



Universidade
do Porto

Faculdade de
Engenharia

FEUP

The Natural Neighbour Radial Point Interpolation Method.

Solid Mechanics and Mechanobiology Applications.

by

Jorge Américo Oliveira Pinto Belinha

Thesis submitted to

Faculdade de Engenharia da Universidade do Porto

as a requirement to obtain the PhD degree in Mechanical Engineering.

under the supervision of

Professor Renato Manuel Natal Jorge

and

Professor Lúcia Maria de Jesus Simas Dinis

Porto, September 2010

Agradecimentos

Na vida, por vezes, tudo quanto precisamos é apenas de uma oportunidade. Cabe-nos reconhecê-la e agarrá-la com as duas mãos. Esta tese de doutoramento apenas foi possível porque duas pessoas me deram essa oportunidade.

Em primeiro lugar gostaria sinceramente de agradecer ao Professor Renato Manuel Natal Jorge, orientador. Obrigado por me ter permitido prosseguir o trabalho de investigação. Obrigado pela disponibilidade constante e por ter proporcionado as condições logísticas essenciais à concretização desta tese. Obrigado pela sua boa-disposição e pela sua confiança nas minhas capacidades.

Os meus mais profundos agradecimentos são dirigidos à Professora Lúcia Maria de Jesus Simas Dinis, co-orientadora, por me ter aberto a porta da investigação. Ainda me recordo, como se tivesse sido ontem, do dia e do local onde me encontrava quando a Professora Lúcia me comunicou que iria passar a fazer investigação com ela, lembro-me que por dentro explodia de alegria. Desde então, ao longo destes oito anos, a Professora Lúcia tem sido para mim uma tutora, uma conselheira, uma amiga. Agradeço-lhe a serenidade transmitida nos momentos mais difíceis, a sua total disponibilidade e o alento e motivação que desde sempre me incutiu. Obrigado pela presença e pela Amizade.

Gostaria de agradecer ao Professor António Joaquim Mendes Ferreira as oportunidades de crescimento académico que recorrentemente me ofereceu. Assim como ao Marco Parente, ao Pedro Martins, à Carla Roque e ao António Melro, colegas de investigação do departamento de Engenharia Mecânica da Faculdade de Engenharia da Universidade do Porto, a franca amizade e disponibilidade para a troca de ideias e saberes.

Ao Instituto de Engenharia Mecânica (IDMEC, Unidade de Conceção e Validação Experimental), na pessoa do Professor António Augusto Fernandes, bem como à Faculdade de Engenharia da Universidade do Porto, em particular ao Departamento de Engenharia Mecânica, na pessoa do Professor António Torres Marques, agradeço as condições e os meios informáticos dispensados. Estes agradecimentos são extensíveis à Sr.^a

Fernanda Grandão, à Sr.^a Fernanda Fonseca e à Sr.^a Júlia Meira, pela amabilidade e solicitude com que sempre me solucionaram diversos problemas de secretariado.

Cabe-me ainda referir a importância do apoio financeiro proporcionado pela Fundação para a Ciência e Tecnologia por intermédio de atribuição da bolsa de Doutoramento SFRH/BD/31121/2006.

Aos meus amigos e irmãos do *GASde69* (Grupo Académico de Serenatas de Espinho), muito obrigado por todas as sextas-feiras me recarregarem as baterias com malte.

Por fim, gostaria de agradecer à minha família o ânimo e o apoio. Em especial, ao meu tio agradeço o exemplo e à minha avó as lições de vida.

Rita, sem o teu encorajamento e incentivo constante esta caminhada teria sido muito mais difícil de cumprir, obrigado por estares sempre do meu lado.
<3***<3

Ao meu pai, à minha mãe e à minha irmã agradeço profundamente o apoio e as condições que desde sempre me proporcionaram, sem as quais este percurso de vida não teria sido possível. Muito obrigado.

The Natural Neighbour Radial Point Interpolation Method.

Solid Mechanics and Mechanobiology Applications.

by

Jorge Américo Oliveira Pinto Belinha

Thesis submitted in fulfilment of the degree of Doctor of Philosophy in
Mechanical Engineering in the Faculdade de Engenharia da Universidade
do Porto under the supervision of:

Professor **Renato Manuel Natal Jorge**

Associated Professor of Faculdade de Engenharia da Universidade do Porto

and

Professor **Lúcia Maria de Jesus Simas Dinis**

Associated Professor of Faculdade de Engenharia da Universidade do Porto

Abstract

This work presents and develops the Natural Neighbour Radial Point Interpolation Method (NNRPIM), a new truly meshless method. The NNRPIM is extended to several engineering fields, such as solid mechanics static and dynamic linear analysis and structural nonlinear analysis, and bone remodelling biomechanical analysis.

Within the NNRPIM the nodal connectivity is enforced using the Natural Neighbour concept. After the Voronoï diagram construction from the unstructured nodal mesh, which discretizes the problem domain, small cells are created, the “influence-cells”. These cells are in fact influence-domains entirely nodal dependent. The Delaunay triangles are used to create a node-depending background mesh used in the numerical integration of the NNRPIM interpolation functions. The NNRPIM interpolation functions, used in the Galerkin weak form, are constructed with the Radial Point Interpolators. In the construction of the NNRPIM interpolation functions no polynomial base is required, which is an innovation and the used Radial Basis Function (RBF) is the Multiquadric RBF. The NNRPIM interpolation functions posses the delta Kronecker property, which simplify the imposition of the natural and essential boundary conditions.

In the nonlinear analysis, large deformations and elastoplastic material behaviour are considered. The solution of the nonlinear equation system is obtained resorting to incremental/iterative methods, such as the Newton-Raphson method or the orthogonal actualized Ramm's method, this last permitting the analysis of structures that in some point evidence instability phenomena such as the "snap-through" and the "snap-back". The material nonlinear behaviour is dealt considering for the elastoplastic model the Von Mises yield function and the efficient "forward-Euler" procedure is used in order to return the stress to the yield surface.

In the biomechanical analysis, a new mathematical model to obtain the bone material properties is proposed. This material law is based in experimental data, which support the idea that the law governing the mechanical behaviour of the bone tissue is the same for cortical bone and trabecular bone. A bone remodelling algorithm is proposed, adapted from Carter's remodelling algorithm. It is based on the assumption that the adaptation of bony tissue responds mainly to mechanical stimulus. A simple forward Euler scheme is implemented, resulting in an iterative remodelling process.

After the respective exposition, several benchmark solid mechanics static and dynamic linear examples are solved, and well-known demanding nonlinear reference examples are analysed. In biomechanics applications, bone remodelling validation tests are made and then two important bone structures in the human body are studied. The obtained results indicate that NNRPIM is an accurate, flexible and reliable meshless method that can be used in several fields of solid mechanics and biomechanical analysis.

The Natural Neighbour Radial Point Interpolation Method.
Solid Mechanics and Mechanobiology Applications.

por

Jorge Américo Oliveira Pinto Belinha

Tese apresentada à Faculdade de Engenharia da Universidade do Porto,
para a obtenção do grau de Doutor em Engenharia Mecânica, sob a
orientação de:

Professor **Renato Manuel Natal Jorge**

Professor Associado da Faculdade de Engenharia da Universidade do Porto

e

Professor **Lúcia Maria de Jesus Simas Dinis**

Professora Associado da Faculdade de Engenharia da Universidade do Porto

Sumário

Neste trabalho é apresentado e desenvolvido o “Natural Neighbour Radial Point Interpolation Method” (NNRPIM), um método sem malha inovador. O NNRPIM é aplicado a vários domínios da engenharia. Na mecânica dos sólidos a análise linear estática e dinâmica é abordada, tal como a análise não-linear estrutural. Na biomecânica, o NNRPIM é aplicado na remodelação do tecido ósseo.

No NNRPIM a conectividade nodal é imposta utilizando o conceito geométrico dos vizinhos naturais. Partindo da malha nodal que discretiza o domínio do problema, o diagrama de Voronoï é construído. Com base neste diagrama pequenas células são criadas, as denominadas células-de-influência, que na realidade são domínio-de-influência completamente dependentes da discretização nodal. A triangulação de Delaunay é usada para criar uma malha de fundo, também esta dependente somente da discretização nodal, que posteriormente é usada para efeitos de integração das funções de interpolação do NNRPIM. As referidas funções de interpolação, usadas na forma fraca de Galerkin, são construídas por intermédio dos Interpoladores Radiais Pontuais. As funções de interpolação do NNRPIM não requerem base polinomial e a função de base radial usada é

a multiquádrica. No NNRPIM a imposição das condições de fronteira, naturais e essenciais, é facilitada pois as funções de interpolação possuem a propriedade de delta Kronecker.

Na análise não-linear são consideradas as grandes deformações e materiais com comportamento elastoplástico. A solução do sistema de equações não-linear é obtida por intermédio de métodos incrementais e iterativos, como o método de Newton-Raphson ou o método ortogonal actualizado de Ramm, sendo este último um método do arco. O método de Ramm permite a análise de estruturas que em algum ponto do carregamento evidenciem fenómenos de instabilidade, tais como o fenómeno de “snap-through” e/ou de “snap-back”. O comportamento não-linear material é simulado considerando no modelo elastoplástico a função de cedência de Von-Mises e um processo iterativo de retorno das tensões à superfície de cedência.

Na biomecânica é proposto um novo modelo matemático para obter as propriedades materiais do tecido ósseo. A referida lei material é baseada em dados experimentais, que suportam a ideia que a lei que governa o comportamento mecânico do tecido ósseo é a mesma para o osso cortical e trabecular. Baseado no modelo de remodelação óssea de Carter, é proposto um algoritmo de remodelação do tecido ósseo. Este modelo apoia-se na suposição que a reestruturação óssea depende maioritariamente dos estímulos mecânicos. O modelo proposto obtém a solução pretendida por meio de um processo iterativo.

Vários exemplos da mecânica dos sólidos, referentes à análise linear estática e dinâmica, bem como relativos à análise não-linear geométrica e material, são analisados e resolvidos. Também o modelo de remodelação óssea proposto é validado. Duas das mais importantes estruturas ósseas no corpo humano são analisadas com sucesso. Os resultados obtidos indicam que o NNRPIM é um método numérico com um elevado nível de precisão e extremamente flexível, que poderá ser aplicado com sucesso a diversos campos da mecânica dos sólidos e da mecanobiologia.

Table of Contents

1	INTRODUCTION.....	1
1.1	MESHLESS METHODS.....	1
1.1.1	Approximation (or interpolation) functions.....	1
1.1.2	Formulation.....	2
1.1.3	Integration.....	2
1.1.4	Relevant Meshless Methods.....	3
1.2	NATURAL NEIGHBOUR RADIAL POINT INTERPOLATION METHOD	4
1.3	THESIS PURPOSE.....	5
1.4	THESIS ARRANGEMENT.....	6
2	MECHANICAL FUNDAMENTALS.....	9
2.1	CONTINUUM FORMULATION.....	9
2.1.1	Kinematics.....	10
2.1.1.1	Deformation gradient.....	11
2.1.1.2	Strain.....	12
2.1.1.3	Polar decomposition.....	13
2.1.1.4	Stress.....	13
2.1.1.5	Principal stress.....	14
2.1.2	Constitutive Equations.....	18
2.2	WEAK FORM	20
2.2.1	Galerkin Weak Form.....	21
2.3	DISCRETE SYSTEM EQUATION	24
2.3.1	Principle of virtual work.....	25
2.3.2	Stiffness matrix.....	28
2.3.3	Mass matrix.....	29
2.3.4	Force vector.....	30
2.3.5	Imposition of the boundary conditions	31
2.3.6	Dynamic equations.....	32
2.3.6.1	Forced Vibrations.....	35
2.4	ELASTOPLASTIC CONSTITUTIVE EQUATIONS	36
2.4.1	Yield Criterion	36
2.4.2	Hardening Rule.....	38
2.4.3	Plastic Flow	40
2.4.4	Elastoplastic small strain constitutive model.....	40
2.4.5	Stress returning algorithm.....	41
2.5	LINEAR SOLUTION ALGORITHMS.....	42
2.6	NONLINEAR SOLUTION ALGORITHMS.....	44
2.6.1	Newton Raphson Method.....	45
2.6.2	Arc-length method.....	48
2.6.2.1	Generic arc-length algorithm	50
3	THE MESHLESS METHOD.....	53
3.1	MESHLESS GENERIC PROCEDURE	53
3.2	NATURAL NEIGHBOURS	56
3.3	INFLUENCE-CELLS AND NODAL CONNECTIVITY	59
3.4	NUMERICAL INTEGRATION IN THE NNRPIM	60
3.4.1	2D integration scheme	61
3.4.1.1	Integration Scheme – Order 0.....	62
3.4.1.2	Integration Scheme – Order 1.....	63
3.4.1.3	Integration Scheme – Order k.....	63

3.4.1.4	<i>Nodal Integration Scheme</i>	64
3.4.2	<i>3D integration scheme</i>	64
3.4.3	<i>3D Shell-Like integration scheme</i>	65
3.5	RADIAL POINT INTERPOLATORS.....	67
3.5.1	<i>Properties of the RPI function</i>	70
3.5.2	<i>RPI function shape</i>	71
3.6	NNRPIM LINEAR FLOW-CHART	75
3.7	RPI FUNCTION PATCH TEST	76
3.7.1	<i>RPIM patch test</i>	76
3.7.2	<i>NNRPIM patch test</i>	80
3.8	NNRPIM SHELL-LIKE FORMULATION – OPTIMIZATION	87
3.9	CONCLUSIONS AND REMARKS	91
4	<i>LINEAR APPLICATIONS</i>	93
4.1	STATIC ANALYSIS.....	93
4.1.1	<i>The Unit Square Plate under Uniaxial Stress</i>	93
4.1.2	<i>The Cantilever Beam</i>	97
4.1.3	<i>Infinite Plate with a Circular Hole</i>	101
4.1.4	<i>Plates subjected to Transversal Loads Convergence Study</i>	104
4.1.5	<i>Scordelis-Lo Cylindrical Roof</i>	107
4.1.6	<i>Cylindrical Shell</i>	109
4.1.7	<i>Orthotropic Plate and Shell Examples</i>	110
4.1.7.1	<i>Square Symmetric Laminates</i>	111
4.1.7.2	<i>Square Anti-Symmetric Laminates</i>	114
4.1.7.3	<i>Long Cylindrical Laminated Shell</i>	116
4.1.7.4	<i>Simply Supported Cylindrical Laminated Shell</i>	117
4.2	DYNAMIC ANALYSIS.....	118
4.2.1	<i>Cantilever Beam</i>	119
4.2.2	<i>Variable Cross Section Beams</i>	120
4.2.3	<i>Shear-Wall</i>	122
4.2.4	<i>Free Vibrations of Plates</i>	123
4.2.5	<i>Free Vibration of Shells</i>	129
4.2.6	<i>Plates Transient Response</i>	132
4.2.7	<i>Shells Transient Response</i>	135
4.3	CONCLUSIONS AND REMARKS	137
5	<i>NONLINEAR ANALYSIS</i>	139
5.1	MATERIAL NONLINEARITY.....	139
5.1.1	<i>Cantilever Beam</i>	139
5.1.2	<i>Cook Membrane</i>	141
5.1.3	<i>Infinite Plate With a Circular Hole</i>	145
5.1.4	<i>Infinite Cylinder</i>	147
5.1.5	<i>Block Under Concentrated Pressure</i>	148
5.1.6	<i>Square Plate Under Transversal Load</i>	149
5.2	LARGE DEFORMATION ANALYSIS.....	151
5.2.1	<i>Clamped Thick Arch</i>	151
5.2.2	<i>Simply Supported Shallow Arch</i>	155
5.2.3	<i>Pure Bending of a Cantilever Beam</i>	157
5.2.4	<i>Cantilever Beam</i>	159
5.2.5	<i>Square Plate Under Transversal Load</i>	162
5.2.6	<i>Curved Cantilever Beam</i>	165
5.3	GEOMETRIC AND MATERIAL NONLINEARITY	170
5.4	CONCLUSIONS AND REMARKS	174
6	<i>BONE TISSUE</i>	175
6.1	BONE BIOLOGY: BASIC CONCEPTS.....	175
6.1.1	<i>Bone Morphology</i>	176
6.1.2	<i>Composition and Structure of the Bone</i>	179

6.1.2.1	<i>Bone matrix.....</i>	179
6.1.2.2	<i>Bone cells.....</i>	180
6.1.2.3	<i>Reticular and Lamellar Bone.....</i>	181
6.1.3	<i>Cortical and Trabecular Bone.....</i>	182
6.1.3.1	<i>Cortical Bone.....</i>	182
6.1.3.2	<i>Cortical Bone Remodelling.....</i>	183
6.1.3.3	<i>Trabecular Bone.....</i>	184
6.1.3.4	<i>Trabecular Bone Remodelling.....</i>	184
6.2	BONE TISSUE MECHANICAL PROPERTIES	185
6.2.1	<i>Lotz material law</i>	186
6.2.2	<i>Proposed material law.....</i>	187
6.3	BONE REMODELLING ALGORITHMS.....	191
6.3.1	<i>Pauwels's Model.....</i>	192
6.3.2	<i>Cowin's Model.....</i>	192
6.3.3	<i>Carter's Model.....</i>	193
6.3.4	<i>Rodrigues' Models</i>	196
6.3.5	<i>Proposed Adaptation of Carter's Model Models.....</i>	197
7	<i>MECHANOBIOLOGY EXAMPLES.....</i>	203
7.1	<i>THE PLATE MODEL.....</i>	203
7.2	<i>THE CUBIC MODEL.....</i>	212
7.3	<i>THE CALCANEUS BONE.....</i>	221
7.4	<i>THE FEMUR BONE</i>	228
7.4.1	<i>The Femoral Two-Dimensional Analysis.....</i>	228
7.4.2	<i>The Femoral Three-Dimensional Analysis.....</i>	233
7.5	<i>CONCLUSIONS AND REMARKS</i>	237
8	<i>CONCLUSIONS.....</i>	239
8.1	<i>CONCLUSIONS AND REMARKS</i>	239
8.2	<i>FUTURE DEVELOPMENTS.....</i>	241
8.3	<i>EMBRACING THE NNRPIM</i>	243
9	<i>REFERENCES.....</i>	245

List of Figures

FIGURE 1 – GENERAL MOTION OF A DEFORMABLE BODY.	10
FIGURE 2 – (A) THREE-DIMENSIONAL STRESS COMPONENTS. (B) STRESS VECTOR ACTING ON A PLANE WITH NORMAL VECTOR N	16
FIGURE 3 – STRESS TENSOR TRANSFORMATION AND RESPECTIVE ANGLES.	16
FIGURE 4 – PROJECTION OF VECTOR N IN THE COORDINATE AXIS AND IN THE OXY PLANE.	19
FIGURE 5 – CONTINUOUS SOLID SUBJECT TO VOLUME FORCES AND EXTERNAL FORCES.	21
FIGURE 6 – ESSENTIAL BOUNDARY CONDITION NON ALIGNED WITH THE GLOBAL AXIS.	32
FIGURE 7 – HARDENING RULE GEOMETRIC REPRESENTATION. (A) ISOTROPIC HARDENING. (B) KINEMATIC HARDENING. (C) STRAIN HARDENING MODELS MIXTURE.	38
FIGURE 8 – ORTHOGONALITY CONDITION IN A STRESS TWO-DIMENSIONAL SPACE.	39
FIGURE 9 – INCREMENTAL STRESS CHANGES AT A POINT IN AN ELASTOPLASTIC CONTINUUM AT INITIAL YIELD.	42
FIGURE 10 – (A) GAUSS ELIMINATION ALGORITHM FOR MATLAB. (B) JACOBI ELIMINATION ALGORITHM FOR MATLAB.	44
FIGURE 11 – LOAD/DISPLACEMENT DIAGRAM SHOWING INSTABILITIES PHENOMENA.	45
FIGURE 12 – NEWTON-RAPSON METHOD FOR AN ONE-DIMENSIONAL SYSTEM.	47
FIGURE 13 – MODIFIED NEWTON-RAPSON METHOD (A) K_0 AND (B) K_1	48
FIGURE 14 – ORTHOGONAL ACTUALIZED RAMM'S METHOD FOR AN UNIDIMENSIONAL SYSTEM.	49
FIGURE 15 – NODAL DISCRETIZATION EXAMPLE.	54
FIGURE 16 – (A) FITTED GAUSSIAN INTEGRATION MESH. (B) GENERAL GAUSSIAN INTEGRATION MESH. (C) VORONOI DIAGRAM FOR NODAL INTEGRATION.	54
FIGURE 17 – (A) INFLUENCE DOMAINS WITH DIFFERENT SIZES AND SHAPES. (B) EXAMPLE OF A BAD CHOICE IN THE SIZE OF THE INFLUENCE DOMAIN, INTEREST POINT X_1 HAS MUCH MORE NODES INSIDE THAN INTEREST POINT X_2	55
FIGURE 18 – (A) INITIAL NODAL SET OF POTENTIAL NEIGHBOUR NODES. (B) FINAL CELL CONTAINING ONLY NEIGHBOUR NODES. (C) VORONOI CELL. (D) VORONOI DIAGRAM.	57
FIGURE 19 – 3D VORONOI CELL.	58
FIGURE 20 – (A) INITIAL VORONOI DIAGRAM. (B) DELAUNAY TRIANGULATION. (C) NATURAL NEIGHBOUR CIRCUMCIRCLES.	59
FIGURE 21 – (A) FIRST DEGREE INFLUENCE-CELL. (B) SECOND DEGREE INFLUENCE-CELL.	60
FIGURE 22 – (A) VORONOI CELL AND THE RESPECTIVE P_{II} INTERSECTION POINTS. (B) MIDDLE POINTS M_{II} AND THE RESPECTIVE GENERATED QUADRILATERALS. (C) QUADRILATERAL $\overline{M_{I3} P_{I4} M_{I4} n_I}$	61
FIGURE 23 – (A) VORONOI CELL AND THE RESPECTIVE P_{II} INTERSECTION POINTS. (B) MIDDLE POINTS M_{II} AND THE RESPECTIVE GENERATED TRIANGLES. (C) TRIANGLE $\overline{M_{II} M_{II} n_I}$	62
FIGURE 24 – (A) TRIANGULAR SHAPE AND (B) QUADRILATERAL SHAPE AND THE RESPECTIVE INTEGRATION POINTS X_I	62
FIGURE 25 – (A) PARTITION OF THE INITIAL TRIANGULAR SHAPE INTO QUADRILATERALS AND (B) PARTITION OF THE INITIAL QUADRILATERAL SHAPE INTO SMALL QUADRILATERALS WITH THE RESPECTIVE INTEGRATION POINTS X_I	63
FIGURE 26 – (A) INITIAL QUADRILATERAL. (B) TRANSFORMATION OF THE INITIAL QUADRILATERAL INTO AN ISOPARAMETRIC SQUARE SHAPE AND APPLICATION OF THE 6X6 QUADRATURE POINT RULE. (C) RETURN TO THE INITIAL QUADRILATERAL SHAPE.	64
FIGURE 27 – (A) VORONOI CELL AND (B) RESPECTIVE TETRAHEDRONS. (C) INITIAL TETRAHEDRON. (D) INITIAL TETRAHEDRON ISOPARAMETERIZATION AND DETERMINATION OF THE QUADRATURE INTEGRATION POINTS. (E) QUADRATURE INTEGRATION POINTS IN CARTESIANS COORDINATES.	65

FIGURE 28 – PROCEDURE OF THE SHELL-LIKE NODAL AND INTEGRATION MESH CONSTRUCTION AND DISTRIBUTION.	66
FIGURE 29 – (A) NODAL MESH USED. (B) RPI FUNCTION ISOMETRIC VIEW. (C) RPI FUNCTION Oxz VIEW. ($c=0.1$)	72
FIGURE 30 – (A) NODAL MESH USED. (B) RPI FUNCTION ISOMETRIC VIEW. (C) RPI FUNCTION Oxz VIEW. ($c=5$).....	72
FIGURE 31 – (A) NODAL MESH USED. (B) RPI FUNCTION ISOMETRIC VIEW. (C) RPI FUNCTION Oxz VIEW. ($c=20$).....	72
FIGURE 32 – (A) NODAL MESH USED. (D) $\phi(x)$ ISOMETRIC VIEW.....	73
FIGURE 33 – (A) NODAL MESH USED. (D) $\phi_{,x}(x)$ ISOMETRIC VIEW.	73
FIGURE 34 – (A) NODAL MESH USED. (D) $\phi_{,y}(x)$ ISOMETRIC VIEW.	74
FIGURE 35 – (A) NODAL MESH USED. (D) $\phi_{,xx}(x)$ ISOMETRIC VIEW.....	74
FIGURE 36 – (A) NODAL MESH USED. (D) $\phi_{,xy}(x)$ ISOMETRIC VIEW.....	74
FIGURE 37 – (A) NODAL MESH USED. (D) $\phi_{,yy}(x)$ ISOMETRIC VIEW.....	75
FIGURE 38 – (A) CONSTRUCTION EXAMPLES OF THE INFLUENCE DOMAIN. (B) IRREGULAR MESH AND (C) REGULAR MESH USED TO DISCRETIZE THE PROBLEM (“o” = NODES, ”+” = GAUSS POINTS).	77
FIGURE 39 – (A) SHAPE PARAMETER γ EFFECT ON THE ACCURACY AND (B) PARAMETER p EFFECT ON THE ACCURACY FOR THE REGULAR AND IRREGULAR MESH. LOGARITHMIC SCALES.	79
FIGURE 40 – SHAPE PARAMETER γ EFFECTS ON THE ACCURACY USING (A) THE FORMULATION $v1i0p0$, (B) THE FORMULATION $v1i0p1$ AND (C) THE FORMULATION $v1i0p3$. LOGARITHMIC SCALES.	81
FIGURE 41 – SHAPE PARAMETER p EFFECTS ON THE ACCURACY USING (A) THE FORMULATION $v1i0p0$, (B) THE FORMULATION $v1i0p1$ AND (C) THE FORMULATION $v1i0p3$. LOGARITHMIC SCALES.	82
FIGURE 42 – SHAPE PARAMETER γ EFFECTS ON THE ACCURACY USING (A) THE FORMULATION $v2i0p0$, (B) THE FORMULATION $v2i0p1$, (C) THE FORMULATION $v2i0p3$ AND (D) THE FORMULATION $v2i0p6$. LOGARITHMIC SCALES.	83
FIGURE 43 – SHAPE PARAMETER p EFFECTS ON THE ACCURACY USING (A) THE FORMULATION $v2i0p0$, (B) THE FORMULATION $v2i0p1$, (C) THE FORMULATION $v2i0p3$ AND (D) THE FORMULATION $v2i0p6$. LOGARITHMIC SCALES.	84
FIGURE 44 – INFLUENCE OF THE INTEGRATION SCHEME ON THE ACCURACY FOR (A) RPIM (A) AND FOR (B) NNRPIM.	86
FIGURE 45 – GEOMETRIC AND LOAD PROPERTIES OF THE SQUARE PLATE AND REGULAR NODAL MESH DISTRIBUTION.....	88
FIGURE 46 – RELATIVE ERROR OBTAINED IN THE ANALYSIS WHEN THE PARAMETER λ VARIES. .90	
FIGURE 47 – (A) CONSIDERED MATERIAL AND GEOMETRIC CONDITIONS OF THE UNIT SQUARE PLATE PROBLEM. (B) IRREGULAR 169 NODE MESH. (C) REGULAR 169 NODE MESH.	94
FIGURE 48 – DISPLACEMENT VALUES FOR THE INTEREST POINTS.	94
FIGURE 49 – MEDIUM DISPLACEMENT AND STRESS ERRORS IN RELATION TO THE NUMBER OF NODES OBTAINED WITH THE: IRREGULAR MESH (A) AND (C) AND WITH THE REGULAR MESH (B) AND (D). yy AXIS IN LOGARITHMIC SCALE.	96
FIGURE 50 – MEDIUM DISPLACEMENT ERROR, IN RELATION TO THE COMPUTATIONAL COST, IN A LOGARITHMIC SCALE, OBTAINED WITH THE (A) REGULAR MESH AND WITH THE (B) IRREGULAR MESH.	96
FIGURE 51 – CONSIDERED MATERIAL AND GEOMETRIC CONDITIONS OF THE CANTILEVER BEAM PROBLEM.....	97
FIGURE 52 – (A) 2D REGULAR 231 NODE MESH. (B) 2D IRREGULAR 231 NODE MESH. (C) 3D REGULAR 637 NODE MESH. (D) 3D IRREGULAR 637 NODE MESH.....	98
FIGURE 53 – (A)MEDIUM DISPLACEMENT ERROR Θ . (B)MEDIUM STRESS ERROR $\Theta_{\sigma_{xx}}$. (C)MEDIUM STRESS ERROR $\Theta_{\sigma_{xy}}$	99
FIGURE 54 – DISPLACEMENT VALUES OBTAIN WITH IRREGULAR AND REGULAR MESHES IN INTEREST POINTS A AND B.....	99

FIGURE 55 – STRESS VALUES ON THE INTEREST POINTS C AND D FOR IRREGULAR MESHES AND FOR REGULAR MESHES.	100
FIGURE 56 – MATERIAL AND GEOMETRICAL CHARACTERISTICS AND BOUNDARY AND LOAD CONDITIONS OF THE PLATE WITH A CIRCULAR HOLE.	101
FIGURE 57 – EXAMPLES OF THE REGULAR NODAL DISCRETIZATION FOR THE PLATE WITH A CIRCULAR HOLE.	102
FIGURE 58 – DISPLACEMENT VALUES IN INTEREST POINTS A AND B.	102
FIGURE 59 – MEDIUM STRESS ERRORS.	103
FIGURE 60 – NORMAL STRESSES IN INTEREST POINTS A AND B.	103
FIGURE 61 – DISPLACEMENT FIELD AND THE STRESS DISTRIBUTION OBTAINED WITH THE NNRPIM AND WITH THE EXACT SOLUTION.	103
FIGURE 62 – SQUARE PLATE REGULAR AND IRREGULAR MESHES.	104
FIGURE 63 – CENTRE TRANSVERSE DISPLACEMENT VERSUS NUMBER OF USED NODES FOR THE (A) SIMPLY SUPPORTED SQUARE PLATE AND (B) CLAMPED SQUARE PLATE.	105
FIGURE 64 – CIRCULAR PLATE MESHES.	105
FIGURE 65 – CENTRE TRANSVERSE DISPLACEMENT VERSUS NUMBER OF USED NODES FOR THE (A) SIMPLY SUPPORTED CIRCULAR PLATE AND (B) CLAMPED CIRCULAR PLATE.	106
FIGURE 66 – (A) GEOMETRIC PROPERTIES OF THE SKEWED PLATE AND TYPICAL REGULAR MESH. (B) CENTRE TRANSVERSE DISPLACEMENT VERSUS NUMBER OF USED NODES.	107
FIGURE 67 – GEOMETRIC AND MATERIAL PROPERTIES OF THE SCORDELIS-LO CYLINDRICAL ROOF.	107
FIGURE 68 – USED MESHES IN THE ANALYSIS OF THE SCORDELIS-LO CYLINDRICAL ROOF PROBLEM.	108
FIGURE 69 – TRANSVERSE DISPLACEMENT ON POINT A VERSUS NUMBER OF USED NODES.	108
FIGURE 70 – GEOMETRIC AND MATERIAL PROPERTIES OF THE CYLINDRICAL SHELL.	109
FIGURE 71 – USED MESHES IN THE ANALYSIS OF CYLINDRICAL SHELL PROBLEM.	109
FIGURE 72 – TRANSVERSE DISPLACEMENT ON POINT A VERSUS NUMBER OF USED NODES.	110
FIGURE 73 – LAMINATE GEOMETRIC PROPERTIES AND LAMINA MATERIAL PROPERTIES.	111
FIGURE 74 – NORMALIZED CENTRE TRANSVERSE DISPLACEMENT VERSUS NUMBER OF USED NODES DISCRETIZING THE 0/90/90/0 LAMINATE. (A) ULD. (B) SSL.	112
FIGURE 75 – NORMALIZED CENTRE TRANSVERSE DISPLACEMENT VERSUS NUMBER OF USED NODES DISCRETIZING THE 0/90/0 LAMINATE. (A) ULD. (B) SSL.	112
FIGURE 76 – NON-DIMENSIONAL STRESSES VERSUS THICKNESS FOR A SIMPLY SUPPORTED 0/90/90/0 LAMINATE UNDER A SINUSOIDAL LOAD.	113
FIGURE 77 – GEOMETRIC PROPERTIES OF THE CYLINDRICAL LAMINATED SHELL (A) COMPLETE SHELL. (B) UNIT STRIP.	116
FIGURE 78 – MESHES USED IN THE ANALYSIS. (A) STRIP MESH RM1. (B) STRIP MESH IM1. (C) QUARTER MESH RM2.	117
FIGURE 79 – MESHES USED IN THE ANALYSIS. (A) STRIP MESH M1. (B) STRIP MESH M2.	118
FIGURE 80 – NORMALIZED CENTRE TRANSVERSE DISPLACEMENT VERSUS NUMBER OF USED NODES DISCRETIZING THE (A) 0/90 LAMINATE AND (B) 0/90/90/0 LAMINATE.	118
FIGURE 81 – CANTILEVER BEAM. EXAMPLES OF 2D AND 3D, REGULAR AND IRREGULAR MESHES.	119
FIGURE 82 – CROSS SECTION OF THE VARIABLE BEAMS AND RESPECTIVE NODAL ARRANGEMENT FOR THE 2D AND 3D MODULATION. (A) BEAM A AND (B) BEAM B.	121
FIGURE 83 – SHEAR WALL WITH FOUR OPENINGS AND RESPECTIVE MESH DISCRETIZATION FOR THE 2D AND THE 3D MODULATION.	122
FIGURE 84. – SQUARE PLATE CONVERGENCE STUDIES FOR DISTINCT BOUNDARY CONDITIONS.	124
FIGURE 85 – (A) CIRCULAR REGULAR MESH. (B) CIRCULAR IRREGULAR MESH.	126
FIGURE 86 – CONVERGENCE STUDY OF THE CIRCULAR CLAMPED PLATE.	127
FIGURE 87 – GENERIC SKEWED PLATE GEOMETRIC PROPERTIES AND USED REGULAR MESH WITH $\theta=30^\circ$	128
FIGURE 88 – THIN CLAMPED CYLINDRICAL SHELL PANEL GEOMETRIC AND MATERIAL PROPERTIES AND REGULAR AND IRREGULAR MESHES EXAMPLES USED IN THE ANALYSIS.	130
FIGURE 89 – CONVERGENCE STUDY OF THE THIN CLAMPED CYLINDRICAL SHELL PANEL.	131
FIGURE 90 – THIN CLAMPED-FREE CYLINDRICAL SHELL PANEL GEOMETRIC AND MATERIAL PROPERTIES AND REGULAR AND IRREGULAR MESHES EXAMPLES USED IN THE ANALYSIS.	131
FIGURE 91 – CONVERGENCE STUDY OF THE THIN CLAMPED-FREE CYLINDRICAL SHELL PANEL.	132

