i + B_i \beta_{i-1} + C_i \Delta \bar{u}_{i-1} \quad (3.5-21)$$

where,

$$C_i = \frac{\Delta u_{i-1}^T \Delta R_i}{\Delta u_{i-1}^T r_i} \quad (3.5-22)$$

$$A_i = I - C_i = \frac{-\Delta u_{i-1}^T \Delta R_{i-1}}{\Delta u_{i-1}^T r_i} \quad (3.5-23)$$

$$B_i = -C_i - \frac{(\Delta \bar{u}_i - \Delta \bar{u}_{i-1})^T \Delta R_i}{\beta_{i-1} \Delta u_{i-1}^T r_i} + C_i \frac{(\Delta \bar{u}_i - \Delta \bar{u}_{i-1})^T r_i}{\beta_{i-1} \Delta u_{i-1}^T r_i} \quad (3.5-24)$$

At the beginning of cycle, $K_{i-1} = K_c$ where, K_c is an approximate stiffness matrix, which could be the exact matrix at the beginning of the increment and

$$\Delta \bar{u}_i = -K_c \Delta R_i \quad (3.5-25)$$

$$\Delta \bar{u}_{i-1} = -K_c \Delta R_{i-1} \quad (3.5-26)$$

Eq. 3.5-21 defines a three-parameter acceleration procedure. This method is derived from the BFGS technique and implies a stiffness matrix K_i update from K_c which satisfies the secant relationships. Crisfield [1979] called Eq. 3.5-22, SN-2 and derived a simpler two-parameter acceleration (SN-3) for which

$$\Delta u_i = A_i \Delta \bar{u}_i + D_i \beta_{i-1} \Delta u_{i-1} \quad (3.5-27)$$

where,

$$D_i = -C_i - A_i \left(\frac{\Delta \bar{u}_i^T r_i}{\beta_{i-1} \Delta u_{i-1}^T r_i} \right) = A_i \left(1 - \frac{\Delta \bar{u}_i^T r_i}{\beta_{i-1} \Delta u_{i-1}^T r_i} \right) - 1 \quad (3.5-28)$$

By making the approximation

$$\Delta \bar{u}_{i-1} = -K_c^{-1} \Delta R_{i-1} = \beta_{i-1} \Delta u_{i-1} \quad (3.5-29)$$

No approximation is therefore involved if the previous iteration is a standard unaccelerated MNR iteration. However, numerical experience has shown that good solutions can be obtained by applying Eqs. 3.5-27 and 3.5-28. As the coefficient A_i of the dominant $\Delta \bar{u}_i$ is the same for both the two-parameter and the three-parameter accelerators, a further approximation whereby the Δu_{i-1} term in Eq. 3.5-27 is ignored and the acceleration becomes simply

$$\Delta u_i = A_i \Delta \bar{u}_i \quad (3.5-30)$$

This procedure therefore, involves the application of the dominant term in a restarted single-cycle BFGS technique. However it is not called a SN method since it does not degenerate to the standard secant method for one-dimensional problems.

3.5.5 Barnes's Secant Method

The algorithm given by Barnes [1965] has the advantage of not requiring the explicit evaluation of derivatives. This uses instead an approximate value of the Jacobian and corrects this after each function evaluation and is equivalent to the generalized secant method described by Wolfe [1959] with additional advantage of being able to make use of an initial approximation to the Jacobian. The benefit of this last fact is significant since the situation often arises in practice where the same set of equations are to be solved several times with slightly different values of certain parameters. The final solution point and the Jacobian often provide excellent initial conditions for the next and under these circumstances the method may prove to be many times faster than the NR method. Both theoretical and experimental results have shown that this method is in general about twice as good as NR procedure in the neighborhood of a solution [Barnes, 1965].

Assume J_1 and u_1 are an initial guess of the Jacobian and an initial point, respectively, where the function value is f_1 , then the first step for Δu_1 is defined by

$$J_1 \Delta u_1 = -f_1 \quad (3.5-31)$$

Here, if the assumption J_1 were correct, it would give the Newton-Raphson (NR) step and rise to the point $u_2 = u_1 + \Delta u_1$ at which the function value is f_2 .

The correction to be applied to the Jacobian J_1 is determined by considering the behavior of a linear system which has values f_1, f_2 at u_1, u_2 , respectively. For Jacobian J of a system, it can be written as

$$f_2 = f_1 + J\Delta u_1 \quad (3.5-31)$$

The corrected Jacobian J_2 is chosen to satisfy above the equation where d_1 is a correction so that $J_2 = J_1 + d_1$. Then d_1 satisfies $f_2 = f_1 + (J_1 + d_1)\Delta u_1$ and the above equation becomes

$$f_2 = d_1\Delta u_1 \quad (3.5-32)$$

A solution of this where z_1 is an arbitrary vector is

$$d_1 = \frac{f_2 z_1^T}{z_1^T \Delta u_1} \quad (3.5-33)$$

A general iteration process where the vectors z_i are as yet undefined is thus

$$\begin{aligned}
\Delta u_i &= -J_i^{-1} f_i \\
u_{i+1} &= u_i + \Delta u_i \\
d_i &= \frac{f_{i+1} z_i^T}{z_i^T \Delta u_i} \\
J_{i+1} &= J_i + d_i
\end{aligned} \tag{3.5-34}$$

where, $d_i \Delta u_i = f_{i+1}$ and $J_{i+1} \Delta u_i = f_{i+1} - f_i$.

A desirable feature of any method of solving nonlinear equations is to rapidly solve a linear set of equations. In fact, function evaluation $n+1$ should suffice to determine the Jacobian of the system exactly and hence the solution ought to be found on the function evaluation $n+2$. This means that u_{n+2} ought to be the solution. The magnitude of z_i is irrelevant. For convenience it is taken to be unit vector and the above computation is then readily referred in either case by the well known Gram-Schmidt orthogonalization process.

The important consequence of the choice of z_i is that

$$d_i \Delta u_j = 0 \quad 1 \leq i - j < n \tag{3.5-35}$$

Therefore,

$$\begin{aligned}
J_{i+1} \Delta u_j &= (J_{j+1} + d_{j+1} + \dots + d_i) \Delta u_j \\
&= J_{j+1} \Delta u_j \quad 1 \leq i - j < n
\end{aligned} \tag{3.5-36}$$

Considering the behavior with the general set of nonlinear equations (i.e., defined by $f_i(u_j) = 0$) the above equation is expressed as

$$\begin{aligned} J_k \Delta u_{k-i} &= J_{k-i+1} \Delta u_{k-i} \\ &= f_{k-i+1} - f_{k-i} \\ &= \Delta f_{k-i} \quad i < k, \quad 0 < i \leq n \end{aligned} \quad (3.5-37)$$

In particular if $k \geq n$

$$J_k (\Delta u_{k-n}, \Delta u_{k-n+1}, \dots, \Delta u_{k-1}) = (\Delta f_{k-n}, \Delta f_{k-n+1}, \dots, \Delta f_{k-1}) \quad (3.5-38)$$

The value of Jacobian J_k is therefore that of the linear system defined by the $n+1$ pairs of points and function values, $u_{k-n}, f_{k-n}, \dots, u_k, f_k$. This method is therefore identified with the generalized secant method. As mentioned earlier, the present representation of secant method however has the advantages of being able to use an initial value of J and in practice has been found to be more reliable.

3.5.6 Chen's Method

A consistent approach for carrying out the iterative computation of nonlinear FE problems has been suggested [Chen, 1990, 1992]. In this computation procedure, the direction of incremental response vector is defined by the constant stiffness prediction. Then a relaxation parameter obtained by minimizing certain error quantity is adopted for defining an improved incremental response vector in iteratively updating the total response vector. The secant formulation is adopted for constructing the error quantity by

introducing an implicit secant stiffness matrix. This approach is considered to be consistent with the geometrical sense inherent in the relaxation process. The resulting algorithm requires only a small amount of post data for carrying out the relaxation process. The relaxation parameters are simply defined by the inner products of a certain combination involving incremental response vector obtained by the constant stiffness prediction and residual force vectors. Moreover, it is easy to implement this computation procedure into a FEA code. The computational procedure can be applied to the collapse analysis of a structure. This procedure can accomplish the computation task economically, while maintaining good numerical stability, though a large load step is adopted for this analysis.

As described in Chapter 2, the Full Newton-Raphson iteration is defined in different notations (where, K_n^i = global tangent stiffness matrix, Δu_i^{i+1} = incremental displacement vector and R_i^i = global residual force vector at iteration step i) as

$$\begin{aligned} K_n^i \Delta u_i^{i+1} &= R_i^i \\ u_i^{i+1} &= u_i^i + \Delta u_i^{i+1} \end{aligned} \quad (3.5-39)$$

To improve the performance of the elasto-plastic analysis, a relaxation technique is adopted for replacing the above iteration procedures. The constant stiffness matrix, denoted as K_n^0 formed at the beginning of the iteration process is used to predict the

direction of displacement increments for all subsequent iteration steps. Letting β^{i+1} denote the accelerating parameter, then this procedure can be as follows:

$$\begin{aligned} K_n^0 \Delta u_i^{i+1} &= R_r^i \\ u_i^{i+1} &= u_i^i + \beta^{i+1} \Delta u_i^{i+1} \end{aligned} \quad (3.5-40)$$

A series of iteration schemes using different accelerating parameters has been proposed. Some of these methods adopted a dynamic way in which different accelerating parameters are used for scaling different degrees of freedom in the global discrete system to improve the convergence. Taking β^{i+1} equal to 1, the relaxation algorithm stated by above equation lead to the Modified Newton-Raphson (MNR) scheme.

In this approach, the data obtained from the MNR method are used for constructing a secant relation. Two different error quantities defined by adopting the secant relation are used to determine two accelerating parameters, which lead to two iteration algorithms. Let \bar{R}_r^i and \bar{K}_n^i denote, respectively, the residual force vector after the MNR prediction and the stiffness matrix that establishes the secant relation for the period with incremental displacement vector Δu_i^{i+1} . Then \bar{R}_r^i is related to R_r^i through the following equation

$$\bar{R}_r^i = R_r^i \pm \bar{K}_n^i \Delta u_i^{i+1} \quad (3.5-41)$$

Selecting the scaled incremental displacement vector $\beta^{i+1} \Delta u_i^{i+1}$ as the true response increment, the residual vector at the end of the current iteration step can be obtained by

$$R_e^{i+1} = R_e^i \pm \beta^{i+1} \bar{K}_n^i \Delta u_e^{i+1} \quad (3.5-42)$$

In the first algorithm to be derived, a physical parameter defined as the inner product of R_e^{i+1} and the displacement increment vector caused by R_e^{i+1} along the linear deformation path predicted by \bar{K}_n^i is selected to be the error quantity. Letting E_e^{i+1} denote this error quantity, the following expression can be defined

$$E_e^{i+1} = R_e^{i+1} (\bar{K}_n^i)^{-1} R_e^{i+1} \quad (3.5-43)$$

Introducing Eqs. 3.5-42 and 3.3-43, it shows that E_e^{i+1} is related to β^{i+1} stated by Eq. 3.5-44

$$E_e^{i+1} = (\beta^{i+1})^2 \bar{K}_n^i \Delta u_e^{i+1} \Delta u_e^{i+1} \pm 2\beta^{i+1} \Delta u_e^{i+1} R_e^i + (\bar{K}_n^i)^{-1} R_e^i R_e^i \quad (3.5-44)$$

Minimizing E_e^{i+1} with respect to β^{i+1} and using β_e^{i+1} to replace β^{i+1} , the accelerating parameter β_e^{i+1} is defined as follows:

$$\beta_e^{i+1} = \left| \frac{\Delta u_e^{i+1} R_e^i}{\Delta u_e^{i+1} (R_e^i - \bar{R}_e^i)} \right| \quad (3.5-45)$$

In deriving the second iteration algorithm, the self-inner product of R_e^{i+1} is selected to be the error quantity. Letting E_f^{i+1} denote this error quantity, then E_f^{i+1} can be expressed as

$$E_f^{i+1} = R_e^{i+1} R_e^{i+1} \quad (3.5-46)$$

Introducing Eqs. 3.5-41 and 3.5-42 into the above, E_f^{i+1} appears to be a quadratic form of β^{i+1} stated by the following equation

$$E_f^{i+1} = (\beta^{i+1})^2 (R_i' - \bar{R}_i')(R_i' - \bar{R}_i') \pm 2\beta^{i+1} R_i'(R_i' - \bar{R}_i') + R_i' R_i' \quad (3.5-47)$$

Minimizing E_f^{i+1} with respect to β^{i+1} and using β_f^{i+1} to replace β^{i+1} , the second acceleration parameter can be found to have the form of

$$\beta_f^{i+1} = \left| \frac{R_i'(R_i' - \bar{R}_i')}{(R_i' - \bar{R}_i')(R_i' - \bar{R}_i')} \right| \quad (3.5-48)$$

Eqs. 3.5-45 and 3.5-48 represent the two accelerating parameters that are derived for improving the convergence rate and numerical stability of the MNR iteration. It is noted that the essence inherent in these two algorithms is the introduction of a secant relation.

3.5.7 Tangent-Secant Approach

A tangent-secant technique for nonlinear FE equations in small elasto-plastic structural problems is developed by Alfano, et al. [1999]. Its peculiar feature lies in the choice of the most suitable consecutive operator to be adopted at each iteration of a generic load step. It ensures the utmost stability and convergence rate. Namely, the consistent tangent operator is replaced by a secant one or vice versa, whether the adopted form of the residual does not, or does conveniently decrease at the current iteration. The secant operator is defined as to recover the finite step increment of the plastically admissible

stress from the total, not iterative, strain increment. The original formulation of the original solution procedure consists of alternative tangent and secant iterations. It can extend to achieve an effective coupling with line searches as well.

The displacement-based FEA of elasto-plastic structural models relies upon two main ingredients: the numerical integration of the rate constitutive equations over a generic time step and the iterative algorithm exploited for solving the nonlinear equilibrium equations. To enhance the overall efficiency of the solution procedure, a greater attention has to be devoted to the iterative algorithm exploited for solving the nonlinear equilibrium equations since the numerical integration of the rate constitutive equations over a generic time step is currently carried out by fairly standard methods. This is the outcome of the extensive attention, which has been devoted, in the last 15 years.

On the contrary, the choice of most economical iterative scheme for the solution of nonlinear FE equations is still the argument of an open debate, as it is usually problem dependent. It is well known that NR method is conditionally convergent and as a general rule, the convergence rate decreases as the degree of stability increases. The MNR is extremely robust but requires a large number of iterations to ensure convergence. The method is very economical since the Jacobian matrix can be assembled and factorized once per load increment but the rate of convergence is so poor that usually offsets any other computational advantages. On the contrary, the FNR provides the highest rate of convergence (i.e., asymptotically quadratic) among the iterative methods currently employed to solve nonlinear sets of equations [Luenberger, 1984]. Stability is however

rather critical since small load steps are required to ensure convergence. Further the Jacobian matrix needs to be computed and factorized afresh at each iteration thus considerably increasing the numerical cost of the whole strategy.

Looking for more stable solution strategies, attention was naturally drawn to the so-called Quasi-Newton or secant approaches, which exhibit intermediate performance between the MNR and FNR methods. The Quasi-Newton methods are usually very economical since the Jacobian does not need to be inverted at each iteration. Rather the Jacobian inverse is periodically updated by rank one or rank two (BFP, BFGS) corrections [Matthies and Strang, 1979]. The convergence rate of Quasi-Newton methods is only linear but the stability properties are significantly greater than the ones characterizing a NR approach. The previous considerations make one reasonably feel that a solution strategy encompassing both the high convergence rate of FNR method and the stability of the secant one can be very effective. However as they stand, Quasi-Newton methods don't seem to be well suited to a simple straightforward merging with the traditional implementations of the NR method, although some proposals in this sense can be found [Geradin, et al., 1981]. This led Alfano, et al. to search for alternative formulations of the secant method, a goal already pursued [Martin and Bird, 1986] from a different perspective. Thus, the expression of a symmetric secant operator which provides the total, not iterative increment in the step of the plasticity admissible stress associates with a given strain increment is derived. Hence the method described by Alfano, et al. [1997] belongs to the class of the so-called Picard or direct procedures. Actually the structural stiffness operator associated with secant elasto-plastic operator establishes an explicit

nonlinear relation between the total displacement increment in the step and the assigned load increment. The stability properties of secant method exploited turned out to be excellent and in some cases it was possible to assign load steps several times greater than the ones that make the NR method converge. However, as expected the number of iterations required at each load step was comparable with the one pertaining to a MNR method and hence unacceptable for large-scale computations.

The main objective is to present a general and robust solution procedure for small elasto-plastic structural problems, which can encompass both the high convergence rate of the tangent approach and the remarkable stability properties of the secant one. To this end, the secant procedure originally formulated in terms of the total increment of the displacements in the step is conveniently transformed so as to assume the iterative displacement increments as the primary unknowns. This greatly facilitates the implementations of the tangent-secant strategy since just a logical switch needs to be added to the routine in which the constitutive operator is evaluated at the element level. In implementation, the most convenient choice between the consistent tangent and the secant operator to be made at each structural iteration is assumed to depend upon the energy norm of the residual. Specially, the tangent operator is tried to use so as to speed up the calculations by switching to the secant one at those iterations in which the energy norm does not reduce according to a user defined ratio, with respect to the least value achieved at the previous iterations. Only subsequently, when the energy norm has conveniently decreased, the converse switch from the secant to the tangent operator is performed. The numerical performances of the tangent-secant strategy turned out to be

comparable with the ones achievable with a pure tangent approach supplemented by line searches. This prompted to find out the most convenient way of coupling this classical technique [Crisfield, 1991] with the proposed one.

The basic idea underlying this strategy is different from the tangent methods and belongs to the class of direct or Picard procedures. A solution of the nonlinear system is iteratively sought for by defining a secant, which associates the total rather than iterative displacement increments in the step. The values of the state variables at the beginning of the step satisfy the equilibrium equation:

$$R_0 - \int_V B^T E (B u_0 - p_0) dV = 0 \quad (3.5-49)$$

The structural equilibrium equation is expressed in terms of the numerical vector u of displacement parameters through the residual $\Delta R(u)$ defined by

$$\Delta R(u) = R - \int_V B^T E (B u - p) dV \quad (3.5-50)$$

where, R and $\Delta R(u)$ are the applied and unbalanced loads, respectively. The difference between applied loads and the forces associated with the internal stress $\sigma = E(Bu - p)$ gives the unbalanced loads. B and p are the strain-displacement operator and nonlinear function of displacement u , respectively. The integration is extended to the domain V of the structural model.

The combination leads to solving the nonlinear system in the unknown Δu .

$$\Delta R(\Delta u) = \Delta R_e - \int_V B^T E (B \Delta u - \Delta p) dV = 0 \quad (3.5-51)$$

It is noted that ΔR_e is an applied load increment. Defining a linear secant operator E_s , where, $E_s \Delta \epsilon = E(\Delta \epsilon - \Delta p)$, the previous system is recast in the equivalent form of

$$\Delta R(\Delta u) = \Delta R_e - \int_V B^T E_s B \Delta u dV = 0 \quad (3.5-52)$$

Accordingly, when applied to the given total strain increment $\Delta \epsilon = B \Delta u$, the secant operator provides the associated stress increment in the step. A solution of the nonlinear system is then iteratively sought for by solving the linear system of the equations,

$$\Delta R_e - \left\{ \int_V B^T E_s(\Delta u_i) B dV \right\} \Delta u_{i+1} = 0 \quad (3.5-53)$$

Setting

$$K_{s,i} = K_s(\Delta u_i) = \int_V B^T E_s(\Delta u_i) B dV \quad (3.5-54)$$

Eq. 3.5-53 becomes

$$\Delta R_e - K_{s,i} \Delta u_{i+1} = 0 \quad (3.5-55)$$

The previous linear system of equations expressed in terms of the trial values $\Delta u_i = u_i - u_0$ of the total displacement increment in the step, can be reformulated in terms of the iterative increments δu_{i+1} by letting $\delta u_{i+1} = \Delta u_{i+1} - \Delta u_i$ so that Eq. 3.5-55 becomes

$$K_{i+1} \delta u_{i+1} = \Delta R_i - K_{i+1} \Delta u_i \quad (3.5-56)$$

or,

$$\begin{aligned} \Delta R_i - K_{i+1} \Delta u_i &= R - R_0 - \int_V B^T E [(B u_i - u_0) - (p_i - p_0)] dV \\ &= R - \int_V B^T E (B u_i - p_i) dV \\ &= \Delta R(u_i) \end{aligned} \quad (3.5-57)$$

Therefore, finally

$$K_{i+1} \delta u_{i+1} = \Delta R_i \quad (3.5-58)$$

Different implementations of line search techniques are possible to accelerate the convergence rate. It has been shown that the tangent-secant approach and its coupling with line searches provide valuable tools to analyze elasto-plastic structural problems in the small strain regime. In particular the proposed strategies exhibited numerical performances appreciably better than a tangent approach with line searches [Alfano, et al., 1999].

3.6 Summary

A review of nonlinear solution methods based on secant methods is presented in this Chapter [Wolfe, 1959; Barnes, 1961; Matthies and Strang, 1978; Owen and Gomez, 1981; Powell and Simons, 1981; Alfano, et al., 1997, 1999]. They include direct and path following secant techniques (BFGS, SN, MSN, etc.) [Crisfield, 1981-84; Zhang and Owen, 1982; Bathe, 1996]. Most of them are a combination of incremental and iterative procedures. However, they can be either incremental or iterative alone. Incremental procedures are adopted using stiffness matrix with updating co-ordinates and initial displacements using a number of load steps, and operations are predetermined as a series of linear problems. As in the case of tangent methods described in Chapter 2, errors are likely to accumulate after several steps unless very fine steps are adopted. Therefore, the accuracy can be improved by applying equilibrium corrections and thus preventing divergence considerably. An iterative procedure can be assumed to have converged when the unbalanced load becomes acceptably small, judged by the Euclidean norm (a scalar measure of the magnitude of the vector). In pure iterative methods, the total load is applied at a time and equilibrium is restored by iteration. Most of the time, path following techniques devise mixed schemes combining the features of both procedures. Secant type methods (e.g., Quasi-Newton) converge almost always in a larger number of steps than an optimal Newton strategy. They become competitive when the cost of Jacobian evaluation is significantly larger than that of the residual vector calculation. The procedure of using secant approximation to the derivative in Newton's iterative scheme for finding the solution of simultaneous equations generally takes less computer

time than the classical techniques [Jeeves, 1958]. Although the secant method requires more iterations, each iteration requires less time since there is no evaluation of the derivative of the function. Conversely, strong nonlinearity could lead to an ill-conditioned iteration matrix. However, for complex material nonlinearities, secant type methods (e.g., BFGS) are preferable. A line search can be applied to reduce the number of iterations significantly. The line search also costs the analysis, but most of the time, it makes the analysis more efficient and effective. Some approximate methods (Neuber, EGLOSS, etc.) in this Chapter based on direct secant techniques (robust methods based on elastic moduli adjustment techniques) have been studied [Neuber, 1961; Molski and Glinka, 1981; Seshadri, 1991, 1995; Alwar and Babu, 1995; Babu and Iyer, 1998-99; Raghavan, 1998; Knop, et al., 2000; Seshadri and Babu, 2000]. Most direct secant (e.g., EGLOSS) methods involve elastic moduli adjustment techniques that directly recalculate the stiffness matrix of the structures. All robust methods have the some considerations of relaxation locus, residual energy, etc. Traditional path following secant methods all have some kind of accelerations (line searches) associated with them. In the next Chapter, possible alternative robust methods combining the features of traditional secant line searches and ideas of relaxation locus are explored.

Chapter 4

Line Search Techniques for Secant Analyses

4.1 Introduction

Various robust techniques have been developed to predict local inelastic strains as well as limit load estimations based on elastic modulus adjustment procedures, e.g., Neuber [1961], Molski and Glinka [1981] and Seshadri [1991]. Robustness in the present context can be viewed as an ability to provide acceptable results on the basis of conceptual insight and the availability of a less than ideal material. Many of the robust techniques are based on the direct secant method for determining inelastic effects. In these methods, simple linear elastic FEA is carried out for solving elasto-plastic problems considering material parameters as field variables. The nonlinear response is obtained by employing two elastic analyses rather than carries out full nonlinear analysis. Highly stressed regions of the structure are systematically weakened by a reduction of their moduli in an attempt to simulate local inelastic softening. Robust methods don't require detailed constitutive relationships describing the inelastic flow. Since robust techniques are based on linear elastic FEA, they can be easily used to evaluate inelastic strains in practical structures having complicated geometry. The use of robust methods based on linear elastic analysis is significant from the design point of view. The design community has adapted these techniques as a design tools for some applications. Such

secant techniques have also been used successfully for the estimation of limit loads, e.g., Mackenzie, et al. [1993] and Shi, et al. [1993]. The present Chapter discusses some of these methods and possible extensions.

As described previously, Babu and Iyer [1998] developed a procedure called the MARS method using relaxation based on GLOSS analysis. There, an attempt was made to satisfy force equilibrium in the plastic range for a specific set of applications. In this approach, the residual force after every iteration is calculated and applied to figure out the equilibrium modified Young's modulus. This method requires several iterations. The calculation of unbalanced force seems to be application specific. In case of the EGLOSS method, the final state described by local inelastic strain does not satisfy the equilibrium of the structure for a given loading condition. The residual force after first linear elastic FEA is not directly accounted for although there is an implied localized compensation of the excess energy. Usually, the compensation obtained by modifying the modulus of elasticity of the yielded elements from the first linear elastic FEA is not enough. The increase in the volume of the yield zone is only indirectly accounted for. At larger loads, the discrepancy between the initial analysis yield zone size and the actual yield zone is very significant. This discrepancy increases with increase in loading except in the case of sudden stiffening (which is even more difficult to track through an initial elastic analysis). Hence, the inelastic strains are not estimated accurately and the error in the estimation increases with increase in the load level. Besides, at larger load levels, using EGLOSS modifications, Young's modulus could become negative thus rendering the method inapplicable.

Several other approximate methods have been developed to predict the local inelastic stresses and strains, e.g., Neuber's rule [Neuber, 1961], variations of Neuber's rule [Seeger and Heuler, 1985], strain energy density approach [Molski and Glinka, 1981], etc.

Neuber's rule predicts inelastic strains reasonably well for certain applications, e.g., plane stress problems. In plane strain situations, Neuber's rule has been widely reported to overestimate the inelastic strain. This could lead to significant errors in fatigue life predictions.

