INFORMATION TO USERS

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

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

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

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

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

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

UMI®

**Analysis of Geometrically Nonlinear and Softening
Response of Thin Structures by a new Facet Shell Element**

By

Adrian Vasilescu

Dipl.Ing. Civil Engineering Institute Bucharest

A thesis submitted to
the Faculty of Graduate Studies and Research
in partial fulfillment of
the requirements for the degree of

Master of Engineering

Department of Civil & Environmental Engineering
Carleton University
Ottawa, Ontario
September, 2000

The Master of Engineering in Civil Engineering is a joint program with the University of Ottawa, administrated by the Ottawa-Carleton Institute for Civil Engineering

© 2000, Adrian Vasilescu

**National Library
of Canada**

**Acquisitions and
Bibliographic Services**

**395 Wellington Street
Ottawa ON K1A 0N4
Canada**

**Bibliothèque nationale
du Canada**

**Acquisitions et
services bibliographiques**

**395, rue Wellington
Ottawa ON K1A 0N4
Canada**

Your file Votre référence

Our file Notre référence

The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de cette thèse sous la forme de microfiche/film, de reproduction sur papier ou sur format électronique.

L'auteur conserve la propriété du droit d'auteur qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

0-612-57743-0

Canada

The undersigned recommend to the Faculty of Graduate Studies and Research acceptance of the thesis *Analysis of Geometrically Nonlinear and Softening Response of Thin Structures by a new Facet Shell Element*, submitted by Adrian Vasilescu, B.Eng; in partial fulfillment of the requirements for the degree of Master of Engineering.

Thesis Supervisor

Thesis Co-Supervisor

Chairman,
Department of Civil Engineering

Carleton University
Ottawa
September 6, 2000

Abstract

This thesis presents the theory and implementation details of the Quadrilateral Facet-Shell Finite Element (QFSE) in simulating geometrically nonlinear deformation effects. The complete formulation of QFSE based on the Discrete Kirchhoff assumptions and geometric nonlinear effects are provided with special consideration accorded to the through-thickness layered model. Special reference is made to the computational efficiency and accuracy of the RQUAD finite element (the membrane component of QFSE facet element) in approximating certain classes of membrane type structures. The large deformation equations are obtained in terms of the Total Lagrangian (TL) formulation based on the Green-Lagrange strain tensor and its energy conjugate Second Piola-Kirchhoff stresses. The complete description of the Newton-Raphson incremental-iterative procedure (N-R) used to trace the nonlinear equilibrium path is also provided. An extension of the incremental-iterative procedure in handling the critical equilibrium states is adopted in order to detect the ultimate load capacity and the post critical structural behavior. In a separate section, a short description of the object oriented code implementation which focuses on some aspects related to the code development and computational efficiency is given. Finally, a selected number of sample analyses are provided to demonstrate the significance of the geometric nonlinear effects in the finite element analysis of thin and very thin plate and shell structures and the ability of QFSE in modeling it. Finally, conclusions and recommendations for further possible improvements and extensions of the existing development are offered in closure.

Acknowledgements

The author wishes to express his gratitude and respect to his supervisors Prof. A.G. Razaqpur and Dr. M. Nofal for their advice, guidance and encouragement throughout the study and preparation of this thesis. A special thank is due to NRC-IRC (National Research Council – Institute for Research in Constructions) Canada for using their facilities during the course of this study.

The author also wishes to express his appreciation to Prof. J. Goldak (M&A, Carleton University) for his many inspiring discussions and suggestions in the field of FEM.

Thanks are due to the staff of both the Civil Engineering Department at Carleton University and to the Buildings Envelope and Structure Program (BES) at NRC-IRC Canada. I am thankful to Dr. Ralph Paroli, the Director of the BES program for his support. I am thankful also to Dr. R. Djebbar, Dr. W. Maref, Dr. P. Mukhopadhyaya, Dr. A. Laouadi and Dr. M. Bomberg, at NRC-IRC for their friendship and many helpful discussions. Thanks are finally extended to my Carleton fellow students Anton Tzenkov, Holger Spiess, Mo Zhang, Ahmed Farag Tolba and Mostafa Amin for their cooperation and help.

The author is thankful to Mrs. A. Galasiu (NRC-IRC) for her help and suggestions in preparing the text and the color plots.

Table of Contents

1	INTRODUCTION	1
1.1	GENERAL	1
1.2	MOTIVATION FOR THE PRESENT WORK	2
1.3	STATE OF THE ART	4
1.4	LITERATURE REVIEW	8
1.5	OBJECTIVES OF THE PRESENT INVESTIGATION	12
1.6	CONTRIBUTIONS AND THESIS LAYOUT.....	13
2	GENERAL FORMULATION OF NONLINEAR FEM.....	16
2.1	GENERAL	16
2.2	THE FINITE ELEMENT METHOD.....	16
2.3	FINITE ELEMENT METHOD FOR PLATES AND SHELLS.....	18
2.4	EQUILIBRIUM CONDITIONS.....	21
2.5	DESCRIPTION OF MOTION AND DEFORMATION	29
2.6	CHOICE OF DESCRIPTION OF MOTION AND DEFORMATION.....	31
2.7	DEFORMATION AND STRAINS.....	33
2.8	DEFINITION OF STRESSES.....	39
2.9	SHELLS AS AN ASSEMBLY OF FLAT ELEMENTS.....	42
2.10	THE LAYERED MODEL.....	45
3	LARGE DISPLACEMENT PROBLEMS	51
3.1	GENERAL	51
3.2	RELATION OF MEMBRANE-BENDING INTERACTION IN PLATES TO THE LARGE DEFLECTION PROBLEM	51
3.3	RELATION OF THE LARGE DISPLACEMENT PROBLEM TO BUCKLING AND COLLAPSE ANALYSIS OF THIN STRUCTURES.....	54

4	FORMULATION OF QFSE FACET-SHELL ELEMENT	57
4.1	GENERAL	57
4.2	REVIEW OF ISOPARAMETRIC QUADRILATERAL ELEMENTS	58
4.2.1	Isoparametric Membrane Element QUAD8	59
4.2.2	Isoparametric Bending Element QUAD9.....	65
4.3	REFINED QUADRILATERAL MEMBRANE RQUAD.....	72
4.4	IMPROVED DISCRETE KIRCHHOFF QUADRILATERAL IDKQ.....	81
4.5	FACET-SHELL QUADRILATERAL FINITE ELEMENT QFSE.....	91
	The nonlinear membrane strain components are represented by.....	95
4	GENERAL PROCEDURE FOR NONLINEAR ANALYSIS	105
5.1	GENERAL	105
5.2	NUMERICAL SOLUTIONS TO THE NONLINEAR PROBLEM	105
5.3	THE FULL NEWTON-RAPHSON ALGORITHM	113
5.4	CONVERGENCE CRITERIA.....	116
5.4.1	Displacement Criterion:	117
5.4.2	Force Criterion:	118
5.4.3	Energy Criterion:.....	119
5.4.5	Alternative Force Criterion:	120
6	ARC-LENGTH CONTROL IN FOLLOWING THE EQUILIBRIUM PATH	
6.1	GENERAL	122
6.2	EQUILIBRIUM PATH AND CRITICAL EQUILIBRIUM STATES	122
6.3	THE ARC-LENGTH CONTROL	131
6.4	THE PREDICTOR SOLUTION.....	138
6.6	AUTOMATIC INCREMENT CUTTING.....	141

7	COMPUTATIONAL IMPLEMENTATION ASPECTS OF FEM	142
7.1	GENERAL	142
7.2	THE MODULAR STRUCTURE OF THE NONLACS FE – PROGRAM.....	145
8	EXAMPLES	150
8.1	GENERAL	150
8.2	THICK CANTILEVER BEAM.....	150
8.3	RESPONSE OF A BEAM-COLUMN.....	152
8.4	RESPONSE OF A THIN PLATE/SHELL	154
8.5	ELASTIC BEAM-COLUMNS LOADED ECCENTRICALLY	155
8.6	TWO HINGED DEEP ARCH	160
8.7	SIMPLY SUPPORTED SQUARE PLATE UNDER UNIFORM LOAD.....	162
8.8	FOLDED PLATE STRUCTURE	164
8.10	WILLIAMS TOGGLE FRAME.....	169
8.11	SHALLOW CIRCULAR ARCH SUBJECT TO NEAR CENTRAL LOAD.....	170
8.12	SHALLOW HINGED SINUSOIDAL ARCH SUBJECT TO UNIFORM PRESSURE	171
9	CONCLUSIONS AND RECOMMENDATIONS	187
9.1	CONCLUSIONS.....	187
9.2	RECOMMENDATIONS FOR FURTHER RESEARCH WORK.....	191
	APPENDIX I.....	194
	APPENDIX II	195
	APPENDIX III.....	198
	REFERENCES	203

List of Figures

2.1 DEFORMATION OF AN INFINITESIMAL MEMBRANE ELEMENT	34
2.2 DEFORMATION OF AN INFINITESIMAL PLATE ELEMENT	36
2.3 DISCRETIZATION OF A CURVED SURFACE BY FACET ELEMENTS.....	43
2.4 DISCRETIZATION OF SHELLS BY TRIANGULAR AND RECTANGULAR FINITE ELEMENTS	44
2.5 LAYERED PLATE OR SHELL MODELS.....	46
4.1 STRAIGHT AND CURVED SIDED QUAD8 MEMBRANE ELEMENT	59
4.2 STRAIGHT AND CURVED SIDED QUAD9 BENDING ELEMENT	65
4.3 DEFORMATION OF MINDLIN PLATE ELEMENT	66
4.4 TRANSFORMATION OF QUAD8 ELEMENT TO RQUAD ELEMENT.....	73
4.5 TYPICAL NODAL DISPLACEMENTS ALONG THE SIDES OF QUAD8 AND RQUAD.....	74
4.6 DRILLING ROTATIONS VERSUS ELASTIC ROTATION.....	80
4.7 TRANSFORMATION OF ELEMENT QUAD9 TO IDKQ	82
4.8 COORDINATE TRANSFORMATION ALONG OF A QUAD9 SIDE	85
4.9 TYPICAL NODAL DOF OF RQUAD AND IDKQ ELEMENTS	92
4.10 INCOMPATIBLE CONNECTION OF TWO PERPENDICULAR FACET ELEMENTS.....	103

5.1 DISCRETIZATION OF A CURVED SURFACE BY FACET ELEMENTS.....	112
5.2 INCREMENTAL ANALYSIS WITH AND WITHOUT EQUILIBRIUM CHECK.....	112
5.3 INCREMENTAL ANALYSIS USING INITIAL OR TANGENT STIFFNESS APPROACH.....	113
6.1 EQUILIBRIUM PATH FOR THE SNAP-THROUGH PROBLEM.....	125
6.2 MODIFIED NEWTON-RAPHSON SCHEME	127
6.3 INCREMENT CUTTING PROCESS	128
6.4 OVERSTEPPING OF A SNAP-THROUGH.....	129
6.5 RIKS'S ORIGINAL METHOD WITH A) FN-R AND B) MN-R.....	132
6.6 SPHERICAL ARC-LENGTH METHOD OF 1 DOF PROBLEM.....	134
6.7 SCHEMATIC ILLUSTRATION OF TWO POSSIBLE SOLUTIONS FOR $\delta\lambda$	136
7.1 THE FE - PROCESS.....	143
7.2 EXAMPLES OF DATA MODULES	146
III.1 PATCH TEST PROBLEM	198
III.2 INFLUENCE OF MESH DISTORTIONS ON IDKQ COMPEARED TO DKQ [65]	199
III.3 SKEW PLATE WITH EQUAL SIDE LENGTHS AND SKEW ANGLE $\beta = 30^0$	201

List of Tables

III.1	CENTER DEFLECTIONS OF SQUARE PLATES	199
III.2.1	SKEW PLATE WITH ALL EDGES SIMPLY SUPPORTED UNDER UNIFORM LOADING.....	202
III.2.2	SKEW PLATE WITH ALL EDGES SIMPLY SUPPORTED UNDER CENTRAL POINT LOAD	202
III.3.1	CLAMPED SKEW PLATE UNDER UNIFORM LOADING	202
III.3.2	CLAMPED SKEW PLATE UNDER CENTRAL POINT LOAD	202

List of Symbols

A_e	Area of an element
A_s	Steel area of a reinforcing bar
A_m	Matrix of first order derivatives of in-plane displacements, u, v
A_b	Matrix of first order derivatives of out-of-plane displacements, w
$[B]$	Strain - displacement matrix
$[B_0]$	Linear strain - displacement matrix
$[B_L]$	Nonlinear strain - displacement matrix
$[B_0^m], [B_L^m]$	Membrane linear and nonlinear strain - displacement matrix
$[B_0^b], [B_L^b]$	Bending linear and nonlinear curvature - displacement matrix
$[B_0^b], [B_L^b]$	Bending linear and nonlinear curvature - displacement matrix
$[D]$	Material (constitutive) matrix
$[D_T]$	Tangential Material (constitutive) matrix
$[D_{mm}], [D_m]$	Membrane material (constitutive) matrix
$[D_{bb}], [D_b]$	Bending material (constitutive) matrix
$[D_{mb}]$	Coupling material (constitutive) matrix

D_{ijpq}	Constitutive tensor
E	Elasticity Modulus
$\{F_i\}, F$	vector of external applied loading forces
$[G_m]$	Matrix of first order derivatives of shape functions of membrane element
$[G_b]$	Matrix of first order derivatives of shape functions of bending element
$[J]$	Jacobi transformation matrix
$[K], K$	structural stiffness matrix
K_0	Elastic (linear) stiffness matrix
K_L	Initial displacement stiffness matrix
K_σ	Initial stress matrix (geometric stiffness matrix)
K_T	Tangent stiffness matrix
K^e	Element stiffness matrix
K_m	Membrane stiffness matrix
K_b	Bending stiffness matrix
K_{mb}	Coupling stiffness matrix
K_s	Shear stiffness
$[Q_i^e]$	element internal stress resultants
$[L]$	Differential Operator
$[N]$	matrix of shape functions
N, N_i	Shape function, shape function at node i
W_i, W_e	internal and external virtual work
U	internal virtual work
W	total virtual work
T_{RQUAD}, T_{IDKQ}	Element transformation matrix
T_i	Fictitious traction components on the reference configuration

t_i	Physical traction forces in the current configuration
S_{ij}	Second Piola-Kirchhoff stress tensor
σ_{ij}	Cauchy stress tensor
$\sigma_p, \sigma_{actual}$	in-plane stress which satisfies the yielding criterion
$\{\sigma\}$	vector of internal stresses
u, v	in-plane displacements
u_i, v_i	in-plane nodal displacements
w	out-of-plane displacements
w_i	out-of-plane nodal displacements
θ_x, θ_y	rotations of the normals to the mid-surface of the element
θ_{xi}, θ_{yi}	nodal rotations of the normals to the mid-surface of the element
θ_z, ω	drilling rotations
θ_{zi}, ω_i	nodal drilling rotations
θ_m, θ_b	vectors of first order derivatives of the in-plane and out-of-plane displacements
$\varepsilon_m, \varepsilon_b$	membrane and bending strains
$\varepsilon_0^m, \varepsilon_L^m$	linear and nonlinear membrane strain vectors
$\varepsilon_0^b, \varepsilon_L^b$	linear and nonlinear bending strain vectors
$\varepsilon_0, \varepsilon_L$	linear and nonlinear strain vectors
t	thickness of a smeared steel layer, thickness of the layer
s	length coordinate, spacing between reinforcement bars
z_i, z_{i+1}	Thickness coordinates of the layer's bottom and top respectively
$\{\delta\}$	vector of virtual displacements
$\{\delta_i\}$	vector of nodal displacements
$\{b\}$	vector of body forces
$\{q\}$	vector of distributed loads
$\{P_i\}$	vector of isolated loading forces

d	vector of nodal displacements
f^n	external incremental applied loading forces
g, g_i, g_i^n	vectors of residual forces, residual forces in iteration i of load increment n
Δr_i	current incremental displacements vector
r_i	iterative displacements vector
Δq	incremental external applied load vector
p	generalized displacements
λ	load level parameter
$d\lambda$	change of the load level parameter
dp	change in the generalized displacements
Δp	incremental generalized displacement
q_{ext}	effective external applied load
Δq_{ext}	incremental external applied load
$s, ds, \Delta s$	arc-length, change in the arc-length, incremental arc-length
δp_T	tangential displacement change
$\delta \bar{p}$	iterative displacement change
$\delta \lambda$	iterative change of the load level parameter
λ	scalar load level parameter
$\Delta \lambda$	incremental change in the load level parameter
C_s	Bergan's Current stiffness parameter
I_d	desired number of iterations
I_0	number of iterations achieved in the previous converged solution
k, k_0	stiffness parameters
ε_D	displacement convergence parameter
ε_F	force convergence parameter
ε_E	energy convergence parameter

1

Introduction

1.1 General

The importance of investigating the nature of nonlinear response of structures in design is recognized to an increasing extent especially in metallic structures. In the last two decades, considerable research effort has been directed towards developing reliable and consistent analysis methods that would ensure the safety and integrity of structural designs. The increasing need for appropriate analysis methods is generated mainly by the complex aspects of the design of cable stayed and box girder bridges or nuclear and off-shore structures where testing on full scale models may be impossible or prohibitive by expensive costs. In such cases, numerical studies based on the nonlinear Finite Element Method (FEM) offer satisfactory practical solutions.

The finite element analysis of structures has been proven as being very effective in linear analysis for many years [18], [19], [20], [29]. With regard to nonlinear analysis, practical solutions to many different problems have also been obtained [16], [18], [20], [21], [24], [29], [32], [35]. The earliest nonlinear finite element analyses have been developed for specific applications, mainly on an intuitive basis, obtaining solutions to problems of particular interest [16]. In the last three decades, major progress has been made in the field of nonlinear finite element procedures providing general analysis capabilities that can handle complex nonlinear

structural problems. However, research to improve the stability, accuracy and effectiveness of the nonlinear mathematical models is still in progress. In addition, the computational efficiency and implementation of the algorithms is reconsidered based on the newest software technologies.

1.2 Motivation for the Present Work

The use of advanced numerical tools is mainly justified by their high efficiency and low cost obtained through significant reductions in the design cycles and required facilities. Based on preliminary numerical studies, feasible design directions and experimental testing strategies can be developed, leading to safe and economical designs.

The most important aspect of many structural problems is closely related to the structural response at ultimate states, which are largely dominated by nonlinear effects. Many different factors may introduce nonlinear effects in structures. Earthquake action is one such factor that can bring a structure to its resistance limit, depending on the particular circumstances. In this case, the ultimate load capacity is of special interest in assessing the safety and functional requirements of structures, since their collapse may result in loss of human lives and incalculable damage. A deeper understanding of the nonlinear behavior of a structural system may lead to better design solutions in satisfying the safety and integrity requirements imposed by

the design codes. The resistance evaluation of structures that suffered noticeable damage due to accidental overloading or expired service life is also of interest in establishing economic rehabilitation strategies. Consequently, establishing the ultimate load and collapse modes become desirable and constitutes the principal motivation of the present work. However, the study will be focused on large displacement rather than large strain.

In this thesis, the extension of the quadrilateral facet-shell element QFSE (Razaqpur and Nofal, [9], [46]) to geometric and material nonlinear capabilities is investigated. This finite element is remarkably simple in formulation and computationally highly efficient in estimating the structural response of thin plates and shells with reasonable accuracy. The need of the QFSE is justified by the fact that for this particular class of thin structures the traditional isoparametric finite elements QUAD8 or QUAD9 lock in the shear mode. The original QFSE element based on the Discrete Kirchhoff Assumption is able to avoid the locking problem. Knowing that thin structures are generally prone to large deformations, the extension of the QFSE to geometric nonlinear problems is desirable. The QFSE element is best suitable for nonlinear computations due to its reduced amount of computational effort. The computational advantage of QFSE can be easily observed by inspecting the size of its stiffness matrix $K_{24 \times 24}^{QFSE}$ and compared to the size of the similar plate bending finite element QUAD9, $K_{45 \times 45}^{QUAD9}$. The computational economy of QFSE can be really exploited in the nonlinear Full Newton-Raphson procedure that requires the solution of the equilibrium equation system in each iteration.

The QFSE element development for geometric nonlinear effects is further completed by the Arc-Length Method in establishing the pre- and post-buckling equilibrium path. The need for the equilibrium path continuation facility is motivated by the structural behavior of thin plates and shells, mainly characterized by large displacements under small strains and structural instability. The Arc-Length Method makes possible the structural analysis in the vicinity of the bifurcation and limit points, permitting the determination of the ultimate load capacity accurately. It should be mentioned that the divergence of the nonlinear Newton-Raphson procedure is not a reliable indication of structural collapse. A local buckling or the failure of an element in the structure may cause severe numerical problems generating the divergence of the numerical procedure. The Arc-Length Method is able to overcome these numerical difficulties in an elegant manner as it will be shown later.

1.3 State of the Art

The nonlinear response of structures is caused by a multitude of factors. For analysis purposes, those factors can be generally reduced to two main categories. First, the elasto-plastic material behavior, which in certain cases becomes significantly nonlinear even at relatively small loading levels, may cause important changes of the overall structural response through development of plastic hinges that may change the initial structural system. Second, the changes in the geometry of the

structure due to large deformations may cause loads to alter their distribution and magnitude, the equilibrium of the structure being reestablished on the deformed configuration. Such effects determine significant changes of the overall structural response and their underestimation may lead to false design solutions based on unrealistic structural evaluations. As mentioned before, many other factors may influence the structural response, but all these factors can be expressed numerically in terms of material and geometric nonlinearities.

The research interest in the field of nonlinear numerical methods is motivated by the changes in the design of composite materials, which may behave completely different as their isolated constituents. For example, it is well known that high-performance concrete fails in a brittle manner and it is very important to optimize its design taking into consideration the ultimate structural states before collapse. It should be noted that the traditional way of design offers reasonably accurate analytical models for some particular structural element-types, depending on certain assumptions that may not consider the nonlinear effects rigorously. Traditional design methods generally limit the nonlinearity of structural components rather than the entire structural system.

In a nonlinear analysis, the stability and accuracy of the numerical procedures depend on various factors and are more difficult to achieve than in a linear analysis. An important factor is the use of a consistent continuum mechanics formulation associated with an effective finite element discretisation. The isoparametric finite element class has been proven as being effective in the implementation of general continuum mechanics formulations [24]. Another important aspect directly related to

the nonlinear continuum mechanics is the use of material models that can accurately represent materials under field conditions [31]. In the last decades, extensive research has been done in the field of material science and material models formulation. The traditional elasto-plastic material models were significantly improved and completed by creep, thermal and visco-plasticity capabilities. In addition, the increased interest in using composite materials has made necessary the consideration of material anisotropy effects on the behavior of the structural elements. In such circumstances, the implementation of advanced material models becomes more complicated and special attention needs to be paid to avoid numerical difficulties that may lead to unstable nonlinear solutions [24], [31].

In the field of structural analysis, shell structures are by far the most complex and difficult structures to analyze. Traditional analytical methods have been found limited in scope and in practical use for this kind of structures. Currently, the FEM is the most commonly used analysis tool for general shells due to its ease in dealing with arbitrary shapes, material properties, loads and boundary conditions and all the other matters related to practical design. Although the finite element procedures based on the isoparametric shell finite elements are satisfactorily applicable to general shell structures, the search for general shell elements is still in progress [15], [17], [18], [26], [29], [37], [46], [47]. This is because the classical curved shell elements, despite their accuracy, have difficulty in convergence and compatibility requirements, and are quite expensive when used in nonlinear analyses. A major progress was achieved with the degenerate shell concept in formulating general shell elements for thick and moderately thin structures. In reference [24], a series of isoparametric plates and

shells are discussed in the context of combined geometric and material analysis. Despite their efficiency and optimal balance between accuracy and computational cost, these elements are difficult to use in the analysis of composite thin laminates where the ratio of the elastic modulus to shear modulus becomes very large. In this context, different theories based on the Discrete Kirchhoff Hypothesis were proposed to develop reliable finite elements that can satisfy a certain degree of accuracy when used to analyze thin and very thin shell structures [15], [17], [18]. Apart from the quality of these finite elements, the stability of thin shell structures is associated with significant geometric nonlinear effects that can not be simply ignored, even for relatively small deflections (5% of the shell thickness) [29]. Disregard of the geometric nonlinear effects may result in considerable stiffening or softening of the analyzed shell structures, leading to unrealistic structural assessments. In the design of moderately thin and very thin plate or shell structural elements, this aspect should not be ignored. In conclusion, the accurate evaluation of the structural response plays a key role in satisfying the safety, durability and functionality requirements of structures as stated in the design codes. In this context, the numerical procedures based on the nonlinear FEM have the capacity to offer reliable and practical solutions to the increasing complexity of the structural designs.

1.4 Literature Review

The first investigations in the field of the nonlinear FEM originate from late 1960's. Large displacement analysis by means of FEM was first proposed by Turner [1]. "Initial stress" matrices (K_σ) were suggested to account for the effect of initial stress in truss and plane stress assemblies. Subsequent work on the derivation of the initial stress matrices for different types of finite elements was reported by Argyris [2]. First applications of large displacement analyses for thin plates and shells were published by Gallagher [3], Kapur and Hartz [4]. "Initial displacement" matrices (K_L), which account for the contribution of the initial displacements to the stiffness matrices, were introduced first by Marcal [5]. He used nonlinear displacement-strain relationships from the classical theory of elasticity and demonstrated that the initial stress and initial displacement matrices are of the same order [5]. Since then, in the last two decades, significant progress in developing reliable computational algorithms for nonlinear FEM has been made. Remarkable theoretical works in the field of FEM were offered by Clough [45], Argyris [2], Zienkiewicz [19], Bathe [20], Cook [18] and many others. Basic procedures largely used in the nonlinear finite element analysis are of incremental/iterative nature and are known as Newton-Raphson procedures. In recent years, significant interest has emerged in completing the Newton-Raphson procedure with the capability of following the equilibrium path beyond the critical equilibrium points. Riks [34] used a normal to the tangent stiffness

in order to detect the next equilibrium point. Crisfield [40] in the early 1980's improved the Riks's method by using a sphere of convergence instead of the normal to the tangent stiffness to ensure the convergence of the solution. Crisfield also introduced the line searches concept in calibrating the magnitude of the load increments to overcome the inherent convergence difficulties [16]. This technique is known as the Arc-Length Method. It represents a form of generalized displacement control realized by means of a fixed imposed radius of the sphere of convergence, denoted as arc-length parameter, that physically represents a finite segment of the equilibrium path. During the twenty years, extensive research effort has been made in improving the Arc-Length Method. Difficulties were detected in travelling the snap-backs. Hellweg and Crisfield [10] improved the existing Arc-Length Method; Bergan [11], introduced the so-called current stiffness parameter to predict the position of the local maximum and minimum points on the equilibrium path. Crisfield [16] used the Bergan's current stiffness parameter as an indicator in switching from the load control to arc-length control in the vicinity of the limit points. In the present work, the current stiffness parameter will be used to increase or decrease the load level following the snap-through forms. However, it is recognized that following equilibrium paths is more complex in the case of the thin shells that are characterized as multi DOF systems prone to bifurcation and severe stability problems.

In the field of FEM research, special attention is devoted to the finite element development. Despite the remarkable achievements in the many areas of FEM, the search for a general and defect free facet-shell finite element is still in progress. Allman [56] and Cook [18] introduced the first effective finite elements for in-plane

stress analysis which were of triangular shape. These special triangular elements passed the patch-test and were computationally efficient. Several four node finite elements were also developed. Allman [62] introduced also a quadrilateral membrane (AQ2) based on the assemblage of four Allman triangular elements (AT). This finite element was found to be accurate even for coarse meshes, but by far too complex in formulation. Cook [18] used a kinematic transformation procedure to generate the Allman's triangle and he mentioned the possibility of extending his transformation concept for a quadrilateral having the QUAD8 as the parent element. This possibility was followed by Razaqpur *et al.* [9], and was extensively investigated in [26], [46], and [48]. They used the classical beam displacement function in deriving the kinematic transformation matrix instead of the Allman's functions. The resulting quadrilateral membrane element with drilling DOF's was called RQD4 (referred as RQUAD later in the thesis) and it was found reliable and posing the same accuracy level as its parent element in linear analyses. Robinson [63] reviewed Razaqpur and Nofal's formulation of RQD4. He noted that the RQD4 is sensitive to aspect ratio and he detected the presence of a spurious mode. However, RQD4 in Razaqpur and Nofal's formulation was found to be very simple in formulation compared with the Allman's quadrilateral and had the same accuracy level of QUAD8 and its drawbacks were of little practical importance.

A distinct research direction in the development of simple and reliable finite elements in analyzing thin plate and shell structures is represented by the development of the facet elements [19]. This element type was suggested as early as 1961 by Greene [6]. At that time, the success of such analysis was hampered by the

poor stiffness matrix of the triangular bending element and the singularity of the global stiffness matrix of coplanar elements. Since then, these problems have been solved in different ways [19]. In analyzing general cylindrical shells, it was found that the flat elements of rectangular or quadrilateral shape represent a good alternative to the classical shell theories [7], [8]. In [9], Razaqpur *et al.* presented a facet shell finite element based on the combination of RQD4 membrane and the IDKQ (Chinniah [15]) plate bending elements. This element was found extremely efficient in analyzing thin shell structures for nonlinear material behavior and small displacements. However, the IDKQ plate bending element was found to be computationally quite expensive in comparison with RQD4. In the present work, IDKQ is reformulated following the same kinematic transformation procedure as was done in Cook [18] for the triangular plate bending element. This new formulation was mentioned by Crisfield [17] as a possible way in imposing the Discrete Kirchhoff Assumptions, but no implementation details or examples were offered. The new form of IDKQ is developed by Nofal (1999) and it will be presented in a later section of this work. The resulting facet element is called QFSE and it will be extended in this thesis for geometric nonlinear problems.

The extension of the quadrilateral facet elements for geometric nonlinear effects was not at the top of the related research agenda. The main reason is that the quadrilateral shapes are prone to distortions that can not be easily avoided. To overcome the problem of element distortions, several four node elements were developed based on the Marguerre's shallow shell theory. Further improvements in the kinematics of the four node quadrilaterals were proposed by Belytschko [12]. He

introduced a nodal projection to account for the warped configurations of the four node quadrilateral shell element. However, the need for simple, accurate and computationally effective finite elements for thin plate and shell structures remains.

1.5 Objectives of the Present Investigation

The objective of this work is to analyze the large displacement effects of membrane, plate bending and thin shell structures. The study is aimed at investigating the influence of the geometric nonlinear effects on the structural response of thin plate and shell structures by means of FEM, particularly employing the quadrilateral facet finite element QFSE. Besides the central objective, it is intended to highlight the significance of the geometric nonlinear effects on the reliability and accuracy of methods which attempt to predict the overall response of structures. Special attention will be paid to aspects closely related to the nonlinear deformation effects such as the interaction between the combined membrane-bending actions. Finally, the influence of the large deformation effects on the pre- and post-buckling behavior of thin cylindrical shell structures using the facet quadrilateral QFSE element is analyzed.

1.6 Contributions and Thesis Layout

In pursuing the objective of this work, the NONLACS (Razaqpur and Nofal, [64]) finite element program was employed. The structure of the original code, written in Fortran'77, was revised and improved, by exploiting the advantages of the Fortran'99 in terms of data structure management. In this sense, an original concept for the modularity of the code is proposed in the last chapter of this work. The previous Discrete Kirchhoff (DK) plate-bending element, as developed by Chinniah [15], was replaced by a new one, the Improved Discrete Kirchhoff Quadrilateral and denoted IDKQ, (see section 4.2.4). This new formulation follows the same idea of introducing a stiffness transformation matrix as in Razaqpur&Nofal [46] and Cook [18], but applied for a quadrilateral DK plate-bending element. To the author's knowledge, this formulation has not been previously published. A complete set of benchmark tests showing the quality of IDKQ is presented by the author in Appendix III. In the present development, the author has implemented the geometrically nonlinear module based on the Total Lagrangian formulation for the QFSE element. In addition, Crisfield's Arc-Length Procedure in following the equilibrium path was implemented by the author, fulfilling the objective of this thesis.

In the first part of the thesis, an introduction to the general nonlinear finite element formulation based on the standard matrix expressions used in the usual finite element developments is presented. The complete formulation of the facet-shell finite

element QFSE obtained by coupling two finite elements used successfully in plane stress analysis is further presented. The two constituents of the facet-shell finite element QFSE, namely, the membrane element with drilling DOF's (RQD4, Razaqpur&Nofal-[46]) and the improved plate bending element based on the Discrete Kirchhoff Hypothesis (IDKQ, Nofal-[64]) are presented separately in a consistent matrix formulation. The special issues that make these two finite elements very attractive in the analysis of thin and very thin plates and cylindrical shell structures are discussed.

The second part of the thesis is devoted to the different Newton-Raphson (*N-R) incremental-iterative procedures largely employed in solving nonlinear structural problems. Due to the complex behavior of composite materials, as example prestressed and reinforced concrete, special remark is made to the Full Newton-Raphson (fN-R) procedure that employs the actual tangent stiffness in each iteration. This numerical procedure is further complemented by the Arc Length Method (ALM), the generalized displacement control version of the *N-R procedures. The ALM makes possible the tracing of the equilibrium path of the analyzed structure in the vicinity of the critical equilibrium states. Possible improvements of the standard ALM are discussed in order to avoid the numerical difficulties at the bifurcation points on the equilibrium path. In a separate section, some aspects related to the computational efficiency and code implementation are discussed in the context of the new software technologies available on the market.

In the last part of the thesis, sample examples are provided to demonstrate the importance of the geometric nonlinear effects and related aspects on the numerical

simulation of the structural response. The performance of the RQUAD membrane finite element is tested when loaded in the in-plane bending mode knowing that the standard membrane elements perform rather poorly for this particular load case. The behavior of the IDKQ plate-bending element is further analyzed in the context of large deflections highlighting the plate stiffening effect. Special attention is paid to the idealization aspects of shell structures by means of the facet-shell element QFSE, applicability limits and possible improvements. The stability problems of shells approximated by facet finite elements are taken into discussion, these problems being much more relevant to thin shells than to simple plates.