FIGURE 92 – SQUARE PLATE MAXIMUM DEFLECTION VARYING ALONG TIME FOR A SUDDENLY APPLIED LOAD.	133
FIGURE 93 – SQUARE PLATE MAXIMUM DEFLECTION VARYING ALONG TIME FOR AN APPLIED LOAD THAT SUDDENLY VANISHES.	134
FIGURE 94 – SQUARE PLATE ABSOLUTE VALUE OF THE AMPLITUDE VERSUS THE LOAD NORMALIZED FREQUENCY.	135
FIGURE 95 – MAXIMUM DEFLECTION VARYING ALONG TIME FOR A SUDDENLY APPLIED LOAD FOR (A) THE THIN CLAMPED CYLINDRICAL SHELL PANEL AND FOR (B) THE CLAMPED-FREE CYLINDRICAL SHELL.	136
FIGURE 96 – MAXIMUM DEFLECTION VARYING ALONG TIME FOR AN APPLIED LOAD THAT SUDDENLY VANISHES FOR (A) THE THIN CLAMPED CYLINDRICAL SHELL PANEL AND FOR (B) THE CLAMPED-FREE CYLINDRICAL SHELL.	136
FIGURE 97 – ABSOLUTE VALUE OF THE AMPLITUDE VERSUS THE LOAD NORMALIZED FREQUENCY FOR (A) THE THIN CLAMPED CYLINDRICAL SHELL PANEL AND FOR (B) THE CLAMPED-FREE CYLINDRICAL SHELL.	137
FIGURE 98 – (A) CANTILEVER BEAM. (B) REGULAR MESH. (C) IRREGULAR MESH.	140
FIGURE 99 – LOAD/DISPLACEMENT DIAGRAM FOR THE VERTICAL DISPLACEMENT IN POINT A... 140	
FIGURE 100 – NORMAL STRESS DISTRIBUTION ALONG THE ESSENTIAL BOUNDARY FOR INCREASING LOAD LEVELS.	141
FIGURE 101 – (A) COOK MEMBRANE. (B) IRREGULAR 462 NODE MESH.	142
FIGURE 102 – (A) v_A AND (B) u_A DISPLACEMENT COMPONENTS IN RELATION TO THE APPLIED LOAD.	142
FIGURE 103 – NORMAL STRESS DISTRIBUTION ALONG THE MEMBRANE ESSENTIAL BOUNDARY FOR INCREASING LOAD LEVELS.	143
FIGURE 104 – SHEAR STRESS DISTRIBUTION ALONG THE MEMBRANE ESSENTIAL BOUNDARY FOR INCREASING LOAD LEVELS.	144
FIGURE 105 – (A) QUARTER OF THE PLATE WITH THE CIRCULAR HOLE. (B) 660 NODE MESH.	145
FIGURE 106 – (A) v_B , (B) u_D AND (C) Δ_C DISPLACEMENT COMPONENTS IN RELATION TO THE APPLIED LOAD.	145
FIGURE 107 – NORMAL STRESS DISTRIBUTION ALONG ESSENTIAL BOUNDARY DEFINE BY AB FOR INCREASING LOAD LEVELS.	146
FIGURE 108 – (A) CYLINDER GEOMETRIC CHARACTERISTICS. (B) TWO-DIMENSIONAL MESH WITH 315 NODES AND (C) THREE-DIMENSIONAL REGULAR MESH WITH 2205 NODES.	147
FIGURE 109 – LOAD/DISPLACEMENT DIAGRAM.	148
FIGURE 110 – (A) BLOCK UNDER PRESSURE. (B) REGULAR MESH OF 1331 NODES.	148
FIGURE 111 – (A) LOAD/DISPLACEMENT w_A DIAGRAM. (B) LOAD/DISPLACEMENT w DIAGRAM FOR LINE AB	149
FIGURE 112 – EFFECTIVE STRESS MAPS.	149
FIGURE 113 – (A) STUDIED SQUARE PLATE. (B) REGULAR MESHES WITH 363 NODES (M2) AND REGULAR MESH WITH 605 NODES (M4).	150
FIGURE 114 – NORMALIZED LOAD/VERTICAL CENTRAL DISPLACEMENT w_C DIAGRAM FOR THE (A) SSSS PLATE AND THE (B) CCCC PLATE.	150
FIGURE 115 – EFFECTIVE STRESS MAP FOR (A) THE SSSS PLATE AND (B) THE CCCC PLATE.	151
FIGURE 116 – GEOMETRICAL MODEL OF PROPOSED THICK ARCH.	152
FIGURE 117 – MESHES USED IN THE ANALYSIS. (A) 2D REGULAR 73 NODAL MESH. (B) 2D REGULAR 255 NODAL MESH. (C) 2D REGULAR 909 NODAL MESH. (D) 2D IRREGULAR 303 NODAL MESH. (E) 3D REGULAR 510 NODAL MESH. (F) 3D IRREGULAR 606 NODAL MESH. ..	152
FIGURE 118 – EVOLUTION OF THE DISPLACEMENT ON THE MID SPAN OF THE ARCH FOR DISTINCT MESHES.	153
FIGURE 119 – EVOLUTION OF THE DISPLACEMENT ON THE MID SPAN OF THE ARCH WHEN TWO DISTINCT NONLINEAR SOLUTION METHODS ARE USED.	154
FIGURE 120 – EVOLUTION OF THE DISPLACEMENT ON THE MID SPAN OF THE ARCH FOR SEVERAL VALUES OF THE POISSON RATIO.	154
FIGURE 121 – EVOLUTION OF THE DISPLACEMENT ON THE MID SPAN OF THE ARCH OBTAINED WITH A 3D FORMULATION AND WITH IRREGULAR MESHES.	155
FIGURE 122 – GEOMETRICAL MODEL OF THE PROPOSED SHALLOW ARCH.	156
FIGURE 123 – MESHES USED IN THE ANALYSIS.	156
FIGURE 124 – DISPLACEMENT ON THE MID SPAN OF THE ARCH.	157

FIGURE 125 – GEOMETRICAL MODEL OF THE ROLL-BEAM PROPOSED.....	158
FIGURE 126 – 2D AND 3D MESHES USED IN THE ANALYSIS.....	158
FIGURE 127 – DEFORMED MESHES FOR INCREASING TIP MOMENTS FOR THE 2D ANALYSIS.....	159
FIGURE 128 – DEFORMED MESHES FOR INCREASING TIP MOMENTS FOR THE 3D ANALYSIS.....	159
FIGURE 129 – GEOMETRICAL MODEL OF THE CANTILEVER BEAM PROPOSED AND MESHES CONSIDERED IN THE ANALYSIS.	160
FIGURE 130 – EVOLUTION OF THE DISPLACEMENT ON POINT A WITH THE LOAD LEVEL FOR DISTINCT MESHES, FOR THE 2D AND THE 3D ANALYSIS.....	161
FIGURE 131 – DEFORMATION OF THE CANTILEVER BEAM FOR SEVERAL INCREASING LOAD LEVELS FOR THE 2D ANALYSIS.	161
FIGURE 132 – DEFORMATION OF THE CANTILEVER BEAM FOR SEVERAL INCREASING LOAD LEVELS FOR THE 3D ANALYSIS.	162
FIGURE 133 – GEOMETRICAL MODEL OF THE PLATE AND APPLIED LOAD.	162
FIGURE 134 – (A)REGULAR MESH WITH 363 NODES AND (D) IRREGULAR MESH WITH 474 NODES.	163
FIGURE 135 – EVOLUTION OF THE CENTRAL DISPLACEMENT OF THE SIMPLY SUPPORTED PLATE WITH THE LOAD LEVEL, FOR THE 3D ANALYSIS.	163
FIGURE 136 – EVOLUTION OF THE CENTRAL DISPLACEMENT OF THE CLAMPED PLATE WITH THE LOAD LEVEL, FOR THE 3D ANALYSIS.	164
FIGURE 137 – DEFORMATION OF THE CLAMPED SQUARE PLATE FOR SEVERAL INCREASING LOAD LEVELS. (A) SIMPLY SUPPORTED PLATE. (B) CLAMPED PLATE.	165
FIGURE 138 – GEOMETRICAL MODEL OF THE CURVED CANTILEVER BEAM AND APPLIED LOAD. .	166
FIGURE 139 – MESHES USED IN THE ANALYSIS.....	167
FIGURE 140 – EVOLUTION OF THE DISPLACEMENT ALONG X DIRECTION ON THE END TIP OF THE CURVED CANTILEVER BEAM WITH THE LOAD LEVEL, FOR DISTINCT MESHES.	168
FIGURE 141 – EVOLUTION OF THE DISPLACEMENT ALONG Y DIRECTION ON THE END TIP OF THE CURVED CANTILEVER BEAM WITH THE LOAD LEVEL, FOR DISTINCT MESHES.	168
FIGURE 142 – EVOLUTION OF THE DISPLACEMENT ALONG Z DIRECTION ON THE END TIP OF THE CURVED CANTILEVER BEAM WITH THE LOAD LEVEL, FOR DISTINCT MESHES.	169
FIGURE 143 – DEFORMATION OF THE CURVED BEAM FOR SEVERAL INCREASING LOAD LEVELS.	169
FIGURE 144 – FRAME DOME GEOMETRIC PROPERTIES AND SECTION MESHES USED IN THE ANALYSIS.	170
FIGURE 145 – 3D MESHES USED IN THE ANALYSIS.	171
FIGURE 146 – GLOBAL 3D MESH BASED ON MESH2.....	171
FIGURE 147 – EVOLUTION OF THE DISPLACEMENT w_A ON THE TOP OF THE DOME WITH THE LOAD LEVEL, FOR DISTINCT MESHES AND DISTINCT NUMERICAL METHODS.	172
FIGURE 148 – DEFORMATION AND STRESS FIELD OF THE DOME FOR SEVERAL INCREASING LOAD LEVELS. (A) LARGE DEFORMATIONS WITH ELASTIC MATERIAL BEHAVIOUR. (B) LARGE DEFORMATIONS WITH ELASTOPLASTIC MATERIAL BEHAVIOUR.	173
FIGURE 149. – BONE SHAPE MAJOR GROUPS.	177
FIGURE 150 – LONG BONE. HUMAN FEMUR WITH THE IDENTIFICATION OF THE PRIMARY STRUCTURES AND INNER FEATURES OF A DIAPHYSIS SECTION.....	178
FIGURE 151 – BONE REABSORPTION PROCESS.	181
FIGURE 152 – TRANSVERSAL CROSS-SECTION PIE-SHAPED OF AN OSTEON AND TRANSVERSAL CROSS-SECTION OF A TRABECULAR BRANCH. DETAILED VIEW OF A TYPICAL BONE CORTEX.	183
FIGURE 153 – ELASTICITY MODULUS IN THE AXIAL DIRECTION. EXPERIMENTAL DATA OBTAIN IN ZIOUPOS WORK COMPARED WITH LOTZ LAW FOR CORTICAL AND TRABECULAR BONE AND WITH THE MATHEMATICAL MODEL PROPOSED IN THIS WORK.	189
FIGURE 154 - ELASTICITY MODULUS IN THE TRANSVERSAL DIRECTION. LOTZ LAW FOR CORTICAL AND TRABECULAR BONE COMPARED WITH THE MATHEMATICAL MODEL PROPOSED IN THIS WORK.....	190
FIGURE 155 – COMPRESSION STRESS IN THE AXIAL DIRECTION. LOTZ LAW FOR CORTICAL AND TRABECULAR BONE COMPARED WITH THE MATHEMATICAL MODEL PROPOSED IN THIS WORK.	190
FIGURE 156 - COMPRESSION STRESS IN THE TRANSVERSAL DIRECTION. LOTZ LAW FOR CORTICAL AND TRABECULAR BONE COMPARED WITH THE MATHEMATICAL MODEL PROPOSED IN THIS WORK.....	191

FIGURE 157 – PROPOSED BONE REMODELLING ALGORITHM.	201
FIGURE 158 – (A) PLATE MODEL GEOMETRY AND ESSENTIAL AND NATURAL BOUNDARY CONDITIONS. (B) REGULAR NODAL MESH (1681 NODES). (C) IRREGULAR NODAL MESH (1952 NODES). (D) SECOND LOAD CASE CONSIDERED IN THE ANALYSIS.....	204
FIGURE 159 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 10% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ISOTROPIC MATERIAL PROPERTIES).....	204
FIGURE 160 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 5% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ISOTROPIC MATERIAL PROPERTIES).....	205
FIGURE 161 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 2% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ISOTROPIC MATERIAL PROPERTIES).....	205
FIGURE 162 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 1% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ISOTROPIC MATERIAL PROPERTIES).....	205
FIGURE 163 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 10% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	206
FIGURE 164 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 5% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	206
FIGURE 165 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 2% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	206
FIGURE 166 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 1% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	206
FIGURE 167 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 10% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	207
FIGURE 168 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 5% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	207
FIGURE 169 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 2% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	207
FIGURE 170 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 1% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (PROPOSED MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	207
FIGURE 171 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 10% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	208
FIGURE 172 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 5% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	208
FIGURE 173 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 2% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	208
FIGURE 174 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 1% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, REGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	209
FIGURE 175 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 10% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	209
FIGURE 176 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 5% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES).....	209

FIGURE 177 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 2% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	209
FIGURE 178 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WHEN 1% OF INTEREST POINTS ARE REMODELLED IN EACH ITERATION. (LOTZ MATERIAL LAW, IRREGULAR MESH AND ANISOTROPIC MATERIAL PROPERTIES)	210
FIGURE 179 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WITH A REGULAR NODAL MESH FOR THE TWO LOAD CASES: L1 WITH 1000 CYCLES AND L2 WITH 1000 CYCLES.	211
FIGURE 180 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WITH A IRREGULAR NODAL MESH FOR THE TWO LOAD CASES: L1 WITH 1000 CYCLES AND L2 WITH 1000 CYCLES.	211
FIGURE 181 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WITH A REGULAR NODAL MESH FOR THE TWO LOAD CASES: L1 WITH 1000 CYCLES AND L2 WITH 5000 CYCLES.	211
FIGURE 182 – EVOLUTION OF THE APPARENT DENSITY IN THE SQUARE BONE PATCH WITH A IRREGULAR NODAL MESH FOR THE TWO LOAD CASES: L1 WITH 1000 CYCLES AND L2 WITH 5000 CYCLES.	212
FIGURE 183 – (A) CUBIC PATCH MODEL SUBMITTED TO VERTICAL LOADS. (B) CUBIC PATCH MODEL SUBMITTED TO A DIAGONAL LOAD. (C) REGULAR NODAL MESH (2681 NODES). (D) PATCH SECTION CUT.	213
FIGURE 184 – EVOLUTION OF THE APPARENT DENSITY IN THE BONE CUBIC PATCH FOR VERTICAL LOADS.	214
FIGURE 185 – EVOLUTION OF THE APPARENT DENSITY IN THE BONE CUBIC PATCH FOR DIAGONAL LOAD.	214
FIGURE 186 – CUBIC PATCH APPARENT DENSITY THREE-DIMENSIONAL VIEW.	214
FIGURE 187 – (A) CUBIC PATCH MODEL SUBMITTED TO A DIAGONAL LOAD. (B) REGULAR NODAL MESH (2744 NODES). (C) PATCH SECTION CUT.	215
FIGURE 188 – EVOLUTION OF THE APPARENT DENSITY IN THE BONE CUBIC PATCH FOR DIAGONAL LOAD.	216
FIGURE 189 – (A) CUBIC PATCH MODEL SUBMITTED TO CROSS DIAGONAL LOADS. (B) PATCH SECTIONS CUTS.	217
FIGURE 190 – EVOLUTION OF THE APPARENT DENSITY IN THE BONE CUBIC PATCH FOR THE CROSS DIAGONAL LOADS. (ISOTROPIC MATERIAL CONSIDERING THE PROPOSED MATERIAL LAW).	218
FIGURE 191 – EVOLUTION OF THE APPARENT DENSITY IN THE BONE CUBIC PATCH FOR THE CROSS DIAGONAL LOADS. (ANISOTROPIC MATERIAL CONSIDERING THE PROPOSED MATERIAL LAW).	219
FIGURE 192 – EVOLUTION OF THE APPARENT DENSITY IN THE BONE CUBIC PATCH FOR THE CROSS DIAGONAL LOADS. (ANISOTROPIC MATERIAL CONSIDERING LOTZ MATERIAL LAW).....	220
FIGURE 193 – (A) FOOT X-RAY PLATE. (B) CALCANEUS INTERNAL PRINCIPAL TRABECULAR STRUCTURES.....	221
FIGURE 194 – TENDON FORCES. ACHILLES TENDON FORCE (T1), PLANTAR FASCIA FAN-TENDON FORCE (T2) AND CALCANEUS-METATARSAL TENDON FORCE (T3).	222
FIGURE 195 – GEOMETRY AND NODAL MESH USED IN THE ANALYSIS OF THE CALCANEUS BONE.	222
FIGURE 196 – LOADS AND CONSTRAINS OF THE FIRST MECHANICAL CASE OF THE CALCANEUS BONE.	223
FIGURE 197 – LOADS AND CONSTRAINS OF THE SECOND MECHANICAL CASE OF THE CALCANEUS BONE.	224
FIGURE 198 – LOADS AND CONSTRAINS OF THE THIRD MECHANICAL CASE OF THE CALCANEUS BONE.	224
FIGURE 199 – EVOLUTION OF THE APPARENT DENSITY IN THE CALCANEUS BONE FOR THE (A) FIRST MECHANICAL CASE, (B) SECOND MECHANICAL CASE AND (C) THIRD MECHANICAL CASE.	226
FIGURE 200 – EVOLUTION OF THE APPARENT DENSITY IN THE CALCANEUS BONE FOR THE MIXING CASES. (A) MECHANICAL LOAD 1 - 6000 CYCLES; MECHANICAL LOAD 2 - 6000 CYCLES; MECHANICAL LOAD 3 - 4200 CYCLES. (B) MECHANICAL LOAD 1 - 6000 CYCLES; MECHANICAL LOAD 2 - 6000 CYCLES; MECHANICAL LOAD 3 - 6000 CYCLES.	227

FIGURE 201 – (A) FEMORAL X-RAY PLATE. (B) FEMORAL INTERNAL PRINCIPAL TRABECULAR STRUCTURES. (C) NODAL MESH.	229
FIGURE 202 – LOADS AND CONSTRAINS OF THE VARIOUS MECHANICAL CASE OF THE FEMUR BONE. (A) FIRST MECHANICAL CASE, (B) SECOND MECHANICAL CASE AND (C) THIRD MECHANICAL CASE.	229
FIGURE 203 – EVOLUTION OF THE APPARENT DENSITY IN THE FEMORAL BONE FOR THE (A) FIRST MECHANICAL CASE, (B) SECOND MECHANICAL CASE AND (C) THIRD MECHANICAL CASE. ...	231
FIGURE 204 – EVOLUTION OF THE APPARENT DENSITY IN THE FEMORAL BONE WHEN THE THREE MECHANICAL CASES ARE APPLIED SIMULTANEOUSLY.	232
FIGURE 205 – THREE-DIMENSIONAL MODEL OF THE FEMUR BONE.	233
FIGURE 206 – (A) FIRST MECHANICAL CASE. (B) SECOND MECHANICAL CASE. (C) THIRD MECHANICAL CASE.	234
FIGURE 207 – THREE-DIMENSIONAL MODEL SECTIONAL CUT AA.....	234
FIGURE 208 – EVOLUTION OF THE APPARENT DENSITY IN THE FEMORAL BONE FOR THE (A) FIRST MECHANICAL CASE, (B) SECOND MECHANICAL CASE AND (C) THIRD MECHANICAL CASE. (D) THREE MECHANICAL CASES APPLIED SIMULTANEOUSLY.	236

List of tables

TABLE 1 – NNRPIM STATIC LINEAR FLOW CHART.	75
TABLE 2 – COORDINATES OF THE NODES THAT DISCRETIZE THE PATCH DOMAIN.	78
TABLE 3 – OBTAINED RESULTS FOR OPTIMAL SHAPE PARAMETERS FOR THE 2D ANALYSIS.	85
TABLE 4 – OBTAINED RESULTS FOR OPTIMAL SHAPE PARAMETERS FOR THE 3D ANALYSIS.	87
TABLE 5 – SELECTED OPTIMAL SHAPE PARAMETERS.	87
TABLE 6 – CENTRE TRANSVERSE DISPLACEMENT FOR DISTINCT FORMULATIONS AND INCREASING NODAL DISCRETIZATION.	90
TABLE 7 – COMPUTATIONAL TIME IN SECONDS FOR THE DISTINCT NNRPIM FORMULATIONS AND INCREASING NODAL DISCRETIZATION.	90
TABLE 8 – NORMALIZED DISPLACEMENT IN THE CENTRE OF THE LAMINATES FOR DISTINCT FORMULATIONS AND NODAL MESHES.	113
TABLE 9 – NORMALIZED DISPLACEMENT IN THE CENTRE OF THE LAMINATE FOR DISTINCT FORMULATIONS AND NODAL MESHES, VARYING THE BOUNDARY CONDITIONS.	114
TABLE 10 – NORMALIZED DISPLACEMENT IN THE CENTRE OF THE LAMINATE FOR DISTINCT FORMULATIONS AND NODAL MESHES VARYING THE BOUNDARY CONDITIONS.	115
TABLE 11 – NORMALIZED DISPLACEMENT IN THE CENTRE OF THE ANGLE-PLY LAMINATES FOR DISTINCT THICKNESS.	115
TABLE 12 – NORMALIZED DISPLACEMENT IN THE CENTRE OF THE CROSS-PLY SHELL LAMINATES FOR DISTINCT FORMULATIONS AND NODAL MESHES.	117
TABLE 13 – CONVERGENCE OF THE FIRST NATURAL FREQUENCY ω (RAD/S) VARYING THE NUMBER OF NODES DISCRETIZING THE CANTILEVER BEAM DOMAIN.	119
TABLE 14 – FIRST TEN NATURAL FREQUENCIES ω (RAD/S) OBTAINED FOR THE CANTILEVER BEAM.	120
TABLE 15 – OBTAINED NATURAL FREQUENCIES ω (RAD/S) WITH THE MESHLESS SOLUTIONS AND THE FEM FOR BEAM A.	121
TABLE 16 – OBTAINED NATURAL FREQUENCIES ω (RAD/S) WITH THE MESHLESS SOLUTIONS AND THE FEM FOR BEAM B.	122
TABLE 17 – FIRST FIVE NATURAL FREQUENCIES ω (RAD/S) OBTAINED FOR THE SHEAR-WALL... 123	123
TABLE 18 – NORMALIZED NATURAL FREQUENCIES OF THE SSSS SQUARE PLATE.	125
TABLE 19 – NORMALIZED NATURAL FREQUENCIES OF THE CCCC SQUARE PLATE.	125
TABLE 20 – NORMALIZED NATURAL FREQUENCIES OF THE SCSC SQUARE PLATE.	125
TABLE 21 – NORMALIZED NATURAL FREQUENCIES OF THE SCSS SQUARE PLATE.	126
TABLE 22 – NORMALIZED NATURAL FREQUENCIES OF THE CIRCULAR CLAMPED PLATE.	127
TABLE 23 – SKEWED PLATES NORMALIZED NATURAL FREQUENCIES.	129
TABLE 24 – LOTZ LAW COEFFICIENTS.	187
TABLE 25 – COEFFICIENTS OF THE PROPOSE BONE MODEL.	189

Chapter 1

1 Introduction

The engineering design process can be divided in many specific phases. The present work lean over three of this phases: the Modulation, the Simulation and the Analysis. All this process is recurrent by nature, always looking for the numerical approach which better reproduces the studied phenomenon. At the present time, there are many numerical methods available and capable to successfully handle the referred engineering design process phases. This work presents, develop and extends to many engineering fields a new numerical method. From the simple solid mechanical problems to the complex nonlinear analysis in biomechanics, this work shows that the proposed numerical method is flexible and accurate.

1.1 Meshless Methods

In the last few years meshless methods for numerically solving partial differential equations came into focus of interest, especially in the engineering community. In the meshless methods [1-3] the nodes can be arbitrary distributed, once the field functions are approximated within an influence domain rather than an element. In opposition to the no-overlap rule between elements in the Finite Element Method (FEM) [4, 5], in meshless methods the influence domains may and must overlap each other. A numerical method can be divided in three fundamental phases: the field approximation (or interpolation) function, the formulation and the integration.

1.1.1 Approximation (or interpolation) functions

There are many approximation (or interpolation) functions. The most relevant are the Taylor approximation, the moving least-square approximation, the reproducing kernel approximation, the hp-cloud approximation function, the polynomial interpolation, the parametric

interpolation, the radial interpolation and the Sibson interpolation. The approximation, or interpolation, function requires a domain of applicability, outside this domain the function assumes zero values. In the FEM this domain is the ‘element’ and in the meshless methods this domain is called ‘influence-domain’. In meshless methods it is necessary to determine the influence-domain for each node within the nodal mesh discretizing the problem domain, as a consequence the shape and the size of the influence-domain vary with the considered node. The method in which each influence-domain is obtained varies with the used meshless method.

1.1.2 Formulation

Meshless methods can be classified in two categories, a first category that pursues the strong form solution and another that seeks the weak form solution. The strong form formulation uses directly the partial differential equations in order to obtain the solution. The weak form formulation uses a variational principle to minimize the residual weight of the differential equations ruling the phenomenon. The residual is obtained by substituting the exact solution by an approximated function affected by a test function. The existent distinct weak form solution methods are dependent on the used test function. Surprisingly a differential equation may have solutions which are not exactly differentiable and the weak formulation allows to find such solutions. Weak form solutions are very important since many differential equations governing the real world phenomena do not admit sufficiently smooth solutions, then the only way of solving such equations is using the weak formulation.

1.1.3 Integration

In order to obtain the integral of the residual weight of the differential equations it is necessary to select an integration scheme. The integration can be made using a background mesh, covering the entire problem domain, composed by integration points. These integration points must have a defined influence area and weight (which corresponds to the theoretical infinitesimal mass portion defined in the integral expression) and should not overlap each other. This background mesh for integration proposes generally is nodal independent, which jeopardises the ‘meshless’ denomination of such numerical methods. There are another integration schemes often used, the point collocation and the nodal integration. In these integration schemes the node represents the integration point, the influence area is the node influence-domain and the integration weight is the node influence volume. In this case the integration mesh is the nodal mesh itself.

Some authors have the opinion that such integration schemes are less accurate, which is not true. Meshless methods using these integrations schemes are considered ‘truly’ meshless methods. In this work a hybrid integration scheme, totally nodal dependent, is presented and applied, warranting that the proposed meshless method is ‘truly’ meshless.

1.1.4 Relevant Meshless Methods

The initially created meshless methods used approximation functions, since it produces more smooth solutions, the implementation of the influence-domain concept was easier and the background integration scheme was nodal independent. The first meshless method using the Moving Least Square approximants (MLS) in the construction of the approximation function was the Diffuse Element Method (DEM) [6]. The MLS was proposed by Lancaster and Salkauskas [7] for surface fitting. Belytschko evolved the DEM and developed one of the most popular meshless methods, the Element Free Galerkin Method (EFGM) [8], which uses a nodal independent background integration mesh. One of the oldest meshless methods is the Smooth Particle Hydrodynamics Method (SPH) [9], which is in the origin of the Reproducing Kernel Particle Method (RKPM) [10]. Another very popular approximant meshless method is the meshless local Petrov-Galerkin method (MLPG) [11], initially created to solve linear and nonlinear potential problems, which later evolved towards the Method of the Finite Spheres (MFS) [12]. The Finite Point Method (FPM) [13, 14] uses for integration proposes a stabilization technique in the collocation point method. Another approximation method, distinct from the previous, is the Radial Basis Function Method (RBFM) [15]. It uses the radial basis functions, respecting a Euclidean norm, to approximate the variable fields within the entire domain or in small domains. It does not require an integration mesh and, in opposition to the previous referred meshless methods, uses the strong form formulation. Initially used to approximate multidimensional data [16] only latter it was applied by others [17, 18] to the analysis of differential equations.

Although approximants meshless methods have been successfully applied in computational mechanics there were several problems not completely solved. One of these problems, and perhaps the most important unsolved issue, was the lack of the delta Kronecker property, $\varphi_i(\mathbf{x}_j) \neq \delta_{ij}$, on the approximation functions, which difficult the imposition of essential and natural boundary conditions.

To address the above problem, several interpolant meshless methods were developed in the last few years. The most relevant are the Point Interpolation Method (PIM) [19], the Point Assembly Method [20], the Radial Point Interpolation Method (RPIM) [21, 22], Meshless Finite Element Method (MFEM) [23]. The Natural Neighbour Finite Element Method (NNFEM) [24, 25] or the Natural Element Method (NEM) [26-28] are meshless methods that use the Sibson interpolation functions. More recently, the combination between the NEM and the RPIM originated the Natural Neighbour Radial Point Interpolation Method (NNRPIM) [29], which is the object of this work.

1.2 Natural Neighbour Radial Point Interpolation Method

The Natural Neighbour Radial Point Interpolation Method (NNRPIM) is the product of the combination of the Radial Point Interpolators (RPI) with the Natural Neighbours geometric concept. The RPI have its origin in the Point Interpolation Method (PIM) [19]. The technique consisted in constructing polynomial interpolants, possessing the delta Kronecker property, based only on a group of arbitrarily distributed points. These equations are then derived using variational principles. However this technique has too many numerical problems, for instance the perfect alignment of the nodes produces singular solutions in the interpolation function construction process. As so, this technique evolved and the Radial Point Interpolation Method (RPIM) [21] was created. In this meshless method the Radial Basis Function (RBF) was added in the construction process of the interpolation function, stabilizing the method. The RBF used in these early works were the Gaussian and the multiquadric RBF. Initially the RBF was developed for data surface fitting, and later, with the work developed by Kansa [15], the RBF was used for solving partial differential equations. However the RPIM uses, unlike Kansa's algorithm, the concept of "influence domain" instead of "global domain", generating sparse and banded stiffness matrices, more adequated to complex geometry problems.

The NNRPIM is the next step in the RPI. In order to impose the nodal connectivity, the 'influence-domain' is substituted by the 'influence-cell' concept. In order to obtain the 'influence-cells' the NNRPIM relies on geometrical and mathematical constructions such as the Voronoï diagrams [30] and the Delaunay tessellation [31]. Thus, resorting to Voronoï cells, a set of influence cells are created departing from an unstructured set of nodes. The Delaunay triangles, which are the dual of the Voronoï cells, are applied to create a node-depending background mesh used in the numerical

integration of the NNRPIM interpolation functions. Due to the integration mesh total dependency on the nodal mesh, the NNRPIM can be considered a truly meshless method. Unlike the FEM, where geometrical restrictions on elements are imposed for the convergence of the method, in the NNRPIM there are no such restrictions, which permits a total random node distribution for the discretized problem. The NNRPIM interpolation functions, used in the Galerkin weak form, are constructed in a similar process to the RPIM, with some differences that modify the method performance.

Although the NNRPIM is a recent developed meshless method [29] it has been extended to many fields of the computational mechanics, such as the static analysis of isotropic and orthotropic plates [32] and the functionally graded material plate analysis [33], the 3D shell-like approach [34] for laminated plates and shells [35]. The dynamic analysis of several solid-mechanic problems was also studied [36-38]. The NNRPIM was also tested in more demanding applications such as the material nonlinearity [39] and the large deformation analysis [40].

1.3 Thesis Purpose

In 2005 a dilemma emerged. The numerical method used until that point [41-46], the Element Free Galerkin Method (EFGM), was declining in interest and in fields of application, as well as all the others meshless methods based on approximation functions. It was urgent or to embrace a new emergent meshless method or to develop a new one. Hence a decision was made: the development of a new meshless method. Therefore the Natural Neighbour Radial Point Interpolation Method (NNRPIM) was created. In 2006 the work finally started to produce promising results, and the method was finally extended to several engineering fields. From then on, ten articles in reviewed publications were accepted and complimented, as well as more than thirty conferences communications on the subjected were presented in this period. However, this was only the visible result of the work developed along this period. The numerical tools developed are numerous and the acquired experience is intangible.

During the development of this work several programs in MATLAB and FORTRAN language were built. From an early stage, the MATLAB programs were developed to test and calibrate the several parameters of the NNRPIM, for the several engineering applications. Only when all the

processes reach an optimized level, the package could be added to the global NNRPIM analysis program in FORTRAN language.

This global NNRPIM analysis program is a original code, which contains more than 37.000 code lines, and is able to solve various engineering problems, such as:

- i.* Static and dynamic linear problems.
- ii.* Nonlinear static problems. Considering large deformation analysis and inelastic materials.
- iii.* Structural topology problems.
- iv.* Crack evolution path.
- v.* Bone density optimization problems.

The program permits to analyse the problem using:

- i.* the three-dimensional deformation theory,
- ii.* the plain stress and the plain strain two-dimensional deformation theory,
- iii.* several plate shear deformation theories, such as the Reissner-Mindlin deformation theory, the third-order shear deformation theory and the unconstrained third-order shear deformation theory.

The program allows the user to choose the material disposition along the solid domain, as well as the principal material orientation for the case of an anisotropic analysis. The program reads external mesh files generated in the commercial software FEMAP. Nonetheless it also possesses an algorithm which permits the user to create the mesh discretizing the problem domain without using the FEMAP. In order to visualize the obtained results the program produces data files which can be open in the FEMAP software or in the MATLAB, permitting to visually study and understand the deformation field and the stress distribution along the solid domain. Besides the pre-processing phase and the pos-processing phase the global NNRPIM analysis program is independent from any other commercial software.

1.4 Thesis Arrangement

This thesis is divided in eight chapters. In the first chapter it is introduced the thesis purposed. A general overview on the meshless method world is shown and the used meshless method is briefly presented, as well as the

most relevant meshless methods.