Since Neuber's rule overestimates the local plastic strains, a new method of elastic-plastic stress and strain calculation based on strain energy density was developed by Molski and Glinka [1980]. This approach is based on the assumption that the strain energy density distribution in the plastic zone ahead of a notch tip is the same as that determined on the basis of an elastic solution. When the stress near the notch increases beyond the yield stress, plastic deformation takes place. It is assumed that the energy ratio (the ratio of the strain energy per unit volume due to the local stress and the nominal remote stress, respectively) does not change due to the small plastic region. The relatively high volume of the elastic material surrounding the small plastic zone controls the amount of strain energy absorbed by the plastic zone. The results of an elastic analysis can be used in combination with the material constitutive relationship to estimate the strains in plastic zone without actually carrying out a plastic analysis. The application of this method is limited to the cases in which the plastic zone is small in comparison

with the surrounding elastic field. Neuber's rule has the same energy density interpretation (as Molski and Glinka) in the elastic regime. However, Neuber's stress-strain product does not represent the same energy density in the elastic-plastic regime. Therefore, the difference between these two methods is higher for plastic materials and high stress concentration factors.

Knop, et al. [2000] has done extensive work on both Neuber's rule and Molski and Glinka's approach for the prediction of inelastic stresses and strains near the notches and cracks. Neuber's rule tends to overestimate the notch tip stress and strain while Glinka's approach underestimates them. Neuber's rule makes better predictions for plane stress loading conditions, while Glinka's method makes better predictions for plane strain conditions. As the value of the stress concentration factor increases the predictions made by the Glinka's method improve. For tension loading the predictions made by Neuber's rule are better than for bending loading. The converse is true for Glinka. For torsion loading, the prediction made by Glinka method is better than Neuber's rule.

4.2 Line Search

In all the methods discussed above, one important aspect is either not recognized or is recognized but not accounted for explicitly. It is concerned with the difference between the size of the plastic zone indicated by the initial elastic analysis and the actual plastic zone. In the following, extensions to the concepts presented by previous researchers are

discussed. These extensions are mainly dependent on using line search to estimate the spread of the yield zone beyond that indicated by the first linear elastic FEA.

The basic steps involved in line search are described in Chapter 3. Several methods, e.g., BFGS, Crisfield MSN, basic or modified Newton-Raphson, etc. can use this technique. Several variants of this technique are possible. The procedure adopted in the present study is described below.

4.2.1 Basic Line Search Technique

The basic line search technique involves, as in the case of all other techniques, carrying out an elastic analysis on the original structure with all the given loading R and boundary conditions. Let the deformations, equivalent stresses and strains of this analysis be indicated by $[u_i]$, $[\sigma_i]$, $[\epsilon_i]$, respectively. If structure was loaded beyond the elastic limit, the stresses in the inelastic zone would show values higher than the yield stress of the material. Let the highest stressed element in the structure has equivalent (von Mises) stress σ_{e1} and equivalent strain ϵ_{e1} . This is represented by point D in Fig. 4.1. At point D, equilibrium and compatibility conditions are satisfied while the constitutive equation is violated. The excess stress at such points needs to be removed.

Once the stress level is brought down to yield stress, constitutive and compatibility conditions are satisfied but equilibrium is violated, i.e., removal of these stresses results in a net force imbalance. Let F represent the internal forces corresponding to this

reduced stress level. It can be estimated easily as [Zienkiewicz, et al., 1969; Owen and Hinton, 1986].

$$F = \int_V B^T \sigma dV \quad (4.2-1)$$

where, $[B]$ is the derivative matrix defining strain-displacement relationship,
 $[\sigma]$ is the stress

The difference between the applied load and F can be viewed to constitute an 'unbalanced' or a 'residual' force (ΔR).

$$\Delta R = R - F \quad (4.2-2)$$

This is the force due to the excess stresses beyond the yield point. This unbalanced force needs to be redistributed to the remainder of the structure in order to establish equilibrium. Such redistribution involves increasing the stresses at several points of the structure. As a result, some of the areas that were elastic in the initial analysis might be rendered inelastic. This means the yield boundary as indicated by the initial analysis expands upon redistribution. Also, the strain in the already yielded zones increases to maintain compatibility.

In line search technique, the residual force is applied as an external load to the original structure and the deformation $[\Delta u]$ due to the unbalanced forces is calculated. This does not involve a fresh analysis since the structure stiffness (and hence $[K^{-1}]$) is not altered.

$$\Delta u = K^{-1} \Delta R \quad (4.2-3)$$

Computation of $[\Delta u]$ is like finding the displacements to an alternate load case without disturbing the original matrix. This however, implies that the matrix (or its decomposed form) must be stored and hence requires an extension of the storage time. If such extra storage time is not needed, the matrix might be disposed off after the initial linear analysis. It must be noted that most practical techniques such as FNR, MNR, BFGS, etc., do not actually discard the matrix $[K]$. They store it (in its decomposed or inverse form) and 'update' it to carry out further iterations. Some of the techniques used in this regard are explained in Chapters 2 and 3. In that sense, the present computation of $[\Delta u]$ does not involve any additional storage or analysis except the computations implied by Eq. 4.2-1 to 4.2-3. It must be pointed out that this differs from the implementation of current 'robust' techniques such as EGLOSS, etc. These techniques carry out the initial analysis for the given loading and immediately discard the global matrix and other such data. They do not need to store it since the second analysis involves the assembly of a mostly 'new' stiffness matrix. Although they have not recognized it, these techniques can be modified to update only the necessary portions of the 'old' stiffness matrix to obtain a 'new' secant matrix based on the results of the initial analysis.

These displacements $[\Delta u]$ are considered as increments to the displacements $[u_i]$ obtained by the initial linear analysis. These displacement increments will not be the correct increments to fully balance the unbalanced force since the application of the increment

might result in further expansion of the yielded zone. However, this increment vector is thought to provide the 'direction' in which further increments can be carried out. A very good estimate of the exact increment is calculated by linearly scaling the computed increment $[\Delta u]$. If the scaling parameter is β , the line search displacement can be calculated as

$$u_{it} = u_i + \beta \Delta u \quad (4.2-4)$$

The parameter β is computed such that the product of the unbalanced load due to such increased displacement and $[\Delta u]$ is a minimum. In order to obtain the optimum value for β , a trial value is first assumed and trial values of $[u_{it}]$ are calculated. Strains and stresses for this displacement field are estimated. For this trial field of stresses, the unbalanced force is calculated (in the same manner as that indicated by Eq. 4.2-2). The work done by these unbalanced forces on the displacement increment $[\Delta u]$ is minimized by iteratively adjusting β . In other words,

$$\Delta u^T \Delta R = 0 \quad (4.2-5)$$

or,

$$\Delta u^T (R - F_{it}) \leq tol \Delta u^T (R - F_{i-1}) \quad (4.2-6)$$

where, tol is the tolerance number,
 i refers to the trial number.

The trail for determining the value of β is called as the line search. The final value of displacement at the end of line search is given by Eq. 4.2-4 where the converged or 'optimal' value of β needs to be used. Let the highest stressed element at this stage has the corresponding equivalent (von Mises) stress σ_e and equivalent (von Mises) strain ϵ_e .

The new displacement field $[u_b]$ represents significant expansion of the yielded zone compared to that predicted by the initial linear analysis $[u_l]$. This increase in the yielded zone is especially significant at large loads. As mentioned above, different variations of the line search technique can be used to accelerate the convergence rate in the FNR, MNR, Initial Stress, BFGS methods, as well as some other nonlinear solution algorithms.

However, previously described 'robust' techniques have not recognized the utility of the line search or its variants in estimating the inelastic zone size. Adluri [2001] showed that if the inelastic zone size is estimated with good accuracy, the analysis becomes linear and is completely determinable. The procedures described below apply a variety of simple geometrical constructions using the power of line search. These procedures give good predictions of inelastic strain for a variety of structures subject to a wide range of load levels.

After the line search is carried out, for all elements with equivalent stress exceeding yield stress, the a new secant modulus is estimated as below:

$$E_H = \frac{\sigma_y}{\epsilon_e} \quad (4.2-7)$$

A new yield stress akin to the reduced yield stress or equilibrium primary stress can be computed by lowering the yield stress level as below:

$$\sigma_y = \frac{\sigma_{e1}}{E_t} E_{II} \quad (4.2-8)$$

This equation is derived on the basis of line search and basic secant modulus concept explained in Chapter 3. A line search is carried out to expand the yield zone size (close to actual yield zone size) after initial elastic analysis. To satisfy the constitutive condition at this stage, the stress level is brought down to the specified yield stress. To carry out the second elastic analysis, the moduli of all yielded elements are reduced based on the specified yield stress and line search strain. Bringing σ_{e1} (the pseudo stress after initial elastic analysis) down to this secant line establishes the modified yield stress level defined by the Eq. 4.2.8.

All the work described above can be carried out after the initial linear analysis involving the assembly and processing of a single global stiffness equation. After the line search and the estimation of a secant modulus, a second linear elastic analysis is carried out (using the modified secant moduli of Eq. 4.2-7). The Poisson's ratio is left unchanged. The elastic properties of all other elements where the secant modulus is not used are left unchanged in the second FEA. The corresponding equivalent stress and strain of highest stressed element are determined and denoted as σ_{e2} and ϵ_{e2} .

It must be noted that sometimes, line search could overestimate the strain. In this case, the strain ϵ_{e2} obtained by second linear elastic FEA will be less than the strain ϵ_B . Computation of the optimum value β depends on the minimization of the product of the unbalanced load due to increased displacement and $[\Delta u]$. Therefore, the position of the strain after second linear elastic FEA depends on the prediction of line search.

OAC is the elastic-perfectly plastic stress-strain curve and OD is the elastic line. The pseudo elastic point D ($\sigma_{e1}, \epsilon_{e1}$) of the local element after first linear FEA is represented on this elastic line. For a particular element, the pseudo elastic point D' (σ_B, ϵ_B) shown in Fig. 4.2 (in Figs. 4.1 and 4.3, this point is not shown) is obtained after line search. This point has crossed the yield stress. Keeping the strain value the same and projecting the point D' on the stress-strain curve, the effective value of Young's modulus for the second FEA is obtained. The point M ($\sigma_{e2}, \epsilon_{e2}$) locates the stress and strain of the local element obtained from the second linear FEA. The slope of OM denoted, as E_{II} is known as secant modulus.

There are several ways of combinations to represent the pseudo-elastic stresses and strains after second linear FEA. To do this, several combinations are explored here. These are outlined as LSM1 to LSM7.

4.2.2 LSM1

In this procedure, an initial elastic analysis, a line search and a second linear analysis are carried out as described above. The stress-strain points D (from the first pseudo linear elastic analysis) and M (from the second pseudo linear elastic analysis) are joined together as shown in Fig. 4.1. The approximate relaxation line DM is extended to the point F on the yield stress line AC. If the point M is lower than the yield stress line, the point F is taken at the intersection of DM and the yield stress line AC. The maximum strain could be found either at point F or M depends upon the direction of line search. This maximum strain gives the approximate estimation including the plastic effects.

4.2.3 LSM2

In this procedure, the same analyses as described above are carried out. A modified yield stress is evaluated using Eq. 4.2-8.

As in the case of LSM1, the line DM is extended to intersect the new line representing the modified yield stress at point F' as shown in Fig. 4.1. The strain at point F' gives the approximate equivalent strain including the plastic effects.

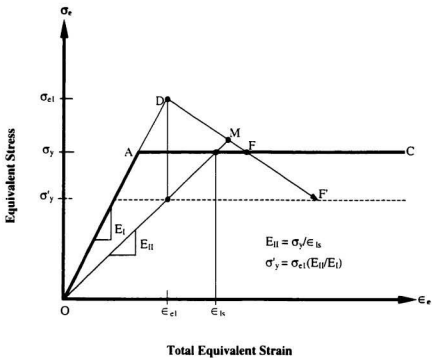


Fig. 4.1: LSM1 (Point F) and LSM2 (Point F') - Inelastic Strain Predictions

4.2.4 LSM3

At the end of the line search, we obtain an approximate estimate of equivalent strain. Using this strain, we can obtain an elastic stress by multiplying it with the Young's modulus. Let this strain and the corresponding pseudo linear elastic stress denote the point D' in Fig. 4.2. The line D'M is extended to the point F on the yield stress line. As before, the strain at point F is assumed to give an approximate equivalent strain including the plastic effects. Note that the point F in this analysis is not the same as the F in LSM1 procedure (Fig. 4.1).

4.2.5 LSM4

In this approach, the analysis is the same as in LSM3. The line D'M is extended to the lowered yield stress line (similar to that in LSM2) as shown in Fig. 4.2. The intersection is at F' which gives an approximate estimate of inelastic strain. Note that the point F' in this analysis is not the same as the F' in LSM2 procedure (Fig. 4.1).

4.2.6 LSM5

In this approach, an initial elastic analysis ($\sigma_{e1}, \epsilon_{e1}$), a line search (σ_b, ϵ_b) and a second linear analysis ($\sigma_{e2}, \epsilon_{e2}$) are carried out as described above for the previous methods. An approximate estimate of inelastic strain (ϵ) is evaluated using

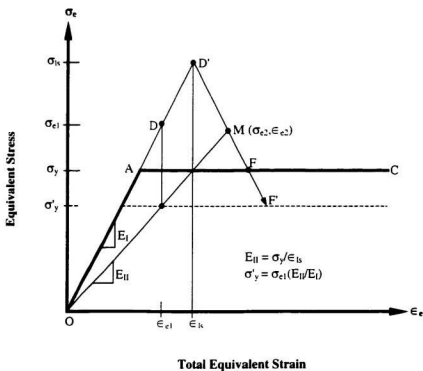


Fig. 4.2: LSM3 (Point F) and LSM4 (Point F') - Inelastic Strain Predictions

$$\epsilon = \frac{\sigma_{e2} \epsilon_{e2}}{\sigma_{e1} \epsilon_{e1}} \epsilon_{i1} \quad (4.2-9)$$

This is a simple extrapolation of a new strain based on the two elastic analyses [Adluri 2001]. It assumes that a second line search would increase the strain in the production of the ratios of strain energy in the two analyses. Instead of conducting a second line search, this equation is used to extrapolate the strain.

4.2.7 LSM6

In this procedure, the same analyses as described in LSM2 are carried out. A modified yield stress instead of Eq. 4.2-8 after second elastic analysis can be computed by modifying the yield stress level as below:

$$\sigma_y' = 2\sigma_y - \sigma_{e2} \quad (4.2-10)$$

This equation reflects a simple modification to compensate for the local strain energy loss in secant modification to the modulus [Adluri, 2001]. This is similar to that used by eq.3.3-8. That equation is based on pseudo-equilibrium of the local element.

Similar to others, the line DM is extended to intersect the new line representing the modified yield stress (defined by Eq. 4.2-10) at point F' as shown in Fig. 4.3. The strain at point F' is explored for the approximate equivalent strain including the plastic effects. Note that the point F' in this analysis is not the same as the F' in LSM2 and LSM4

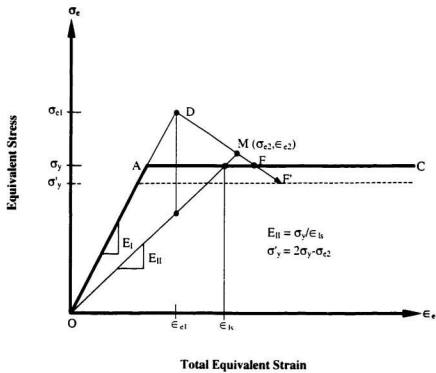


Fig. 4.3: LSM6 (Point F') - Inelastic Strain Predictions

procedures. It must be noted that sometimes, the effective yield stress level could be higher than the yield stress level.

4.2.8 LSM7

In this procedure, the same analyses as described in LSM4 are carried out. A modified yield stress is evaluated using Eq. 4.2-10 (as in LSM6) in stead of Eq. 4.2-8. As in the case of LSM4, the line D'M is extended to intersect the new line representing the modified yield stress at point F' as shown in Fig. 4.2. The strain at point F' gives the approximate equivalent strain including the plastic effects. It must be noted that the points F' and the effective stress level in LSM4 and LSM7 are not the same.

4.3 Procedures Based on Extensions to Neuber's Rule

The usefulness of Neuber's rule is explored in this study. The rule is not traditionally used to carry out iterative nonlinear analyses. But the simple concept of the rule can be extended to carry out progressive refinement of the initial analysis results. In the procedure used here, an initial analysis is carried out as usual (shown as point D in Fig. 4.4). For all the points above yield limit, the modulii are changed using

$$E_{II} = \frac{\sigma_y^2}{\sigma_{eI} \epsilon_{eI}} \quad (4.3-1)$$

This is the same as Eq. 3.3-1. For points that remain elastic, the original modulus is retained. Besides, an effective yield stress is obtained as defined in Eq. 4.2-8 above.

A second linear analysis is carried out using the moduli modified as appropriate. The result is represented by point M in Fig. 4.4.

Two combinations (based on Neuber's rule) to evaluate equivalent strains after second linear FEA are explored here. These are named as N1 and N2.

4.3.1 N1 Approach

In this procedure, an initial elastic analysis and a second linear analysis are carried out as described above. The stress-strain points D (from the first pseudo-elastic analysis) and M (from the second pseudo-elastic analysis) are joined together as shown in Fig. 4.4. The point F is found at the intersection of DM and the yield stress line AC. The strain at point F is assumed to be an approximate equivalent strain including the plastic effects.

4.3.2 N2 Approach

In this approach, the analysis is the same as in N1. The approximate relaxation line DM is extended to the modified yield stress line (defined in Eq. 4.2-8) as shown in Fig. 4.4. The intersection of DM and the modified yield stress line (point F') is considered as the approximation to the inelastic strain. Note that the point F' in this analysis is not the same as the F' in LSM2, LSM4, LAM6 and LSM7 procedures.

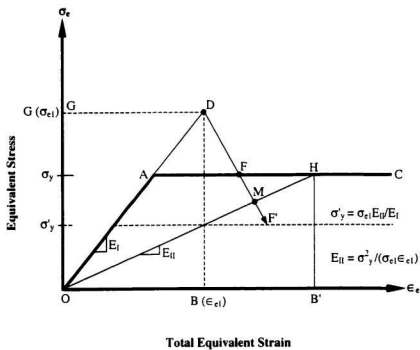


Fig. 4.4: Inelastic Strain Prediction Based on Extensions to Neuber's Rule

It must be noted that several other combinations of geometric constructions are possible. All those will have some physical justification. All of them are approximate. However, at least some of them will be robust in the sense that they will predict reasonably accurate strains with a relatively small effort and can absorb imperfections in data measurement.

The methods are loosely based on ideas from previous research. They include Neuber's rule, EGLOSS extensions to compute modified yield stress, line search to estimate the increase in yield boundary beyond that predicted by the initial linear analysis, etc. The individual researchers who developed the above techniques did not choose to make the combinations -especially the use of line search with robust techniques. It can be argued that the use or non-use of special techniques depends on the requirements of the analysis. For example, a quick estimate of strain can be obtained using simple Neuber's rule and nothing else. A much better estimation can be obtained using LSM1. But this would involve additional cost of line search. Regular nonlinear finite element analysis itself can benefit by using the techniques of robust methods to accelerate the analysis and or to check the accuracy of the iterative updates. The present study is attempt at exploring some of these possibilities.

All the combinations (except LSM5 and LSM6) described above are compared with the nonlinear FEA and summarized in Chapter 5.

Chapter 5

Numerical Examples

5.1 Introduction

In the previous Chapter, a number of alternative procedures have been outlined. These are robust techniques largely based on line search to minimize the work done by updated unbalanced forces on the displacements due to initial unbalanced forces. In addition, these techniques use ideas developed in other methods such as Neuber, EGLOSS, etc. To demonstrate the applicability of the proposed robust techniques to the solution of practical problems, a variety of sample analyses have been carried out.

The objective of this Chapter is to present the comparison of equivalent inelastic strain predictions for material non-linearity evaluated by the proposed methods with those of detailed inelastic FEA and theoretical results, where available. The relative merits of the different alternatives are discussed.

The problems are modeled by using ANSYS Finite Element software (Release 5.5) [ANSYS, 1998]. Four-noded isoparametric quadrilateral elements (PLANE42) are used for FE modeling for two-dimensional problems. PLANE42 has two degrees of freedom at each node. Elements of this kind have proven to be very effective and efficient in linear as well as general nonlinear continuum mechanics formulations. LINK1 and

SOLID45 element types are used for truss and rectangular plate problems, respectively. LINK1 is a two-noded element with two degrees of freedom at each node. SOLID45 is eight-noded solid element with three degrees of freedom at each node. The materials used in the modeling are considered to be homogeneous, isotropic and elastic-perfectly plastic. Wherever relevant, the models have been refined to obtain mesh convergence.

ANSYS FE software does not perform line searches after the first linear elastic analysis. The algorithm used by ANSYS seems to need at least two analyses to establish search directions and meaningful updates. However, basic line search can always be carried out irrespective of the condition of the analysis -although in some cases, line search may not show significant change in the yield boundary. In order to carry out the line search outside the purview of ANSYS, a set of procedures reflecting the basic ideas of line search parameters have been developed. These procedures are purely for the ease of carrying out line search and do not in any way influence the effectiveness (or the lack thereof) of the methods being discussed. If one were to program these methods directly, these extra procedures need not be resorted to. All these extra procedures have been programmed using the ANSYS Design Parametric Language (ADPL) provided with the ANSYS package [ANSYS, 1998]. The language is somewhat similar to Fortran and can be easily understood. Sample macros involving the extra programming are included in Appendix A. It lists input files for the analysis of simply supported beam under uniformly distributed load (UDL). The input files of linear and nonlinear analyses are listed in Appendix B. The macros to perform the necessary elastic modulus changes and post-processor routines for EGLOSS and N2 methods are listed in Appendix C. It must

be pointed out that the equivalent strain values obtained from ANSYS do not include the Poisson's ratio factor. Therefore, elastic and plastic equivalent strain values obtained using ANSYS commands [ANSYS, 1998], respectively must be divided by $(1+\nu)$ in order to get the correct elastic and plastic equivalent strains. Details are given in Appendix D.

5.2 Beams with UDL

Beams form an important class of structural components since they are commonly used in many kinds of applications. Several times during the life of a beam the load is expected to increase beyond the initial yield there by creating accumulation of plastic strain. In the present work, three beams with different end conditions subjected to UDL are selected to carry out the equivalent inelastic strain predictions. All three beams have the same dimensions and properties. The beams have a span of 508 mm (20 in.), depth of 25.4 mm (1 in.) and unit thickness. Although the dimensions are odd, this problem has been selected for analysis since it was used by previous researchers [Raghavan, 1998; Seshadri and Fernando, 1992].

All beams are modeled as two-dimensional problems. Therefore, these beams are assumed to have unit width in the direction normal to the paper. The material of all beams has a yield stress of 206.85 MPa (30 ksi) and a Young's modulus of 206,850 MPa (30,000 ksi). The Poisson's ratio is taken as 0.3.

It is known that individual robust techniques are good for certain load levels. Most are applicable for low levels of plastic strains since the relaxation locus is relatively straight line. In this study, different load levels ranging from just above yield to nearly limit loads have been considered to ascertain the effectiveness of the various techniques under consideration.

5.2.1 Simply Supported Beam

This is a determinate beam. Elastic results for the beam are known from elementary texts. A theoretical nonlinear analysis (detail analysis is given in Appendix E) has also been carried out for this beam.

The FE model of the beam has 60 equal divisions along the beam span and 10 equal divisions along the depth. PLANE42 elements of ANSYS are selected. The finite element mesh and the boundary conditions are shown in Figs. 5.1a and b. The beam model has zero displacement in X and Y directions at the mid side node of left support. At the right support, the mid side node is restrained in the Y direction. Despite the aspect ratio of the elements seems to be very high, the mesh has been used to carry out an elastic analysis and a detailed inelastic analysis, because the results of both the analyses are in excellent agreement with the theoretical analysis as shown in Table 5.1. Thus, the model can be used to test the effectiveness of the different possible techniques proposed in Chapter 4.

Extreme fibers of mid-span constitute the critical zone where yielding begins. The beam remains elastic when the load is less than the yield load. When the applied load equals the yield load, the extreme fibers of the section at mid-span start to yield. When the load is increased beyond the yield load, the maximum moment at mid-span exceeds the yield moment, thus spreading the yield boundary both in the length direction and depth direction. This spreading of the yield zone continues until the entire section at mid-span is yielded. The limit load for the problem is 1034.4 kPa (150 psi). The theoretical load at the first onset of yielding at extreme fiber is 689.6 kPa (100 psi). Different load stages (shown in Table 5.1) between the yield and limit load are considered to compare the different proposed techniques with the exact analysis and nonlinear FEA. These loads are chosen arbitrarily to demonstrate the applicability of the proposed techniques at different load levels. The input files of linear elastic and nonlinear analyses are given in Appendices B.1.1 and B.1.2, respectively.

The highest strain is identified at mid-span corresponding to element number 300 or 310. Since ANSYS gives one strain per element, this strain is taken to be at the centre of the element. At a load of 827.5 kPa (120 psi, 20% higher than the yield load and 40% of the interval between yield and ultimate loads), the predictions made by LSM1, LSM3, LSM4, LSM7 and N1 are in excellent agreement with those of the nonlinear analysis (less than 0.5% below). The EGLOSS, N2 and LSM2 give strains more than 5% (conservative) above the nonlinear analysis results. At a load of 896.5 kPa (130 psi, 30% higher than the yield load), LSM1, LSM3, LSM4 and LSM7 show errors well below 2% while the others are relatively acceptable. Even at load stage of 965.4 kPa (140 psi), the

strain prediction given by LSM1, LSM3, LSM4 and LSM7 is 8 to 9%. The prediction by the EGLOSS and N2 show marginally higher error. As the load level increases the error also increases, because at larger load levels, the discrepancy between the initial analysis yield zone size and the actual yield size is very significant. This discrepancy increases with increase in loading.