Finally, conclusions related to the topics of this study are presented. Remarks and recommendations on how further research can improve the present work are presented.

2 General Formulation of Nonlinear FEM

2.1 General

In this chapter, an introduction to nonlinear FEM is provided. Special reference is made for thin plate and shell structures in section 2.3. General equilibrium conditions based on the Principle of Virtual Work are formulated in section 2.4. Basic concepts from continuum mechanics such as description of motion and deformation, definition of strains and stresses used in describing nonlinear structural effects are reviewed. In section 2.9, shells approximated by flat finite elements are discussed. In the last section, a short description of the layered model associated with the QFSE is given

2.2 The Finite Element Method

In the last three decades, the Finite Element Method (FEM) has become the most powerful numerical analysis tool, being widely used in all engineering fields and in a wide range of applications. In the field of structural engineering, the main

advantage of FEM is its flexibility in dealing with complex engineering problems, involving arbitrary geometry and support conditions, diverse material properties and all kinds of external actions. The formulation of the FEM is based on the discretisation of the continuous material domain into a set of finite continuous subdomains, approximating the original domain, which are usually denoted simply as finite elements. The elements are called finite in order to distinguish them from the differential elements used in the analytical calculus. By means of discretisation techniques, the behavior of the complex structural continuum is reduced to an assemblage of finite elements, comprising a simple geometry for which the numerical description of the structural behavior becomes straightforward. The structural characteristics of each finite element, such as strains and stresses, are expressed in terms of nodal degrees of freedom (DOF) - the problem unknowns. The properties of each type of finite element are uniquely determined and stored in the element stiffness matrix. A load vector with possible force components acting in the direction of the element DOF is defined for each finite element type. Following a systematic procedure based on matrix and vector algebra, the finite element matrices, the DOF vector and the load vector are combined according to finite element model of the continuum domain. The resulting set of simultaneous linear equilibrium equations is solved for the DOF unknowns after imposing the problem boundary conditions. This procedure belongs to the displacement-based finite elements category that governs the structural behavior. For special problems, where certain requirements have to be satisfied, other formulations of the FEM are possible.

2.3 Finite Element Method for Plates and Shells

In the field of structural engineering based on finite elements, a special attention is paid to the two-dimensional representation (2D) of the structural continuum in terms of membrane and plate structural elements. This class of finite elements is derived on the plane stress ($\sigma_z = \sigma_{yz} = \sigma_{zx} = 0$) assumption. For plate bending, it is assumed that the material particles situated originally on a straight line through the element thickness remain on a straight line during and after deformation. For thin plates, the shear deformations are usually neglected and the normals to element mid-surface before deformation remain normal after deformation (Kirchhoff plate bending theory). For thick plates, the shear deformations can not be neglected. In order to consider the shear deformation effect for the case of thick plates, it is assumed that normals to the element mid-surface remain straight but not necessarily normal during and after the deformation process (Mindlin plate bending theory). Based on these assumptions, diverse 2D finite elements have been developed. The first elements used to model the in-plane actions were the constant strain triangle and the bilinear quadrilateral. The first plate bending finite elements used to model the out-of-plane actions were based on the Kirchhoff plate theory, applicable to thin plates only. The so-called Kirchhoff elements were found to perform rather poorly, being non-compatible and showing slow convergence [18]. In recent years, it was

recognized that more effective finite elements for plate bending could be formulated based on Mindlin plate theory [18], [20].

Due to their importance in civil and mechanical structural applications, facet-shell structures have received a special attention in the last decades. The aim of research in the field of shell structures was to develop a reliable finite element capable of simulating the combined in-plane and out-of-plane actions accurately. Therefore, complex finite elements based on specialized shell theories or three dimensional continuum mechanics were developed, but these elements were found to be computationally inefficient due to their required large amount of CPU time, especially in nonlinear analysis. Consequently, during the last decade renewed interest has developed in the development of facet-shell elements, [46]. It has been found that the superposition of a membrane and a plate element are desirable in order to satisfy the design requirements of structural applications. It must be noted that from a physical point of view, both the membrane and bending effects contribute at the same time to the carrying capacity of the structure. In addition, significant in-plane forces may be generated if the deflections w are more than a few tenths of the plate thickness [18], [22]. Therefore, the complete analysis (including also the critical equilibrium states) of moderately thin facet-shell structures should be able to represent both, the in-plane and out-of-plane deformations, especially when large displacements are involved. Clearly, this suggests the combination of the displacement patterns of two existing 2D finite elements which are found to be successful in the analysis of plane stress and plate bending. In this manner, a whole class of facet-shell structural elements can be efficiently developed.

In addition, it must be noted that the effective use, accuracy and computational efficiency of the combined element are determined by the characteristics of the constituent finite elements. The effective use refers to the general utilization of the combined finite element in modeling of shell structures, as well as to its coupling with other types of finite elements such as beams. The connection to other finite element types requires the compatibility of the displacements along the element boundaries. This compatibility requirement is difficult to satisfy in plate bending analysis of thin shells where the rotational DOF are derived from the transverse displacements. For this reason, the plate and shell finite elements derived from independent displacements and rotations are of interest. The accuracy of the results may play an important role when these finite elements are used in modeling the strong nonlinear behavior of certain materials like reinforced concrete or in determining the critical equilibrium states.

In the field of FEM, computational accuracy is a distinct research topic, being closely related to numerical methods, programming techniques, mesh adaptivity and error estimates. The computational cost is another factor that should be considered particularly in nonlinear problems where it might rise exponentially with the number of finite elements in the mesh, making the analysis dependent on the available computational facilities. In this respect, finite elements with lower number of nodes are suitable for the desired computational efficiency. A proper balance between effectiveness, accuracy and efficiency of the proposed facet shell element will be demonstrated in the following sections.

2.4 Equilibrium Conditions

The earliest finite element formulations were in the field of structural applications, and were based on the direct approach, applying discrete force equilibrium conditions. The major disadvantage of this approach is that practically it is limited to some very simple finite elements. For more complex elements, variational approaches are usually employed in deriving the discrete equilibrium equations.

For the purpose of this work, the Principle of Virtual Work will be employed. It is not only simple to use but it is also very general and more suitable when complex structural behavior, such as material and geometric nonlinearity is involved. Alternatively, the Principle of Stationary Potential Energy can be employed, the equilibrium equations being established by setting to zero the derivatives of the total potential energy functional with respect to each generalized coefficient. It should be noted that the Principle of Virtual Work requires only the total virtual work W produced during the virtual deformation of the structure to vanish for a stable equilibrium condition

$$W = W_i + W_e = -U + W_e = 0 \quad (2.1)$$

where W_i is the internal virtual work, always negative, the internal forces undergoing their corresponding virtual displacements in opposite sense. The external virtual work

W_e consists of the virtual works done by the external loading forces acting on the structure

$$W_e = \int_V \{\delta\}^T \{b\} dV + \int_A \{\delta\}^T \{q\} dA + \{\delta\}_j^T \{P\}_j \quad (2.2)$$

$\{\delta\}$ is the vector of virtual displacements, which are imposed on the continuous system, and satisfying the boundary conditions. As a remark, from a theoretical point of view it is not required for virtual displacements to satisfy the boundary conditions. However, to be able to obtain a statically determinate system of equations, we are required to consider virtual displacements compatible with the support conditions only, avoiding in this manner to introduce virtual work done by unknown reaction forces. Also, it must be mentioned that this principle of virtual work has nothing whatsoever to do with the conservation of mechanical energy. It applies even for cases where the mechanical energy is not conserved like in plastic deformation processes, [20]. The virtual work equation in Eq. (2.1) becomes

$$\int_V \{\delta\epsilon\} \{\sigma\}^T dV = \int_V \{\delta\}^T \{b\} dV + \int_A \{\delta\}^T \{q\} dA + \{\delta\}_j^T \{P\}_j \quad (2.3)$$

in which $\{\delta\epsilon\}$ are the virtual strains related to the imposed virtual displacements by means of a differential operator $[L]$

$$\{\delta\epsilon\} = [L]\{\delta\} \quad (2.4)$$

and $\{\sigma\}$ are the internal stresses generated in the deformation process.

Equation (2.3) represents the continuous form of the equilibrium conditions, which constitute the basis of the general formulation of the equilibrium problem in structural mechanics. Equation (2.3) is not only an equivalent way of stating the equilibrium conditions, but it offers some computational advantages. The volume and area integrals refer to the reference configuration, which is always known. The virtual displacement field $\{\delta\}$ is an arbitrary kinematically admissible one satisfying the boundary conditions. Also, it can be observed that Eq. (2.3) is valid for large displacements and it is not dependent on material properties.

In the finite element method, the continuous structural body is discretised into an assemblage of finite elements, connected to each other only at selected nodes where degrees of freedom (denoted simply as DOF's) are defined. In this manner, the infinite number of DOF's of the continuous body is reduced to a manageable finite number. The finite elements are designed to be able to represent the general behavior of the body between the nodal points. The displacement field $\{\delta\}$ within an element is approximated by a set of interpolation functions, called also shape functions, $[N]$. These functions are chosen to reflect the deformation patterns of the continuous material as accurately as possible

$$\{\delta\} = [N]\{\delta_i\} \quad (2.5)$$

where $\{\delta_i\}$ is the vector of discrete nodal displacements. Substituting Eqs. (2.4) and (2.5) into Eq. (2.3), the virtual work equation becomes

$$\sum_e \int_{V_e} ([L][N]\{\delta_i\})^T \{\sigma\} dV_e = \sum_e \int_{V_e} \{\delta_i\}^T [N]^T \{b\} dV_e + \sum_e \int_{A_e} \{\delta_i\}^T [N]^T \{q\} dA_e + \sum_e \{\delta_i\}^T \{P_i\} \quad (2.6)$$

or

$$\sum_e \int_{V_e} [B]^T \{\sigma\} dV_e = \sum_e \int_{V_e} [N]^T \{b\} dV_e + \sum_e \int_{A_e} [N]^T \{q\} dA_e + \sum_e \{P_i\} \quad (2.7)$$

The matrix $[B] = [L][N]$ represents the strain-displacement relationships. For linear structural behavior where the small strains theory applies, the matrix $[B]$ consisting of the first order derivatives of the shape functions $[N]$ is constant and not dependent on the values of the nodal displacements $\{\delta_i\}$. Consequently, the equilibrium conditions are stated in terms of the undeformed configuration. For other cases where the strains are non-proportionally dependent on the magnitude of the nodal displacements, the behavior of the structure is classified as geometrically nonlinear and the matrix $[B]$ is no longer constant. For such cases, the equilibrium equations should be derived rather on the basis of the deformed and not on the initial configuration of the structure.

Ignoring the initial stresses or strains for the sake of simplicity, the actual stresses $\{\sigma\}$ are obtained according to the constitutive relationship as

$$\{\sigma\} = [D]\{\epsilon\} \quad (2.8)$$

in which $\{\epsilon\} = [B]\{d_i\}$ are the actual strains in the structure and $\{d_i\}$ corresponds to the true nodal displacement vector. The matrix $[D]$ is the constitutive (material) matrix containing the actual material properties.

Substituting Eq. (2.8) into Eq. (2.7), the virtual work equation becomes

$$\sum_e \int_{V_e} [B]^T [D] [B] dV_e \{d_i\} = \sum_e \left(\int_{V_e} [N]^T \{b\} dV_e + \int_{A_e} [N]^T \{q\} dA_e + \{P_i\}^e \right) \quad (2.9)$$

or

$$[K] \{d_i\} = \{F_i\} \quad (2.10)$$

where

$$[K] = \sum_e \int_{V_e} [B]^T [D] [B] dV_e \quad (2.11)$$

is the structural stiffness matrix and

$$\{F_i\} = \sum_e \left(\int_{V_e} [N]^T \{b\} dV_e + \int_{A_e} [N]^T \{q\} dA_e + \{P_i\}^e \right) \quad (2.12)$$

is the vector of work equivalent nodal loads.

Equation (2.10) represents the raw form of the structural equilibrium conditions developed on the discretized continuum in terms of nodal displacements $\{d_i\}$ as basic unknowns. This is a very general representation of the equilibrium conditions since at this stage no assumptions other than the general assumption of equilibrium were considered. At this stage, it must be noted that the strain-displacement matrix $[B]$ in certain circumstances becomes dependent on the magnitude of the nodal displacements $\{d_i\}$. It can be shown that in terms of the Total Lagrange (TL) formulation, $[B]$ can be efficiently expressed as

$$[B] = [B_0] + [B_L] \quad (2.13)$$

where $[B_0]$ is the constant component accounting for small strains and $[B_L]$ is the non-constant component accounting for large deformations and depending on the magnitude of the total nodal displacements. In addition, the constitutive matrix $[D]$ representing the material strain-stress relationship in Eq. (2.11) changes in accordance with the strain level in the strain-stress relationship. Because the general equilibrium equations stated in Eq. (2.10) are of nonlinear nature, they will not be directly satisfied and an incremental procedure of predictor-corrector type is usually employed. Roughly, the total load is divided into load increments and the structural analysis is performed for each load increment (predictor) considering a linear correspondence between the incremental load and its corresponding incremental nodal displacements. However, the equilibrium equations will not be exactly satisfied by the incremental solution and further, by successive iterative improvements (correctors), a solution with an imposed degree of accuracy is obtained.

Thus, taking appropriate variations of Eq. (2.7) with respect to $\{d_i\}$, the incremental equilibrium equation can be established as

$$\sum_e \left(\int_{V_e} d([B]^T) \{\sigma\} dV_e + \int_{V_e} [B]^T d(\{\sigma\}) dV_e \right) = d(\{F_i\}) \quad (2.14)$$

and according to Eqs. (2.8) and (2.13)

$$d(\{\sigma\}) = [D]d(\{\epsilon\}) = [D][B]\{d_i\} \quad (2.15)$$

where

$$d([B]) = d([B_L])$$

Substituting Eq. (2.15) into Eq. (2.14), the incremental equilibrium equation can be expressed as

$$\sum_e \left(\int_{V_e} [B]^T [D] [B] dV_e + \int_{V_e} d[B_L]^T \{\sigma\} dV_e \right) d(\{d_i\}) = d(\{F_i\}) \quad (2.16)$$

or

$$([K_0] + [K_L] + [K_\sigma]) d(\{d_i\}) = d(\{F_i\}) \quad (2.17)$$

with $[K_0]$ representing the elastic stiffness matrix associated with the elastic material behavior and small strains, $[K_L]$ representing the stiffness corrections due to large displacements and is known also as large displacement matrix. Finally $[K_\sigma]$ is a symmetric matrix depending on the actual stress level, element geometry and displacement field, usually denoted as the geometric stiffness matrix. The effect of the membrane-type forces on the bending stiffness is accounted for by the $[K_\sigma]$, [18]. For further development of the numerical procedure, these stiffness matrices are expressed as

$$[K_0] = \sum_e \int [B_0]^T [D] [B_0] dV_e \quad (2.18)$$

$$[K_L] = \sum_e \int ([B_0]^T [D] [B_L] + [B_L]^T [D] [B_L] + [B_L]^T [D] [B_0]) dV_e \quad (2.19)$$

$$[K_\sigma] = \sum_e \int d([B_L]^T) \{\sigma\} dV_e \quad (2.20)$$

It should be noted that $[K_0]$ and $[K_L]$ are computed together in order to reduce the amount of unnecessary matrix multiplication but also in minimizing the round-off errors.

The matrix sum

$$[K_0 + K_L] + [K_\sigma] = [K_T] \quad (2.21)$$

is denoted as the tangential stiffness matrix which represents the incremental stiffness matrix corresponding to the load level of the last converged solution. Because of changes in $[B_L]$ and $[D]$, the tangential stiffness matrix $[K_T]$ should be updated correspondingly during the load incrementation. This aspect might constitute a source of computational inefficiency. Traditionally, the tangential stiffness matrix is kept constant for all iterations in the subsequent load increment saving in this manner the computational cost of rebuilding the element stiffness matrices.

Equations (2.18), (2.19) and (2.20) are very general expressions of the structural stiffness. Moreover, they offer the advantage of incorporating both material and geometrical non-linear behavior separately by means of the tangent constitutive matrix $[D_T]$ and the nonlinear strain-displacement matrix $[B_L]$ respectively in a very concise and efficient way.

2.5 Description of Motion and Deformation

A continuum under external loading undergoes deformations, the particles of the continuum being displaced following various spatial deformation patterns depending on the particular characteristics of the external action. The general motion of the continuum particles is expressed in continuum mechanics in two ways, the Total Lagrangian (TL) and the Update Lagrangian (UL).

In the TL formulation, the Green-Lagrange strain and its energy conjugate - 2nd Piola-Kirchhoff stress field are referred to the original undeformed geometric configuration. In this formulation, the displacement field giving the current configuration of the system is related to its original position. The independent variables are expressed in this case in terms of the material coordinates (Lagrangian Description), a set of fixed base vectors relating to the undeformed configuration of the material.

For the UL, formulation in which the Almansi strain and its energy conjugate Cauchy stress fields are defined on the basis of the current geometric configuration of the system. The geometry of the structure is successively updated during the incremental process, the strain and stress fields referring to the last evaluated configuration. In this case, the displacement field giving the current system configuration is related to the last geometric configuration corresponding to the last converged equilibrium state. The independent variables are expressed in terms of the

material coordinates (Lagrangian Description), a set of updated base vectors relating to the actual configuration of the material.

Both of the above mentioned formulations are similar, the transformation of all quantities from TL to UL and vice-versa could be performed by means of a simple coordinate transformation. The Green-Lagrange and Almansi strains reduce to the usual engineering strain definition in the case of small deformation problems. It should be noted that the above-mentioned stress definitions do not have the same physical meaning. The Cauchy stresses being defined on the current deformed configuration keeps its physical significance but the physical meaning of the 2nd Piola-Kirchhoff may be lost if large strains occur, $\varepsilon > 0.02$ as suggested in [20]. However, the large strain behavior can be modeled properly only by considering the constitutive relationships of the materials. The advantage of using one formulation or the other is closely related to their numerical effectiveness in the context of particular characteristics of the analyzed structural problem. In the case of the composite materials, material anisotropy and the very different stress-strain relationships of the constituent materials might require special treatment in order to optimize the computational effort. In a general purpose finite element program, both of the formulations should be provided.

2.6 Choice of Description of Motion and Deformation

The choice of the most appropriate description of the large deformation process has a special significance in the context of composite materials, a significance that is not easily obvious. The computational effectiveness is only one aspect that should not be ignored, but the correct physical treatment of the problem is the most important one. The dominant geometrically nonlinear effects in thin plate and shell structures are mainly provided by the finite rotations consisting practically of rigid body motions, [18], [19], [20]. For this reason, the small strain assumption has been found acceptable in thin plates and shells analysis, [47]. In this respect, the TL formulation, associated with the elastic material behavior is most effectively employed for large displacements and large rotations but small strains analysis. The computational advantages are obvious: no coordinate or other physical properties transformation needs to be performed to or from the current configuration. The implementation of TL for the small strain assumption (but large displacements and rotations) is relatively straightforward by simply substituting in the material characterization subroutine the 2nd Piola-Kirchhoff stresses and the Green-Lagrange strains in lieu of the usual engineering strains [20]. This computational advantage of TL may not be fully exploitable in the case of composite materials in which the constituent materials might behave differently and the small strain assumption may no longer be acceptable for all materials in the composite system. In the case of large strains, the changes in the volume and configuration of the body become significant

and adequate strain and stress measures have to be further considered. Consequently, the anisotropic constitutive relationship of the composite materials should be transformed to the original geometric configuration, a transformation that can not be carried out easily in the TL formulation. It should be noted that in practice the generalized Hooke's law can be applied only for small strains analysis as pointed out in [20].

The use of the UL formulation implies that the last displaced configuration of the structure becomes the reference configuration for any following deformation process. The element deformations are evaluated in the displaced element local coordinate system, since nodal displacements and rotations are assumed to be small with respect to the local coordinate system. Consequently, the element displacements and rotations have to be adjusted for the rigid body motions that the element undergoes. It should be noted that displacement transformation between the coordinate systems does not pose any problems, for it is being performed by means of the usual orthogonal transformation matrix. However, the finite rotations require special treatment when transformed from one coordinate system to another, and great care must be exercised when they must be added or subtracted. In addition, the elastic finite rotations have to be separated from the rigid body rotations because the Almansi strains do not vanish in the case of the rigid body motions. Because for thin plates and shells the finite rotations consist of almost rigid body rotations and small elastic strains, the use of UL formulation seems to be of less practical interest. Another important aspect considered in choosing the most appropriate Lagrangian formulation is referring to the difficulty of computing the initial curvatures of a

deformed four node quadrilateral finite element. The UL requires the updated geometric configuration at each stage of the computation.

Based on the above observations, the choice of the most suitable description of motion is not always simple, but an optimal balance between accuracy and computational efficiency can be established by combining the TL and UL formulations.

For the purpose of this work, only the TL is considered. This choice is motivated by the fact that the quadrilateral shape of QFSE may be distorted by the large displacement effects. In this view, TL offers the advantage that all calculations are performed on the element original configuration. In a UL formulation, all calculations being performed on the actual element configuration, the possible element distortion must be included in the mathematical model. The main use of the facet element QFSE is aimed for the practical design of concrete plates where the nonlinear deformations are kept in the limits imposed by the design code, limits which do not permit significant distortions in plates. In such circumstances, the TL formulation seems to be the better choice.

2.7 Deformation and Strains

In the mechanics of solids based on the TL approach, the basic measure of deformation is represented by Green's strain expressed in the reference coordinates.

The geometric nonlinear effects are introduced in the computational model by reporting the changes in the geometric configuration, to the last obtained displaced configuration but in terms of the material coordinates of the reference configuration. The expression of the Green's strains can be easily determined following the ordinary strain definition of change in length per initial length, where the initial length represents a displaced configuration. For the scope of this work, the Green's strain definition is provided in terms of membrane and bending components separately.

Figure 2.1 shows the deformed shape of an infinitesimal membrane element $dXdY$. In the following, the derivation of the expression for the membrane strain components $[\epsilon_m]$ is given for the X direction only, but for the other strain components, the calculus is similar

Figure 2.1: Deformation of an Infinitesimal Membrane Element.

$$dx^2 = (dX + u_{,x}dX)^2 + (v_{,x}dX)^2$$

$$dx^2 = dX^2 \left\{ 1 + 2 \left[u_{,x} + \frac{1}{2} u_{,x}^2 + \frac{1}{2} v_{,x}^2 \right] \right\}$$

$$dx = dX \sqrt{1 + 2 \left[u_{,x} + \frac{1}{2} u_{,x}^2 + \frac{1}{2} v_{,x}^2 \right]} \equiv dX \left[1 + u_{,x} + \frac{1}{2} u_{,x}^2 + \frac{1}{2} v_{,x}^2 \right]$$

where dx and dy are the deformed lengths of the dX and dY , and $u_{,x}$ and $v_{,x}$ are the X derivatives of the displacement components in the X and Y directions respectively.

By definition, the strain component in X direction is given by

$$\varepsilon_x^m = \frac{dx - dX}{dX} = u_{,x} + \frac{1}{2} u_{,x}^2 + \frac{1}{2} v_{,x}^2 = \varepsilon_{x0}^m + \varepsilon_{xL}^m \quad (2.22)$$

in which the superscript m denotes the membrane strain.

In Eq. (2.22) it can be observed that the so-called engineering strains $\varepsilon_{x0}^m = u_{,x}$, are complemented by the quadratic terms $\varepsilon_{xL}^m = \frac{1}{2} u_{,x}^2 + \frac{1}{2} v_{,x}^2$, representing corrections to the engineering strains, and they account for the geometric changes produced in the deformation process. It should be mentioned that the engineering strains ε_0 are proportional to the displacements and computed for the initial element geometry. For this reason, the engineering strains are also termed linear strains, implying small deformation levels. On the other hand, the quadratic terms ε_L are dependent on the deformation levels, they being generated by the significant changes in the configuration of the element (Figure 2.1). Usually, they are denoted as nonlinear strains. It should be noted that the membrane strain components

are assumed constant over the element thickness. Therefore, in the membrane element idealization, the unknown displacements are computed at the element mid-surface.

The bending strain components are obtained in the same manner considering the deformed shape of the infinitesimal plate element $dXdY$ as follows

Figure 2.2: Deformation of an Infinitesimal Plate Element

$$dx^2 = dX^2 + (w + w_{,x}dX - w)^2.$$

$$dx = dX \sqrt{1 + w_{,x}^2} \cong dX \left[1 + \frac{1}{2} w_{,x}^2 \right]$$

$$\varepsilon_x^b = \frac{dx - dX}{dX} = \frac{1}{2} w_{,x}^2 = \varepsilon_{xL}^b, \quad \text{at mid-surface } z=0 \text{ and } \varepsilon_{x0}^b = 0$$

where $w_{,x}$ is the X derivative of the lateral displacement w of the plate.

For an arbitrary point located at z distance from the element reference surface, the bending strains are given by

$$\varepsilon_x^b = (-z)\kappa_x + \frac{1}{2} w_{,x}^2 = \varepsilon_{x0}^b + \varepsilon_{xL}^b \quad (2.23)$$

From Eq. (2.23) it can be observed that the bending strain components are not constant over the element thickness, they vary with the z thickness coordinate. The strain component $\varepsilon_{x0}^b = (-z)\kappa_x$ represents the usual engineering bending strains in the X direction. The term κ_x denotes the curvature of the mid-surface of the element along X direction. These strains are not dependent on the deformation level. The nonlinear strain component $\varepsilon_{xL}^b = \frac{1}{2}w_{,x}^2$ represents the corrections added to the engineering strains, corrections that are determined when large out-of-plane element deformations occur as shown in Figure 2.2. The magnitude of these strains depends on the deformation level. It should be noted that the above strain definition follows the plate bending theory which states that points on the mid-surface of the element ($z = 0$) move only in the z direction as the plate deforms in bending. For this reason, no in-plane nonlinear deformation effects from bending action can be accounted for in this strain definition. In what follows, for simplicity the material coordinates X,Y,Z will be replaced by x,y,z .

According to Eqs. (2.22) and (2.23), the complete definition of the facet-shell Green's strains comprising membrane and bending actions, including small and large deformations, can be written as

$$[\varepsilon] = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} u_{,x} \\ v_{,y} \\ u_{,y} + v_{,x} \end{bmatrix}_0^m + \begin{bmatrix} \frac{1}{2}(u_{,x}^2 + v_{,x}^2) \\ \frac{1}{2}(u_{,y}^2 + v_{,y}^2) \\ u_{,x}u_{,y} + v_{,x}v_{,y} \end{bmatrix}_L^m + (-z) \begin{bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{bmatrix}_0^b + \begin{bmatrix} \frac{1}{2}w_{,x}^2 \\ \frac{1}{2}w_{,y}^2 \\ w_{,x}w_{,y} \end{bmatrix}_L^b \quad (2.24a)$$

or

$$[\varepsilon] = [\varepsilon_0^m] + [\varepsilon_L^m] + [\varepsilon_0^b] + [\varepsilon_L^b] = [\varepsilon_m] + [\varepsilon_b] \quad (2.24b)$$

where the subscripts O and L indicate the linear and the nonlinear strain components and the superscripts m and b indicate the strain types, membrane and bending actions.

The strain measure in Eq. (2.24), expressed in general displacement terms u, v, w as a tensor of rank two, is known as the Green-Lagrange strain measure (usually just referred to as Green's strain). This strain definition is not only convenient for the element development but it has a very important property, the strains vanish for rigid body motions.

The Green's strain measure can be used in a wide range of nonlinear problems from small to large deformations, but when used in a TL formulation, it is particularly effective for acceptable small strains and large displacement and rotation problems. The material nonlinear behavior is usually related to the stress-strain relationship and it is introduced in the computational model by means of the material (constitutive) matrix. The TL formulation is computationally effective for isotropic materials because it does not require any transformation of the constitutive matrix to the reference configuration. Moreover, the linear strain-displacement matrices $[B_0^m]$ and $[B_0^b]$ are computed once and they are kept unchanged during the analysis. Only the nonlinear strain-displacement matrices $[B_L^m]$ and $[B_L^b]$, accounting for the large displacements and nonlinear geometric effects, will change according to the changes of the incremental load level. The incremental strain-displacement matrices $[B_m]$ and $[B_b]$ are then obtained through a simple addition of $[B_0]$ matrices to the actual values of $[B_L]$ matrices. It should be noted that in TL formulation the $[B_0]$ matrices remain unchanged, a considerable computational effort being saved.

2.8 Definition of Stresses

The concept of stress was introduced in continuum mechanics to approximate the internal virtual work in terms of a volume integral. In the context of FEM, the internal work is expressed as a sum of integrals over the individual finite elements as stated in Eq. (2.14). As long as the configuration of the continuum does not undergo significant changes, the ordinary stress representation of “force per area” gives the stress measure. In a large deformation analysis the configuration of the continuum changes continuously and the simple stress representation mentioned before should be reconsidered in the light of expressing the incremental internal virtual work, in a different manner.

In the TL formulation, the Green’s strain measure is energy conjugate with the 2nd Piola-Kirchhoff stress, both of them referring to the reference configuration. It should be noted that in the case of a combined TL-UL, the reference configuration is the last updated geometric configuration in which the Green’s strains are referred for the next load step (or steps). In the TL formulation, the reference configuration is represented by the original geometric configuration at time zero.

In the usual tensor notation, the 2nd Piola-Kirchhoff stress tensor S_{ji} is generally defined by the component relationship

$$dT_i = S_{ji} n_j \quad (2.25)$$

where dT_i are traction components related to the reference configuration and n_j are the outward unit normals to the surface area dA in the reference configuration. Note that, the meaning of the entities in Eq. (2.25) is not of physical nature because they are altered by the coordinate transformation from the current to the reference configuration

$$dT_j = \frac{\partial X_i}{\partial x_j} dt_i \quad (2.26)$$

where $\frac{\partial X_i}{\partial x_j}$ is the deformation gradient and dt_i is the vector of physical traction forces measured in the current geometric configuration from the following relationship

$$dt_i = \sigma_{ij} n_j \quad (2.27)$$

in which n_j is the outward unit normal vector to the surface area da and σ_{ij} is the physical stress <force per final area> denoted as the Cauchy stress tensor, both entities being related to the current configuration.

According to Eqs. (2.25), (2.26) and (2.27), a relationship between the 2nd Piola-Kirchhoff and Cauchy stress tensors can be established

$$S_{ji} = \left| \frac{\partial x}{\partial X} \right| \frac{\partial X_i}{\partial x_m} \sigma_{mn} \frac{\partial X_j}{\partial x_n} \quad (2.28)$$

In Eq. (2.28), since σ_{ij} is symmetric, the 2nd Piola-Kirchhoff stress tensor is also symmetric signifying the fact that for rigid body motions they both vanish. This is a very important aspect highlighting complete agreement between the computational model with the physical reality.

Generally, the 2nd Piola-Kirchhoff stresses are related to the Green's strains by means of a constitutive relationship reflecting the material properties at the actual deformation state. In practice, the constitutive relationship can be approximated with the generalized Hook's law so long as the strains remain in the domain of the small strain assumption ($\varepsilon < 2\%$ for the usual materials as suggested in [20]). This is not a limitation in the applicability of the TL, the numerical solution being obtained as a series of linear incremental solutions based on the small strain assumption [20]. For large strain levels and highly inelastic material behavior, the constitutive tensor should be updated according to the strain-stress relationship of the material. It should be noted that for isotropic materials and small strains there is no need for coordinate transformation of the constitutive relationship. Significant computational advantages can be achieved by using the isotropic constitutive relations for each constituent material of the composite section that is anisotropic by employing the layered model as will be shown in the section describing the adopted layered model.

Under the assumption of small strains in the reference configuration that might be the original or an intermediate one, the 2nd Piola-Kirchhoff and Cauchy stresses are approximately equal in magnitude and direction $S_{ij} \approx \sigma_{ij}$. For elastic material behavior, the constitutive matrix does not change during the deformation process and the 2nd Piola-Kirchhoff stress tensor can be simply expressed as

$$\Delta S_{ij} = D_{ijpq} \Delta \varepsilon_{pq} \quad (2.29)$$

where ΔS_{ij} and $\Delta \varepsilon_{pq}$ are the incremental quantities of the 2nd Piola-Kirchhoff stress and Green's strain tensors used in TL. For elasto-plastic material behavior, under the

small strain assumption of TL formulation, Eq. (2.29) can be extended for the material nonlinear analysis easily by employing the tangent constitutive matrix usually used in the material-nonlinear-analysis-only. Equation (2.29) is of practical interest because of its simplicity and low computational cost in the framework of the incremental TL formulation. In addition, Eq. (2.29) is especially advantageous in the context of the layered model associated with the composite section. In the layered model, each layer may represent different isotropic materials through the thickness for which generalized Hooke's laws are well established.

2.9 Shells as an Assembly of Flat Elements

The formulation of the governing equations for curved general shells is recognized to be a very difficult problem [19]. In the classical shell theory many different alternative formulations were developed based on various approximations. In the context of FEM, the discretisation of the continuum is able to overcome some difficulties related to the classical differential geometry by introducing the isoparametric interpolation concept for example. However, in the solution of a practical shell problem, further approximations need to be invoked in order to skip the theoretical difficulties encountered in the classical shell formulations, yet concurrently achieving a reasonable degree of accuracy. In this manner, the approximation of the curved shell surface by means of small flat elements is of practical importance, Figure 2.3. The reliability of this physical approximation can be

intuitively proven. The convergence of the facet assemblage is ensured as the size of the facet subdivisions represented by the flat finite elements is reduced to the limit at which the facet assembly fits the curved surface exactly. Extensive numerical experience shows such a convergence [15], [16], [18], [19], [27], [46].