In the second chapter the mechanical fundamentals are developed. The continuum formulation is presented, along with the used weak form formulation and the discrete system equations obtained for each solid mechanical problem. The elastoplastic constitutive equations are shown, and in the end of the chapter the linear and the nonlinear solution algorithms used are presented.

The NNRPIM is fully presented in the third chapter. The nodal connectivity, the integration mesh and the interpolation function construction are shown. In this chapter it is possible to observe the NNRPIM patch test results, which permits to gauge the most important parameters in the NNRPIM interpolation functions, allowing to improve and stabilize the meshless method.

The fourth chapter is dedicated to the two-dimensional and the three-dimensional linear analysis. Static and dynamic problems are solved and important conclusions are made, permitting advances and improvements in the NNRPIM analysis.

The nonlinear analysis can be found in the fifth chapter. First it is study the material nonlinearity, solids exhibiting elastoplastic material properties are analysed and the results are compared with the available literature. Afterwards the large deformation analysis, considering material elasticity, is studied. In the final part of the chapter an example which combines the elastoplastic analysis with the large deformation is presented.

The mechanobiology analysis can be found in the sixth chapter, first the basic concepts of the bone biology are introduced. Then, the bone biologic mechanical laws are presented, as well as the developed law for this work. In the end the bone remodelling algorithm is presented.

In the seventh chapter one can find mechanobiologic applications, where the proposed material law and remodelling algorithm are applied. Two-dimensional and three-dimensional benchmark examples are used to validate the bone trabecular remodelling algorithm. The calcaneus bone and the femoral bone are simulated. In both examples the obtained trabecular bone architecture is in very good agreement with the real bone X-ray images. The last chapter is dedicated to the conclusions and future developments on the subject.

Chapter 2

2 Mechanical Fundamentals

In this chapter the mechanical fundamentals behind the numerical applications presented in this work are developed. Firstly it is present a brief exposition of the used continuum formulation, where the solid kinematics and constitutive equations are shown. Following it is presented the used weak form and the consequent generated discrete equation system. Next, the dynamic analysis equations are presented and elastoplastic basic concepts are introduced. At the end of the chapter the linear solution algorithms for solving linear equations systems and eigenvalues problems are shown, as well as the nonlinear solution algorithms used and the stress returning algorithm for material nonlinear problems.

2.1 Continuum Formulation

The continuum mechanics is the foundation of the nonlinear numerical analysis. Solids and structures subjected to loads or forces become stressed. The stresses lead to strains, which can be interpreted as deformations or relative displacements. Solid Mechanics and Structural Mechanics deals, for a given solid and boundary condition (external forces and displacements constrains), with the relationship between stress and strain and the relationship between strain and displacements [47-49]. Solids can show different behaviours, depending on the solid material stress-strain curve. In this work only elastic and plastic materials are considered. In elastic materials the deformation in the solid caused by loading disappears fully with the unloading. In plastic materials a residual deformation (which can not be naturally recovered) remains after the total unload process. The material properties on the solid can also be anisotropic, i.e., the material property varies with the direction [50]. On an anisotropic material, the deformation caused by a load applied in a given direction causes a different deformation if the same load is applied in a distinct direction. Composite laminates are generally constituted by layers of anisotropic material. There

are many material constants to be considered and defined in order to fully describe an anisotropic material, which is the reason why so often the engineering problems reduce the analysis to an isotropic material analysis. Isotropic materials are a special case of anisotropic materials, where only two independent material properties need to be known, the Young modulus and the Poisson ratio.

In this section the rigid solid motion and deformation are described, with emphasis on rotation, which plays an important role in nonlinear continuum mechanics. Also the concepts of strain and stress in nonlinear mechanics are introduced. The equilibrium and the constitutive equations are presented afterwards.

2.1.1 Kinematics

The general motion of a deformable body is represented in figure 1. The body, in the initial position $t=0$, is considered to be an assemblage of material particles, labelled by the coordinates $\mathbf{X}$, with respect to the Cartesian basis $\mathbf{e}$. The current position of a particle is defined at time t by the coordinates $\mathbf{x}$.

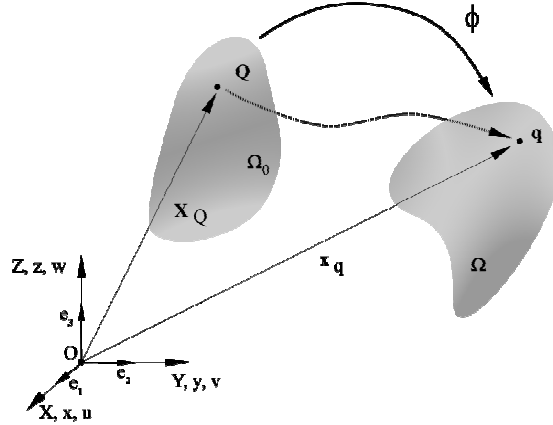


figure 1 – General motion of a deformable body.

The motion can be mathematically described by a mapping function ϕ between initial and current particle positions,

$$\mathbf{x} = \phi(\mathbf{X}, t) \quad (1)$$

It is considered the material description, the Lagrangian description, since the variation of the solid deformation is described with respect to the initial coordinates $\mathbf{X}$, at time $t=0$.

2.1.1.1 Deformation gradient

The deformation gradient $\mathbf{F}$, is a key quantity in finite deformation analysis, since it is involved in all equations relating quantities before deformation (initial configuration) with the correspondent quantities after the finite deformation (current configuration). The deformation gradient tensor $\mathbf{F}$ can be defined as,

$$\mathbf{F} = \frac{\partial \phi}{\partial \mathbf{X}} = \nabla \phi \quad (2)$$

Alternatively to equation (1) the motion can be expressed by,

$$\mathbf{x} = \mathbf{x}(\mathbf{X}, t) \quad (3)$$

which permits the deformation gradient to be written as,

$$\mathbf{F} = \frac{\partial \mathbf{x}}{\partial \mathbf{X}} \quad (4)$$

As so, for a three-dimensional deformation problem the deformation gradient tensor of an initial material position $\mathbf{X} = \{X \ Y \ Z\}$ in respect to a current material position $\mathbf{x} = \{x \ y \ z\}$ can be presented as,

$$\mathbf{F} = \begin{bmatrix} \frac{\partial x}{\partial X} & \frac{\partial x}{\partial Y} & \frac{\partial x}{\partial Z} \\ \frac{\partial y}{\partial X} & \frac{\partial y}{\partial Y} & \frac{\partial y}{\partial Z} \\ \frac{\partial z}{\partial X} & \frac{\partial z}{\partial Y} & \frac{\partial z}{\partial Z} \end{bmatrix} \quad (5)$$

The determinant of the $\mathbf{F}$ is denoted by J and is called the ‘Jacobian determinant’,

$$J = \det(\mathbf{F}) \quad (6)$$

The Jacobian determinant can be used to relate integral of a given functional f in the current and in the initial configuration by,

$$\int_{\Omega} f(\mathbf{x}, t) d\Omega = \int_{\Omega_0} f(\phi(\mathbf{X}, t), t) J d\Omega_0 \quad (7)$$

2.1.1.2 Strain

Consider the change of the scalar product of the two elemental vectors from $d\mathbf{X}_1 = \mathbf{Q}_1 - \mathbf{P}$ and $d\mathbf{X}_2 = \mathbf{Q}_2 - \mathbf{P}$, initial configuration, to $d\mathbf{x}_1 = \mathbf{q}_1 - \mathbf{p}$ and $d\mathbf{x}_2 = \mathbf{q}_2 - \mathbf{p}$, current configuration, as a general measure of deformation. Where $\mathbf{Q}_1$ and $\mathbf{Q}_2$ are two material particles in the neighbourhood of a material particle $\mathbf{P}$ for the initial configuration and $\mathbf{q}_1$ and $\mathbf{q}_2$ and $\mathbf{p}$ the same respective material particles in the current configuration. Equation (4) permits the following relations $d\mathbf{x}_1 = \mathbf{F} d\mathbf{X}_1$ and $d\mathbf{x}_2 = \mathbf{F} d\mathbf{X}_2$, and the spatial scalar product $d\mathbf{x}_1 \cdot d\mathbf{x}_2$ can be found in terms of the material vectors $d\mathbf{X}_1$ and $d\mathbf{X}_2$ as,

$$d\mathbf{x}_1 \cdot d\mathbf{x}_2 = d\mathbf{X}_1 \cdot \mathbf{F}^T \mathbf{F} d\mathbf{X}_2 \quad (8)$$

The right Cauchy-Green deformation tensor is defined as,

$$\mathbf{C} = \mathbf{F}^T \mathbf{F} \quad (9)$$

Which operates directly on the material vectors $d\mathbf{X}_1$ and $d\mathbf{X}_2$. Alternatively the initial scalar product $d\mathbf{X}_1 \cdot d\mathbf{X}_2$ can be obtained in terms of spatial vectors $d\mathbf{x}_1$ and $d\mathbf{x}_2$ using the left Cauchy-Green deformation tensor $\mathbf{b}$,

$$\mathbf{b} = \mathbf{F} \mathbf{F}^T \quad (10)$$

The change in scalar product can now be found in terms of the material vectors $d\mathbf{X}_1$ and $d\mathbf{X}_2$ and the Lagrange or Green strain tensor $\mathbf{E}$ can be defined as,

$$\frac{1}{2} (d\mathbf{x}_1 \cdot d\mathbf{x}_2 - d\mathbf{X}_1 \cdot d\mathbf{X}_2) = d\mathbf{X}_1 \cdot \mathbf{E} d\mathbf{X}_2 \quad (11)$$

where the strain tensor $\mathbf{E}$ is,

$$\mathbf{E} = \frac{1}{2} (\mathbf{C} - \mathbf{I}) \quad (12)$$

2.1.1.3 Polar decomposition

The tensor $\mathbf{F}$ can be expressed as the product of the orthogonal rotation tensor $\mathbf{R}$ by the symmetric stretch tensor $\mathbf{U}$,

$$\mathbf{F} = \mathbf{R}\mathbf{U} \quad (13)$$

where,

$$\mathbf{R}^T \mathbf{R} = \mathbf{I} \quad \text{and} \quad \mathbf{U} = \mathbf{U}^T \quad (14)$$

Such decomposition is called, polar decomposition, and Equation (9) can be expressed as,

$$\mathbf{C} = \mathbf{F}^T \mathbf{F} = \mathbf{U}^T \mathbf{R}^T \mathbf{R} \mathbf{U} = \mathbf{U}^T \mathbf{I} \mathbf{U} = \mathbf{U} \mathbf{U} \quad (15)$$

In order to actually obtain $\mathbf{U}$ from equation (15) it is first necessary to evaluate the principal directions of $\mathbf{C}$, represented by the eigenvectors set $\mathbf{W}_i$ and the correspondent eigenvalues λ_i^2 , with $i = \{1, 2, 3\}$ for the three-dimensional case. In this manner $\mathbf{C}$ can be defined as,

$$\mathbf{C} = \sum_{i=1}^3 \lambda_i^2 \mathbf{W}_i \otimes \mathbf{W}_i \quad (16)$$

since the eigenvectors $\mathbf{W}_i$ are in fact orthogonal unit vectors, because $\mathbf{C} = \mathbf{C}^T$. As so, with equations (15) and (16) it is possible to write the material stretch tensor $\mathbf{U}$ as,

$$\mathbf{U} = \sum_{i=1}^3 \lambda_i \mathbf{W}_i \otimes \mathbf{W}_i \quad (17)$$

Once the stretch tensor $\mathbf{U}$ is known, the rotation tensor $\mathbf{R}$ can be obtained without difficulty from equation (13),

$$\mathbf{R} = \mathbf{F} \mathbf{U}^{-1} \quad (18)$$

2.1.1.4 Stress

In a large deformation analysis a body can experience a large rotation and/or a large strain. The defined stress terms together with the obtained

strain terms make it possible to express the virtual work as an integral over the known body volume, expressing in this manner the change in the body configuration. Both strain tensor and stress tensor are referred to the same deformed state. The Cauchy stress tensor is defined as $\mathbf{\Lambda}$ is a symmetric tensor and it represents the stresses of the current configuration, for the three-dimensional case it can be defined as,

$$\mathbf{\Lambda} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \quad (19)$$

In this work it is used the Voigt notation, since the development of fourth order tensors is less practical. In Voigt notation the tensors are expressed in column vectors, so the stress tensor $\mathbf{\Lambda}$ is reduced to the stress vector $\boldsymbol{\sigma}$,

$$\boldsymbol{\sigma} = \{\sigma_{xx} \quad \sigma_{yy} \quad \sigma_{zz} \quad \sigma_{xy} \quad \sigma_{yz} \quad \sigma_{zx}\}^T \quad (20)$$

and the strain tensor $\mathbf{E}$ to the strain vector $\boldsymbol{\varepsilon}$,

$$\boldsymbol{\varepsilon} = \{\varepsilon_{xx} \quad \varepsilon_{yy} \quad \varepsilon_{zz} \quad \varepsilon_{xy} \quad \varepsilon_{yz} \quad \varepsilon_{zx}\}^T \quad (21)$$

The use of vectors is more practical in the programming process.

2.1.1.5 Principal stress

Another way of describing the Cauchy stress tensor, which completely defines the stress state in an interest point, is by,

$$\mathbf{\Lambda} = \begin{bmatrix} \mathbf{t}^{(\hat{e}_1)} \\ \mathbf{t}^{(\hat{e}_2)} \\ \mathbf{t}^{(\hat{e}_3)} \end{bmatrix} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \quad (22)$$

where $\hat{e}_1$, $\hat{e}_2$ and $\hat{e}_3$ are the versors of the coordinate system and $\mathbf{t}^{(\hat{e}_i)}$ is the stress vector on a plane normal to $\hat{e}_i$ passing through the interest point, figure 2(a). Following Cauchy's stress theorem, if the stress vectors of three orthogonal planes, with a common point, are known, then the stress vector on any other plane passing through that point can be found through the coordinate transformation equations [51]. Thus the stress vector $\mathbf{t}^{(n)}$ in a point belonging to an inclined plane, figure 2(b), can be defined by,

$$\mathbf{t}^{(n)} = \mathbf{n} \cdot \boldsymbol{\sigma}_{ij} = \begin{bmatrix} n_1 & n_2 & n_3 \end{bmatrix} \cdot \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \quad (23)$$

where $\mathbf{n}$ is the inclined plane normal vector. The relation in equation (23) leads to the transformation rule of the stress tensor. The initial stress tensor $\boldsymbol{\sigma}_{ij}$, defined in the $\mathbf{x}_i$ coordinate system, can be transformed in a new stress tensor $\boldsymbol{\sigma}'_{ij}$, defined in another $\mathbf{x}'_i$ coordinate system by the relation,

$$\boldsymbol{\Lambda}' = \mathbf{A} \boldsymbol{\Lambda} \mathbf{A}^T \quad (24)$$

being $\mathbf{A}$ the rotation matrix. Developing equation (24),

$$\begin{bmatrix} \sigma'_{xx} & \sigma'_{xy} & \sigma'_{xz} \\ \sigma'_{yx} & \sigma'_{yy} & \sigma'_{yz} \\ \sigma'_{zx} & \sigma'_{zy} & \sigma'_{zz} \end{bmatrix} = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \begin{bmatrix} a_{11} & a_{21} & a_{31} \\ a_{12} & a_{22} & a_{32} \\ a_{13} & a_{23} & a_{33} \end{bmatrix} \quad (25)$$

The a_{ij} coefficients can be understood as the projection of the $\mathbf{x}'_i$ coordinate system versors in the $\mathbf{x}_i$ coordinate system versors. Therefore, the angle between the versors of each coordinate system can be defined as,

$$\gamma_{ij} = \cos^{-1}(a_{ij}) \quad (26)$$

Through equation (23) and figure 3 it is possible to comprehend better the physical meaning of the a_{ij} coefficients and the respective angles.

Let P be an interest point of a considered stressed body. There are at least three planes, orthogonal with each others, crossing P where the corresponding stress vector is normal to the plane. These planes are called principal planes and the normal vectors of each plane are called principal directions. The stress vectors are parallel to the plane normal vectors and are called principal stresses.

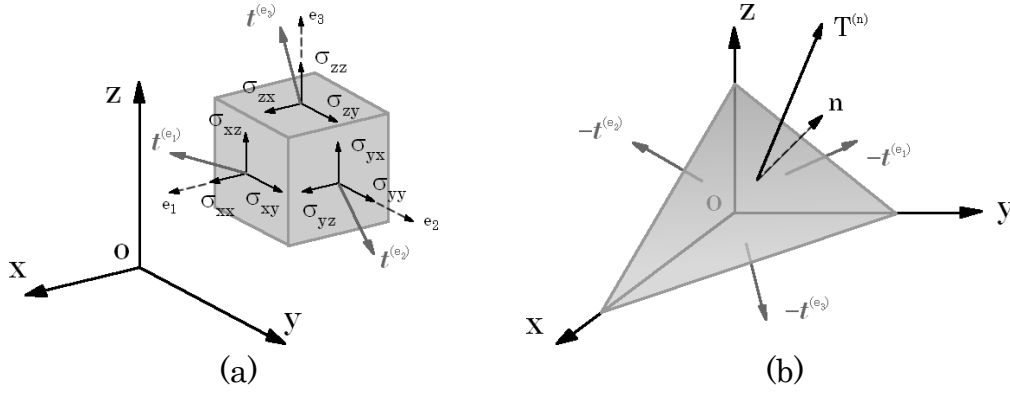


figure 2 – (a) Three-dimensional stress components. (b) Stress vector acting on a plane with normal vector $\mathbf{n}$.

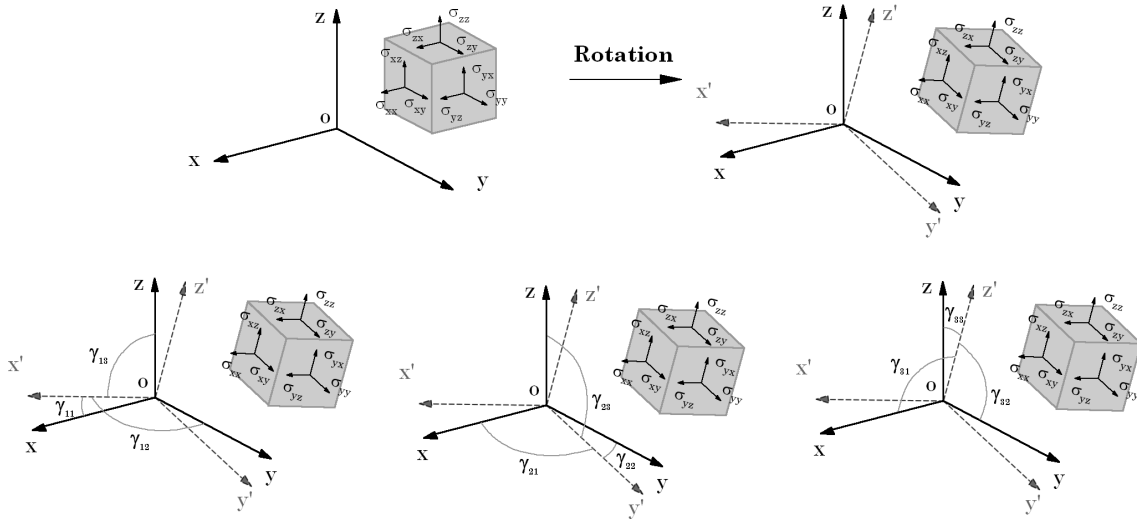


figure 3 – Stress tensor transformation and respective angles.

The stress tensor is a physical quantity, independent of the coordinate system chosen to represent it. Therefore there are certain invariants associated with it which are also independent of the coordinate system. Being a second order tensor the stress tensor has associated three independent invariant quantities. One set of such invariants are the principal stresses of the stress tensor, which are just the eigenvalues of the stress tensor. Their direction vectors are the principal directions or eigenvectors. A stress vector parallel to the normal vector $\mathbf{n}$ is given by,

$$\mathbf{t}^{(n)} = \lambda \mathbf{n} = \sigma_n \mathbf{n} \quad (27)$$

being λ a constant of proportionality, and in this particular case the magnitude of σ_n , the principal stress in $\mathbf{n}$ direction. Knowing $t_i^{(n)} = \sigma_{ij} n_j$ and

$n_i = \delta_{ij} n_j$, where δ_{ij} is the delta Kronecker, the following development can be made,

$$t_i^{(n)} = \lambda n_i \Rightarrow \sigma_{ij} n_j = \lambda \delta_{ij} n_j \Rightarrow (\sigma_{ij} - \lambda \delta_{ij}) n_j = 0 \quad (28)$$

which is a homogeneous system, three linear equations for three n_j unknowns. To obtain the n_j non-zero solution, the matrix determinant must be equal to zero,

$$|\sigma_{ij} - \lambda \delta_{ij}| = \begin{vmatrix} \sigma_{xx} - \lambda & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} - \lambda & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} - \lambda \end{vmatrix} = 0 \quad (29)$$

which leads to the following cubic equation,

$$|\sigma_{ij} - \lambda \delta_{ij}| = -\lambda^3 + I_1 \lambda^2 - I_2 \lambda + I_3 = 0 \quad (30)$$

being I_1 , I_2 and I_3 the stress invariants,

$$I_1 = \sigma_{kk} \quad (31)$$

$$I_2 = \frac{1}{2} (\sigma_{ii} \sigma_{jj} - \sigma_{ij} \sigma_{ji}) \quad (32)$$

$$I_3 = \det(\sigma_{ij}) \quad (33)$$

The three roots $\lambda_1 = \sigma_1$, $\lambda_2 = \sigma_2$ and $\lambda_3 = \sigma_3$ of equation (30) are the eigenvalues or principal stresses, which are unique. Therefore the stress invariants have always the same value regardless of the orientation of the chosen coordinate system. For each eigenvalue, there is for n_j a non-trivial solution on equation (28). These n_j solutions, called eigenvectors, are the principal directions, which defines the plane where the respective stress acts. Applying equation (25),

$$\begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix} = \begin{bmatrix} n_{11} & n_{12} & n_{13} \\ n_{21} & n_{22} & n_{23} \\ n_{31} & n_{32} & n_{33} \end{bmatrix} \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \begin{bmatrix} n_{11} & n_{21} & n_{31} \\ n_{12} & n_{22} & n_{32} \\ n_{13} & n_{23} & n_{33} \end{bmatrix} \quad (34)$$

The principal stresses and principal directions characterize the stress in P and are independent of the orientation of the coordinate system.

2.1.2 Constitutive Equations

The following relation between the stress rate and the strain rate is assumed,

$$d\boldsymbol{\sigma} = \mathbf{c} \, d\boldsymbol{\varepsilon} \quad (35)$$

The material matrix is defined by $\mathbf{c}$ and if material nonlinear relations exists between $\boldsymbol{\sigma}$ and $\boldsymbol{\varepsilon}$, then $\mathbf{c} = \mathbf{c}^{\text{ep}}$. With equation (35) the following relation can be established,

$$d\boldsymbol{\varepsilon} = \mathbf{c}^{-1} \, d\boldsymbol{\sigma} \quad (36)$$

being $\mathbf{s} = \mathbf{c}^{-1}$ and defined for the three-dimensional case as,

$$\mathbf{s} = \begin{bmatrix} \frac{1}{E_{xx}} & -\frac{\nu_{yx}}{E_{yy}} & -\frac{\nu_{zx}}{E_{zz}} & 0 & 0 & 0 \\ -\frac{\nu_{xy}}{E_{xx}} & \frac{1}{E_{yy}} & -\frac{\nu_{zy}}{E_{zz}} & 0 & 0 & 0 \\ -\frac{\nu_{xz}}{E_{xx}} & -\frac{\nu_{yz}}{E_{yy}} & \frac{1}{E_{zz}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \quad (37)$$

The material matrix $\mathbf{c}$ is obtained by inverting matrix $\mathbf{s}$, which is defined for an anisotropic material. The elements on matrix $\mathbf{s}$ are obtained experimentally. E_{ii} is the Young modulus in direction i , ν_{ij} is the Poisson ratio which characterizes the deformation rate in direction j when a force is applied in direction i , G_{ij} is the shear modulus which characterizes the variation angle between directions i and j . Due to symmetry the following relation can be established,

$$E_i v_{ji} = E_j v_{ij} \quad (38)$$

For the two-dimensional case the plane stress and plane strain [51] deformation theory assumptions are assumed.

In the case of an anisotropic material, it is possible to rotate the material matrix c and orientate the material directions with a vector. Consider a known vector $\mathbf{n}$ in the Euclidean space $\mathbb{R}^3$, figure 4, and the respective projections on the coordinate axis and in the oxy plane. As it is known,

$$\theta = \cos^{-1} \left(\frac{\mathbf{n}_{oxy} \cdot \mathbf{n}_{ox}}{\|\mathbf{n}_{oxy}\| \cdot \|\mathbf{n}_{ox}\|} \right) \quad \text{and} \quad \omega = \cos^{-1} \left(\frac{\mathbf{n}_{oxy} \cdot \mathbf{n}}{\|\mathbf{n}_{oxy}\| \cdot \|\mathbf{n}\|} \right) \quad (39)$$

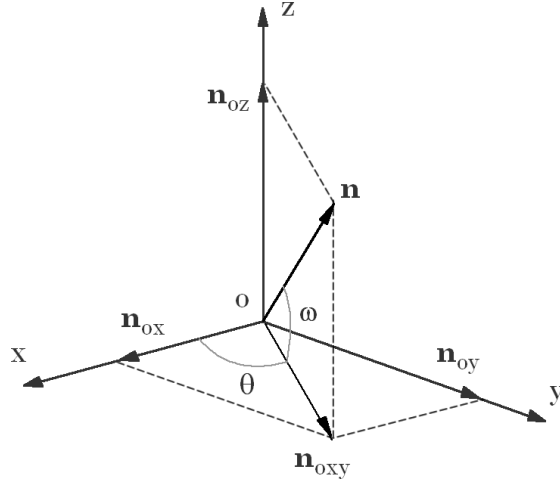


figure 4 – Projection of vector $\mathbf{n}$ in the coordinate axis and in the oxy plane.

With the obtained angle information it is now possible to rotate the material matrix using the rotational transformation matrix and therefore align the material ox axis with the known vector $\mathbf{n}$. The rotational transformation matrix that permits an anticlockwise rotation along the ox axis of a known angle β can be defined as,

$$\mathbf{T}_{ox} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & \cos^2 \beta & \sin^2 \beta & 0 & -\sin 2\beta & 0 \\ 0 & \sin^2 \beta & \cos^2 \beta & 0 & \sin 2\beta & 0 \\ 0 & 0 & 0 & \cos \beta & 0 & \sin \beta \\ 0 & \sin \beta \cdot \cos \beta & -\sin \beta \cdot \cos \beta & 0 & \cos^2 \beta - \sin^2 \beta & 0 \\ 0 & 0 & 0 & -\sin \beta & 0 & \cos \beta \end{bmatrix} \quad (40)$$

Along the oy axis,

$$\mathbf{T}_{oy} = \begin{bmatrix} \cos^2 \beta & 0 & \sin^2 \beta & 0 & 0 & \sin 2\beta \\ 0 & 1 & 0 & 0 & 0 & 0 \\ \sin^2 \beta & 0 & \cos^2 \beta & 0 & 0 & -\sin 2\beta \\ 0 & 0 & 0 & \cos \beta & -\sin \beta & 0 \\ 0 & 0 & 0 & \sin \beta & \cos \beta & 0 \\ -\sin \beta \cdot \cos \beta & 0 & \sin \beta \cdot \cos \beta & 0 & 0 & \cos^2 \beta - \sin^2 \beta \end{bmatrix} \quad (41)$$

And along the oz axis,

$$\mathbf{T}_{oz} = \begin{bmatrix} \cos^2 \beta & \sin^2 \beta & 0 & -\sin 2\beta & 0 & 0 \\ \sin^2 \beta & \cos^2 \beta & 0 & \sin 2\beta & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ \sin \beta \cdot \cos \beta & -\sin \beta \cdot \cos \beta & 0 & \cos^2 \beta - \sin^2 \beta & 0 & 0 \\ 0 & 0 & 0 & 0 & \cos \beta & -\sin \beta \\ 0 & 0 & 0 & 0 & \sin \beta & \cos \beta \end{bmatrix} \quad (42)$$

The material matrix after rotation can be defined as,

$$\mathbf{c}^{current} = [\mathbf{T}_{oz}]_{\theta}^T [\mathbf{T}_{oy}]_{\omega}^T \mathbf{c}^{initial} [\mathbf{T}_{oy}]_{\omega} [\mathbf{T}_{oz}]_{\theta} \quad (43)$$

2.2 Weak Form

The strong form system equations are the partial differential system equations governing the studied physic phenomenon. In contrast, the weak form requires a weaker consistency on the adopted approximation (or interpolation) functions. The ideal would be obtaining the exact solution from strong form system equations, however this is usually an extremely difficult task in complex practical engineering problems. Formulations based on weak forms are able to produce stable algebraic system equations and to give a discretized system of equations which leads to more accurate results. These are the reasons why so many prefer the weak form to obtain the approximated solution.

In this work the discrete equation system is obtained using the Galerkin weak form, which is a variational method [52]. In the NNRPIM the discrete system equation is obtained similarly with the FEM, with some differences inherent to the meshless approach. The discrete equations for the static and the dynamic approach are developed and shown for the distinct deformation theories used on this work.

2.2.1 Galerkin Weak Form

Consider the solid with a domain Ω bounded by Γ , figure 5. The continuous solid surface on which the external forces $\bar{\mathbf{t}}$ are applied is denoted as Γ_t (natural boundary) and the surface where the displacements are constrained is denoted as Γ_u (essential boundary).

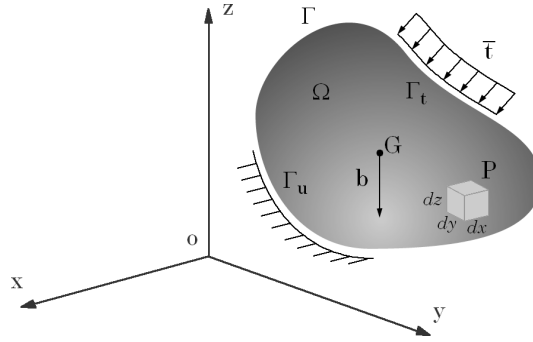


figure 5 – Continuous solid subject to volume forces and external forces.