It must be pointed out that the EGLOSS and N2 errors change signs. Near the load levels where the error trends change signs, the error may seem to be very low. Thus, it must be noted that a low error level only at a particular load does not necessarily indicate the effectiveness of a certain method. At the same load level, LSM2 and N1 show more than 20% error. Similar trends with increased error continue at a load of 1000 kPa (145 psi close to the limit load). It is noted that at every load stage, the techniques based on line searches (LSM1, LSM3, LSM4 and LSM7) give better estimation though unconservative, compared to the EGLOSS, N1 and N2 methods. The remaining line search techniques such as LSM2 were not as good as the rest of the line search techniques. For comparison purposes, the full nonlinear analysis (Newton-Raphson) with line search option has been employed by restricting the analysis to two iterations. The results are reported in the Table above against NFEA-2itr. The results show zero error for load case 1. However, for subsequent load cases, the error is significantly high. The zero error in load case 1 is probably due to the fact that the error trend is changing sign (from negative to the positive) at that particular load.

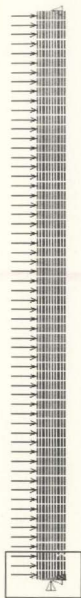


Fig. 5.1a: Finite Element Model - Simply Supported Beam with UDL

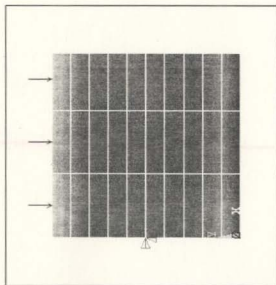


Fig. 5.1b: Boundary condition at Left Support

Table 5.1: Simply Supported Beam with UDL -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1			*Load Case 2			*Load Case 3			*Load Case 4		
	Strain	Error1	Error2	Strain	Error1	Error2	Strain	Error1	Error2	Strain	Error1	Error2
LSM 1	1156.2	0.28	0.43	1393.4	1.36	1.99	1813.7	8.85	9.7	2174.7	22.91	23.28
LSM 2	12188	-5.11	-4.95	1588.1	-12.41	-11.70	2439.2	-22.62	-21.43	3298.5	-16.91	-16.36
LSM 3	1156.2	0.29	0.43	1393.5	1.35	1.98	1813.9	8.81	9.68	2161.7	23.38	23.73
LSM 4	1155.8	0.32	0.46	1389.9	1.61	2.23	1806.9	9.16	10.04	2165.1	23.26	23.61
LSM7	1156.2	0.28	0.43	1393.7	1.34	1.97	1814.2	8.79	9.67	2161.8	23.37	23.73
N1	1155.2	0.37	0.52	1331	5.78	6.38	1506.7	24.25	24.98	1617.7	42.66	42.92
N2	1225.1	-5.65	-5.49	1468.8	-3.97	-3.31	1702.9	14.39	15.21	1857.6	34.16	34.46
E-GLOSS	1230.6	-6.13	-5.97	1492.2	-5.62	-4.95	1753.7	11.83	12.68	1925.3	31.76	32.07
NFEA-2itr	1159.4	0.00	0.15	1335.1	5.49	6.09	1673.1	15.88	16.70	1805.7	36.00	36.30
Nonlinear FEA	1159.5		0.14	1412.7		0.63	1989.1		0.96	2821.4		0.46
Exact Analysis	1161.2			1421.7			2008.5			2834.6		

*Load Case	Load (Pa)	β
1	827.5 (120 psi)	1.815
2	896.5 (130 psi)	2.62
3	965.4 (140 psi)	3.89
4	1000.0(145 psi)	4.673

Note: All strains indicated above are in micro-strain units.

Errors are in percent (%). Error1 and Error2 are in comparison with nonlinear FEA and exact analysis, respectively.

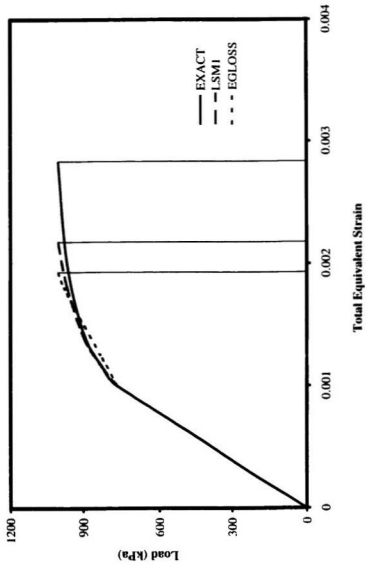


Fig. 5.1c: Inelastic Strain Prediction - Simply Supported Beam

The techniques LSM5 and LSM6 were also examined for various problems. Generally their results are not as good as those of the other line search techniques listed in the Tables in this Chapter.

Representative results for LSM1 and EGLOSS are plotted in Fig. 5.1c. It must be noted that the three curves shown appear to be 'close' to each other. However, the appearance is somewhat deceptive. The error must be looked as the difference in strain for a particular stress level. To illustrate the point, three strain lines are drawn for the three curves at load case 4. The difference indicated by these three vertical lines must be considered to understand the effectiveness of a particular method.

5.2.2 Propped Cantilever Beam with UDL

The propped cantilever beam has the left end built-in and the other end simply supported. It is subjected to a distributed load. With increasing load the first plastic hinge forms at the built-in support and the structure subsequently becomes determinate. As the load is increased further, an additional plastic hinge forms in the span leading to the collapse of the structure. The limit load for the problem is 1551 kPa (225 psi). The load at the first onset of yielding is 793 kPa (115 psi) as per ANSYS. As in the case of simply supported beam with UDL, different loads (ranging between the yield and limit loads) are considered to evaluate the effectiveness of different procedures. The input files of linear elastic and nonlinear analyses are given in Appendices B.2.1 and B.2.2, respectively. The results for different methods are presented in Table 5.2.

The FE model of this beam has 100 equal divisions along the beam span and 10 equal divisions along the depth. PLANE42 elements of ANSYS are used. The finite element mesh and the boundary conditions are shown in Figs. 5.2a and b. The beam model has zero displacement in X and Y directions at the each node of left support. At the right support, the bottom node (if the middle node is taken, the response does not change significantly) is restrained in the Y direction. The mesh has been used to carry out an elastic analysis, NFEA-2itr (nonlinear response obtained after two iterations) and a detailed inelastic analysis.

The highest strain is identified at the built-in support corresponding to element number 1. At each load stage except at the lower end, the EGLOSS method gives slightly better predictions than the other methods, even when the load is significant. This is in spite of the fact that the EGLOSS method is developed for low load levels. At the load of 896.5 kPa (130 psi), LSM1, LSM3, LSM7, N1 and NFEA-2itr give good results compared to other techniques. LSM2 gives good prediction at high loads. However, this seems to be due to the fact that the error trend for this method changed sign and hence momentarily seems to be better. Representative results for LSM1 and EGLOSS are plotted in Fig. 5.2c.

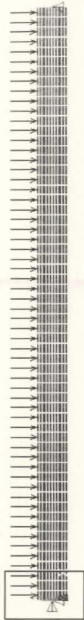


Fig. 5.1a: Finite Element Model - Simply Supported Beam with UDL

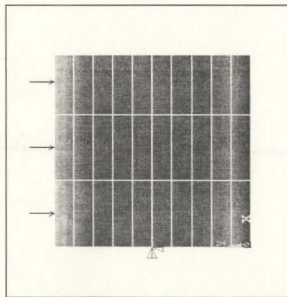


Fig. 5.1b: Boundary condition at Left Support

Table 5.2: Propped Cantilever Beam with UDL -Inelastic Strain Prediction at Different loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	1297.3	2.89	1508.4	5.29	2486.8	19.55	2930.5	26.86
LSM 2	1438.6	-7.68	1740.9	-9.30	3321.7	-7.447	4003.8	0.078
LSM 3	1298.5	2.79	1509.3	5.23	2469.8	20.11	2880.4	28.11
LSM 4	1288.5	3.54	1498.6	5.90	2475.3	19.93	2895.1	27.75
LSM7	1299.7	2.70	1510.3	5.17	2469.9	20.10	2881.1	28.09
N1	1274.2	4.62	1440.1	9.57	2011.6	34.93	2271.1	43.32
N2	1401.2	-4.88	1662.5	-4.38	2372.1	23.27	2721.8	32.07
E-GLOSS	1419.7	-6.27	1656.4	-4.00	2557.7	17.26	2989.1	25.40
NFEA-2itr	1323.4	0.94	1438.4	9.68	2204.3	28.7	2665.1	33.49
Nonlinear FEA	1339.3		1592.7		3091.5		4007.0	

*Load Case	Load (kPa)	β
1	896.5 (130 psi)	2.2
2	965.4 (140 psi)	2.545
3	1172.3 (170 psi)	3.6283
4	1241.3 (180 psi)	3.9

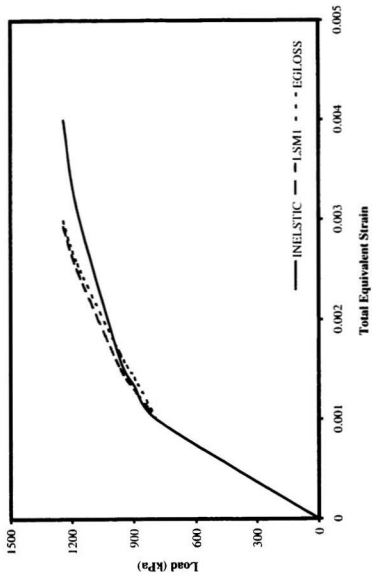


Fig. 5.2c: Inelastic Strain Prediction - Propped Cantilever Beam

5.2.3 Fixed-Fixed Beam with UDL

A fixed-fixed beam has been analyzed for the same properties as those for the two other beams described above. At a load of 1230.6 kPa (178.5 psi), the extreme fibers at the end supports start to yield. The spreading of the yielding zone continues in to the span and along the depth with increasing load and first plastic hinge is formed at the supports. As the load is increased further, an additional plastic hinge forms at the middle of the span leading to collapse. The limit load for the problem is 2221 kPa (322 psi). The results of different methods at different load stages between the yield and limit loads are presented in Table 5.

The FE model of this beam has 60 equal divisions along the beam span and 10 equal divisions along the depth. PLANE42 elements of ANSYS are used. The beam model has zero displacement in X and Y directions at the each node of left and right supports. The mesh has been used to carry out an elastic analysis, NEFA-2itr and a detailed inelastic analysis. The input files of linear elastic and nonlinear analyses are given in Appendices B.3.1 and B.3.2, respectively.

The highest local strain is identified at the built-in support corresponding to the element number 1. In prediction of critical strain at low load, all line search methods except LSM2 give good comparison with nonlinear FEA. As in the case of propped cantilever beam, LSM2 is somewhat inconsistent.

Table 5.3: Fixed-Fixed Beam -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	1148.2	1.61	1402.5	2.94	2330.9	11.51	3955.9	31.61
LSM 2	1227.6	-5.19	1577.3	-9.14	3219.8	-22.22	6102.5	-5.50
LSM 3	1149.1	1.54	1404.5	2.81	2331.1	11.51	3894.1	32.67
LSM 4	1142.9	2.06	1390.1	3.81	2323.0	11.81	3907.2	32.45
LSM7	1149.8	1.47	1406.6	2.66	2331.3	11.50	3894.5	32.67
N1	1135.9	2.66	1355.3	6.22	1827.1	30.64	2562.9	55.7
N2	1202.9	-3.07	1507.0	-4.28	2131.3	19.09	3140.0	45.71
E-GLOSS	1207.2	-3.44	1536.2	-6.29	2256.1	14.35	3468.9	40.02
NFEA-2itr	1162.6	0.37	1359.4	5.93	2011.09	23.65	3015.7	47.86
Nonlinear FEA	1167.1		1445.2		2634.3		5784.3	

*Load Case	Load (MPa)	β
1	1.31 (190 psi)	1.835
2	1.45 (210 psi)	2.605
3	1.73 (250 psi)	4.156
4	2.00(290 psi)	5.8055

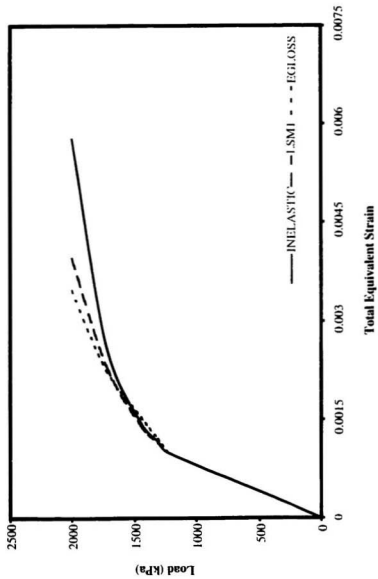


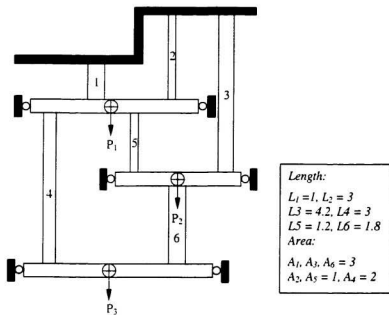
Fig. 5.3a: Inelastic Strain Prediction - Fixed-Fixed Beam

At a load of 1724 kPa (250 psi), the errors in the EGLOSS, N1, N2 and NFEA-2itr (except at low load level) methods are quite significant while LSM1, LSM3, LSM4 and LSM7 are reasonable. At a load of 2000 kPa (290 psi), the trends continue. Representative results for LSM1 and EGLOSS are plotted in Fig. 5.3a. It has been noted that in all three beams, at load just above the yield load level, NFEA-2itr shows excellent agreement with a detailed nonlinear analysis compared to the other methods described above. But at higher loads, this excellency is dissipated quickly.

5.3 Simple Multi-Bar Truss

In the previous section, three cases of a beam have been explored. The predominant mode of non-linearity is due to bending. In this section, a simple truss with six members arranged parallel to each other is selected to apply the different analytical procedures.

In trusses of this kind, the failure is by the direct yielding of entire members. Compression failure by buckling is excluded by design. Such 'multi-bar' structures are popularly used to study the effectiveness of inelastic analysis procedures, e.g., Chen [1996] and Seshadri [1991]. It is customary to take two or three bars parallel to each other to carry out such analyses. In the present case, a truss with six bars is taken. This was used by Adluri [2001] to demonstrate the limit load estimation techniques.





- Note: (1) Figures not to scale
 (2) All dimension in arbitrary
 (3) P_1, P_2 and P_3 are loads
 (4)  Roller support
 Fixed support

Fig. 5.4a: A Simple Truss Model

Table 5.4: Simple Multi-Bar Truss -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain	Error (%)	Strain	Error (%)	Strain	Error (%)	Strain	Error (%)
LSM 1	1.318	2.65	2.189	8.00	2.488	8.57	2.997	3.58
LSM 2	1.802	-33.06	3.189	-34.00	3.601	-32.31	4.562	-46.75
LSM 3	1.277	5.73	2.016	15.25	2.275	16.40	2.729	12.22
LSM 4	1.286	5.01	2.058	13.49	2.326	14.50	2.787	10.35
LSM7	1.277	5.67	2.024	14.93	2.285	16.02	2.739	11.88
N1	1.276	5.77	2.062	13.32	2.342	13.93	2.629	15.43
N2	1.464	-8.11	2.662	-11.88	3.063	-12.56	3.465	-11.45
E-GLOSS	1.128	16.67	2.871	-20.62	3.388	-24.49	3.9360	-26.61
NFEA-2itr	1.384	-2.20	2.436	-2.39	2.787	-2.45	3.138	-0.94
Nonlinear FEA	1.354		2.379		2.722		3.109	
Exact Analysis	1.354		2.379		2.722		3.109	

*Load Case	P1	P2	P3	β
1	1.5	2.25	1.5	3.43634
2	1.8	2.7	1.8	2.98711
3	1.9	2.85	1.9	2.96013
4	2	3	2	3.58684

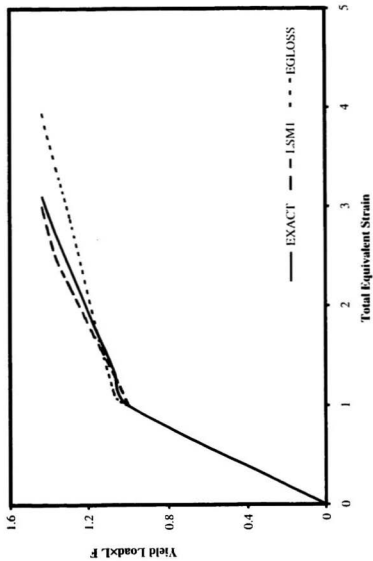


Fig. 5.4b: Inelastic Strain Prediction - Simple Truss

This truss is unlike regular truss like structures in the sense that it exhibits sudden stiffening during iterative secant analysis where as it does not exhibit sudden stiffening in an exact analysis.

If the parameters are changed slightly, even the exact analysis indicates sudden stiffening. Traditionally, problems with sudden stiffening are cumbersome to deal with. The configuration of truss and the dimensions of the members are shown in Fig. 5.4a. The yield stress of the material is assumed to be 1 and the Young's modulus is also taken as 1.

In FE modeling for the truss, LINK1 is used. LINK1 is a two-noded element with two degrees of freedom at each node. The input file of nonlinear analysis is given in Appendix B.4.1. The loads shown in Table 5.4 are applied arbitrarily and the equivalent inelastic strains are evaluated.

Both exact analysis and inelastic FEA are carried out for this problem. Both these show identical results. The results of the analyses are shown in Table 5.4. Member 1 first yields in the initial analysis and is considered as a critical member for equivalent inelastic strain approximation. At all load levels (including close to the limit load), LSM1 and NFEA-2itr give good prediction. Since the full nonlinear analysis for this problem converges in three iterations, NFEA-2itr (after two nonlinear iterations) gives very good estimates. At each load stage, N2 overestimates the strain but is reasonably good. LSM2 is consistently too high in its predictions at all load stages. The prediction made by N1 and EGLOSS methods is also not quite acceptable. LSM3, LSM4 and LSM7 show

results that are slightly inconsistent but are acceptable. Representative results for LSM1 and EGLOSS are plotted in Fig. 5.4b.

5.4 Plate with a Central Hole under Uniform Tension

A flat plate of length 76.2 mm and width 38.1 mm with a central hole of radius 6.375 mm (Fig. 5.5a) is solved for equivalent inelastic strain prediction based on the GLOSS method by Raghavan [1998]. Babu and Iyer [1998] have taken the same problem for inelastic analysis of components using a modulus adjustment scheme called MARS. Seshadri [1991] had earlier solved that problem for showing the effectiveness of GLOSS analysis. Chen [1992] used it for an accelerated method in elastic-plastic finite element computation. They used a uniform pressure of 200 MPa applied to the plate. In the present study, four different load cases are used. The yield stress of the material is 363.2 MPa and the Young's modulus is 72,368 MPa. The Poisson's ratio is assumed to be equal to 0.3.

In FE modeling, the key points are defined along the edges of the plate. The key points are connected by line in Cartesian and Cylindrical coordinates. The mesh is made denser near the hole. The corresponding key points define areas and AMESH command performs the automatic meshing. Due to symmetry, only one quarter of the plate is modeled. Four-noded isoparametric quadrilateral element (PLANE42) is used. Symmetric boundary conditions are given for the relevant edges. The model is shown in Figs. 5.5b and c. The input files of linear elastic and nonlinear analyses are given in

Appendices B.5.1 and B.5.2, respectively. The results for different methods are presented in Table 5.5.

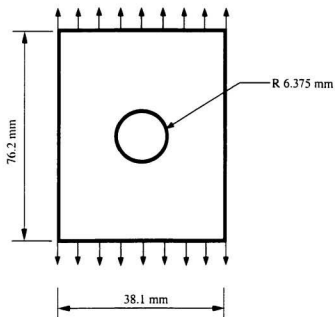
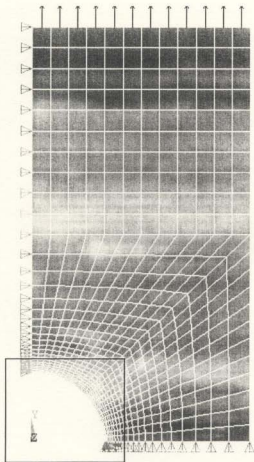


Fig. 5.5a: Plate with a Circular Hole



**Fig. 5.5b: Finite Element Model -
Plate with a Hole**

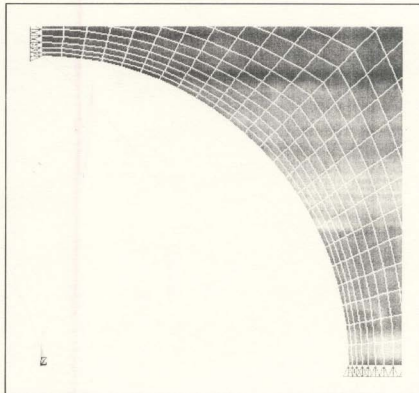


Fig. 5.5c: Detail near the hole

Table 5.5: Plate with a Central Hole -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	8454.5	6.48	10501.3	9.82	15663.3	13.59	17290.2	14.19
LSM 2	8998.6	0.47	11459.1	1.60	17762.2	2.01	19786.7	1.80
LSM 3	8454.5	6.48	10481.9	9.99	15583.9	14.03	17186.2	14.71
LSM 4	8452.1	6.51	10492.5	9.90	15620.4	13.83	17231.7	14.48
LSM7	8454.5	6.48	10482.1	9.99	15585.3	14.02	17188.1	14.70
N1	8080.4	10.63	9817.6	15.70	13918.9	23.22	15093.5	25.09
N2	8791.4	2.76	10941.4	6.04	15950.0	12.01	17372.0	13.79
E-GLOSS	9086.2	-0.50	11717.4	-0.62	20486.7	-13.01	-N/A- **	-N/A- **
NFEA-2itr	8752.9	3.18	11276.5	3.16	19744.7	-8.92	21654.4	-7.46
Nonlinear FEA	9041.0		11645.6		18127.6		20150.4	

*Load Case	Load (MPa)	β
1	150	2.05
2	170	2.225
3	210	2.73
4	220	2.897

Note: ** Analysis could not be carried out because of negative modulus update

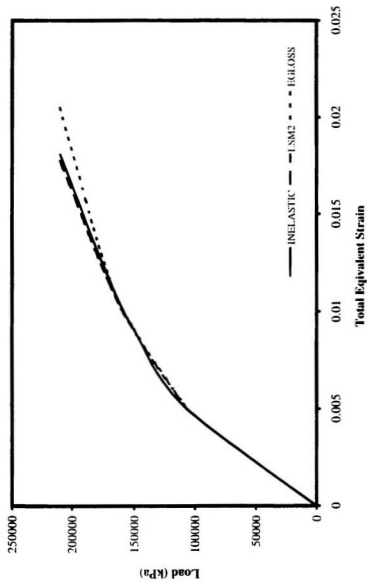
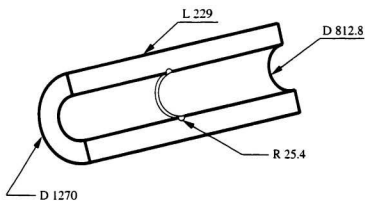


Fig.5.5d: Inelastic Strain Prediction - Flat Plate with a Central Hole

After first linear elastic FEA, the highest critical strain is found near the hole region. The pressure at the first onset of yielding is 106.4 MPa. The limit load for this problem is 242.3 MPa. At low load levels, EGLOSS shows excellent results in comparison to other methods. Similarly, NFEA-2itr is also acceptably good at low loads. For higher loads such as a load of 220 MPa, the EGLOSS method fails since the modified moduli become negative. At a load of 210 MPa, the EGLOSS method gives a high error like other methods except LSM2. The LSM2 method gives surprisingly excellent results at all load levels for this strain concentration type problem. However, it must be noted that the method does not give consistently good results for other types of problems (e.g., beams). The other line search techniques although not as good as LSM2, give consistent trends. Representative results for LSM2 and EGLOSS are plotted in Fig. 5.5d.

5.5 Thick Cylinder with a Circumferential Notch

A thick cylinder of length 228.6 mm (9 in), inner diameter 812.8 mm (32 in.) and outer diameter 1270 mm (50 in.) with a circumferential notch radius of 25.4 mm (1 in.) on the inside surface under internal is considered. The cylinder is subjected to plane strain condition. Different load stages under plane strain condition are considered to compare the line search techniques. The material has a yield stress of 200 MPa (29 ksi) and a Young's modulus of 190,000 MPa (27,500 ksi). The Poisson's ratio is 0.3. Raghavan [1998], and Seshadri and Kizhatil [1993] solved this problem for equivalent inelastic strain estimation based on the GLOSS method.

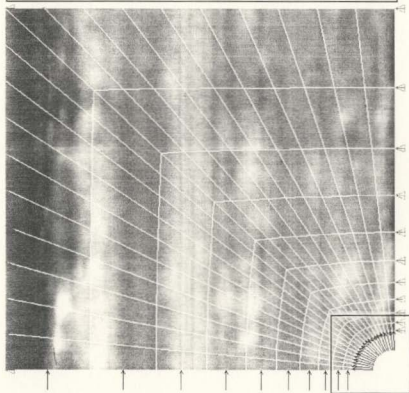


Note: Figures not to scale
All dimensions in mm

Fig. 5.6a: Thick Cylinder with a Circumferential Notch

Part of the cylinder is modeled under axisymmetry condition. The mesh chosen is finer near the circumferential notch. Isoparametric quadrilateral element PLANE42 with axisymmetric option is used. Symmetric boundary conditions are given for the relevant edges. The model is shown in Figs. 5.5b and c. The input files of linear elastic and nonlinear analyses are given in Appendices B.6.1 and B.6.2, respectively. The results for different methods are presented in Table 5.5.

The pressure at the first onset of yielding is 60.7 MPa (8.8 ksi) and the limit load is 175 MPa (25.4 ksi) as per ANSYS. Table 5.6 shows the equivalent inelastic strain predictions obtained by different methods for various loading stages. It can be seen that the EGLOSS method over predicts the equivalent inelastic strain at each load stage, while N2 gives over estimation at low load stages but is not as conservative as the EGLOSS method. Even at higher load, the error obtained by N2 is found to be no more than five percent. Methods LSM2, N2 (based on Neuber's rule) and NFEA-2itr are in very good agreement with nonlinear FEA. LSM1 and LSM3 give consistent equivalent inelastic strain trends as the error increases with increasing load. The estimation given by the EGLOSS method at low load levels is a little better than that of LSM1 but not as much as that by the modified Neuber method (N2) and LSM2. At a load of 96.5 MPa (14 ksi), the EGLOSS method shows nearly 5% higher error than that obtained by LSM1. Considering higher load stages, LSM1 has shown better prediction than the EGLOSS method. The overall predictions obtained by NFEA-2itr at all load levels are also acceptable. Representative results for LSM2 and EGLOSS are plotted in Fig. 5.6d.