Figure 2.3 : Discretisation of a Curved Surface by Facet Elements

Generally, the shell structures are subjected to both bending and membrane actions. The usual flat elements used in building of shell assemblages are based on the independent membrane and bending deformation patterns, but for a correct physical representation of the deformation process they should be coupled. The coupling process consists of combining the membrane and bending strains in the strain energy expression. The importance of the coupling process becomes more important in relationship with the layered model adopted in the discretisation of composite sections, [18]. This aspect will be presented later in detail in the section devoted to the development of QFSE finite element.

The quality of the discretisation of general shells into flat finite elements is mesh dependent, special attention should be paid in the meshing process in order to minimize the geometric errors. Theoretically, this can be done only by a mesh

generator that is able to calibrate the element size as function of the local geometric error estimate. Accordingly, the discretisation of an arbitrary shell surface can be effectively performed only by triangular flat elements where the three corner nodes of the elements always remain coplanar (Fig. 2.4). It should be noted that the standard triangular elements suffer from lack of a good bending stiffness matrix [18], [19]. However, some shells of general cylindrical shapes can be well approximated by flat quadrilateral element. In this respect, RQUAD and IDKQ that promise high accuracy at low computational costs are considered in the formulation of the facet quadrilateral element QFSE.

Figure 2.4: Discretisation of Shells by Triangular and Rectangular Finite Elements

The junction of the flat elements in Fig 2.4 may pose difficulties in the case of the standard finite elements with 5 DOF's per node, especially in assemblages of perpendicular elements or in coupling with elements having θ_z DOF. Closely related to this phenomenon, it should be mentioned that because of the drilling DOF of RQUAD membrane element, the assemblages QFSE flat elements would not result in a singularity of the structural stiffness matrix. Also, it should be noted that the drilling

DOF's of RQUAD do not have the meaning of a physical degree of freedom, they being transformation parameters that will be discussed in detail in the section devoted for the RQUAD development.

2.10 The Layered Model

The usual computational models, able to approximate accurately the composite nature of the structural materials like reinforced concrete, are based on a through-thickness discretisation approach. The plate finite element is divided into a number of material layers of different thickness and material properties, each assumed to behave in a state of plane stress in the case of thin plates.

As can be observed in Figure 2.5, the reinforcement represented usually by a series of bars disposed at equal space intervals in the section is converted to an equivalent smeared reinforcing layer with an equivalent thickness determined by

$$t = \frac{A_s}{s} \quad (2.30)$$

where A_s is the steel area of one reinforcing bar and s is the spacing between bars. The smeared reinforcing layers are defined in the local coordinate system of the element accounting for different reinforcing directions and different layer thickness. The locations of the layers are specified by means of z_i local coordinates measured from the element reference surface to the mid-surface of the i -th layer. Due to

Kirchhoff assumption that normals to the element reference surface before deformation remain also normal and straight after deformation, the deformations compatibility between adjacent layers is ensured. For this reason, no relative displacements between the reinforcing layers and the body layers can be explicitly accounted for in this model. Thus, in the present study it is assumed that no bond slip takes place during the deformation process.

Figure 2.5: Layered Plate or Shell models

In the adopted layered model, the computation of the element stiffness matrix requires the evaluation of the strain-displacement matrix $[B]$ and the constitutive matrix $[D]$ for each layer. It must be noted that both of these matrices are dependent on the state of deformation of each layer. Because $[B_0]$ matrix is depending mainly on the first derivatives of the element shape functions, it applies for all element layers. The matrices that must be separately evaluated for each layer are the constitutive matrix $[D]$ and the large strain-displacement matrix $[B_L]$. The element tangent stiffness matrix, $[K_T]$, is then obtained by adding together the layer stiffness contributions carrying out the integration over the element thickness with respect to the local z coordinate. For the facet element QFSE, constant thickness over the element surface is assumed resulting in a constant Jacobian matrix through the layers. This assumption is appropriate for thin plate or shell structures.

The layers are numbered sequentially, starting from the bottom of the element. Each layer contains Gauss integration stations on its mid-surface specified in the input data file for all layers in the layer system. The stress components are computed at the Gauss stations and are considered constant over the layer thickness as shown in Figure 2.4. As can be observed, the thickness discretisation is expensive from a computational point of view, the same element stiffness matrix being employed for each layer separately. However, this layered model permits an accurate approximation of the sectional characteristics of composite reinforced concrete sections. Each layer has different deformation and stress states depending on the strain variation over the element thickness due to the combined membrane-bending

actions. The actual physical state of each layer can be checked and in case that the failure limit of the material is exceeded, the respective layer is considered damaged and it will not contribute longer to the element stiffness. In this manner, the layered system makes possible the simulation of the progressive softening and cracking or crushing of the composite sections.

It must be mentioned that the material damage levels are determined in an average sense at the integration points according to the material nonlinear model. Because the strains are evaluated at the integration points over the element area, the same strain level is extrapolated over the tributary area of the integration point. Therefore, the damage accumulated at an integration point in a layer is considered uniformly distributed over the tributary area. The material properties are set by means of average values determined statistically from many experimental tests and specified in the design codes.

This layered model offers important advantages in the finite element idealization of the composite structural members, especially for members of reinforced concrete. The main advantage of this layered model is represented by its flexibility in layer definition that allows one to account accurately for the different material properties composing the member section. In this way, the element stiffness matrix is assembled by adding together the layer stiffness contributions, leading finally to a reliable prediction of the finite element physical state, cracked or uncracked and intermediate states. This issue plays the most important role in the nonlinear analysis of the post-critical equilibrium paths where significant changes in the structural integrity are expected. In addition, it should be noted that the degree of

accuracy in analysis is determined by the mid-ordinate rule used to integrate the stiffness matrices and the residual forces through the thickness. The integration over the element thickness is carried out by summing up the contributions of all layers as follows

$$\begin{aligned}
 D_{mm} &= \int_h [D_i] dz = \sum_{i=1}^{nl} (z_{i+1} - z_i) [D_i] \\
 D_{mb} &= \int_h (-z_i) [D_i] dz = -\frac{1}{2} \sum_{i=1}^{nl} (z_{i+1}^2 - z_i^2) [D_i] \\
 D_{bb} &= \int_h z_i^2 [D_i] dz = \frac{1}{3} \sum_{i=1}^{nl} (z_{i+1}^3 - z_i^3) [D_i]
 \end{aligned} \tag{2.31}$$

where z_i and z_{i+1} represent the thickness coordinates of the top and bottom of the layer margins to the element reference surface. Following the same rule, the element internal forces necessary to evaluate the residuals are computed from

$$Q_i^e = \int_{A_e} [B_m]_i^T N_i^e dA_e + \int_{A_e} [B_b]_i^T M_i^e dA_e \tag{2.32}$$

where N_i^e and M_i^e are stress resultants of layer i computed at the integration points as follows

$$N_i^e = \sum_{p=1}^{nl} [\sigma_p]_i t_i \tag{2.33}$$

$$M_i^e = \sum_{i=1}^{nl} z_i [\sigma_p]_i t_i$$

where $[\sigma_p]$ are the actual in-plane stresses that satisfy the constitutive relation and t_i is the thickness of the layer i .

3 Large Displacement Problems

3.1 General

Two large displacement problems are discussed in this chapter. The first one is characteristic for plates and is treated in the next section. It relates to the development of internal membrane type forces generated by the bending actions in plates. In the last section, the influence of the geometrically nonlinear effects on the buckling phenomena in thin structures is investigated.

3.2 Relation of Membrane-Bending Interaction in Plates to the Large Deflection Problem

The large deflection plate problem belongs to a special class of finite deformation problems in which the strains but not the rotations can be considered infinitesimal [36]. In the special case of plates, the consideration that differentiates between small and large deflection assumption is represented by the measure in which the developed membrane-type forces can alter the bending stiffness [18]. Large deflections may cause the development of membrane-type strains and stresses that are not accounted in the small deflection theory (because of the Kirchhoff assumptions).

Consequently, the small deflection assumptions are adequate only if the membrane-type forces developed by the out-of-plane actions are small enough to not alter the bending stiffness. From this point of view, the distinguishing feature separating the small from large deflection assumption in plates can not be associated with any absolute geometric representation of the displacements or rotations. It is generally accepted that the applicability limit of the small deflection assumption may be valid for a ratio of maximum deflection to plate thickness less than $1/3$ to $1/2$, [22], [25], but these limits might be influenced by factors such as boundary conditions and the type of curvature developed in the plate structure. For example, in a simply supported square plate with unrestrained horizontal DOF's, membrane-type forces in the plate mid-surface will not be developed. From a theoretical point of view, the small deflection assumption is valid so long the nonlinear part of the Green's strains is orders lower than the linear one so that it does not influence magnitude of the total strain.

In the large deflection problem, the membrane-type forces can have a stiffening or a softening effect on the plate bending, depending on particular circumstances. The tensile membrane-type internal forces developed by the out-of-plane actions stabilize the plate structure contributing to its overall resistance. In the case of the development of compressive membrane-type internal forces, the membrane stiffness may be orders higher than the bending stiffness contributing significantly to the bending resistance. However, structural stability problems can occur because of the development of compressive membrane-type forces. This situation is characteristic for thin shell structures.

The membrane-bending interaction in plates (also in shells) is a complex process that is not governed by the absolute magnitude of the out-of-plane deflections because the membrane-type actions developed in the plate mid-surface occur at deflections that are rather small. The problem becomes somewhat more complex when the distribution of the membrane-bending interaction over the plate thickness is considered in the modeling. On the other hand, the large deflection problem in plates should be considered from the point of view of the progressive geometric changes and their magnitude that in certain circumstances overstep the limits for what is considered.

The particular finite element model adopted in this study is based on approximating the actual plate or shell structure by an idealized model obtained by coupling of two quadrilateral finite elements previously used in bending and in-plane linear analysis [35]. The idealized model is based on the general definition of strain in terms of Green's strain tensor and TL formulation taking into account the membrane-bending interaction. The basic concept will be presented in a separate section describing the QFSE finite element.

3.3 Relation of the Large Displacement Problem to Buckling and Collapse Analysis of Thin Structures

Compared to plates, when considering large deformations, the stability problem of shell structures is much more important than the membrane-bending interaction problem developed in plates. It is extremely important to emphasize that in structures governed by stability, the most important design objective is the evaluation of their ultimate load capacity and collapse modes.

The estimation of the maximum load that a structure can support prior to structural instability or collapse depends strongly on the particular behavior characteristics of the structure as they pertain to the magnitude of the displacements, load redistribution, construction imperfections, etc. In the analysis, the load distribution is always known based on pertinent approximations offered by specialized design codes, the design requirements referring usually to structural safety, structural integrity and functionality. In certain situations, analysis of the structural instability modes and ultimate load capacity may become also part of the design requirements. Usually, it is considered that the structural collapse occurs when indefinitely large displacements are produced by a small load increment, [20]. While this notion has some practical significance within the concept of safe design, from the structural point of view, the ultimate load capacity can not be reasonably assessed without a pre- and post-buckling analysis. It should be mentioned that the determination of the actual collapse load of shells or other slender structures present

obvious difficulties as convergence of displacements can no longer be obtained under load control conditions.

The linearised buckling analysis is based on assumptions that usually do not meet accurately the physical reality. For example, it assumes structural geometry free of imperfections. It assumes also that the elastic stiffness matrix K_0 does not change significantly before buckling and the geometric stiffness matrix K_σ is taken simply as a multiple of its initial value by means of the eigenvalue parameters λ_i . In addition, the load redistribution due to the large geometric changes in the structures are not accounted for. Consequently, the linearised buckling analysis can be effectively used to determine the load level at which the structure becomes unstable only if the pre-buckling displacements and their effects are negligible. In the case of the real structures, the structural instability problem is somewhat more difficult; the accuracy of the structural estimations being significantly affected by the absolute magnitude of displacements, load redistribution or material inelasticity.

The stability problem of shells is more complex in analysis due to their high sensitivity to manufacturing imperfections. In [53] is shown an example that a particular specified imperfection within normal engineering tolerances can reduce the critical load by a factor varying from 2 to 10, depending on circumstances. In design, the estimation of the ultimate load capacity of the structural elements can be obtained only quantitatively. Based on certain assumptions that make the analysis simple (perfect geometry, symmetry, etc.), useful approximations of the critical loads can be obtained. At least, the magnitude of such approximate critical loads can be of real help in assessing the safety of a structure.

From a qualitative point of view, the numerical solution of the stability problem of symmetric shells seems to be difficult due to the existence of many simultaneous modes (multiple eigenvalues). Moreover, the nonlinear coupling of unrelated and mild failure modes (characteristic of shells) can lead also to extremely severe post-buckling structural behavior as was predicted by Koiter and Skaloud in [54], a situation largely met in composite structures. In this connection, it can be concluded that the current design procedures based on linear buckling studies (which are uncoupled linear analyses) will simply fail to locate the danger accurately, which if improperly interpreted could lead to unsafe design.

The general procedure in establishing the equilibrium path of a structure is based on the incremental-iterative approach, which applies to the analysis of both the geometric and material nonlinear effects. At this juncture, it should be mentioned that only a large displacement analysis could detect accurately the load level at which buckling has been initiated. Moreover, it permits the evaluation of the post-buckling behavior based on a displacement control technique. In this respect, two illustrative test problems were chosen to highlight the influence of the geometrically nonlinear effects on the structural stability problem, sections 8.3 and 8.4.

4 Formulation of QFSE Facet-Shell Element

4.1 General

This chapter is devoted to the finite element matrix formulation of facet-shell element QFSE developed to analyze the geometrically and material nonlinear response in plate and shell structures. In the first two sections, a brief review of the parent elements QUAD8 and QUAD9 of RQUAD (Refined Quadrilateral Membrane, [46]) and IDKQ (Improved Discrete Kirchhoff Quadrilateral), respectively is provided. In the third section, the matrix formulation of the membrane element RQUAD, which accounts for large in-plane displacements, is presented. In the fourth section, the matrix formulation of the plate bending element IDKQ, which accounts for large out-of-plane displacements, is given. Finally, in the last section of this chapter the facet-shell element QFSE formulation for geometrically and material nonlinear response, with special reference to the membrane-bending effect and based on the combination of the aforementioned RQUAD and IDKQ, is provided. The formulation of IDKQ as presented here is new and the geometrically nonlinear formulation of the layered QFSE has not been previously presented by others.

4.2 Review of Isoparametric Quadrilateral Elements

During the last two decades, it has been recognized that higher-order finite elements, particularly the parabolic isoparametric elements, are quite effective for both in-plane and bending analysis [18], [20]. The finite elements of the isoparametric class were first preferred because of their simplicity in formulation and their relatively good balance between accuracy and computational effort. These elements are also advantageous in modeling 2D structures with curved boundaries and surfaces.

For membrane analysis, the eight-node isoparametric finite element (QUAD8) is largely used [18]. It does not exhibit any spurious mechanisms when selective or full integration schemes are employed [18] and it performs well for quadrilateral shapes. Moreover, QUAD8 can be used for problems involving in-plane bending because of its quadratic shape functions, although it does not have any rotational degree of freedom.

For plate bending analysis, the nine-node Lagrange finite element (QUAD9) based on Mindlin plate theory is considered as offering the best performance for this class of problems [24]. It is mechanism-free when used with full integration schemes [18]. It has also the advantage to be used for both thick and moderately thin shells through appropriate adjustments of the shear-strain energy contribution to the total strain energy of the element [18], [24]. Using finite elements of higher-order than quadratic is not easy to justify as shown through significant examples in [18] where it

was found that their accuracy compared to the required computational effort is not of practical importance.

In the following two sections, both, QUAD8 and QUAD9 will be reviewed separately, highlighting some important aspects of their formulation for linear and nonlinear structural behavior.

4.2.1 Isoparametric Membrane Element QUAD8

The sides of the undeformed finite element QUAD8 (16 DOF) can be straight or curved as shown in Fig 4.1.

Figure 4.1: Straight and Curved Sided QUAD8 Membrane Element

In this figure, the r and s are the dimensionless natural coordinates that vary between -1 and $+1$ over each element. The introduction of the natural coordinate system is required by the numerical integration schemes that are normally applied to

determine the element stiffness or other relevant quantities. The main characteristic of the isoparametric finite elements is the interpolation of the element Cartesian coordinates $x(r,s)$ and $y(r,s)$ in the same way as the element displacements u and v using the same shape functions $N_i(r,s)$

$$\begin{aligned} \begin{bmatrix} x \\ y \end{bmatrix} &= \sum_{i=1}^8 \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix} \begin{bmatrix} x_i \\ y_i \end{bmatrix} \\ \begin{bmatrix} u \\ v \end{bmatrix} &= \sum_{i=1}^8 \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix} \begin{bmatrix} u_i \\ v_i \end{bmatrix} \end{aligned} \quad (4.1)$$

where $i=1,8$ indicates the node numbers and x_i , y_i and u_i , v_i are nodal values of the element Cartesian coordinates and displacements, respectively.

The shape functions N_i , $i=1,8$ expressed in natural coordinates (r,s) are given as

$$\begin{aligned} N_1 &= \frac{1}{4}(1-r)(1-s) - \frac{1}{2}(N_8 + N_5) & N_5 &= \frac{1}{2}(1-r^2)(1-s) \\ N_2 &= \frac{1}{4}(1+r)(1+s) - \frac{1}{2}(N_5 + N_6) & N_6 &= \frac{1}{2}(1+r)(1-s^2) \\ N_3 &= \frac{1}{4}(1+r)(1+s) - \frac{1}{2}(N_6 + N_7) & N_7 &= \frac{1}{2}(1-r^2)(1+s) \\ N_4 &= \frac{1}{4}(1-r)(1+r) - \frac{1}{2}(N_7 + N_8) & N_8 &= \frac{1}{2}(1-r)(1-s^2) \end{aligned} \quad (4.2)$$

The expressions of the shape functions in Eq.(4.2) are especially useful in developing and implementing the so called transition elements by omitting some of the mid-nodes. The transition finite elements are mainly used in mesh refinement techniques in order to ensure the compatibility of the connected element boundaries. Note that the transition finite elements can be connected to both bilinear and biquadratic elements without loss of compatibility.

As stated in Eqs. (4.25), the membrane type strains $[\varepsilon_m]$ could be provided by the QUAD8 element as

$$[\varepsilon_m] = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{bmatrix} = [\varepsilon_0^m] + [\varepsilon_L^m] = \begin{bmatrix} u_{,x} \\ v_{,y} \\ u_{,y} + v_{,x} \end{bmatrix}_0^m + \frac{1}{2} \begin{bmatrix} u_{,x}^2 + v_{,x}^2 \\ u_{,y}^2 + v_{,y}^2 \\ u_{,x}u_{,y} + v_{,x}v_{,y} \end{bmatrix}_L^m \quad (4.3)$$

According to Eq. (4.1), the element displacements are computed interpolating their nodal values by means of the shape functions in Eq. (4.2) as follows

$$\begin{bmatrix} u \\ v \end{bmatrix} = \sum_{i=1}^8 \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix} [a_i^m] = [N_m^m]_{(2 \times 16)} [a_m]_{(16 \times 1)} \quad (4.4)$$

The linear membrane strain components $[\varepsilon_0^m]$ in Eq. (4.3) can be expressed according to Eq. (4.4) in terms of Cartesian derivatives of the shape functions $[N_m]$ and nodal displacements $[a_m]$ as

$$[\varepsilon_0^m] = \begin{bmatrix} u_{,x} \\ v_{,y} \\ u_{,y} + v_{,x} \end{bmatrix} = \sum_{i=1}^8 [B_0^m]_i [a_i^m] = [B_0^m]_{(3 \times 16)} \times [a_m]_{(16 \times 1)} \quad (4.5)$$

where $[B_0^m]_i$ represents the strain-displacement transformation matrix for node i and may be expressed as

$$[B_0^m]_i = \begin{bmatrix} N_{i,x} & 0 \\ 0 & N_{i,y} \\ N_{i,y} & N_{i,x} \end{bmatrix} \quad (4.6)$$

Thus, the total strain-displacement transformation matrix of element QUAD8 may be written as

$$[B_0^m] = \begin{bmatrix} [B_0^m]_1 & [B_0^m]_2 & \cdots & [B_0^m]_7 & [B_0^m]_8 \end{bmatrix}$$

To evaluate the Cartesian shape function derivatives in Eq. (4.6), the chain rule of differentiation must be employed

$$\begin{aligned} N_{i,x} &= N_{i,r} r_{,x} + N_{i,s} s_{,x} \\ N_{i,y} &= N_{i,r} r_{,y} + N_{i,s} s_{,y} \end{aligned} \quad \text{or} \quad (4.7)$$

$$\begin{bmatrix} N_{i,x} \\ N_{i,y} \end{bmatrix} = [J]^{-1} \begin{bmatrix} N_{i,r} \\ N_{i,s} \end{bmatrix}$$

where $[J]^{-1}$ is the inverse of the Jacobian transformation matrix of element QUAD8 that is defined as

$$[J] = \begin{bmatrix} x_{,r} & y_{,r} \\ x_{,s} & y_{,s} \end{bmatrix} \quad (4.8)$$

Using matrix inversion, the inverse of $[J]$ may be expressed as

$$[J]^{-1} = \begin{bmatrix} r_{,x} & s_{,x} \\ r_{,y} & s_{,y} \end{bmatrix} = \frac{1}{|J|} \begin{bmatrix} y_{,s} & -x_{,s} \\ -y_{,r} & x_{,r} \end{bmatrix} = \frac{1}{|J|} \begin{bmatrix} \sum_{i=1}^8 N_{i,s} y_i & -\sum_{i=1}^8 N_{i,s} x_i \\ -\sum_{i=1}^8 N_{i,r} y_i & \sum_{i=1}^8 N_{i,r} x_i \end{bmatrix} \quad (4.9)$$

where

$$|J| = x_{,r} y_{,s} - x_{,s} y_{,r} = \left(\sum_{i=1}^8 N_{i,r} x_i \right) \left(\sum_{i=1}^8 N_{i,s} y_i \right) - \left(\sum_{i=1}^8 N_{i,s} x_i \right) \left(\sum_{i=1}^8 N_{i,r} y_i \right) \neq 0$$

is the determinant of the Jacobian transformation matrix.

As can be observed in Eq. (4.9), the inverse of the Jacobian matrix can be easily computed from the nodal values of the shape function derivatives and nodal

Cartesian coordinates. This might be the most important advantage of the isoparametric formulation of element geometry.

The nonlinear membrane strain components $[\varepsilon_L^m]$ in Eq. (4.3) can be also expressed also in terms of nodal displacements $[a_m]$ as follows

$$[\varepsilon_L^m] = \frac{1}{2} \begin{bmatrix} u_x^2 + v_x^2 \\ u_y^2 + v_y^2 \\ u_x u_y + v_x v_y \end{bmatrix} = \frac{1}{2} \begin{bmatrix} u_x & 0 & v_x & 0 \\ 0 & u_y & 0 & v_y \\ u_y & u_x & v_y & v_x \end{bmatrix} \begin{bmatrix} u_x \\ u_y \\ v_x \\ v_y \end{bmatrix} = \frac{1}{2} [A_m] \times [\theta_m] \quad (4.10)$$

where

$$[\theta_m] = \sum_{i=1}^4 \begin{bmatrix} N_{i,x} & 0 \\ N_{i,y} & 0 \\ 0 & N_{i,x} \\ 0 & N_{i,y} \end{bmatrix} \times [a_i] = [G_m]_{(4 \times 16)} \times [a_m]_{(16 \times 1)} \quad (4.11)$$

As can be observed, all terms of the $[G_m]$ matrix in Eq. (4.11) are constant; they depend on the Cartesian first order derivatives of the shape functions that are computed once at the beginning of the analysis. In the computer code, the $[G_m]$ matrix is assembled with the terms of the linear strain-displacement matrix $[B_0^m]$ determined from Eq. (4.6).

According to Eqs. (4.10), (4.11) and knowing that $A(d\theta) = (dA)\theta$, the nonlinear membrane strain-displacement transformation matrix $[B_L^m]$ can be easily found taking a strain variation of $[\varepsilon_L^m]$ with respect to the nodal displacements $[a_m]$ as follows

$$d[\varepsilon_L^m] = \frac{1}{2} d([A_m] \times [G_m])[a_m] + \frac{1}{2} [A_m] \times [G_m] d([a_m]) = [A_m] \times [G_m] \times d([a_m]) = [B_L^m] \times d([a_m])$$

or

$$[B_L^m] = [A_m]_{(3 \times 4)} \times [G_m]_{(4 \times 16)} \quad (4.12)$$

The components of matrix $[A_m]$ are known quantities that are obtained from the last converged solution. The nonlinear procedure is started with the solution of a linear computation, usually the predictor solution obtained from the first load increment.

Finally, the element stiffness matrix of QUAD8 can be typically expressed as

$$[K_m]_{(16 \times 16)} = t \int_{-1}^{+1} \int_{-1}^{+1} [B_m(r, s)]^T [D_m] [B_m(r, s)] |J| dr ds \quad (4.13)$$

where $t = \text{constant}$ is the element thickness, $[B_m] = [B_0^m] + [B_L^m]$, as in Eq. (1.13), and the differential element area $dx dy = |J| dr ds$.

The work equivalent nodal load vector $[q_m]$ associated with QUAD8 element is obtained from specified external membrane actions $[q] = [q(x, y)]$ as

$$[q_m] = |J| \int_{-1}^{+1} \int_{-1}^{+1} [q] dr ds \quad (4.14)$$

where

$$[q] = \sum_{i=1}^8 \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix}^T \begin{bmatrix} q_x^i \\ q_y^i \end{bmatrix} \quad (4.15)$$

where q_x^i and q_y^i are the values of the continuous external load $[q(x, y)]$ at node i .

4.2.2 Isoparametric Bending Element QUAD9

The Lagrange isoparametric plate bending finite element QUAD9 (27 DOF's) is a nine-node element with products of complete second order Lagrangean polynomials as shape functions that can properly represent pure bending even for nonrectangular element shapes. The geometry of QUAD9 is derived from the eight boundary nodes as in the case of the quadratic serendipity element and it might have straight or curved boundaries as shown in Figure 4.2. However, QUAD9 is much less sensitive to nonrectangular shapes and curvature of sides than the serendipity element, due to the completeness property of the Lagrange polynomials that are used to construct the shape functions.

Figure 4.2: Straight and Curved Sided QUAD9 Bending Element

The behavior of the QUAD9 element is based on the assumption of the Mindlin plate bending theory, namely that normals on the mid-surface of the element remain straight during the element deformation process, but not necessarily normal. According to the plane stress assumptions, no transverse normal stress is developed (*i.e.*, $\sigma_z = 0$), but transverse shear deformation is allowed. In addition, the motion of a particle located not on the element mid-surface is governed by the rotations θ_x and θ_y of the lines that were normal to the element mid-surface of the undeformed plate as shown in Figure 4.3.

Figure 4.3: Deformation of Mindlin Plate Element

Thus, with θ_x and θ_y small rotation angles, the displacements of a point P in Figure 4.3 can be expressed as

$$u = -z\theta_x \quad v = -z\theta_y \quad w = w \quad (4.16)$$

The DOF's of QUAD9 consist of out-of-plane nodal displacements w_i and nodal rotations θ_{xi} and θ_{yi} of the normal at the element mid-surface. According to the Mindlin assumption, the shears are expressed as $\gamma_{xz} = w_{,x} - \theta_x$ and $\gamma_{yz} = w_{,y} - \theta_y$ which implies $\theta_x \neq w_{,x}$ and $\theta_y \neq w_{,y}$ unless the thin plate assumption is involved for which it can be assumed $\gamma_{yz} = \gamma_{xz} = 0$ (Kirchhoff theory).

Thus, from Eq. (2.25), the bending strains may be expressed as

$$[\varepsilon_b] = (-z) \begin{bmatrix} \theta_{x,x} \\ \theta_{y,y} \\ \theta_{x,y} + \theta_{y,x} \end{bmatrix} + \frac{1}{2} \begin{bmatrix} w_{,x}^2 \\ w_{,y}^2 \\ w_{,x}w_{,y} \end{bmatrix} = [\varepsilon_0^b] + [\varepsilon_L^b] \quad (4.17)$$

In Eq. (4.17), the transverse shear-strains $\gamma_{yz} = w_{,y} - \theta_y$ and $\gamma_{xz} = w_{,x} - \theta_x$ were not included. The QUAD9 element based on Mindlin theory is mainly applicable to thick and moderately thin plates. Through proper adjustments of the shear strain contributions to the strain energy, this finite element can be successfully used for thin plates also [24]. The main advantage of the elements based on Mindlin theory is that the out-of-plane displacements are independent of the rotations θ_x and θ_y of the normals, these elements requiring only a $C^{(0)}$ -continuity that can be easily satisfied. It should be noted that QUAD9 can represent pure bending properly with any number of Gauss points.

Thus, the displacements w and the rotations of the normals θ_x , θ_y at any point (r, s) may be interpolated by the same shape functions $N_i = N_i(r, s)$ as follows

$$\begin{bmatrix} w \\ \theta_x \\ \theta_y \end{bmatrix} = \sum_{i=1}^9 \begin{bmatrix} N_i & 0 & 0 \\ 0 & N_i & 0 \\ 0 & 0 & N_i \end{bmatrix} \begin{bmatrix} w_i \\ \theta_{xi} \\ \theta_{yi} \end{bmatrix} \quad (4.18)$$

where i represents node numbers.

The expressions of the shape functions in Eq. (4.18) are given in natural coordinates (r, s) as follows

$$\begin{aligned} N_1 &= \frac{1}{4}(1-r)(1-s) - \frac{1}{2}(N_8 + N_5) - \frac{1}{4}N_9 & N_5 &= \frac{1}{2}(1-r^2)(1-s) - \frac{1}{4}N_9 \\ N_2 &= \frac{1}{4}(1+r)(1+s) - \frac{1}{2}(N_5 + N_6) - \frac{1}{4}N_9 & N_6 &= \frac{1}{2}(1+r)(1-s^2) - \frac{1}{4}N_9 \\ N_3 &= \frac{1}{4}(1+r)(1+s) - \frac{1}{2}(N_6 + N_7) - \frac{1}{4}N_9 & N_7 &= \frac{1}{2}(1-r^2)(1+s) - \frac{1}{4}N_9 \\ N_4 &= \frac{1}{4}(1-r)(1+r) - \frac{1}{2}(N_7 + N_8) - \frac{1}{4}N_9 & N_8 &= \frac{1}{2}(1-r)(1-s^2) - \frac{1}{4}N_9 \\ N_9 &= (1-r^2)(1-s^2) \end{aligned} \quad (4.19)$$

By substitution of Eq. (4.18) into (4.17), the linear bending strains $[\varepsilon_0^b]$ of QUAD9 can be expressed as

$$[\varepsilon_0^b] = (-z) \begin{bmatrix} \theta_{x,x} \\ \theta_{y,y} \\ \theta_{y,x} + \theta_{x,y} \end{bmatrix} = (-z) \sum_{i=1}^9 [B_0^b]_i \times [a_i^b] = (-z) [B_0^b]_{(3 \times 27)} [a_b]_{(27 \times 1)} \quad (4.20)$$

where $[B_0^b]_i$ is the curvature-displacement transformation matrix of node i written as

$$[B_0^b]_i = \begin{bmatrix} 0 & N_{i,x} & 0 \\ 0 & 0 & N_{i,y} \\ 0 & N_{i,y} & N_{i,x} \end{bmatrix} \quad (4.21)$$

Thus, the total curvature-displacement transformation matrix $[B_0^b]_{(3 \times 27)}$ of element QUAD9 can be easily assembled as $[B_0^b] = [[B_0^b]_1 \quad [B_0^b]_2 \quad \dots \quad [B_0^b]_8 \quad [B_0^b]_9]$.

The shear-strains can be derived from Eq. (4.18) in terms of a (shear) strain-displacement transformation matrix $[B^s]_i$ in the following manner

$$[\varepsilon^s] = \begin{bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{bmatrix} = \begin{bmatrix} \theta_x - w_{,x} \\ \theta_y - w_{,y} \end{bmatrix} = \sum_{i=1}^9 \begin{bmatrix} -N_{i,x} & N_i & 0 \\ -N_{i,y} & 0 & N_i \end{bmatrix} \begin{bmatrix} w_i \\ \theta_{xi} \\ \theta_{yi} \end{bmatrix} = \sum_{i=1}^9 [B^s]_i [a_i^b] = [B^s]_{(2 \times 27)} [a_b]_{(27 \times 1)} \quad (4.22)$$

The isoparametric coordinate transformation is given by

$$\begin{bmatrix} x \\ y \end{bmatrix} = \sum_{i=1}^9 \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix} \begin{bmatrix} x_i \\ y_i \end{bmatrix} \quad (4.23)$$

where x_i and y_i are the nodal coordinates of the QUAD9 element. The evaluation of the Cartesian shape function derivatives follows the chain rule of differentiation

$$\begin{aligned} N_{i,x} &= N_{i,r} r_{,x} + N_{i,s} s_{,x} \\ N_{i,y} &= N_{i,r} r_{,y} + N_{i,s} s_{,y} \end{aligned} \quad (4.24)$$

or

$$\begin{bmatrix} N_{i,x} \\ N_{i,y} \end{bmatrix} = [J]^{-1} \begin{bmatrix} N_{i,r} \\ N_{i,s} \end{bmatrix}$$

The Jacobian of the coordinate transformation and its inverse are expressed similarly to Eqs. (4.8) and (4.9) as

$$[J] = \begin{bmatrix} x_{,r} & y_{,r} \\ x_{,s} & y_{,s} \end{bmatrix} \quad (4.25)$$

$$[J]^{-1} = \begin{bmatrix} r_{,x} & s_{,x} \\ r_{,y} & s_{,y} \end{bmatrix} = \frac{1}{|J|} \begin{bmatrix} y_{,s} & -x_{,s} \\ -y_{,r} & x_{,r} \end{bmatrix} = \frac{1}{|J|} \begin{bmatrix} \sum_{i=1}^9 N_{i,s} y_i & -\sum_{i=1}^9 N_{i,s} x_i \\ -\sum_{i=1}^9 N_{i,r} y_i & \sum_{i=1}^9 N_{i,r} x_i \end{bmatrix} \quad (4.26)$$

where

$$|J| = x_{,r} y_{,s} - x_{,s} y_{,r} = \left(\sum_{i=1}^9 N_{i,r} x_i \right) \left(\sum_{i=1}^9 N_{i,s} y_i \right) - \left(\sum_{i=1}^9 N_{i,s} x_i \right) \left(\sum_{i=1}^9 N_{i,r} y_i \right) \neq 0$$

The nonlinear strain components $[\varepsilon_L^b]$ in Eq. (4.17) can be expressed also in terms of nodal displacements $[a_b]$ as follows

$$[\varepsilon_L^b] = \begin{bmatrix} \frac{1}{2} w_{,x}^2 \\ \frac{1}{2} w_{,x}^2 \\ w_{,x} w_{,y} \end{bmatrix} = \frac{1}{2} \begin{bmatrix} w_{,x} & 0 \\ 0 & w_{,y} \\ w_{,y} & w_{,x} \end{bmatrix} \begin{bmatrix} w_{,x} \\ w_{,y} \end{bmatrix} = \frac{1}{2} [A_b] \times [\theta_b] \quad (4.27)$$

where

$$[\theta_b] = \sum_{i=1}^9 \begin{bmatrix} N_{i,x} & 0 & 0 \\ N_{i,y} & 0 & 0 \end{bmatrix} [a_i^b] = [G_b]_{(2 \times 27)} \times [a_b]_{(27 \times 1)} \quad (4.28)$$

As can be observed, all terms of $[G_b]$ matrix in Eq. (4.28) are constant; they depend on the Cartesian derivatives of the shape functions that are computed once at the beginning of the analysis. In the computer code, $[G_b]$ matrix is assembled with the terms of the linear strain-displacement matrix $[B_0^b]$ determined from Eq. (4.21).