The Galerkin weak form is a variational principle based on the energy principle. Of all possible displacement configurations satisfying the compatibility conditions, the essential boundary conditions (kinematical and displacement) and the initial and final time conditions, the real solution correspondent configuration is the one which minimizes the Lagrangian functional L ,

$$L = T - U + W_f \quad (44)$$

being T the kinetic energy, U is the strain energy and W_f is the work produced by the external forces. The kinetic energy is defined by,

$$T = \frac{1}{2} \int_{\Omega} \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} d\Omega \quad (45)$$

where the solid volume is defined by Ω and $\dot{\mathbf{u}}$ is the displacement first derivative in order to time, i.e. the velocity. The solid mass density is defined by ρ . The strain energy, for elastic materials, is defined as,

$$U = \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega \quad (46)$$

being $\boldsymbol{\varepsilon}$ the strain vector and $\boldsymbol{\sigma}$ the stress vector. The work produced by the external forces can be expressed as,

$$W_f = \int_{\Omega} \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \quad (47)$$

where $\mathbf{u}$ represents the displacement, $\mathbf{b}$ the body forces and Γ_t the traction boundary where the external forces $\bar{\mathbf{t}}$ are applied. By substitution the Lagrangian functional L can be rewritten as,

$$L = \frac{1}{2} \int_{\Omega} \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} d\Omega - \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \quad (48)$$

and then minimized,

$$\delta \int_{t_1}^{t_2} \left[\frac{1}{2} \int_{\Omega} \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} d\Omega - \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \right] dt = 0 \quad (49)$$

Moving the variation operator δ inside the integrals,

$$\int_{t_1}^{t_2} \left[\frac{1}{2} \int_{\Omega} \delta(\rho \dot{\mathbf{u}}^T \dot{\mathbf{u}}) d\Omega - \frac{1}{2} \int_{\Omega} \delta(\boldsymbol{\varepsilon}^T \boldsymbol{\sigma}) d\Omega + \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \right] dt = 0 \quad (50)$$

Since all operations are linear, changing the order of operation does not affect the result. In the first term of equation (50) the time integral can be moved inside the spatial integral,

$$\int_{t_1}^{t_2} \left[\frac{1}{2} \int_{\Omega} \delta(\rho \dot{\mathbf{u}}^T \dot{\mathbf{u}}) d\Omega \right] dt = \frac{1}{2} \int_{\Omega} \left[\int_{t_1}^{t_2} \delta(\rho \dot{\mathbf{u}}^T \dot{\mathbf{u}}) dt \right] d\Omega \quad (51)$$

Using the chain rule of variation and then the scalar property, the integral can be rewritten as,

$$\int_{t_1}^{t_2} \delta(\rho \dot{\mathbf{u}}^T \dot{\mathbf{u}}) dt = \rho \int_{t_1}^{t_2} (\delta \dot{\mathbf{u}}^T \dot{\mathbf{u}} + \dot{\mathbf{u}}^T \delta \dot{\mathbf{u}}) dt = 2\rho \int_{t_1}^{t_2} (\delta \dot{\mathbf{u}}^T \dot{\mathbf{u}}) dt \quad (52)$$

And knowing that $\dot{\mathbf{u}}^T \dot{\mathbf{u}}$ is a scalar and $\dot{\mathbf{u}} = \partial \mathbf{u} / \partial t$,

$$\int_{t_1}^{t_2} (\delta \dot{\mathbf{u}}^T \dot{\mathbf{u}}) dt = \int_{t_1}^{t_2} \left(\frac{\partial \delta \mathbf{u}^T}{\partial t} \frac{\partial \mathbf{u}}{\partial t} \right) dt \quad (53)$$

Then integrating by parts, with respect to time,

$$\int_{t_1}^{t_2} \left(\frac{\partial \delta \mathbf{u}^T}{\partial t} \frac{\partial \mathbf{u}}{\partial t} \right) dt = \int_{t_1}^{t_2} \left(-\delta \mathbf{u}^T \frac{\partial^2 \mathbf{u}}{\partial t^2} \right) dt + \left(\delta \mathbf{u}^T \frac{\partial \mathbf{u}}{\partial t} \right) \Big|_{t_1}^{t_2} \quad (54)$$

Notice that $\mathbf{u}$ satisfies, by imposition, the conditions at the initial time, t_1 , and final time, t_2 , leading to a null $\delta \mathbf{u}^T$ at t_1 and t_2 . Therefore the last term in equation (54) vanishes. Considering the last development and switching the integration order again, equation (51) becomes,

$$\int_{t_1}^{t_2} \left[\frac{1}{2} \int_{\Omega} \delta (\rho \dot{\mathbf{u}}^T \dot{\mathbf{u}}) d\Omega \right] dt = - \int_{t_1}^{t_2} \left[\rho \int_{\Omega} (\delta \mathbf{u}^T \ddot{\mathbf{u}}) \right] dt \quad (55)$$

being $\ddot{\mathbf{u}} = \partial^2 \mathbf{u} / \partial t^2$ the acceleration. The second term on equation (50) can also be developed. The integrand function in the second integral term can be written as follows,

$$\delta (\boldsymbol{\varepsilon}^T \boldsymbol{\sigma}) = \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} + \boldsymbol{\varepsilon}^T \delta \boldsymbol{\sigma} \quad (56)$$

as the two terms in equation (56) are in fact scalars, the transpose does not affect the result, as so,

$$\boldsymbol{\varepsilon}^T \delta \boldsymbol{\sigma} = (\boldsymbol{\varepsilon}^T \delta \boldsymbol{\sigma})^T = \delta \boldsymbol{\sigma}^T \boldsymbol{\varepsilon} \quad (57)$$

Using the constitutive equation $\boldsymbol{\sigma} = \mathbf{c} \boldsymbol{\varepsilon}$ and the symmetric property of the material matrix, $\mathbf{c}^T = \mathbf{c}$, it is possible to write,

$$\delta \boldsymbol{\sigma}^T \boldsymbol{\varepsilon} = \delta (\mathbf{c} \boldsymbol{\varepsilon})^T \boldsymbol{\varepsilon} = \delta \boldsymbol{\varepsilon}^T \mathbf{c}^T \boldsymbol{\varepsilon} = \delta \boldsymbol{\varepsilon}^T \mathbf{c} \boldsymbol{\varepsilon} = \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} \quad (58)$$

Therefore equation (56) becomes,

$$\delta (\boldsymbol{\varepsilon}^T \boldsymbol{\sigma}) = 2 \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} \quad (59)$$

simplifying the second term in equation (50),

$$\int_{t_1}^{t_2} \left[\frac{1}{2} \int_{\Omega} \delta(\boldsymbol{\epsilon}^T \boldsymbol{\sigma}) d\Omega \right] dt = \int_{t_1}^{t_2} \left[\int_{\Omega} \delta \boldsymbol{\epsilon}^T \boldsymbol{\sigma} d\Omega \right] dt \quad (60)$$

Equation (50) now becomes,

$$\int_{t_1}^{t_2} \left[-\rho \int_{\Omega} (\delta \mathbf{u}^T \ddot{\mathbf{u}}) d\Omega - \int_{\Omega} \delta \boldsymbol{\epsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \right] dt = 0 \quad (61)$$

To satisfy equation (61) for all possible choices of $\mathbf{u}$ the integrand of the time integration has to be null, leading to the following expression,

$$-\rho \int_{\Omega} (\delta \mathbf{u}^T \ddot{\mathbf{u}}) d\Omega - \int_{\Omega} \delta \boldsymbol{\epsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma = 0 \quad (62)$$

This last equation is known as the ‘Galerkin weak form’, which can also be viewed as the principle of virtual work. The principle of virtual work states that if a solid body is in equilibrium, the virtual work produced by the body inner stresses and the body applied external forces should vanish when the body experiments a virtual displacement. Considering the stress-strain relation, $\boldsymbol{\sigma} = \mathbf{c} \boldsymbol{\epsilon}$, and the strain-displacement relation, $\boldsymbol{\epsilon} = \mathbf{L} \mathbf{u}$, equation (62) can be rearranged in the following expression,

$$\int_{\Omega} \delta (\mathbf{L} \mathbf{u})^T \mathbf{c} (\mathbf{L} \mathbf{u}) d\Omega - \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega - \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma + \int_{\Omega} \rho (\delta \mathbf{u}^T \ddot{\mathbf{u}}) d\Omega = 0 \quad (63)$$

which is the generic Galerkin weak form written in terms of displacement, very useful in solid mechanical problems. In static problems the fourth term of equation (63) disappears.

2.3 Discrete System Equation

The discrete equations for the NNRPIM are obtained from the principle of virtual work by using the NNRPIM interpolator functions for the trial and test functions. The domain Ω is discretized in a nodal mesh, and each node possesses an ‘influence-domain’ which imposes the nodal connectivity between the neighbouring nodes. The NNRPIM trial function $\mathbf{u}(\mathbf{x}_I)$ is given by,

$$\mathbf{u}(\mathbf{x}_I) = \sum_{i=1}^n \phi_i(\mathbf{x}_I) u_i \quad (64)$$

being $\varphi_i(\mathbf{x}_I)$ the NNRPIM C^∞ interpolation function and u_i are the nodal displacements of the n nodes belonging to the ‘influence-domain’ of interest node $\mathbf{x}_I$. The NNRPIM interpolation function satisfies the condition,

$$\varphi_i(\mathbf{x}_j) = \delta_{ij} \quad (65)$$

where δ_{ij} is the delta Kronecker, $\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$. Following equation (64), the test functions (or virtual displacements) are defined as,

$$d\mathbf{u}(\mathbf{x}_I) = \sum_{i=1}^n \varphi_i(\mathbf{x}_I) du_i \quad (66)$$

where du_i are the nodal values for the test function.

2.3.1 Principle of virtual work

The NNRPIM formulation is established in terms of a weak form of the differential equation under consideration, equation (62). In the solid mechanics context this implies the use of the virtual work equation.

$$L = \int_{\Omega} \boldsymbol{\sigma} d\boldsymbol{\epsilon} d\Omega - \int_{\Omega} \mathbf{b} \cdot d\mathbf{u} d\Omega - \int_{\Gamma} \bar{\mathbf{t}} \cdot d\mathbf{u} d\Gamma + \rho \int_{\Omega} (d\mathbf{u}^T \ddot{\mathbf{u}}) d\Omega = 0 \quad (67)$$

The virtual deformation $d\boldsymbol{\epsilon}$ is defined by,

$$d\boldsymbol{\epsilon} = \bar{\mathbf{B}} d\mathbf{u} \quad (68)$$

where $\bar{\mathbf{B}}$ is the deformation matrix. Thus the virtual work of the first term in equation (67), using equation (68) can be expressed as,

$$L_1 = \int_{\Omega} d\mathbf{u}^T \bar{\mathbf{B}}^T \boldsymbol{\sigma} d\Omega \quad (69)$$

The strain vector can be divided in two parts, the linear part and the nonlinear part,

$$\boldsymbol{\epsilon} = \boldsymbol{\epsilon}_0 + \boldsymbol{\epsilon}_{NL} \quad (70)$$

which can also be presented as,

$$\boldsymbol{\varepsilon} = \underbrace{\mathbf{L} \boldsymbol{\theta}}_{\boldsymbol{\varepsilon}_0} + \underbrace{\frac{1}{2} \mathbf{A} \boldsymbol{\theta}}_{\boldsymbol{\varepsilon}_{NL}} = \left(\mathbf{L} + \frac{1}{2} \mathbf{A} \right) \boldsymbol{\theta} \quad (71)$$

Matrix $\mathbf{L}$ is defined as,

$$\mathbf{L} = \begin{bmatrix} \mathbf{e}_1^T & \mathbf{0} & \mathbf{0} \\ [1 \times 3] & [1 \times 3] & [1 \times 3] \\ \mathbf{0} & \mathbf{e}_2^T & \mathbf{0} \\ [1 \times 3] & [1 \times 3] & [1 \times 3] \\ \mathbf{0} & \mathbf{0} & \mathbf{e}_3^T \\ [1 \times 3] & [1 \times 3] & [1 \times 3] \\ \mathbf{e}_2^T & \mathbf{e}_1^T & \mathbf{0} \\ [1 \times 3] & [1 \times 3] & [1 \times 3] \\ \mathbf{0} & \mathbf{e}_3^T & \mathbf{e}_2^T \\ [1 \times 3] & [1 \times 3] & [1 \times 3] \\ \mathbf{e}_3^T & \mathbf{0} & \mathbf{e}_1^T \\ [1 \times 3] & [1 \times 3] & [1 \times 3] \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 & 1 & 0 & 0 \end{bmatrix} \quad (72)$$

Being $\mathbf{e}_i$ the coordinate i director column vector,

$$\mathbf{I} = [\mathbf{e}_1 \quad \mathbf{e}_2 \quad \mathbf{e}_3] \quad (73)$$

The column vector $\boldsymbol{\theta}$ is defined by,

$$\boldsymbol{\theta} = \mathbf{G} \mathbf{u} \quad (74)$$

The geometric matrix $\mathbf{G}$ is defined by,

$$\mathbf{G}^T = \begin{bmatrix} \frac{\partial \varphi}{\partial x} & 0 & 0 & \frac{\partial \varphi}{\partial y} & 0 & 0 & \frac{\partial \varphi}{\partial z} & 0 & 0 \\ 0 & \frac{\partial \varphi}{\partial x} & 0 & 0 & \frac{\partial \varphi}{\partial y} & 0 & 0 & \frac{\partial \varphi}{\partial z} & 0 \\ 0 & 0 & \frac{\partial \varphi}{\partial x} & 0 & 0 & \frac{\partial \varphi}{\partial y} & 0 & 0 & \frac{\partial \varphi}{\partial z} \end{bmatrix} \quad (75)$$

which produces the following column vector $\boldsymbol{\theta}$,

$$\boldsymbol{\theta} = \begin{bmatrix} \boldsymbol{\theta}_x \\ \boldsymbol{\theta}_y \\ \boldsymbol{\theta}_z \end{bmatrix} \quad \text{being} \quad \boldsymbol{\theta}_\xi = \begin{bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial v}{\partial \xi} \\ \frac{\partial w}{\partial \xi} \end{bmatrix} \quad (76)$$

The current configuration displacement is considered in matrix $\mathbf{A}$, which corresponds in equation (71) to the actualized component.

$$\mathbf{A} = \begin{bmatrix} \boldsymbol{\theta}_x^T & \mathbf{0}_{[1 \times 3]} & \mathbf{0}_{[1 \times 3]} \\ \mathbf{0}_{[1 \times 3]} & \boldsymbol{\theta}_y^T & \mathbf{0}_{[1 \times 3]} \\ \mathbf{0}_{[1 \times 3]} & \mathbf{0}_{[1 \times 3]} & \boldsymbol{\theta}_z^T \\ \boldsymbol{\theta}_y^T & \boldsymbol{\theta}_x^T & \mathbf{0}_{[1 \times 3]} \\ \mathbf{0}_{[1 \times 3]} & \boldsymbol{\theta}_z^T & \boldsymbol{\theta}_y^T \\ \boldsymbol{\theta}_z^T & \mathbf{0}_{[1 \times 3]} & \boldsymbol{\theta}_x^T \end{bmatrix} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} & \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} & 0 & 0 & 0 & \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \end{bmatrix} \quad (77)$$

The deformation matrix $\bar{\mathbf{B}}$, dependent of $\mathbf{u}$, can be defined as,

$$\bar{\mathbf{B}} = \mathbf{B}_0 + \mathbf{B}_{\text{NL}}(\mathbf{u}) \quad (78)$$

since it varies with the deformation of the solid. The linear part of the deformation matrix is represented by $\mathbf{B}_0$ and the nonlinear contribution by $\mathbf{B}_{\text{NL}}$. For the three-dimensional case,

$$\mathbf{B}_0^T = \begin{bmatrix} \frac{\partial \phi}{\partial x} & 0 & 0 & \frac{\partial \phi}{\partial y} & 0 & \frac{\partial \phi}{\partial z} \\ 0 & \frac{\partial \phi}{\partial y} & 0 & \frac{\partial \phi}{\partial x} & \frac{\partial \phi}{\partial z} & 0 \\ 0 & 0 & \frac{\partial \phi}{\partial z} & 0 & \frac{\partial \phi}{\partial y} & \frac{\partial \phi}{\partial x} \end{bmatrix} \quad (79)$$

and

$$\mathbf{B}_{\text{NL}} = \mathbf{A} \mathbf{G} \quad (80)$$

The non linear deformation is actualized through matrix $\mathbf{A}$, which contains the displacement current configuration.

2.3.2 *Stiffness matrix*

The tangential stiffness matrix $\mathbf{K}_{\text{T}}$ can be determined considering the variation of the virtual work of equation (69), in order to the generalized displacements $d\mathbf{u}$,

$$dL_1 = d \left[\int_{\Omega} \bar{\mathbf{B}}^{\text{T}} \boldsymbol{\sigma} d\Omega \right] \quad (81)$$

which can be developed as,

$$dL_1 = \int_{\Omega} d\bar{\mathbf{B}}^{\text{T}} \boldsymbol{\sigma} d\Omega + \int_{\Omega} \bar{\mathbf{B}}^{\text{T}} d\boldsymbol{\sigma} d\Omega = \mathbf{K}_{\text{T}} d\mathbf{u} \quad (82)$$

Using equation (35) and equation (68) the following relation is obtained,

$$d\boldsymbol{\sigma} = \mathbf{c} \bar{\mathbf{B}} d\mathbf{u} \quad (83)$$

In the deformation matrix $\bar{\mathbf{B}}$ only the nonlinear part $d\mathbf{B}_{\text{NL}}$ is dependent of $\mathbf{u}$, equation (78), thus $d\bar{\mathbf{B}} = d\mathbf{B}_{\text{NL}}$ and therefore,

$$dL_1 = \int_{\Omega} d\mathbf{B}_{\text{NL}}^{\text{T}} \boldsymbol{\sigma} d\Omega + \int_{\Omega} \bar{\mathbf{B}}^{\text{T}} \mathbf{c} \bar{\mathbf{B}} d\Omega \quad (84)$$

Where the stiffness matrix can be presented as,

$$\mathbf{K}_{\text{T}} = \mathbf{K}_{\sigma} + \mathbf{K}_0 + \mathbf{K}_{\text{NL}} \quad (85)$$

Being,

$$\mathbf{K}_{\sigma} = \int_{\Omega} d\mathbf{B}_{\text{NL}}^{\text{T}} \boldsymbol{\sigma} d\Omega \quad (86)$$

$$\mathbf{K}_0 = \int_{\Omega} \mathbf{B}_0^{\text{T}} \mathbf{c} \mathbf{B}_0 d\Omega \quad (87)$$

$$\mathbf{K}_{\text{NL}} = \int_{\Omega} \left(\mathbf{B}_0^{\text{T}} \mathbf{c} \mathbf{B}_{\text{NL}} + \mathbf{B}_{\text{NL}}^{\text{T}} \mathbf{c} \mathbf{B}_{\text{NL}} + \mathbf{B}_{\text{NL}}^{\text{T}} \mathbf{c} \mathbf{B}_0 \right) d\Omega \quad (88)$$

The initial stress matrix or geometric matrix $\mathbf{K}_{\sigma}$ is defined as,

$$\mathbf{K}_{\sigma} d\mathbf{u} = \int_{\Omega} \mathbf{G}^{\text{T}} d\mathbf{A}^{\text{T}} \boldsymbol{\sigma} d\Omega \quad (89)$$

The variation of matrix $\mathbf{A}$ in order to $\mathbf{u}$ can be defined as,

$$d\mathbf{A}^{\text{T}} = \begin{bmatrix} d\boldsymbol{\theta}_x & \mathbf{0}_{[3 \times 1]} & \mathbf{0}_{[3 \times 1]} & d\boldsymbol{\theta}_y & \mathbf{0}_{[3 \times 1]} & d\boldsymbol{\theta}_z \\ \mathbf{0}_{[3 \times 1]} & d\boldsymbol{\theta}_y & \mathbf{0}_{[3 \times 1]} & d\boldsymbol{\theta}_x & d\boldsymbol{\theta}_z & \mathbf{0}_{[3 \times 1]} \\ \mathbf{0}_{[3 \times 1]} & \mathbf{0}_{[3 \times 1]} & d\boldsymbol{\theta}_z & \mathbf{0}_{[3 \times 1]} & d\boldsymbol{\theta}_y & d\boldsymbol{\theta}_x \end{bmatrix} \quad (90)$$

As so, the term $d\mathbf{A}^{\text{T}} \boldsymbol{\sigma}$ can be represented as,

$$d\mathbf{A}^{\text{T}} \boldsymbol{\sigma} = \begin{bmatrix} \sigma_{xx} \mathbf{I}_{[3 \times 3]} & \tau_{zx} \mathbf{I}_{[3 \times 3]} & \tau_{yz} \mathbf{I}_{[3 \times 3]} \\ \tau_{zx} \mathbf{I}_{[3 \times 3]} & \sigma_{yy} \mathbf{I}_{[3 \times 3]} & \tau_{xy} \mathbf{I}_{[3 \times 3]} \\ \tau_{yz} \mathbf{I}_{[3 \times 3]} & \tau_{xy} \mathbf{I}_{[3 \times 3]} & \sigma_{zz} \mathbf{I}_{[3 \times 3]} \end{bmatrix} d\boldsymbol{\theta} = \mathbf{Z} d\boldsymbol{\theta} = \mathbf{Z} \mathbf{G} d\mathbf{u} \quad (91)$$

and therefore equation (86) can be presented as,

$$\mathbf{K}_{\sigma} = \int_{\Omega} \mathbf{G}^{\text{T}} \mathbf{Z} \mathbf{G} d\Omega \quad (92)$$

Therefore, the initial stress matrix $\mathbf{K}_{\sigma}$ takes into consideration the actualized stress field.

2.3.3 Mass matrix

The virtual work of the last term in equation (67) can be expressed and developed as,

$$dL_4 = d \left[\rho \int_{\Omega} \left(d\mathbf{u}^{\text{T}} \ddot{\mathbf{u}} \right) d\Omega \right] = \mathbf{M} d\ddot{\mathbf{u}} \quad (93)$$

where the mass matrix $\mathbf{M}$ can be defined as,

$$\mathbf{M} = \int_{\Omega} \mathbf{H}^T \boldsymbol{\rho} \mathbf{H} d\Omega \quad (94)$$

being $\mathbf{H}$ the interpolation function matrix for the interest point i defined as,

$$\mathbf{H}_i = \varphi_i \mathbf{I} \quad (95)$$

Where φ_i is the interpolation function for interest node i and $\mathbf{I}$ is the identity matrix defined in equation (73). The density diagonal matrix can be defined as,

$$\boldsymbol{\rho} = \rho \mathbf{I} \quad (96)$$

being ρ the solid material density.

2.3.4 Force vector

The virtual work of the middle terms in equation (67) can be expressed and developed as,

$$dL_2 = d \left[\int_{\Omega} \mathbf{b} \cdot d\mathbf{u} d\Omega \right] = \mathbf{f}_b \quad (97)$$

and

$$dL_3 = d \left[\int_{\Gamma} \bar{\mathbf{t}} \cdot d\mathbf{u} d\Gamma \right] = \mathbf{f}_{\bar{\mathbf{t}}} \quad (98)$$

being the total force vector $\mathbf{f}$ defined as,

$$\mathbf{f}_b + \mathbf{f}_{\bar{\mathbf{t}}} = \mathbf{f} \quad (99)$$

Thus the total force vector $\mathbf{f}$ can be developed in a matrix form,

$$\mathbf{f} = \int_{\Omega} \mathbf{H}^T \mathbf{b} d\Omega + \int_{\Gamma} \mathbf{H}^T \bar{\mathbf{t}} d\Gamma \quad (100)$$

2.3.5 Imposition of the boundary conditions

The boundary conditions can be imposed directly as in the FEM, once the NNRPIM interpolation functions possess the delta Kronecker property. The continuum analysis involves two types of boundary conditions, the essential boundary conditions (displacement related) and the natural boundary conditions (force related). Neglecting damping effects and assuming that the matricial form of the equilibrium equations resulting from virtual work expression, equation (67), can be presented as,

$$\mathbf{K}\mathbf{u} + \mathbf{M}\ddot{\mathbf{u}} = \mathbf{f} \quad (101)$$

Such equation can be rewritten as,

$$\begin{bmatrix} \mathbf{K}_{cc} & \mathbf{K}_{cd} \\ \mathbf{K}_{dc} & \mathbf{K}_{dd} \end{bmatrix} \begin{bmatrix} \mathbf{u}_c \\ \mathbf{u}_d \end{bmatrix} + \begin{bmatrix} \mathbf{M}_{cc} & \mathbf{M}_{cd} \\ \mathbf{M}_{dc} & \mathbf{M}_{dd} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{u}}_c \\ \ddot{\mathbf{u}}_d \end{bmatrix} = \begin{bmatrix} \mathbf{f}_c \\ \mathbf{f}_d \end{bmatrix} \quad (102)$$

where $\mathbf{u}_c$ are the unknown displacements and $\mathbf{u}_d$ the known, or prescribed, displacements. The vectors $\mathbf{f}_c$ and $\mathbf{f}_d$ correspond respectively to the known applied loads (external and body forces) and to the unknown reactions due the imposed displacement constraints. With the equation (102) it is assumed that the displacement components considered are axial aligned with the prescribed displacements. If this is not the case it is required the identification of all prescribed displacement orientations and transform locally the discrete equilibrium equations to correspond to the global axis. Thus,

$$\mathbf{u} = \mathbf{T} \bar{\mathbf{u}} \quad (103)$$

where $\bar{\mathbf{u}}$ is the vector of nodal point degrees of freedom in the required directions. The transformation matrix $\mathbf{T}$ is defined by equation (104) and figure 6, which is a typical representation of the constrained displacements in 2D and 3D analysis.

$$\mathbf{T}_{2D} = \begin{bmatrix} u_x & v_x \\ u_y & v_y \end{bmatrix} \quad \text{and} \quad \mathbf{T}_{3D} = \begin{bmatrix} u_x & v_x & w_x \\ u_y & v_y & w_y \\ u_z & v_z & w_z \end{bmatrix} \quad (104)$$

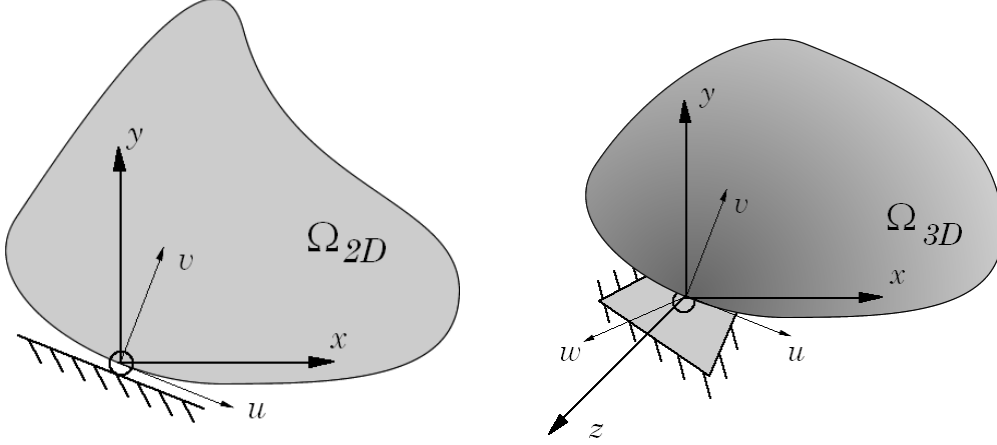


figure 6 – Essential boundary condition non aligned with the global axis.

Using equations (103) and (101) it is possible to write,

$$\bar{\mathbf{K}} \bar{\mathbf{u}} + \bar{\mathbf{M}} \ddot{\bar{\mathbf{u}}} = \bar{\mathbf{f}} \quad (105)$$

Where,

$$\bar{\mathbf{M}} = \mathbf{T}^T \mathbf{M} \mathbf{T} \quad (106)$$

$$\bar{\mathbf{K}} = \mathbf{T}^T \mathbf{K} \mathbf{T} \quad (107)$$

$$\bar{\mathbf{f}} = \mathbf{T}^T \mathbf{f} \quad (108)$$

Notice that the matrix multiplications in equations (106), (107) and (108) involve changes only in those columns and rows of $\mathbf{M}$, $\mathbf{K}$ and $\mathbf{f}$ that are actually affected by the prescribed displacement. In practice, the transformation is effectively carried out on the local level just prior to adding the local matrices to the global assembled matrices.

2.3.6 Dynamic equations

The equilibrium equations governing the linear dynamic response can be represented as in equation (101). The fundamental mathematical method used to solve equation (101) is the separation of variables. In order to change the equilibrium equations to the modal generalized displacements [5] it is proposed the following transformation,

$$\mathbf{u}(t) = \Phi \mathbf{x}(t) \quad (109)$$

where Φ is a $m \times m$ square matrix containing m spatial vectors independent of the time variable t , $\mathbf{x}(t)$ is a time dependent vector and $m = 2N$ for the 2D case and $m = 3N$ for the 3D case, being N the total number of nodes in the problem domain. From equation (109) also follows that $\dot{\mathbf{u}}(t) = \Phi \dot{\mathbf{x}}(t)$ and $\ddot{\mathbf{u}}(t) = \Phi \ddot{\mathbf{x}}(t)$. The components of $\mathbf{u}(t)$ are called generalized displacements. For which the solution can be presented in the form,

$$\mathbf{u}(t) = \phi \sin(\omega(t - t_0)) \quad (110)$$

being ϕ the vector of order m , t the time variable, t_0 the constant initial time and ω is the vibration frequency vector. Substituting (110) into (101) the generalized eigenproblem is obtained, from which ϕ and ω must be determined,

$$K \phi = \omega^2 M \phi \quad (111)$$

Equation (111) yields the m eigensolutions,

$$\begin{cases} K \phi_1 = \omega_1^2 M \phi_1 \\ K \phi_2 = \omega_2^2 M \phi_2 \\ : \\ K \phi_m = \omega_m^2 M \phi_m \end{cases} \quad (112)$$

The vector ϕ_i is called the i^{th} mode shape vector and ω_i is the corresponding frequency of vibration. Defining a matrix Φ whose columns are the eigenvectors ϕ_i ,

$$\Phi = [\phi_1 \quad \phi_2 \quad \dots \quad \phi_m] \quad (113)$$

and a diagonal matrix Ω which stores the eigenvalues ω_i ,

$$\Omega = \begin{bmatrix} \omega_1^2 & & & \\ & \omega_2^2 & & \\ & & \dots & \\ & & & \omega_m^2 \end{bmatrix} \quad (114)$$

the m solutions can be written as,

$$\mathbf{K} \Phi = \mathbf{M} \Phi \Omega \quad (115)$$

It is required that the space functions satisfy the following stiffness and mass orthogonality conditions,

$$\Phi^T \mathbf{K} \Phi = \Omega \quad \text{and} \quad \Phi^T \mathbf{M} \Phi = \mathbf{I} \quad (116)$$

After substituting equations (109) and its time derivatives into equation (101) and pre-multiplying it by Φ^T , the equilibrium equation that corresponds to the modal generalized displacement is obtained,

$$\ddot{\mathbf{x}}(t) + \Omega \mathbf{x}(t) = \Phi^T \mathbf{F}(t) \quad (117)$$

The initial conditions on $\mathbf{x}(t)$ are obtained using (109) and considering the the $\mathbf{M}$ -orthonormality of Φ at time $t = 0$,

$$\begin{cases} \mathbf{x}_0 = \Phi^T \mathbf{M} \mathbf{u}_0 \\ \dot{\mathbf{x}}_0 = \Phi^T \mathbf{M} \dot{\mathbf{u}}_0 \end{cases} \quad (118)$$

Equation (117) can be represented as m individual equations of the form,

$$\begin{cases} \ddot{x}_i(t) + \omega_i^2 x_i(t) = f_i(t) \\ f_i(t) = \phi_i^T \mathbf{F}(t) \end{cases} \quad (119)$$

with the initial conditions,

$$\begin{cases} x_i^{t=0} = \phi_i^T \mathbf{M} \mathbf{u}_0 \\ \dot{x}_i^{t=0} = \phi_i^T \mathbf{M} \dot{\mathbf{u}}_0 \end{cases} \quad (120)$$

For the complete response, the solution to all m equations in equation (119) must be calculated and then the modal point displacements are obtained by superposition of the response in each mode.

$$\mathbf{u}(t) = \sum_{i=1}^m \phi_i x_i(t) \quad (121)$$

Therefore the response analysis requires, first, the solution of the eigenvalues and eigenvectors of the problem, equation (111), then the solution of the decoupled equilibrium equations in equation (119) and, finally, the superposition of the response in each eigenvector as expressed in equation (121).

2.3.6.1 Forced Vibrations

In this work when forced vibrations are imposed only three different time-dependent loading conditions are considered, $\mathbf{f}(t) = \mathbf{f} \times g(t)$. A time constant load – **load case A**,

$$g_A(t) = 1 \quad (122)$$

A transient load – **load case B**,

$$\begin{cases} g_B(t) = 1 & \text{if } t \leq t_i \\ g_B(t) = 0 & \text{if } t > t_i \end{cases} \quad (123)$$

And a harmonic load – **load case C**,

$$g_C(t) = \sin(\gamma t) \quad (124)$$

The solution of each equation in equation (119) can be calculated using the Duhamel integral,

$$x_i(t) = \frac{1}{\omega_i} \int_0^t f_i(\tau) \sin(\omega_i(t - \tau)) d\tau + \alpha_i \sin(\omega_i t) + \beta_i \cos(\omega_i t) \quad (125)$$

where α_i and β_i are determined from the initial conditions, equation (120) and $f_i(t) = \phi_i^T \mathbf{f}(t)$. For load case A and load case B the obtained solution is,

$$x_i(t) = \frac{f_i(t)}{\omega_i^2} (1 - \cos(\omega_i t)) + \frac{\dot{x}_i^{t=0}}{\omega_i} \sin(\omega_i t) + x_i^{t=0} \cos(\omega_i t) \quad (126)$$

For load case C the obtained solution is,

$$x_i(t) = \frac{f_i(t)}{\omega_i^2 - \gamma^2} \left(\sin(\gamma t) - \frac{\gamma}{\omega_i} \sin(\omega_i t) \right) \quad (127)$$

2.4 Elastoplastic Constitutive Equations

In the mathematical theory of plasticity the main purpose is to describe the theoretical relationships between stresses and strains for a material with elastoplastic behaviour [53]. The elastoplastic materials are materials exhibiting elastic behaviour until a stress level (with total strain recovery), called yield stress, and beyond the yield stress level start to develop plastic strains. Therefore, once unloaded, the material exhibits a permanent deformation, which can not be naturally recovered, due to the developed plastic strains. The elastoplastic materials can be classified as rate-independent or rate-dependent. In this work only rate-independent elastoplastic materials are considered, for which the stress is independent on the strain rate. The elastoplastic theoretical formulation requires three fundamental concepts,

Yield Criterion, which defines the material yield surface that delineates the material elastic limit for a multiaxial stress state.

Hardening Rule, which describes the evolution equations for internal variables, including the strain hardening which governs the evolution of the material yield surface.

Plastic Flow Rule, which is related with the plastic functional and defines the direction of the incremented plastic deformation vector.

In the elastoplastic deformation theory the total strain can be divided in two parts: a elastic strain component, $\boldsymbol{\epsilon}^e$, and a plastic strain component, $\boldsymbol{\epsilon}^p$.

$$\boldsymbol{\epsilon} = \boldsymbol{\epsilon}^e + \boldsymbol{\epsilon}^p \quad (128)$$

Before initializing the plastic flow the material respects the generalized Hook law, equation (35). However, once the stress level reaches the stress limit imposed by the *yield criterion* the linear Hook law is no longer valid and *plastic flow rule* starts to govern the stress state. In addition, beyond this stress level the yield surface begins to evolve due to the plastic strain component, governed by the *hardening rule*.

2.4.1 Yield Criterion

The yield criterion defines mathematically the stress elastic behaviour limit [54]. The yield criterion is a functional of the stress and the material loading history, which can be generically defined as,

$$F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, k) = f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa) - Y(\kappa) = 0 \quad (129)$$

Where $f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa)$ is the considered yield function, dependent on the stress $\boldsymbol{\sigma}$, on the strain plastic component, $\boldsymbol{\varepsilon}^p$, and a hardening parameter κ . The material yield stress is defined with $Y(\kappa)$. In this phase, for simplicity sake, it is assumed that the material yield stress $Y(\kappa)$ is independent of the hardening parameter κ and the material exhibits isotropic properties, which means that the functional F becomes independent on the plastic strain,

$$F(\boldsymbol{\sigma}) = f(\boldsymbol{\sigma}) - Y = 0 \quad (130)$$

In this work it was used only the Von-Mises yield criterion [55]. Von-Mises suggested that the yielding occurs when the second deviatoric stress invariant, equation (32), reaches a critical value [56],

$$I_2 = \frac{\sigma_Y(\kappa)}{\sqrt{3}} \quad (131)$$

Which can be rewritten as,

$$(\sigma_{xx} - \sigma_{yy})^2 + (\sigma_{yy} - \sigma_{zz})^2 + (\sigma_{zz} - \sigma_{xx})^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2) = 2\sigma_Y^2 \quad (132)$$

or,

$$(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 = 2\sigma_Y^2 \quad (133)$$

From which can be obtained,

$$\bar{\sigma} = \sqrt{3I_2} = \sigma_Y \quad (134)$$

being $\bar{\sigma}$ the effective stress. The Von-Mises criterion is the generalization of Tresca criterion [57]. There are numerous yielding criterions, depending on the material used in the analysis [58].

2.4.2 Hardening Rule

After initial yielding, the stress level at which further plastic deformation occurs may be dependent on the current degree of plastic strain. This phenomenon is called strain hardening. Therefore, the yield surface will vary at each stage of the plastic deformation with the plastic strain. In figure 7 some models are presented describing the material strain hardening.