**Fig. 5.6b: Finite Element Model -
Cylinder with Circumferential Notch**

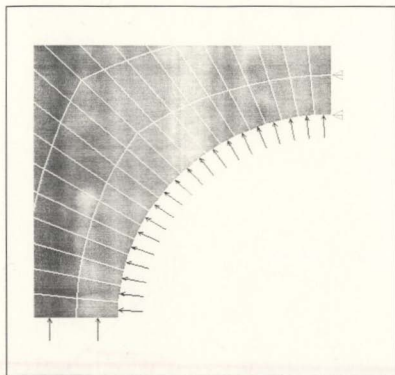


Fig. 5.6c: Detail at Notch

Table 5.6: Cylinder with Circumferential Notch -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	1306.0	2.70	1502.9	5.42	1945.2	7.78	2723.6	9.43
LSM 2	1357.6	-1.13	1580.9	0.52	2151.2	-1.98	3307.1	-9.97
LSM 3	1311.0	2.33	1506.8	5.18	1951.4	7.48	2729.3	9.25
LSM 4	1294.3	3.58	1490.0	6.24	1920.6	8.94	2689.1	10.59
LSM7	1316.0	1.95	1510.7	4.93	1957.6	7.19	2735.0	9.05
N1	1285.1	4.26	1480.5	6.84	1861.3	11.75	2415.3	19.69
N2	1362.4	-1.50	1611.4	-1.40	2118.3	-0.43	2880.2	4.23
E-GLOSS	1376.0	-2.51	1662.1	-4.59	2319.9	-9.98	3536.6	-17.59
NFEA-2itr	1334.9	0.54	1570.6	1.16	2044.9	3.05	2702.8	10.13
Nonlinear FEA	1342.3		1589.1		2109.3		3007.4	

*Load Case	Load (MPa)	β
1	68.9 (10 ksi)	1.94
2	75.8 (11 ksi)	1.665
3	86.2 (12.5 ksi)	2.1226
4	96.5 (14 ksi)	2.9632

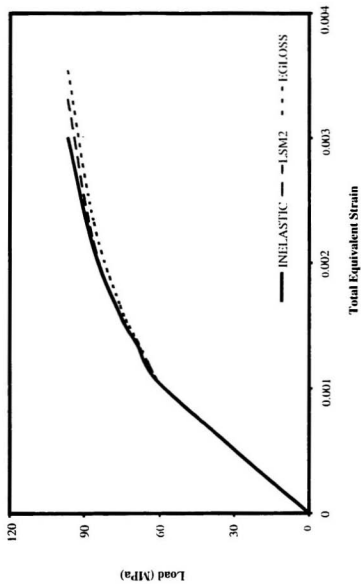


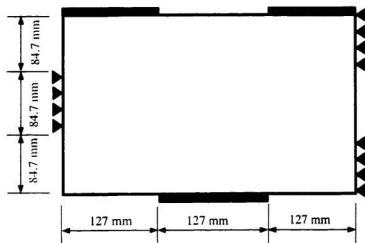
Fig. 5.6d: Inelastic Strain Prediction -Cylinder with Circumferential Notch

5.6 Bending of Rectangular Plate with Partial Fixity

A rectangular plate 382 mm (15 in.) long, 254 mm (10 in.) wide and 12.7 mm (0.5 in.) thick has been selected to carry out nonlinear strain prediction. The plate is subjected to uniform lateral pressure throughout. The plate is partially fixed and partially simply supported as shown in Fig. 5.7a. Plates are widely used as important structural components such as flat heads of pressure components, internals of pressure vessels, heat exchangers, and a variety of building structure applications. The complex boundary conditions have been chosen intentionally for this problem to investigate the versatility and robustness of the procedures under consideration, since analytical solutions for such configurations are difficult to obtain. For instance, the complex geometry along with the boundary conditions can cause shear interactions thus rendering an analytical elastic-plastic analysis intractable.

The plate material has a yield stress of 206.85 MPa (30 ksi) and a Young's modulus of 206,850 MPa (30,000 ksi). The Poisson's ratio is 0.3. These properties are the same as those for the beam type problems described earlier.

This problem has no symmetry. It is modeled as a three-dimensional solid using SOLID45 elements of ANSYS. These are eight-noded solid elements with three degrees of freedom at each node. The FE model is shown in Fig. 5.7b.



- Note: (1) Figures not to scale
 (2) Plate subjected to uniform pressure
 (3)
- Fixed supported edge
 - ▲ Simply supported edge
 - Free edge

Fig. 5.7a: Rectangular Plate Partially Fixed, Partially Free and Partially Supported

Due to the complexity of boundary conditions, before first linear elastic FEA, it is difficult to say where the critical strain occurs. Twelve equal divisions along the length and width are considered to model the plate. The thickness is divided into 5 divisions. Displacements at each fixed node are restrained in all X, Y and Z directions. At simply supported nodes, only Y direction is restrained. The input listings of linear elastic and nonlinear analyses are given in Appendices B.7.1 and B.7.2, respectively. Mangalaramanan [1997] and Bolar [2002] have studied this problem for limit load behavior. Note that the problem could have been solved using plate-bending elements as well.

The limit load for the problem is 3483 kPa (505 psi) [Mangalaramanan, 1997]. The load at the first onset of yielding is 1547 kPa (224.3 psi) as per ANSYS. Different load stages between the yield and limit load are selected to compare the different line search techniques with the nonlinear FEA. These load levels are 1862 kPa (270 psi), 2068.5 kPa (300 psi), 2206.7 kPa (320 psi) and 3103 kPa (450 psi).

In spite of the complex boundary conditions, at low loads all methods show good estimation of inelastic strain. All methods including NFEA-2itr (nonlinear results after two iterations) show good results even at moderately large loads such as 1106 kPa (320 psi). The EGLOSS method predicts negative moduli at higher load of 3103 kPa (450 psi) and thus fails. Surprisingly, LSM2 shows a remarkable consistency throughout the load increase. Representative results for LSM1 and EGLOSS are plotted in Fig. 5.

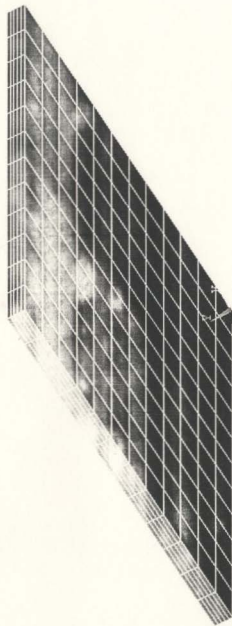


Fig. 5.7b: Finite Element Model - Rectangular Plate with Partial Fixity

Table 5.7: Bending of Rectangular Plate -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	1324.7	0.12	1552.0	1.19	1720.3	2.17	3775.3	17.33
LSM 2	1367.6	-3.12	1622.4	-3.28	1813.9	-3.15	4504.5	1.36
LSM 3	1330.5	-0.32	1560.2	0.67	1730.4	1.59	3780.2	17.225
LSM 4	1312.6	1.03	1534.1	2.33	1697.7	3.45	3740.2	18.10
LSM7	1336.4	-0.76	1568.3	0.15	1740.5	1.02	3785.1	17.11
N1	1286.4	3.00	1501.7	4.40	1646.8	6.35	2821.0	38.22
N2	1355.2	-2.17	1624.5	-3.42	1801.2	-2.42	3227.2	29.33
E-GLOSS	1365.9	-2.98	1657.9	-5.55	1856.3	-5.56	-N/A- **	-N/A- **
NFEA-2itr	1315.9	0.78	1534.4	2.31	1687.4	4.04	3268.6	28.42
Nonlinear FEA	1326.3		1570.7		1758.5		-4576.8	

*Load Case	Load (kPa)	β
1	1862 (270 psi)	1.7813
2	2069 (300 psi)	2.033
3	2207 (320 psi)	2.2143
4	3103 (450 psi)	3.8866

Note: ** Analysis could not be carried out because of negative modulus update

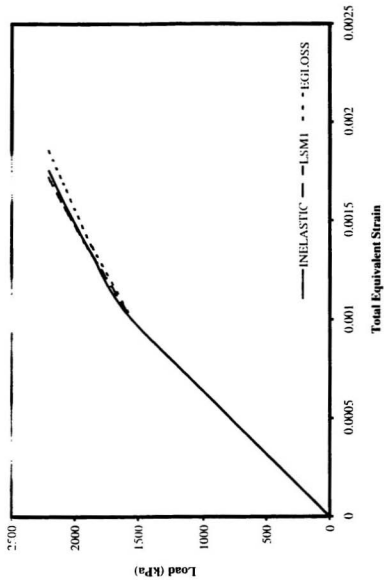


Fig. 5.7c: Inelastic Strain Prediction - Bending of Rectangular Plate

5.7 Thick Cylinder Subjected to Internal Pressure

A considerable amount of research over the decades has been devoted to the design of thick-walled cylinder. In this section, one of the benchmark problems of this kind is selected. Many researchers used this problem for a variety of purposes. It has an inner radius of 76.2 mm (3 in.) and an outer radius of 228.6 mm (9 in.) shown in Fig. 5.8a. Different internal pressures are considered to evaluate the proposed methods. The material has a yield stress of 206.85 MPa (30 ksi), Young's modulus of 206,850 MPa (30,000 ksi) and a Poisson's ratio of 0.3. The thick-walled cylinder is solved using axisymmetric PLANE42 elements. Since the problem is axisymmetric, only a small section is modeled.

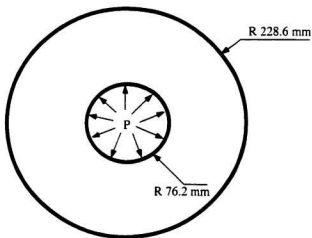


Fig. 5.8a: Thick-Walled Cylinder Subjected to Internal Pressure

It is advisable to use a finer mesh near the inner surface to evaluate inelastic effects. However, for simplicity, the mesh is made uniform through out. The FE modeling of this problem (with and without boundary conditions) is shown in Figs. 5.7b, c and d. The restrains have been applied along the radial direction. The input listings of linear elastic and nonlinear analyses are given in Appendices B.8.1 and B.8.2, respectively.

The limit load for this problem is 261.6 MPa (37.9 ksi) and the load at the first onset of yielding is 108.4 MPa (15.7 ksi) as per ANSYS. Critical strain after the first linear FEA is found at the inner radius. None of the methods have given good results at high load stages. At the lower load of 138 MPa (20 ksi), the error in LSM1, LSM3, LSM4, LSM7 and NFEA-2itr have been found to be less than 6%, while the errors in the EGLOSS and N2 methods are quite high. The EGLOSS and N2 methods to evaluate equivalent inelastic strain for this problem are not very good even at low loads. The evaluation obtained by LSM2 is also not acceptable. All methods over predict the strain for this problem. The predictions obtained by LSM3, LSM7, N1 and NFEA-2itr at all load stages are perhaps better than those of the rest. Except at higher load level, nonlinear analysis after two iterations (NFEA-2itr) shows better prediction for this problem. This behaviour of NFEA-2itr is similar to that in the case of a truss. The behaviour of trusses and thick cylinders have several similarities which might explain the reasons for the performance of NFEA-2itr. As in the case of the flat plate with a hole, for the thick cylinder problem too the EGLOSS method predicts negative moduli at higher loads and thus fails. Representative results for LSM1 and EGLOSS are plotted in Fig. 5.8e.



Fig. 5.8b: Finite Element Model - Thick Cylinder (without Boundary Conditions)



Fig. 5.8c: Finite Element Model - Thick Cylinder (with Boundary Conditions)



Fig. 5.8d: Detail at left side

Table 5.8: Thick Cylinder with Internal Pressure - Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	1483.5	-5.94	2478.1	-18.30	3430.0	-27.33	7512.7	-39.08
LSM 2	1588.1	-13.41	2847.2	-35.91	4264.3	-58.30	10011.9	-85.35
LSM 3	1477.9	-5.54	2402.2	-14.67	3238.9	-20.23	6579.2	-21.80
LSM 4	1481.1	-5.76	2426.7	-15.84	3300.2	-22.51	7547.2	-25.05
LSM7	14781.4	-5.55	2407.6	-14.92	3254.1	-20.79	6649.7	-23.11
N1	1491.9	-6.54	2451.2	-17.01	3276	-21.62	6105.6	-13.04
N2	1655.4	-18.93	2993.8	-42.91	4116.8	-52.82	7850.3	-45.33
E-GLOSS	1735.5	-23.94	3930.5	-87.62	7951.5	-195.18	N/A-**	N/A-**
NFEA-2Iir	1396.6	0.26	2047.6	2.25	2545.1	5.52	4136.2	23.42
Nonlinear FEA	1400.3		2094.8		2693.7		5401.4	

*Load Case	Load (MPa)	β
1	137.9 (20ksi)	1.85
2	172.4 (25ksi)	2.375
3	193.1 (28ksi)	2.926
4	241.4 (35ksi)	4.333

Note: ** Analysis could not be carried out because of negative modulus update

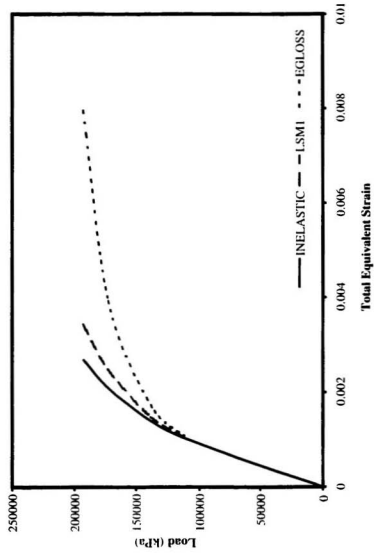


Fig. 5.8e: Inelastic Strain Prediction - Thick Cylinder

5.8 Torispherical Head under Uniform Internal Pressure

Torispherical heads are an important class of problems and a significant amount of research has been devoted them. Drucker and Shield [1959], Mangalaramanan [1997] among others had carried out approximate analyses of torispherical heads. For the present study, a wall thickness of 25.4 mm is used. The ratio of the average diameter of the torispherical head to the thickness is taken to be equal to 300. The yield stress of the material is assumed to be 206.85 MPa and Young's modulus is 206,850 MPa. The Poisson's ratio is 0.3.

Due to symmetry, a quarter model is selected. Four-noded isoparametric quadrilateral elements (PLANE42) used in the FE modeling of other problems described above are used for this problem under axisymmetric condition as well. Six divisions along the thickness are considered. Pressures are applied inside the surface with proper boundary conditions indicated in Figs. 5.9a, b and c. The input files of linear elastic and nonlinear analyses are given in Appendices B.9.1 and B.9.2, respectively.

The pressure at the first onset of yielding is 533.4 kPa and the limit pressure is 858 kPa as per ANSYS. At higher load stages, the local strain prediction given by N1, N2 and EGLOSS methods is not acceptable in comparison to that of the nonlinear FEA. Prediction obtained by LSM1, LSM3, LSM4 and LSM7 at each stage is not bad. They are especially good at low loads although the structure is quite complex. LSM2 is not acceptable, as it gives a huge error even at low loads.

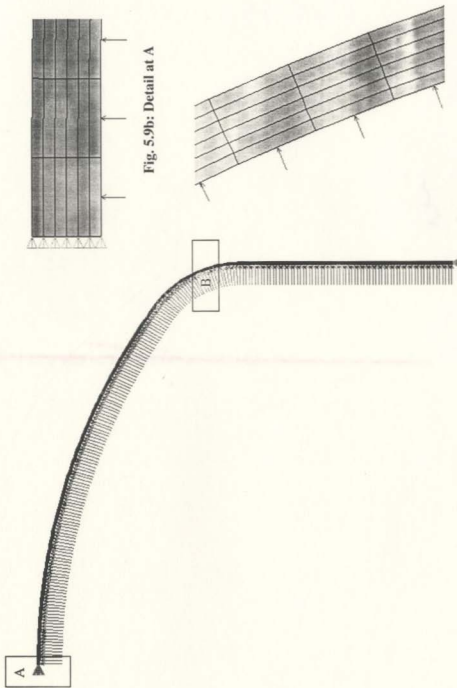


Fig. 5.9a: Finite Element Model - Torispherical Head

Fig. 5.9c: Detail at B

Table 5.9: Torispherical Head with Internal Pressure -Inelastic Strain Prediction at Different Loads

Methods	*Load Case 1		*Load Case 2		*Load Case 3		*Load Case 4	
	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)	Strain (μ)	Error (%)
LSM 1	1183.7	-2.03	1709.6	-4.76	2339.9	-2.49	3444.1	12.83
LSM 2	1206.9	-4.03	1998.6	-22.46	3224.0	-41.22	5536.6	-40.12
LSM 3	1183.6	-2.02	1704.9	-4.46	2312.5	-1.29	3337.7	15.52
LSM 4	1183.7	-2.03	1706.9	-4.59	2330.8	-1.65	3359.5	14.97
LSM7	1183.6	-2.02	1704.9	-3.54	2312.8	-1.30	3338.8	15.49
N1	1173.8	-1.18	1560.8	4.35	1829	19.88	2177.1	44.90
N2	1217.3	-4.92	1750.2	-7.24	2129.7	6.71	2628.7	33.47
E-GLOSS	1222.6	-5.38	1799.9	-10.29	2229.3	2.34	2802.5	29.07
NFEA-2tir	1236.1	-6.55	1789.4	-9.65	2278.9	0.17	3074.9	22.17
Nonlinear FEA	1160.1		1631.9		2282.9		3951.1	

*Load Case	Load (kPa)	β
1	600	1.588
2	700	2.953
3	750	4.5137
4	800	6.4894

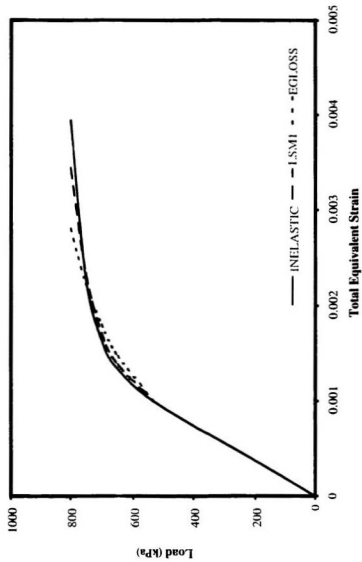


Fig. 5.9d: Inelastic Strain Prediction - Torispherical Head

At a high load of 800 kPa, the estimation given by LSM1 is quite reasonable with a 12% error where as the EGLOSS gives more than 29%. NFEA-2itr gives good prediction at high load (750 kPa). However, this seems to be due to the fact that the error trend changed sign and hence momentarily gave a better result. Representative results for LSM1 and EGLOSS are plotted in Fig. 5.9d.

5.9 Summary

To generalize the proposed techniques, various configurations of structural components with different loading categories and boundary conditions have been solved. The results are compared with those of nonlinear FEA. For the same problem, different load stages are investigated because one method can show its best performance for certain load stage but may not be that good for other load stages. Several times in the analysis, individual methods showed error trends that change signs. When the sign change occurs, the error may appear to be very small. However, this is not reflective of the effectiveness of that method. Therefore, to generalize the conclusions, it is necessary to investigate the methods for a variety of load stages.

In general, it has been observed that LSM1 and LSM7 are the best in comparison with nonlinear FEA at almost all load levels. LSM2 is not acceptable for any problems except those with significant strain concentration such as plates with holes and notch problems. For those cases, LSM2 is better than any other methods studies above. The EGLOSS method is said to be specifically good for low load levels. It is a simple technique and

requires less effort than the line search techniques. However, in general it shows considerably more error than the line search techniques LSM1 and LSM7 for regular problems and LSM2 for strain concentration problems. It has also been seen that NFEA-2itr (response obtained by nonlinear analysis after two iterations when full Newton-Raphson along with line search option as per ANSYS are applied) shows excellent agreement for those load levels just above the yield.

It must be noted that the EGLOSS method is acceptable at low load levels while it becomes unacceptable at higher load stages. In certain cases, the method fails because of the creation of negative modified moduli. For thick cylinders, the error by EGLOSS and N2 is quite high even at low load level, while LSM1 and LSM7 show reasonably good estimation in comparison and are acceptable. Even for such problems as the irregular plate and the torispherical head, the overall prediction obtained by LSM1, and LSM7 is acceptable at almost all load levels. In addition to these, as mentioned earlier, LSM5 and LSM6 have also been studied for the above problems. They show somewhat inconsistent error trends and hence are not reported here in detail. They are not consistently better than LSM7 or LSM1.

Chapter 6

Conclusions and Recommendations

6.1 Summary

Approximate inelastic strain estimation is of great use in several types of applications. Besides classical nonlinear FEA, robust techniques such as Neuber's rule, EGLOSS, etc., are available for this purpose. These robust techniques are applicable for small loads just above the initial yield. These methods find secant modulus based on unbalanced local element energy. They do not account for change in the yield boundary while computing secant modulus. Several traditional secant techniques were developed to update the secant stiffness directly in FEA based nonlinear schemes. The present study explores simple and systematic alternatives for robust determination of inelastic strains based on line search and direct secant modulus. The main concept of these methods is the minimization of the energy due to residual force vector acting on a pseudo displacement after first linear FEA. A line search with the displacements due to the unbalanced forces spreads the yield zone considerably close to the actual state.

In this study, different numerical solution techniques (iterative, incremental or combination of both, etc.) have been reviewed. Incremental procedures are adopted using stiffness matrix with updating co-ordinates and initial displacements using a

number of load steps, and operations are predetermined as a series of linear problems. Errors are likely to accumulate after several steps unless very fine steps are adopted. The solution may therefore, diverge considerably from the true response. The accuracy is improved by applying equilibrium corrections. Pure iterative procedures (without increments) are generally assumed to have converged when the unbalanced load becomes acceptably small, judged by the Euclidean norm. The total load is applied at a time and equilibrium is restored by iteration. Most practical procedures implement a mixed scheme combining the features of both pure iteration and pure incremental procedures. The most popular of such schemes is the well-known Newton-Raphson method. Since in its pure form, it involves updating of stiffness matrix in every iteration of each step, a modified Newton-Raphson method is often used to reduce the number of matrix updates. The conventional incremental procedure is a single iteration of its modified version wherein the unbalanced forces in the previous load increment are neglected.

Secant type methods (e.g., Quasi-Newton) are also very popular since the matrix updates avoid the difficulties associated with the finding of a tangent. However, in general, they are somewhat slower than an optimal Newton strategy (such as Full Newton-Raphson). They become competitive when the cost of Jacobian evaluation is significantly larger than that of the residual vector calculation. In all these, strong nonlinearities could lead to an ill-conditioned iteration matrix. However, for complex material nonlinearities, secant type methods (e.g., BFGS) are preferable. To reduce the iteration numbers significantly, a line search can be applied. Although these were originally developed for secant methods, they can be extended to almost all other types of methods. This line

search has a cost associated with it, but most of the time, it makes the analysis considerably more efficient and effective by reducing the total number of iterations substantially.

Full nonlinear FEA is not always suitable or necessary in several practical cases. Although full nonlinear analysis gives the best representation of structural plasticity, obtaining the solution may be difficult. The process could lead numerical convergence problems and undetectable errors and even to solution instabilities. Thus, it requires the analysis be restarted with necessary modifications made to the geometry, applied loading conditions or the predefined convergence criteria. Besides, the accuracy of the solution obtained depends on the size of load increments taken and the degree of non-linearity of the problem involved. Consequently, there is no guarantee of a numerical solution. Therefore, it is very desirable to have the means for obtaining easy, approximate and yet robust estimates of nonlinear strains.

Such methods will be useful for initial design or feasibility study where several repeated analyses are needed. They are also useful for quick estimating critical strains without the elaborate analytical evaluations for the entire structure. They can also be used as independent checks for full nonlinear analyses results.

Several such techniques have been developed. Most approximate methods involve elastic moduli adjustment techniques that redistribute pseudo-elastic stresses. They have some advantages over conventional nonlinear methods. These methods are fast and efficient. Besides, they are based on a series of linear analyses, and hence avoid convergence

difficulties. However, caution must be exercised while using those methods (EGLOSS, Neuber's Rule, energy density approach, etc.). The EGLOSS method predicts strain reasonably well only at low load levels. On the other hand, Neuber's rule predicts inelastic strains reasonably well for certain applications (e.g., plane stress problems). In plane strain situations, Neuber's rule has been widely reported to overestimate the inelastic strain while energy density approach underestimates them. This could lead to significant errors in fatigue life predictions.

This study is aimed at studying different possibilities and developing a simple set of methods for evaluating the inelastic strain largely based on line search and direct secant method. Line search is concerned with the difference between the size of the plastic zone indicated by the initial elastic analysis and the actual plastic zone. Seven possible techniques (LSM1 to LSM7) are explored to estimate this. They can be applied to most load levels. These techniques involve carrying out an elastic analysis on the original structure for a given loading and boundary conditions. The difference between the applied load and the internal forces corresponding to the reduced stress level constitutes an unbalanced force vector. This unbalanced force needs to be redistributed to the remainder of the structure in order to establish equilibrium with the help of a line search. Using line search, the yield boundary as indicated by the initial analysis can be expanded through redistribution. Previously described robust techniques have not recognized the utility of the line search or its variants in estimating the inelastic zone size. After the line search is carried out, for all elements with equivalent stress exceeding yield stress, the new secant modulus is estimated. After the line search and the estimation of a secant

modulus, a second linear elastic analysis is carried out. The elastic properties of all other elements where the secant modulus is not used are left unchanged in the second FEA.

The results of the first analysis and the second analysis are used to project an approximate relaxation line and hence, a nonlinear strain estimate. For such projections, several combinations of geometric constructions are possible. All those (i.e., geometric combinations) will have some physical justification. All of them are approximate. However, at least some of them will be robust in the sense that they will predict reasonably accurate strains with a relatively small effort and can absorb imperfections in data measurement. Such projections are not recognized by traditional nonlinear FEA based on Newton or Quasi-Newton techniques. However, such projections are used routinely during the estimation of primary stress for mechanical design. EGLOSS and other such methods are based on them. The individual researchers who developed these techniques did not make the combination of line search and geometric projections for relaxation lines. On the basis of first and second analyses, the approximation relaxation locus can be obtained.