According to Eqs. (4.27), (4.28) and knowing that $A(d\theta) = (dA)\theta$, the nonlinear bending strain-displacement transformation matrix $[B_L^b]$ can be easily found taking a strain variation of $[\varepsilon_L^b]$ with respect to the nodal displacements $[a_b]$ as follows

$$d[\varepsilon_L^b] = \frac{1}{2} d([A_b] \times [G_b])[a_b] + \frac{1}{2} [A_b] \times [G_b] d([a_b]) = [A_b] \times [G_b] \times d([a_b]) = [B_L^b] \times d([a_b])$$

or

$$[B_L^b] = [A_b]_{(3 \times 2)} \times [G_b]_{(2 \times 27)} \quad (4.29)$$

In Eq. (4.29), the matrix $[A_b]$ is assembled with the displacement derivatives obtained in the previous converged solution. In order to be able to construct the matrix $[A_b]$, the incremental-iterative procedure is started from the solution of a linear computation, usually the predictor solution obtained from the first load increment.

The bending and the shear stiffness matrices of QUAD9 can be computed from

$$\begin{aligned} [K_b]_{(27 \times 27)} &= \int_{-1}^{+1} \int_{-1}^{+1} [B_b(r, s)]^T [D_b] [B_b(r, s)] |J| dr ds \\ [K_s]_{(18 \times 18)} &= \int_{-1}^{+1} \int_{-1}^{+1} [B_s(r, s)]^T [D_s] [B_s(r, s)] |J| dr ds \end{aligned} \quad (4.30)$$

where t is the element thickness assumed constant, $[B_b] = [B_0^b] + [B_L^b]$ as in Eq. (1.13)

and $[B^s]$ is obtained from Eq. (4.22). The differential element area is $dxdy = |J| dr ds$.

For the special case of linear elastic and isotropic plate, the elasticity matrices in Eq. (4.30) are given by

$$[D_b] = \frac{Et^3}{12(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \quad [D_s] = \frac{Et}{2(1+\nu)} \begin{bmatrix} \gamma & 0 \\ 0 & \gamma \end{bmatrix}$$

in which E is the Young's modulus, ν is the Poisson's ratio and γ is a shear modification factor that takes into account the warping of the element section. Note

that for rectangular sections $\gamma = 5/6$. Finally, the stiffness matrix assemblage of the QUAD9 element is given by

$$[K]_{(45 \times 45)}^{QUAD9} = \int_{-1}^{+1} \int_{-1}^{+1} \begin{bmatrix} [K]_{27 \times 27}^b & 0 \\ 0 & [K]_{18 \times 18}^s \end{bmatrix} |J| dr ds \quad (4.31)$$

The work equivalent nodal loads $[q_b]$ of the uniform distributed loads associated with QUAD9 element is obtained from specified external out-of-plane actions only, so that

$$[q_b]_i = \int_{-1}^{+1} \int_{-1}^{+1} \begin{bmatrix} N_i & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix}^T \begin{bmatrix} q_{\bar{x}} \\ 0 \\ 0 \end{bmatrix} |J| dr ds \quad (4.33)$$

where $q_{\bar{x}}$ are the values of the distributed load at node i .

4.3 Refined Quadrilateral Membrane RQUAD

Razaqpur and Nofal (1987) developed a quadrilateral membrane element with drilling DOF's called RQUAD using the classical beam displacement function. The formulation of this element is based on a DOF's transformation approach that maps the in-plane mid-node displacements u_{i+4}, v_{i+4} ($i = 1, 2, 3, 4$) of the straight-sided

serendipity element QUAD8 to a four-node quadrilateral RQUAD with in-plane displacements u_i , v_i and drilling rotations ω_i as shown in Figure 4.4.

Figure 4.4: Transformation of Element QUAD8 to Element RQUAD

Thus, the DOF's transformation from QUAD8 to RQUAD can be expressed by means of a transformation matrix $[T]$ as follows

$$[d]_{(16 \times 1)}^{QUAD8} = [T]_{(16 \times 12)} [d]_{(12 \times 1)}^{RQUAD4} \quad (4.34)$$

In the same manner, the nodal loads transformation may be expressed by means of matrix $[T]$ as

$$[T]_{(12 \times 16)}^T [F]_{(16 \times 1)}^{QUAD8} = [F]_{(12 \times 1)}^{RQUAD4} \quad (4.35)$$

The transformation matrix $[T]$ maps the mid-node displacements u_{i+4} , v_{i+4} of QUAD8 element into equivalent drilling (in-plane) rotations ω_i , ω_{i+} at the

corresponding corner nodes for $i = 1, 2, 3, 4$ and $i+ = 2, 3, 4, 1$ (Crisfield's notation, [16]).

In [46], Razaqpur and Nofal used the classical beam displacement function to reproduce the variation of the normal displacement δ_n along the element sides and a linear function to represent the variation of the tangential displacements δ_t as shown in Figure 4.5.

Figure 4.5: Typical Nodal Displacements along the Sides of QUAD8 and RQUAD

For a QUAD8 element side $(i, i+)$ in Figure 4.5, the in-plane displacements u_{i+4} and v_{i+4} can be related to the normal displacement δ^n_{i+4} at the mid-side node $i+4$ by a simple displacement transformation as

$$u_{i+4} = \frac{u_i + u_{i+}}{2} - \delta^n_{i+4} \sin \theta_i \quad (4.36)$$

$$v_{i+4} = \frac{v_i + v_{i+}}{2} + \delta_{i+4}^n \cos \theta_i$$

In Eq. (4.3), δ_{i+4}^n is obtained through a superposition of the normal displacements $(\delta_n)_{\omega_i}$ and $(\delta_n)_{\omega_{i+}}$ that are produced by independent drilling rotations ω_i and ω_{i+} as

$$\delta_{i+4}^n = \left\langle (\delta_n)_{\omega_i} + (\delta_n)_{\omega_{i+}} \right\rangle_{s=\frac{1}{2}l_i} \quad (4.37)$$

where

$$(\delta_n)_{\omega_i} = \omega_i \left((\delta_n)_{\omega_i=1} \right) \quad \text{and} \quad (\delta_n)_{\omega_{i+}} = \omega_{i+} \left((\delta_n)_{\omega_{i+}=1} \right) \quad (4.38)$$

From the classical beam displacement function, the normal displacements $(\delta_n)_{\omega_i=1}$ and $(\delta_n)_{\omega_{i+}=1}$ produced by unit end-rotations $\omega_i = 1$ and $\omega_{i+} = 1$ respectively, are

$$(\delta_n)_{\omega_i=1} = -\frac{s(l_i - s)^2}{l_i^2} \quad (\delta_n)_{\omega_{i+}=1} = \frac{(l_i - s)s^2}{l_i^2} \quad (4.39)$$

where l_i is the length of the element side $(i, i+)$ and s is the local coordinate $0 \leq s \leq l_i$ as shown in Figure 4.5.

Thus, the normal displacement δ_{i+4}^n can be expressed in terms of in-plane rotations ω_i and ω_{i+} by substituting Eqs. (4.39) into (4.38) and (4.38) into (4.37) for $s = \frac{1}{2}l_i$

$$\delta_{i+4}^n = \frac{l_i}{8} (\omega_{i+} - \omega_i) \quad (4.40)$$

According to Eqs. (4.36) and (4.40), the in-plane displacements u_{i+4} and v_{i+4} , ($i = 1, \dots, 4$) at the mid-nodes of QUAD8 element can be directly related to the in-plane rotations ω_i and ω_{i+} of the RQUAD element as follows

$$\begin{aligned} u_{i+4} &= \frac{u_i + u_{i+}}{2} - (y_i - y_{i+}) \frac{(\omega_{i+} - \omega_i)}{8} \\ v_{i+4} &= \frac{v_i + v_{i+}}{2} + (x_{i+} - x_i) \frac{(\omega_{i+} - \omega_i)}{8} \end{aligned} \quad (4.41)$$

where $\sin \theta_i = \frac{y_{i+} - y_i}{l_i}$ and $\cos \theta_i = \frac{x_{i+} - x_i}{l_i}$. In matrix form, Eq. (4.41) may be expressed as

$$\begin{bmatrix} u_{i+4} \\ v_{i+4} \end{bmatrix} = \begin{bmatrix} \frac{1}{2} & 0 & a_i & \frac{1}{2} & 0 & -a_i \\ 0 & \frac{1}{2} & -b_i & 0 & \frac{1}{2} & b_i \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ \omega_i \\ u_{i+} \\ v_{i+} \\ \omega_{i+} \end{bmatrix} \quad \text{for } \begin{matrix} i = 1, 2, 3, 4 \\ i+ = 2, 3, 4, 1 \end{matrix} \quad (4.42)$$

where

$$a_i = \frac{y_i - y_{i+}}{8} \quad \text{and} \quad b_i = \frac{x_i - x_{i+}}{8} \quad \text{for } \begin{matrix} i = 1, 2, 3, 4 \\ i+ = 2, 3, 4, 1 \end{matrix} \quad (4.43)$$

Finally, according to Eq. (4.34), all DOF's in QUAD8 can be related to the corresponding DOF's in RQUAD by means of the transformation matrix $[T]_{(16 \times 12)}$ (explicitly given in [26], Appendix I) as

$$[d]_{(16 \times 1)}^{QUAD8} = [T]_{(16 \times 12)} [d]_{(12 \times 1)}^{RQUAD4}$$

The transformed stiffness can be easily derived by substitution of Eqs. (4.34) and (4.35) in the stiffness relationship of element QUAD8

$$[K]^{(QUAD8)} [T] [d]^{(RQUAD4)} = [F]^{(QUAD8)} \quad (4.44)$$

and by pre- multiplying Eq. (4.11) by $[T]^T$, the stiffness matrix of element RQUAD may be written as

$$[K]^{(RQUAD4)} = [T]^T [K]^{(QUAD8)} [T] \quad (4.45)$$

As it can be observed, Eq. (4.45) does not change the element properties or its orientation in the global coordinate system by any means. Rather, it represents a static condensation of the in-plane mid-nodes displacements of QUAD8 element in terms of in-plane displacements and rotations at the corner nodes of RQUAD element.

In Eq. (4.45) further simplifications are possible in order to reduce the amount of calculations required by the numerical integration by rewriting $[K]^{RQUAD4}$ as suggested by Razaqpur and Nofal in [46]

$$[K]^{(RQUAD4)} = [T]^T \left(\int [B]_{QUAD8}^T [D] [B]_{QUAD8} dV \right) [T]$$

or

$$[K]^{(RQUAD4)} = \int [T]^T [B]_{QUAD8}^T [D] [B]_{QUAD8} [T] dV = \int [B]_{RQUAD4}^T [D] [B]_{RQUAD4} dV$$

where

$$[B]_{(3 \times 12)}^{RQUAD4} = [B]_{(3 \times 16)}^{QUAD8} [T]_{(16 \times 12)} \quad (4.46)$$

The matrix multiplication in Eq. (4.46) can be further optimized by a selective component-by-component matrix multiplication of the non-zero terms. Note that $\approx 75\%$ of the $[T]$ matrix consists of zero components.

Finally, the stiffness matrix of the RQUAD element may be expressed as

$$[K]_{(12 \times 12)}^{RQUAD4} = \int_{-1}^{+1} \int_{-1}^{+1} [B]_{RQUAD4}^T [D]_m [B]_{RQUAD4} |J| d\xi d\eta \quad (4.47)$$

in which $[B]_{RQUAD4}$ is the strain-transformation matrix of RQUAD element, $|J|$ is the determinant of the Jacobian matrix of the coordinate transformation and $[D]$ is the plane stress elasticity matrix

$$[D]_m = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

in which E and ν are the Young's modulus and the Poisson's ratio respectively.

It should be noted that $[T]_{(16 \times 12)}$ not being square, it can not be inverted. For this reason, the DOF's transformation by means of $[T]$ matrix can be done only from QUAD8 to RQUAD.

It should be noted that the rotational DOF's ω_i and ω_{i+} of RQUAD do not represent the elastic rotations of the element nodes as defined in the theory of elasticity, i.e.

$$\Omega = \frac{1}{2} \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) \quad (4.48)$$

The beam formula in Eqs. (4.38) and (4.39) involves rotations ω_i and ω_{i+} that physically represent the rotations of the tangents of the deformed shape from the chord joining the nodes i and $i+$ as shown in Figure 4.6 .

From geometry in Figure 4.6, the relationship between the drilling rotations ω_i and the total corresponding rotations Ω_i can be established as

$$\Omega_i = \omega_i + \varphi_i$$

Extensive numerical analysis tests on linear elasticity problems performed on RQUAD in [26], [46] and [48] have shown its high reliability and computational effectiveness. It was also found that RQUAD passes the patch test and due to the rotational DOF's, this element does not exhibit the so-called "Irons' paradox" as shown in [48]. Because the in-plane forces and the moments are in equilibrium independently, the RQUAD element has a zero energy mode that may be obtained for $u_i = 0, v_i = 0$ and $\omega_i = \text{constant}$. However, this zero energy mode can be suppressed by fixing a rotational DOF in the mesh that might be required by the boundary conditions of many real structures.

Based on the above element development of RQUAD, its extension for geometric nonlinear effects in terms of TL becomes straightforward. In the formulation of the stiffness matrix, the usual engineering strains are replaced by the

Green's strains accounting for large displacements. According to Eq. (2.13), the elastic and the large displacement stiffness matrices of RQUAD are obtained together as shown in Eq. (2.21). In addition, the Eq. (4.14) remains unchanged when geometric nonlinear effects are considered. Because of computational reasons, it is necessary to follow Eq. (2.13) before computing the element stiffness matrix, minimizing in this manner the amount of matrix multiplication and reducing the round-off errors at the same time. The tangent stiffness matrix of RQUAD is completed by the geometric stiffness matrix that takes into account the effect of initial stresses according to Eq. (2.20). The geometric stiffness matrix plays the key role in the buckling and large displacements analysis.

Figure 4.6: Drilling Rotations versus Elastic Rotations

4.4 Improved Discrete Kirchhoff Quadrilateral IDKQ

Finite elements based on the Discrete Kirchhoff Approach are designed for the analysis of thin plates where the transverse shear energy may be considered negligible compared to the bending energy. The essential step in their formulation is the enforcement of the so-called Kirchhoff constraints ($\gamma_{yz} = \gamma_{zx} = 0$) at certain discrete points in such a manner that all shear-strain modes are totally suppressed. Feasible discrete points are the element nodes where the displacement modes are represented by the discrete values of the nodal displacements.

The starting point in the formulation of the IDKQ is the Mindlin isoparametric finite element QUAD9 that accounts for shear deformations and requires only $C^{(0)}$ -continuity. By imposing the Kirchhoff constraints on all shear strain modes, the 27 DOF's (9×3) of the QUAD9 may be transformed into 12 equivalent DOF's (4×3) of the IDKQ element. It should be noted that according to the Discrete Kirchhoff constraints, the shear stiffness in Eq. (4.31) vanishes. In Eq. (4.30), the bending stiffness depends only on the rotational DOF's according to Eq. (4.21). Consequently, the size of the bending stiffness matrix of the QUAD9 element under Kirchhoff mode can be reduced to a (18×18) matrix. In what follows, note that the displacement and load vectors of the original QUAD9 element are reduced correspondingly to (18×1) vectors by removing all out-of-plane nodal displacements.

Similarly to the displacement transformation of RQUAD4, the aim of the DOF's transformation in Figure 4.7 is to derive the transformation matrix $[T]$ that maps the DOF's of the straight sided QUAD9 into equivalent DOF's of IDKQ element.

$$[d]^{QUAD9} = [T] [d]^{IDKQ4} \quad (4.49)$$

Figure 4.7: Transformation of Element QUAD9 to Element IDKQ

The nodal loads transformation may be expressed also by means of matrix $[T]$ as

$$[T]^T [F]^{QUAD9} = [F]^{IDKQ4} \quad (4.50)$$

but, this transformation is of less interest as it will be shown later in this section.

The transformation matrix $[T]$ in Eqs. (4.49) and (4.50) is determined relating the mid-side nodes and central node DOF's of QUAD9 in terms of the corner nodes DOF's by imposing the Kirchhoff constraints in such a way that $\gamma_z = 0$ is exactly

represented over the element. It should be noted that the element strains and the strain energy in QUAD9 depend only on θ_x and θ_y rotations that are independent of the out-of-plane displacements w . Consequently, only the eighteen rotational DOF's $\theta_{xi} \neq w_{,xi}$ and $\theta_{yi} \neq w_{,yi}$ at nodes 1 through 9 of the QUAD9 must be expressed in terms of w_i , $w_{,xi} = \theta_{xi}$ and $w_{,yi} = \theta_{yi}$ DOF's at the corners of IDKQ element. This DOF's transformation is achieved by imposing the Kirchhoff constraints at the nodal points (corresponding to the element sides and centerlines) as follows

- 1) Transverse shear strains γ_{yz} and γ_{zx} vanish at the corner nodes

$$\theta_{xi} = w_{,xi} \quad \theta_{yi} = w_{,yi} \quad \text{at nodes } i = 1, 2, 3, 4, 9 \quad (4.51)$$

- 2) Transverse shear strains γ_{sz} (expressed in edge tangent coordinate s) vanish at the mid-side nodes so that

$$\theta_{sk} = w_{,sk} \quad \text{at nodes } k = 5, 6, 7, 8 \quad (4.52)$$

- 3) Normal slopes $w_{,n} = \theta_n$ of the cubic displacement function along the element sides are assumed to vary linearly

$$\theta_{nk} = \frac{1}{2}(\theta_{ni} + \theta_{nj}) \quad \text{at nodes } k = 5, 6, 7, 8 \quad \text{for } \begin{matrix} i = 1, 2, 3, 4 \\ j = 2, 3, 4, 1 \end{matrix} \quad (4.53)$$

According to the Eqs. (4.51), (4.52) and (4.53), the eighteen DOF's θ_{xm} and θ_{ym} , ($m = 1, \dots, 9$) of the QUAD9 element may be related in terms of the twelve

DOF's w_i , $w_{,xi} = \theta_{xi}$ and $w_{,yi} = \theta_{yi}$ ($i = 1, \dots, 4$) at the corners of the IDKQ element. Symbolically, this DOF's transformation is represented by Eq. (4.1).

In order to determine the transformation matrix $[T]$, the foregoing Kirchhoff constraints are applied along the element sides and centerlines making use of the corresponding coordinate transformations and a redefinition of the displacement field w .

As stated in Eq. (4.18), the rotations of the QUAD9 mid-surface are interpolated using quadratic Lagrangian polynomials.

$$\begin{bmatrix} \theta_x \\ \theta_y \end{bmatrix} = \sum_{i=1}^9 \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix} \begin{bmatrix} \theta_{xi} \\ \theta_{yi} \end{bmatrix} \quad (4.54)$$

The out-of-plane displacements w must be assumed cubic in the edge tangent coordinate s along the element sides in order to be able to satisfy the Kirchhoff constraints as stated in Eqs. (4.51), (4.52) and (4.53) ($w_{,x} = \theta_x$, $w_{,y} = \theta_y$)

$$w_{\zeta} |_{side(ij)} = \frac{1}{4} \begin{bmatrix} 2 - 3\zeta + \zeta^3 & 2 + 3\zeta - \zeta^3 \end{bmatrix} \begin{bmatrix} w_i \\ w_j \end{bmatrix} + \frac{l_{ij}}{8} (\zeta^2 - 1) \begin{bmatrix} \zeta - 1 & \zeta + 1 \end{bmatrix} \begin{bmatrix} \theta_{si} \\ \theta_{sj} \end{bmatrix} \quad (4.55)$$

for $\begin{matrix} i = 1, 2, 3, 4 \\ j = 2, 3, 4, 1 \end{matrix}$

where ζ and l_{ij} are the natural coordinate and the length of the element side ij ; θ_{si} , and θ_{sj} are the nodal tangential rotations along the element side ij at the end nodes i and j respectively.

The expression in Eq. (4.55) is easily obtained from a standard cubic Lagrangian in natural coordinate $-1 \leq \zeta \leq +1$ so that $\zeta = 0$ at the mid-side nodes. In order to express the rotations $\theta_s = w_{,s}$, the derivatives of w_ζ in Eq. (4.55) must be related to the physical coordinate s by means of a coordinate transformation as shown in Figure 4.8.

Figure 4.8: Coordinate Transformation along of a QUAD9 Side

Thus, the rotations $\theta_{sk} = w_{,sk}$, ($k = 5, 6, 7, 8$) at the mid-nodes of QUAD9 element may be expressed after differentiation in Eq. (4.55) with respect to s , substituting first $\zeta = \left(\frac{2}{l_{ij}}\right)s - 1$

$$\theta_{sk} = -\frac{3}{2l_{ij}}w_i - \frac{1}{4}\theta_{si} + \frac{3}{2l_{ij}}w_j - \frac{1}{4}\theta_{sj} \quad (4.56)$$

or in matrix form

$$\begin{bmatrix} \theta_{sk} \\ \theta_{nk} \end{bmatrix} = \begin{bmatrix} -\frac{3}{2l_{ij}} & -\frac{1}{4} & 0 & \frac{3}{2l_{ij}} & -\frac{1}{4} & 0 \\ 0 & 0 & \frac{1}{2} & 0 & 0 & \frac{1}{2} \end{bmatrix} \begin{bmatrix} w_i \\ \theta_{si} \\ \theta_{ni} \\ w_j \\ \theta_{sj} \\ \theta_{nj} \end{bmatrix} \quad (4.57)$$

where $i = 1, 2, 3, 4$, $j = 2, 3, 4, 1$ and $k = 5, 6, 7, 8$.

Two other equations similar to Eq. (4.56) can be expressed for the centerlines of the QUAD9 element as

$$\begin{aligned} \theta_{x9} &= -\frac{3}{2l_{68}} w_8 - \frac{1}{4} \theta_{x8} + \frac{3}{2l_{68}} w_6 - \frac{1}{4} \theta_{x6} \\ \theta_{y9} &= -\frac{3}{2l_{57}} w_7 - \frac{1}{4} \theta_{y7} + \frac{3}{2l_{57}} w_5 - \frac{1}{4} \theta_{y5} \end{aligned} \quad (4.58)$$

The set of six equations stated in (4.57) and (4.58) provides the transformation relationships of the rotational DOF's θ_{xk} and θ_{yk} , ($k = 5, \dots, 9$) of QUAD9 element in terms of the out-of-plane displacements w_i and rotational $w_{,xi} = \theta_{xi}$ and $w_{,yi} = \theta_{yi}$ ($i = 1, \dots, 4$) DOF's at the corners of the IDKQ element.

Equation (4.57) requires the transformation from the side coordinates s, n to the element Cartesian x, y coordinates. In matrix form, these coordinate transformations may be expressed by means of the Jacobian transformation matrices of each element side as follows

$$[J]_{ij} = \begin{bmatrix} x_{,s} & y_{,s} \\ x_{,r} & y_{,r} \end{bmatrix}_{ij} = \begin{bmatrix} \cos \alpha_{ij} & \sin \alpha_{ij} \\ \sin \alpha_{ij} & -\cos \alpha_{ij} \end{bmatrix} \quad (4.59)$$

where $i = 1, 2, 3, 4$; $j = 2, 3, 4, 1$ and $[J]_{ij} = [J]_{ij}^{-1}$.

Thus,

$$\begin{bmatrix} \theta_{sk} \\ \theta_{nk} \end{bmatrix} = [J]_{ij} \begin{bmatrix} \theta_{sk} \\ \theta_{yk} \end{bmatrix} \quad (4.60)$$

and Eq. (4.57) may be expressed in the element Cartesian coordinates as

$$\begin{bmatrix} \theta_{sk} \\ \theta_{yk} \end{bmatrix} = [J]_{ij} \begin{bmatrix} -\frac{3}{2l_{ij}} & -\frac{1}{4} & 0 & \frac{3}{2l_{ij}} & -\frac{1}{4} & 0 \\ 0 & 0 & \frac{1}{2} & 0 & 0 & \frac{1}{2} \end{bmatrix} \begin{bmatrix} w_i \\ [J]_{ij} \theta_{si} \\ [J]_{ij} \theta_{ni} \\ w_j \\ [J]_{ij} \theta_{sj} \\ [J]_{ij} \theta_{ni} \end{bmatrix} \quad (4.61)$$

According to Eq. (4.49), all rotational DOF's in QUAD9 can be explicitly related to the corresponding DOF's in IDKQ by means of the transformation matrix $[T]_{(18 \times 12)}$ given in Appendix II.

The stiffness transformation can be easily derived by substitution of Eqs. (4.49) and (4.50) into the stiffness relationship for element QUAD9

$$[K]^{(QUAD9)} [T] [d]^{(IDKQ)} = [F]^{(QUAD9)} \quad (4.62)$$

and by pre-multiplying Eq. (4.14) with $[T]^T$, the stiffness matrix of the element IDKQ may be written as

$$[K]^{(IDKQ)} = [T]^T [K]^{(QUAD9)} [T] \quad (4.63)$$

As can be observed, Eq. (4.63) does not change the element properties or its orientation in the global coordinate system by any means. Rather, it represents a static condensation of the central and mid-node DOF's of QUAD9 element in terms of DOF's at the corner nodes of the IDKQ element.

Further simplifications, similar to that already shown for the RQUAD element, are possible in order to reduce the amount of calculations required in the numerical integration process by rewriting $[K]^{(IDKQ)}$ as follows

$$[K]^{(IDKQ)} = [T]^T \left(\int [B]_{QUAD9}^T [D][B]_{QUAD9} dV \right) [T]$$

or

$$[K]^{(IDKQ)} = \int [T]^T [B]_{QUAD9}^T [D][B]_{QUAD9} [T] dV = \int [B]_{IDKQ}^T [D][B]_{IDKQ} dV$$

where

$$[B]_{IDKQ}^{(3 \times 12)} = [B]_{QUAD9}^{(3 \times 18)} [T]_{(18 \times 12)} \quad (4.64)$$

in which the matrix multiplication is performed selectively, component-by-component, only for the non-zero components. Note that about 60% of the components in $[T]$ matrix are zero.

Finally, the stiffness matrix of the IDKQ element may be expressed as

$$[K]_{(12 \times 12)}^{(IDKQ)} = \int_{-1}^{+1} \int_{-1}^{+1} [B]_{IDKQ}^T [D]_b [B]_{IDKQ} |J| d\xi d\eta \quad (4.65)$$

in which $[B]_{(IDKQ)}$ is the curvature-transformation matrix of IDKQ element, $[D]_b$ is the bending elasticity matrix and $|J|$ is the determinant of the Jacobian matrix of the coordinate transformation.

As can be observed, $[T]_{(18 \times 12)}$ not being a square matrix, it could not be inverted. For this reason, the DOF's transformation by means of $[T]$ matrix can be done only from QUAD9 to IDKQ.

Crisfield [17] found this element as behaving very well, but no examples were provided in his work. He mentioned also that this quadrilateral based on the Discrete Kirchhoff Assumptions passes the patch test. Crisfield's formulation is similar to that of Batoz *et al.* in [14], formulation which is not able to produce the work equivalent load vector for the DKQ element.

The IDKQ element may have the same shortcoming as Crisfield's formulation of DKQ. The fact that in the DOF's transformation matrix $[T]_{(18 \times 12)}$ is related to the rotational DOF's only makes it difficult to formulate the work equivalent nodal loads corresponding to the distributed load. The cubic out-of-plane displacements w in Eq. (4.55) apply only on the border and center lines of the element. Therefore, it is not simple to formulate the work equivalent load vector for the distributed loads. However, a lumped nodal load vector F_l^{IDKQ} can be employed as an approximation for the normal distributed load $q_z^T = [q_z \quad 0 \quad 0]$. For quadrilateral elements, bilinear shape functions can be used to represent the equivalent lumped nodal loads of the distributed load. For a rectangular element the lumped nodal loads are

$$[F_l^{IDQ}] = q_z \begin{pmatrix} \frac{A_e}{4} & 0 & 0 & \frac{A_e}{4} & 0 & 0 & \frac{A_e}{4} & 0 & 0 & \frac{A_e}{4} & 0 & 0 \end{pmatrix}$$

where A_e is the area of the element. Since w is represented by a cubic interpolation as shown in Eq. (4.55), a cubic polynomial expansion $\bar{N}_c$ may be more consistent with the derivation of the equivalent nodal loads F_c^{IDKQ} from

$$[F_c^{IDKQ}] = \int_{A_e} \bar{N}_c^T q_z dA_e$$

where

$$\bar{N}_c^T = [1 \quad \xi \quad \eta \quad \xi^2 \quad \xi\eta \quad \eta^2 \quad \xi^3 \quad \xi^2\eta \quad \xi\eta^2 \quad \eta^3 \quad \xi^3\eta \quad \xi\eta^3] \{a\}$$

is a complete 2D expansion from the Pascal Triangle completed by the two bi-cubic extra terms $\xi^3\eta$ and $\xi\eta^3$ corresponding to the 12 generalized coordinates $\{a\}$ that must be exchanged with the 12 DOF's $\{d^{IDKQ}\}$. Such exchange corresponds to the derivation of the shape function for w interpolation in the Kirchhoff Plate Bending Theory. For a general quadrilateral shape of the element, numerical integration must be employed to obtain either F_l^{IDKQ} or F_c^{IDKQ} . However, the numerical experiments on IDKQ show that the lumped nodal loads F_l^{IDKQ} is a very good approximation for practical applications.

A special problem that is characteristic of all four node quadrilaterals has to be considered, namely, if large deformations occur, the quadrilaterals get distorted or warped. The main type of distortion refers to the fact that the original four coplanar corner nodes of the quadrilateral no longer remain coplanar (more refined explanations can be found in [18]). With respect to these two remarks, it can be

concluded that the practical use of IDKQ element for large deformations has certain limitations that can not be ignored. More research work is needed in order to make this element more attractive for a larger class of structural problems involving thin plate and shallow shell structures undergoing moderately large deformations. Mesh refinement is able to improve the shape of the distorted elements to a certain level. However, in the areas on the structure where geometric distortions are prominent, high-order shell elements can be automatically employed to replace the existing distorted facet elements. Moreover, two or four compatible triangular finite elements can substitute a distorted quadrilateral based on a measure of the distortion level and re-meshing.

4.5 Facet-Shell Quadrilateral Finite Element QFSE

The large deflection problem of plates and shells requires a special treatment of the unknown nodal displacements in the context of the Finite Element Method [18], [22]. From a general point of view, both the membrane and bending effects contribute at the same time to the carrying capacity of the structure. Therefore, the complete analysis of plate and shell problems must be able to represent both, the in-plane and out-of-plane deformations. At this point, it is necessary to distinguish between deformation in the general sense and displacements and deflections in the context of the membrane and plate theory, respectively. The distinction between the

deformation patterns is necessary due to the fact that the in-plane and the out of plane displacement modes are independent in case of flat finite elements based on the membrane-plate element combination.

Figure 4.9: Typical Nodal DOF of RQUAD and IDKQ Elements

As a starting point in the development of the QFSE facet element, the superposition of the displacement modes of RQUAD membrane and IDKQ plate elements is considered as follows

$$\begin{aligned}
 u &= u_m + u_b \\
 v &= v_m + v_b \\
 w &= w_b
 \end{aligned}
 \tag{4.66}$$

where the subscripts m and b account for membrane and bending components respectively. Because the displacement components are computed from different finite elements, the general representation in Eq.(4.66) is not very useful for the development of the combined finite element. In what will follow, the displacement components and their corresponding strains are computed and kept distinctly.

Consequently, the resulting element stiffness matrix is composed of distinct parts accounting for membrane and bending actions as well as a coupling part accounting for the interaction between them. The coupling stiffness component is generated when the element stiffness matrix is formulated at locations other than the element mid-plane.