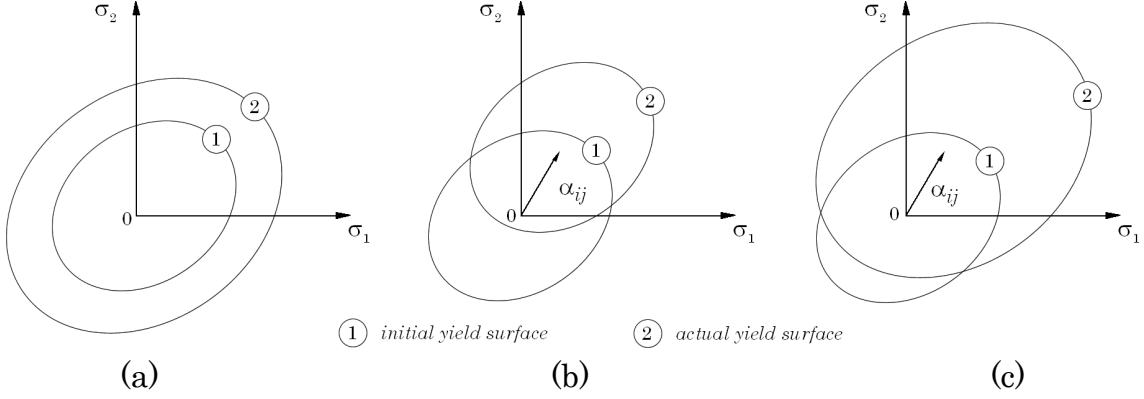


figure 7 – Hardening rule geometric representation. (a) Isotropic hardening. (b) Kinematic hardening. (c) Strain hardening models mixture.

A perfect plastic material is defined when the initial yield surface never suffers any translation, rotation or expansion. If the subsequent yield surface is a uniform expansion of the original yield function, then the strain hardening model is isotropic, figure 7(a). In the case of the subsequent yield surface preserve its shape and orientation but suffers a translation the model is known as kinematics, figure 7(b). Based on the rotation, translation and expansion of the original yield function several models can be constructed, figure 7(c). Thus, the yield criterion can be rewritten,

$$F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}, \kappa) = f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}, \kappa) - k^2(\kappa) = 0 \quad (135)$$

where k^2 represents the yield surface size, $f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}, \kappa)$ the yield surface shape and $\boldsymbol{\alpha}$ the yield surface centre. The hardening rule used in this work is the isotropic rule, which is the simplest one. Therefore equation (135) simplifies and becomes,

$$f(\boldsymbol{\sigma}) = k^2(\kappa) \quad (136)$$

Applying the Von-Mises criterion,

$$F(\boldsymbol{\sigma}, k) = 3I_2 - \sigma_Y^2(\kappa) \quad (137)$$

Where $\sigma_Y^2(\kappa)$ can be associated with the effective plastic strain $\bar{\epsilon}^p$ through the material stress/strain uniaxial curve. The progressive development of the yield surface can be defined by relating the yield stress k to the plastic deformation by means of the hardening parameter κ [56], which can be obtain using the effective plastic strain, $\bar{\epsilon}^p$,

$$\bar{\epsilon}^p = \int d\bar{\epsilon}^p = \int \sqrt{\frac{2}{3}} d\epsilon^p d\epsilon^p \quad (138)$$

Then the hardening parameter κ is defined as,

$$\kappa = \bar{\epsilon}^p \quad (139)$$

Stress states for which $f = k$ represent plastic stress states, while elastic behaviour is characterized by $f < k$. At the plastic state the material behaviour becomes conditioned by the stress variation of f ,

$$df = \left(\frac{\partial f}{\partial \sigma} \right)^T d\sigma \quad (140)$$

being $\partial f / \partial \sigma$ the gradient of f , therefore orthogonal to the yield surface, figure 8.

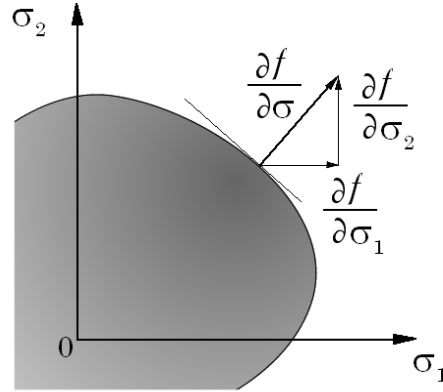


figure 8 – Orthogonality condition in a stress two-dimensional space.

Then, if $\partial f < 0$ elastic unloading occurs and the stress point returns inside the yield surface. If $\partial f = 0$ neutral loading occurs and the stress point remains on the yield surface. If $\partial f > 0$ plastic loading occurs and the stress point remains on the expanding yield surface.

2.4.3 Plastic Flow

Considering the plastic potential theory [53], derived by Von Mises from the plastic flow theory of Prandtl-Reuss for a perfect plastic rigid material, a material is in a plastic flow state when $\partial f \geq 0$. Introducing the concept of plastic potential, $g(\boldsymbol{\sigma}, \boldsymbol{\epsilon}^p, \kappa)$, the following flow rule is defined,

$$d\boldsymbol{\epsilon}^p = d\lambda \frac{\partial g}{\partial \boldsymbol{\sigma}} \quad (141)$$

It is assumed that the plastic strain increment is proportional to the stress gradient of the plastic potential. Where $d\lambda > 0$ is a proportional constant called plastic multiplier. The gradient of the plastic potential surface $\partial g / \partial \boldsymbol{\sigma}$ defines the direction of the plastic strain incremental vector, $d\boldsymbol{\epsilon}^p$, and the magnitude is defined by the plastic multiplier. In the associated plasticity theory the plastic potential can be coincident with the yield function $f(\boldsymbol{\sigma}, \boldsymbol{\epsilon}^p, \kappa)$, therefore equation (141) becomes [54],

$$d\boldsymbol{\epsilon}^p = d\lambda \frac{\partial f}{\partial \boldsymbol{\sigma}} \quad (142)$$

This equation is termed the *normality condition* since $\partial f / \partial \boldsymbol{\sigma}$ is a vector normal to the yield surface at the stress point under consideration, figure 8. It is the orthogonality between the plastic flow vector and the yield surface that warrant the solution uniqueness.

2.4.4 Elastoplastic small strain constitutive model

The small-strain formulation can be matricially summarized in the following items,

- i. Additive decomposition of rate strain into elastic and plastic parts:

$$\dot{\boldsymbol{\epsilon}} = \dot{\boldsymbol{\epsilon}}^e + \dot{\boldsymbol{\epsilon}}^p \quad (143)$$

- ii. Stress rate and elastic strain relation:

$$\dot{\boldsymbol{\sigma}} = \mathbf{c} : \dot{\boldsymbol{\epsilon}}^e = \mathbf{c} : (\dot{\boldsymbol{\epsilon}} - \dot{\boldsymbol{\epsilon}}^p) \quad (144)$$

- iii. The Plastic Prandtl-Reuss flow rule, where $\dot{\lambda}$ is the plastic rate multiplier and $\mathbf{a}$ is the flow vector:

$$\dot{\boldsymbol{\epsilon}}^p = \dot{\lambda} \frac{\partial f}{\partial \boldsymbol{\sigma}} = \dot{\lambda} \mathbf{a} \quad (145)$$

iv. The Von-Mises yield condition [55], which only depends on the second deviatoric stress invariant magnitude I_2 , and of a hardening parameter κ :

$$F(\boldsymbol{\sigma}, \kappa) = 3I_2 - \sigma_Y^2(\kappa) \quad (146)$$

v. The hardening functional, depending on the hardening rule:

$$A = \dot{\lambda}^{-1} \frac{\partial \sigma_Y}{\partial \kappa} \dot{\kappa} \quad (147)$$

vi. The plastic rate parameter:

$$\dot{\lambda} = \frac{\mathbf{a}^T : \mathbf{c} : \dot{\boldsymbol{\epsilon}}}{A + \mathbf{a}^T : \mathbf{c} : \mathbf{a}} \quad (148)$$

vii. Stress rate and strain rate relation:

$$\dot{\boldsymbol{\sigma}} = \mathbf{c}^{ep} : \dot{\boldsymbol{\epsilon}} \quad (149)$$

viii. Continuum elastoplastic tangent modulus:

$$\mathbf{c}^{ep} = \mathbf{c} - \frac{(\mathbf{c} : \mathbf{a}) \otimes (\mathbf{c} : \dot{\boldsymbol{\epsilon}})}{A + \mathbf{a}^T : \mathbf{c} : \mathbf{a}} \quad (150)$$

2.4.5 Stress returning algorithm

In this work the material behaviour is modelled in the form of an incremental relation between the incremental stress vector and the strain increment. In order to force the stress to return to the yield surface the “forward-Euler” procedure [58] presented in figure 9 is considered.

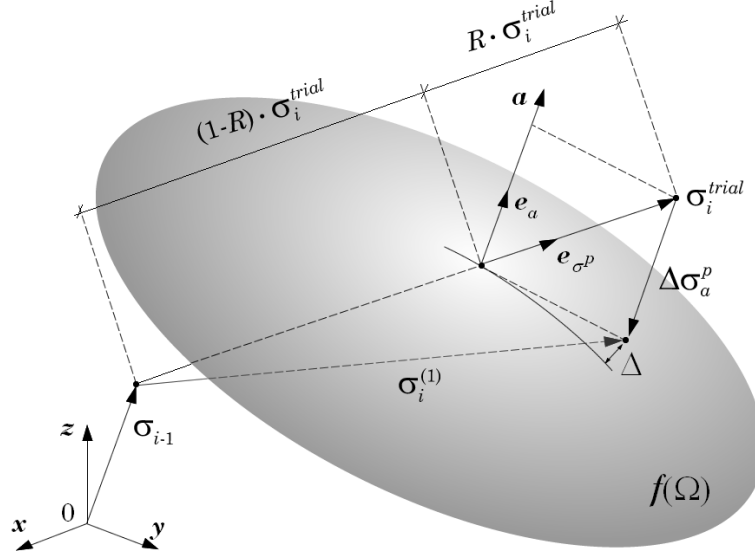


figure 9 – Incremental stress changes at a point in an elastoplastic continuum at initial yield.

A trial stress σ_i^{trial} is obtained and it passes beyond the actualized yield surface $f(\sigma)$. The trial stress is divided in two parts, $(1-R) \cdot \sigma_i^{trial}$ inside the yield surface and $R \cdot \sigma_i^{trial}$ outside the yield surface. Where R is the reduction factor,

$$R = \frac{\bar{\sigma}_i^{trial} - \sigma_Y}{\bar{\sigma}_i^{trial} - \bar{\sigma}_{i-1}} \quad (151)$$

Being $\bar{\sigma}$ the σ effective stress and σ_Y the actualized yield stress. The flow vector $\mathbf{a}$, as is visible, is normal to the yield surface at the intersection of σ_i^{trial} with the yield surface. The part of the stress that passes beyond the yield surface it is projected into the flow vector, originating $\Delta\sigma_a^p$, which is subtracted to $R \cdot \sigma_i^{trial}$, returning the stress to the yield surface. However it is visible and understandable that in fact the yield surface has not been reached yet, thus the process is repeated until $\Delta \cong 0$.

2.5 Linear Solution algorithms

In this work, to solve the static and dynamic linear problems it were used standard solvers. To solve static problems, where the system of linear equations to be solved can be represented as,

$$\mathbf{A}\mathbf{b} = \mathbf{c} \quad (152)$$

the Gaussian elimination algorithm can be applied. The high-performance linear algebra library LAPACK [59] contains a very efficient implementation of this algorithm to solve equation (152). With the Galerkin weak form the unknown displacements $\mathbf{u}$ can be determined directly for a linear elastic problem by solving the equation,

$$\mathbf{K} \mathbf{u} = \mathbf{f} \quad (153)$$

With the NNRPIM the stiffness matrix $\mathbf{K}$ is generally sparse, due to the compact support of the NNRPIM interpolation functions and always symmetric. The solution of equation (153) can only be found if $\mathbf{K}$ is non-singular. Since $\mathbf{K}$ is symmetric and non-singular another linear solution procedure can be applied to equation (153), the $\bar{\mathbf{L}} \bar{\mathbf{D}} \bar{\mathbf{L}}^T$ factorization [5],

$$\mathbf{K} = \bar{\mathbf{L}} \bar{\mathbf{D}} \bar{\mathbf{L}}^T \quad (154)$$

where $\bar{\mathbf{L}}$ is a triangular matrix and $\bar{\mathbf{D}}$ is diagonal. Then, the unknown displacement vector $\mathbf{u}$ can be obtained by using the backward substitution,

$$\bar{\mathbf{L}} \mathbf{v} = \mathbf{f} \quad (155)$$

and,

$$\bar{\mathbf{D}} \bar{\mathbf{L}} \mathbf{u} = \mathbf{v} \quad (156)$$

For larger $\mathbf{K}$ matrices this process is faster than the blind Gaussian elimination algorithm.

To solve eigenproblems, such are the dynamic problems, the Jacobi algorithm is used. To eigenproblem, where the eigenvalues and eigenvectors must be found, can be represented as,

$$(\mathbf{A} - \lambda \mathbf{I}) \mathbf{b} = \mathbf{0} \quad (157)$$

where $\mathbf{b}$ is an eigenvector from the system of linear equations, $\mathbf{A}$, and λ is the correspondent eigenvalue. With a system $\mathbf{A}$ with the size $m \times m$ one should obtain m pairs of eigenvalues and eigenvectors. The Jacobi algorithm is an iterative convergent method [5]. Next the Gauss elimination algorithm and the Jacobi algorithm are presented.

```

% Gauss elimination routine
% A b = c

% example variables
A = [ 1 3 4 ; 2 3 8 ; -1 0 2 ]; % system of linear equations
b = [ -1 3 2 ]; % independent vector
n = 3; % size of matrix A

% definition of matrix MATT
MATT = A;
for jj = 1:n
    MATT(jj,jj+1) = b(jj);
end

% triangular matrix
for jj = 1:n
    for ii = (1+jj):n
        if MATT(ii,jj) ~= 0;
            coef = MATT(ii,jj) / MATT(jj,jj);
            for kk = 1:(n+1)
                MATT(jj,kk) = coef * MATT(jj,kk);
            end
            for kk = 1:(n+1)
                MATT(ii,kk) = MATT(ii,kk) - MATT(jj,kk);
            end
        end
    end
end

% diagonal matrix
for jj = 1:n
    for ii = jj:(n-1)
        coef = MATT(jj,ii+1) / MATT(ii+1,ii+1);
        for kk = 1:(n+1)
            MATT(jj,kk) = MATT(jj,kk) - coef * MATT(ii+1,kk);
        end
    end
end

% identity matrix
for ii = 1:n
    coef = 1 / MATT(ii,ii);
    for kk = 1:(n+1)
        MATT(ii,kk) = coef * MATT(ii,kk);
    end
end

% solution vector c
for ii = 1:n
    c = MATT(ii,n+1);
end

% Jacobi eigenproblem routine
% (A - H) C = 0

% example variables
A = [5 -4 1 0; -4 6 -4 1; 1 -4 6 -4; 0 1 -4 5]; % system of linear equations
n = 4; % size of matrix A

% matrix that will become in the end the diagonal eigenvalue matrix
D = A

% identity matrix that will become in the end the diagonal eigenvector matrix
FI = 0;
for ii = 1:n
    FI(ii,ii) = 1;
end

val = 1; % convergence error
ind = 0; % iterative counter

% Jacobi iterative procedure
while abs(val) > 10^-6 % it stops when the convergence error is lower 10-6
    ind = ind + 1;

    for ii = 1:n
        for jj = 1:n
            if jj > ii

% iterative angle
                teta = (atan(2*DD(ii,jj)/(DD(ii,ii)-DD(jj,jj))))/2;

% identity matrix
                PP = 0;
                for ii = 1:n
                    PP(ii,ii) = 1;
                end

                PP(ii,ii) = cos(teta);
                PP(ii,jj) = -sin(teta);
                PP(jj,ii) = sin(teta);
                PP(jj,jj) = cos(teta);
                DD = PP' * DD * PP;
                FI = FI * PP;
            end
        end
    end

    val = 0;
    for ii = 1:n
        for jj = 1:n
            if jj > ii
                val = val + DD(ii,jj);
            end
        end
    end

    H = D; % diagonal eigenvalue matrix
    C = FI; % eigenvector matrix
end
    
```

(a)

(b)

figure 10 – (a) Gauss elimination algorithm for MATLAB. (b) Jacobi elimination algorithm for MATLAB.

2.6 Nonlinear Solution algorithms

In nonlinear analysis the solution is achieved via an iterative process. In this work the iteration method used is the actualized normal plan method, considering in each new load increment the previous increment tangent stiffness matrix. Such method, which is a variant of the arc-length method, permits the solution of problems where there are the possibility of occurring structural instabilities, as the “snap-through” and “snap-back” phenomenon, figure 11.

These instability phenomena cannot be predicted with standard nonlinear Newton-Raphson solution methods. For such nonlinear solution methods, in the equilibrium curve, once reached point A, the solution would pass directly to point C, neglecting point B, a dynamic solution leap to a new displacement state without load variation. If a control displacement analysis was used the “snap-through” phenomenon would be solved, however the

“snap-back” phenomenon would still occur, a dynamic leap from point D to F, without predicting point E.

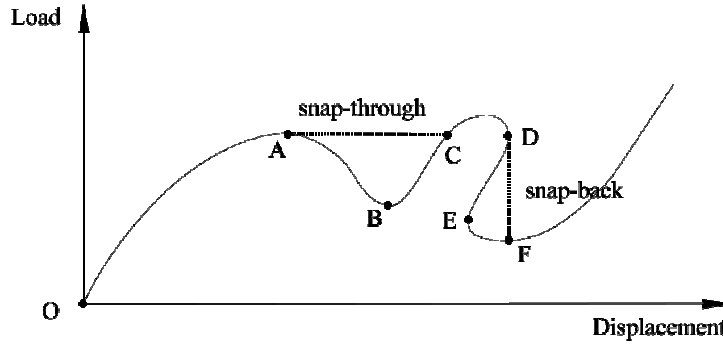


figure 11 – Load/displacement diagram showing instabilities phenomena.

2.6.1 Newton Raphson Method

The solution of a system of equilibrium equations is searched in the static analysis of a structural problem. If the analysis is formulated in displacements of the problem spatial volumetric discretization, the equilibrium equations can be written as,

$$\mathbf{K} \mathbf{u} = \mathbf{f} \quad (158)$$

If the problem to be solved is linear in order to $\mathbf{u}$ the solution of equation (158) is obtained directly by $\mathbf{u} = \mathbf{K}^{-1} \mathbf{f}$. However in the presence of a nonlinear problem the stiffness matrix $\mathbf{K}$ becomes itself $\mathbf{u}$ dependent, making useless the direct solution methods for solving equation (158). In order to overcome this difficulty incremental and iterative methods are adopted, as the Newton-Raphson Method, the most used solution method for nonlinear analysis. Thus the nonlinear equilibrium equation system, for load increment n , can be written as,

$$\Psi({}^n \mathbf{u}) = \Upsilon({}^n \mathbf{u}) - {}^n \mathbf{f} \quad (159)$$

Being ${}^n \mathbf{f}$ the applied external force vector associated with the load increment n , the searched nodal displacement vector is defined by ${}^n \mathbf{u}$ and $\Upsilon({}^n \mathbf{u})$ is the nodal force vector equivalent to the stress field found after applying the load increment n .

$$\Upsilon({}^n\mathbf{u}) = \int_{\Omega} {}^n\bar{\mathbf{B}}^T {}^n\boldsymbol{\sigma} \, d\Omega \quad (160)$$

Vector $\psi({}^n\mathbf{u})$ becomes the residual force vector associated with load increment n . To the residual force vector $\psi({}^n\mathbf{u}^i) \geq \mathbf{0}$ of iteration i corresponds a trial displacement vector ${}^n\mathbf{u}^i$. An improved displacement vector ${}^n\mathbf{u}^{i+1}$ can be obtained using a Taylor series,

$$\psi({}^n\mathbf{u}^{i+1}) = \psi({}^n\mathbf{u}^i) + \left. \frac{\partial \psi(\mathbf{u})}{\partial \mathbf{u}} \right|_i {}^n d\mathbf{u}^i = \mathbf{0} \quad (161)$$

which can be written as,

$$\psi({}^n\mathbf{u}^i) + {}^n\mathbf{K}^i {}^n d\mathbf{u}^i = \mathbf{0} \quad (162)$$

As so, the variation of the displacement ${}^n d\mathbf{u}^i$ can be obtained using,

$${}^n d\mathbf{u}^i = -[{}^n\mathbf{K}^i]^{-1} \psi({}^n\mathbf{u}^i) \quad (163)$$

the tangential stiffness matrix ${}^n\mathbf{K}^i$ can be obtained by,

$${}^n\mathbf{K}^i = \left. \frac{\partial \psi(\mathbf{u})}{\partial \mathbf{u}} \right|_i \quad (164)$$

Thus, an improved solution for the displacement field can be obtained with,

$${}^n\mathbf{u}^{i+1} = {}^n\mathbf{u}^i + {}^n d\mathbf{u}^i \quad (165)$$

The equations (163) and (165) represents the Newton-Raphson solution method for the nonlinear system equation. The schematic description of the method can be found in figure 12 for the case of an one-dimensional system, one degree of freedom. The iterative process is concluded when a certain convergence criterion is reached, achieving the converged solution for increment n .

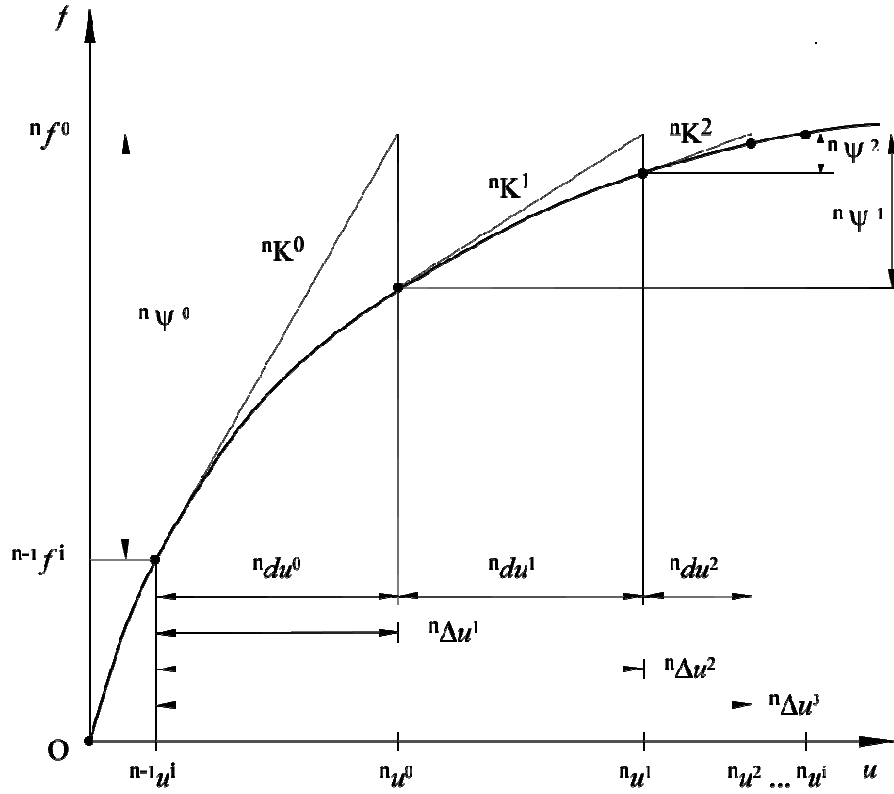


figure 12 – Newton-Raphson Method for an one-dimensional system.

There are several Newton-Raphson method variants, which seek to reduce the computational effort for redefining the tangential stiffness matrix in each iteration step of the incremental process. The most common are probably the *Initial Stiffness Method* (K0) and the *Initial Increment Stiffness Method* (K1).

The K0 method permits a great reduction of the computational cost, since it uses the same stiffness matrix during all the incremental-iterative process, the initial linear elastic stiffness matrix, $\mathbf{K}_0$, determined in the first iteration of the first load increment. However this method presents a low iterative convergence rate when the structure is significantly nonlinear, leading to the divergence of the iterative process. Equation (163) can be then presented as,

$${}^n d\mathbf{u}^i = -[\mathbf{K}_0]^{-1} \psi({}^n \mathbf{u}^i) \quad (166)$$

The K1 method is itself a variant of the K0 method, here the tangential stiffness matrix used is determined in the first iteration of each load increment. This method has the advantage of presenting a fast iterative

convergence and permits more robust analysis of nonlinear structures. Thus the variation of the displacement ${}^n du^i$ can be obtained with,

$${}^n du^i = -[{}^n K_0]^{-1} \psi({}^n u^i) \quad (167)$$

In figure 13 both *Initial Stiffness Method* (K0) and the *Initial Increment Stiffness Method* (K1) are presented.

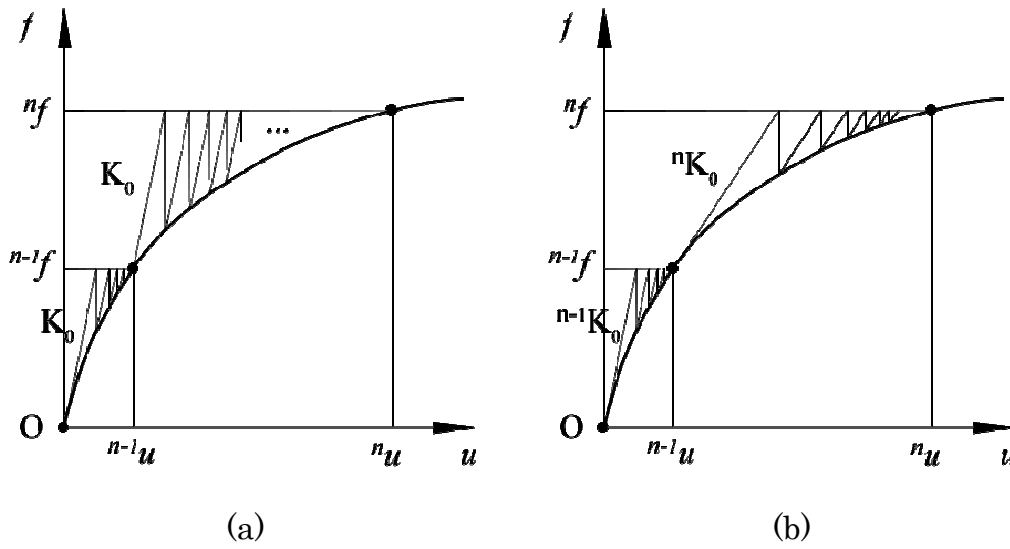


figure 13 – Modified Newton-Rapson Method (a) K0 and (b) K1.

As figure 13 illustrate, for a higher level of nonlinearity the K0 method will need much more iterations in each load increment when compared with the K1 method.

2.6.2 Arc-length method

The instability phenomena presented in figure 11 cannot be predicted using the classical nonlinear Newton-Rapson solution method approach, for those cases the arc-length methods are more adequate. In this work it is used the Orthogonal Actualized Ramm's method, illustrated in figure 14. In this method the iterative process is forced to follow a plane orthogonal to vector ${}^n t^i$. For the simplicity sake the scheme of an one-dimensional system solution is presented in figure 14, reducing the orthogonal plane to a two-dimensional vector ${}^n n^i$, where ${}^n n^i \perp {}^n t^i$.

$$\left[{}^n\Delta\mathbf{u}^i \right]^T {}^n d\mathbf{u}^i + \left({}^n\Delta\lambda^i \quad {}^n d\lambda^i \right) \mathbf{f}^T \mathbf{f} = 0 \quad (170)$$

resulting after development in,

$${}^n d\lambda^i = \frac{-\left[{}^n\Delta\mathbf{u}^i \right]^T {}^n \overline{d\mathbf{u}}^i}{\left[{}^n\Delta\mathbf{u}^i \right]^T {}^n d\mathbf{u}_l^i + \left({}^n\Delta\lambda^i \right) \mathbf{f}^T \mathbf{f}} \quad (171)$$

In this version of the orthogonal actualized Ramm's method the tangential stiffness matrix used is determined in the first iteration of each load increment, as the K1 method. Being so, ${}^n d\mathbf{u}_l^i$ is constant along all the iterative process,

$${}^n d\mathbf{u}_l^i = {}^n \Delta\mathbf{u}^1 = \left[{}^n \mathbf{K}^0 \right]^{-1} {}^n \Delta\lambda^1 \mathbf{f} \quad (172)$$

The vector ${}^n \overline{d\mathbf{u}}^i$ can be obtained with,

$${}^n \overline{d\mathbf{u}}^i = \left[{}^n \mathbf{K}^0 \right]^{-1} {}^n \psi^i \quad (173)$$

And the displacement vector in the current iteration is defined as,

$${}^n d\mathbf{u}^i = \left[{}^n \mathbf{K}^0 \right]^{-1} \left({}^n \Delta\lambda^i \mathbf{f} - {}^n \psi^i \right) \quad (174)$$

The displacement vector is accumulated in each iteration,

$${}^n \Delta\mathbf{u}^{i+1} = {}^n \Delta\mathbf{u}^i + {}^n d\mathbf{u}^i \quad (175)$$

The actualized incremental parameter is obtained with,

$${}^n \Delta\lambda^{i+1} = {}^n \Delta\lambda^i - {}^n d\lambda^i \quad (176)$$

2.6.2.1 Generic arc-length algorithm

The generic arc-length algorithm can be summarized in the following seven steps,

- i. In the beginning of the load incremental process it is determined,

$${}^n\overline{d\mathbf{u}}^0 = \left[{}^n\mathbf{K}^0 \right]^{-1} {}^n\Delta\lambda^1 \mathbf{f} = {}^n\Delta\mathbf{u}^1 = {}^nd\mathbf{u}_I^1 = {}^nd\mathbf{u}^0 \quad (177)$$

note that ${}^nd\mathbf{u}_I^1$ is constant along all the iterative process.

ii. With the obtained displacement field the stress field is determined in each iteration,

$${}^nd\boldsymbol{\sigma}^i = \mathbf{c} \, {}^n\overline{\mathbf{B}}^i \, {}^nd\mathbf{u}^i \quad (178)$$

iii. The residual force vector is obtained,

$${}^n\boldsymbol{\psi}^i = {}^n\Delta\lambda^i \mathbf{f} - \int_{\Omega} {}^n\overline{\mathbf{B}}^i \, {}^nd\boldsymbol{\sigma}^i \, d\Omega \quad (179)$$

iv. The iterative process is concluded when a certain convergence criterion is reached, achieving the converged solution for increment n . It is used a force convergence criterion,

$$e_t = \frac{\| {}^n\boldsymbol{\psi}^i \|}{\| {}^n\Delta\lambda^i \mathbf{f} \|} \quad (180)$$

If $e_t \leq \text{toler}$ the process stops and it moves to the next load increment $n+1$. The *toler* parameter is a scalar within the interval $\text{toler} \in [10^{-4}, 10^{-2}]$.

v. Else if $e_t > \text{toler}$ the iteration process continues and it is determined the ${}^nd\lambda^i$. First ${}^n\overline{d\mathbf{u}}^i$ is obtained,

$${}^n\overline{d\mathbf{u}}^i = \left[{}^n\mathbf{K}^0 \right]^{-1} {}^n\boldsymbol{\psi}^i \quad (181)$$

and then,

$${}^nd\lambda^i = \frac{-\left[{}^n\Delta\mathbf{u}^i \right]^T {}^n\overline{d\mathbf{u}}^i}{\left[{}^n\Delta\mathbf{u}^i \right]^T {}^nd\mathbf{u}_I^i + \left({}^n\Delta\lambda^i \right) \mathbf{f}^T \mathbf{f}} \quad (182)$$

vi. The displacement vector of current iteration ${}^n d\mathbf{u}^i$ is determined, equation (174), and accumulated ${}^n \Delta \mathbf{u}^{i+1}$, equation (175). A new load level of the next iteration is establish ${}^n \Delta \lambda^{i+1}$, equation (176).

vii. With the displacement vector of current iteration determined one can move to step **ii**, and re-start the iterative process until the convergence criterion is reached.

Chapter 3

3 The Meshless Method

In this chapter the Natural Neighbour Radial Point Interpolation Method (NNRPIM) is presented. The chapter starts with an introduction to the generic meshless procedure. Afterwards the natural neighbour concept is explained, along with the nodal connectivity enforcement and integration scheme process. Next the interpolation function construction process is presented. Then it is shown the weak form application for several solid mechanic problems and the NNRPIM flow chart for the linear analysis. This chapter ends with a patch test and some useful conclusions and remarks on the NNRPIM features.

3.1 Meshless Generic Procedure

Most of the meshless approaches respect the following outline. First the problem domain is discretized in a nodal mesh. The nodal density of the mesh, as well as the nodal distribution, affects the method performance. A fine nodal mesh conducts generally to more accurate results, however the computational cost grows with the increase of the node total number. The unbalanced distribution of the nodes discretizing the problem domain could lead to less accurate results. Generally locations with predictable stress concentrations (such as crack tips or constrained boundaries) should have a higher nodal density than locations with a more expected smooth stress distribution [60], figure 15.

After the nodal discretization a background integration mesh is constructed, nodal dependent or nodal independent many numerical methods require it in order to numerically integrate the weak form equations governing the physic phenomenon. The integration mesh can have the size of the problem domain or even a larger one, without affecting too much the final results [8].

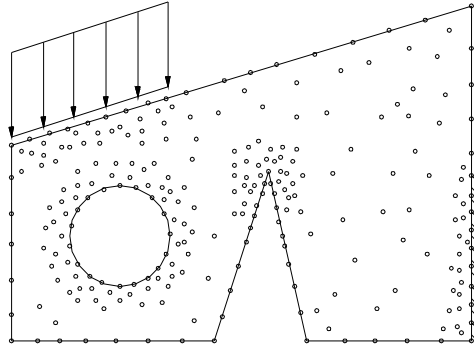


figure 15 – Nodal discretization example.

It is common to use Gaussian integration meshes, as in the FEM, fitted to the problem domain, figure 16(a), however other approaches in meshless methods are also valid, figure 16(b). Another way to integrate the weak form equations is using the nodal integration, which resorts to the Voronoi diagrams in order to obtain the integration weight on each node, figure 16(c). As it is perceptible in this procedure the nodal mesh will serve also as the integration mesh. Nonetheless the nodal integration leads generally to less accurate results, being necessary to implement a stabilization method which increases the computational cost.