Proposed method LSM1 computes the approximate strain at the intersection point of the relaxation locus with the yield stress after the second analysis. Instead of yield stress, a modified yield stress is explored in the LSM2. At the end of the line search, an approximate estimate of equivalent strain can be obtained. Multiplying this strain with the Young's modulus, an elastic stress can be obtained. On this pseudo stress strain point, a relaxation locus can be obtained. The same procedures as in the cases of LSM1 and

LSM2 are repeated to approximate the equivalent strain including the plastic effects in the LSM3 and LSM4, respectively. An approximate estimate of inelastic strain can be obtained at the end of the line search by multiplying a factor. This method is named as LSM5. Another different effective yield stress (akin to EGLOSS) after second elastic analysis can be computed. The same procedures as in the cases of LSM2 and LSM4 are followed to approximate the inelastic strains in the LSM6 and LSM7 (where new effective yield stress level is used), respectively.

The usefulness of Neuber's rule is also explored in this study. This rule is not traditionally used to carry out iterative nonlinear analyses. But the simple concept of the rule can be extended to carry out progressive refinement of the initial analysis results. A second linear analysis is carried out using the moduli modified by Neuber's rule after first linear elastic analysis. An effective yield stress can also be obtained. Two combinations (named as N1 and N2) to evaluate inelastic strains after second linear FEA are explored here. The same procedures as in the cases of LSM1 and LSM2 are repeated in N1 and N2 approaches (where different modified yield level is used), respectively.

All these proposed schemes are applied to study several example problems. They include bending beams (simply supported beam, propped cantilever, fixed beam -all with UDL), bending of rectangular plate (with irregular boundary), simple truss, stretching of a plate with a hole, thick cylinder with internal pressure, thick cylinder with a circumferential notch, and a torispherical shell. These problems were studied for loads ranging from just

above initial yield to nearly limit loads. All results were compared with those obtained by EGLOSS and nonlinear FEA.

6.2 Conclusions

Based on the present study, the following conclusions can be drawn.

1. Secant type and Newton-Raphson based tangent methods are available for nonlinear FEA. All of them are based on incremental iterative procedures and are prone to numerical problems. Their accuracy depends on the size of load increments taken and the degree of non-linearity of the problem involved. All of them update the stiffness matrix directly.
2. Line search significantly reduces the number of iterations. Although the line search involves some extra cost, the analysis becomes significantly more efficient and effective.
3. Robust techniques such as EGLOSS are dependent on direct estimation of secant modulus (as opposed to secant stiffness). They also use relaxation line projections to approximate the inelastic strain. They are reasonably effective at loads just above the yield level. They are very cost effective and are not prone to numerical convergence difficulties.
4. In this study, seven possible techniques (LSM1 to LSM7) based on line search, direct estimation of secant modulus, relaxation line projections are studied. In

addition, two combinations (N1 and N2) based on extensions of Neuber's rule have been studied. All these were applied to a variety of numerical examples. They include beams, truss, plate with hole, cylinder with notch, bending of plate, thick cylinder and torispherical head, etc. They include problems with general bending and stretching as well as problems having strain concentration.

5. LSM1 and LSM7 have been found to give generally similar results and are quite good in giving a robust estimation of inelastic strain for general problems such as bending and stretching. They give consistently better results than other approximate methods (including EGLOSS, etc.) for all load levels. They are also better than full nonlinear analysis (with line search) restricted to two iterations. LSM1 is a little better at higher load levels while LSM7 is slightly better at lower load levels.
6. LSM2 has been found to be the best approximation for strain concentration type problems and maintains an overall consistency in predicting inelastic effects at different load levels. For plate with a central hole, the error for this method is less than 2% throughout (even at 90% of limit load). In these particular types of problems, LSM1 and LSM7 have been also found to be reasonable but not as good as LSM2.
7. All other methods (LSM3, LSM4, LSM5, LSM6, N1, and N2) give trends that are not consistent. While they may be good at certain loads and for certain problems, general trends are difficult to establish.

8. As can be expected, methods based on line search (LSM1/LSM7 or LSM2) consistently fared better than other methods that do not use the line search.
9. It must be noted that some of the techniques examined show very low error for certain load levels. This occurs where the error trends change signs. Thus, a low error level only at a particular load does not necessarily indicate the effectiveness of a certain method.
10. It has been noticed that at larger load levels, using EGLOSS modifications, Young's modulus could become negative thus rendering the method inapplicable. At low load levels just above the yield, EGLOSS has been found to give reasonably good estimations. At larger loads, the discrepancy between the initial analysis yield zone size and the actual yield zone is very significant. This discrepancy increases with increase in loading except in the case of sudden stiffening. Hence, the inelastic strains are not estimated accurately and the error in the estimation increases with increase in the load level. Generally, the compensation obtained by modifying the modulus of elasticity of the yielded elements from the first linear elastic FEA is not enough.

6.3 Recommendations

It has been found that the proposed techniques (LSM1, LSM2 and LSM7) based on line search provide good estimates of inelastic strain for a given load and geometry. Further research in this area would be worthwhile. As LSM2 gives good approximations only for

strain concentration type problems, extension of limitations could be an area worth pursuing. Further research in nonlinear conventional techniques to extend these techniques and implementation of relaxation line after iteration could improve conventional methods.

References

- Adlun, S. M. R., 2001. *Basic Secant Techniques for Limit Loads and Nonlinear Analyses*, Internal Research Report 02-01, Faculty of Engineering, Memorial University of Newfoundland, Canada.
- Adinarayana, N., and Seshadri, R., 1996. "Simplified Thermal Fatigue Evaluations Using the GLOSS Method," *Fatigue and Fracture*, ASME, Vol. 1, pp. 151-163.
- Alfano, G., Rosati, L., and Valoroso, N., 1997. "A Secant-Like Strategy for Finite Element Solutions in Elastoplasticity," *XIII Congresso Nazionale AIMETA*, SIENA, Vol. 3, pp. 145-150.
- Alfano, G., Rosati, L., and Valoroso, N., 1999. "A Tangent-Secant Approach to Rate-Independent Elastoplasticity: Formulations and Computational Issues," *Computer Methods in Applied Mechanics and Engineering*, Vol. 169, pp. 379-405.
- Alwar, R. S., and Babu, S., 1995. "Inelastic Strain Concentration in Directionally Solidified Materials," *Transactions of the CSME*, Vol. 19, No. 3, pp. 331-346.
- ANSYS, 1998. *ANSYS Engineering Analysis System User's Manual*, Release 5.5, Swanson Analysis Systems Inc., Houston, Pa.
- Babu, S., and Iyer, P. K., 1999. "A Robust Method for Inelastic Analysis of Components Made of Anisotropic Material," *Journal of Pressure Vessel Technology*, ASME, Vol. 121, pp. 154-159.
- Babu, S., and Iyer, P. K., 1998. "Inelastic Analysis of Components Using a Modulus Adjustment Scheme," *Journal of Pressure Vessel Technology*, ASME, Vol. 120, pp. 1-5.
- Barnes, J. G. P., 1965. "An Algorithm for Solving Nonlinear Equations Based on the Secant Method," *Computer Journal*, Vol. 8, pp. 66-72.
- Bathe, K. J., and Dvorkin, E. N., 1983. "On the Automatic Solution of Nonlinear Finite Element Equations," *Computers & Structures*, Vol. 17, No. 5-6, pp. 871-879.
- Bathe, K. J., 1996. *Finite Element Procedures*, Prentice-Hall, Englewood Cliffs, New Jersey, USA.
- Batoz, J. L., and Dhatt, G., 1979. "Incremental Displacement Algorithms for Nonlinear Problems," *International Journal for Numerical Methods in Engineering*, Vol. 14, No.8, pp. 1262-1267.

- Bergan, P. G., and Clough, R. W., 1972. "Convergence Criteria for Iterative Processes," *AIAA Journal*, Vol. 10, pp. 1107-1108.
- Bird, W. W., and Martin, J. B., 1985. "Secant Approximation for Holonomic Elastic Plastic Incremental Analysis with von-Mises Yield Condition," *Conference on Finite Element Methods in South Africa*, pp. 233-268.
- Bolar, A., 2002. *Robust Estimation of Limit Loads of Plates Using Secant Rigidity*, Master's Thesis, Faculty of Engineering and Applied Science, Memorial University of Newfoundland, Canada.
- Boyle, E. F., and Jennings, A., 1972. "Accelerating the Convergence of Elastic-Plastic Stress Analysis," *International Journal for Numerical Methods in Engineering*, Vol. 7, No. 2, pp. 232-235.
- Broyden, C. G., 1970. "The Convergence of a Class of Double-Rank Minimization Algorithms," *Institute of Mathematics and its Applications*, Vol. 6, pp. 76-90.
- Chen, C. N., 1990. "Improved Constant Stiffness Algorithms for the Finite Element Analysis," *International Conference on Numerical Methods in Engineering: Theory and Applications*, Swansea, Wales, pp. 566-573.
- Chen, C. N., 1992. "An Acceleration Method in Elastic-Plastic Finite Element Computation," *Computers & Structures*, Vol. 44, No. 1/2, pp. 125-132.
- Chen, W. F., and Sohal, I., 1995. *Plastic Design and Second-Order Analysis of Steel Frames*, Springer-Verlag.
- Crisfield, M. A., 1979. *Iterative Solution Procedures for Linear and Nonlinear Structural Analysis*, TRRL Laboratory Report 900, Structures Department, Transport and Road Research Laboratory, Crowthorne, Berkshire.
- Crisfield, M. A., 1980. "Incremental/Iterative Solution Procedures for Nonlinear Structural Analysis," *Proceedings of the International Conference*, University College Swansea, pp. 261-290.
- Crisfield, M. A., 1981. "A Fast Incremental/Iterative Solution Procedure that Handles the Snap-Through," *Computers & Structures*, Vol. 13, pp. 55-62.
- Crisfield, M. A., 1982. *Variable Step Lengths for Nonlinear Structural Analysis*, TRRL Laboratory Report 1049, Structures Department, Transport and Road Research Laboratory, Crowthorne, Berkshire.

- Crisfield, M. A., 1983. "An Arc-Length Method Including Line Searches and Accelerations," *International Journal for Numerical Methods in Engineering*, Vol.19, No. 9, pp.1269-1289.
- Crisfield, M. A., 1984. "Accelerating and Damping the Modified Newton-Raphson Method," *Computers & Structures*, Vol. 18, No. 3, pp. 395-407.
- Crisfield, M. A., 1991. *Nonlinear Finite Element Analysis of Solids and Structures*, Volume-I, John Wiley & Sons, England.
- Dennis, J. E. JR., and More, J. J., 1977. "Quasi-Newton Methods, Motivation and Theory," *SIAM REVIEW*, Vol. 19, No. 1, pp. 46-89.
- Desikan, V., and Sethuraman, R., 2000. "Analysis of Material Nonlinear Problems Using Pseudo-Elastic Finite Element Method," *Journal of Pressure Vessel Technology*, ASME, Vol. 122, pp. 457-461.
- Dhalla, A. K., 1984. "Verification of an Elastic Procedure to Verify Follow-up," *Design of Elevated Temperature Piping*, ASME, New York, pp. 81-96.
- Dhalla, A. K., and Jones, G. L., 1986. "ASME Code Classification of Pipe Stresses: A Simplified Elastic Procedure," *International Journal of Pressure Vessel and Piping*, Vol. 26, pp. 145-166.
- Dhalla, A. K., 1987. "A Simplified Procedure to Classify Stresses for Elevated Temperature Service," *Proceedings of the ASME-PVP Conference*, San Diego, Vol. 120, pp. 177-188.
- Elsawaf, A. F. M. Y., 1979. *The Conjugate-Newton Method for Nonlinear Finite Element Problems*, Ph. D. Thesis, Department of Civil Engineering, University of Calgary, Alberta, Canada
- Fletcher, R., and Powell, M. J. D., 1963. "A Rapidly Convergent Descent Method for Minimization," *Computer Journal*, Vol. 6, pp. 163-168.
- Garcia, J. A., 1998. "Evaluation of the Newton Secant Method in Hydraulic Design Applications," *International Water Resources Engineering Conference*, Memphis, Tenn., pp. 1840-1851.
- Geradin, M., Idelsohn, S., and Hogge, M., 1981. "Computational Strategies for the Solution of Large Nonlinear Problem via Quasi-Newton Methods," *Computers & Structures*, Vol. 13, pp. 73-81.

- Glinka, G., 1985. "Energy Density Approach to Calculation of Inelastic Strain-Stress Near Notches and Crack." *Engineering Fracture Mechanics*, Vol. 22, No. 3, pp. 485-508.
- Haisler, H. W., Stricklin, J. A., and Stebbins, F. J., 1972. "Development and Evaluation of Solution Procedures for Geometrically Nonlinear Structural Analysis," *AIAA Journal*, Vol. 10, pp.264-272.
- Haisler, H. W., Stricklin, J. A., and Key, J. E., 1977. "Displacement Incrementation in Nonlinear Structural Analysis by the Self-Correcting Method," *International Journal for Numerical Methods in Engineering*, Vol. 11, No. 1, Pp.3-10.
- Henshell, R. D., and Shaw, K. G., 1975. "Crack Tip Finite Elements are Unnecessary," *International Journal for Numerical Methods in Engineering*, Vol. 9, pp. 495-507.
- Hoffman and Seeger, P., 1985. "Multiaxial Elasto-Plastic Stress Strain Relationships," *Journal of Testing and Evaluation*, Vol. 10, No. 4, pp. 199-204.
- Hu, P., Liu, Y., and Cheng, S., 1992. "Improvement of the Common Elastic-Plastic FEM and Hybrid/Mixed Quasi-Newton Modulus Method," *Computers & Structures*, Vol. 44, No. 5, pp. 1073-1081.
- Jahed, H., Sethuraman, R., and Dubey, R.N., 1997. "A Variable Material Property Approach for Solving Elastic-Plastic Problems," *International Journal of Pressure Vessel & Piping*, Vol. 71, pp. 285-291.
- Jeeves, T. A., 1958. "Secant Modification of Newton's Method," *Association for Computing Machinery and Communication*, Vol. 1, No. 8, pp. 9-10.
- Kao, R., 1974. "A Comparison of Newton-Raphson Methods and Incremental Procedures for Geometrically Nonlinear Analysis," *Computers & Structures*, Vol. 4, pp. 1091-1097.
- Knop, M., Jones, M, Molent, L. and Wang, C., 2000. "On the Glinka and Neuber Methods for Calculating Notch Tip Strains under Cyclic Load Spectra," *International Journal of Fatigue*, Vol. 22 Pp.743-755.
- Kizhatil, K. R., and Seshadri, R., 1994. "Multiaxial Stress Relaxation Using the Local Region Constraint Parameter," *International Journal of Pressure Vessel & Piping*, Vol. 63, pp. 99-110.
- Kowalczyk, M., and Bojczuk, D., 1996. "Nonlinear Incremental Analysis of Loading and Design Sensitivity Problems," *Mechanics of Structures and Machines*, Vol. 24, pp. 331-360.

- Logan, D. L., 1992. *A First Course in the Finite Element Method*. PWS Publishing Company, Boston.
- Lee, L. S., and Basu, P. K., 1989. "Secant Method for Nonlinear Semi-Rigid Frames," *J. Construct. Steel Research*, Vol. 14, pp. 273-299.
- Mackenzie, D. and Boyle, J. T., 1993. "A Method of Estimating Limit Loads by Iterative Elastic Analysis. I-Simple Examples," *International Journal of Pressure Vessel & Piping*, Vol. 53, pp. 77-95.
- Mangalaramanan, S., P., 1997. *Robust Limit Loads Using Elastic Modulus Adjustment Techniques*, Ph.D. Thesis. Faculty of Engineering and Applied Science, Memorial University of Newfoundland, Canada.
- Marriot, D. L., 1988. "Evaluation of Deformation or Load Control of Stresses under Inelastic Conditions Using Elastic Finite Element Stress Analysis," *Proceedings of the ASME-PVP Conference*, Pittsburgh, Vol. 136.
- Masset, D. A., and Stricklin, J. A., 1971. "Self-Correcting Incremental Approach in Nonlinear Structural Mechanics," *AIAA Journal*, Vol.9, No. 12, pp. 2464-2466
- Matthies, H., and Strang, G., 1978. "The Solution of Nonlinear Finite Element Equations," *International Journal for Numerical Methods in Engineering*, Vol.14, pp. 1613-1626.
- Martinez, M. J., 2000. "Practical Quasi-Newton Methods for Solving Nonlinear Systems," *Journal of Computational and Applied Mathematics*, Vol. 124, pp. 97-121.
- Molski, K. and Glinka, G., 1981. "A Method of Elastic-Plastic Stress and Strain Calculation at a Notch Root," *Material Science and Engineering*, Vol. 50, pp. 93-100.
- Nayak, G. C., and Zienkiewicz, O. C., 1972. "Note on the 'Alpha'-Constant Stiffness Method for the Analysis of Nonlinear Problems," *International Journal for Numerical Methods in Engineering*, Vol. 4, pp. 579-582.
- Neuber, H., 1961 "Theory of Stress Concentration for Shear-Strained Prismatical Bodies with Arbitrary Nonlinear Stress-Strain Laws," *Journal of Applied Mechanics*, ASME, Vol. E28, pp. 554-550.
- Owen, D. R. J., and Gomez, C. M. B., 1981. "An Appraisal of Numerical Solution Techniques for Elasto-Plastic and Elasto-Viscoplastic Material Problems," *International Conference on Structural Mechanics in Reactor Technology*, Paris, France.

- Powell, G., and Simons, J., 1981. "Improved Iteration Strategy for Nonlinear Structures." *International Journal for Numerical Methods in Engineering*, Vol.17, Pp.1455-1467.
- Raghavan, P., and Seshadri, R., 1997. "Simplified Inelastic Analysis of Notched Components Subjected to Mechanical and Thermal Loads." *Fatigue and Fracture*, ASME, Vol. 1, pp. 425-432.
- Raghavan, P., 1998. *Simplified Inelastic Analysis of Notched Components Subjected To Mechanical and Thermal Loads*, Master's Thesis, Faculty of Engineering and Applied Science, Memorial University of Newfoundland, Canada.
- Ralph, F. E., 2000. *Robust Methods of Finite Element Analysis: Evaluation of Nonlinear, Lower Bound Limit Loads of Plated Structures and Stiffening Members*, Master's Thesis, Faculty of Engineering and Applied Science, Memorial University of Newfoundland, Canada.
- Ramm, E., 1982. "The Riks/Wempner Approach-An Extension of the Displacement Control Method in Nonlinear Analysis." *Recent Advances in Nonlinear Computational Mechanics*, University College of Swansea, U.K. pp. 63-86.
- Risk, E., 1979. "An Incremental Approach to the Solution of Snapping and Buckling Problems," *International Journal of Solids Structures*, Vol. 15, pp. 529-551.
- Roberts, T. M., and Ashwell, D. G., 1971. "The Use of Finite Element Mid-Increment Stiffness Matrices in the Post-Buckling Analysis of Imperfect Structures," *International Journal Solids & Structures*, Vol. 7, pp. 805-823.
- Schmidt, W. F., 1978. "Adaptive Step Size Selection for Use with the Continuation Method," *International Journal for Numerical Methods in Engineering*, Vol. 12, pp. 677-694.
- Schweizerhof, K. H., and Wriggers, P., 1986. "Consistent Linearization for Path Following Methods in Nonlinear Finite Element Analysis," *Computer Methods in Applied Mechanics and Engineering*, Vol. 49, pp. 261-279.
- Seeger, T. and Heuler, P., 1980. "Generalized Application of Neuber's Rule," *Journal of Testing and Evaluation*, Vol. 8, No. 4, pp. 199-204.
- Seshadri, R., 1990. "The Effect of Multiaxiality and Follow-Up on Creep Damage," *Journal of Pressure Vessel Technology*, ASME, Vol. 112, pp. 378-385.
- Seshadri, R., 1991. "The Generalized Local Stress Strain (GLOSS) Analysis- Theory and Applications," *Journal of Pressure Vessel Technology*, ASME, Vol. 113, pp. 219-227.

- Seshadri, R., and Fernando, C. P. D., 1992. "Limit Loads of Mechanical Components and Structures Using the GLOSS R-Node Method," *Journal of Pressure Vessel Technology*, ASME, Vol. 114, pp. 201-208.
- Seshadri, R., and Kizhatil, K. R., 1993. "Notch Root Inelastic Strain Estimates Using GLOSS Analysis," *Journal of Pressure Vessel Technology*, ASME, pp. 397-411
- Seshadri, R., 1995. "Inelastic Evaluation of Mechanical and Structural Components Using Generalized Local Stress Strain Method of Analysis," *Nuclear Engineering and Design*, Vol. 153, pp. 287-303.
- Seshadri, R., and Babu, S., 2000. "Extended GLOSS Method for Determining Inelastic Effects in Mechanical Components and Structures: Isotropic Materials," *Journal of Pressure Vessel Technology*, ASME, Vol. 122, pp. 413-420.
- Severud, L. K., 1984. "Simplified Evaluation for Piping Elastic Follow-Up," *Proceedings of the 5th International Congress of Pressure Vessel Technology*, ASME, San Francisco, pp. 367-387.
- Shames, I., H., and Cozzarelli, F. A., 1992. *Elastic and Inelastic Stress Analysis*, Prentice Hall, Inc., NJ.
- Stricklin, J. A., Haisler, W. E., and Riesenmann, W. A. V., 1971. "Self-Correcting Initial Value Formulations in Nonlinear Structural Mechanics," *AIAA Journal*, Vol. 9, pp. 2000-2001.
- Stricklin, J. A., Haisler, W. E., and Riesenmann, W. A. V., 1973. "Evaluation of Solution Procedures for Material and/or Geometrically Nonlinear Structural Analysis," *AIAA Journal*, Vol. 8, pp. 292-299.
- Stricklin, J. A., and Haisler, W. E., 1977. "Formulations and Solution Procedures for Nonlinear Structural Analysis," *Computers & Structures*, Vol. 7, pp. 125-136
- Thomas, G. R., 1973. "A Variable Step Incremental Procedure," *International Journal for Numerical Methods in Engineering*, Vol. 7, pp. 563-566.
- Ugural, A. C., and Fenster, S. K., 1987. *Advanced Strength and Applied Elasticity*, Elsevier, Second SI Edition.
- Wellford, L. C., and Sen, H., 1981. "A Selective Relaxation Iterative Solution Technique for Nonlinear Structural Analysis Problems," *International Journal for Numerical Methods in Engineering*, Vol. 17, pp. 773-795.
- Wolfe, P., 1959. "The Secant Method for Simultaneous Nonlinear Equations," *Communications of the ACM*, Vol. 2, pp. 12-13.

- Zienkiewicz, O. C., Valliappan, S., and King, I. P., 1969. "Elasto-Plastic Solutions of Engineering Problems - 'Initial Stress' Finite Element Method," *International Journal for Numerical Methods in Engineering*, Vol. 1, pp. 75-100.
- Zienkiewicz, O. C., 1971. "Incremental Displacement in Nonlinear Analysis," *International Journal for Numerical Methods in Engineering*, Vol. 3, pp. 587-592.
- Zienkiewicz, O. C., and Gallagher, R. H., 1998. "A General Procedure for Deriving Symmetric Expressions for the Secant and Tangent Stiffness Matrices in Finite Element Analysis," *International Journal for Numerical Methods in Engineering*, Vol. 42, pp. 219-236.
- Zhang, L., and Owen, D. R. J., 1982. "A Modified Secant Newton Method for Nonlinear Problems," *Computers & Structures*, Vol. 15, No. 5, pp. 543-547.

Appendix A

ANSYS Files for Proposed Techniques

All ANSYS input and commands listing for the problems discussed in Chapter 5 are provided in the following Appendices. As mentioned earlier in Chapter 5, ANSYS FE software does not perform line searches after the first linear elastic analysis. The algorithm used by ANSYS seems to need at least two analyses to establish search directions and meaningful updates. However, basic line search can always be carried out irrespective of the condition of the analysis. In order to carry out the line search outside the purview of ANSYS, a set of procedures reflecting the basic ideas of line search parameters have been developed. These procedures are purely for the ease of carrying out line search outside ANSYS and do not in any way influence the effectiveness (or the lack thereof) of the methods being discussed. If one were to program these methods directly, these extra procedures need not be resorted to. All these extra procedures have been programmed using the ANSYS Design Parametric Language (ADPL) provided with the ANSYS package [ANSYS, 1998]. The language is somewhat similar to Fortran and can be easily understood. Sample macros involving the extra programming are included in this Appendix. If one were to use another software package, say, ABAQUS, the system of procedures would change. Most of the procedures reported below would not be required if the particular software package is equipped with commands to supply the

decomposed matrix (or the matrix inverse) or is programmed to carry out the line search after the first linear analysis.

A complete set of analysis files for the simply supported beam with UDL has been listed in the present appendix. Similar files for other problems are not listed since they are similar to those listed here except for the modeling input.

In the following, section A.1 lists the input data to perform the first linear elastic analysis. It stores stresses and strains as output in the file 'stress_strain_1'. A file 'd_val' is created to calculate nodal displacements for a duplicate analysis. Section A.2 lists a duplicate input data. The unbalanced forces at the nodes are output to the file 'f_val1-f_valn' for further processing. The applied nodal displacements in a different format are also restored in the file 'd_val1' for a link file to be used in section A.4. Listing A.3 gives the nodal displacements due to the unbalanced loads written in the file 'f_val1-f_valn'. Section A.4 lists a simple Fortran routine to compute line search parameter β . This routine and the APDL listing in A.5 are run repeatedly till the parameter β is determined. Section A.5 has a listing for nonlinear analysis to find out the equivalent balance forces (compiled in the file 'f_vali') for an applied displacement field (compiled in the file 'd_sum') obtained by using the current value of the line search parameter β . After the line search has converged, modification of the Young's modulus is carried out and the input file to carry out linear elastic analysis is listed in A.6. After running A.6, line search stresses and strains are stored in the file 'stress_strain_1s'. A second linear elastic

FEA with modified moduli is carried out using the listing in A.7. It stores the stresses and strains after the second linear analysis in the file 'stress_strain_2'.

As mentioned above, this pattern of procedures is similar to all the example problems.