According to the large deflection theory, the changes in the element configuration produced by the external actions become significant and the two problems of in-plane and out-of-plane deformations can no longer be treated separately. Usually, the term large deflections refers to displacements w which are affected by the nonlinear part of the Green's strain definition as mentioned in section 3.1. However, there are no fixed limits based on geometric restrictions on displacements or rotations in order to distinguish the applicability domain between small and large deflection theories. It is generally accepted that the small deflection theory applies for out-of-plane deflections that are less than one half of the plate thickness. However, this ratio can be affected by other factors such as boundary conditions and the type of curvature developed in the plate [20], [28].

The physical consideration that differentiates between small and large deformation theories in the context of plates is the stretching of the middle plate surface because of the out-of-plane displacements. This stretching leads to the development of membrane stresses which for small out-of-plane displacements are not accounted for, but in the case of large deflections, the membrane stresses become comparable with the bending stresses. Consequently, the coupling effects between

membrane and bending actions have to be considered in the computational model in order to obtain the desired analysis accuracy.

The displacement components in vector representation are split in separate components of linear and nonlinear membrane and bending types. The element deformations are computed interpolating their nodal values by means of appropriate shape functions as follows

$$\left\{ \begin{array}{c} \left\{ \begin{array}{c} u \\ v \\ \theta_z \end{array} \right\}_m \\ \left\{ \begin{array}{c} w \\ \beta_x \\ \beta_y \end{array} \right\}_b \end{array} \right\} = [N] \{a_i\} \quad (4.67)$$

where

$$[N] = \begin{bmatrix} \underline{N}_i^m & \underline{0} \\ \underline{0} & \underline{N}_i^b \end{bmatrix} \quad \{a_i\} = \begin{Bmatrix} \underline{a}_i^m \\ \underline{a}_i^b \end{Bmatrix} \quad (4.68)$$

with

$$\{a_i^m\} = \begin{Bmatrix} u_i \\ v_i \\ \theta_{zi} \end{Bmatrix}_m \quad \{a_i^b\} = \begin{Bmatrix} w_i \\ \beta_{xi} \\ \beta_{yi} \end{Bmatrix}_b \quad (4.69)$$

and i representing the node number.

According to Eqs. (4.67), (4.68), (4.69), the strain representation can be expressed in a concise manner as follows

$$\begin{aligned} \{\varepsilon\} &= \{\varepsilon_0^m\} + \{\varepsilon_L^m\} + \{\varepsilon_0^b\} + \{\varepsilon_L^b\} = [B_m] \times \{a_i^m\} + [B_b] \times \{a_i^b\} = \\ &= (\underline{B}_0^m + \underline{B}_L^m) \{a_i^m\} + ((-z)\underline{B}_0^b + \underline{B}_L^b) \{a_i^b\} \end{aligned} \quad (4.70)$$

The nonlinear membrane strain components are represented by

$$\{\epsilon_i^m\} = \frac{1}{2} \left\{ \begin{bmatrix} \left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial x} \right)^2 \\ \left(\frac{\partial u}{\partial y} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 \\ \frac{\partial u}{\partial x} \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \frac{\partial v}{\partial y} \end{bmatrix} \right\} = \frac{1}{2} \begin{bmatrix} \frac{\partial u}{\partial x} & 0 & \frac{\partial v}{\partial x} & 0 \\ 0 & \frac{\partial u}{\partial y} & 0 & \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} \end{bmatrix} \times \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial y} \end{bmatrix} = \frac{1}{2} [A_m] \times [\theta_m] \quad (4.71)$$

where

$$[\theta_m] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 & 0 \\ 0 & 0 & \frac{\partial}{\partial x} & 0 \\ 0 & 0 & 0 & \frac{\partial}{\partial y} \end{bmatrix} \times$$

$$\times \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 & N_5 & 0 & N_6 & 0 & N_7 & 0 & N_8 & 0 \\ N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 & N_5 & 0 & N_6 & 0 & N_7 & 0 & N_8 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 & N_5 & 0 & N_6 & 0 & N_7 & 0 & N_8 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 & N_5 & 0 & N_6 & 0 & N_7 & 0 & N_8 \end{bmatrix}_{(4 \times 16)}^{QUAD 8} \times$$

$$\times [T]_{(16 \times 12)} \times \{a_i^m\}_{(12 \times 1)} = [G_m]_{(4 \times 12)} \times \{a_i^m\}_{(12 \times 1)} \quad (4.72)$$

The nonlinear membrane strain-displacement matrix $[B_L^m]$ can be easily found taking a variation of Eq. (4.71) with respect to the nodal displacements $\{a_i^m\}$

$$d\{\epsilon_L^m\} = \frac{1}{2} d([A_m] \times [G_m]) \{a_i^m\} + \frac{1}{2} [A_m] \times [G_m] d\{\{a_i^m\}\} = [A_m] \times [G_m] \times d\{\{a_i^m\}\} = [B_L^m] \times d\{\{a_i^m\}\}$$

or

$$[B_L^m]_{(3 \times 12)} = [A_m]_{(3 \times 4)} \times [G_m]_{(4 \times 12)} \quad (4.73)$$

The nonlinear bending strains are represented by

$$\{\varepsilon_L^b\} = \frac{1}{2} \left\{ \begin{matrix} \left(\frac{\partial w}{\partial x} \right)^2 \\ \left(\frac{\partial w}{\partial y} \right)^2 \\ \frac{\partial w}{\partial x} \frac{\partial w}{\partial y} \end{matrix} \right\} = \frac{1}{2} \begin{bmatrix} \frac{\partial w}{\partial x} & 0 \\ 0 & \frac{\partial w}{\partial y} \\ \frac{\partial w}{\partial y} & \frac{\partial w}{\partial x} \end{bmatrix} \times \begin{bmatrix} \frac{\partial w}{\partial x} \\ \frac{\partial w}{\partial y} \end{bmatrix} = \frac{1}{2} [A_b] \times [\theta_b] \quad (4.74)$$

where

$$\begin{aligned} [\theta_b] &= \begin{bmatrix} \frac{\partial w}{\partial x} \\ \frac{\partial w}{\partial y} \end{bmatrix} = \begin{bmatrix} \sum_{i=1}^9 N_i \theta_i^b \\ \sum_{i=1}^9 N_i \theta_i^b \end{bmatrix} = \\ &= \begin{bmatrix} N_1 & 0 & N_2 & 0 & \dots & \dots & \dots & \dots & N_8 & 0 & N_9 & 0 \\ 0 & N_1 & 0 & N_2 & \dots & \dots & \dots & \dots & 0 & N_8 & 0 & N_9 \end{bmatrix}_{(2 \times 18)}^{QUAD9} \times \{\theta_i^b\}_{(18 \times 1)} = \\ &= [N_b]_{(2 \times 18)} \times \{\theta_i^b\}_{(18 \times 1)} = [N_b]_{(2 \times 18)} \times [T_b]_{(18 \times 12)} \times \{a_i^b\}_{(12 \times 1)} = [G_b]_{(2 \times 12)} \times \{a_i^b\}_{(12 \times 1)} \end{aligned} \quad (4.75)$$

From Eqs. (4.74) and (4.75), the nonlinear curvature-displacement $[\beta_L^b]$ can be easily expressed taking a variation of Eq. (4.74) with respect to the nodal displacement $\{a_i^b\}$

$$d\{\varepsilon_L^b\} = \frac{1}{2} d([A_b] \times [G_b]) \{a_i^b\} + \frac{1}{2} [A_b] \times [G_b] d\{\{a_i^b\}\} = [A_b] \times [G_b] \times d\{\{a_i^b\}\} = [B_L^b] \times d\{\{a_i^b\}\}$$

or

$$[B_L^b]_{(3 \times 12)} = [A_b]_{(3 \times 2)} \times [G_b]_{(2 \times 12)} \quad (4.76)$$

It should be noted that the components of the matrices $[A_m]$ and $[A_b]$ are known from the last converged solution obtained at the end of the previous load increment. The incremental-iterative procedure starts first with a linear analysis with a small enough load increment and after this first step, the nonlinear analysis is performed.

Further, in order to be able to determine the element stiffness matrix, the incremental plane stress constitutive (or material) matrix $[D]_{(3 \times 3)}$, reflecting the incremental stress-strain relationship must be specified. This matrix has to be updated for each load increment according to the tangent modulus E determined from the specific stress-strain material relationship. All aspects related to the material behavior can be accounted for in the finite element analysis by means of an appropriate formulation of the constitutive matrix. The material nonlinear behavior incorporated in the constitutive model is not closely related to the scope of this study.

According to Eq. (4.70), the total strain-displacement matrix $[B]$ can be now expressed in terms of membrane and bending components as follows

$$\{\varepsilon\} = \begin{bmatrix} \underline{B}_m & \underline{B}_b \end{bmatrix} \begin{Bmatrix} \underline{a}_i^m \\ \underline{a}_i^b \end{Bmatrix} = [B] \{a_i\} \quad (4.77)$$

Finally, the evaluation of the tangent stiffness matrix can be easily obtained following Eqs. (2.18), (2.19), (2.20) and (2.21)

$$\begin{aligned}
[K] &= \int_{V_e} (\underline{B})^T \underline{D}(\underline{B}) dV_e = \int_{V_e} \begin{bmatrix} \underline{B}_m^T \\ \underline{B}_b^T \end{bmatrix} \underline{D} \begin{bmatrix} \underline{B}_m & \underline{B}_b \end{bmatrix} dV_e = \\
&= \int_{V_e} \begin{bmatrix} \underline{B}_m^T \underline{D} \underline{B}_m & \underline{B}_m^T \underline{D} \underline{B}_b \\ \underline{B}_b^T \underline{D} \underline{B}_m & \underline{B}_b^T \underline{D} \underline{B}_b \end{bmatrix} dV_e = \int_{V_e} \begin{bmatrix} (\underline{B}_0^m)^T + (\underline{B}_L^m)^T & (\underline{B}_0^m)^T + (\underline{B}_L^m)^T \\ (-z)(\underline{B}_0^b)^T + (\underline{B}_L^b)^T & (-z)(\underline{B}_0^b)^T + (\underline{B}_L^b)^T \end{bmatrix} \underline{D} \begin{bmatrix} \underline{B}_0^m + \underline{B}_L^m & \underline{B}_0^b + \underline{B}_L^b \\ (-z)\underline{B}_0^b + \underline{B}_L^b & (-z)\underline{B}_0^b + \underline{B}_L^b \end{bmatrix} dV_e = \\
&= [K_0] + [K_L]
\end{aligned} \tag{4.78}$$

where

$$[K_0] = \begin{bmatrix} \underline{K}_0^{mm} & \underline{K}_0^{mb} \\ \underline{K}_0^{bm} & \underline{K}_0^{bb} \end{bmatrix} = \int_{V_e} \begin{bmatrix} (\underline{B}_0^m)^T \underline{C} \underline{B}_0^m & -z (\underline{B}_0^m)^T \underline{C} \underline{B}_0^b \\ -z (\underline{B}_0^b)^T \underline{C} \underline{B}_0^m & z^2 (\underline{B}_0^b)^T \underline{C} \underline{B}_0^b \end{bmatrix} dV_e. \tag{4.79}$$

$$[K_L] = \begin{bmatrix} \underline{K}_L^{mm} & \underline{K}_L^{mb} \\ \underline{K}_L^{bm} & \underline{K}_L^{bb} \end{bmatrix} = \int_{V_e} \begin{bmatrix} (\underline{B}_L^m)^T \underline{D} \underline{B}_L^m + (\underline{B}_L^m)^T \underline{D} (\underline{B}_0^m + \underline{B}_L^m) & (\underline{B}_L^m)^T \underline{D} \underline{B}_L^b + (\underline{B}_L^m)^T \underline{D} (-z) \underline{B}_0^b + \underline{B}_L^b \\ (-z)(\underline{B}_L^b)^T \underline{D} \underline{B}_L^m + (\underline{B}_L^b)^T \underline{D} (\underline{B}_0^m + \underline{B}_L^m) & (-z)(\underline{B}_L^b)^T \underline{D} \underline{B}_L^b + (\underline{B}_L^b)^T \underline{D} (-z) \underline{B}_0^b + \underline{B}_L^b \end{bmatrix} dV_e \tag{4.80}$$

As can be observed, the linear membrane and bending stiffness components in Eq. (4.79) are coupled in this formulation. The coupling terms κ_0^{mb} and κ_0^{bm} are generated by the fact that in Eq. (4.71) the expression of strains is considered for a plane other than the neutral surface of the element. For an element stiffness formulation referred only to the deformations of the element neutral surface for which $z=0$, the linear membrane and bending stiffness components become uncoupled as in [19]. The integration process in the thickness direction is performed layer-wise by superposition of the layer contributions. The element stiffness matrix is then obtained by adding the stiffness contributions of the element layers. For this reason, the usual 2D finite element formulation based only on the deformation of the element neutral

surface is not acceptable in idealizing correctly the state of deformation over the element thickness when composite materials are involved. The large displacement stiffness matrix $[K_L]$ obtained in Eq. (4.80) is generated by the nonlinear strain-displacement matrices inserted in the general strain formulation according to Eq. (4.71). It should be noted that significant geometric changes occur also in the plastic range of the materials. The relationship between the geometric nonlinear effects and the material behavior is much more difficult to analyze. The reason is that large deformations might cause nonlinear material behavior, but the plastic flow of the material generates large deformations as well. Consequently, for a complete and an accurate structural analysis, the problem of material nonlinear behavior should be treated in the context of nonlinear geometric behavior and conversely.

The geometric (or initial stress) stiffness contribution K_σ to the tangent stiffness matrix is obtained according to Eq. (2.20). Knowing that G_m and G_b in Eqs. (4.73) and (4.76) are constant expressions, they being dependent only on the nodal coordinates, then

$$\begin{aligned} d(\underline{B}_L^m)^T &= [G_m]^T d([A_m])^T \\ d(\underline{B}_L^b)^T &= [G_m]^T d([A_m])^T \end{aligned} \tag{4.81}$$

According to the special property of the A_m and A_b matrices $d(A)\theta = Ad(\theta)$, the matrix products which give the geometric stiffness matrix become

$$d([A_m])^T \times \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} \sigma_x & \sigma_{xy} & 0 \\ \sigma_{yx} & \sigma_y & 0 \\ 0 & \sigma_x & \sigma_{xy} \\ 0 & \sigma_{yx} & \sigma_y \end{bmatrix} \times d(\theta_m) = [G_m] d(\{a_i^m\}) = [\sigma]_{(4 \times 4)} [G_m]_{(4 \times 12)} d(\{a_i^m\}_{(12 \times 1)}) \quad (4.82)$$

$$d([A_b])^T \times \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{Bmatrix} = \begin{bmatrix} \sigma_x & \sigma_{xy} \\ \sigma_{yx} & \sigma_y \end{bmatrix} \times d(\theta_b) = [G_b] d(\{a_i^b\}) = [\sigma]_{(2 \times 2)} [G_b]_{(2 \times 12)} d(\{a_i^b\}_{(12 \times 1)})$$

Finally, the element geometric stiffness matrix is expressed as

$$[K_\sigma] = \begin{bmatrix} \underline{K}_\sigma^m & 0 \\ 0 & \underline{K}_\sigma^b \end{bmatrix} = \int_{V_e} \begin{bmatrix} \underline{G}_m^T [\sigma]_{(4 \times 4)} \underline{G}_m & 0 \\ 0 & \underline{G}_b^T [\sigma]_{(2 \times 2)} \underline{G}_b \end{bmatrix} dV_e \quad (4.83)$$

The tangent stiffness matrix can be easily obtained according to Eq. (2.21) by matrix summation of the elastic, large displacement and geometric stiffness matrices

$$K_T = K_0 + K_L + K_\sigma$$

Because $\underline{D}$ is a symmetric matrix, from the expressions of the stiffness matrices in Eqs. (4.79) and (4.80), the elastic stiffness matrix K_0 and the large displacement matrix K_L are also symmetric, they being composed of symmetric matrix products disposed symmetrically in the element stiffness matrix. The geometric stiffness matrix in Eq. (4.83) is also symmetric because of the symmetry of the stress matrix of the in-plane action $[\sigma]$ and symmetric matrix products. For instance, the tangent stiffness matrix K_T is symmetric. Note that the symmetry of the structural stiffness matrix is an important requirement in structural engineering according to the reciprocity of the mechanical work stated by Betti's Law. For the

program implementation, this property offers a significant advantage, the memory space required to store the structural stiffness matrix being reduced to a half. Also, very efficient solution strategies can be developed in solving large systems of equations by reducing symmetric non-singular matrices [16], [20]. The integration of the element stiffness matrices is carried out numerically using the Gauss quadrature rule. In Eqs. (4.79), (4.80) and (4.83), the layer thickness coordinate z_l ($z_l \neq z$) is part of the integration process being associated with the material matrix of its layer. It should be noted that z represents the distance of the middle plane of the layer to the mid-surface of the element. The integration over the element volume v_e is performed in two stages. In the first stage, because the strain-displacement matrices are not dependent on the element thickness, the integration in the thickness direction is performed analytically by adding together the layer contributions. It must be mentioned that the actual in-plane stresses are not constant over the element thickness. The stresses stored in the matrix $[\sigma]$ of Eq. (4.83) correspond to the actual stresses that satisfy the yielding conditions and are constant over the thickness of each layer. On the other hand, the matrices G_m and G_b being dependent only by the nodal coordinates, they are not varying over the element thickness. The geometric stiffness matrix becomes

$$[K_\sigma] = \begin{bmatrix} \underline{K}_\sigma^m & \underline{0} \\ \underline{0} & \underline{K}_\sigma^b \end{bmatrix} = \int_{A_e} \begin{bmatrix} \underline{G}_m^T \left(\sum_{l=1}^n (z_{l+1} - z_l) [\sigma_l]_{(4 \times 4)} \right) \underline{G}_m & \underline{0} \\ \underline{0} & \underline{G}_b^T \left(\sum_{l=1}^n (z_{l+1} - z_l) [\sigma_l]_{(2 \times 2)} \right) \underline{G}_b \end{bmatrix} dA_e \quad (4.84)$$

The stiffness expressions in Eqs. (4.78), (4.79) and (4.84) are evaluated in the local Cartesian coordinate system of the element, the strains and the stresses having a physical meaning in the element local coordinate system only. The element tangent stiffness matrix is constructed by adding together the elastic, large-displacement and geometric element stiffness matrices. Finally, in order to be able to assemble the tangent stiffness matrix, the element stiffness matrices must be first transformed from local to the global Cartesian coordinate system. This transformation is carried out by means of a coordinates transformation matrix which can be found in standard engineering textbooks.

The QFSE element described in this section has the advantage of having six DOF's per node, advantage that makes it powerful when used in arbitrary finite element assemblages. Although, the QFSE element behaves well as demonstrated by the numerical tests shown in the sample examples (see the Folded Plate example later), the problem of the sixth degree of freedom is still open. The present trend is to include the in-plane rotations as effective DOF's coupled to the in-plane DOF's u_i, v_i , [55], [56], [57]. The introduction of the drilling DOF's is justified by the fact that the classical approach of substituting small fictitious stiffness quantities for the in-plane rotational DOF's leads sometimes to wrong results [18], [19]. In the present element development, the coupling of the drilling DOF's to the in-plane displacement DOF's is performed according to Eq. (4.41). For coplanar finite element assemblages, it is easy to observe that the displacement fields are continuous from one element to another. It should be pointed out again that the drilling DOF's do not have the physical meaning of nodal rotations. To find the meaning of the ω_i drilling DOF's in

a three-dimensional structure, the case of two non-coplanar finite elements intersecting at 90° is considered.

Figure 4.10
Incompatible Connection of two Perpendicular Facet Elements

From Eqs. (4.36), (4.39) and (4.55), it can be concluded that the displacement in the two elements in Figure 4.9 are equal at A and B but not along the edge AB, irrespective of the drilling DOF's.

As mentioned in [48], the facet element QFSE poses a spurious mode provided by the ω_i DOF's

$$\begin{aligned} u_i &= v_i = 0 \\ \omega_i &= C \neq 0 \end{aligned} \tag{4.85}$$

Usually, this spurious mode is locked. In a three-dimensional structure, it is always locked. For a planar structure, this spurious mode can be locked by fixing one of the ω_i DOF's to zero. Consequently, the introduction of the drilling DOF's ω_i can be considered as a stabilization technique capable of solving the problem of the sixth

DOF, a problem characteristic of the facet-shell finite elements. However, the drilling DOF's improve the in-plane behavior of the membrane element, the performance of the RQUAD element being similar to that of QUAD8 membrane element, but to a significantly lower computational effort compared to QUAD8.

5 General Procedure for Nonlinear Analysis

5.1 General

In this chapter, the full Newton-Raphson solution strategy used in solving the nonlinear problems in this work is presented. In the first section, a brief description of the most used solution strategies currently employed in solving nonlinear structural problems is provided. The full Newton-Raphson algorithm is described in the next section. In the last section, different convergence criteria are presented and analyzed from the point of view of their practical importance.

5.2 Numerical Solutions to the Nonlinear Problem

The solution techniques of nonlinear problems are of iterative nature which consist of a series of approximate solutions of the incremental equilibrium equations expressed as

$$[K_{i-1}^n] \{\Delta r_i^n\} = \{F_i^n\} \quad (5.1)$$

The most important nonlinear effects that govern the structural response are caused by the nonlinear material response and geometric nonlinear effects. Other kinds of nonlinear effects, which may influence the structural response, can be expressed in terms of material and geometric nonlinear effects only and easily included in the analysis. A convenient classification of the nonlinear structural behavior is given in [20] in which the material and kinematic nonlinear effects are treated separately.

For a material nonlinear analysis only, the nonlinear effect is due to the nonlinear stress-strain relationship of the material. The displacement and strain fields are assumed small enough, so that the engineering strain and stress measures may be employed to predict the structural response. This kind of analysis is restricted only to materials that behave nonlinear under small deformations. Plain concrete is such a material. On the other hand, when large deformations take place, the small strain formulation is no longer adequate to predict accurately the structural response, the strain-displacement relationship being also another source of nonlinear behavior. In these circumstances, the structural analysis becomes more complex and computationally very intensive depending on the particular characteristics of the problem. Consequently, the classification of the problem and the choice of the analysis type are important parts of the design process. In the practical design activity it is indicated that a restrictive analysis may be performed, getting first some insight into the response prediction and after this, establishing the most efficient analysis strategy as suggested in [20]. The use of the most general nonlinear analysis formulation will lead always to highly accurate results, but at the same time the

computational cost may be prohibitive. However, the analysis strategy belongs to the modeling process and depends on the particular characteristics of the structural system and user experience.

The nonlinear structural response is computationally obtained using a step-by-step incremental/iterative procedure which reduces to one-step analysis in the case of a linear and time independent analysis. The nonlinear solution is obtained automatically through a number of load steps until the maximum applied load is reached. At each load increment, the analysis is performed based on a predictor-corrector technique. The predictor solution is given by a linear solution with the corresponding tangential stiffness of the load increment. Since the equilibrium equations are not exactly satisfied by the predictor solution, the applied loading being yet not balanced by the computed element stresses, a corrector procedure based on successive iterations is employed to reduce the out-of-balance forces to acceptably small levels.

The difference between the force vector of the external loads $\{F\}$ and the internal force vector $\{Q\}$ provides a system of out-of-balance forces denoted as residual forces. Through successive adjustments, the out-of-balance load vector is reduced in magnitude until it loses its structural significance. For each load increment, the series of iterations are performed according to

$$\{g\}_i^n = \{F\}^n - \{Q\}_i^n = \{F\}^n - \int_V [B]^T \{\sigma\}_i^n dV \neq 0 \quad (5.2)$$

until the convergence criteria expressed in terms of a ε tolerance limit is satisfied. For the iteration i of the load increment n , $\{g\}_i^n$ is the out-of-balance load vector, $\{F\}^n$ is the external load vector and $\{Q\}_i^n$ refers to the internal force vector. The stress vector $\{\sigma\}_i^n$ represents the actual stresses satisfying the appropriate constitutive law.

The vector of the total displacements is updated in each iteration according to

$$\{r\}_i^n = \{r\}_{i-1}^n + \{\Delta r\}_i^n \quad (5.3)$$

where $\{\Delta r\}_i^n$ is the displacement vector obtained in the last iteration i and produced by the out-of-balance load vector obtained in the previous iteration $i-1$. Thus,

$$\{\Delta r\}_i^n = -([K_T]_{i-1}^n)^{-1} \{g\}_{i-1}^n \quad (5.4)$$

where $[K_T]_{i-1}^n$ is the tangent stiffness, which corresponds to the geometric and material conditions, established in the iteration $i-1$. It must be noted that in the layered model, the layers stress must be checked at each iteration. For this reason, the element tangent stiffness matrix must be also updated according to the physical state of each layer of the finite element. In the traditional incremental-iterative procedures, the tangent stiffness is kept unchanged for all iterations in order to reduce the computational cost. Further, with the updated displacements $\{r\}_i^n$, the actual strains and stresses can be computed and finally the new out-of-balance load vector $\{g\}_{i+1}^n$ is established according to Eq. (5.2).

In the predictor-corrector procedures the correct evaluation of the residual forces $\{Q\}_i^n$ based on $\{r\}_i^n$ is of crucial importance, any error in this computation

resulting in an inaccurate response prediction. In certain situations, neglecting the stiffness terms relating the large deformation or initial stresses, the displacements $\{r\}_i^n$ are directly affected and consequently, the final results as well.

The use of iterative solutions requires appropriate convergence criteria, which more or less are able to determine the quality of the results. The iterations are stopped when the norm of the out-of-balance load vector becomes smaller than a fixed limit ε , called the tolerance. It should be noted that it is not necessary to satisfy the incremental equilibrium equations exactly. To obtain the desired accuracy, realistic convergence criteria must be used in order to terminate the iterations and to advance the solution to the next load increment.

Generally, three incremental/iterative solution strategies can be distinguished in predicting the nonlinear structural response. These methods are classified as follows

- 1) **Iterative Methods.** These methods are also known also as Newton-Raphson Iterations. The full load is applied in one step. The out-of-balance forces are applied iteratively until equilibrium is achieved. Depending on how the stiffness matrix is selected, three further classifications of the iterative methods are possible
  - a) **Constant or Initial Stiffness.** In this method, the initial stiffness matrix is used to solve for the displacement increment $\{\Delta r\}_i$. The initial stiffness matrix is computed once before the first iteration and it

is used unchanged for all iterations. From a computational point of view, this strategy seems to be very advantageous. The stiffness matrix is assembled and inverted only once, but using a constant stiffness matrix, a large number of iterations is required to obtain convergence, Figure 5.1(a).

b) **Tangent Stiffness.** For each iteration, a new stiffness matrix is formed and used to solve for the next iterative displacement vector. In this case the convergence is reached rapidly, but with a significant computational effort required by the complete solution of the problem in every iteration Figure 5.1(b)

c) **Secant Stiffness.** The high computational effort required by the use of tangent stiffness can be reduced using the secant stiffness established according to the secant to the load-deflection curve as shown in Figure 5.1(c). The steps in the secant directions are not quite as advantageous as the steps in the tangent directions, but the secant stiffness steps are much cheaper and stable compared to the tangent stiffness steps, [7].

2) **Incremental Methods.** In these methods, the full load vector $\{F\}$ is divided into a number of load increments $\{\Delta F\}$, usually equal in magnitude. At the beginning of each load increment, a tangent stiffness matrix is formed and kept unchanged for all iterations in the subsequent increment. If no attempt is

made to satisfy the equilibrium equations accurately, the predicted structural response of the incremental procedure could have significant deviations from the correct response as can be observed in Figure 5.2(a). In order to improve the method, an equilibrium correction procedure is employed so that the out-of-balance loads are added to the next load increment. The obtained results are a better approximation of the true or exact load-deflection path as shown in Figure 5.2(b)

- 3) **Combined Methods.** In the practice, the methods mentioned in (1) and (2) are combined in a variety of ways trying to find an accurate and computationally efficient solution scheme. Usually, the total load vector $\{F\}$ is divided into a number of load increments according to the Incremental Method and tangent stiffness iterations according to the Iterative Method are performed in each load step as shown in Figure 5.3. In the commercial finite element codes, the user is asked to select the best suitable solution strategy for the problem at hand indicating the number of load steps, the maximal number of iterations and in which conditions the tangent stiffness matrix should be updated.

Figure 5.1 : Different Incremental Methods for Nonlinear Analysis

Figure 5.2 : Incremental Analysis with and without Equilibrium Check

Figure 5.3: Incremental Analysis Using Initial or Tangent Stiffness Approach

5.3 The Full Newton-Raphson Algorithm

Two principal version of Newton-Raphson (N-R) procedures are currently employed, depending on the selection of the initial stiffness scheme. The so-called Full Newton-Raphson (fN-R) procedure employs the assembly of the tangent stiffness and the inversion of the equilibrium equations in each iteration. This procedure is computationally intensive, but for hard nonlinear problems is usually preferred. A variant of the fN-R, is offered by the so-called Modified Newton-Raphson (mN-R) procedure based on the initial stiffness scheme. In mN-R procedure, the tangent stiffness obtained at the last converged solution is kept constant for all iterations in the subsequent load step. This procedure reduces drastically the computational cost per iteration, but it requires more iterations for convergence than fN-R. In practice, a

combination of the above-mentioned iteration schemes is used by updating the tangent stiffness of the system at selected iterative intervals only.

Knowing the displacements r^{n-1} , the stresses σ^{n-1} at the last converged solution, the principal steps of the fN-R algorithm are summarized as follows

- 1) Evaluation of the residual forces corresponding to the predictor solution

$$g_0^n = f^n + Q_0^n$$

- 2) Updating the tangent stiffness matrix $[K_T]$ depending on the iterative scheme employed.

- 3) Computing the incremental displacements Δr_i (at iteration i) by solving the equilibrium equations

$$\Delta r_i = -[K_T]^{-1} g_{i-1}$$

- 4) Updating the total displacement vector r_i

$$r_i = r_{i-1} + \Delta r_i$$

- 5) Computing the incremental strains $\Delta \varepsilon_i$ and the total strains ε_i

$$\Delta \varepsilon_i = B \Delta r_i \quad \varepsilon_i = B' r_i$$

where

$$B = B_0 + B_L \quad \text{and} \quad B' = B_0 + \frac{1}{2} B_L$$

Note that the strain-displacement matrices B and B' are different, and a distinction must be made between the total strains ε and the finite strains $\Delta\varepsilon$.

- 6) Evaluation of the incremental stresses and the total stresses

$$\begin{aligned} \Delta\sigma_i &= D_T \Delta\varepsilon_i \\ \sigma_i &= \sigma_{i-1} + \Delta\sigma_i \end{aligned}$$

Note that the stress σ_{i-1} satisfies the yielding conditions and the matrix D_T is the tangent constitutive matrix corresponding to the strain-stress level obtained at the last converged solution.

- 7) Evaluate of the actual stress σ_{actual} that satisfies the yielding conditions (the plasticity criterion)
- 8) Computation of the equivalent internal nodal forces corresponding to iteration i

$$Q_i = \int_V B^T \sigma_{actual} dV$$

- 9) Computation of the out-of-balance vector g_i resulting from iteration i

$$g_i = F - Q_i$$

where F represents the total loads vector.

10) Checking the convergence criterion.

In case of convergence, proceed to the next load increment starting the fN-R algorithm from step 1. If the convergence criterion is not satisfied, proceed to the next iteration starting from step 2 of the fN-R algorithm. The choice of the convergence criterion is important and its satisfaction does not always guarantee that the structural equilibrium is satisfied to a certain level of accuracy.

5.4 Convergence Criteria

The efficiency and reliability of the incremental/iterative procedures are dependent on the convergence criteria used in terminating the iteration process when the desired degree of accuracy is achieved. At the end of each iteration, accuracy of the solution is checked and if it falls in the specified tolerance limits, the iterations are stopped and the solution procedure is continued with the next load increment. The tolerance limits that are usually set by the user play an important role in achieving the desired accuracy. In the case of a too large convergence tolerance, the iterations are terminated prematurely leading to inaccurate solution. When the convergence tolerance is too severe, the increase of the computational effort may not be justified by the gains in the accuracy of the solution. Similarly, an ineffective divergence check may stop the iterations prematurely indicating divergence when in fact the

solution is slowly converging. In conclusion, the choice of the convergence criteria may influence the quality of the results significantly. For practical purposes, the user should check the structural response using different convergence criteria for the same problem in order to get more confidence in the solution accuracy.

Generally, three different procedures have been advocated as convergence criteria in terminating the iterative process. These procedures are presented below

5.4.1 Displacement Criterion:

$$\|\delta_i\|_E = (\delta_i^T \delta_i)^{1/2} < \varepsilon_D \|\Delta r_i\|_E \quad (5.5)$$

where $\|\cdot\|_E$ represents the Euclidean norm, $\delta_i = -K_T^{-1} g_i$ denotes the iterative displacement-change vector and Δr_i is the vector of the total incremental nodal displacements. Convergence is achieved when the iterative displacements become smaller than a fraction ε_D of the incremental displacements expressed in Euclidean vector norms. This convergence criterion does not reflect directly the structural equilibrium-state stated in Eq. (5.1). For this reason, it must be used with care.

The Displacement Criterion as expressed in Eq. (5.5) may be not effective in situations in which the iterative displacements δ_i have small variations but continue to change for a large number of iterations. Particularly, if line searches are adopted, for very small load steps, very low changes in the iterative displacements are produced, displacements which might satisfy the convergence criterion from the

numerical point of view only. This situation is characteristic to hypoelastic and elasto-plastic analysis problems in which high stresses are achieved at very small strain levels. However, a very small load step obtained through line searches adjustment indicates that the solution tends to achieve its resistance limit where convergence becomes difficult to achieve.

5.4.2 Force Criterion:

$$\|g_i\|_E < \varepsilon_F \text{MAX}(\|F_i\|_E, \|R_i\|_E) \quad (5.6)$$

where R_i and F_i are the vector of the reaction forces and the load vector at iteration i , respectively. The introduction of the reaction forces in Eq (5.6) allows the Force Criterion to be applied to problems involving prescribed displacements. The convergence is achieved when the norm of the out-of-balance forces g_i obtained in iteration i become less than a fraction ε_F of the norm of the applied external load F .