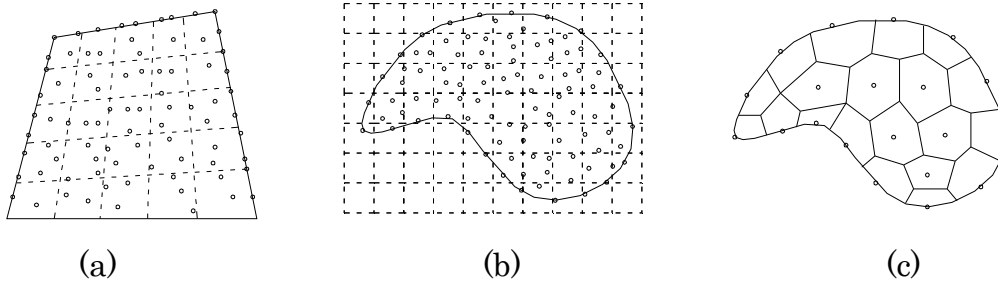


figure 16 – (a) Fitted Gaussian integration mesh. (b) General Gaussian integration mesh. (c) Voronoi diagram for nodal integration.

With the nodal mesh defined and the integration mesh constructed the nodal connectivity can be imposed. In the FEM the nodal connectivity is predefined with the ‘element’ but in the meshless methods there are no elements. Thus, for each interest point x_i of the problem domain concentric areas or volumes are defined, and the nodes inside these areas or volumes belong to the ‘influence-domain’ of node x_i . In the majority of the meshless methods the interest points are the integration points from the background integration mesh, however there are meshless methods, which use the collocation point methodology or the nodal integration scheme, where the interest points are the nodes of the nodal mesh. The shape and size of the

influence domain, which depends on the relative position of the interest point, affects the quality of the results, figure 17(a). It is recommend that all ‘influence-domains’ possess the same number of nodes inside, as so the size of the influence domain should be dependent on the nodal density around the interest point. In figure 17(b) it is shown a bad choice in the ‘influence-domain’ strategy.

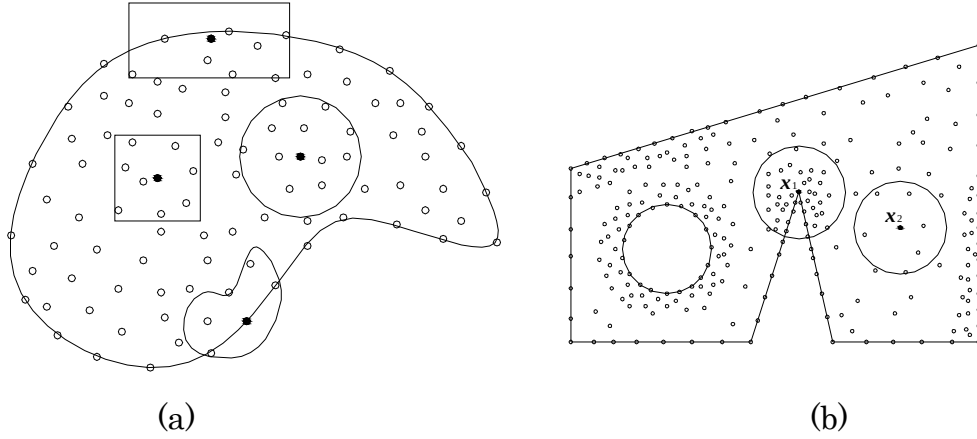


figure 17 – (a) Influence domains with different sizes and shapes. (b) Example of a bad choice in the size of the influence domain, interest point x_1 has much more nodes inside than interest point x_2 .

Afterwards the field variables can be obtained with the approximation or interpolation function. Take for example the displacement field $\mathbf{u}$ as the field variable under consideration. In meshless methods the displacement components $\mathbf{u}_i = (u_x \ u_y \ u_z)$ at any interest point $\mathbf{x}_i$ within the problem domain are approximated or interpolated using the nodal displacement of the nodes inside the ‘influence-domain’ of such interest point $\mathbf{x}_i$,

$$\mathbf{u}_i(\mathbf{x}) = \sum_{j=1}^n \phi_j(\mathbf{x}) \mathbf{u}_j(\mathbf{x}) \quad (183)$$

being n the number of nodes inside the ‘influence-domain’ of the interest point $\mathbf{x}_i$, $\mathbf{u}_j(\mathbf{x})$ is the displacement components of each node within the ‘influence-domain’ and $\phi_j(\mathbf{x})$ is the approximation or interpolation function of the j th node obtained using only the n nodes inside the ‘influence-domain’.

The equation system is the next step. In the meshless method discrete equations can be formulated using the approximation or interpolation functions applied to the strong or weak form formulation. In the case of the meshless methods using the weak form of Galerkin it can be obtained

applying, to the differential equation governing the physic phenomenon, the weighted residual method of Galerkin. The produced equations are then arranged in a local nodal matrix form and assembled into a global equation system matrix. In the case of a static problem the global equation system matrix is a set of algebraic equations, for the case of the free vibration analysis or buckling analysis is a set of eigenvalue equations and in the case of a dynamic (time dependent) analysis is a set of differential equations.

In order to obtain the distinct solutions for the distinct types of analysis it must be choose the appropriate solver. For static problems with the global equation system matrix the displacement field is obtained. To obtain it a linear algebraic equation solver is required. In this work for small systems it is used a Gauss elimination method and for larger equation systems a LU decomposition method. For the free-vibration and buckling analysis it is required the use of an eigenvalue equation solver to obtain the eigenvalue and the eigenvector solution. In this work it was used the Jacobi's method. In the dynamic analysis the variation through time of the displacement, velocity and acceleration are the results to be obtained. In this work the solution of the dynamic equations is obtained using the modal superposition method.

3.2 Natural Neighbours

In the NNRPIM the Natural Neighbour concept is used to enforce the nodal connectivity and to aid the construction of the integration mesh. The concept of natural neighbours was firstly introduced by Sibson [61] for data fitting and smoothing. The Voronoï diagrams and the Delaunay triangulation are useful mathematical tools in the determination of the natural neighbours for each node belonging to the global nodal set. This theory is applicable to a D -dimensional space, however for simplicity in this work an example of a two-dimensional Euclidian space $\square^2$ is presented. Consider a set N of N distinct nodes,

$$N = \{n_1, n_2, \dots, n_N\} \in \square^2 \quad (184)$$

The Voronoï diagram of N is the partition of the domain defined by N in sub-regions V_I , closed and convex. Each sub-region V_I is associated to the node I , n_I , in a way that any point in the interior of the V_I is closer to n_I than any other node n_J , where $n_J \in N \wedge J \neq I$. In other words, V_I is the geometric place where all points are closer to n_I than to any other node. The

sub-regions V_k are the “Voronoi cells” which form the Voronoi diagram, $k = 1, 2, \dots, N$. In mathematical terms the Voronoi cell is defined by,

$$V_I = \{ \mathbf{x} \in \square^2 : E_n(\mathbf{x}, \mathbf{x}_I) < E_n(\mathbf{x}, \mathbf{x}_J) \quad \forall J \neq I \} \quad (185)$$

being $E_n(\mathbf{x}, \mathbf{x}_I)$, the Euclidian metric norm, i.e., the distance between a interest point and a node I , with coordinates defined by $\mathbf{x}$ and $\mathbf{x}_I$. In figure 18 the construction of the sub-region V_I is represented, which starts with a nodal set of potential neighbours represented in figure 18(a).

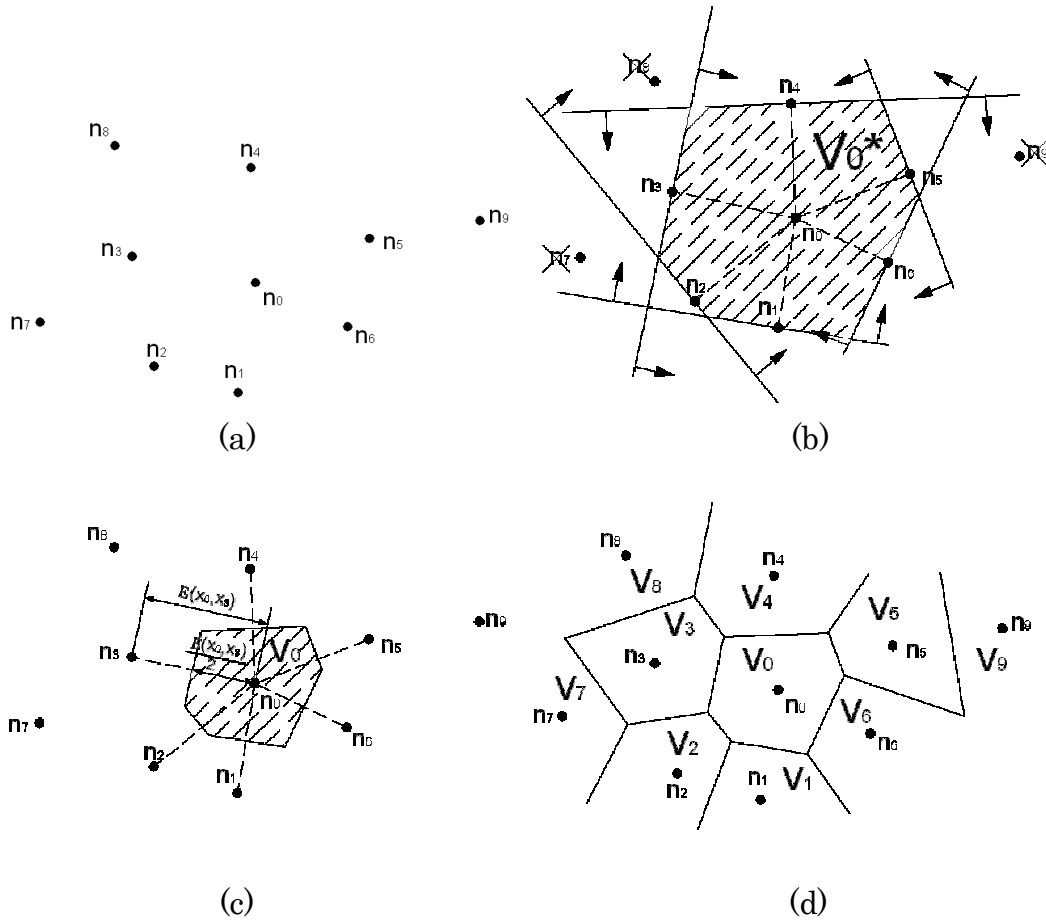


figure 18 – (a) Initial nodal set of potential neighbour nodes. (b) Final cell containing only neighbour nodes. (c) Voronoi cell. (d) Voronoi diagram.

As figure 18(b) indicates the neighbour nodes are obtained by the intersection of domains whose limits are defined by the line that intersects node J (potential neighbour node) and is normal to the dashed line OJ , where O is the interest or central node. By definition, only the nodes in the perimeter of the obtained final domain, V_0^* , are considered as neighbour nodes. The Voronoi cell, V_0 , obtained is determined as figure 18(c) indicates.

The cell V_o is the homothetic form of V_o^* . A similar procedure is applied in order to obtain the remaining Voronoï cells, figure 18(d). This procedure can easily be extrapolated to the three-dimensional Euclidian space $\mathbb{R}^3$. In the $\mathbb{R}^3$ space the solid lines in figure 18(b) become planes, normal to the vectors defined by the dash lines, which connects the potential neighbours J with the central node O . Therefore, in the end, the obtained Voronoï cell, V_o , is similar with the one presented in figure 19.

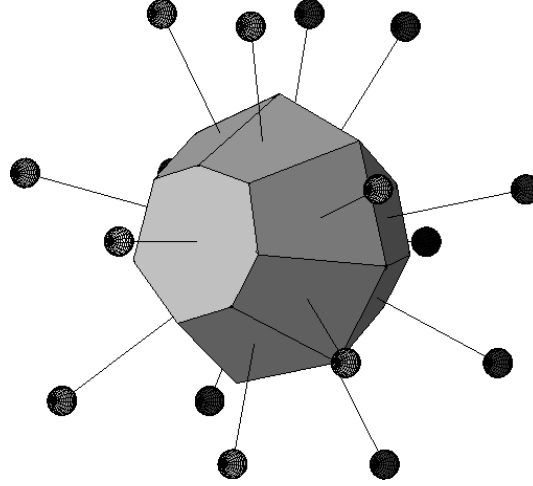


figure 19 – 3D Voronoï cell.

The Delaunay triangulation is the geometrical dual of the Voronoï diagram and it is constructed by connecting the nodes whose Voronoï cells have common boundaries. The duality between the Voronoï diagram and the Delaunay triangulation implies that a Delaunay edge exists between two nodes in the plane if and only if their Voronoï cells share a common edge. An important property of the Delaunay triangles is the “empty circumcircle criterion” [62]. If a set of nodes $\mathbf{n}_t = \{n_j, n_k, n_l\} \in N$ forms a Delaunay triangle then the circumcircle formed by the triangle N_t contains no other nodes of N . In the context of the natural neighbour interpolation these circles are known as “natural neighbour circumcircles” [63]. The centre of the natural neighbour circumcircle is the vertex of the respective Voronoï cell. These features are presented in figure 20.

In the NNRPIM the Voronoï diagram is used to create the “influence-cells”, which enforces the connectivity between the nodes in N . On the other hand the duality between the Voronoï cells and the Delaunay triangles is used in order to construct a nodal dependent background integration mesh.

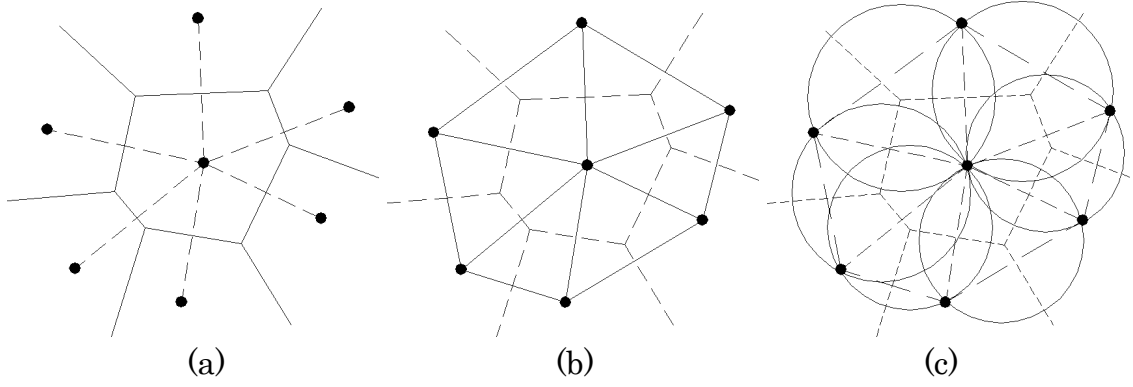


figure 20 – (a) Initial Voronoi diagram. (b) Delaunay triangulation. (c) Natural neighbour circumcircles.

3.3 Influence-Cells and Nodal Connectivity

In early works related with the RPI [19, 20, 22, 64] the nodal connectivity is obtained by the overlap of the influence domain of each node. These influence domains are found by searching enough nodes inside a fixed area or a fixed volume, respectively for the 2D problem and for the 3D problem. Because of its simplicity many other meshless methods use this concept [8, 10, 11]. However the size or shape variation of these influence domains along with the problem domain affects the performance and the final solution of the meshless method. It is important that all the influence domains in the problem contain approximately the same number of nodes. Irregular domain boundaries or node clusters in the nodal mesh can lead to unbalanced influence domains. Fixed and regular shaped influence domains are not suitable to the meshless procedure. Therefore in the NNRPI the nodal connectivity is imposed by the overlapping of the ‘influence-cells’ [29, 32], which can be considered as a new concept, similar to the influence domain, obtained from the Voronoï cells. The cell formed by n nodes that contributes to the interpolation of the interest point x_i is called ‘influence-cell’. Since it is simpler to represent, only the determination of the 2D influence-cell is presented, however this concept is applicable to a D -dimensional space. Two distinct types of influence-cells are presented:

First degree influence-cell: A point of interest, x_i , search for its neighbour nodes following the Natural Neighbour Voronoi construction presented early. Thus the *first degree influence-cell* is composed by these first natural neighbours.

Second degree influence-cell: A point of interest, x_i , searches for its neighbour nodes, in the same manner as in the *first degree influence-cell*.

Then, based on a previous construction of the Voronoi diagram for the node mesh, the natural neighbours of the first natural neighbours of $\mathbf{x}_I$ are added to the influence-cell.

The schematic representation of these two types of influence-cell is presented in figure 21. As it is possible to observe the *first degree influence-cell* is naturally smaller than the *second degree influence-cell*.

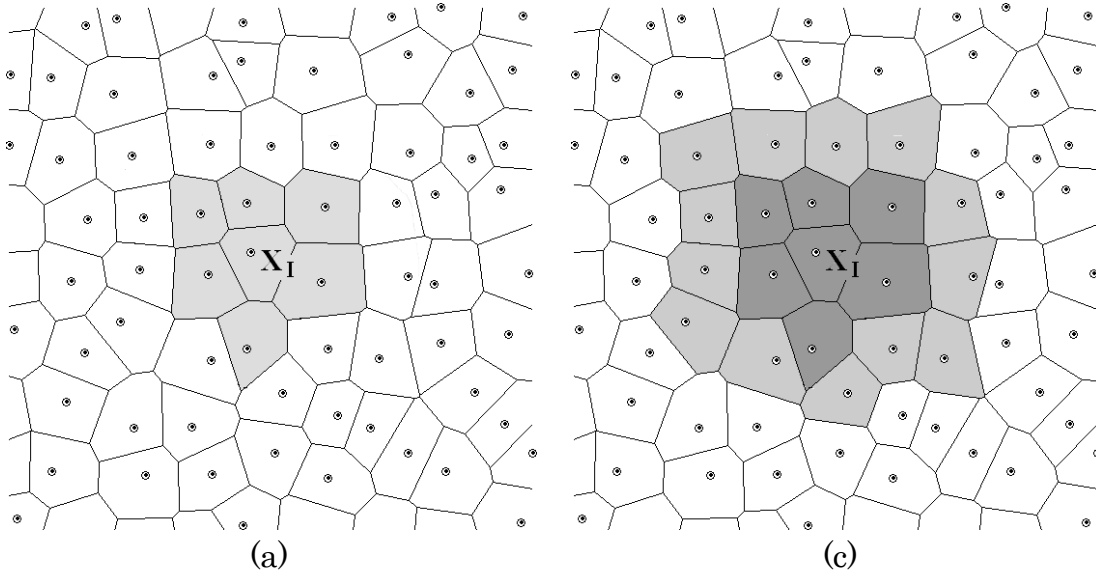


figure 21 – (a) First degree influence-cell. (b) Second degree influence-cell.

In an initial phase, after the domain discretization in a regular or an irregular nodal mesh, the Voronoï cells of each node are constructed. These cells can be considered as a background mesh for integration purpose, being determined the influence-cell for each one of these integration points.

3.4 Numerical Integration in the NRPIM

Recently a meshless method, based on the RPI, using a stabilized nodal integration [65] was successfully implemented and the obtained results proved to be better than the other meshless RPI approaches based on Gauss integration schemes [21, 22]. However the extra time spent in stabilizing the nodal integration does not pay the increased accuracy of the final solution. Thus, in this work an innovate integration scheme based on the Voronoï tessellation and the Delaunay triangulation is proposed. Once again, since it is easier to present, first it is demonstrated the 2D integration scheme.

3.4.1 2D integration scheme

Using the referred geometric constructions small areas are established, which can be quadrilaterals or triangles respectively consistent with an irregular or a regular mesh. Then, with the construction of the Voronoï cells, V_i , the intersection points, P_{ij} , of the neighbour edges of V_i can be settled, figure 22. Afterwards the middle points, M_{ij} , between node n_i and its neighbour nodes are obtained. Thus the Voronoï cells are divided in n quadrilateral sub-cell, S_{ij} , as figure 22 indicates.

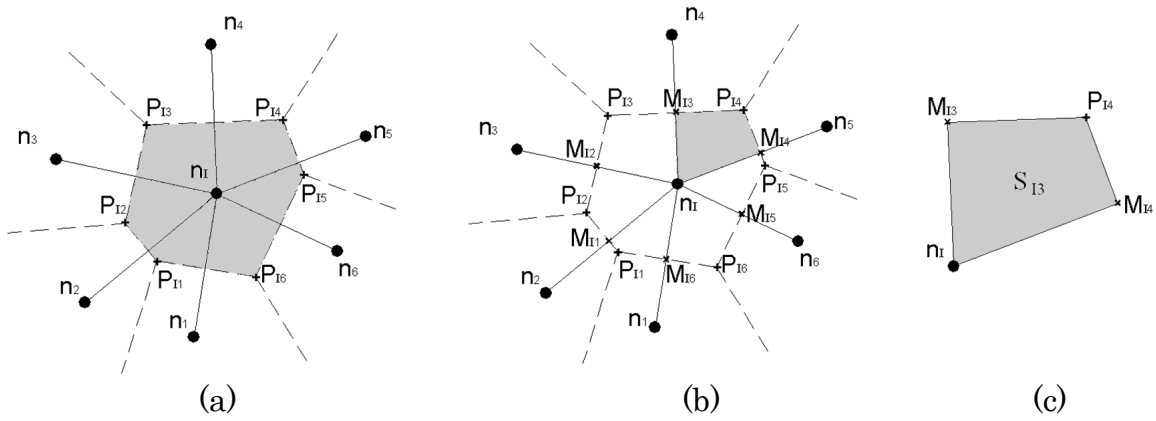


figure 22 – (a) Voronoï cell and the respective P_{ij} intersection points. (b) Middle points M_{ij} and the respective generated quadrilaterals. (c) Quadrilateral $\overline{M_{i3} P_{i4} M_{i4} n_i}$.

For the regular mesh, figure 23, the middle points M_{ij} are coincident with the edge intersection points P_{ij} . It is visible that this geometric coincidence leads to the formation of triangles instead of the quadrilateral sub-cells, as in the irregular mesh case.

Any n_i node with J neighbour nodes, forming a V_i Voronoï cell, possesses J sub-cells, S_{ij} , where,

$$A_{V_i} = \sum_{i=1}^n A_{S_{ij}}, \quad \forall A_{S_{ij}} \geq 0 \quad (186)$$

being A_{V_i} the Voronoï cell area and $A_{S_{ij}}$ the sub-cell area. If the set of Voronoï cells are a partition, without gaps, of the global domain then, the set of sub-cells are also a partition, without gaps, of the global domain.

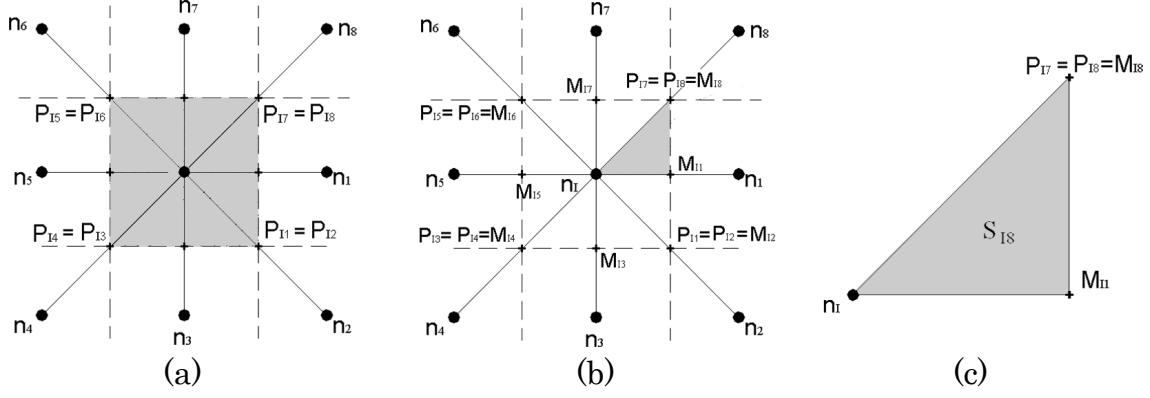


figure 23 – (a) Voronoï cell and the respective P_I intersection points. (b) Middle points M_I and the respective generated triangles. (c) Triangle $\overline{M_{11}M_{18}n_{11}}$.

It is clear now how the construction of the sub-cells generates two types of basic shapes - triangles or quadrilaterals. Starting with these two shapes, numerous integrations schemes can be constructed. In this work it is proposed an ordered scheme, based on the Gauss-Legendre numerical integration.

3.4.1.1 Integration Scheme – Order 0

The coordinates of each integration point are calculated on each sub-cell, as indicated in figure 24, being the weight of each integration point the area of the respective sub-cell.

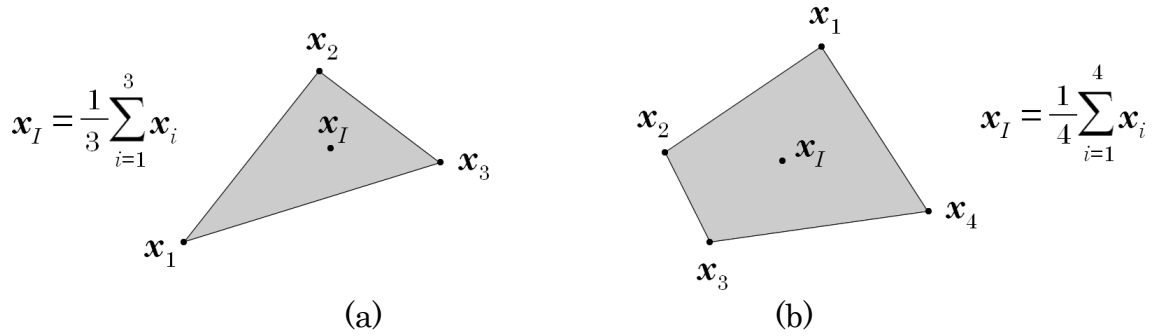


figure 24 – (a) Triangular shape and (b) Quadrilateral shape and the respective integration points x_I .

Therefore, considering figure 24, the area of the triangle shape sub-cell is defined by,

$$A_I = \frac{1}{2} \left| \det \begin{bmatrix} x_2 - x_1 & y_2 - y_1 \\ x_3 - x_1 & y_3 - y_1 \end{bmatrix} \right| \quad (187)$$

and for the quadrilateral shape the area is,

$$A_I^\square = \frac{1}{2} \left| \det \begin{bmatrix} x_2 - x_1 & y_2 - y_1 \\ x_3 - x_1 & y_3 - y_1 \end{bmatrix} + \det \begin{bmatrix} x_4 - x_1 & y_4 - y_1 \\ x_3 - x_1 & y_3 - y_1 \end{bmatrix} \right| \quad (188)$$

The process is equivalent with the 1x1 integration point Gauss-Legendre scheme for triangle and quadrilateral shapes, respectively.

3.4.1.2 Integration Scheme – Order 1

In this integration scheme the basic geometric forms are sub-divided again. However, in this case only in quadrilateral shapes, as figure 25 indicates. Firstly, the centre of the geometric shape is determined, afterwards middle points on the edges are determined and then new quadrilaterals are constructed.

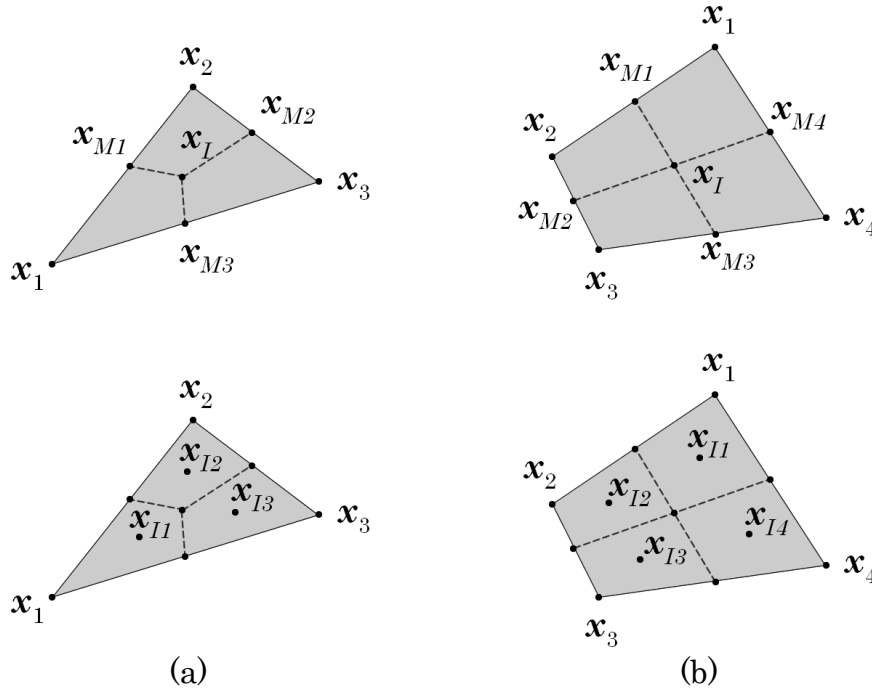


figure 25 – (a) Partition of the initial triangular shape into quadrilaterals and (b) partition of the initial quadrilateral shape into small quadrilaterals with the respective integration points x_I .

For each new quadrilateral shape obtained the ‘*Integration Scheme of Order 0*’ is applied, as shown in figure 24(b).

3.4.1.3 Integration Scheme – Order k

Considering the quadrilateral shapes constructed for the “Integration Scheme of Order 1” a Gauss-Legendre quadrature scheme of $k \times k$ integration points is applied, figure 26.

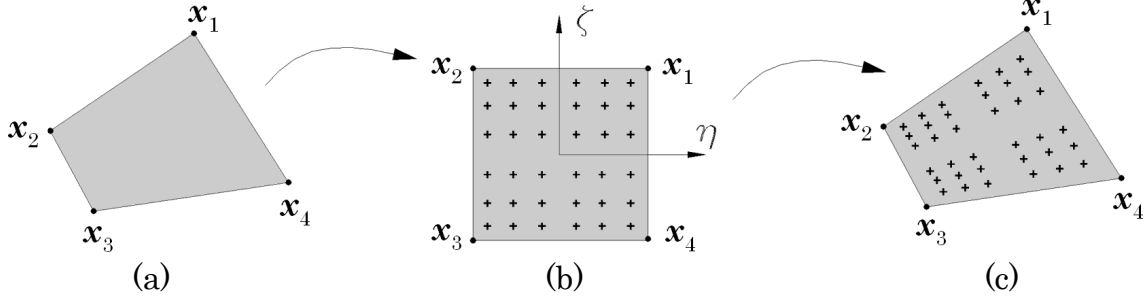


figure 26 – (a) Initial quadrilateral. (b) Transformation of the initial quadrilateral into an isoparametric square shape and application of the 6x6 quadrature point rule. (c) Return to the initial quadrilateral shape.

The integration weight is defined by,

$$A_I = w_\eta w_\zeta \left(\frac{A^\square}{4} \right) \quad (189)$$

Where $A^\square$ is defined in equation (188) and w_η and w_ζ are the Gauss quadrature weights for an isoparametric quadrilateral element of FEM.

3.4.1.4 Nodal Integration Scheme

The nodal integration scheme is as simple as considering the integration point coincident with each node. Thus, the weight of each integration point is the sum of the each of the areas of the respective sub-cells, i.e., it is the area of the respective Voronoï cell, obtained with equation (186). This type of integration is not sufficient to integrate the NNRPIM interpolation functions, it is necessary to apply a stabilization procedure [65], which increases the method computational cost, reducing consequently the NNRPIM efficiency.

3.4.2 3D integration scheme

The extension of the presented 2D integration scheme is strait-forward for the 3D space. In the irregular mesh, the Voronoï cell is subdivided in hexahedrons and in the regular mesh in tetrahedrons. Then the distinct integration scheme orders are obtained following the 2D procedure. In figure 27 it is described the 3D procedure. First small volumes are established, figure 27(b). These volumes can be isoparameterized, figure 27(c), and the Gauss-Legendre quadrature scheme applied, figure 27(d).

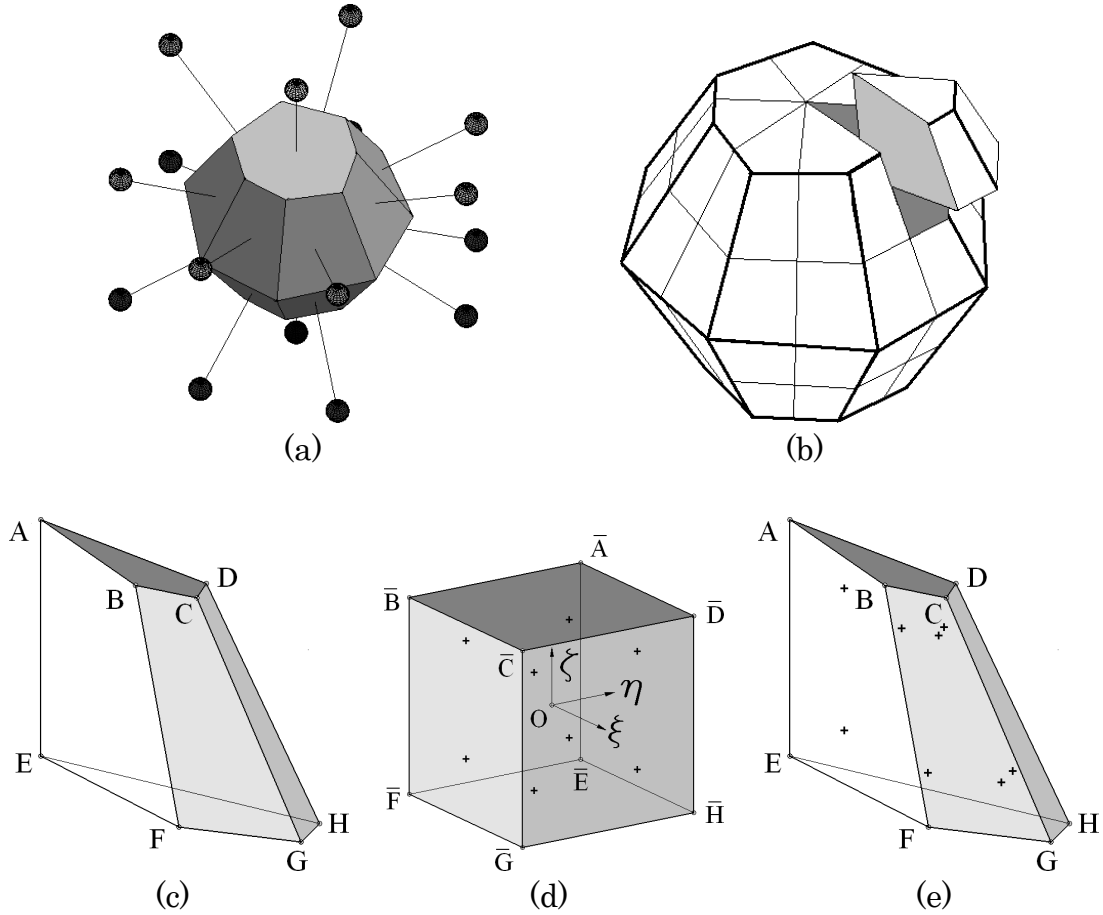


figure 27 – (a) Voronoï cell and (b) respective tetrahedrons. (c) Initial tetrahedron. (d) Initial tetrahedron isoparameterization and determination of the quadrature integration points. (e) Quadrature integration points in Cartesian coordinates.