A.1 First Linear Elastic Analysis

```
/batch
```

```
/title, simply supported beam with udl
```

```
! set basic material constants
```

```
*set, ym, 30e06 ! Young's modulus
```

```
*set, ys, 30e03 ! yield stress
```

```
*set, poisson, 0.3 ! Poisson's ratio
```

```
! set basic geometric inputs
```

```
*set, ln, 20 ! beam span
```

```
*set, dt, 1 ! beam depth
```

```
*set, ndiv1, 60 ! no of divisions along beam span
```

```
*set, ndiv2, 10 ! no of divisions along beam depth
```

```
*set, pr, 120 ! applied udl on the beam surface
```

```
! enter preprocessor
```

```
/prep7
```

```
! define analysis type as 'static' and
```

```
! element type as 'four-noded isoparametric' element
```

```
antype, 0
```

```
et, 1, 42
```

```
! add elastic material properties
```

```
mp, ex, 1, ym
```

```
mp, nuxy, 1, poisson
```

```
key, 1, 3, 0
```

```
! start modeling and defining keypoints
```

```
k, 1
```

```
k, 2, ln
```

```

k, 3, ln, dt
k, 4, 0, dt

! divide lines
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define areas
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions at left support
nset, s, loc, x, 0
nset, r, loc, y, dt/2
d, all, ux
d, all, uy

! add boundary conditions at right support
nset, s, loc, x, ln
nset, r, loc, y, dt/2
d, all, uy

! add udl on the beam surface
nset, s, loc, y, dt
sf, all, pres, pr
nset, all

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

```



```

! end of solving, and exit solution routine
fini

! enter post-processor
/post1

set, 1

! create element tables for equivalent stresses and strains
etable, eqvst, s, eqv
etable, estrn, epel, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, num, max

! create arrays as 'dummy1' and 'dummy2'
*dim, dummy1, array, max1
*dim, dummy2, array, max1

! open a file as 'stress_strain_1' and
! store first linear stresses and strains in corresponding arrays
*cfdopen, stress_strain_1
*do, kk, 1, max1
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*set, dummy1(kk), sig
*set, dummy2(kk), epsl/(1+poisson)
*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vwrite, kk, dummy1(kk), dummy2(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos

fini

! get maximum node number as 'max2'
*get, max2, node, 0, num, max

! create arrays as 'dummy3' and 'dummy4'
*dim, dummy3, array, max2
*dim, dummy4, array, max2

! set material number as 'mnum'
*set, mnum, 1

```

```

! open a file as 'd_val' and store nodal displacements
*cfopen, d_val
*do, kk, 1, max2
*get, d1, node, kk, u, x
*get, d2, node, kk, u, y
*set, dummy3(kk), d1
*set, dummy4(kk), d2
*cfwrite, d, mnum, ux, dummy3(kk)
*cfwrite, d, mnum, uy, dummy4(kk)
*set, mnum, mnum+1
*enddo
*cfclos

fini

```

A.2 Calculation of Unbalanced Nodal Forces

```

/batch

/title, calculation of unbalanced nodal forces

! set the same basic material constants and geometric inputs
*set, ym, 30e06
*set, ys, 30e03
*set, poisson, 0.3
*set, ln, 20
*set, dt, 1
*set, ndiv1, 60
*set, ndiv2, 10

! enter preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric' element
antype, 0
et, 1, 42

! define elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

key, 1, 3, 0

```

```

! start modeling
! define keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! devide lines
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define areas
a, 4, 1, 2, 3

! start meshing
amesh, all

! select all nodes of the model and
! apply equivalent nodal displacements instead of udl
nset, all
/inp, d_val
nset, all

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
outres, all, all
save

! start solving
solve

! end of solving, and exit solution routine
fini

! enter post-processor
/post1

set, 1

```

```

! get maximum node number as 'max1'
*get, max1, node, 0, num, max

! create arrays as 'dummy1', 'dummy2', etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'd_val1' and
! store applied nodal displacements in a different format
*cfdopen, d_val1
*do, kk, 1, max1
*get, d1, node, kk, u, x
*get, d2, node, kk, u, y
*set, dummy1(kk), d1
*set, dummy2(kk), d2
*cfdwrite, 1, mnum, dummy1(kk)
*cfdwrite, 2, mnum, dummy2(kk)
*set, mnum, mnum+1
*enddo
*cfclos

fini

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'f_val1' and
! store nodal forces (reaction forces) in a different format
*cfdopen, f_val1
*do, kk, 1, max1
*get, a1, node, kk, rf, fx
*get, a2, node, kk, rf, fy
*set, dummy3(kk), a1
*set, dummy4(kk), a2
*cfdwrite, 1, mnum, dummy3(kk)
*cfdwrite, 2, mnum, dummy4(kk)
*set, mnum, mnum+1
*enddo
*cfclos

save

```

```

fini

! start another inelastic analysis

! enter preprocessor
/prep7

! the same model and applied displacements are restored
resume

! define inelastic properties
tb, bkin, 1, 1
tbdata, 1,ys, 0

! nonlinear solver options
autots, on
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
outres, all, all
save

! start solving
solve

! end of solveing, and exit solution routine
fini

! enter post-processor
/post1

set, last

! get maximum node number as 'max2'
*get, max2, node, 0, num, max

! create arrays as 'dummy5' and 'dummy6'
*dim, dummy5, array, max2

```

```

*dim, dummy6, array, max2

! set material number as 'mnum'
*set, mnum, 1

! open as file as 'f_valn' and
! store balanced nodal forces in a different format
*cfopen, f_valn
*do, kk, 1, max2
*get, a3, node, kk, rf, fx
*get, a4, node, kk, rf, fy
*set, dummy5(kk), a3
*set, dummy6(kk), a4
*cfwrite, 1, mnum, dummy5(kk)
*cfwrite, 2, mnum, dummy6(kk)
*set, mnum, mnum+1
*enddo
*cfclos
save

fini

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'f_val1-f_valn' and store unbalanced nodal forces
*cfopen, f_val1-f_valn
*do, kk, 1, max2
*cfwrite, f, mnum, fx, dummy3(kk)-dummy5(kk)
*cfwrite, f, mnum, fy, dummy4(kk)-dummy6(kk)
*set, mnum, mnum+1
*enddo
*cfclos

fini

```

A.3 Calculation of Nodal Displacements

```

/batch

/title, calculation of nodal displacements due to unbalanced nodal forces

! set the same basic constants and inputs

```

```

*set, ym, 30e06
*set, ys, 30e03
*set, poisson, 0.3
*set, ln, 20
*set, dt, 1
*set, ndiv1, 60
*set, ndiv2, 10

! enter preprocessor
/prep7

! define analysis and element type
antype, 0
et, 1, plane42

! define elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

key, 1, 3, 0

! start modeling
! define keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! divide lines
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define areas
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions at left support
nset, s, loc, x, 0
nset, r, loc, y, dt/2
d, all, ux
d, all, uy

```

```

! add boundary conditions at right support
nset, s, loc, x, ln
nset, r, loc, y, dt/2
d, all, uy

! select all nodes of the model and
! apply unbalanced nodal forces
nset, all
/inp, f_val1-f_valn
nset, all

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
outres, all, all
save

! start solving
solve

! end of solving and exit solution routine
fini

! enter post-processor
/post1

set, 1

! get maximum node number as 'max1'
*get, max1, node, 0, num, max

! create arrays as 'dummy1' and 'dummy2'
*dim, dummy1, array, max1
*dim, dummy2, array, max1

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'del_d' and
! store displacements due to unbalanced forces
*cfopen, del_d

```



```

*do, kk, 1, max1
*get, d1, node, kk, u, x
*get, d2, node, kk, u, y
*set, dummy1(kk), d1
*set, dummy2(kk), d2
*cfwrite, 1, mnum, dummy1(kk)
*cfwrite, 2, mnum, dummy2(kk)
*set, mnum, mnum+1
*enddo
*cfclos

fini

```

A.4 Calculation of Line Search Parameter β

```

*****
* obtain the applied force at each dof as a vector: force1
* obtain the balanced force sustained by the updated displacement: force_i
* obtain the displacement due to initial unbalanced force disp_d
* find the error energy
* change  $\beta$  value and find the error energy again
* repeat the process to minimize the error
*****

*   character*80 fname
    dimension errorval(100), betaval(100), changeval(100)

*****
* the following is to prepare post-processing data
* the same file will also be used after rewinding
* to create a file containing segments of input for the reanalysis
*****

* output file containing post processing data

    open (unit=06,file='d_val1')
    open (unit=07,file='del_d')
    open (unit=08,file='d_sum')
    open (unit=09,file='beta')
    open (unit=10,file='f_val1')
    open (unit=11,file='f_vali')

```

```

        error = 0.0
* error is the residual energy

        beta_tol = 0.01
* tolerance for convergence of residual energy

        node_dof = 2
* node_dof is the number of degrees of freedom per node

        read (09,*) nodes, no_trials
* nodes is total number of nodes
* no_trials is the number of trial iteration carried out
* to find the value of beta, initially the file beta will
* have initial trial beta value as zero. i.e.,
* the beta file will have an input that looks like
* 'number of nodes', 0

        if ( no_trials .ne. 0) then

            read (09,*) (errorval(i), betaval(i), changeval(i), i=1, no_trials)

            do 200 i = 1, node_dof*nodes

                read (07,*) idir_d, n_d, disp_d
                read (10,*) idir1, n1, force1
                read (11,*) idir_i, ni, force_i

                error = error + disp_d * (force_i - force1)

200      continue

            change = -100.0*(errorval(no_trials)-error)/errorval(no_trials)

            if ( abs(change) .le. beta_tol .and. no_trials .ne. 1 ) then

                write (05,300) beta_tol
300      format("beta value converged with a tolerance of ", F6.4)

                stop
            end if

            write (05,360)
360      format (1x," trial#  error value  beta      %change ")

```

```

write (05,370)(i, errorval(i), betaval(i), changeval(i), i=1, no_trials)
370 format (1x,3x,I2.3x,E13.3,2x,F6.3,2x,F9.3)

```

```

write (05,380) error, change
380 format (1x,"the current values of error and change ratio are",
          /9x,E13.3,2x,6x,2x,F9.3/)

```

```

else

```

```

error = 10000000000.0
change = 1000000000.0

```

```

end if

```

```

write (05,400)
400 format(1x,"please type in the next guess for beta: "S)

```

```

read (05,*) beta
no_trials = no_trials+1
errorval(no_trials) = error
betaval(no_trials) = beta
changeval(no_trials)= change

```

```

close(unit=09)
open (unit=09,file='beta')

```

```

write (09,*) nodes, no_trials
write (09,420)(errorval(i), betaval(i), changeval(i), i=1, no_trials)
420 format (1x,E13.3,2x,F7.3,2x,F15.3)

```

```

*****

```

```

* the following is to prepare a file containing total displacements
* of input for the reanalysis
* this utilizes the post-processing data files created earlier
* and the beta value calculated above

```

```

*****

```

```

rewind(unit=06)
rewind(unit=07)

```

```

do 1000 i = 1, node_dof*nodes

```

```

read (06,*) idirection1, n1, disp1
read (07,*) idirection2, n2, disp2

disp = disp1 + beta * disp2

if (idirection1 .eq. 1) then

  write (08,510) n1, disp
510  format ('d','l4','.ux','e18.9)

  else

    write (08,520) n1, disp
520  format ('d','l4','.uy','e18.9)

  end if

1000 continue

close (unit=06)
close (unit=07)
close (unit=08)
close (unit=09)
close (unit=10)
close (unit=11)

stop
end

```

A.5 Calculation of Balanced Nodal Forces

```
/batch
```

```
/title, calculation of balanced nodal forces corresponding nodal displacements
```

```

! set the basic material and geometric constants and inputs
*set, ym, 30e06
*set, ys, 30e03
*set, poisson, 0.3
*set, ln, 20

```

```

*set, dt, 1
*set, ndiv1, 60
*set, ndiv2, 10

! enter preprocessor
/prep7

! define analysis and element type
antype, 0
et, 1, plane42

! define material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! define nonlinear properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

key, 1, 3, 0

! start modeling by defining keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! divide suitable lines
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define areas
a, 4, 1, 2, 3

! start meshing
amesh, all

! select all nodes and
! apply corresponding nodal displacements
nset, all
/inp, d_sum
nset, all

```

```

! nonlinear solver options
autots, on
lnsrch, on
nropt, full, . off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
outres, all, all
save

! begin solving
solve

! end of solving, and exit solution routine
fini

! enter post-processor
/post1

set, 1

! get total node number as 'max1'
*get, max1, node, 0, num, max

! create arrays as 'dummy1' and 'dummy2'
*dim, dummy1, array, max1
*dim, dummy2, array, max1

! set material number
*set, mnum, 1

! open a file as 'f_vali' and
! store balanced nodal forces in a different format for line search
*cfopen, f_vali
*do, kk, 1, max1
*get, a1, node, kk, rf, fx
*get, a2, node, kk, rf, fy
*set, dummy1(kk), a1
*set, dummy2(kk), a2

```

```

*cfwrite, 1, mnum, dummy1(kk)
*cfwrite, 2, mnum, dummy2(kk)
*set, mnum, mnum+1
*enddo
*cfclos
save

fini

```

A.6 Modification of Young's Modulus

```

/batch

/title, modification of Young's modulus after line search

! set the same basic constants and inputs
*set, ym, 30e06
*set, ys, 30e03
*set, poisson, 0.3
*set, ln, 20
*set, dt, 1
*set, ndiv1, 60
*set, ndiv2, 10

! enter preprocessor

/prep7

! set analysis type as 'static' and
! element type as 'four-noded isoparametric (plane42)' element
antype, 0
et, 1, 42

! set material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

key, 1, 3, 0

! start modeling by setting keypoint
k, 1
k, 2, ln
k, 3, ln, dt

```

```

k, 4, 0, dt

! devide lines suitable for meshing
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3,4, ndiv1
l, 4, 1, ndiv2

! define areas
a, 4, 1, 2, 3

! start meshing
amesh, all

! select all nodes and
! apply the nodal displacement (obtained by line search)
nsel, all
/inp, d_sum
nsel, all

! end of modeling and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
outres, all, all
save

! start solving
solve

! end of solving, and exit solution routine
fini

! enter post-processor
/post1

set, 1

! create tables as 'eqvst' and 'estrn'
etable, eqvst, s, eqv
etable, estrn, epel, eqv

! get total element number as 'max1'

```



```

*get, maxl, elem, 0, num, max

! create arrays as 'dummy1' and 'dummy2'
*dim, dummy1, array, maxl
*dim, dummy2, array, maxl

! open a file as 'stress_strain_ls' and
! store the stresses and strains in corresponding arrays (after line search)
*cfcopen, stress_strain_ls
*do, kk, 1, maxl
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*set, dummy1(kk), sig
*set, dummy2(kk), epsl/(1+poisson)
*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vwrite, kk, dummy1(kk), dummy2(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos

fini

! select all elements of the model
esel, all

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'ym_val' and update the Young's modulii
*cfcopen, ym_val
*do, kk, 1, maxl
*if, dummy1(kk), ge, ys, then
*set, esec, ys/dummy2(kk)
*else
*set, esec, ym
*endif
*cfwrite, mp, ex, mnum, esec
*cfwrite, mp, nuxy, mnum, poisson
*set, mnum, mnum+1
*enddo
*cfclos

fini

```

```

! select all elements of the model
esel, all

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'ym_mod' and modify material properties
*cfdopen, ym_mod
*do, kk, 1, maxl
*cfwite, mat, mnum
*cfwite, emodif, kk
*set, mnum, mnum+1
*enddo
*cfclos

fini

```

A.7 Second Linear Elastic Analysis

```

/batch

/title, simply supported beam with udl

! set the same material constants and geometric inputs
*set, ym, 30e06
*set, ys, 30e03
*set, poisson, 0.3
*set, ln, 20
*set, dt, 1
*set, ndiv1, 60
*set, ndiv2, 10
*set, pr, 120

! enter preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric(plane42)' element
antype, 0
et, 1, 42

key, 1, 3, 0

```

```

! start modeling and defining keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! devide suitable lines
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define areas
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions at left support
nset, s, loc, x, 0
nset, r, loc, y, dt/2
d, all, ux
d, all, uy

! add boundary conditions at right support
nset, s, loc, x, ln
nset, r, loc, y, dt/2
d, all, uy

! apply udl on the beam
nset, s, loc, y, dt
sf, all, pres, pr
nset, all

! add modified material properties (Young's modulii)
/inp, ym_val
/inp, ym_mod

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0

```

```

outress, all, all
save

! start solving
solve

! end of solving and exit solution routine
fini

! enter post-processor
/post1

set, 1

! define element tables as 'eqvst' and 'estrm'
etable, eqvst, s, eqv
etable, estrm, epel, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, count

! create arrays as 'dummy1' and 'dummy2'
*dim, dummy1, array, max1
*dim, dummy2, array, max1

! open a file as 'stress_strain_2' and
! store the second elastic stresses and strains in corresponding arrays
*cfcopen, stress_strain_2
*do, kk, 1, max1
  *get, sig, elem, kk, etab, eqvst
  *get, epsl, elem, kk, etab, estrm
  *set, dummy1(kk), sig
  *set, dummy2(kk), epsl/(1+poisson)
  *vmask, dummy1(kk)
  *vmask, dummy2(kk)
  *vwrite, kk, dummy1(kk), dummy2(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos

fini

```

Appendix B

ANSYS Files for Linear and Nonlinear Analyses

ANSYS input and command listings for the analysis of all the problems discussed in Chapter 5 are provided in this Appendix. The listing includes files for linear elastic and nonlinear analyses. To perform EGLOSS and N2 analyses, macros ('e_gloss' and 'e_neuber') are called in linear elastic analysis. The listings below have appropriate comments at the end to carryout EGLOSS, and N2. It should be noted that prediction of inelastic strain based on N1 method is approximated by collecting data (the first and second linear elastic analyses results) performed in 'e_neuber' macro. The macros 'e_gloss' and 'e_neuber' are listed in Appendix C. The nonlinear equivalent (von Mises) stresses and strains are stored in the file 'stress_strain_nl'.

B.1 Simply Supported Beam with UDL

B.1.1 Linear Elastic Analysis

```
/batch
```

```
/title, simply supported beam subjected to udl
```

```
! set basic material constants
```

```
*set, ym, 30e06
```

```
! Young's modulus
```

```
*set, ys, 30e03
```

```
! yield stress
```

```
*set, poisson, 0.3
```

```
! Poisson's ratio
```

```
! set basic geometric inputs
```

```
*set, ln, 20
```

```
! beam span
```

```

*set, dt, 1 ! beam depth
*set, ndiv1, 60 ! no of divisions along the beam span
*set, ndiv2, 10 ! no of divisions along the beam depth
*set, pr, 120 ! applied udl on the beam surface

! enter preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric (plane42)' for the model
antype, 0
et, 1, 42

! define elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

key, 1, 3, 0

! start modeling by defining keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! divide lines suitable for meshing
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define area and meshing
a, 4, 1, 2, 3

! start modeling
amesh, all

! apply boundary conditions at left support
nsel, s, loc, x, 0
nsel, r, loc, y, dt/2
d, all, ux
d, all, uy

! apply boundary conditions at right support
nsel, s, loc, x, ln

```

```

nset, r, loc, y, dt/2
d, all, uy

! apply udl
nset, s, loc, y, dt
sf, all, pres, pr
nset, all

!end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
time, pr
outres, all
save

! start solving
solve

! end of solving, and exit solution routine
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.1.2 Nonlinear Analysis

```

/batch

/title, simply supported beam subjected to udl

! set basic material constants
*set, ym, 30e06           ! Young's modulus
*set, ys, 30e03           ! yield stress
*set, poisson, 0.3        ! Poisson's ratio

```

```

! define Poisson's ratio in case of inelastic strain calculation
*set, poi, 0.5

! set basic geometric inputs
*set, ln, 20          ! beam span
*set, dt, 1           ! beam depth
*set, ndiv1, 60        ! no of divisions along beam span
*set, ndiv2, 10        ! no of divisions along beam depth
*set, pr, 120          ! applied udl on the beam surface

! enter preprocessor
/prep7

! define analysis type as 'static' analysis and
! element type as 'four- noded isoparametric (plane42)' for the model
antype, 0
et, 1, 42

! add elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! add material nonlinear properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

key, 1, 3, 0

! start modeling
! define keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! divide lines suitable for meshing
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define area
a, 4, 1, 2, 3

! start meshing

```



```

amesh, all

! add boundary conditions at left support
nset, s, loc, x, 0
nset, r, loc, y, dt/2
d, all, ux
d, all, uy

! add boundary conditions at right support
nset, s, loc, x, ln
nset, r, loc, y, dt/2
d, a ll, uy

! apply loads
nset, s, loc, y, dt
sf, all, pres, pr
nset, all

! nonlinear solver options
autots, on
nsubst, 20
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solution and exit solution routine
fini

! enter post-processor
/post1

```

```

set, last

! create element tables for equivalent stresses and strains
etable, eqvst, s, eqv
etable, estrn, epel, eqv
etable, pstrn, eppl, eqv
etable, eqvstrn, epto, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, count

! define arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1
*dim, dummy5, array, max1

! open a file as 'stress_strain_n1' and
! store stresses and strains in corresponding arrays
*cfcopen, stress_strain_n1
*do, kk, 1, max1
  *get, sig, elem, kk, etab, eqvst
  *get, epsl, elem, kk, etab, estrn
  *get, epsp, elem, kk, etab, pstrn
  *get, epst, elem, kk, etab, eqvstrn

  dummy1(kk)=sig
  dummy2(kk)=epsl/(1+poisson)
  dummy3(kk)=epsp/(1+poi)
  dummy4(kk)=dummy2(kk)+dummy3(kk)
  dummy5(kk)=epst

  *vmask, dummy1(kk)
  *vmask, dummy2(kk)
  *vmask, dummy3(kk)
  *vmask, dummy4(kk)
  *vmask, dummy5(kk)

  *vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
  (3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
fini

```

exit

B.2 Propped Cantilever Beam - Uniformly Distributed Loads

B.2.1 Linear Elastic Analysis

/batch

/title, propped cantilever beam subjected to udl

! set basic material constants and geometric inputs

*set, ym, 30e06 ! Young's modulus

*set, ys, 30e03 ! yield stress

*set, poisson, 0.3 ! Poisson's ratio

*set, ln, 20 ! beam length

*set, ht, 1 ! beam depth

*set, pr, 170 ! applied loads

! set element size parameters

*set, ndiv1, 100 ! no of divisions along length

*set, ndiv2, 10 ! no of divisions along depth

! enter preprocessor

/prep7

! define anlysis type as 'static' and

! element type as 'four-noded isoparametric (plane42)' element for the model
antype, 0

et, 1, 42

! add elastic material properties

mp, ex, 1, ym

mp, nuxy, 1, poisson

! start modeling

! define keypoints

k, 1

k, 2, ln

k, 3, ln, ht

k, 4, 0, ht

```

! define line divisions
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define area
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions at left support
nset, s, loc, x, 0
d, all, all, 0
nset, all

! add boundary conditions at right support
nset, s, loc, x, ln
nset, r, loc, y, 0
d, all, uy, 0
nset, all

! apply udl on the surface
nset, s, loc, y, ht
sf, all, pres, pr
nset, all

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solution and exit solution module
fini

```

```

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.2.2 Nonlinear Analysis

```

/batch

/title, propped cantilever beam subjected to udl

! set basic material constants
*set, ym, 30e06           ! Young's modulus
*set, ys, 30e03           ! yield stress
*set, poisson, 0.3        ! Poisson's ratio

! define Poisson's ratio in case of inelastic strain calculation
*set, poi, 0.5

! set basic geometric inputs
*set, ln, 20              ! beam span
*set, ht, 1               ! beam depth
*set, pr, 150             ! applied udl on the beam surface
*set, ndiv1, 100          ! no of division along the beam span
*set, ndiv2, 10           ! no of divisions along the beam depth

! enter preprocessor
/prep7

! set analysis type as 'static' and
! element type as 'four-noded isoparametric plane42' element
antype, 0
et, 1, 42

! add elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! add nonlinear properties
tb, bkin, 1, 1

```

```

tbdata, 1, ys, 0

! start modeling
! define keypoints
k, 1
k, 2, ln
k, 3, ln, ht
k, 4, 0, ht

! divide lines suitable for meshing
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define area
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions at left support
nsel, s, loc, x, 0
d, all, all, 0
nsel, all

! add boundary conditions at right support
nsel, s, loc, x, ln
nsel, r, loc, y, 0
d, all, uy, 0
nsel, all

! apply udl on the beam surface
nsel, s, loc, y, ht
sf, all, pres, pr
nsel, all

! add nonlinear solver options
autots, on
nsubst, 20
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor

```

```

fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

!start solving
solve

! end of solution and exit solution module
fini

! enter post-processor
/post1

set, last

! create element tables as 'eqvst', 'estm', 'pstrn' and 'eqvstrn'
etable, eqvst, s, eqv
etable, estm, epel, eqv
etable, pstrn, eppl, eqv
etable, eqvstrn, epto, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, count

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1
*dim, dummy5, array, max1

! open a file as 'stress_strain_nl' and
! store inelastic stresses and strains in corresponding arrays
*cfopen, stress_strain_nl
*do, kk, 1, max1
  *get, sig, elem, kk, etab, eqvst
  *get, epsl, elem, kk, etab, estm
  *get, epsp, elem, kk, etab, pstrn
  *get, epst, elem, kk, etab, eqvstrn

```

```

dummy1(kk)=sig
dummy2(kk)=eps1/(1+poisson)
dummy3(kk)=epsp/(1+poi)
dummy4(kk)=dummy2(kk)+dummy3(kk)
dummy5(kk)=epst

*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vmask, dummy4(kk)

*vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
fini

exit

```

B.3 Fixed- Fixed Beam under UDL

B.3.1 Linear Elastic Analysis

```

/batch

/title, fixed-fixed beam with udl

! set basic material constants
*set, ym, 30e06           ! Young's modulus
*set, ys, 30e03           ! yield strength
*set, poisson, 0.3        ! Poisson's ratio

! set basic geometric inputs
*set, ln, 20              ! beam length
*set, dt, 1               ! beam depth
*set, ndiv1, 60           ! no of divisions along length
*set, ndiv2, 10           ! no of divisions along depth
*set, pr, 190             ! udl on the beam surface

```



```

! entet preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric (plane42)' element for the model
antype, 0
et, 1, 42

! add elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! start modeling
! define keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! define line segments suitable for meshing
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define area
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions
nsel, s, loc, x, 0
d, all, all, 0
nsel, all
nsel, s, loc, x, ln
d, all, all, 0
nsel, all

! add uniformly distributed loads
nsel, s, loc, y, dt
sf, all, pres, pr
nsel, all

! end of modeling, and exit preprocessor

```

```

fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
/inp, e_gloss

exit