The Force Criterion, in contrast to the Displacement Criterion, relates directly to the equilibrium equations in Eq. (5.1). This convergence criterion has the advantage of reflecting the equilibrium-state accurately and is not sensitive to the magnitude of the displacements. For practical purposes, the out-of-balance forces representing a fraction of 1% ($\varepsilon_F = 0.01$) of its applied loads are more than sufficient in satisfying the equilibrium conditions with a high degree of accuracy. However,

special attention must be paid to the inconsistency of the measurement units for forces and moments in the case of the facet element QFSE. Difficulties appear when the out-of-balance forces become very small but the changes in displacements do not diminish. This case corresponds to an elasto-plastic analysis when entering the plastic path under a small hardening modulus.

5.4.3 Energy Criterion:

$$\delta_i^T g_i < \varepsilon_E (\Delta r_{(i)}^n)^T \Delta F^n \quad (5.7)$$

where $\Delta r_{(i)}^n$ is the displacement vector corresponding to the predictor solution (first iteration) of the current increment n ; ΔF^n represents the incremental load vector.

This criterion, apart from the square root, may be seen as combination of the force and displacement criteria since they have the same dimensions. Convergence is assumed to occur when the amount of work done by the out-of-balance forces g_i undergoing their corresponding iterative displacements δ_i is less than a fraction ε_E of the initial internal energy increment developed in the predictor solution. From a practical point of view, this criterion may overcome some of the weaknesses of the above two criteria. However, it has the same disadvantage of the displacement criterion when the iterative displacements are very small and the out-of-balance forces are significantly high.

For practical purposes, the best way in overcoming the possible convergence difficulties that might appear is to employ both the Displacement and the Force Criteria independently, considering the iterative process to have converged when both criteria are satisfied concurrently. This strategy will cost some more iterations, but the obtained solution is surely accurate.

5.4.5 Alternative Force Criterion:

Obvious alternatives to the above mentioned convergence criteria are still possible. From a practical point of view, the choice of the best suitable convergence criterion for a particular analysis problem is not simple. First, it is recommended to perform some preliminary analysis employing different convergence criteria and comparing the solutions, then it can be decided which criterion is most suitable for a particular.

The facet element QFSE, when has both displacement and rotation DOF's, requires that all parameters in the above enumerated convergence criteria have the same dimension. In the case of the thin shells, this aspect should not be ignored knowing that the rotations are dominant. If the force criterion is used, the following scaling technique introduced by Peano and Riccioni found in [17] seems to be optimal.

$$\frac{\|g\|}{\max(\lambda\|q\|, \|r\|)} \leq \varepsilon \quad (5.8)$$

where $\bar{g}$, $\bar{q}$ and $\bar{r}$ are the Euclidean vector norms corresponding to the scaled residual forces, total applied load and reaction vectors. The scaling as introduced by Peano and Riccioni is given by

$$\|\bar{g}\| = \sqrt{g^T D^{-1} g} \quad , \quad etc \quad (5.9)$$

in which D^{-1} is the inverse of the diagonal matrix containing the leading diagonal elements of the tangent stiffness matrix obtained at the beginning of the load increment. However, this criterion is computationally expensive and based on the own experience with Nonlacs, it can be successfully replaced by the Energy Criterion.

6 Arc-Length Control in Following the Equilibrium Path

6.1 General

In this chapter, the main idea of the Crisfield's Arc-Length Procedure [16] is presented. In the first section, general notions related to the description of the equilibrium path, critical equilibrium states and the phenomenon of snap-through are explained. In the second section, the concept of the arc-length control technique in following the equilibrium path beyond the load limit points is described. Automatic arc-length control is introduced together with an increment cutting strategy in case of lack of convergence within the desired number of iterations. Reference is made to the Bergan's Current Stiffness Parameter in passing the bifurcation points.

6.2 Equilibrium Path and Critical Equilibrium States

In both physical and mathematical applications, the phenomena of interest are usually represented by means of real-valued smooth functions. The numerical analysis based on the mathematical descriptions of the physical phenomena exhibit

special characteristics at certain points along the solution path at which the function's derivatives vanish. Such locations are denoted as critical points. The commonest type of critical points of a smooth function is represented by its local maximum and minimum where the function's derivatives change signs. These critical points are denoted as limit points. In structural engineering, the limit points indicate a limit state of equilibrium. Another category of critical equilibrium points is represented by the so-called bifurcation points. At the bifurcation load, one or more equilibrium configurations are possible, making the numerical analysis very difficult at this equilibrium point. However, due to the existence of constructive or material imperfections in the real structures, the bifurcation points will not occur in the structural response of a real structure.

For engineering purposes, the complete structural response can be described by load-displacement curves corresponding to specific degrees of freedom of the analyzed structure. Such a curve is referred to as equilibrium path or deformation path. An example of equilibrium path is shown in Figure 6.1. According to the type of relationship between the different load levels and their corresponding deformations, two principal types of structural response can be distinguished. One type represents the proportional load-deformation branch on the equilibrium path, the other type representing the non-proportional load-deformation branches. In non-linear analysis, equilibrium path is obtained by loading progressively the considered structure over its limit of proportional behavior until collapse occurs. The nonlinear response of structures is provided by significant changes of the two major factors that affect the structural behavior: material properties and changes in the structure's configuration

due to large displacements and deformations. Significant changes of the material properties occur when the elastic limit of the loaded material is exceeded in accordance with the material stress-strain relationship. Depending on the particular characteristics of the analyzed structure, changes in the structure's geometry can affect significantly the equilibrium state by redistribution of the loads, these changes being associated more or less with the changes of the material properties.

Generally, the structural problems represented as $[K]\{\delta\} = \{F\}$, in which both $[K]$ and $\{F\}$ are not affected by the magnitude of the deformation $\{\delta\}$, are denoted as linear. In the case of problems in which $[K]$ or $\{F\}$ become dependent on the deformation level $\{\delta\}$, the problem is termed nonlinear. From a theoretical point of view, the classification of the analysis as linear or nonlinear becomes meaningless, since a theoretically correct computational model should handle all aspects related to the physical and geometric behavior during the analysis. The prediction of the post-critical response (e.g. Figure. 6.1) is useful for studies on imperfection sensitivity and collapse modes. In certain circumstances and for limited analysis scope, satisfactory solutions of many structural problems could be obtained by linear equations only, ignoring all aspects related to the nonlinear structural behavior [7]. However, the linear solutions can constitute only the starting point for a more advanced analysis of the same problem.

Structural collapse, contact and impact problems require structural analyses invoking all aspects related to large deformations, nonlinear constitutive laws and nonlinear boundary conditions. All these problems introduce various complexities in the finite element formulation and solution process. One particular complexity is

introduced by the singularity of the stiffness matrix at the critical equilibrium points. Other complexities are related to the fact that the problem solution becomes non-unique on the post-critical path in the sense that to a certain load level may correspond different displacement solutions as shown in Figure. 6.1.

Figure 6.1: Equilibrium Path for the Snap-Through Problem

The critical equilibrium points in Figure. 6.1 refer to the equilibrium limit states for which maximum or minimum loads or displacements are achieved. In such circumstances, the numerical analysis becomes especially difficult requiring adequate solution strategies based on load and/or displacement control. The detection of the critical equilibrium points is closely related to the concept of structural stability. The locations on the equilibrium path where changes of its convexity take place under

loads that are not limit loads are denoted as bifurcation points and they are closely connected with the physical concept of local buckling phenomenon. At the limit points, the structural analysis becomes especially difficult because of the singularity of the structural stiffness matrix. It should be noted that a mathematically exact solution at these limit points does not exist, but from a practical point of view, an exact solution is not needed at all. An approximate solution in the vicinity of the limit points is sufficient in order to trace with enough accuracy the equilibrium path and to start an adequate numerical procedure to overcome the limit points. In this respect, solution methods for structural analysis should include the following capabilities:

- 1) Detection of critical equilibrium points on the load-displacement path.
- 2) Tracing the pre- and post-critical equilibrium paths including both softening and stiffening behavior.
- 3) Handling of snaps on the equilibrium path.

The most efficient computational methods capable to satisfy these requirements are of incremental/iterative nature known as Newton-Raphson methods. Their solution strategy is based on a progressive loading by means of an incremental load technique and iterative solution corrections in establishing the structural equilibrium at each corresponding load level. In this context, the modified Newton-Raphson scheme (m-NR) is largely employed in practical applications based on accumulated experience showing that it can predict accurate solutions [16], [17], [19], [21], [24], [33], [34]. If the iterations are performed within the sphere of convergence,

the full Newton-Raphson (f-NR) procedure poses the property of quadratic convergence [16], [24], [25].

Figure 6.2: Modified Newton-Raphson (m-NR) Scheme

According to the standard N-R procedure, the incremental/iterative process can be continued until the load level required in analysis is achieved or until divergence occurs. It must be noted that the occurrence of divergence does not indicate always that the structural collapse load level was achieved. There are other reasons associated with the divergence of the numerical procedure than structural failure, e.g. the selected FEM model, numerical integration scheme and the occurrence of spurious zero-energy modes, [19]. Particularly, local buckling phenomena or an insignificant local failure in the analyzed structure can lead to severe numerical problems or divergence of the iterative procedure. For this reason, the failure of the numerical analysis alone can not be used to assess the performance of the whole structure. Improvements to the existing numerical techniques are required in order to be able to perform a complete and reliable structural analysis.

A particular situation leading to divergence of the numerical procedure occurs when applying a new load increment and the total load level oversteps the load

capacity of the structure as shown in Figure 6.3. Such a situation can not be easily anticipated because the maximal load that can be carried by the analyzed structure is not known in advance.

Figure 6.3: Increment Cutting Process

An indication of divergence can be considered the increasing tendency of the out-of-balance forces $g_i^{(n)}$ in consecutive iterations. Based on this concept, a significant computational time can be saved by restarting the load incrementation from the last converged solution with a smaller load increment. Such a technique is usually denoted as increment cutting. In [16], this strategy is invoked when the convergence of the structural equilibrium is not achieved within a specified number of iterations. Usually, the number of iterations per load increment is specified by the user, based on experience and the particular characteristics of the problem. It can be easily observed that such a criterion is not always a reliable one. Sometimes, the convergence of the structural equilibrium equations can be achieved in fewer iterations than the maximal number specified by the user, but in fact the apparent convergence may have leaped over a snap-through A-B-C as shown in Figure. 6.4.

Figure 6.4: Overstepping of a Snap-Through

Increment cutting can be computationally inefficient, recognizing that at each iteration the complete solution of the equilibrium equation system is required. In order to obtain a better solution strategy, the size of the load increments can be adjusted based on the iteration history. For increments requiring a larger number of iterations, the size of the next load increment is reduced proportionally. For this reason, usually, line searches are introduced at a very low computational cost by means of a η load parameter as shown in [16]. It should be noted that line searches are load controls which are introduced to keep the numerical procedure in a feasible direction.

The most difficult aspect in tracing the equilibrium path is related to the special numerical treatment that must be applied beyond the first limit point. The equilibrium path until the first limit point is uniquely determined, which means that to a certain load corresponds only one displacement solution. Beyond the critical equilibrium point, the analysis becomes more complex because the solution is not unique, in the sense that to a load level corresponds more than one possible displacement solution and vice versa. In such circumstances, forms of load or displacement control solution strategies expressed in terms of fixed increments become ineffective because of the impossibility of distinguishing among the possible

multiple solutions and their order of occurrence. This aspect is suggestively illustrated in Figure 6.1 by the LL* and DD* branches. The switch to some forms of displacement control may be useful in overstepping limit or bifurcation points.

The original forms of displacement control were introduced simulating a physical testing procedure as described in [37], but its major disadvantage relates to the selection of the appropriate displacement variables that are supposed to be controlled. In this case, some knowledge of the failure mode is required for a proper choice of the controlling displacements [39]. It should be noted that the displacement field might exercise significant changes from a load level to another, especially when softening occurs. Other numerical procedures invoke the use of fictitious springs, replacing the original problem with an equivalent one for which the structural stiffness matrix remains positive definite [38]. The limitation of this technique is related to the trial and error invoked in the selection of the appropriate springs stiffness, making it applicable to simple cases only.

6.3 The Arc-Length Control

As shown in the previous section, the major problem in tracing the equilibrium path is engendered by the non-uniqueness of the solution beyond the first limit point, which makes the numerical analysis difficult when snap-through and snap-back occur. For this reason, in recent research more attention has been paid to numerical procedures that focus more on the path properties and less on the

traditional load or displacement controls. These methods are known as arc-length methods, they being originally introduced by Riks [34] and Crisfield [41] independently. The aim of these methods is to adjust the size of the load increment step in the load-deflection space in order to ensure the convergence of the numerical solution. This objective can be achieved by fixing the search of the iterative process on the equilibrium path in the vicinity of the last converged solution by means of a constant parameter called arc-length. In this manner, the search for the numerical solution can be restricted to a domain-of-convergence expressed in terms of a constraint equation that relates the effective load q_{ext} and the displacement iterative change dp to the fixed arc-length parameter $s = \text{constant}$. The arc-length parameter s should be seen as the radius of a sphere of convergence in which the solution is sought. The constraint equation is generally introduced by means of a scalar load multiplier $d\lambda$, a new independent variable that represents the iterative change in the load level λ and a constant s , representing the arc length defined as

$$s = \int ds = \text{constant} \quad (6.1)$$

and

$$ds^2 = dp^T dp + d\lambda^2 q_{ext}^T q_{ext} \quad (6.2)$$

where ds is the arc-length change on the equilibrium path corresponding to the iterative displacement change dp .

Figure 6.5 illustrates suggestively the main idea of Riks's arc-length control for a one DOF problem. Riks used to adjust the size of the load increment for each iteration step following the normal to the tangent solution (predictor). This basic concept is the basis for all linearised arc-length methods developed by a number of authors [23], [37], [39].

Figure 6.5 : Riks's original method with (a) Full Newton-Raphson Iterations (fN-R) and (b) Modified Newton-Raphson Iterations (mN-R)

In Figure 6.5, the domain of convergence delimited by the normal to the tangent solution (or to the secant solution as in [39]) may not be always effective when snap-through and snap-back dominate the structural response. These forms of arc-length have the disadvantage that they do not ensure always the convergence of the solution or they miss the equilibrium path as reported in [16].

A significant improvement to the original arc-length method was first introduced by Crisfield [16]. He replaced the normal to the tangential solution with a sphere of a fixed radius (Δs) and centered on the equilibrium path at the last converged solution, resulting in the so-called spherical arc-length method shown in Figure 6.6. This new method reduces the solution domain to a feasible one for which at least theoretically the solution converges always to a unique equilibrium point, even for cases when the linearised forms miss the equilibrium path [16].

In order to be able to incorporate the arc-length method in the incremental - iterative procedure, the differential form of the constraint equation (6.2) should be replaced with an incremental one that can be stated as

$$\Delta s^2 = \Delta p_{i+1}^T \Delta p_{i+1} + \Delta \lambda_{i+1}^2 q_{ex}^T q_{ex} \quad (6.3a)$$

where Δs is the incremental constant arc-length parameter.

In Eq. (6.3a), Δs is the fixed radius imposed for the solution domain and represents an approximation of the incremental arc-length, while Δp_{i+1} and $\Delta \lambda_{i+1}$ are incremental quantities that relate to the previously determined incremental parameters Δp_i and $\Delta \lambda_i$ as shown in Figure 6.6. The numerical experience in [40] has shown that instead of applying Eq. (6.3a), it is preferably to fix the arc-length parameter Δs in the space of the problem unknowns as

$$\Delta s^2 = \Delta p_{i+1}^T \Delta p_{i+1} \quad (6.3b)$$

for all iterations.

The N-R iterations are introduced by updating the new unknown load parameter $\Delta \lambda_{i+1}$ and its corresponding new incremental displacements Δp_{i+1} as

$$\Delta \lambda_{i+1} = \Delta \lambda_i + \delta \lambda_i \quad (6.4)$$

$$\Delta p_{i+1} = \Delta p_i + \delta p_i \quad (6.5)$$

where $\Delta \lambda_i$, Δp_i are obtained from the previous incremental step as shown in Figure 6.6, while $\delta \lambda_i$ and δp_i are the iterative changes still to be determined. At this point, it should be mentioned that $\delta \lambda_i$ is the only independent unknown, since δp_i can be expressed in terms of $\delta \lambda_i$ similarly to the relationship (5.4), but updating the out-of-balance force vector g_i according to the variable load level controlled by $\delta \lambda_i$

Figure 6.6: Spherical Arc-Length Method of 1 DOF Problem

$$\delta p_i = -K_T^{-1}(g_i - \delta \lambda_i q_{ext}) = \delta \bar{p}_i + \delta \lambda_i \delta p_T \quad (6.6)$$

where K_T is obtained at $(p_0, \lambda_0 q_{ext})$ and kept constant for all iterations in the case of mN-R procedure as shown in Figure (6.6). The out-of-balance force vector g_i is computed as usual according to Eq.(5.2). In Eq.(6.6) $\delta \lambda_i$ is still unknown and $\delta p_T = K_T^{-1} q_{ext}$ represents the tangential displacement change; q_{ext} is the actual load level. A similar form of Eq.(6.6) can be also obtained from a truncated Taylor series expression of the Newton-Raphson procedure as shown in [16].

By substituting of Eq.(6.6) into Eq.(6.5), the next incremental displacement level can be expressed as

$$\Delta p_{i+1} = \Delta p_i + \delta p_i = \Delta p_i + \delta \bar{p}_i + \delta \lambda_i \delta p_T \quad (6.7)$$

in which Δp_i is known from the previous increment, δp_T is known and $\delta \lambda_i$ is the unique unknown.

Finally, by substituting Eqs.(6.5) and (6.7) into Eq.(6.3b), the constraint equation can be re-expressed in terms of known parameters determined in the previous increment denoted as i and solved for the only unknown $\delta \lambda_i$ as follows

$$(\Delta p_i + \delta \bar{p}_i + \delta \lambda_i \delta p_T)^T (\Delta p_i + \delta \bar{p}_i + \delta \lambda_i \delta p_T) = \Delta s^2 \quad (6.8)$$

or

$$\alpha(\delta \lambda_i)^2 + \beta(\delta \lambda_i) + \gamma = 0 \quad (6.9)$$

where

$$\begin{aligned} \alpha &= \delta p_T^T \delta p_T \\ \beta &= 2(\Delta p_0 + \delta \bar{p}_i)^T \delta p_T \quad \gamma = (\Delta p_0 + \delta \bar{p}_i)^T (\Delta p_0 + \delta \bar{p}_i) - \Delta s^2 \end{aligned} \quad (6.10)$$

From Eq.(6.9), a solution for $\delta \lambda_i$ can be easily obtained and a feasible prediction for the iterative load level $\Delta \lambda_{i+1}$ becomes possible according to Eq.(6.4). As can be observed, the equilibrium equation system given by Eq. (6.6) becomes uncoupled from Eq. (6.9). In this way, the banded symmetric structure of the tangential stiffness matrix K_T is preserved, the solution of the equation system being obtained as usual by factorization or Gauss elimination. As mentioned before, the singularity of the stiffness matrix should not pose any problem because it is less

probable to achieve exactly the limit point. If it happens, then by an increment-cutting strategy a solution in the near of the limit point can be obtained. A major difficulty of this procedure is the existence of two solutions for the iterative load parameter $\delta\lambda_i$. Therefore, some investigations must be done in order to choose the right parameter $\delta\lambda_i$ that corresponds to the physical reality.

Figure 6.7: Schematic Illustration of the Two Possible Solutions for $\delta\lambda$

The two real roots of Eq.(6.9) are denoted as $\delta\lambda_1$, $\delta\lambda_2$. The success of the arc-length procedure depends on the choice of $\delta\lambda$ from the two roots $\delta\lambda_1$, $\delta\lambda_2$. A simple criterion suggested by Crisfield is to consider $\delta\lambda$ as the root that gives a positive pseudo-angle between the incremental displacement vector Δp_{i-1}^n before the present iteration and one of the incremental displacement vectors $(\Delta p_i^n)_{1,2}$ after the current iteration. The cosines of the two pseudo-angles are

$$\cos(\theta_1) = \frac{(\Delta p_{i-1}^n)^T (\Delta p_i^n)_1}{\Delta s^2} \quad \cos(\theta_2) = \frac{(\Delta p_{i-1}^n)^T (\Delta p_i^n)_2}{\Delta s^2}$$

The appropriate root of the Eq.(6.9) is the one that gives the smallest cosine value corresponding to a positive pseudo-angle. If both of the pseudo-angles are positive, the best suitable root is the one that is closest to the linear solution of Eq.(6.9)

$$\delta \lambda_i = -\frac{\gamma}{\beta}$$

The arc-length method described in this section is considered as a generalized displacement control method as mentioned in [16], although, physically, the problem does not involve any kind of displacements control. The major advantage of the continuation methods based on the arc-length parameter is stated in Eq. (6.3). It relies on the fact that these methods are very general and do not require user's interference in selecting appropriate stiffness or displacement control parameters. The load parameters $\Delta \lambda_i$ being scalars, they do not alter the load pattern of the externally applied loads q_{ex} . Also, Eqs.(6.3a) or (6.3b) are scalar equations which do not require the direct control of any physical displacement. The control on the size of the nodal displacements is exercised only in an average sense by limiting the Euclidean norm $\Delta p^T \Delta p$ of the incremental displacement change Δp to the fixed multiple of the squared arc-length parameter Δs^2 . In this manner, no direct knowledge related to the prediction of the deformation pattern is needed. It should be noted that the deformation pattern might change significantly after passing the limit point because of the formation of internal hinges in the overloaded structural system. For this

reason, the direct control of the displacement field Δp_{i+1} for which convergence should be ensured might become very difficult and unfeasible.

6.4 The Predictor Solution

In order to start the Newton-Raphson iterations, a tangential solution, usually denoted predictor, is needed. It must be noted that the size of the new load increment $\Delta q_p = \Delta \lambda_p q_{ext}$ has to be adapted to the size of the arc-length Δs . This requirement must be satisfied in order to make sure that the iterations, usually denoted corrections, are performed inside of the sphere of convergence delimited by the arc-length Δs as its radius. With the tangent stiffness matrix K_T determined at the beginning of the new load increment $\Delta q_{ext} = \Delta \lambda q_{ext}$, the displacements corresponding to the predictor solution are given by

$$\Delta p_p = K_T^{-1} \Delta q_p = \Delta \lambda_p K_T^{-1} q_{ext} = \Delta \lambda_p \delta p_T \quad (6.11)$$

where $\Delta \lambda_p$ and Δp_p correspond in Figure 6.6 to $\Delta \lambda_1$ and Δp_1 respectively. Substituting Eq. (6.11) into Eq. (6.3b), an estimation of the load increment $\Delta \lambda_p$ for a given arc-length Δs can be obtained

$$\Delta \lambda_p = \frac{\Delta s}{\sqrt{\delta p^T \delta p}} \quad (6.12)$$

Equation (6.12) gives an approximation of the required load increment $\Delta q_p = \Delta \lambda_p q_{ext}$ that ensures convergence of the iterations. Attention must be paid on how this load increment must be applied because it might represent an increase or a decrease of the applied external load, depending on the definition of the tangent

stiffness. The switch of the sign of the $\Delta\lambda_p$ can be automatically controlled by means of the so-called current stiffness parameter C_s introduced in [42] as a scalar measure of the stiffness of the system

$$C_s = \frac{k}{k_0} \quad (6.13)$$

where k and k_0 are scalar stiffness parameters obtained from the actual and the initial load increments respectively and defined in [16] as

$$k = \frac{\Delta q_p^T \Delta p_p}{\Delta p_p^T \Delta p_p} \quad (6.14)$$

As can be observed from Eqs. (6.13) and (6.14), the current stiffness parameter tends to zero in the vicinity of a limit point. This aspect constitutes a good indication in switching from the load control to arc-length control of the *N-R procedure as the limit point is reached. By imposing a value for the current stiffness parameter C_s , the solution is forced to switch automatically to the arc-length control, depending on the extent of the path non-linearity. The main advantage of introducing the current stiffness parameter C_s is that it is stable when reaching the bifurcation points where it is not simple to decide if the next load increment increases or decreases [16]. Difficulties are encountered if the sign of the determinant of the stiffness matrix is used in increasing or decreasing the load level.

6.5 Automatic Incremental Arc-Lengths

In the presentation of the arc-length procedure, it was assumed a constant arc-length parameter Δs that may be fixed by the user. This aspect is not very advantageous in conducting the numerical analysis. First, some knowledge and user

expertise is required in order to set a proper size for the arc-length parameter. Secondly, it is not efficient to use the same size of the increments for linear and nonlinear portions of the path. Accordingly, it can be concluded that a significant improvement of the numerical procedure can be obtained if the arc-length parameter is adjusted according to the degree of non-linearity of the corresponding path segment.

A simple technique that can be employed in calibrating the incremental arc-lengths was introduced by Ramm in [39]. He used to adjust the load parameter $\Delta\lambda_n$ based on the iteration history. This technique was introduced in the context of the arc-length procedure by Crisfield in [16] as follows

$$\Delta s_{i+1} = \Delta s_i \left(\frac{I_0}{I_i} \right)^{\frac{1}{2}} \quad (6.15)$$

in which Δs_{i+1} represents the prediction of the arc-length parameter for the next load increment. The constant I_i is the number of iterations required for convergence in the previous load increment based on Δs_i . I_0 is the desired number of iterations to achieve convergence for a load increment and is prescribed by user; it typically ranges in value from 3 to 5, but generally is problem dependent. As can be observed, Eq. (6.15) leads to the provision of small increments when the structural response becomes mostly nonlinear and larger increments when the structural response is almost linear. This technique is simple and does not require any estimation of the curvature of the equilibrium path in order to predict the arc-length for the next load step. For practical use, the user has to input a suitable value for the starting arc-length parameter with the addition of minimal and maximal step sizes. It should be noted that for the first few load increments the load control technique should be used, the switch to arc-length control being performed only in the vicinity of the limit points. The switch to arc-length control can be done automatically with the help of the current stiffness parameter C_i introduced in the previous section.

6.6 Automatic Increment Cutting

In certain cases, the convergence of the equilibrium iterations can not be achieved because the actual load increment is higher than the limit load that the structure can carry. In order to determine the limit load, an increment cutting procedure is employed when the actual load increment oversteps the limit load that the structure can support. The increment cutting strategy used in this work is simply based on the reduction of the actual load increment to half and starting the iterations again. This procedure can be applied in a cycle of three consecutive increment cuttings before to abort the numerical procedure. However, the computational efficiency can be improved by adjusting the size of the load increment according to the ratio of the desired convergence factor (the user-specified tolerance ε) and the current convergence factor ε_c that does not satisfy the imposed tolerance. This improvement can be easily achieved following

$$\frac{\Delta\lambda_{new}}{\Delta\lambda_{old}} = \frac{\varepsilon}{\varepsilon_c} \quad \text{or} \quad \frac{\Delta s_{new}}{\Delta s_{old}} = \frac{\varepsilon}{\varepsilon_c} \quad (6.16)$$

Automatic increment cutting can be also employed in the case that the spherical arc-length procedure fails because of non-existent real valued roots in Eq. (6.9). However, numerical experience shows that the simple increment cutting works very well for problems characterized by mild nonlinearities.

7 Computational Implementation Aspects of FEM

7.1 General

The FEM software is widely used in different engineering and scientific disciplines, where analysis of mechanical designs represents the main application area. Since the analysis requirements vary widely depending on the complexity of the designs and their intended functionality, the software requirements of FEM programs vary in a similar manner. This complexity makes FE analysis strongly dependent on an efficient computing environment, including both hardware and software facilities. For the software area this not only includes the FE programs, but involves a much broader spectrum of software engineering issues. Firstly, considering internal aspects of an autonomous FE program, one must recognize the necessity of designing the FE software in a way that supports the effective usage as well as the maintenance and further development of the program. Secondly, one must consider external aspects, i.e. where FE software is designed to support effective integration and communication with other engineering software environments and applications.

As illustrated in Figure 7.1, the FE application process is typically divided into four major activities: specification, discretisation, analysis and evaluation. It should be mentioned that each of these activities constitutes alone a formidable task

in translating it into computer code. A more difficult task is to assemble them for readability, maintainability and extendibility of the code.

Figure 7.1 The FE - Process

Modern commercial FE programs have already integrated the complete analysis process from modeling to evaluation and take advantage of graphical user interfaces where the analysis model can be specified in domain-specific terminology. The commercial FE programs are available as “black box” products, mainly designed for conventional and direct use, their major development requirements being related to the analysis functionality and efficient user interfaces and not to their internal organization.

In the field of engineering research and FE development, efficient processing implies that efficient algorithms and corresponding data structures are available. In this case, the program internal structure and its flexibility play the most important role

in the program development. The code readability and algorithm understandability are principal factors that may ensure the program extendibility, data communication or data composition into new derived information. The same holds for efficient development and maintenance where the overall complexity of this type of software can be elegantly reduced, ensuring a higher level of reusability by refining the program structure and increasing the code readability.

The major commercial FE programs have their origin in the 1960's. In contrast to the dramatic development in hardware performance during the last 15 years, the basic structure of the commercial FE programs and their development have not undergone dramatic improvements since their inception. This is partially explained by the fact that the advantage of the hardware performance is commercially more profitable than to redesign and re-implement the engineering software. On the other hand, the development of general FE programs meeting the commercial and industrial requirements implies a close interaction between several scientific and engineering disciplines. The scientific fields involve mathematics for numerical analysis and computational geometry, computer science for implementation and visualization techniques and not at the end, several fields of physics and mechanics where domain-specific applications are studied. In the engineering field, several classes of the FEM are applied to solve problems in areas such as mechanical analysis, heat conduction, fluid flow, etc. Accordingly, the investment in the FE developments usually involve large direct costs as well educational costs and additional costs of refreshing the old code. In addition, the software vendors have large investments in the existing software system where a technology change in the

implementation technique may become very costly. In such circumstances, sudden changes in the FE software become prohibitive.

As stated earlier, the FE software development is a very complex, interdisciplinary, time consuming and resource-dependent activity. Therefore, the conceptual design of FE software and its data management are of crucial importance in ensuring its long-term usage and development at high productivity and sustainable costs. With a well structured FE code, high quality research in large scale scientific computing environment becomes possible. Research software involving testing of new ideas and new computational models can be rapidly prototyped and developed leading to a faster pace of engineering research, technology transfer and valuable technological achievements.

7.2   The Modular Structure of the NONLACS FE – Program

At the present stage, the software development called NONLACS is limited to the scope of a research tool for 2D finite elements. The actual structure of the program is based on a first level object oriented (OO) concept intended to reduce the overall program complexity for an easy implementation of the nonlinear capabilities. In the actual program, the engineering data is expressed in terms of objects and their relationships. The features of the FORTRAN'99 programming language were found to serve well the scope of this development.

The engineering data and the program parameters are stored in a free format input file. Unconventionally, data needed in the nonlinear computations is read every time from the input file. In this manner, the need for saving the same data more than once is avoided. The objects are generated by means of the TYPE feature of the FORTRAN'99 programming language. Further, several objects of similar functionality are grouped into modules by means of the MODULE feature of the FORTRAN'99 programming language. For example, in Figure 7.2 all element data is structured in a module containing different sub-objects relating to the particular finite element structure of QFSE.

Figure 7.2: Examples of Data Modules.

In the ELEMENT_DATA module, QUAD_8 and RQUAD are distinct finite elements that can be used independently as well. The only difference between them is that the RQUAD uses the data of QUAD_8. In such circumstances, the data of the

QUAD_8 element is shared within the RQUAD element by inserting a pointer of the QUAD_8 object into the RQUAD data structure. The main benefit of this development is not only the economic memory space management at element level, but also the reuse of the code corresponding to QUAD_8 element. The same procedure is used to build the QFSE facet element that largely reuses the code of RQUAD and IDKQ elements.

This hierarchical data organization expressed in terms of FORTRAN'90 features corresponds to the inheritance concept of the most promising OO programming language C++. In the same manner, the boundary conditions, material data, load cases, the different analysis types and the program control data are structured in specialized modules that can be easily accessed everywhere in the program. For an easy access and code readability during the program development, all module definitions were stored together in a F90 file named TYPES.F90. The flow of the program is controlled by means of functions defined in specialized routines and subroutines. Each specialized function with its routines and subroutines is stored in a separate F90 file in order to obtain a better modular structure of the program. In this arrangement, the program flow can be easily followed from the MAIN program routine. For example, the arc-length control was attached to the program by means of a separate routine consisting of several subroutines. The arc-length control parameters were introduced in the program by means of a control data structure attached to the program control module. Consequently, the extension of the program was realized without any interaction with the existing code. Special attention was paid to the variables memory space allocation and initialization problems. A specialized

subroutine, structured on several hierarchical levels, is responsible for this task permitting easy control on the actual values of the variables, especially in the iterative computations. In addition, new variables could be easily introduced and initialized in the program in a very consistent manner. In the nonlinear computations, in case of divergence or lack of convergence after the maximum number of iterations is achieved, an automatic program restart from the last converged solution with a properly reduced load increment is provided. Such an automatic restart is obligatory when the arc-length control is invoked.

The above described program structure was found efficient for the scope of the QFSE finite element development. It is simple in implementation, the standard FORTRAN intrinsic mathematical functions being of great help. FORTRAN'99 provides the basic mathematical operations on multidimensional arrays. The FORTRAN'99 does not impose any restriction on the length of the specification names of the variables or data structures, this aspect being of great advantage in improving the code readability by using full specification names. Also, in FORTRAN'99 the memory space can be allocated and de-allocated on the fly by specialized language instructions `ALLOCATE()` and `DEALLOCATE()`, preserving local scope for the allocated data. However, for large-scale software systems, the complexity in data management increases dramatically and the choice of the programming language and its software environment can be easily considered as variables of the software design problem.