The 3D analysis is computational very demanding. Although it is possible to apply all the integration schemes referred in section 3.4.1, there is no need to use a Gauss-Legendre quadrature scheme higher than 1x1x1 per tetrahedron. This integration scheme is sufficient for the 3D NNRPIM formulation, since higher integration schemes raise enormously the computational cost and the 3D nodal integration without stabilization is not sufficient to integrate the NNRPIM interpolation functions.

3.4.3 3D Shell-Like integration scheme

In this work a 3D approach to solve the particular case of thin structures, such as plates and shells, was developed. Due to its geometry this kind of structures can be discretized only with two layers of nodes: a top nodal layer and a bottom nodal layer. No nodes in the middle (along the plate or shell thickness) are necessary to simulate the deformation field. Thus the problem is reduced in a first stage to a two-dimensional problem, considering only the two larger dimensions and neglecting the plate or shell thickness, figure 28(a). Therefore a two-dimensional nodal mesh, coincident

with the middle surface of the plate or of the shell, is firstly created, figure 28(b). Based in this two-dimensional mesh the background integration mesh is constructed, considering the integration schemes proposed in section 3.4.1, figure 28(c). Afterwards the middle surface nodal mesh constructed in the first step, figure 28(b), is projected to the top surface and bottom surface of the plate or shell, figure 28(d). The two-dimensional integration mesh is distributed along the plate or shell thickness respecting a Gauss quadrature scheme.

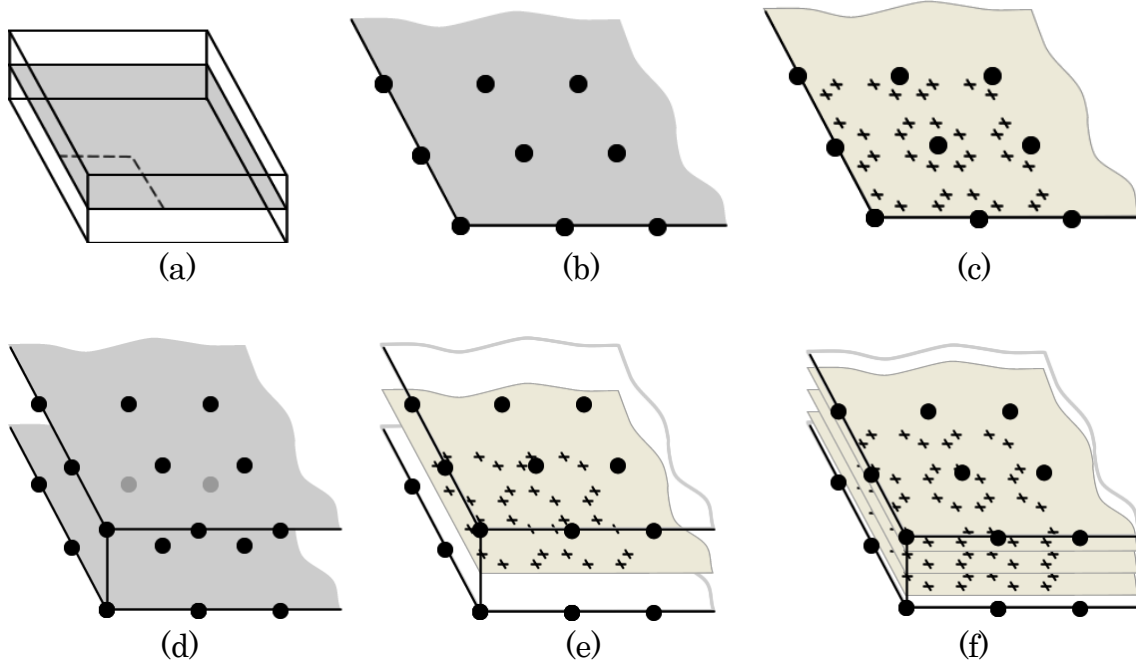


figure 28 – Procedure of the shell-like nodal and integration mesh construction and distribution.

In figure 28(e) is shown the result of only considering one repetition of the two-dimensional integration mesh, obtained in figure 28(c), along the plate thickness. In figure 28(f) is shown the result of considering three repetitions of the two-dimensional integration mesh along the thickness. The weight of each integration point has to consider the repetition along the thickness (always respecting a Gauss quadrature scheme) and the thickness of the plate,

$$W_k = t w_i A_j \quad (190)$$

being t the thickness of the plate or shell, w_i is the i th weight of the Gauss quadrature (there will be as many w_i as repetitions of the two-dimensional integration mesh along the thickness), A_j is the weight area of interest point j obtained with the integration scheme adopted from section 3.4.1

and W_k is the integration weight of interest point k . Note that only for the case of figure 28(e) $j = k$. In the end $m = \kappa \times n$, being m the total number of integration points in the problem domain, n the number of integration points of the two-dimensional integration mesh and κ the number of repetitions.

3.5 Radial Point Interpolators

Consider a function $\mathbf{u}(\mathbf{x})$ defined in the domain Ω , which is discretized by a set of N nodes. In the NNRPIM the function $\mathbf{u}(\mathbf{x})$ passes through all nodes using a radial basis function (RBF). It is assumed that only the nodes within the influence-cell of the point of interest $\mathbf{x}_I$ have effect on $\mathbf{u}(\mathbf{x}_I)$. The value of function $\mathbf{u}(\mathbf{x}_I)$ at the point of interest $\mathbf{x}_I$ is obtained by,

$$\mathbf{u}(\mathbf{x}_I) = \sum_{i=1}^n R_i(\mathbf{x}_I) a_i(\mathbf{x}_I) + \sum_{j=1}^m p_j(\mathbf{x}_I) b_j(\mathbf{x}_I) = \mathbf{R}^T(\mathbf{x}_I) \mathbf{a}(\mathbf{x}_I) + \mathbf{p}^T(\mathbf{x}_I) \mathbf{b}(\mathbf{x}_I) \quad (191)$$

where $R_i(\mathbf{x}_I)$ is the RBF and n is the number of nodes inside the influence-cell of $\mathbf{x}_I$. The coefficients $a_i(\mathbf{x}_I)$ and $b_j(\mathbf{x}_I)$ are non constant coefficients of $R_i(\mathbf{x}_I)$ and $p_j(\mathbf{x}_I)$, respectively. The monomials of the polynomial basis are defined by $p_j(\mathbf{x}_I)$ and m is the basis monomial number. The vectors in equation (191) are defined as,

$$\mathbf{R}(\mathbf{x}_I) = \{R_1(\mathbf{x}_I) \quad R_2(\mathbf{x}_I) \quad \cdots \quad R_n(\mathbf{x}_I)\}^T \quad (192)$$

$$\mathbf{p}(\mathbf{x}_I) = \{p_1(\mathbf{x}_I) \quad p_2(\mathbf{x}_I) \quad \cdots \quad p_m(\mathbf{x}_I)\}^T \quad (193)$$

$$\mathbf{a}(\mathbf{x}_I) = \{a_1(\mathbf{x}_I) \quad a_2(\mathbf{x}_I) \quad \cdots \quad a_n(\mathbf{x}_I)\}^T \quad (194)$$

$$\mathbf{b}(\mathbf{x}_I) = \{b_1(\mathbf{x}_I) \quad b_2(\mathbf{x}_I) \quad \cdots \quad b_m(\mathbf{x}_I)\}^T \quad (195)$$

The number of monomials m of the polynomial basis should be $m < n$ to obtain a more stable function. Early works on the RPI [21, 60] sustain that the RPI with pure radial functions are not consistent and have a problem passing the standard patch test, i.e. it fails to reconstruct the linear field exactly. The initial purpose of adding the polynomials into the basis function

was to insure the consistency of the RPI functions. Thus, adding to the RBF a linear polynomial ensures the reproducing of the linear field (C^1 consistency) and consequently help the RPI passing the standard patch test. However in section 3.7 it is proved that the RPI do not require a polynomial basis function [29, 32] to pass the standard patch test, reducing in this manner the computational cost of the RPI methods. In the RBF the variable is the distance r_{li} between the relevant node $\mathbf{x}_l$ and the neighbour node $\mathbf{x}_i$. For the three-dimensional space is defines as,

$$r_{li} = \sqrt{(x_l - x_i)^2 + (y_l - y_i)^2 + (z_l - z_i)^2} \quad (196)$$

Several known RBFs are well studied and developed in [21, 22]. This work [29] uses the Multiquadric (MQ) function proposed initially by Hardy [16]. Other functions were also used in an initial phase, however the best results were obtained with the MQ. The form of the MQ-RBF is,

$$R(r_{li}) = \left(r_{li}^2 + c^2 \right)^p \quad (197)$$

where c and p are two shape parameters that need to be optimized [22, 29]. In early works on the RPI [21, 22] the optimal value for the shape parameters is achieved $c = 1.42$ and $p = 1.03$. However the achieved values are inaccurate and lack in precision and stability [29, 32]. This is the major disadvantage of meshless methods using RBFs. The variation of the shape parameters affects the performance of the RBFs and there are no efficient rigorous theoretical methods for obtaining the optimal values of c and p . In section 3.7 an optimization study is presented and more suitable shape parameters are obtained. In this work some distinct polynomial basis are experimented. In the 2D analysis,

$$\begin{aligned} \text{Constant basis } (m=1): \quad & \mathbf{p}^T(\mathbf{x}) = \{1\} \\ \text{Linear basis } (m=3): \quad & \mathbf{p}^T(\mathbf{x}) = \{1 \quad x \quad y\} \\ \text{Quadratic basis } (m=6): \quad & \mathbf{p}^T(\mathbf{x}) = \{1 \quad x \quad y \quad x^2 \quad xy \quad y^2\} \end{aligned} \quad (198)$$

In the 3D analysis,

$$\begin{aligned} \text{Constant basis } (m=1): \quad & \mathbf{p}^T(\mathbf{x}) = \{1\} \\ \text{Linear basis } (m=4): \quad & \mathbf{p}^T(\mathbf{x}) = \{1 \quad x \quad y \quad z\} \\ \text{Quadratic basis } (m=10): \quad & \mathbf{p}^T(\mathbf{x}) = \{1 \quad x \quad y \quad z \quad x^2 \quad xy \quad y^2 \quad yz \quad z^2 \quad zx\} \end{aligned} \quad (199)$$

Another consideration regarding the polynomial basis is used in this work, the *null base*, which is the same as the absence of polynomial basis. In the case of the *null basis* equation (191) becomes simpler, once the polynomial terms disappear, i.e., $p_j(\mathbf{x}_i)$ and $b_j(\mathbf{x}_i)$ vanish from the equation. In the existing case of a polynomial basis, the polynomial basis has to satisfy an extra requirement in order to obtain a unique solution:

$$\sum_{i=1}^n p_j(\mathbf{x}_i) a_i(\mathbf{x}_i) = 0 \Leftrightarrow \mathbf{p}^T(\mathbf{x}_i) \mathbf{a}(\mathbf{x}_i) = 0 \quad (200)$$

being $j = \{1, 2, \dots, m\}$. Therefore, combining equation (191) with equation (200) a new equation matrix can be written,

$$\begin{Bmatrix} \mathbf{u}_s \\ \mathbf{0} \end{Bmatrix} = \begin{bmatrix} \mathbf{R}_Q & \mathbf{P}_m \\ \mathbf{P}_m^T & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} = \mathbf{G} \begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} \quad (201)$$

where $\mathbf{u}_s$ is given by,

$$\mathbf{u}_s = \{u_1, u_2, \dots, u_n\}^T \quad (202)$$

the moment matrix $\mathbf{R}_Q$ of the RBF is obtained with,

$$\mathbf{R}_Q = \begin{bmatrix} R(r_{11}) & R(r_{12}) & \dots & R(r_{1n}) \\ R(r_{21}) & R(r_{22}) & \dots & R(r_{2n}) \\ \vdots & \vdots & \ddots & \vdots \\ R(r_{n1}) & R(r_{n2}) & \dots & R(r_{nn}) \end{bmatrix} \quad (203)$$

And the moment matrix $\mathbf{P}_m$ of the polynomial basis, a $n \times m$, is given by,

$$\mathbf{P}_m = \begin{bmatrix} P_1(\mathbf{x}_1) & P_2(\mathbf{x}_1) & \dots & P_m(\mathbf{x}_1) \\ P_1(\mathbf{x}_2) & P_2(\mathbf{x}_2) & \dots & P_m(\mathbf{x}_2) \\ \vdots & \vdots & \ddots & \vdots \\ P_1(\mathbf{x}_n) & P_2(\mathbf{x}_n) & \dots & P_m(\mathbf{x}_n) \end{bmatrix} \quad (204)$$

As example the linear polynomial basis for the 3D problem is presented,

$$\mathbf{P}_m = \begin{bmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ \vdots & \vdots & \vdots & \vdots \\ 1 & x_n & y_n & z_n \end{bmatrix} \quad (205)$$

The moment matrix $\mathbf{P}_m$ of the polynomial constant basis is a column vector $n \times 1$ with only unit cells. Notice that, in equation (201), if the polynomial basis does not exist $\mathbf{R}_Q = \mathbf{G}$, and the system becomes smaller. The geometric matrix $\mathbf{G}$ is a symmetric matrix because the distance is directional independent, i.e., $R(r_{ij}) = R(r_{ji})$. By solving equation (201),

$$\begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} = \mathbf{G}^{-1} \begin{Bmatrix} \mathbf{u}_s \\ \mathbf{0} \end{Bmatrix} \quad (206)$$

and substituting on equation (191),

$$\mathbf{u}(\mathbf{x}_I) = \left\{ \mathbf{R}^T(\mathbf{x}_I), \mathbf{p}^T(\mathbf{x}_I) \right\} \mathbf{G}^{-1} \begin{Bmatrix} \mathbf{u}_s \\ 0 \end{Bmatrix} = \varphi(\mathbf{x}_I) \mathbf{u}_s \quad (207)$$

the interpolation function, $\varphi(\mathbf{x}_I)$, on interest point $\mathbf{x}_I$ is obtained,

$$\varphi(\mathbf{x}_I) = \left\{ \mathbf{R}^T(\mathbf{x}_I), \mathbf{p}^T(\mathbf{x}_I) \right\} \mathbf{G}^{-1} = \{\varphi_1(\mathbf{x}_I), \varphi_2(\mathbf{x}_I), \dots, \varphi_n(\mathbf{x}_I)\} \quad (208)$$

The partial derivative of $\varphi(\mathbf{x}_I)$ in order to a variable ξ is defined as,

$$\varphi_{,\xi}(\mathbf{x}_I) = \left\{ \mathbf{R}_{,\xi}^T(\mathbf{x}_I), \mathbf{p}_{,\xi}^T(\mathbf{x}_I) \right\} \mathbf{G}^{-1} \quad (209)$$

The partial derivatives of the MQ-RBF in order to the same variable ξ are,

$$\mathbf{R}_{,\xi}(r_{ij}) = 2p \left(r_{ij}^2 + c^2 \right)^{p-1} (\xi_j - \xi_i) \quad (210)$$

3.5.1 Properties of the RPI function

The RPI possess many convenient properties [60], from which some are referred. As long as $\mathbf{G}$ is invertible the constructed RPI function, $\varphi(\mathbf{x}_I)$, depends uniquely on the distribution of the scatted nodes. The interpolation function has a compact support as long as it is constructed using only the

nodes within the influence-cell, which is important to create sparse and banded system matrices. The RPI functions possess the delta Kronecker property,

$$\varphi_i(\mathbf{x}_j) = \delta_{ij} = \begin{cases} 1 & (i = j) \\ 0 & (i \neq j) \end{cases} \quad i, j = 1, \dots, n \quad (211)$$

which is an important property in the imposition of the essential and natural boundary conditions. The partition of unity is satisfied,

$$\sum_{i=1}^n \varphi_i(\mathbf{x}_i) = 1 \quad (212)$$

if the constant is included in the polynomial basis and the interpolation function $\varphi(\mathbf{x}_i)$ possesses reproducing properties

$$\sum_{i=1}^n \varphi_i(\mathbf{x}_i) \mathbf{x}_i = \mathbf{x} \quad (213)$$

if the first order monomial is included in the polynomial basis. These two last properties permit to pass the standard patch test.

The interpolation function has C^∞ , once the MQ-RBF can be infinity derivable. An inconvenient property of the RPIMs interpolation functions is the lack of compatibility. This property is achieved in the RPIMs using the conforming RPIM [66]. However this same study concluded that the RPIM is much more simple and efficient than the conforming RPIM. The lack of consistency in the RPIMs interpolation functions, constructed without a polynomial basis, is the reason why this formulation cannot pass the standard path test, although it is proved that it can approach polynomials in the desired accuracy and the convergence is guaranteed when the nodes are refined [60].

3.5.2 RPI function shape

Examples of 2D NNRPIM interpolation functions were computed using the above described formulation and are presented in this section. The two distinct influence cells were used, a *first degree influence-cell* and a *second degree influence-cell*.

The first study is regarding the effects of the shape parameters (particular parameter c) in the shape of the interpolation function. Using a cell of 7

neighbours an interpolation function was constructed for the central node. It was used a linear polynomial basis and variable shape parameters $c = 0.1$, $c = 5$ and $c = 20$ and a constant shape parameter $p = 0.9$. In figure 29 it is shown the results for $c = 0.1$, and figure 30 and figure 31 are respectively concerning $c = 5$ and $c = 20$.

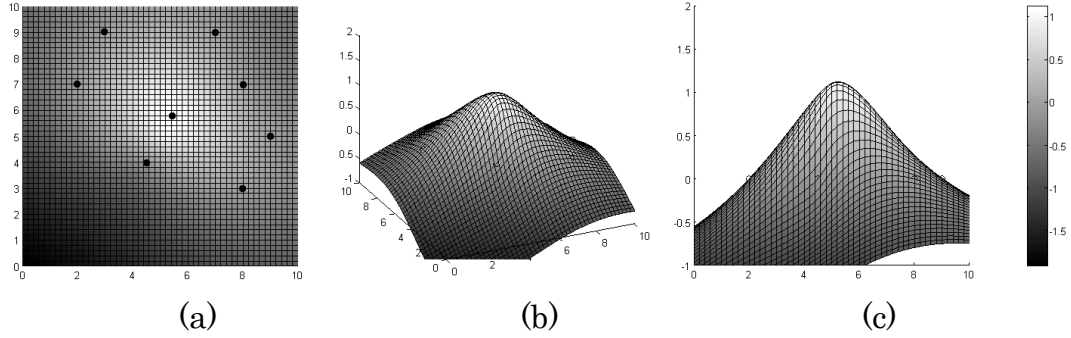


figure 29 – (a) Nodal mesh used. (b) RPI function isometric view. (c) RPI function O_{xx} view. ($c=0.1$)

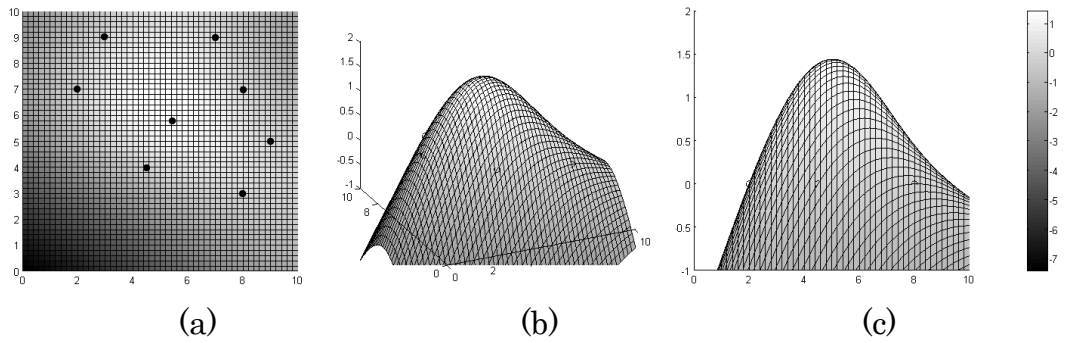


figure 30 – (a) Nodal mesh used. (b) RPI function isometric view. (c) RPI function O_{xx} view. ($c=5$)

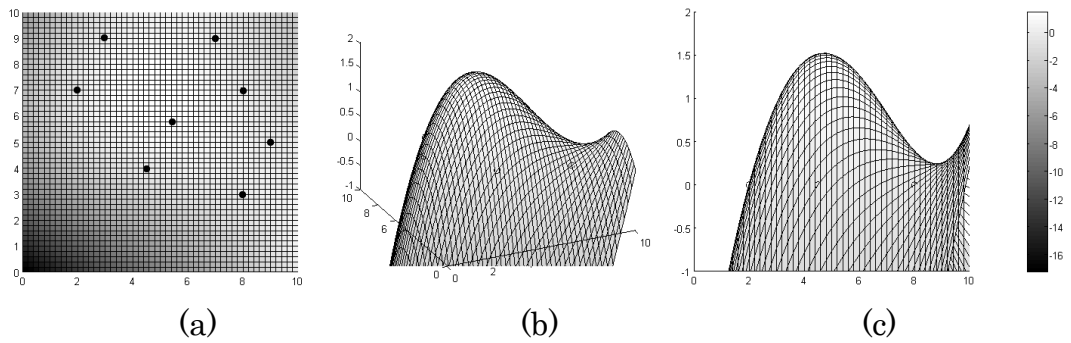


figure 31 – (a) Nodal mesh used. (b) RPI function isometric view. (c) RPI function O_{xx} view. ($c=20$)

As it is possible to observe the parameter c affects the interpolation function shape, with the increase of c the shape function turns more flat and more bell shaped. As it is shown in section 3.7 the increase of c removes from the RPI functions the delta Kronecker property.

The next figures show the smoothness of the RPI functions, and its respective partial derivatives (first order and second order). In this study 12 nodes were used and the interpolation functions were obtained for the middle node. It was used a linear polynomial basis and $c = 0.1$ and $p = 0.9$. In figure 32 it is possible to observe the interpolation function $\phi(\mathbf{x})$. The first partial derivatives $\phi_{,x}(\mathbf{x})$ and $\phi_{,y}(\mathbf{x}_I)$ are shown in figure 33 and figure 34, respectively. The second partial derivatives $\phi_{,xx}(\mathbf{x})$, $\phi_{,xy}(\mathbf{x}_I)$ and $\phi_{,yy}(\mathbf{x}_I)$ are shown from figure 35 to figure 37, respectively.

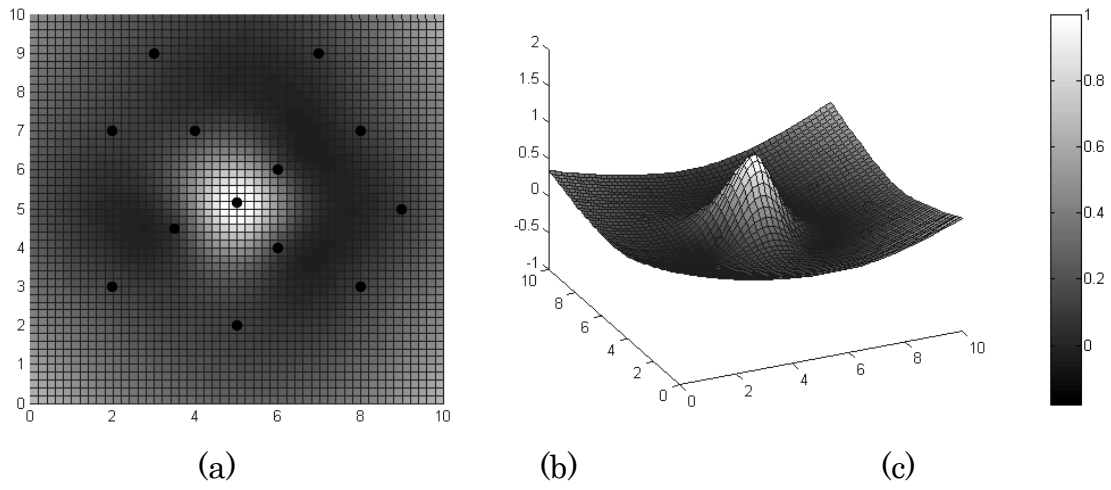


figure 32 – (a) Nodal mesh used. (d) $\phi(\mathbf{x})$ isometric view.

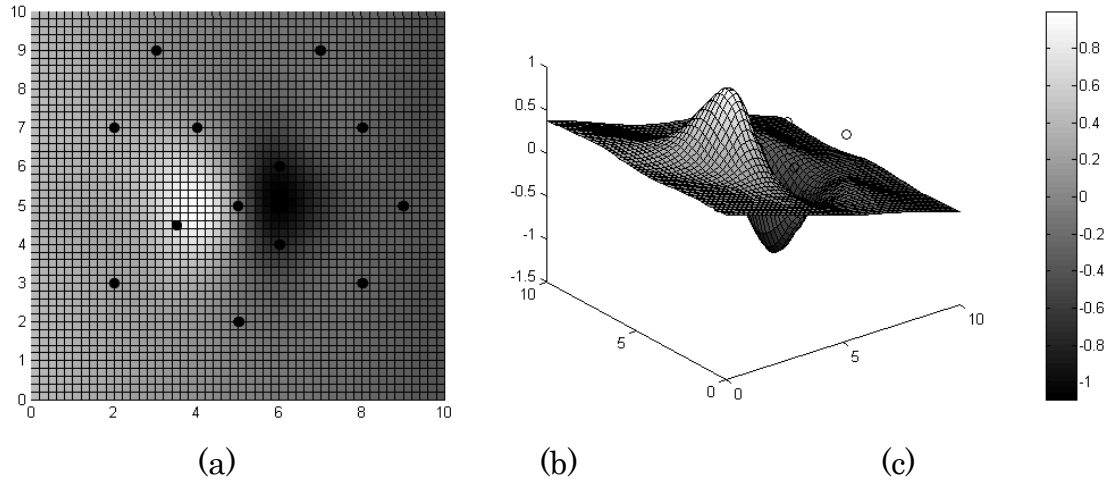


figure 33 – (a) Nodal mesh used. (d) $\phi_{,x}(\mathbf{x})$ isometric view.

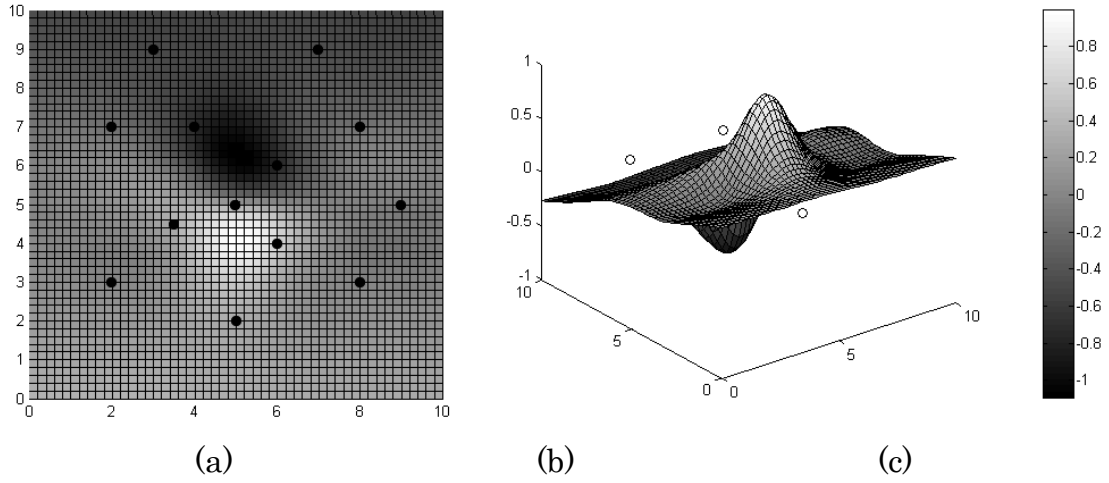


figure 34 – (a) Nodal mesh used. (d) $\phi_y(x)$ isometric view.

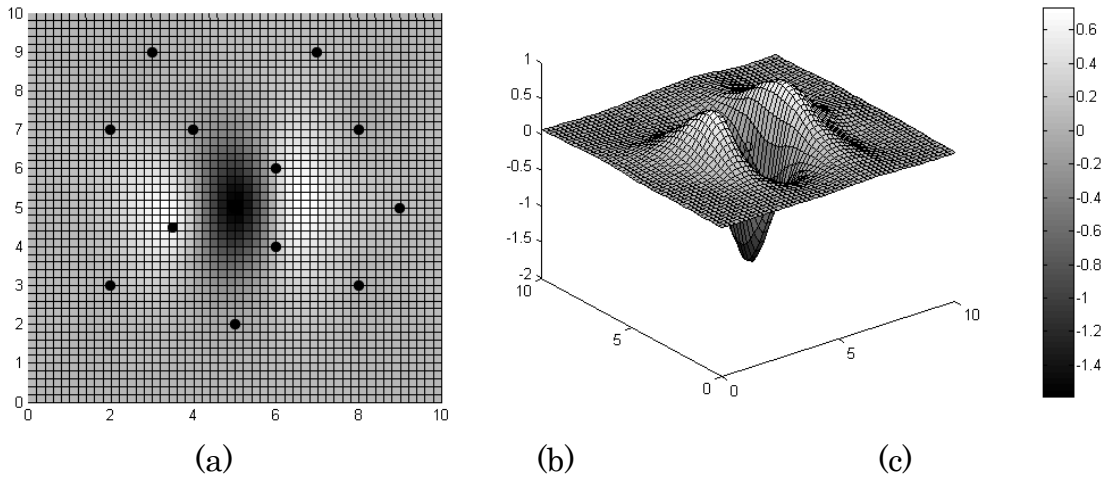


figure 35 – (a) Nodal mesh used. (d) $\phi_{xx}(x)$ isometric view.

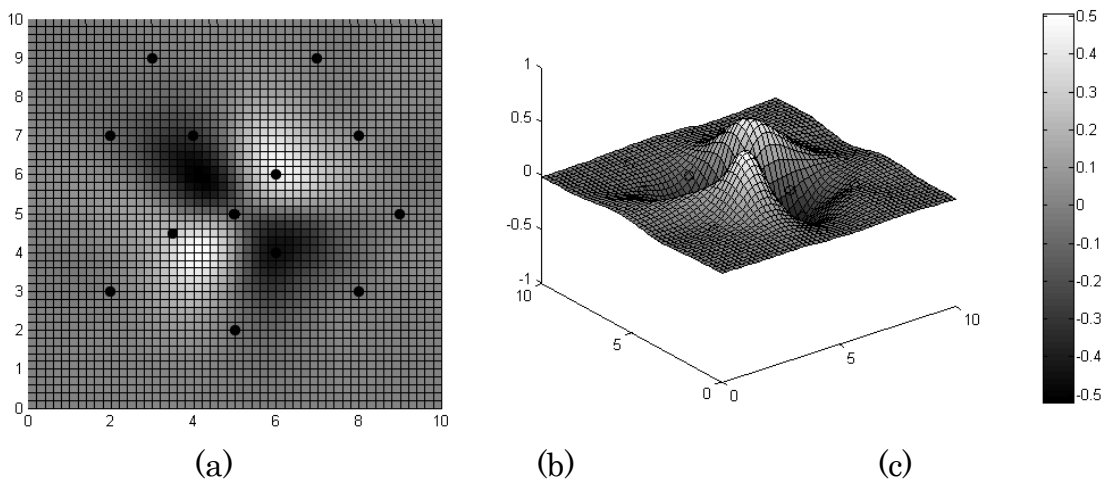


figure 36 – (a) Nodal mesh used. (d) $\phi_{xy}(x)$ isometric view.

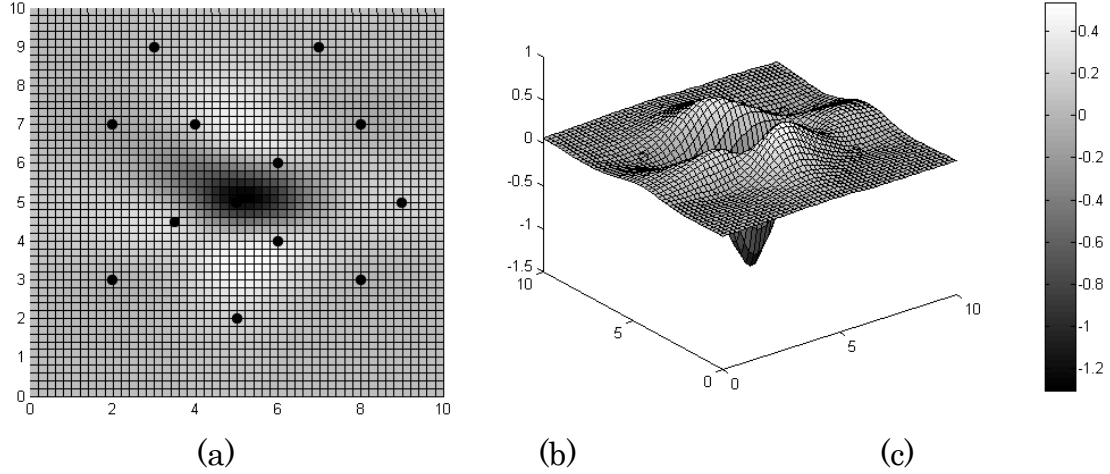


figure 37 – (a) Nodal mesh used. (b) $\phi_{yy}(\mathbf{x})$ isometric view.

With these results it is possible to observe that the constructed RPI functions are smooth and continuous interpolation functions.

3.6 NNRPIM Linear Flow-Chart

The NNRPIM numerical implementation can be summarized in the steps suggested in table 1.