```

B.3.2 Nonlinear Analysis

```

/batch

/title, fixed-fixed beam subjected to udl

! set basic constants as material properties
*set, ym, 30e06           ! Young's modulus
*set, ys, 30e03           ! yield stress
*set, poisson, 0.3        ! Poisson's ratio

! define Poisson's ratio in case of inelastic strain calculation
*set, poi, 0.5

! set basic inputs as geometric properties
*set, ln, 20              ! beam length
*set, dt, 1               ! beam depth
*set, ndivl, 60           ! no of divisions along length

```

```

*set, ndiv2, 10                ! no of divisions along depth
*set, pr, 190                  ! udl on beam

! enter preprocessor
/prep7

! define 'static' type analysis and
! 'four-noded isoparametric (plane42)' element type for the model
antype, 0
et, 1, 42

! add elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! define material nonlinear properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

! start modeling
! define keypoints
k, 1
k, 2, ln
k, 3, ln, dt
k, 4, 0, dt

! define line segments for meshing
l, 1, 2, ndiv1
l, 2, 3, ndiv2
l, 3, 4, ndiv1
l, 4, 1, ndiv2

! define area
a, 4, 1, 2, 3

! start meshing
amesh, all

! add boundary conditions at left support
nset, s, loc, x, 0
d, all, all, 0
nset, all

! add boundary conditions at right support
nset, s, loc, x, ln

```

```

d, all, all, 0
nset, all

! apply loads
nset, s, loc, y, dt
sf, all, pres, pr
nset, all

! add nonlinear solver options
autots, on
nsubst, 20
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving and exit solution module
fini

! enter post-processor
/post1

set, last

! create element tables as 'eqvst', 'estm', 'pstrn', and 'eqvstrn'
etable, eqvst, s, eqv
etable, estm, epel, eqv
etable, pstrn, cppl, eqv
etable, eqvstrn, epto, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, num, max

```

```

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, maxl
*dim, dummy2, array, maxl
*dim, dummy3, array, maxl
*dim, dummy4, array, maxl
*dim, dummy5, array, maxl

! open a file as 'stress_strain_nl' and
! store the inelastic stresses and strains in corresponding arrays
*cfdopen, stress_strain_nl
*do, kk, 1, maxl
  *get, sig, elem, kk, etab, eqvst
  *get, epsl, elem, kk, etab, estrn
  *get, epsp, elem, kk, etab, pstrn
  *get, epst, elem, kk, etab, eqvstrn

  dummy1(kk)=sig
  dummy2(kk)=epsl/(1+poisson)
  dummy3(kk)=epsp/(1+poi)
  dummy4(kk)=dummy2(kk)+dummy3(kk)
  dummy5(kk)=epst

  *vmask, dummy1(kk)
  *vmask, dummy2(kk)
  *vmask, dummy3(kk)
  *vmask, dummy4(kk)
  *vmask, dummy5(kk)

  *vwrte, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
  (3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfcloss
fini

exit

```

B.4 Simple Multibar Truss

B.4.1 Nonlinear Analysis

For the simple truss, linear analysis was carried out manually. The nonlinear analysis can also be carried out manually. However, for comparison purposes, ANSYS has been used and the input file is given below.

```
/batch

/title, simple multibar truss problem

! set Young's modulus and yield stress
*set, ym, 1 ! Young's modulus
*set, ys, 1 ! yield stress

! enter preprocessor
/prep7

! set analysis type as 'static'
antype, 0

! add stress-strain curve
tb, bkin, 1, 1
tbdata, 1, ys, 0

! define element type as 'link1' and Young's modulus as 'ym'
et, 1, link1
mp, ex, 1, ym

! start modeling by defining member areas
r, 1, 3
r, 2, 2
r, 3, 1

! create nodes
n, 1, 0, 0
n, 2, 0, 0
n, 3, 0, -2
n, 4, 0, -3
```

n, 5, 0, -4, 2

n, 6, 0, -6

! create elements corresponding member areas and nodes

real, 1

e, 3, 4

real, 3

e, 2, 4

real, 1

e, 1, 5

real, 2

e, 4, 6

real, 3

e, 4, 5

real, 1

e, 5, 6

! add boundary conditions

d, 1, all

d, 2, all

d, 3, all

d, 4, ux

d, 5, ux

d, 6, ux

! apply forces on the truss

f, 4, fy, -2

f, 5, fy, -3

f, 6, fy, -2

! end of modeling, and exit preprocessor

fini

! enter solution module

/solu

! nonlinear solver options

autots, on

nsbst, 20

lnsrch, on

nropt, full, , off

ncnv, 1

outress, all, all

save

```

! start solving
solve

! end of solving and exit solution module
fini

! enter post-processor
/post1

set, last

! create element tables as 'astr', 'aestm' and 'apstrm'
etable, astr, ls, 1
etable, aestm, lepl, 1
etable, apstrm, lepl, 1

! get maximum element number as 'max1'
*get, max1, elem, 0, num, max

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1

! open a file as 'stress_strain_n1' and
! store the inelastic stresses and strains in corresponding arrays
*cfopen, stress_strain_n1
*do, kk, 1, max1
  *get, sig1, elem, kk, etab, astr
  *get, epslon1, elem, kk, etab, aestm
  *get, epslonp1, elem, kk, etab, apstrm

  *set, dummy1(kk), sig1
  *set, dummy2(kk), epslon1
  *set, dummy3(kk), epslonp1
  *set, dummy4(kk), dummy2(kk)+dummy3(kk)

*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vmask, dummy3(kk)
*vmask, dummy4(kk)

*vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)

```



```

*enddo
*cfclos
fini

exit

```

B.5 Plate with a Central Hole

B.5.1 Linear Elastic Analysis

```

/batch

/title, plate with a central hole subjected to uniform tension

! set basic material constants and geometric parameters
*set, ys, 3632e05           ! yield strength
*set, ym, 72368e06         ! Young's modulus
*set, poisson, 0.3         ! Poisson's ratio
*set, r, 6375e-06          ! central hole radius
*set, w, 1905e-05          ! plate width
*set, d, 381e-04           ! plate depth
*set, pr, -220e06          ! applied pressure

! enter preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric (plane42)' element for the model
antype, 0
et, 1, 42

! define elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! start modeling
! define keypoints and corresponding line segments for meshing
k, 1, r, 0
k, 2, w, 0

l, 1, 2, 22, 22
k, 3, w, d/2

```

```

l, 2, 3, 12
k, 4, 0, d/2
l, 4, 3, 12
k, 5, 0, r
l, 5, 4, 22, 22

csys, 1
k, 6, r, 45
l, 5, 6, 12
l, 6, 1, 12
csys, 0
l, 6, 3, 22, 22

csys, 0
k, 7, w, d
k, 8, 0, d
l, 3, 7, 10
l, 4, 8, 10
l, 7, 8, 12

! define areas
a, 1, 2, 3, 6
a, 6, 3, 4, 5
a, 3, 7, 8, 4

! start meshing
amesh, all

! apply boundary conditions at the edges
csys, 0
nsel, s, loc, y, 0
d, all, uy, 0
nsel, all

nsel, s, loc, x, 0
d, all, ux, 0
nsel, all

! apply pressures
nsel, s, loc, y, d
sf, all, pres, pr
nsel, all

! end of modeling, and exit preprocessor
fini

```

```

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.5.2 Nonlinear Analysis

```

/batch

/title, plate with a central hole subjected to uniform tension

! set basic constants and inputs for material and geometric parameters
*set, ys, 3632e05           ! yield stress
*set, ym, 72368e06         ! Young's modulus
*set, poisson, 0.3         ! Poisson's ratio
*set, r, 6375e-06          ! central hole radius
*set, w, 1905e-05          ! plate width
*set, d, 381e-04           ! plate depth
*set, pr, -220e06          ! applied pressure

! define Poisson's ratio in case of inelastic strain calculation
*set, poi, 0.5

! enter preprocessor
/prep7

```

```

! define analysis type as 'static' and
! element type as 'four-noded isoparametric (plane42)' element for the model
antype,0
et, 1, plane42

! define elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! define plastic properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

! start modeling
! define keypoints and line segments

k, 1, r, 0
k, 2, w, 0

l, 1, 2, 22, 22
k, 3, w, d/2
l, 2, 3, 12
k, 4, 0, d/2
l, 4, 3, 12
k, 5, 0, r
l, 5, 4, 22, 22

csys, 1
k, 6, r, 45
l, 5, 6, 12
l, 6, 1, 12

csys, 0
l, 6, 3, 22, 22

csys, 0
k, 7, w, d
k, 8, 0, d
l, 3, 7, 10
l, 4, 8, 10
l, 7, 8, 12

! define areas
a, 1, 2, 3, 6
a, 6, 3, 4, 5

```

```

a. 3, 7, 8, 4

! start meshing
amesh, all

! add boundary conditions at the edges
csys, 0
nsel, s, loc, y, 0
d, all, uy, 0
nall

nsel, s, loc, x, 0
d, all, ux, 0
nall

! apply pressures
nsel, s, loc, y, d
sf, all, pres, pr
nall

! add nonlinear solver options
autots, on
nsubst, 20
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! enter post-processor

```

```

/postl

set, last

! create element tables as 'eqvst', 'estrm', 'pstrn' and 'eqvstrn'
etable, eqvst, s, eqv
etable, estrn, epel, eqv
etable, pstrn, eppl, eqv
etable, eqvstrn, epto, eqv

! get maximum element number as 'maxl'
*get, maxl, elem, 0, num, max

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, maxl
*dim, dummy2, array, maxl
*dim, dummy3, array, maxl
*dim, dummy4, array, maxl
*dim, dummy5, array, maxl

! open a file as 'stress_strain_nl' and
! store the inelastic stresses and strains in corresponding arrays
*cfcopen, stress_strain_nl
*do, kk, 1, maxl
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*get, epsp, elem, kk, etab, pstrn
*get, epst, elem, kk, etab, eqvstrn

dummy1(kk)=sig
dummy2(kk)=epsl/(1+poisson)
dummy3(kk)=epsp/(1+poi)
dummy4(kk)=dummy2(kk)+dummy3(kk)
dummy5(kk)=epst

*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vmask, dummy5(kk)

*vwrte, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfcloss

```

fini

exit

B.6 Thick Cylinder with a Circumferential Notch

B.6.1 Linear Elastic Analysis

/batch

/title, thick cylinder with a circumferential notch subjected to internal pressure

! set basic material constants

*set, ym, 275e05

! Young's modulus

*set, ys, 29e03

! yield strength

*set, poisson, 0.3

! Poisson's ratio

! set basic geometric inputs

*set, ri, 16

! inner radius

*set, ro, 25

! outer radius

*set, rn, 1

! notch radius

*set, rin, ri+rn

*set, ln, 9

! cylinder length

*set, pr, 14000

! applied pressure

! enter preprocessor

/prep7

! define analysis type as 'static' and element type as

! 'four-noded isoparametric (plane42) element under plane strain condition

antype, 0

et, 1, 42, , , 1

! define elastic material properties

mp, ex, 1, ym

mp, nuxy, 1, poisson

! start modeling and define keypoints and

! line segments corresponding local coordinate system

k, 1, rin

k, 2, ro

```

l, 1, 2, 10, 10
k, 3, ro, ln
l, 2, 3, 0
k, 4, ri, ln
l, 3, 4, 10
k, 5, ri, m
l, 5, 4, 1 0, 10

local, l1, 1, ri
k, 6, m, 45

l, 1, 6, 10
l, 5, 6, 10

csys, 0
l, 6, 3, 10, 10

! define areas
a, 1, 2, 3, 6
a, 6, 3, 4, 5

! start meshing
amesh, all

! add boundary conditions at the edges
nset, s, loc, y, ln
d, all, uy, 0
nall

nset, s, loc, y, 0
d, all, uy, 0
nall

! apply pressures
nset, s, loc, x, ri
sf, all, pres, pr
nall

csys, 11
nset, s, loc, x, m
sf, all, pres, pr
nall

! end of modeling, and exit preprocessor
/fini

```



```

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solution, and exit solution module
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.6.2 Nonlinear Analysis

```

/batch

/title, thick cylinder with a circumferential notch subjected to internal pressure

! define basic material constants and geometric inputs
*set, ym, 275e05                ! Young's modulus
*set, ys, 29e03                 ! yield strength
*set, poisson, 0.3              ! Poisson's ratio
*set, ri, 16                    ! inner radius
*set, ro, 25                    ! outer radius
*set, rn, 1                     ! notch radius
*set, rin, ri+rn
*set, ln, 9                      ! cylinder length
*set, pr, 14000                 ! applied pressure

! define Poisson ratio in case of inelastic strain calculation
*set, poi, 0.5

! enter preprocessor

```

```

/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric' element under plane strain condition
antype, 0
et, 1, 42, , , 1

! define material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! add material nonlinear properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

! start modeling
! define keypoints and line segments

k, 1, rin
k, 2, ro

l, 1, 2, 10, 10
k, 3, ro, ln
l, 2, 3, 10
k, 4, ri, ln
l, 3, 4, 10
k, 5, ri, rm
l, 5, 4, 10, 10
local, 11, 1, ri
k, 6, rm, 45

l, 1, 6, 10
l, 5, 6, 10

csys, 0
l, 6, 3, 10, 10

! define areas
a, 1, 2, 3, 6
a, 6, 3, 4, 5

! start meshing
amesh, all

! add boundary conditions at the specific edges

```

```

nset, s, loc, y, 0
d, all, uy, 0
nall

nset, s, loc, y, 0
d, all, uy, 0
nall

! apply pressures
nset, s, loc, x, ri
sf, all, pres, pr
nall

csys, 11
nset, s, loc, x, m
sf, all, pres, pr
nall

! nonlinear solving options
autots, on
nsubst, 20
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solution, and exit solution module
fini

! enter post-processor
/post1

```

```

set, last

! create element tables as 'eqvst', 'estrm', 'pstrm' and 'eqvstrm'
etable, eqvst, s, eqv
etable, estrm, epel, eqv
etable, pstrm, eppl, eqv
etable, eqvstrm, epto, eqv

! get maximum element number as 'maxl'
*get, maxl, elem, 0, count

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, maxl
*dim, dummy2, array, maxl
*dim, dummy3, array, maxl
*dim, dummy4, array, maxl
*dim, dummy5, array, maxl

! open a file as 'stress_strain_nl' and
! store inelastic stresses and strains in corresponding arrays
*cfdopen, stress_strain_nl
*do, kk, 1, maxl
  *get, sig, elem, kk, etab, eqvst
  *get, epsl, elem, kk, etab, estrm
  *get, epsp, elem, kk, etab, pstrm
  *get, epst, elem, kk, etab, eqvstrm

  dummy1(kk)=sig
  dummy2(kk)=epsl/(1+poisson)
  dummy3(kk)=epsp/(1+poi)
  dummy4(kk)=dummy2(kk)+dummy3(kk)
  dummy5(kk)=epst

  *vmask, dummy1(kk)
  *vmask, dummy2(kk)
  *vmask, dummy3(kk)
  *vmask, dummy4(kk)
  *vmask, dummy5(kk)

  *vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
  (3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
fini

```

exit

B.7 Bending of Rectangular Plate with Partially Fixity

B.7.1 Linear Elastic Analysis

/batch

/title, bending of a rectangular plate subjected to uniform pressure

! set basic constants and inputs for material and geometric properties

*set, ys, 30e03 ! yield strength

*set, ym, 30e06 ! Young's modulus

*set, poisson, 0.3 ! Poisson's ratio

*set, thik, 0.5 ! plate thickness

*set, leng, 15 ! plate length

*set, wdh, 10 ! plate width

*set, pr, 320 ! applied pressure

! enter preprocessor

/prep7

! define analysis type as 'static' and

! element type as 'eight-noded isoparametric (solid45)' solid element

antype, 0

et, 1, solid45

! add material properties

mp, ex, 1, ym

mp, nuxy, 1, poisson

! start modeling

! define keypoints

k, 1, 0, 0, 0

k, 2, wdh, 0, 0

k, 3, wdh, thik, 0

k, 4, 0, thik, 0

k, 5, 0, 0, leng

k, 6, wdh, 0, leng

k, 7, wdh, thik, leng

k, 8, 0, thik, leng

! divide lines suitable for meshing

```

l, 1, 2, 1.2*width
l, 1, 4, 5
l, 2, 3, 5
l, 4, 3, 1.2*width

l, 1, 5, 1.2*width
l, 4, 8, 1.2*width
l, 3, 7, 1.2*width
l, 2, 6, 1.2*width
l, 5, 8, 5
l, 7, 6, 5
l, 7, 8, 1.2*width
l, 5, 6, 1.2*width

! define volume
v, 1,4,8,5,2,3,7,6

! start meshing
vmesh, all

! add boundary conditions at the specific edges
nsel, s, loc, x, 0.3333*width, 0.66667*width
nsel, r, loc, y, 0
nsel, r, loc, z, 0
d, all, uy, 0
nsel, all

nsel, s, loc, x, width
nsel, r, loc, z, 0,0.33333*leng
d, all, all, 0
nsel, all

nsel, s, loc, x, width
nsel, r, loc, z, 0.666667*leng, leng
d, all, all, 0
nsel, all

nsel, s, loc, x, 0
nsel, r, loc, z, 0.3333*leng, 0.666667*leng
d, all, all, 0
nsel, all

nsel, s, loc, x, 0, 0.3333*width
nsel, r, loc, y, 0
nsel, r, loc, z, leng

```

```

d, all, uy, 0
nset, all

nset, s, loc, x, 0.666667*width, width
nset, r, loc, y, 0
nset, r, loc, z, leng
d, all, uy, 0
nset, all

! apply pressures
nset, s, loc, y, thick
sf, all, pres, pr
nset, all

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.7.2 Nonlinear Analysis

```

/batch

```

```

/title, bending of a rectangular plate subjected to uniform pressure

! set basic constants and inputs for material and geometric properties
*set, ys, 30e03 ! yield strength
*set, ym, 30e06 ! Young's modulus
*set, poisson, 0.3 ! Poisson's ratio
*set, thik, 0.5 ! plate thickness
*set, leng, 15 ! plate length
*set, wdh, 10 ! plate width
*set, pr, 320 ! applied pressure

! define Poisson's ratio in case of inelastic strain calculation
*set, poi, 0.5

! enter preprocessor

/prep7

! define static analysis and eight-noded isoparametric solid element
antype, 0
et, 1, solid45

! add elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! add nonlinear properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

! start modeling
! define keypoints
k, 1, 0, 0, 0
k, 2, wdh, 0, 0
k, 3, wdh, thik, 0
k, 4, 0, thik, 0
k, 5, 0, 0, leng
k, 6, wdh, 0, leng
k, 7, wdh, thik, leng
k, 8, 0, thik, leng

! divide lines suitable for meshing
l, 1, 2, 1.2*wdh
l, 1, 4, 5
l, 2, 3, 5

```



```

l, 4, 3, 1.2*width

l, 1, 5, 1.2*width
l, 4, 8, 1.2*width
l, 3, 7, 1.2*width
l, 2, 6, 1.2*width
l, 5, 8, 5
l, 7, 6, 5
l, 7, 8, 1.2*width
l, 5, 6, 1.2*width

! define volume
v, 1, 4, 8, 5, 2, 3, 7, 6

! start meshing
vmesh, all

! add boundary conditions at the specific edges
nsel, s, loc, x, 0.3333*width, 0.66667*width
nsel, r, loc, y, 0
nsel, r, loc, z, 0
d, all, uy, 0
nsel, all

nsel, s, loc, x, width
nsel, r, loc, z, 0, 0.33333*leng
d, all, all, 0
nsel, all

nsel, s, loc, x, width
nsel, r, loc, z, 0.666667*leng, leng
d, all, all, 0
nsel, all

nsel, s, loc, x, 0
nsel, r, loc, z, 0.3333*leng, 0.666667*leng
d, all, all, 0
nsel, all

nsel, s, loc, x, 0, 0.3333*width
nsel, r, loc, y, 0
nsel, r, loc, z, leng
d, all, uy, 0
nsel, all

```

```

nset, s, loc, x, 0.666667*width, width
nset, r, loc, y, 0
nset, r, loc, z, leng
d, all, uy, 0
nset, all

! apply pressures
nset, s, loc, y, thick
sf, all, pres, pr
nset, all

! add nonlinear solver options
autots, on
nsubst, 20
lnsrch, on
nropt, full, , off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! enter post-processor
/post1

set, last

! create element tables as 'eqvst', 'estm', 'pstrn' and 'eqvstrn'
etable, eqvst, s, eqv
etable, estm, epel, eqv
etable, pstrn, eppl, eqv
etable, eqvstrn, epto, eqv

```

```

! get maximum element number as 'max1'
*get, max1, elem, 0, count

! define arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1
*dim, dummy5, array, max1

! open a file as 'stress_strain_n1' and
! store inelastic stresses and strains in corresponding arrays
*cfopen, stress_strain_n1
*do, kk, 1, max1
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*get, epsp, elem, kk, etab, pstrn
*get, epst, elem, kk, etab, eqvstrn

dummy1(kk)=sig
dummy2(kk)=epsl/(1+poisson)
dummy3(kk)=epsp/(1+poi)
dummy4(kk)=dummy2(kk)+dummy3(kk)
dummy5(kk)=epst

*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vmask, dummy5(kk)

*vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfclose
fini

exit

```

B.8 Thick-Walled Cylinder under Internal Pressure

B.8.1 Linear Elastic Analysis

```

/batch

/title, a thick-walled cylinder under internal pressure

! set t basic constants and inputs for material and geometric properties
*set, ys, 30e03 ! yield strength
*set, ym, 30e06 ! Young's modulus
*set, poisson, 0.3 ! Poisson's ratio
*set, ri, 3 ! inner radius
*set, ro, 9 ! outer radius
*set, elno, 90 ! no of elements across the cross-section
*set, th, (ro-ri)/elno ! thickness
*set, pr, 28e03 ! internal pressure

! enter preprocessor

/prep7

! define analysis type as 'static' and
! element type as 'four noded isoparametric (plane42)' with axisymetric options
antype, 0
et, 1, 42, , , 1

! add material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! start modeling
! create nodes
n, 1, ri
n, elno+1, ro
fill, 1, elno+1
n, elno+2, ri, th
n, 2*(elno+1), ro, th
fill, elno+2, 2*(elno+1)

! create elements
*do, kk, 1, elno
e, kk, kk+1, elno+kk+2, elno+kk+1
*enddo

! add boundary conditions
d, all, uy, 0
nsel, s, loc, x, ri

```

```

! apply uniform internal pressure
sf, all, pres, pr
nset, all

! end of modeling, and exit preprocessor
fini

! enter solution routine
/solu

antpe, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving and solution routine
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.8.2 Nonlinear Analysis

```

/batch

!title, a thick-walled cylinder under internal pressure

! set basic material and geometric properties
*set, ys, 30e03 ! yield strength
*set, ym, 30e06 ! Young's modulus
*set, poisson, 0.3 ! Poisson's ratio
*set, ri, 3 ! inner radius
*set, ro, 9 ! outer radius
*set, elno, 90 ! no of elements across the cross-section
*set, th, (ro-ri)/elno ! thickness
*set, pr, 28e03 ! internal pressure

```

```

! define Poisson's ratio in case of inelastic strain calculation
*set, poi, 0.5

! enter preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric (plane42)' with axisymmetric options
antype, 0
et, 1, 42, ...1

! add elastic material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! add material nonlinear properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

! start modeling
! create nodes
n, 1, ri
n, elno+1, ro
fill, 1, elno+1
n, elno+2, ri, th
n, 2*(elno+1), ro, th
fill, elno+2, 2*(elno+1)

! create elements
*do, kk, 1, elno
e, kk, kk+1, elno+kk+2, elno+kk+1
*enddo

! add boundary conditions
d, all, uy, 0
nset, s, loc, x, ri

! apply uniform internal pressure
sf, all, pres, pr
nset, all

! solver options
autots, on
nsubst, 20
lnsrch, on

```

```

nropt, full, off
nenv, 1

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! ente post-processor
/post1

set, last

! create element tables as 'eqvst', 'estm' 'pstrn' and 'eqvstrn'
etable, eqvst, s, eqv
etable, estm, epel, eqv
etable, pstrn, eppl, eqv
etable, eqvstrn, epto, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, count

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1
*dim, dummy5, array, max1

! open a file as 'stress_strain_nl' and
! store inelastic stresses and strains in corresponding arrays
*cfoopen, stress_strain_nl
*do, kk, 1, max1

```

```

*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*get, epsp, elem, kk, etab, pstrn
*get, epst, elem, kk, etab, eqvstrn

dummy1(kk)=sig
dummy2(kk)=epsl/(1+poisson)
dummy3(kk)=epsp/(1+poi)
dummy4(kk)=dummy2(kk)+dummy3(kk)
dummy5(kk)=epst

*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vmask, dummy5(kk)

*vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
fini

exit

```

B.9 Torispherical Head

B.9.1 Linear Elastic Analysis

```

/batch

/title, a torispherical head subjected to uniform pressure

! set basic constants for material properties
*set, pi, 22/7
*set, ym, 20685e07           ! Young's modulus
*set, ys, 20685e04           ! yield strength
*set, poisson, 0.3           ! Poisson's ratio

! set basic inputs for geometric properties
*set, th, 254e-04            ! wall thickness
*set, rhbyd, 0.80            ! rh/d (rh is a radius of head)
*set, rtbyd, 0.12            ! rt/d (rt is a radius of torus)

```



```

*set, thbyd, 1/300 ! th/d (d is a cylinder diameter)
*set, pr, 650e03 ! applied pressure

! derived dimensions
*set, phi2, asin ((0.5-rtbyd)/(rhbyd-rtbyd))*180/pi
*set, phi1, 90-phi2
*set, d, th/thbyd ! inside diameter of cylinder
*set, rk, rtbyd*d ! knuckle radius
*set, rh, rhbyd*d ! head radius
*set, h, th, rh- (rh-rk)*cos (phi2*pi/180) ! height of torispherical head
*set, dist, d/2-rk ! distance from axis to knuckle center
*set, ri, d/2 ! cylindrical inner radius portion
*set, ro, ri+th ! cylindrical outer radius portion
*set, ht, 1.2*5*sqrt (ro*th) ! height from base to lower knuckle

! element size parameters
*set, ndiv1, 6
*set, ndiv2, 70
*set, ndiv3, 30
*set, ndiv4, 120

! enter preprocessor
/prep7

! define analysis type as 'static' and
! element type as 'four-noded isoparametric element' with axisymmetric options
antype, 0
et, 1, plane42, , , 1

! add material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! start modeling
! define keypoints
k, 1, ri
k, 2, ro
k, 3, ri, ht
k, 4, ro, ht

! local co-ordinate system for knuckle
local, 11, 1, dist, ht

csys, 11
k, 5, rk, phi1