Because the FEM software design is a very complex activity, requiring more creativity than pure knowledge, the conceptual software development plays the

central role in meeting the basic requirements of a quality FE - software that were enumerated above. It was not intended in this thesis work to focus on such aspects, but the experience gained in this development opens surely a new way in designing and appreciating the complexities of FEM programming.

8

Examples

8.1 General

In this chapter several example problems are solved to demonstrate the degree of accuracy that can be achieved using the geometrically and materially nonlinear formulation of the layered QFSE described in Chapter 4. The geometrically nonlinear version of this element is part of the original work in this thesis. Furthermore, the coupling of the arc-length control to the nonlinear analysis in order to detect bifurcation and softening phenomena are part of the contributions made in this work.

8.2 Thick Cantilever Beam

This example highlights the accuracy and computational efficiency of the membrane element RQUAD when loaded in the in-plane bending mode considering the geometric nonlinear effects. Linear elastic material behavior is assumed despite the appreciable strains due to the thick shape of the cantilever beam.

The beam was modelled using two different mesh sizes as illustrated in Figure 8.1 (Thick Cantilever Beam). The tip-deflections versus the load diagram is plotted in the same figure, which also shows the theoretical solution of Bishopp and Drucker for this problem.

Numerical results indicate that the membrane element RQUAD offers a very good alternative to solid or plate/shell elements used in the discretisation of structures similar to the cantilever beam. RQUAD shows fast convergence for geometric nonlinear response. A number of 4 to 20 iterations per load increment were needed for a 0.01 tolerance of the convergence criteria in energy norm used. The numerical tests on the thick beam were performed for different number of layers varying from one to ten. No significant differences between the obtained results were detected even when large deformations (five to six times the thickness of the cantilever beam) occurred. In addition, tests with variable load increments were performed on the thick cantilever beam. The obtained results were found to be very close to each other.

The effect of the mesh size was also taken into account. Two meshes were considered, one mesh consisting of 10 elements disposed in one row and the other consisting of 40 elements disposed in two rows of 20 elements each. As can be observed from the plotted results in Figure 8.1, the mesh with 40 elements gives a better approximation. This is because there are two elements disposed over the beam thickness. The mesh with 10 elements is rather poor because the single elements disposed over the beam thickness are stressed in tension and compression. However, the numerical results show reasonable accuracy despite the element distortions that

inevitably appear due to the large deformation effects on the fixed mesh. In all examples, the Poisson's ratio was set to zero to facilitate comparison with an available analytical solution [58]. In the original analytical solution, the shear deformation and the Poisson's ratio are neglected.

Comparison with the results obtained by means of QUAD8 isoparametric membrane elements, as reported in Reference [49], shows that RQUAD has the same accuracy as QUAD8, but at a significantly lower computational cost ($\approx 50\%$).

It is of interest to remark that for thin sections, a reduction of the integration order or at least a correction factor for the shear strain energy should be used to avoid spurious shear stiffness. Reduced integration orders are of general applicability and it is known that this technique is able to improve the overall performance as will be shown later.

8.3 Response of a Beam-Column

The response of a beam-column is chosen in this example to highlight the influence of the large displacement effect on structural stability. The beam-column shown in Figure 8.2(a) of square section of $A = 1.0$, height $L = 10.0$ and $E = 120000$ was modelled using a 1×10 mesh of RQUAD elements. The vertical central load of $V = 24.7$ per load step was distributed equally at the nodes of the

RQUAD element, at the tip of the beam-column. Three iterations per load step corresponding to V were necessary for a 0.1 tolerance of the convergence criterion based on energy norm, showing again the effectiveness of RQUAD element.

In Figure 8.2(a), the response of the beam-column loaded axially by the load V shows a total different structural response for different magnitudes of the horizontally applied load $H = \beta V$. For a very small horizontal load, with $\beta = 1/10000$, the loss of stability occurs suddenly indicating a pure buckling behavior that is in good agreement with the linear elastic buckling load $V_{cr} = \frac{\pi^2 EI}{4L^2}$. For an increasing horizontal load, $\beta = 1/100$, $\beta = 1/10$ the explosive loss of the beam stability is replaced by a progressive softening process indicating a rather large displacement behavior than buckling.

In real design situations, the axial loads act usually eccentrically, depending on the particular construction details of the structure. Moreover, normal loads can also act on the structure making the stability analysis more complex. In addition, element geometric imperfections are always present in the real structures and they may influence significantly the stability of the structure.

The slenderness of the beam column under a geometrically fixed load is the most important factor that may lead to a buckling or to a large displacement problem depending on certain circumstances. This aspect is shown in the next set of beam-columns examples.

8.4 Response of a Thin Plate/Shell

This example shows the response of a plate compared to a shell of different curvatures expressed in terms of h/L ratios in Figure 8.2(b), where h is height of the arch and L is the span. A mesh 1×20 , consisting of QFSE elements was used in modelling the two sides clamped thin plate/shell. A thickness to length ratio of $1/100$ was considered ($t = 0.4$, $L = 40$). The distributed load was lumped into equivalent equal nodal point loads. It was assumed an elasticity modulus of $E = 200000$.

The obtained results show clear the plate stiffening effect for $h = 0$ in Figure 8.2(b). On the other hand, for $h = 0.25L$ the structure is a cylindrical shell and the response of the shell in this case is quite different compared to the above plate response. The small displacements of the shell are due to the compressive membrane forces developed by the bending action. In the case of the plate, tensile membrane forces are developed from the bending action, forces which contribute to the overall resistance of the plate. For a shallow shell with $h = 0.01L$, the structural response differs from the above two responses. In the earlier load stages, during the deformation process the shallow shell will buckle due to the development of compressive membrane forces, and following by a rapid increase of the central point displacements. At higher load levels, the curvature of the shallow shell disappears because of the deformations, the shell in the new configuration behaving like a plate.

8.5 Elastic Beam-Columns Loaded Eccentrically

This example is selected to show the nonlinear deformation effect on the buckling load magnitude of beam columns. Comparison with the Euler's elastic buckling load and Timoshenko's analytical solution [50] is considered.

The first column of $L = 10.0$, $B = 1.0$ and thickness $t = 1.0$ is divided into 10 RQUAD finite elements and loaded eccentrically as shown in Figure 8.3. The elastic buckling load for this types of columns is $P_{cr} = 2.47 EI/L^2$ (Timoshenko, [50]) where $E = 120000$, $I = 1/12$. In order to be able to analyze numerically the post-buckling behavior of this short beam-column, a perturbation horizontal load at the top is usually applied. In this example, the principal load P is applied eccentrically as shown in Figure 8.3. In this manner, no additional perturbing load is required in the analysis.

Analyzing the load – displacement curves plotted in Figure 8.3, it can be observed that large displacements occur at load levels, which are much lower than the elastic buckling-load P_{cr} . A rapid increase of the lateral displacement at approximately $0.6P_{cr}$ indicates the appearance of the so-called $P - \Delta$ effect at this load stage. However, it appears that the buckling load is slightly higher than the classical P_{cr} . Furthermore, the finite element results do not show a complete loss of positive stiffness.

It should be noted that this beam column of 1/10 thickness to length ratio is neither thin nor slender. Consequently, problems with the spurious shear stiffness are excluded. Comparison with the Timoshenko's analytical solution shows clearly the effect of the geometric nonlinear behavior on the buckling response of the beam-column in Figure 8.3. It should be noted that for this particular example, the geometric nonlinear effect is pronounced although the column is short and the analytical solution applies rather for slender columns. The FE solution in Figure 8.3 was obtained with the traditional load-control technique of the full Newton-Raphson procedure. This was possible because of the mild geometric nonlinear effect and a sufficient reserve in column's stiffness. A number of 3 to 4 iterations per load increment were needed for a 0.01 tolerance of the energy norm convergence criterion. The numerical tests were performed for different number of layers varying from one to ten and no significant differences between the results were detected even when large deformations (five to six times the thickness of the cantilever beam) occurred. Full and reduced integration orders were employed, the obtained solutions being very close each other. In addition, tests with variable load increments were performed on the short beam-column to test the correctness of the code and element formulation. The analysis results were found to be very close to each other. The numerical results demonstrate again the efficiency and accuracy of the RQUAD membrane element for in-plane large deformation analysis.

A second beam-column, similar but more slender than the first one, with $L = 20.0$, $B = 1.0$ and thickness $t = 1.0$, is further analyzed. The analysis results are plotted in the Figure 8.4. The column was idealized by 1×20 mesh of equal size

RQUAD elements. This example tests the geometric nonlinear capability of RQUAD membrane element and shows clearly the $P-\Delta$ effect. For this example, comparison with the Timoshenko's analytical solution appears more feasible. In this case, the magnitude of the displacements is larger, but the $P-\Delta$ effect manifests itself at $0.6P_{cr}$. For this analysis example, the arc-length control was employed. The size of the load increments was set automatically corresponding to a desired number of 3 iterations needed for convergence, lower load increments being preserved in the zones with pronounced changes of the equilibrium path configuration. A convergence tolerance of $\varepsilon = 0.001$ for the energy criterion was found more than satisfactory. The current stiffness parameter (denoted CSP and plotted on Figure 8.4) has a smooth variation showing the mild nonlinear character of the analyzed problem. The analysis could be done under load control only, but difficulties were encountered in choosing the proper size of the load increment in order to ensure the convergence of the analysis in a reasonable number of iterations. No difficulties in passing the load limit point were detected. However, the FE solution obtained for a 1×20 mesh is slightly stiffer than the Timoshenko's analytical elastic solution. A mesh 2×20 could not improve the overstiff response of the column as shown in Figure CE.2. Also, a reduced integration order was not able to improve the overstiff results significantly. Although, the column being not thick but also not very slender, shear locking phenomenon does not come in the discussion, but some spurious shear stiffness was detected. To reduce the amount of parasitic shear, a reduction of the shear modulus $G = E/60$ was applied. The improved solution shows clear the presence of parasitic shear provided by a too high amount of shear strain energy.

Note that a reduction of the shear modulus to the previous analyzed short column could not improve the FE solution. An elasto-plastic analysis is of less interest in this example, because the Timoshenko's analytical solution refers to a perfect elastic material assuming inextensibility and no shear deformations. In this example, RQUAD shows again its accuracy and computational efficiency.

In a very slender column similar to the above beam-columns, with $L = 100.0$, $B = 1.0$ and thickness $t = 1.0$, the shear locking effect was detected. A plot of the locked FE solution could not be shown because the magnitude of the displacements was extremely small. It was found that the FE solution is very sensitive to changes of the shear modulus G . As a remedy, a reduction of the integration order associated with a severe decrease of the shear modulus ($G \cong E/1000$) was considered. The Poisson's ratio was set to zero. A load controlled FE solution was found and plotted in Figure 8.5(a). Here it appears that the true first buckling mode occurs at a load level ($\approx 0.6P_{cr}$) which is significantly lower than the Euler's elastic buckling load P_{cr} . It should be noted that due to the pronounced slenderness of the beam-column, the load-control incremental technique is no longer adequate to trace the post-buckling load-deflection path. Indeed, this example shows the limitation of the load control technique. The finite element solution failed to converge in 100 iterations after the first buckling mode occurred. In analyzing the very slender beam-column, a mesh of 100 rectangular RQUAD elements was used in the load-controlled analysis. The spurious shear stiffness was avoided by employing a reduced integration order (2×2), but maintaining $G \cong 0.5E$. Significant variation in the required number of iterations required for convergence can be seen in Figure 8.5(a). Also, in the same

figure can be observed that the load-deflection curve of the column tip is not smooth. The reduced integration order (2X2) alone was sufficient to improve the FE solution. The FE solution failed to converge in 100 iterations after the first buckling mode occurred.

In Figure 8.5(b), the same slender beam column was analyzed using a mesh of 20 rectangular RQUAD elements with the sides ratio of 1:5. This analysis shows clearly an overstiff solution when full integration order (3×3) and shear modulus $G \cong 0.5E$ are employed. An efficient way in avoiding the spurious shear stiffness was found by applying a more severe reduction of the shear modulus, particularly for this example, $G \cong E/1000$ was chosen for a full integration order. The generalized displacement control of the Arc-Length Method was employed to trace the post-buckling equilibrium path. Unexpected numerical difficulties were detected in the first three (automatic) load increments provided by the doubling-back of the arc-length procedure. The latter results showed that the efficiency of the arc-length procedure depends strongly on the chosen constraint parameter (Δs), which must be specified in the input data. The load and the displacement limit points of the horizontal displacement of the column's tip were passed without any difficulty. The plotted results in Figure 8.5(b) are in perfect agreement with analytical solution reported in [49]. The accuracy of the RQUAD element can be remarked also for an element sides ratio of 1:5. In addition, the possible mesh distortions might cause numerical problems too, but this aspect may not be of interest for the scope of this study. However, more research work is needed in order to avoid of spurious shear

stiffness and to improve the overall element computational effectiveness for special cases like this.

8.6 Two Hinged Deep Arch

This example was chosen to test the ability of the arc-length method in passing bifurcation equilibrium points. This problem has been extensively analyzed by Huddleston [51] and Da Deppo and Schmit [52]. The example presented in Figure 8.6 shows again the capability of the RQUAD finite element in geometric nonlinear computations, especially in the analysis of buckling and ultimate load capacity problems. The geometric and material properties are given in Figure 8.6. The two-hinged deep arch was modelled using a mesh consisting of a row of 40 RQUAD elements. The two-hinged deep arch was loaded with a concentrated load in the middle to simulate the symmetric deformation mode. In order to obtain the asymmetric deformation mode, a perturbing horizontal load is usually introduced. Alternatively, the asymmetric deformation mode can be introduced by a small perturbation to the radius of the arch ($R/1000$) on one side as in [51]. Plots of the central point load versus central deflection show good agreement with the analytical solutions from [52]. It was also found that the axisymmetric buckling mode occurred first in the finite element simulation for a load level of 1079 *lb* that is very close to the analytical axisymmetric buckling load of $13 EI/R^2 = 1083$ *lb* specified in [51]. The symmetric buckling load occurred in the finite element

simulation for a load level somewhere after 1245 *lb*. The analytical symmetric buckling load specified in [51] is $15.3 EI/R^2 = 1275 \text{ lb}$, showing reasonable accuracy of the finite element simulation. As shown in Figure 8.6, the two buckling modes are close to each other. This generates serious numerical difficulties due to the existence of two possible equilibrium paths that can not be both followed at the same time. In such circumstances, special attention must be paid in choosing the load increment size and its orientation that in the finite element context should be done automatically. For this example, more accuracy of the buckling loads can be easily obtained for smaller load increments in the vicinity of the limit points. However, in tracing the post-buckling behavior a displacement control technique should be employed.

The arc-length control diverged immediately beyond the bifurcation point. The numerical analysis in this example is particularly difficult because of the path ramification beyond the bifurcation point. The current stiffness parameter is not sensitive to bifurcation points. This might be an advantage, but in the particular case of this example, increasing the load beyond the bifurcation point does not correspond with the load decrease dictated by the occurrence of the first (axisymmetric) buckling mode. The symmetric buckling mode can be easily obtained by suppressing the horizontal DOF's at the nodes located on the symmetry axis (the arch vertical center line). Following this procedure, the load limit point of the symmetric buckling mode is obtained, but such procedure is not recommended for practical purposes. The analysis under arc-length control can be also conducted imposing a form of displacement control for a particular displacement. Restarting the

analysis in the vicinity of the bifurcation point with appropriate values for the arc-length parameters may be used as well. However, in a multi-DOF's system the tangent stiffness matrix is no longer tangent to the equilibrium path that may not be a smooth surface. Due to the slenderness of the arch and the loading pattern, some spurious shear stiffness in this example was also detected, leading to slightly increased number of iterations for each load increment.

8.7 Simply Supported Square Plate under Uniform Load

This example demonstrates the effect of membrane-bending interaction in plates. A simply supported square plate under uniform load with all edges horizontally restrained is analyzed. Two meshes of 8X8 and 16X16 QFSE elements are used. All geometric and material characteristics of the plate are shown in Figure 8.7. The cross section of the plate is divided into 10 layers. Since for the IDKQ element the work equivalent nodal loads corresponding to the normal uniform load can not be established, the distributed load was approximated by means of equal concentrated loads applied at each node. A tolerance of 0.1 for the convergence criterion in energy norm was sufficient to achieve reasonable accuracy. Elastic material behavior is assumed to facilitate comparison with Levy's analytical solution.

As can be observed in Figure 8.7, the geometric nonlinear response of the plate results into a pronounced plate stiffening when compared to the linear response

of the same uniformly loaded plate. The plate stiffening effect is generated by the tensile membrane forces that are developed by the bending action.

Comparison between the central deflection obtained from the finite element analysis and the Levy's analytical solution shows good agreement up to maximum deflection of less than 1.5 of the plate thickness. For larger deflections, twice the thickness in this example, an error in the applied load level can be observed. It should be noted that the load used in the actual finite element analysis is an approximation obtained by calibrating the nodal loads to obtain the linear solution accurately (Figure 8.7). On the other side, the Levy's solution does not take into consideration the nonlinear in-plane strain components, numerical evidence showing that these terms could be neglected. A comparison of the stress levels with the analytical solution is also of interest knowing that QFSE is an C^0 element and inter-element stress discontinuity exists.

With regard to the practical use of the quadrilateral facet elements undergoing large deformations should be noted. As known, the quadrilaterals are prone to distortions because their corner nodes do not remain coplanar for the general deformation case [18]. Depending on the magnitude and pattern of deformation, the induced distortions in quadrilateral elements might lead to an ill-conditioned structural stiffness matrix and unrealistic results. A mesh refinement would not be a guaranty for accurate results. Moreover, an excessive mesh refinement may diminish the computational efficiency. Also, mesh controls are not easy to employ in order to ensure that all quadrilateral facet elements become planes after re-meshing. Based on a distortion measure at element level, the elements

accumulating excessive distortions can be replaced by more specialized shell elements. In this manner, an optimal balance between accuracy and computational cost can be established. In the present example, the level of geometric distortions is low and relatively good results were obtained.

An alternative solution to the above mentioned drawbacks of the QFSE can be found by reformulating the facet element for the triangular shape. Precisely, the formulation of the constituent finite elements of QFSE can be done for triangular shaped elements keeping all the advantages of this DK class of finite elements. The triangular elements are not prone to the same geometric distortions as the quadrilaterals, three points defining always a facet. Mesh refinement remains a necessity depending on the particular deformation patterns in the structure.

8.8 Folded Plate Structure

This practical example is able to demonstrate the ability of QFSE element in approximating the physical problem of a non-coplanar plate assemblage accurately. A linear FE solution using triangular flat elements composed of a constant strain membrane and an incompatible plate was presented in [19], but comparison is made with a set of experimental results obtained by Mark and Riesa [19]. Only half of the structure was discretized due to structural symmetric geometry, material properties and boundary conditions about the vertical center line as shown in Figure 8.8. The same mesh size as used in [19] was considered, but using quadrilaterals instead of

the triangular elements. The rigid diaphragm was idealized by suppressing all DOF's on the right margin of the structure. The frame stiffness represented by the 1 in rib disposed on the left margin of the structure was introduced in analysis also by means of QFSE flat elements of 1.5 in. assumed thickness. The FE linear solution obtained with QFSE flat elements is close to the FE solution presented in [19], but the geometric nonlinear displacement solution is closer to the experimental results as shown in Figure 8.8. The gain in accuracy of the solution is not spectacular for the considered load level shown in Figure 8.8. However, the geometric nonlinear displacement solution overlaps the experimental results of Mark & Riesa. The displacement deviation at the left margin in Figure 8.8 is mainly caused by the assumed rib thickness (1.5 in.), the original one being not specified in [19]. The FE internal forces do not meet the experimental results exactly. Moreover, the geometric nonlinear and the linear solutions of the internal forces are almost identical. In Figure 8.8, the average of the internal forces at the same nodes of the neighbor elements was plotted. The membrane-bending effect has little influence on the accuracy of the results due to the low load level. For higher load levels, only a nonlinear FE solution is able to predict the structural response that is controlled by the local buckling modes of the thin rib or vertical wall of the folded plate. In this respect, for such tiny structures the linear solution alone is not very helpful in estimating the structural resistance because the load level corresponding to a linear response can not be easily estimated in advance.

8.9 Fully Clamped Cylindrical Square Shell Uniformly Loaded

A square cylindrical shell under uniform load was analyzed by means of facet element QFSE. This example shows the applicability limits of this facet element in approximating the large deformations of thin shells. The analysis of shells is much more difficult than that of plates due to the stability problems of the bifurcation and local buckling phenomena.

A quarter of the square cylindrical shell was modelled using a 4×4 mesh of QFSE elements. The geometric and material properties of the shell are given in Figure 8.9. The uniform load was approximated by means of equivalent equal nodal point loads.

As can be observed in Figure 8.9, in the early loading stages the membrane-bending interaction results in a progressive softening of the shell structure until the first buckling mode occurs. The example involves no limit points since the equilibrium path is always stable as shown in Crisfield [59].

The first observation is that the facet element QFSE as it is can not approximate reasonably large displacements in shells. There are several reasons, but two of them worth commenting.

First, the independent response of the membrane and bending constituents of the facet element can not approximate accurately the stress-strain situation of the shell undergoing large displacements. The reason is that the initial curvature of the quadrilateral facet element is not taken into account in the strain definition. For the

four-node element geometry, the initial element curvatures can not be determined. Note that this aspect is closely related to the problem of element distortions. Therefore, the choice of the TL formulation is fully justified, the computations being done on the initial element configuration, but the problem of large displacements can not be overcome in this manner. Once the large displacements occur, the element initial undeformed configuration used in TL is no longer valid in approximating the stress-strain state of the deformed element. The good results obtained for the early loading stages are justified by the fact that the element distortions are not prominent and the Green strain definition remains reliable. A remedy for this shortcoming is to insert in the strain definition the effect of the initial curvatures corresponding to the different available shell theories.

The second remark pertains to the ability of the flat assemblage to approximate accurately the smooth shell surface. Theoretically, by means of successive mesh refinement, the solution of the flat assemblage should converge to the shell solution. If the discretisation is based on a coarse mesh, the resulting flat assemblage becomes stiffer as shown in the Figure 8.9 by the bending stiffness ($\nu = 0.3$) plotted against the “exact” solution given by Crisfield [59]. However, through successive mesh refinement a more accurate solution can be obtained. The bending stiffness with $\nu = 0.0$ fails to predict accurate results for this example. Spurious shear stiffness or shear locking ($K_{shear} / K_{bending}$ is high) is not present because of the assumed Discrete Kirchhoff constraints $\gamma_{zi} = 0$.

The results obtained by means of the facet element QFSE are slightly more accurate than the results obtained with the IDKQ bending element, but only behind

on the softening branch. Membrane locking ($K_{\text{membrane}} / K_{\text{bending}}$ is high) characteristic for thin shells was detected. Generally, the locking effects are generated by the high order strain components resulting from the derivatives of the interpolation function of the displacement field. For the special case of the flat element QFSE, the shear locking problem relies in the fact that the in-plane shear strain energy is provided by the membrane and bending shear strains $\gamma_{xy}^m, \gamma_{xy}^b$. The transverse shear strains $\gamma_{xz}^b, \gamma_{yz}^b$ being suppressed by the DK assumptions, they do not contribute in any way to the shear strain energy. To avoid the membrane locking effect, strain constraints for the bending modes $\varepsilon_{ij} = 0, i \neq j$ similar to the DK constraints applied for the transverse shear strains can be introduced. A numerical test considering the bending stiffness only and $\nu = 0.0, \varepsilon_{ij} = 0, i \neq j$ is shown in Figure 8.9, but this simple treatment of the membrane locking problem is not reliable. The reason is that the in-plane membrane and bending strain components can not be reduced in the same manner as the transverse shear strains due to the Poisson's effect. The shear locking problem in 4-node warped elements was solved in [60] by using a one-point integration rule and a nodal projection.

8.10 Williams Toggle Frame

Williams [13] solved this problem analytically and compared the results to experimental data. The selection of this example was motivated by the possibility of testing the ability of the arc-length procedure in following the equilibrium path. The geometry and the cross-sectional properties are given in Figure 8.10. The toggle frame was analyzed using first the load control technique. Numerical tests were performed using different convergence criteria. As shown in Figure 8.10, the displacement norm fails to predict accurate results. This can be explained by the small numbers expressing the displacements in inches. The energy norm gives better results, but due to the presence of the snap-through of the central vertical displacement, the equilibrium path is lost at the beginning of the softening branch. The frame having a slenderness ratio of $\approx 1:100$, spurious shear stiffness was detected. Consequently, a major reduction of the shear modulus was applied, $G \cong \frac{1}{200} E$. The reduced integration order did not offer a significant improvement compared to the reduced G modulus considered. The arc-length control performed well, 4 iterations per load step being necessary for convergence within a tolerance of 0.0001 that was found to be too tight. An overall improvement of the results was obtained by the simple switch to the arc-length control. The current stiffness parameter was also plotted in Figure 8.10. It shows a strong oscillating character, but it indicated correctly the increase or decrease of the load level in following the equilibrium path. The variation of the horizontal reaction force H was also plotted in

Figure 8.10. It shows a displacement limit point. The obtained results are in a good agreement with the experimental data from [13].

8.11 Shallow Circular Arch Subject to Near Central Load

This example is used also as test sample for the arc-length method. The arch is discretized by means of a row of 22 RQUAD membrane elements. The arch section is shown in Figure 8.11 and it indicates that this structure is not a slender one. The concentrated load is applied eccentrically in order to avoid the bifurcation problem met at the Two-Hinged Deep Arch example. Full integration order (3×3) was used. Spurious shear stiffness was not detected, nor expected. The RQUAD membrane element shows again fast convergence and no difficulties in passing the load limit points accurately. In this example, the current stiffness parameter shows a smooth variation having clear negative values for the softening branch. The automatic arc-length parameter defined by the Eq.(6.15) works as expected, in the vicinity of the limit points the size of the load increments being correspondingly reduced. This example shows clearly the good ability of the arc-length concept in predicting the equilibrium path beyond two consecutive load limit points in a mildly nonlinear problem. In addition, it should be noted that the quality of the finite element in predicting the structural behavior determines the accuracy of the results in

following the equilibrium path beyond the critical points. Arc-length technique is responsible for a proper adjustment of the load parameters in order to ensure convergence, provided a solution exists. No experimental or analytical solutions are available for this structure.

8.12 Shallow Hinged Sinusoidal Arch Subject to Uniform Pressure

This example was selected to test the accuracy of the arc-length method based on the automatic load generation of Eq. (6.15) and the current stiffness parameter concept stated in Eqs. (6.13), (6.14). As shown in Figure 8.12, the finite element solution of an elastic shallow arch of sinusoidal shape under uniform pressure is in a good agreement with the analytical solution given by Timoshenko and Gere in [50]. The automatic load increments were activated by setting the maximal values of the arc-length parameters high in order to avoid any limitation in the expansion of the load levels. No difficulties were encountered in passing the load limit points, the current stiffness parameter indicating correctly the increase or decrease of the load following the equilibrium path. Only the symmetric buckling is considered by modeling one half of the structure and imposing the symmetry boundary condition at the top of the arch. Sixteen RQUAD finite elements were used. The uniform load was decomposed into equivalent nodal forces. The Energy Euclidean Norm was used to cut the Newton-Raphson iterations for $\varepsilon = 0.001$. With

dimensions as given in Figure 8.12, the span/thickness ratio is 50, which is in the practical range of the steel structures. From a physical point of view, an elasto-plastic analysis may give more realistic results which can not be easily obtained with the analytical solutions of the classical theory of elasticity. The FE elasto-plastic analysis plotted in Figure 8.12 confirms the fact that although the structure is slender but not very thin plastic strains occur, but their effect is not significant. The variation of the current stiffness parameter CSP as function of loading p is shown in Figure 8.12. The points along the CSP curve have automatically been distributed according to the degree of nonlinearity of the system. This verifies that Eq. (6.15) gives a good estimate for the automatic load increments. In the case of the elasto-plastic analysis the size of the increments was diminished corresponding to an average of 3 iterations for convergence. The arc-length parameters for this example are shown in Figure 8.12.

Load - Displacement Curves

$E = 200000 \text{ N/mm}^2$
 $\nu = 0.0$

Figure 8.1 : Thick Cantilever Beam

Figure 8.2(a) : Response of a Beam-Column

Figure 8.2(b) : Response of a Thin Plate/Shell

Figure 8.3 : Column Elastica

Figure 8.4 : Column Elastic

Figure 8.5(a) : Elastica Problem

Figure 8.5(b) : Elastica Problem

$P_{cr} = 15.2EI/R^2$ (Elastic Symmetric Buckling, Huddleston [51])
 $P_{cr} = 13.0EI/R^2$ (Elastic Asymmetric Buckling, Huddleston [51])

- RQUAD Symmetric Buckling U(X)
- - - RQUAD Symmetric Buckling V(Y)
- RQUAD Axisymmetric Buckling U(X)
- - - RQUAD Axisymmetric Buckling V(Y)
- ▲ Analytical Sym. Buckling V(Y) (Ref. [40])
- ▲ Analytical Axisym. Buckling V(Y) (Ref. [40])
- Number of Iterations
- Arc-Length Sym. Buckling V(Y)
- Arc-Length Axisym. Buckling V(Y)

Figure 8.6 : Two - Hinged Deep Arch

$h = 2.54 \text{ mm}$
 $E = 200000 \text{ N/mm}^2$
 $\nu = 0.316$

Figure 8.7 : Simply Supported Plate

Plate	Vertical load (lb/in ²)
1	0.009
2	0.009
3	0.129

$t = 2.54 \text{ mm}$
 $E = 3560 \text{ lb/in}^2$
 $G = 1420 \text{ lb/in}^2$
 $n = 0.430$

Figure 8.8 : Folded Plate

Figure 8.9 : Fully Clamped Cylindrical Shell under Uniform Load

Figure 8.10 : Williams Toggle Frame

Figure 8.11 : Shallow Circular Arch Subject to Near Central Point Load

9 Conclusions and Recommendations

9.1 Conclusions

Based on the results of the analyzed problems, the following conclusions are reached:

- 1) Both finite elements, RQUAD (membrane-type) and IDKQ (bending-type) perform extremely well in linear analysis problems. They inherit the same accuracy of their parent elements, QUAD8 and QUAD9, respectively, but at a significantly lower computational cost. Their computational efficiency is intensively exploited particularly in the layered model where for each layer a stiffness matrix is computed. The extremely good performance of RQUAD, when loaded in the in-plane bending mode and the accuracy of IDKQ for general quadrilateral element shapes is note worthy.
- 2) The membrane element RQUAD, when used for geometric nonlinear analysis, shows good agreement with the analytical solution of some standard problems, and achieves fast convergence in a numerically stable condition. It shows some form of spurious shear modes in the case of the slender beam-column example, but this problem could be resolved by a proper reduction of the shear modulus. In

addition, special attention must be paid to the severe element distortions caused by the geometric changes during the deformation process.

- 3) The plate-bending element IDKQ, despite its accuracy in linear analysis, has a drawback: its work equivalent load vector can not be established because the corresponding displacement shape functions can not be explicitly stated. The original displacement shape functions of the former element QUAD9 were replaced by the imposed cubic displacement functions on the border and centerlines. However, for practical purposes the distributed normal loads can be well approximated by nodal concentrated loads. The problem of the work equivalent loads becomes more important in nonlinear analysis problems where the errors induced by the lumped (but not work equivalent) loads mentioned before may accumulate.
- 4) The facet element QFSE (RQUAD + IDKQ) can be seen as a good alternative to the traditional plate or shallow shell elements when geometrically nonlinear effects are not taken in account. It has six DOF's per node, this feature makes it easy to connect to other types of finite elements, such as beams. It is computationally cheap, easily to formulate and implement and it possesses the same accuracy level as of its constituent elements. Difficulties arise when severe geometric nonlinear effects occur where element distortions (especially warping distortions) may affect the accuracy of the solution. The quadrilateral shape of the QFSE element does not permit its use in approximating arbitrary shell structures.

It can be successfully used in analyzing general cylindrical shells. However, QFSE can be used in analysis of mildly geometric nonlinear problems, as the nonlinear plate analysis for example. In case of significant mesh distortions, as a remedy, the distorted elements could be replaced by specialized shell elements by means of a mesh refinement technique.

- 5) The arc-length continuation method was found reliable for examples with a low number of DOF's exhibiting a snap-through behavior that can be considered as mildly nonlinear. For multi-DOF's systems, the arc-length control becomes less efficient but it still works. A major problem of the arc-length method occurs at the bifurcation points where a divergence of the equilibrium path may occur. In such situations, it is not simple to decide if the load level beyond the critical point should be increased or decreased. A restart program facility might help in testing what are the possibilities in detecting the equilibrium path. The problem becomes difficult for a multi-DOF's system where the tangential stiffness is generally not tangent to an equilibrium path that may not be represented by smooth surface.
- 6) It was found that the (Bergan's) current stiffness parameter is not sensitive to bifurcation points. For smooth nonlinear problems, it performs well in assessing the stiffness reserve of the structure, its negative value indicating the necessity of a load decrease in order to be able to follow the equilibrium path. However, it oscillates strongly in thin structures as in the Williams Toggle example. In the present work, the current stiffness parameter was used as an indicator for

increasing or decreasing the load level. This was necessary because the sign of the stiffness matrix determinant did not changed in the case of the slender structures. It should be noted that the thin structures are dominated by large displacements but small strains. In addition, it should be mentioned that no difficulties were encountered in passing load limit points. The only difficulty appeared in passing the displacement limit points. Accordingly, it was found that the Bergan's current stiffness parameter may be ineffective at the displacement limit points.

- 7) The arc-length control is considered as a form of generalized displacement control, but in average sense only and determined by the scalar constraint equation $(\Delta s^2 = \Delta p_{i+1}^T \Delta p_{i+1})$. In fact, the load level is updated by means of the λ parameter so that the incremental displacements remain inside of the sphere of convergence of radius Δs , according to the scalar constraint equation. From this point of view, the arc-length control does not represent any form of displacement control. The only advantage of the arc-length control is that the load pattern is controlled during the analysis. The displacement pattern may change, depending on the load level and particular characteristics of the structure. A displacement control is possible only if the structural response is known in advance, otherwise it may lead the analysis in an unfeasible solution direction.

9.2 Recommendations for Further Research Work

Following the results of the present development, the following research directions are suggested

- 8) The formulation of the RQUAD membrane element should be improved by a correction factor for the shear strain energy in order to avoid the spurious shear stiffness detected in thin structures. In the present work, the spurious shear stiffness was reduced by considering a proper reduction of the shear modulus as in the case of the slender beam column example. This simple technique requires a very high amount of testing work and deep analysis and it is not recommended for practical purposes.
- 9) Element distortions may be a source of inaccurate results. They can be measured and kept to a certain tolerance level by means of adaptive re-meshing techniques. The QFSE facet element is very sensitive to intolerable distortions during the large displacement analysis. However, the quadrilateral facet elements detected as being distorted can be replaced by similar triangular elements. This can be done efficiently by local re-meshing.
- 10) Developing of the work equivalent load vector for IDKQ element is of interest. For thin shells, the distributed loads constitute the principal load case. If for a

linear structural analysis the distributed loads can be approximated by concentrated nodal forces, in the case of nonlinear analysis the errors of the load approximation may accumulate leading to undesirable structural responses. A reasonable choice for the shape functions necessary to interpolate the distributed loads was suggested in section 4.4.

- 11) The TL and the UL methods for geometrically nonlinear problems should be combined for computational efficiency. TL has the advantage that it does not require intensive coordinate transformations. UL has the advantage that it can satisfy the small strain criteria and minimizes the errors arising from rigid body motion.
- 12) The arc-length control technique was found reliable for structures characterized by softening or hardening responses. In the case of thin shells, the arc-length control exhibits doubling back of the response. A program restart facility seems to be necessary in analysis of difficult equilibrium paths. In thin shells, buckling occurs rather explosively and at small strains. Further research work is needed in order to adapt the concept of the arc-length control for this particular class of problems.
- 13) The quality of the code development may play an important role in ensuring the success of the research activity. Three principal factors should be considered at the code implementation stage: readability, maintainability, and extendibility.

Programming of the FEM is a very complex and time consuming activity and it is of crucial importance to keep a developer-friendly coding to make possible the reusability of the program for future work and improvements.

APPENDIX I

RQUAD Transformation Matrix $[T]$

$$[d]_{(16 \times 1)}^{QUAD8} = [T]_{(16 \times 12)} [d]_{(12 \times 1)}^{RQUAD}$$

$$\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \\ u_5 \\ v_5 \\ u_6 \\ v_6 \\ u_7 \\ v_7 \\ u_8 \\ v_8 \end{bmatrix}_{(16 \times 1)}^{QUAD8} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\ \frac{1}{2} & 0 & b_1 & \frac{1}{2} & 0 & -b_1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{1}{2} & a_1 & 0 & \frac{1}{2} & -a_1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2} & 0 & b_2 & \frac{1}{2} & 0 & -b_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{2} & a_2 & 0 & \frac{1}{2} & -a_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & \frac{1}{2} & 0 & b_3 & \frac{1}{2} & 0 & -b_3 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{1}{2} & a_3 & 0 & \frac{1}{2} & -a_3 \\ \frac{1}{2} & 0 & -b_4 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{1}{2} & 0 & b_4 \\ 0 & \frac{1}{2} & -a_4 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{1}{2} & a_4 \end{bmatrix}_{(16 \times 12)} \times \begin{bmatrix} u_1 \\ v_1 \\ \omega_1 \\ u_2 \\ v_2 \\ \omega_2 \\ u_3 \\ v_3 \\ \omega_3 \\ u_4 \\ v_4 \\ \omega_4 \end{bmatrix}_{(12 \times 1)}^{RQUAD}$$

M. Nofal [26]

APPENDIX II

IDQK Transformation Matrix $[T]$

$$[d]_{(18 \times 1)}^{QUAD9} = [T]_{(18 \times 12)} [d]_{(12 \times 1)}^{IDKQ}$$

$$\begin{bmatrix} \theta_{x1} \\ \theta_{y1} \\ \theta_{x2} \\ \theta_{y2} \\ \theta_{x3} \\ \theta_{y3} \\ \theta_{x4} \\ \theta_{y4} \\ \theta_{x5} \\ \theta_{y5} \\ \theta_{x6} \\ \theta_{y6} \\ \theta_{x7} \\ \theta_{y7} \\ \theta_{x8} \\ \theta_{y8} \\ \theta_{x9} \\ \theta_{y9} \end{bmatrix}_{(18 \times 1)}^{QUAD9} = \begin{bmatrix} 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 \\ T_{9,1} & T_{9,2} & T_{9,3} & T_{9,4} & T_{9,5} & T_{9,6} & 0 & 0 & 0 & 0 & 0 & 0 \\ T_{10,1} & T_{10,2} & T_{10,3} & T_{10,4} & T_{10,5} & T_{10,6} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & T_{11,4} & T_{11,5} & T_{11,6} & T_{11,7} & T_{11,8} & T_{11,9} & 0 & 0 & 0 \\ 0 & 0 & 0 & T_{12,4} & T_{12,5} & T_{12,6} & T_{12,7} & T_{12,8} & T_{12,9} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & T_{13,7} & T_{13,8} & T_{13,9} & T_{13,10} & T_{13,11} & T_{13,12} \\ 0 & 0 & 0 & 0 & 0 & 0 & T_{14,7} & T_{14,8} & T_{14,9} & T_{14,10} & T_{14,11} & T_{14,12} \\ T_{15,1} & T_{15,2} & T_{15,3} & 0 & 0 & 0 & 0 & 0 & 0 & T_{15,10} & T_{15,11} & T_{15,12} \\ T_{16,1} & T_{16,2} & T_{16,3} & 0 & 0 & 0 & 0 & 0 & 0 & T_{16,10} & T_{16,11} & T_{16,12} \\ T_{17,1} & T_{17,2} & T_{17,3} & T_{17,4} & T_{17,5} & T_{17,6} & T_{17,7} & T_{17,8} & T_{17,9} & T_{17,10} & T_{17,11} & T_{17,12} \\ T_{18,1} & T_{18,2} & T_{18,3} & T_{18,4} & T_{18,5} & T_{18,6} & T_{18,7} & T_{18,8} & T_{18,9} & T_{18,10} & T_{18,11} & T_{18,12} \end{bmatrix}_{(18 \times 12)} \times \begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \\ u_3 \\ v_3 \\ \theta_3 \\ u_4 \\ v_4 \\ \theta_4 \end{bmatrix}_{(12 \times 1)}^{IDKQ}$$

where

$$a(i) = x(i) - x(j) \quad b(i) = y(i) - y(j) \quad i = 1,2,3,4$$

$$j = 2,3,4,1$$

$$c(i) = \sum_{i=1}^4 (a(i)^2) - \sum_{i=1}^4 (b(i)^2)$$

$$\text{SUM}(a) = \sum_{i=1}^4 a(i) \quad \text{SUM}(b) = \sum_{i=1}^4 b(i)$$

$$d(1) = 2[b(1) + b(4)] - \text{SUM}(b) \quad d(2) = 2[a(1) + a(4)] - \text{SUM}(a)$$

$$d(3) = 2[b(1) + b(2)] - \text{SUM}(b) \quad d(4) = 2[a(1) + a(2)] - \text{SUM}(a)$$

$$\text{Delta} = 0.125 / \{ [(a(3) - a(1)) [b(4) - b(2)] - [a(4) - a(2)] [b(3) - b(1)] \}$$

$$\begin{aligned} T(9, 1) &= 1.50*(a(1))/c(1) \\ T(9, 2) &= -0.75*(b(1)*a(1))/c(1) \\ T(9, 3) &= -0.25*(2.0*b(1)**2 - a(1)**2)/c(1) \\ T(9, 4) &= -1.50*(a(1))/c(1) \\ T(9, 5) &= -0.75*(b(1)*a(1))/c(1) \\ T(9, 6) &= -0.25*(2.0*b(1)**2 - a(1)**2)/c(1) \\ T(10, 1) &= 1.50*(b(1))/c(1) \\ T(10, 2) &= 0.25*(2.0*a(1)**2 - b(1)**2)/c(1) \\ T(10, 3) &= 0.75*(b(1)*a(1))/c(1) \\ T(10, 4) &= -1.50*(b(1))/c(1) \\ T(10, 5) &= 0.25*(2.0*a(1)**2 - b(1)**2)/c(1) \\ T(10, 6) &= 0.75*(a(1)*b(1))/c(1) \\ T(11, 4) &= 1.50*(a(2))/c(2) \\ T(11, 5) &= -0.75*(b(2)*a(2))/c(2) \\ T(11, 6) &= -0.25*(2.0*b(2)**2 - a(2)**2)/c(2) \\ T(11, 7) &= -1.50*(a(2))/c(2) \\ T(11, 8) &= -0.75*(b(2)*a(2))/c(2) \\ T(11, 9) &= -0.25*(2.0*b(2)**2 - a(2)**2)/c(2) \\ T(12, 5) &= 0.25*(2.0*a(2)**2 - b(2)**2)/c(2) \\ T(12, 4) &= 1.50*(b(2))/c(2) \\ T(12, 6) &= 0.75*(b(2)*a(2))/c(2) \\ T(12, 7) &= -1.50*(b(2))/c(2) \\ T(12, 8) &= 0.25*(2.0*a(2)**2 - b(2)**2)/c(2) \\ T(12, 9) &= 0.75*(a(2)*b(2))/c(2) \\ T(13, 7) &= 1.50*(a(3))/c(3) \\ T(13, 8) &= -0.75*(b(3)*a(3))/c(3) \\ T(13, 9) &= -0.25*(2.0*b(3)**2 - a(3)**2)/c(3) \\ T(13,10) &= -1.50*(a(3))/c(3) \\ T(13,11) &= -0.75*(b(3)*a(3))/c(3) \\ T(13,12) &= -0.25*(2.0*b(3)**2 - a(3)**2)/c(3) \\ T(14, 7) &= 1.50*(b(3))/c(3) \end{aligned}$$

$$\begin{aligned}
T(14, 8) &= 0.25*(2.0*a(3)**2 - b(3)**2)/c(3) \\
T(14, 9) &= 0.75*(b(3) * a(3))/c(3) \\
T(14,10) &= - 1.50*(b(3))/c(3) \\
T(14,11) &= 0.25*(2.0*a(3)**2-b(3)**2)/c(3) \\
T(14,12) &= 0.75*(a(3) * b(3))/c(3) \\
T(15, 1) &= - 1.50*(a(4))/c(4) \\
T(15, 2) &= - 0.75*(b(4) * a(4))/c(4) \\
T(15, 3) &= - 0.25*(2.0*b(4)**2-a(4)**2)/c(4) \\
T(15,10) &= 1.50*(a(4))/c(4) \\
T(15,11) &= - 0.75*(b(4) * a(4))/c(4) \\
T(15,12) &= - 0.25*(2.0*b(4)**2-a(4)**2)/c(4) \\
T(16, 1) &= - 1.50*(b(4))/c(4) \\
T(16, 2) &= 0.25*(2.0*a(4)**2-b(4)**2)/c(4) \\
T(16, 3) &= 0.75*(b(4) * a(4))/c(4) \\
T(16,10) &= 1.50*(b(4))/c(4) \\
T(16,11) &= 0.25*(2.0*a(4)**2 - b(4)**2)/c(4) \\
T(16,12) &= 0.75*(a(4) * b(4))/c(4) \\
T(17, 1) &= - 12.0*Delta*d(1) \\
T(17, 2) &= Delta*d(1)*(2.0*b(1) - 2.0*b(4) + SUM(b)) \\
T(17, 3) &= -Delta*d(1)*(2.0*a(1) - 2.0*a(4) + SUM(a)) \\
T(17, 4) &= - 12.0*Delta*d(3) \\
T(17, 5) &= - Delta*d(3)*(2.0*b(1) - 2.0*b(2) + SUM(b)) \\
T(17, 6) &= Delta*d(3)*(2.0*a(1) - 2.0*a(2) + SUM(a)) \\
T(17, 7) &= 12.0*Delta*d(1) \\
T(17, 8) &= Delta*d(1)*(2.0*b(2) - 2.0*b(3) - SUM(b)) \\
T(17, 9) &= - Delta*d(1)*(2.0*a(2) - 2.0*a(3) - SUM(a)) \\
T(17,10) &= 12.0*Delta*d(3) \\
T(17,11) &= - Delta*d(3)*(2.0*b(4) - 2.0*b(3) - SUM(b)) \\
T(17,12) &= Delta*d(3)*(2.0*a(4) - 2.0*a(3) - SUM(a)) \\
T(18, 1) &= 12.0*Delta*d(2) \\
T(18, 2) &= - Delta*d(2)*(2.0*b(1) - 2.0*b(4) + SUM(b)) \\
T(18, 3) &= Delta*d(2)*(2.0*a(1) - 2.0*a(4) + SUM(a)); \\
T(18, 4) &= 12.0*Delta*d(4) \\
T(18, 5) &= Delta*d(4)*(2.0*b(1) - 2.0*b(2) + SUM(b)) \\
T(18, 6) &= - Delta*d(4)*(2.0*a(1) - 2.0*a(2) + SUM(a)) \\
T(18, 7) &= - 12.0*Delta*d(2) \\
T(18, 8) &= - Delta*d(2)*(2.0*b(2) - 2.0*b(3) - SUM(b)) \\
T(18, 9) &= Delta*d(2)*(2.0*a(2) - 2.0*a(3) - SUM(a)) \\
T(18,10) &= - 12.0*Delta*d(4) \\
T(18,11) &= Delta*d(4)*(2.0*b(4) - 2.0*b(3) - SUM(b)) \\
T(18,12) &= - Delta*d(4)*(2.0*a(4) - 2.0*a(3) - SUM(a))
\end{aligned}$$

APPENDIX III

Patch test and Benchmark Problems for IDKQ

1. The Patch Test

BC's: $w = 0$ at nodes 1, 2, 4

Loading:

$M_x = b$ at nodes 2, 3

$M_x = -b$ at nodes 1, 4

$M_y = a$ at nodes 1, 2

$M_y = -a$ at nodes 2, 4

$P_z = -2$ at node 3

Analysis Results:

$M_x = M_y = M_{xy} = 1$

For:

$a = 20, b = 10, h = 1,$

$E = 1000$

$w_3 = 12.60, \nu = 0.3$

$w_3 = 9.69, \nu = 0.0$

Figure III.1: Patch Test Problem

The usual test invoked for the evaluation of low order plate bending elements was introduced by Robinson [66] and is shown in Figure III.1. This simple test serves as a necessary and sufficient condition for correct convergence assessment of the new IDKQ finite element. The suggested boundary and load conditions lead theoretically to a uniform constant state of stress in the rectangular plate. For different positions of the internal nodes 3, 4, 5, 6 and various values of Poisson's ratio, the stress resultants M_x , M_y , and M_{xy} display a constant value equal to one everywhere in the plate. The accuracy of the results may be affected more or less by the element distortions.

2. Evaluation of displacements and stresses in square plates

Table III.1
Center Deflections of Square Plates
($w_{exact} = 1.00000$, $\nu = 0.3$)

Square Plate	Uniform Load		Concentrated Load	
Mesh Size	Simply Supported	Clamped	Simply Supported	Clamped
$N = 1$	0.80256	1.23519	1.11187	1.09695
$N = 2$	0.94695	1.06064	1.02348	1.02689
$N = 4$	0.92487	1.02803	1.00769	1.01151
$N = 8$	1.00775	1.01931	1.00207	1.00314
$N = 16$	1.00958	1.01525	1.00029	1.00017
Timoshenko [15]	$w_{max} = 0.00406 \frac{ql^4}{D}$	$w_{max} = 0.00126 \frac{ql^4}{D}$	$w_{max} = 0.0116 \frac{Pl^2}{D}$	$w_{max} = 0.0056 \frac{Pl^2}{D}$

3. Influence of Mesh Distortions.

Mesh		Difference in w_c of the irregular mesh compared the regular mesh (%)
Regular	Irregular	
		Simply Supported Plate - Concentrated Central Load 0.24 % IDKQ - Uniform Load 0.31 % IDKQ 1.05 % DKQ
		Clamped Plate - Concentrated Central Load 0.14 % IDKQ 0.17 % DKQ - Uniform Load 0.43 % IDKQ

Figure III.2: Influence of Mesh Distortions on IDKQ compared to DKQ [65]

4. Skew Plate

A difficult case test is represented by the simply supported rhombic plate as shown in Figure III.3 . At the obtuse corners, the bending moments are theoretically infinite [67]. Many finite elements fail to show that M_x and M_y are of opposite sign near these corners [18]. The IDKQ element is tested for simply supported ($w = 0$) and fully clamped ($w = \theta_x = \theta_y = 0$) boundary conditions on all edges. Good results were obtained on regular meshes of 4×4 and 8×8 sizes for a constant skew angle $\beta = 30^\circ$. However, slightly more accurate results were obtained on an improved mesh shown in Figure III.3. The stress resultants at the nodes of interest were obtained averaging the stresses computed at the centroids of the elements surrounding the node. The stresses at each element centroid were obtained by means of a bilinear stress interpolation of the stresses obtained at the Gauss stations surrounding the node. The thickness to span ratio $2a$ is taken as 0.01. Because of the DK assumptions, the simply supported boundary condition $w = 0$ requires $w_{,x} = 0$ as well, according to the classical thin-plate theory. These boundary conditions can be easily employed for rectangular plates. However, for skew plates accurate results are obtained only by imposing the finite element simply supported boundary conditions $w = \theta_x = \theta_y = 0$. The plates are tested for two different load cases represented by a central point load and a uniform load. The results are shown in Tables III.2.1, III.2.2, III.3.1, III.3.2. The values obtained with IDKQ element are in very good agreement with the Morley's [67] analytical solutions and Sengupta's Mindlin triangular element having 35 DOF's [68]. Comparison with the Zienkiewicz's [68] plate element based on a mixed formulation with three variables w , θ and S is made.

Figure III.3: Skew Plate with Equal Side Lengths and Skew Angle β .

TABLE III.2.1. Skew Plate with all edges simply supported ($w = 0$) under uniform loading.

Element	$\frac{10^2 \times (w_c)}{qa^4 / D}$	$\frac{10 \times (M_x^c)}{qa^2}$	$\frac{10 \times (M_y^c)}{qa^2}$
Morley [16]	0.6528	0.7640	0.4320
Zienkiewicz [19]	0.6841	0.7853	0.4483
IDKQ	0.6771	0.7568	0.4441

TABLE III.2.2. Skew Plate with all edges simply supported ($w = 0$) under central point load.

Element	$\frac{10^2 \times (w_c)}{Pa^2 / D}$	$\frac{10 \times (M_x^c)}{P}$	$\frac{10 \times (M_y^c)}{P}$
Sengupta [17]	1.4092	3.2360	2.6293
Zienkiewicz [19]	1.4306	3.3612	2.6178
IDKQ	1.4209	3.1690	2.7605

TABLE III.3.1. Clamped Skew Plate ($w = \theta_x = \theta_y = 0$) under uniform loading.

Element	$\frac{10^2 \times (w_c)}{qa^4 / D}$	$\frac{10 \times (M_x^c)}{qa^2}$	$\frac{10 \times (M_y^c)}{qa^2}$
Sengupta [17]	0.1743	3.2089	1.8205
Zienkiewicz [19]	0.1737	3.1772	1.7479
IDKQ	0.1746	3.1527	1.8983

TABLE III.3.2. Clamped Skew Plate ($w = \theta_x = \theta_y = 0$) under central point load.

Element	$\frac{10^2 \times (w_c)}{Pa^2 / D}$	$\frac{10 \times (M_x^c)}{P}$	$\frac{10 \times (M_y^c)}{P}$
Sengupta [17]	0.6282	2.5981	2.1130
Zienkiewicz [19]	0.6403	2.6779	2.0820
IDKQ	0.6364	2.5146	2.2292

References

- [1] Turner, M.J., Dill, E.H., Martin, H.C. and Melosh, R.J., "Large Deflection of Structures Subjected to Heating and External Loads," J. Aerospace Sci., 27, 1960, pp. 97-106.
- [2] Argyris, J.H., Kesely, S. and Kamel, H., Matrix Methods of Structural Analysis, AGARDograph No. 72, Pergamon Press, 1964, pp. 105-120.
- [3] Gallager, R.H., Gellatly, R.A., Padlog, J. and Mallett, R.H., "A Discrete Element Procedure for Thin-Shell Instability Analysis," J. AIAA, 5 1, 1967, pp. 138-144.
- [4] Kapur, K.K. and Hartz, B.J., "Stability of Plates Using the Finite Element Method," Journal of the engineering Mechanics Division, ASCE, 92, EM2, 1966, pp. 177-195.
- [5] Marcal, P.V., "Effect of Initial Displacements on Problems of a Large Deflection and Stability," Division of Engineering, Brown University, Materials Research Program, Contract SD-86, 1967.
- [6] Green, B.E., Strome D.R. and Weikel, R.C., "Application of the Stiffness Method to the Analysis of Shell Structures," in Proc. Aviation Conf. Of American Society of Mechanical Engineers, Los Angeles, March 1961.
- [7] Gould, P.L. and Sen, S.K., "Refined mixed method finite elements for shells of revolution," in 3rd Conf. On Matrix Methods in Structural Mechanics, Wright-Patterson AF Base, Ohio, 1971.
- [8] Striklin, J.A., Navaratna, D.R. and Pian, T.H.H., "Improvements in the analysis of shells of revolution by matrix displacement method (curved elements)," JAIAA, 4 2069-72, 1966.
- [9] Razaqpur, A.G., Aziz, O., Nofal, M., "A new quadrilateral facet shell element," FEM in the Design Process, edited by John Robinson, ISBN 0 9507172 6 6, England, 1990.
- [10] Hellweg, H.B. and Crisfield, M.A., "A new arc-length method for handling sharp snap-baks," Computers&Structures, Vol. 66, No. 5, pp. 705-709, 1998.
- [11] Bergan, P.G. and Soreide, T., "Solution of large displacement and instability problems using the the current stiffness parameter," Finite Elements in Nonlinear Mechanics, pp. 647-669, Tapir Press 3, 1978.
- [12] Belytschko, T., Wong, B.L., Chiang, H.Y., "Improvements in low-order shell elements for explicit transient analysis," in Analytical and Computational Models of Shells, The American Society of Mechanical Engineers, San-Francisco, 1989.
- [13] Wood, R.D. and Zienkiewicz, O.C., "Geometrically nonlinear finite element analysis of beams, frames, arches and axisymmetric shells," Computers&Structures, Vol. 7, pp. 725-735, Pergamon Press, 1977.
- [14] J. L. Batoz, M. B. Tahar, Evaluation of a new quadrilateral thin plate bending element, IJNME, Vol. 18, pp. 1655-1677, 1982

- [15] Chinniah, J., Ph.D. Thesis, Finite Element Formulation for Thin Plate and Shallow Structures, Dept. of Mech. and Aeron. Engng., Carleton University, Ottawa, 1985.
- [16] Crisfield, M. A., Non-linear Finite Element Analysis of Solids and Structures, Vol. 1, Essentials, ISBN 0-471-92956-5, John Wiley & Sons Ltd, 1991
- [17] Crisfield, M. A., Finite Elements and Solution Procedures for Structural Analysis, ISBN 0-906674-53-0, Pineridge, Swansea, UK, 1986
- [18] Cook, R. D., Malkus, D. S., Plesha, M. E., Concepts and Applications of Finite Element Analysis, ISBN 0-471-84788-7, John Wiley & Sons, Inc., 1989.
- [19] O. C. Zienkiewicz, R. L. Taylor, The Finite Element Method, Vol.1 and Vol. 2, ISBN 0-07-084175-6, McGraw-Hill, 1988.
- [20] K. J. Bathe, Finite Element Procedures in Engineering Analysis, ISBN 0-13-317305, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, 1982.
- [21] C. Taylor, D. R. G. Owen, E. Hinton, Computational Methods for Non-Linear Problems, ISBN 0-906674-52-2, Pineridge Press Limited, 1987.
- [22] Y. C. Fung, Foundations of Solid Mechanics, ISBN 0-13-329912-0, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1965.
- [23] W. K. Liu, T. Belytschko, K. C. Park, Proceedings of the International Conference on Innovative Methods for Nonlinear Problems, ISBN 0-906674-41-7, Pineridge Press International Limited, 1984.
- [24] E. Hinton, D. R. G. Owen, G. Krause, Finite Elemente Programme fuer Platten und Schalen, ISBN 3-540-51546-1, Springer Verlag Berlin, 1989, (German Edition).
- [25] S. Timoshenko, S. Woinowsky-Krieger, Theory of Plates and Shells (second edition), ISBN 0-07-064779-8, McGraw-Hill Book Company, Inc., 1959.
- [26] M. Nofal, M.Eng. Thesis, Inelastic load distribution of slab-on-steel girder bridges: An analytical and experimental study, Dept. of Civil and Environmental Engineering, Carleton University, Ottawa, 1988.
- [27] M. Nofal, Ph.D. Thesis, Continuum Damage Mechanics for Plain, Fibre-Reinforced, and Reinforced Concrete Materials and Structures, Dept. of Civil and Environmental Engineering, Carleton University, Ottawa, 1997.
- [28] D. Murray, Ph.D. Thesis, Large Deflection Analysis of Plates, Dept. of Civil Engng., University of California, Berkeley, 1967.
- [29] K. J. Bathe, H. Ozdemir, E. L. Wilson, Static and Dynamic Geometric and Material Nonlinear Analysis, Report No. UCSESM 74-4, Structural Engineering Laboratory, University of California at Berkeley, 1974.
- [30] R. de Borst, P. Nauta, Non-orthogonal cracks in a smeared finite element model, Eng. Comput., Vol. 2, March, 1985.
- [31] S.W. Hopkins, Standardization Improves the Structural Integrity, ASTM Standardization News, October, 1998.
- [32] J. Van Greunen, Nonlinear Geometric, Material and Time Dependent Analysis of Reinforced and Prestressed Concrete Slabs and Panels, Report No. UC SESM 79-3, University of California at Berkeley, 1979.

- [33] M. J. Clark, G. J. Hancock, A Study of Incremental-Iterative Strategies for Non-Linear Analyses, *Int. J. Num. Meth. Engng.*, Vol. 29, pp 1365-1391, 1990.
- [34] E. Riks, An Incremental Approach to the Solution of Snapping and Buckling Problem, Paper presented at the 14th Int. Cong. Theoretical and Applied Mechanics, Delft, The Netherlands, 30 August – 4 September 1976.
- [35] Gonheim, G.A.M., Ph.D. Thesis, Nonlinear Finite Element Analysis of Concrete Structures, Dept. of Civ. Engrg., University of Calgary, Calgary, Alta., 249, 1978.
- [36] J.A. Stricklin, W.E. Heisler and W. Von Rieseaman, Evaluation of solution procedures for material and/or geometrically nonlinear structural analysis by the direct stiffness method, *AIAA/ASME 13th Structures, Structural Dynamics and Materials Conf.*, san Antonio, Texas, AIAA Paper No. 72-353 (1972).
- [37] J. L. Batoz, G. Dhatt, Incremental Displacement Algorithms for Nonlinear Problems, *Int. J. Num. Mech. Engng.* 14, 1262-1266 (1979).
- [38] P. Sharifi, E.P. Popov, Non-linear buckling analysis of sandwich arches, *J. Engng. Mech. Div. ASCE'97*, 1397-1312 (1971).
- [39] Ramm, E., Strategies for tracing the non-linear response near limit-points, *Non-linear Finite Element Analysis in Structural Mechanics*, ed. W. Wunderlich, Springer-Verlag, Berlin, pp 63-89 (1981).
- [40] M.A., Crisfield, A Fast Incremental/Iterative Solutions Procedure that Handels "Snap-Through", *Computers & Structures*, Vol. 13, pp. 55-62, 1981.
- [41] M.A., Crisfield, Accelerated solution techniques and concrete cracking, *Comp. Meth. in Appl. Mech. & Engng.*, 33, 585-607 (1982).
- [42] Bergan, P. G., Solution algorithms for non-linear structural problems, *Engineering Applications of the Finite Element Method*, *Computas*, 13.1-13.39 (1979); also *Computers & Structures*, 12, 497-509 (1980).
- [43] Den Heijder, C., Rheinboldt, W.C., On step-length algorithms for a class of continuations methods, *SIAM J. Num. Analysis*, 18, 925-948 (1981).
- [44] R. W., Clough, "The Finite Element in Plane Stress Analysis," *Proceedings, 2nd A.S.C.E. Conference on Electronic Computation*, Pittsburgh, Pa., Sept., 1960.
- [45] R. W., Clough, and J. L. Tocher, "Finite Element Stiffness Matrices for the Analysis of Plate Bending," *Proceedings, Conference on Matrix Methods in Structural Mechanics*, Wright-Patterson A.F.B., Ohio, 1965.
- [46] Razaqpur, A. G. and Nofal, M., "Performance of a new Quadrilateral Finite Element with Rotational Degrees of Freedom," *Proceedings of the Fifth World Congress on Finite Element Methods*, Salzburg, Austria, 1987.
- [47] T. Soreide, "Collapse Behavior of Stiffened Plates Using Alternative Finite Element Formulations", Report No. 77-3, Division of Structural Mechanics, The Norwegian Institute of Technology, The University of Trondheim, March 1977.
- [48] O. Aziz, M.Eng. Thesis, A mechanism free plane quadrilateral element with rotational degrees of freedom and the associated facet shell element, Dept. of Civil and Environmental Engineering, Carleton University, Ottawa, 1988.

- [49] O. C. Zienkiewicz, G. C. Nayak, "A general approach to problems of plasticity and large deformation using isoparametric elements," Proceedings of the Third Conference on Matrix Methods in Structural Mechanics, AFFDL-TR-71-160, Wright-Patterson Air Force Base, Ohio, 19-21 October, 1971.
- [50] S. P. Timoshenko, J. M. Gere, Theory of Elastic Stability, 2nd Ed., McGraw-Hill, New York, 1961.
- [51] J. V. Huddleston, Finite deflection and snap through of high circular arches, J. Appl. Mech. 35, 763-769 (Dec. 1961)
- [52] D. A. Da Deppo and R. Schmidt, Side sway buckling of deep circular arches under a concentrated load, J. Appl. Mech., 36, 325-327, June 1969.
- [53] Thompson, J. M. T. & Hunt, G. W., Dangers of Structural Optimization, Engng. Optimization 1, 99-110, 1974.
- [54] W.T. Koiter and M. Skaloud, Interventions, pages 64-68, 103-104 in Comportement Postcritiques des Plaques Utilisees en Construction Metallique, Colloque International tenu a l'Universite de Liege, 5 eme serie, tome VIII, fascicule 5.
- [55] Bergan, P.G. and Felippa, C.A., A triangular membrane element with rotational degrees of freedom, Comput. Meth. Appl. Mech. Eng., 50, 25-69 (1985).
- [56] Allman, D.J., A compatible triangular element including vertex rotations for plane elasticity analysis, Comput. Struct., 19, 1-8 (1984).
- [57] Carpenter, N., Stolarsky, H. and Belytschko, T.A., A flat triangular shell element with improved membrane interpolation, Commun. Appl. Num. Meth., 1, 161-168 (1985).
- [58] J. Goldak, Finite Element Method I and II, Course notes (88.565, 88.566), Mechanical&Aerospace Dept., Carleton University, Ottawa, 1999-2000.
- [59] M.A. Crisfield, An Arc-Length Method Including Line Searches and Accelerations, *Int. J. Num. Meth. Engng.*, Vol. 19, 1269-1289, (1983).
- [60] T. Belytschko, B. Leong and H.Y. Chiang, Advances in one-point quadrature shell elements, *Comp. Meth. Appl. Mech. Engng.*, 96, 93-107 (1992).
- [61] H. Duddek, G. Griebenow and G. Shaper, Material and time dependent nonlinear behavior of cracked reinforced concrete slabs, in Nonlinear Behavior of Reinforced Concrete Spatial Structures, Vol. 1, Preliminary Report IASS Symposium held in Darmstadt, Eds. G. Mehlhorn, H. Ruhle and W. Zerna, Werner-Verlag Duesseldorf, pp. 101-113, July 1978.
- [62] Allman, D.J. (1988), "A Quadrilateral Finite Element Including Vertex Rotations for Plane Elasticity Analysis," *Int. J. Num. Meth. Engng.* Vol. 26, pp.717-730.
- [63] Robinson, J. and Sheng, L.D., "Some Single Element Testing of Cook's Quadrilateral Element with Rotational Degrees of Freedom," Finite Element News Issue No.1.
- [64] NONLACS (Non-Linear Analysis of Concrete and Steel Structures), User's Manual, Carleton University, Ottawa, 1988.
- [65] Batoz, J-L. and Tahar, M.B., Evaluation of a new quadrilateral thin plate bending element, *Int. J. Num. Meth. Eng.*, 18, 1655-1677 (1982).

- [66] J. Robinson, "Element evaluation. A set of assessment points and standard tests", Proc. F.E.M. in the Commercial Environment, vol. 1, pp. 217-248 (Oct. 1978).
- [67] L.S.D. Morley, *Skew plates and structures*. Pergamon Press, Oxford (1963).
- [68] D. Sengupta, Performance Study of a Simple Finite Element in the Analysis of Skew Rhombic Plates, *Computers&Structures*, Vol. 54, No. 6, pp. 1173-1182, 1995.
- [69] G. Hartley, Finite Element Analysis, Part III, Course notes (82.513*), Civil and Environmental Dept., Carleton University, Ottawa, 1999.