```

```

k, 6, rk+th, phi1

csys, 0

! local co-ordinate system for head
local, 12, 1, 0, ht+hth-rh

csys, 12
k, 7, rh, 90
k, 8, rh+th, 90

! devide lines suitable corresponding coordinate systems

csys, 0
l, 1, 2, ndiv1
l, 3, 4, ndiv1
l, 5, 6, ndiv1
l, 7, 8, ndiv1
l, 1, 3, ndiv2
l, 2, 4, ndiv2

csys, 11
l, 3, 5, ndiv3
l, 4, 6, ndiv3

csys, 12
l, 5, 7, ndiv4
l, 6, 8, ndiv4

! define areas according to coordinate systems

csys, 0
a, 1, 2, 4, 3
amesh, 1

csys, 11
a, 3, 4, 6, 5
amesh, 2

csys, 12
a, 5, 6, 8, 7
amesh, 3

! apply pressures

csys, 0

```

```

sfl, 5, pres, pr

csys, 11
sfl, 7, pres, pr

csys, 0
csys, 12
sfl, 9, pres, pr

csys, 0
sftran

! add boundary conditions

nsel, , loc, x, 0
d, all, ux, 0
nsel, all

nsel, , loc, y, 0
d, all, uy, 0
nsel, all

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution routine
fini

! call the input listing of macro (e_neuber) for N2 analysis
/inp, e_neuber

! call the input listing of macro (e_gloss) for EGLOSS analysis
! /inp, e_gloss

exit

```

B.9.2 Nonlinear Analysis

```
/batch

/title, a torispherical head subjected to uniform pressure

! set basic material constants
*set, pi, 22/7
*set, ym, 20685e07           ! Young's modulus
*set, ys, 20685e04           ! yield strength
*set, poisson, 0.3           ! Poisson's ratio

! define Poisson ratio in case of inelastic strain calculation
*set, poi, 0.5

! set basic inputs for geometric properties
*set, th, 254e-04           ! wall thickness
*set, rhbyd, 0.80           ! rh/d (rh is a radius of head)
*set, rtbyd, 0.12           ! rt/d (rt is a radius of torus)
*set, thbyd, 1/300          ! th/d (d is a cylinder diameter)
*set, pr, 650e03            ! applied pressure

! derived dimensions
*set, phi2, asin ((0.5-rtbyd)/(rhbyd-rtbyd))*180/pi
*set, phi1, 90-phi2
*set, d, th/thbyd           ! inside diameter of cylinder
*set, rk, rtbyd*d           ! knuckle radius
*set, rh, rhbyd*d           ! head radius
*set, h th, rh- (rh-rk)*cos (phi2*pi/180) ! height of torispherical head
*set, dist, d/2-rk          ! distance from axis to knuckle center
*set, ri, d/2              ! cylindrical inner radius portion
*set, ro, ri+th            ! cylindrical outer radius portion
*set, ht, 1.2*5*sqrt (ro*th) ! height from base to lower knuckle

! element size parameters
*set, ndiv1, 6
*set, ndiv2, 70
*set, ndiv3, 30
*set, ndiv4, 120

! enter preprocessor
/prep7

! define analysis type as 'static' and
```

```

! element type as 'four noded isoparametric' element with axisymmetric option
antype,0
et, 1, 42, , , 1

! add material properties
mp, ex, 1, ym
mp, nuxy, 1, poisson

! add inelastic properties
tb, bkin, 1, 1
tbdata, 1, ys, 0

! start modeling
! define keypoints
k, 1, ri
k, 2, ro
k, 3, ri, ht
k, 4, ro, ht

! local co-ordinate system for knuckle
local, 11, 1, dist, ht

csys, 11
k, 5, rk, phi1
k, 6, rk+th, phi1

csys, 0

! local co-ordinate system for head
local, 12, 1, 0, ht+hth-rh

csys, 12
k, 7, rh, 90
k, 8, rh+th, 90

! divide lines according to coordinate systems

csys, 0
l, 1, 2, ndiv1
l, 3, 4, ndiv1
l, 5, 6, ndiv1
l, 7, 8, ndiv1
l, 1, 3, ndiv2
l, 2, 4, ndiv2

```

```

csys, 11
l, 3, 5, ndiv3
l, 4, 6, ndiv3

csys, 12
l, 5, 7, ndiv4
l, 6, 8, ndiv4

! define areas according to co-ordinate systems

csys, 0
a, 1, 2, 4, 3
amesh, 1

csys, 11
a, 3, 4, 6, 5
amesh, 2

csys, 12
a, 5, 6, 8, 7
amesh, 3

! apply pressure

csys, 0
sfl, 5, pres, pr

csys, 11
sfl, 7, pres, pr

csys, 0
csys, 12
sfl, 9, pres, pr

csys, 0
sftran

! add boundary conditions

nsel, loc, x, 0
d, all, ux, 0
nsel, all

nsel, loc, y, 0
d, all, uy, 0
nsel, all

```

```

! solver options for nonlinear analysis
autots, on
nsubst, 20
lnsrch, on
nropt, full, . off
ncnv, 1

! end of modeling, and exit preprocessor
fini

! enter solution module
/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve

! end of solving, and exit solution module
fini

! enter post-processor
/post1

set, last

! create element tables as 'eqvst', 'estm', 'pstrn' and 'eqvstrn'
etable, eqvst, s, eqv
etable, estm, epel, eqv
etable, pstrn, eppl, eqv
etable, eqvstrn, epto, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, count

! create arrays as 'dummy1', 'dummy2' etc.
*dim, dummy1, array, max1
*dim, dummy2, array, max1
*dim, dummy3, array, max1
*dim, dummy4, array, max1
*dim, dummy5, array, max1

```

```

! open a file as 'stress_strain_nl' and
! store inelastic stresses and strains in corresponding arrays
*cfopen, stress_strain_nl
*do, kk, 1, maxl
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*get, epsp, elem, kk, etab, pstrn
*get, epst, elem, kk, etab, eqvstrn

dummy1(kk)=sig
dummy2(kk)=epsl/(1+poisson)
dummy3(kk)=epsp/(1+poi)
dummy4(kk)=dummy2(kk)+dummy3(kk)
dummy5(kk)=epst

*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vmask, dummy5(kk)

*vwrite, kk, dummy1(kk), dummy2(kk), dummy3(kk), dummy4(kk), dummy5(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
fini

exit

```


Appendix C

Elastic Moduli Softening Macros for EGLOSS, N1 and N2 Methods

The macros below ('e_gloss' for EGLOSS and 'e_neuber' for N2) written in ADPL, perform the post-processing in order to determine the equivalent (von Mises) stresses and strains and the Young's modulus changes after the first linear elastic analysis. The equivalent first and second linear elastic (von Mises) stresses and strains are stored as outputs in files 'stress_strain_1' and 'stress_strain_2', respectively. The expected EGLOSS and N2 strains are stored in files 'e_gloss_strain' and 'e_neuber_strain', respectively. The macro for N1 is not presented separately here. The macro file for this will be the same as that for N2. The results are extracted the same way. The geometric construction is slightly different for N1 compared to that of N2. This difference in calculation is taken care of outside the macro using a spread sheet. Therefore, use the 'e_neuber' macro given below for both N2 and N1.

C.1 Elastic Modulus Softening Macro for EGLOSS Analysis

This file is common to all examples.

```
!*****  
!                                     I-Linear Analysis  
!*****  
  
/post1
```

```

set, l

! create element tables such as 'eqvst' and 'estrm'
etable, eqvst, s, eqv
etable, estrm, epel, eqv

! get maximum element number as 'maxl'
*get, maxl, elem, 0, num, max

! create arrays such as 'dummy1' and 'dummy2'
*dim, dummy1, array, maxl
*dim, dummy2, array, maxl

! open a file as 'stress_strain_l' and store the stresses
! and strains in corresponding arrays
*cfdopen, stress_strain_l
*do, kk, 1, maxl
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrm
*set, dummy1(kk), sig
*set, dummy2(kk), epsl/(1+poisson)
*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vwrite, kk, dummy1(kk), dummy2(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
save

fini

! select all elements of the model
esel, all

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'ym_val' and update the Young's
! modulus based on egloss analysis
*cfdopen, ym_val
*do, kk, 1, maxl
*if, dummy1(kk), ge, ys, then
*set, esec, (2*ys/dummy1(kk)-1)*ym
*else

```

```

*set, esec, ym
*endif
*cfwrite, mp, ex, mnum, esec
*set, mnum, mnum+1
*enddo
*cfclos

fini

! select all elements of the model
esel, all

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'ym_mod' and modify material properties
*copen, ym_mod
*do, kk, 1, max1
*cfwrite, mat, mnum
*cfwrite, emodif, kk
*set, mnum, mnum+1
*enddo
*cfclos

fini

```

```

!*****
!                                     II-Linear Analysis
!*****

```

```

/prep7

! the same model created in elastic analysis is restored again
resume

! apply modified moduli
/inp, ym_val
/inp, ym_mod

/solu

antype, 0
time, pr
outres, all, all

```

```

save

! start solving
solve
fini

/post1

set, 1

! create element tables as 'eqvst' and 'estrm'
etable, eqvst, s, eqv
etable, estrm, epel, eqv

! get maximum element number as 'max2'
*get, max2, elem, 0, num, max

! create arrays as 'dummy3', 'dummy4' etc.
*dim, dummy3, array, max2
*dim, dummy4, array, max2
*dim, dummy5, array, max2

! open a file as 'stress_strain_2' and store the
! second linear elastic stresses and strains in corresponding arrays
*cfdopen, stress_strain_2
*do, kk, 1, max2
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrm
*set, dummy3(kk), sig
*set, dummy4(kk), epsl/(1+poisson)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vwrite, kk, dummy3(kk), dummy4(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfdclose
save

fini

! open another file as 'e_gloss_strain' and store the egloss strain
*cfdopen, e_gloss_strain
*do, kk, 1, max2
*if, dummy1(kk), ge, ys, then
*set, sig_mod, 2*ys-dummy1(kk)

```

```

*set, dummy5(kk), dummy2(kk)+(dummy2(kk)-dummy4(kk))*(sig_mod-
dummy1(kk))/(dummy1(kk)-dummy3(kk))
*else
*set, dummy5(kk), dummy4(kk)
*endif
*vmask, dummy5(kk)
*vwrite, kk, dummy5(kk)
(3x, f8.1, 2x, e15.8)
*enddo
*cfclos

fini

```

C.2 Elastic Modulus Softening Macro for N2 Analysis

This file is common to all examples.

```

!*****
!                               I-Linear Analysis
!*****

/post1

set, 1

! create element tables as 'eqvst' and 'estrn'
etable, eqvst, s, eqv
etable, estrn, epel, eqv

! get maximum element number as 'max1'
*get, max1, elem, 0, num, max

! create arrays as 'dummy1' and 'dummy2'
*dim, dummy1, array, max1
*dim, dummy2, array, max1

! open a file as 'stress_strain_1' and store the
! first linear elastic stresses and strains in corresponding arrays
*cfdopen, stress_strain_1
*do, kk, 1, max1
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn

```

```

*set, dummy1(kk), sig
*set, dummy2(kk), epsl/(1+poisson)
*vmask, dummy1(kk)
*vmask, dummy2(kk)
*vwrite, kk, dummy1(kk), dummy2(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
save

fini

! select all elements of the model
esel, all

! set material number as 'mnum'
*set, mnum, 1

! open as file as 'ym_val' and update the Young's
! modulus based on Neuber's rule
*cfopen, ym_val
*do, kk, 1, max1
*if, dummy1(kk), ge, ys, then
*set, esec, ys*ys/(dummy1(kk)*dummy2(kk))
*else
*set, esec, ym
*endif
*cfwrite, mp, ex, mnum, esec
*set, mnum, mnum+1
*enddo
*cfclos
save

fini

! select all elements of the model
esel, all

! set material number as 'mnum'
*set, mnum, 1

! open a file as 'ym_mod' and modify the material properties
*cfopen, ym_mod
*do, kk, 1, max1
*cfwrite, mat, mnum

```

```

*cfwrite, emodif, kk
*set, mnum, mnum+1
*enddo
*cfclos
save

fini

!*****
!                                     II-Linear Analysis
!*****

/prep7

! the same model created in first elastic analysis is restored again
resume

! apply modified modulii
/inp, ym_val
/inp, ym_mod

/solu

antype, 0
time, pr
outres, all, all
save

! start solving
solve
fini

/post1

set, 1

! create element tables as 'eqvst' and 'estrm'
etable, eqvst, s, eqv
etable, estrm, epel, eqv

! get maximum element number as 'max2'
*get, max2, elem, 0, num, max

! define arrays as 'dummy3', 'dummy4' etc.

```

```

*dim, dummy3, array, max2
*dim, dummy4, array, max2
*dim, dummy5, array, max2

! open a file as 'stress_strain_2' and store the
! second linear elastic stresses and strains in corresponding arrays
*cfcopen, stress_strain_2
*do, kk, 1, max2
*get, sig, elem, kk, etab, eqvst
*get, epsl, elem, kk, etab, estrn
*set, dummy3(kk), sig
*set, dummy4(kk), epsl/(1+poisson)
*vmask, dummy3(kk)
*vmask, dummy4(kk)
*vwrite, kk, dummy3(kk), dummy4(kk)
(3x, f8.1, 2x, e15.8, 2x, e15.8)
*enddo
*cfclos
save

! open a file as 'e_neuber_strain' and store the strain
*cfcopen, e_neuber_strain
*do, kk, 1, max2
*if, dummy1(kk), ge, ys, then
*set, ys_mod, ys*ys/(dummy2(kk)*ym)
*set, dummy5(kk), dummy2(kk)+(dummy2(kk)-dummy4(kk))*(ys_mod-
dummy1(kk))/(dummy1(kk)-dummy3(kk))
*else
*set, dummy5(kk), dummy4(kk)
*endif
*vmask, dummy5(kk)
*vwrite, kk, dummy5(kk)
(3x, f8.1, 2x, e15.8)
*enddo
*cfclos

fini

```


Appendix D

Equivalent Strain Calculations in ANSYS 5.5

The total equivalent strain is the sum of elastic equivalent strain and the plastic equivalent strain. The total equivalent strain is given by

$$\epsilon_r = \frac{1}{\sqrt{2(1+\nu)}} \sqrt{(\epsilon_1 - \epsilon_2)^2 + (\epsilon_2 - \epsilon_3)^2 + (\epsilon_3 - \epsilon_1)^2} \quad (D-1)$$

The yield criterion determines the stress level at which yielding is initiated. For multi-component stresses, this is represented as a function of the individual components which is interpreted as an equivalent stress. When the equivalent stress is equal to a material yield parameter, the material develops plastic strains. Since the total strain can be divided into an elastic and plastic part, plastic part is obtained by subtracting the elastic part from the total strain. For some reason, the equivalent strain values obtained from ANSYS 5.5 do not consider the Poisson's ratio factor [ANSYS, 1998]. Therefore, elastic equivalent strain values obtained using 'epel, eqv' command of ANSYS 5.5 should be divided by $1+\nu$ (or, '1+poisson' in the input file) in order to get the correct elastic equivalent strain [Raghavan, 1998]. Similarly, plastic equivalent strain values obtained using 'eppl, eqv' command (used in nonlinear analysis) should be modified taking into account the fact that the Poisson's ratio for plastic flow is considered to be

0.5. The ANSYS 5.5 values for this case must be divided by 1.5 (or, '1+poi' in the input file) in order to get the correct plastic equivalent strain.

The sum of the elastic equivalent strain and the plastic equivalent strain from ANSYS 5.5 (suitably modified as outlined above) gives the required total equivalent strain. In the ANSYS files, 'poisson' = 0.3 (elastic case) and 'poi' = 0.5 (plastic case), respectively.

It must be noted that, the equivalent strain values obtained from ANSYS 6.0, considers the Poisson's ratio factor.

Appendix E

Exact Analysis for the Bending of a Simply Supported Beam

Consider a simply supported beam (shown in Fig. E1a) of span L with a rectangular cross-section subjected to a load intensity of q_0 throughout the beam. The beam remains elastic when the load intensity q_0 is less than the yield load defined by $q_y = 8M_y / L^2$. At the yield moment M_y , only the extreme fibers yield. At the moment higher than the yield moment M_y , yielding spreads to interior fibers too. This yielding of fibers causes the change in stress carried by the fibers as bending moment. When the maximum moment at mid-span and the moment at the section near the mid-span exceeds the yield moment, thus spreading of the yielding over a length of the beam. The spreading of the yielded zones continues until the maximum moment at mid-span reaches plastic moment M_p . At the plastic moment, all fibers are yielded and this yielded zone spreads out over a length called plastic hinge length.

Consider a rectangular segment of width b and depth h . At the elastic-plastic regime, the yielded fibers continue to carry the constant yield stress σ_y while the less stressed interior elastic fibers take additional stresses induced by the increase in moment. A partially yielded section with the elastic core depth of $2h_e$ has been shown in Fig. E1c.

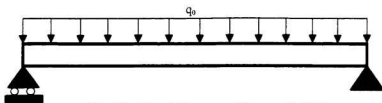


Fig. E1a: Simply Supported Beam with UDL

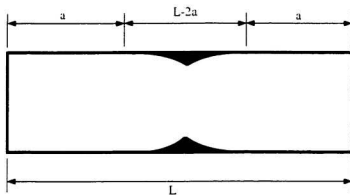


Fig. E1b: Yielded Zones over Beam Length

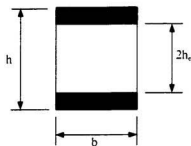


Fig. E1c: Elastic-Plastic Rectangular Section

Deflections of the beam are controlled by the elastic core only since the stiffness is supplied by the elastic portion and not by the plastic zones. Assume that after bending the plane section remains plane and the transverse fiber remains normal to the deflected axis (i.e., shear deformation is negligible).

Defining the moment of inertia of entire section I_0 , the moment of inertia of elastic core I_e , the moment at the initial yielding M_e and the moment M_x at any point x , the following states for rectangular section under pure bending are obtained.

$$\frac{h_e}{h} = \pm \frac{1}{2} \sqrt{3 - \frac{2M_x}{M_e}} \leq \pm \frac{1}{2} \quad (\text{E-1})$$

$$\frac{I_e}{I_0} = 8 \left(\frac{h_e}{h} \right)^3 \leq 1 \quad (\text{E-2})$$

$$\frac{M_x}{M_e} = 4 \left(\frac{h_e}{h} \right)^2 \leq 1 \quad (\text{E-3})$$

Neglecting shearing effects, the bending stress σ_x and strain ϵ_x at C (the distance from neural axis) can be determined by Eqs. E-4 and 5, respectively.

$$\sigma_x = \frac{M_x C}{I_e} \quad (\text{E-4})$$

$$\epsilon_x = \frac{\sigma_x}{E} \quad (\text{E-5})$$

According to the beam theory, if y is the deflection of the neutral axis at any given x , the moment M_x offered by the elastic core is obtained by

$$\frac{M_x}{M_y} = \frac{Eh}{2\sigma_y} \frac{d^2 y}{dx^2} = 1 \quad (\text{E-6})$$

or,

$$\frac{d^2 y}{dx^2} = \frac{\sigma_y}{Eh_y} = \frac{2\sigma_y}{Eh} \quad (\text{E-7})$$

At $h_y/h = 1/2$, the moment at mid-span which is just sufficient to start yielding the extreme beam fibers is given as

$$\frac{M_x}{M_y} = \frac{3q_0 L^2}{\sigma_y b h^2} \left\{ \frac{x}{L} - \left(\frac{x}{L} \right)^2 \right\} = 1 \quad (\text{E-8})$$

or,

$$\frac{x}{L} = \frac{1}{2} \left\{ 1 \pm \sqrt{1 - \frac{M_y}{M_{\max}}} \right\} \quad (\text{E-9})$$

In elastic regime (i.e., $0 \leq x/L \leq a/L$, where $a/L = 1/2(1 - \sqrt{1 - M_y/M_{\max}})$ indicates the elastic span length) the usual beam theory gives

$$\frac{d^2 y}{dx^2} = \frac{M_x}{EI_0} = \frac{q_0 L^2}{2EI_0} \left\{ \frac{x}{L} - \left(\frac{x}{L} \right)^2 \right\} \quad (\text{E-10})$$

Replacing $\xi = x/L$ and integrate with respect to ξ

$$\frac{dy}{d\xi} = \frac{q_0 L^2}{2EI_0} \int (\xi - \xi^2) d\xi + C_1 = \frac{q_0 L^2}{12EI_0} \xi^2 (3 - 2\xi) + C_1 \quad (E-11)$$

or,

$$y = \frac{q_0 L^4}{24EI_0} \xi^3 (2 - \xi) + C_1 \xi + C_2 \quad (E-12)$$

Applying boundary condition, e.g., $\xi = 0$ gives $C_2 = 0$.

$$y = \frac{q_0 L^4}{24EI_0} \xi^3 (2 - \xi) + C_1 \xi \quad (E-13)$$

In the elasto-plastic regime (i.e., $a/L \leq x/L \leq 1/2$, where cross section is partially yielded)

$$\frac{d^2 y}{dx^2} = \frac{2\sigma_y}{Eh \sqrt{3 - 2 \frac{M_y}{M_y}}} \quad (E-14)$$

Again, replacing $\xi = x/L$ (where $a/L \leq \xi \leq 1/2$) gives

$$\frac{d^2 y}{d\xi^2} = \frac{2\sigma_y L^2}{Eh \sqrt{3 - 2 \frac{M_y}{M_y}}} = \frac{2\sigma_y L^2}{Eh} \frac{1}{\sqrt{3 - 8 \frac{M_{max}}{M_y} \xi (1 - \xi)}} \quad (E-15)$$

Integrate with respect to ξ

$$\frac{dy}{d\xi} = -\frac{M_c \sigma_y L^2}{2EhM_{max}} \sqrt{\frac{2M_{max}}{M_c}} \ln \left(u + \sqrt{u^2 + \frac{2M_{max}}{M_c} - 3} \right) + C_1 \quad (\text{E-16})$$

where,

$$u^2 = 3 - 8 \frac{M_{max}}{M_c} \xi(1 - \xi) \quad (\text{E-17})$$

or,

$$\xi = \frac{1}{2} \left\{ 1 \pm \sqrt{1 + \frac{M_c}{2M_{max}} (u^2 - 3)} \right\} \quad (\text{E-18})$$

or,

$$d\xi = -\frac{M_c}{4M_{max}} \frac{udu}{\sqrt{1 + \frac{M_c}{2M_{max}} (u^2 - 3)}} \quad (\text{E-19})$$

Applying boundary condition, e.g., $\xi = 1/2$ (where $u = \sqrt{3 - 2M_{max}/M_c}$) gives the slope at mid-span which is equal to zero.

$$C_1 = -r \ln s \quad (\text{E-20})$$

Therefore,

$$\frac{dy}{d\xi} = r \ln \left(u + \sqrt{u^2 - s^2} \right) - r \ln s = r \ln \left(\frac{u + \sqrt{u^2 - s^2}}{s} \right) \quad (\text{E-21})$$

where,

$$s = \sqrt{3 - \frac{2M_{\max}}{M_v}} \quad (\text{E-22})$$

$$r = -\frac{\sigma_v L^2}{Eh} \sqrt{\frac{M_v}{2M_{\max}}} \quad (\text{E-23})$$

Again integrate with respect to ξ

$$\begin{aligned} y &= r_1 \int \ln \left(\frac{\sqrt{v^2 - s^2} + v}{s} \right) dv + C_4 \\ &= r_1 \left\{ v \ln \left(\frac{\sqrt{v^2 - s^2} + v}{s} \right) - \sqrt{v^2 - s^2} \right\} + C_4 \end{aligned} \quad (\text{E-24})$$

where,

$$r_1 = -\frac{r}{2} \sqrt{\frac{M_v}{2M_{\max}}} \quad (\text{E-25})$$

$$v = \sqrt{u^2 - s^2} \quad (\text{E-26})$$

$$dv = \frac{u du}{\sqrt{u^2 - s^2}} \quad (\text{E-27})$$

The slope $dy/d\xi$ for elastic and partially plastic portions at $\xi = a/L$ (where $u^2 = 1$) must be equal for compatibility and the constant C_1 has been found as

$$C_1 = r \ln \left(\frac{1 + \sqrt{1 - s^2}}{s} \right) - \frac{q_0 L^2}{12EI_0} \left(3 - 2 \frac{a}{L} \right) \left(\frac{a}{L} \right)^2 \quad (\text{E-28})$$

Therefore, the deflection for elastic zone (i.e., $0 \leq x/L \leq a/L$) is

$$y = \frac{q_0 L^4}{24EI_0} \xi \left\{ (2 - \xi)\xi^2 - 2 \left(3 - 2 \frac{a}{L} \right) \left(\frac{a}{L} \right)^2 \right\} + r \xi \ln \left(\frac{1 + \sqrt{1 - s^2}}{s} \right) \quad (\text{E-29})$$

Similarly, the deflection for elastic and partially yielded portions at $\xi = a/L$ (where $u^2 = 1$) must be equal for compatibility and the constant C_4 has been found as

$$C_4 = \frac{q_0 L^4}{24EI_0} \left(\frac{a}{L} \right)^3 \left(3 \frac{a}{L} - 4 \right) + r \frac{a}{L} \ln \left(\frac{1 + \sqrt{1 - s^2}}{s} \right) - r_1 \sqrt{1 - s^2} \ln \left(\frac{1 + \sqrt{1 - s^2}}{s} - 1 \right) \quad (\text{E-30})$$

For partially yielded zones (i.e., $a/L \leq \xi \leq l/2$), the beam deflection is

$$y = r_1 \left\{ \sqrt{u^2 - s^2} \ln \left(\frac{u + \sqrt{u^2 - s^2}}{s} \right) - u - \sqrt{1 - s^2} \ln \left(\frac{1 + \sqrt{1 - s^2}}{s} - 1 \right) \right\} + \frac{q_0 L^4}{24EI_0} \left(\frac{a}{L} \right)^3 \left(3 \frac{a}{L} - 4 \right) + r \frac{a}{L} \ln \left(\frac{1 + \sqrt{1 - s^2}}{s} \right) \quad (\text{E-31})$$

Such analyses can be found in standard references for typical applications. For example, Chen and Sohal [1995] gave examples for beams, frames, etc.