-
- s1.** Determine each node neighbours and construct the correspondent Voronoï cells.
 - s2.** Construct the integration points based on the Voronoï cells.
 - s3.** Set the influence-cells (first or second degree).
 - s4.** Loop over the integration points in order to integrate the terms of the Galerkin weak form.
 - i.** Determine the nodes that directly influence the specified integration point, based on the previously defined integration-cells.
 - ii.** Compute the shape functions and its derivatives for each integration point.
 - iii.** Evaluate stiffness, mass and load at each integration point.
 - iv.** Assemble the contribution of the specified integration point in order to form the system of equations.
 - s5.** Introduce essential and natural boundaries.
 - s6.** Solve the algebraic system to obtain the nodal displacement.
 - s7.** Evaluate strain and stress at each integration point.
-

table 1 – NNRPIM static linear flow chart.

These steps correspond to the static analysis, the dynamic analysis can be performed by substituting steps **s6** and **s7** by an eigenproblem solver, such as the Jacobi solver, and the introduction of the superposition method suggested in section 2.3.6, [36]. The nonlinear analysis assumes a recurring process based in the basic flow chart presented in table 1, [39, 40].

3.7 RPI Function Patch Test

The patch test [67] was originally designed to prove the convergence in non-conforming finite element formulation. Generally the test consists on the imposition of a known displacement field in the boundary of the patch. In this work linear patch tests are used. If the prescribed field is reproduced in the interior of the patch then the test is verified. Although being a benchmark for the evaluation and validation of non-conforming elements, in the context of the meshless methods the relevance of the patch test, from the convergence point of view, is still an open issue.

In the present work it is reproduced the patch test of the RPIM [21] in order to clearly explain the major differences between the RPIM and the NNRPIM and to introduce the NNRPIM procedure for the analysis considering the patch test. The patch test exact solution is compared with the meshless solution and the following medium error is obtained,

$$\Theta = \frac{1}{n} \sum_{i=1}^n \frac{\sqrt{(u_i - u_{i \text{ exact}})^2 + (v_i - v_{i \text{ exact}})^2}}{\sqrt{u_{i \text{ exact}}^2 + v_{i \text{ exact}}^2}} \quad (214)$$

In this work the shape parameter c is obtained with,

$$c = \gamma A_G \quad (215)$$

where γ is a parameter that must be determined and A_G is a geometric parameter dependent on the mesh discretization.

3.7.1 RPIM patch test

The RPIM uses influence domains as figure 38(a) indicates. As in [21, 22], for the RPIM, domains of influence of dimension $dm_i = C_i \times h$ are used, where $C_i = 2.5$ and h is the quadratic norm of the maximum distance between neighbour nodes. The background integration mesh is independent of the node mesh and it is created with integration cells each one with a quadrature scheme of 3x3 Gauss points, as figure 38(b) and (c) indicate for irregular and regular node meshes respectively.

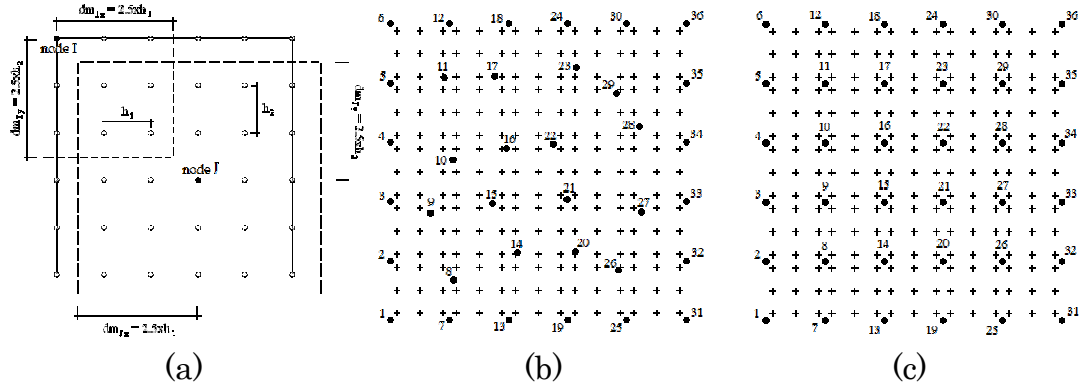


figure 38 – (a) Construction examples of the influence domain. (b) Irregular mesh and (c) Regular mesh used to discretize the problem (“o” = nodes, “+” = Gauss points).

In order to obtain the optimized parameters γ and p , firstly the parameter p is fixed, $p = 0.9999$, and the parameter γ is varied between 50 and 0.04. The shape parameter c is obtained with the expression $c = h \times \gamma$. Once the optimal γ is achieved, the process is repeated but now using the optimal shape parameter c , $c = h \times \gamma$, and varying the shape parameter p , until a optimal p is obtained. In the RPIM it is used the linear polynomial basis, $\mathbf{p}(\mathbf{x}) = \{1 \ x \ y\}$, and the used RBF is the MQ. To perform the patch test, the displacement field imposed on the boundary is defined by,

$$\begin{cases} u = 0.1 + 0.1x \\ v = 0.1 + 0.1y \end{cases} \quad (216)$$

The medium displacement error, Θ , defined in equation (214) is calculated. The sum of the interior equivalent forces is given by the expression,

$$\mathbf{f}_{tot} = \sum_I^{n_i} \mathbf{f}_I^{eq} \quad (217)$$

where n_i is the number of nodes that do not belong to the essential boundary and $\mathbf{f}^{eq}$ is the equivalent force obtained with,

$$\mathbf{f}^{eq} = \int_{\Omega} \mathbf{B}^T \boldsymbol{\sigma} d\Omega \quad (218)$$

The two mesh types indicated in figure 38 are used in this case, an irregular and a regular mesh. The patch has the dimensions of $1.0 \times 1.0 m^2$ and the material properties are $E = 1.0 Pa$ and $\nu = 0.3$. The coordinates of the nodes are presented in table 2.

Node	Irregular Mesh		Regular Mesh		Node	Irregular Mesh		Regular Mesh		Node	Irregular Mesh		Regular Mesh	
	x	y	x	y		x	y	x	y		x	y	x	y
1	0,000	0,000	0,000	0,000	13	0,400	0,000	0,400	0,000	25	0,800	0,000	0,800	0,000
2	0,000	0,200	0,000	0,200	14	0,429	0,226	0,400	0,200	26	0,770	0,167	0,800	0,200
3	0,000	0,400	0,000	0,400	15	0,345	0,394	0,400	0,400	27	0,849	0,364	0,800	0,400
4	0,000	0,600	0,000	0,600	16	0,392	0,580	0,400	0,600	28	0,841	0,655	0,800	0,600
5	0,000	0,800	0,000	0,800	17	0,354	0,823	0,400	0,800	29	0,764	0,765	0,800	0,800
6	0,000	1,000	0,000	1,000	18	0,400	1,000	0,400	1,000	30	0,800	1,000	0,800	1,000
7	0,200	0,000	0,200	0,000	19	0,600	0,000	0,600	0,000	31	1,000	0,000	1,000	0,000
8	0,215	0,135	0,200	0,200	20	0,627	0,230	0,600	0,200	32	1,000	0,200	1,000	0,200
9	0,136	0,359	0,200	0,400	21	0,597	0,407	0,600	0,400	33	1,000	0,400	1,000	0,400
10	0,212	0,541	0,200	0,600	22	0,550	0,593	0,600	0,600	34	1,000	0,600	1,000	0,600
11	0,182	0,818	0,200	0,800	23	0,629	0,852	0,600	0,800	35	1,000	0,800	1,000	0,800
12	0,200	1,000	0,200	1,000	24	0,600	1,000	0,600	1,000	36	1,000	1,000	1,000	1,000

table 2 – Coordinates of the nodes that discretize the patch domain.

In figure 39(a) the medium displacement error Θ is presented, together with the f_{tot} , as function of the shape parameter γ . It is visible that Θ stabilizes for values $\gamma \leq 1.6$. The stabilization and the minimization of f_{tot} is obtained for lower values of γ , $\gamma \leq 0.5$. The parameter value suggested in reference [22] is $c = 1.43$, which corresponds to $\gamma \cong 7$, and $p = 1.03$. It is visible in figure 39(a) that such values for the parameters conducts to a medium displacement error $\Theta = 4 \times 10^{-4}$ and it produces a high value for f_{tot} , $f_{tot} \cong 1.5 \times 10^2$, which is unacceptable, it must be zero. This anomaly is explained with the effective lack of the delta Kronecker property on the RPIM shape functions for high values of γ . It is noted that the value of the shape parameter c determines the fundamental shape of the basis function, with small value of γ the resulting multiquadric radial basis function is cone-shaped and with increasing values of γ the peak of the multiquadric radial basis function becomes flat. As a consequence, the value of the shape parameter γ affects the precision and the accuracy of the multiquadrics interpolation scheme. The increase of the shape parameter implies a increase of the approximation error. Nevertheless to adjust the shape parameter in order to improve the approximation accuracy, low values of γ , causes an ill-conditioned geometric $\mathbf{G}$ matrix or even the singularity of $\mathbf{G}$, [15].

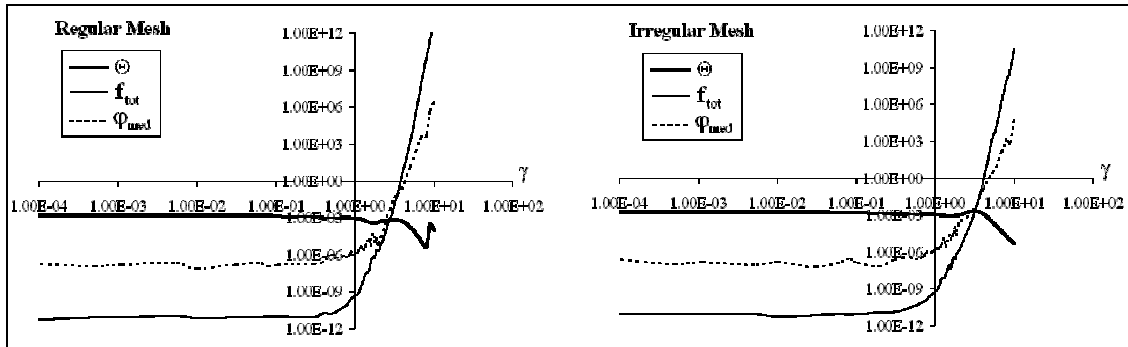
It is now understandable that for high values of γ the shape function does not pass exactly on the nodes and to prove the statement, during previous analysis the shape function vector for the node 15, ϕ_{15} , was saved. The ϕ_{15} values on nodes 9, 14, 16 and 21 were used to calculate ϕ_{med} ,

$$\phi_{med} = \frac{1}{4} (\phi_{(15,9)} + \phi_{(15,14)} + \phi_{(15,16)} + \phi_{(15,21)}) \quad (219)$$

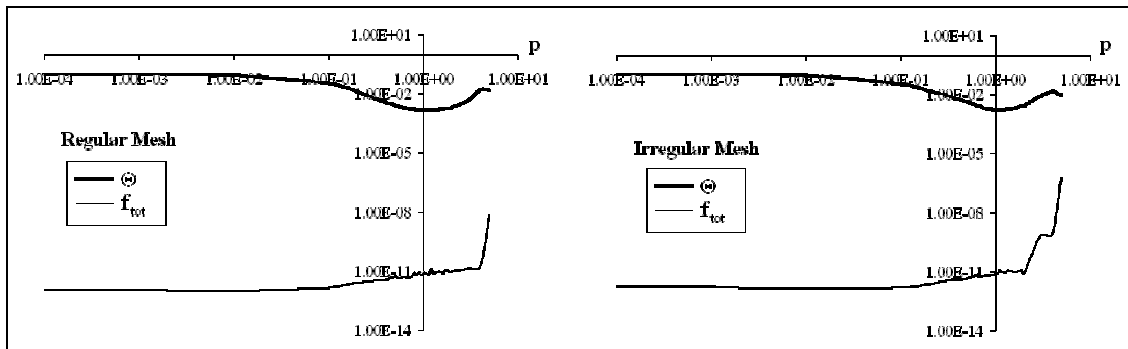
Note that these nodes are the closest nodes to node 15 and if ϕ_{15} has the delta Kronecker property, $\phi_{(15,i)}$ should be zero for all $i \neq 15$, and consequently $\phi_{med} = 0$. However, it is visible in figure 39(a) that ϕ_{med} only stabilizes for $\gamma \leq 0.25$ and even for those values of γ , ϕ_{med} is different of zero, $\phi_{med} \cong 3 \times 10^{-7}$, which indicates the lack of the delta Kronecker property.

As in early works [21, 22] the essential boundary conditions were imposed considering the existence of the delta Kronecker property, as in FEM. This is the reason why f_{tot} is different from zero and even for $\gamma \leq 0.25$ is higher than the machine precision. However, it is acceptable to consider, for $\gamma \leq 0.25$, that the RPIM shape functions possesses the delta Kronecker property.

Considering $\gamma = 0.0001$ and the shape parameter p varying between 10 and 10^{-4} an optimal p is searched. All other considerations regarding to material, geometric and boundary conditions remain the same. The integration scheme and the nodal discretization are also similar to the previous analysis. The results are presented in figure 39(b).



(a)



(b)

figure 39 – (a) Shape parameter γ effect on the accuracy and (b) parameter p effect on the accuracy for the regular and irregular mesh. Logarithmic scales.

It is visible in figure 39(b) that $p \cong 1$ is an optimal value once Θ and γ show a minimum for this value. However, as figure 39(b) illustrates, p can not be exactly equal to 1, or any other integer value, because it will generate a singular $\mathbf{G}$ matrix. Others authors [21, 22] suggest also this value for p . In this work the value used, in the RPIM, for parameter p is $p = 0.9999$.

3.7.2 NNRPIM patch test

In the NNRPIM the procedure to obtain the optimal shape parameters γ and p is similar to the procedure used for the RPIM. All considerations regarding the problem discretization, the material used, the geometry and boundary conditions remain the same as in previous example.

Firstly is presented a study considering the NNRPIM with first degree influence-cell and afterwards it is presented a study of the NNRPIM with the second degree influence-cell. In both formulations only the integration scheme of order 0 is used. The polynomial basis used in both formulations are the null basis, the linear basis and the quadratic basis, equation (198).

In order to identify the applied NNRPIM formulation, acronyms are used for a better understanding. In each acronym the first two characters identifies the degree of the influence cell, first degree = “**v1**” and second degree = “**v2**”. The integration scheme order is identified by “**ik**”, where k is the integration order. The polynomial basis is identified by “**pm**”, where m is the number of monomials in the polynomial basis. For example, the formulation using a second degree influence-cell, an integration scheme of order 1 and a null polynomial basis is called: “**v2i1p0**”.

In the NNRPIM the shape parameter c is obtained with the expression $c = \gamma \times \sqrt{A_{v_i}}$, where A_{v_i} is defined with equation (186). The results of the patch test, concerning the shape parameter γ , for the NNRPIM with influence-cells of first degree are presented in figure 40, for the irregular and regular meshes. For all formulations a parameter $p = 0.9999$ is used.

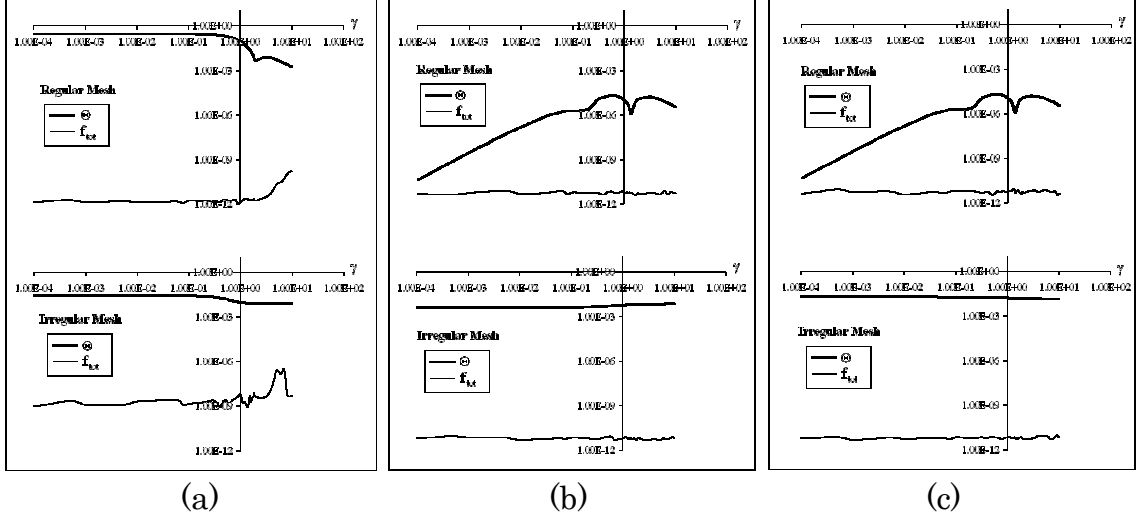


figure 40 – Shape parameter γ effects on the accuracy using (a) the formulation v1i0p0, (b) the formulation v1i0p1 and (c) the formulation v1i0p3. Logarithmic scales.

It was observed that the use of the quadratic polynomial basis on this formulation, v1i0p6, generates a $\mathbf{G}$ matrix singular, i.e. non invertible. Thus the study of this polynomial basis applied to this formulation was abandoned. As figure 40 indicates, for the v1i0p0 formulation the optimal values for the shape parameter γ are $\gamma \geq 2.0$, however the stress field produced with this formulation is highly irregular (it should be constant) and the noted irregularity grows with the increasing of γ . Thus the value suggested for this formulation is $\gamma = 2.0$. The v1i0p1 formulation and the v1i0p3 formulation are quite similar, both show that the optimal value for the shape parameter γ is $\gamma < 0.5$. In further examples, for these two formulations, the shape parameter γ used is $\gamma = 0.0001$.

Using the suggested shape parameters γ for each one of the NNRPIM formulations, the parameter p is varied between 10 and 10^{-4} . The results are presented in figure 41, for the irregular and regular meshes

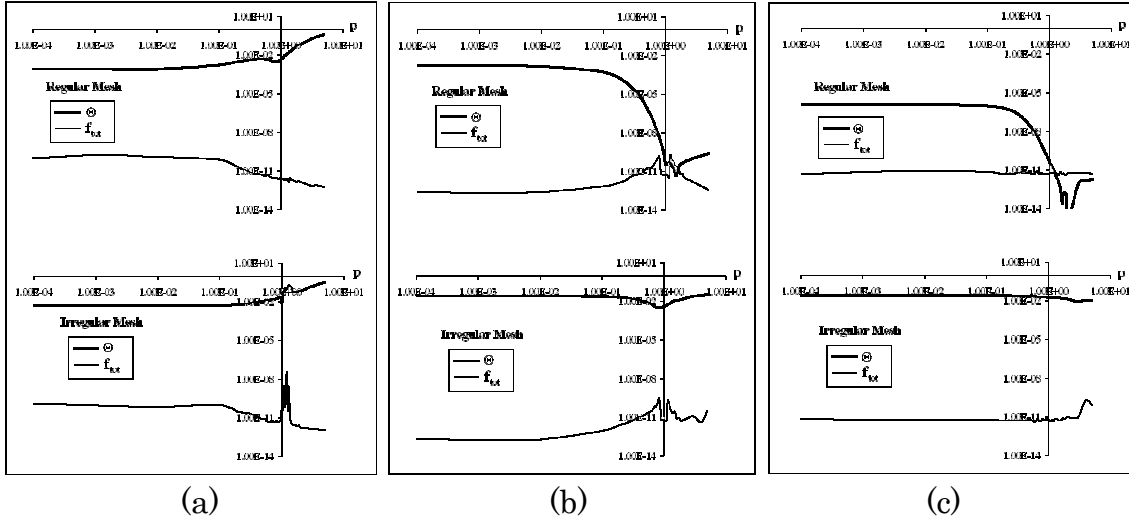


figure 41 – Shape parameter p effects on the accuracy using (a) the formulation v1i0p0, (b) the formulation v1i0p1 and (c) the formulation v1i0p3. Logarithmic scales.

As figure 41(a) shows the solution only stabilizes for $p < 0.015$ in the v1i0p0 formulation. In further examples with this formulation, $p = 0.0001$ is used. For the v1i0p1 and v1i0p3 formulation is visible for both formulations, figure 41(b) and (c), that the solution only stabilizes for $p < 0.1$, however the optimal value for parameter p is achieved near the value 1.0. Thus, the value adopted for parameter p in these two formulations is $p = 0.9999$. It is important to refer that the v1i0p0 fails the aim of producing an acceptable smooth stress field, for this reason the study of this formulation is abandoned.

The NNRPIM with the second degree influence-cell is presented now. The procedure is similar to the previous formulation and the results of the patch test, concerning the shape parameter γ , are presented in figure 42 for the irregular and regular meshes.

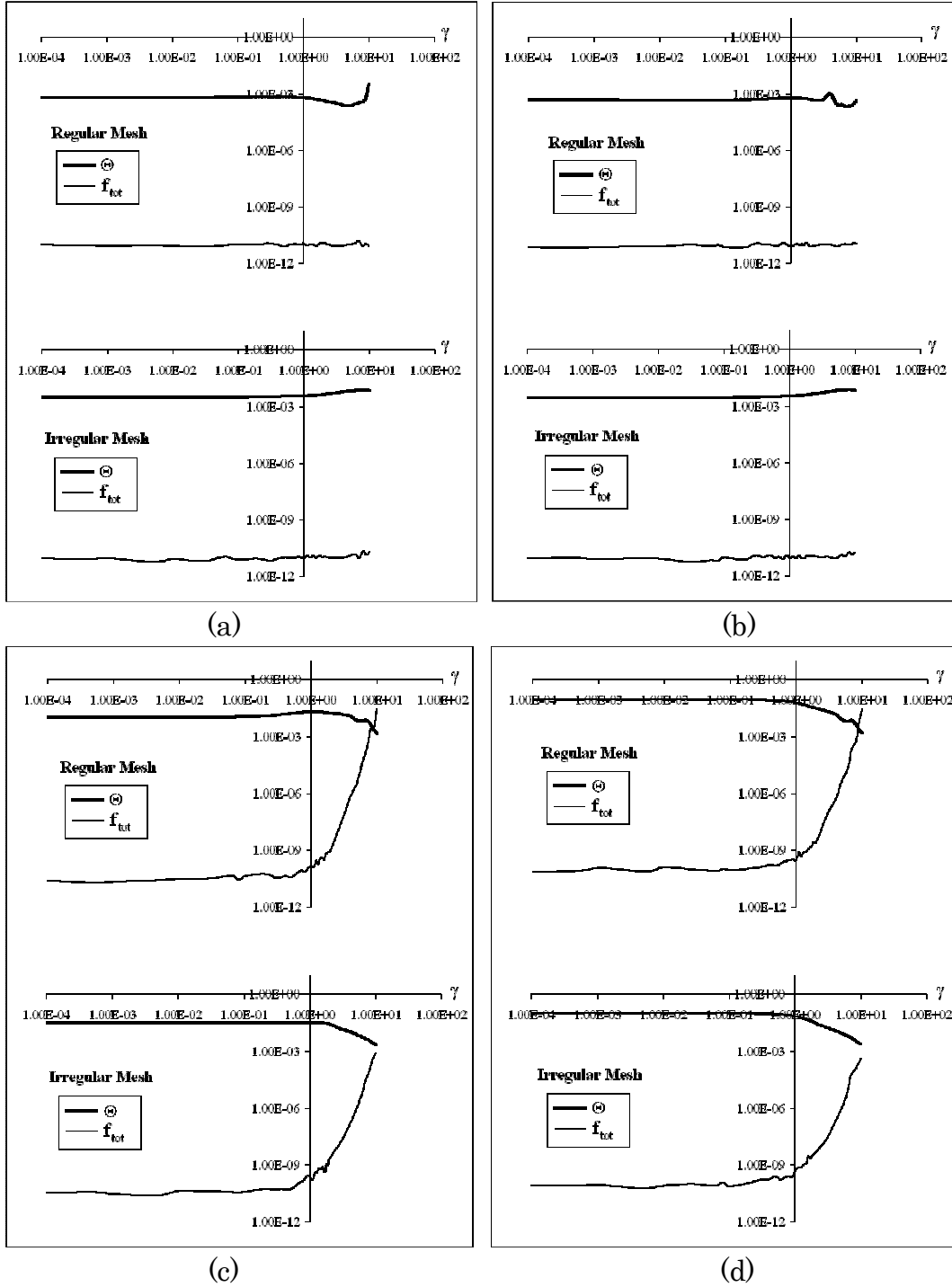


figure 42 – Shape parameter γ effects on the accuracy using (a) the formulation v2i0p0, (b) the formulation v2i0p1, (c) the formulation v2i0p3 and (d) the formulation v2i0p6. Logarithmic scales.

It is visible, as in the RPIM, that the NNRPIM, with the second degree influence-cell formulation, evidences the same problem related with the effective lack of the delta kronecker property. It is also visible that for all formulations the solution only stabilizes for shape parameters values $\gamma < 0.1$.

Using for all formulations a shape parameter $\gamma = 0.0001$ the value of parameter p is varied between 10 and 10^{-4} . The obtained results are presented in figure 43.

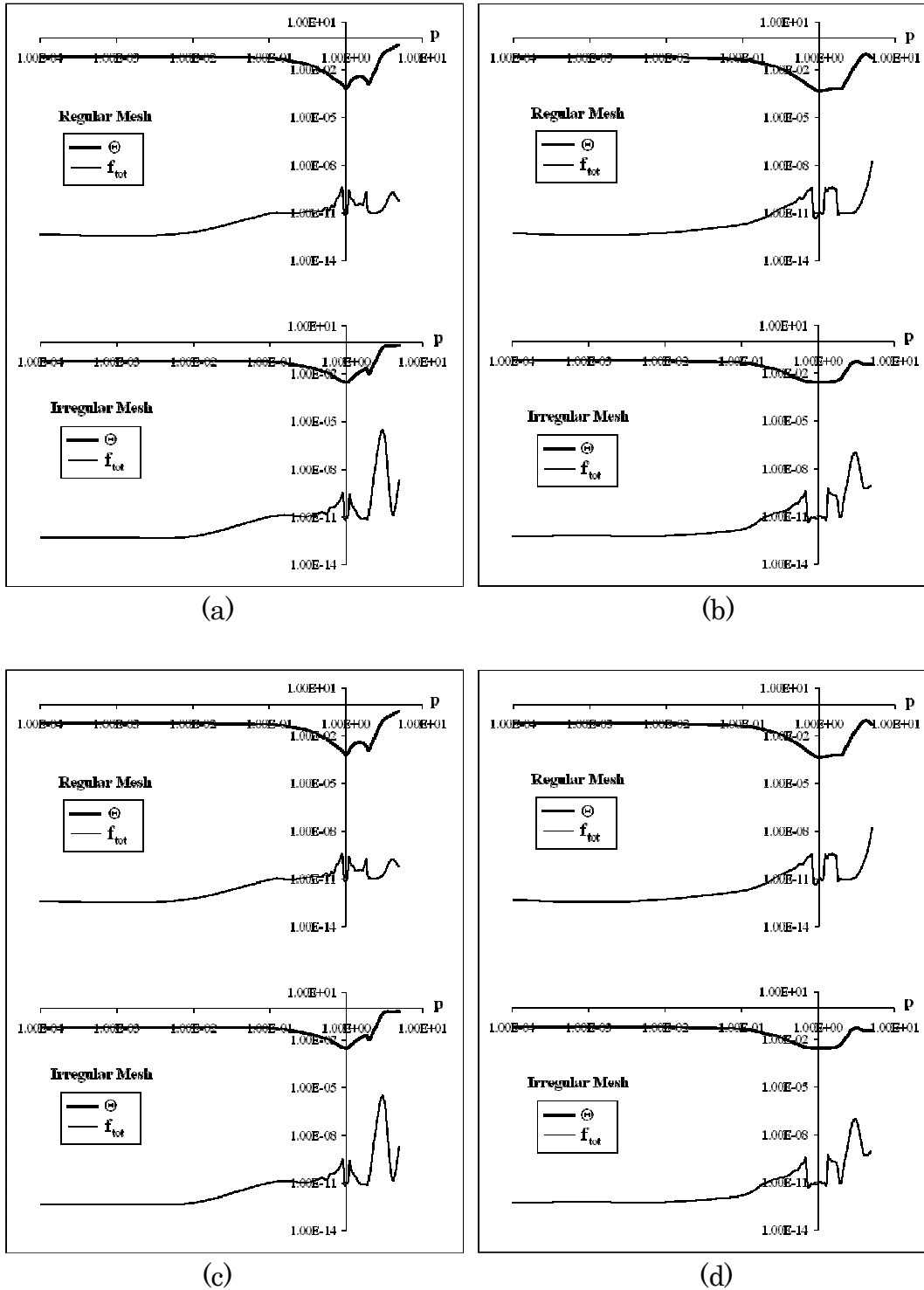


figure 43 – Shape parameter p effects on the accuracy using (a) the formulation v2i0p0, (b) the formulation v2i0p1, (c) the formulation v2i0p3 and (d) the formulation v2i0p6. Logarithmic scales.

Once again, it is desirable to avoid integer values for shape parameter p . In figure 43 is visible that such values produce invalid results, which are clear indicators of the ill-condition of matrix $\mathbf{G}$. It is also shown that $p \cong 1.0$ is an optimal value for all formulations. It seems that for the v2i0p6 formulation $p \cong 2.0$ is a better value, however for this value the generated shape function does not present the delta kronecker property, this notorious deficiency is shown in the f_{tot} values, as in the RPIM. Therefore in further examples with these formulations the parameter p used is $p = 0.9999$.

It was observed that the stress field obtained with the NNRPIM, with the second degree influence-cell formulation, is constant, as it should be, unlike the stress field obtained with the NNRPIM with the first degree influence-cell formulation. This is an important feature, as it is shown in further examples, that influences the accuracy and smoothness of the stress field. In table 3 the obtained Θ and f_{tot} using the optimal shape parameters in the different formulations are summarized.

	Shape Parameters		Regular Mesh		Irregular Mesh	
	γ	p	Θ	f_{tot}	Θ	f_{tot}
RPIM	0.0001	0.9999	1.51E-03	5.74E-12	1.67E-03	9.57E-12
v1i0p0	2.0000	0.0001	8.00E-04	9.87E-11	5.10E-03	1.17E-10
v1i0p1	0.0001	0.9999	4.45E-11	4.88E-12	4.03E-03	6.56E-12
v1i0p3	0.0001	2.9999	1.09E-12	5.79E-12	8.89E-03	9.41E-12
v2i0p0	0.0001	0.9999	6.51E-04	9.63E-12	3.04E-03	8.77E-12
v2i0p1	0.0001	0.9999	4.59E-04	7.26E-12	2.67E-03	8.40E-12
v2i0p3	0.0001	0.9999	1.02E-02	2.52E-11	3.15E-02	3.30E-11
v2i0p6	0.0001	0.9999	6.50E-02	3.38E-12	8.98E-02	2.27E-12

table 3 – Obtained results for optimal shape parameters for the 2D analysis.

As table 3 shows the lowest medium displacement error Θ , for irregular meshes, in the NNRPIM is obtained with the v2i0p1 formulation. This error is very close to the error obtained with the RPIM. It is noted as well that generally the error obtained for regular meshes with NNRPIM is lower than the error obtained with RPIM. In this section the NNRPIM study will continue only with the v1i0p0, v1i0p1, v2i0p0 and v2i0p1 formulations, since the lowest medium displacement error is obtained with these formulations. It is important to refer that the RPIM analysis is an optimized analysis because adequate shape parameters and proper integration schemes were applied. However in the case of the NNRPIM it was applied

only the integration scheme of order 0. Using the optimal shape parameters a study to obtain the optimal integration scheme is carried out.

The irregular mesh presented in figure 38(b) is used and the optimal shape parameters presented in table 3 are applied to the RPIM and NNRPIM formulations. The problem is studied for various integration schemes for each formulation. The obtained medium displacement errors are presented in figure 44.

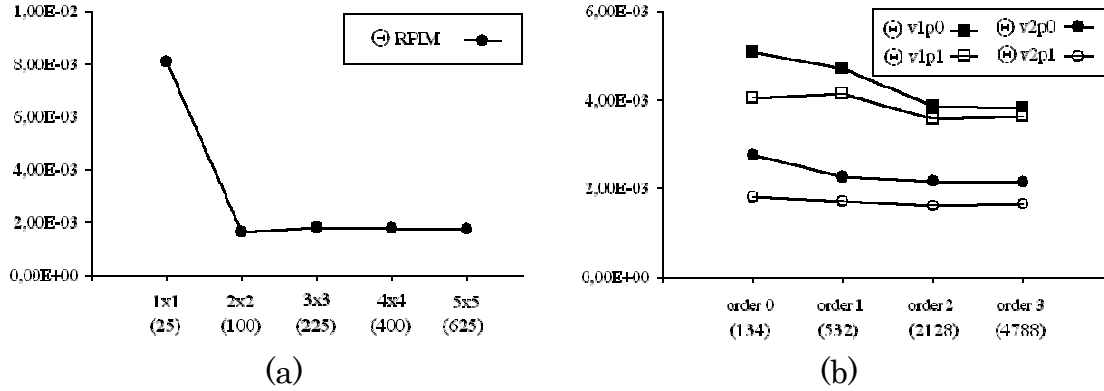


figure 44 – Influence of the integration scheme on the accuracy for (a) RPIM (a) and for (b) NNRPIM.

It must be noticed that the number of integration points used in the distinct integration schemes in the NNRPIM analysis increases by a factor of 4, which means that the computational cost will increase in the same proportionality. It is visible that the solution only stabilizes for integration orders greater than “*order 1*”, however it is reasonable to use the integration scheme of order 0 as an acceptable integration scheme, since it minimizes the computational cost without jeopardizing to much the method accuracy.

The three-dimensional patch test was also studied, an unit cubic solid with a volume of $1.0 \times 1.0 \times 1.0 m^3$ and with the same material properties as the two-dimensional example was submitted to an enforce displacement in all essential boundaries,

$$\begin{cases} u = 0.1 + 0.1x \\ v = 0.1 + 0.2y \\ w = 0.1 + 0.3z \end{cases} \quad (220)$$

Once again the intend of the analysis was to obtain the optimal shape parameters, c and p , for the 3D formulation. The solid was discretized with two distinct meshes, a regular mesh and an irregular mesh, both with $6 \times 6 \times 6$

nodes (216 nodes). The procedure was similar with the 2D patch-tests examples. The optimal shape parameters obtained [29] are presented in table 4.

	Shape Parameters		Regular Mesh		Irregular Mesh	
	γ	p	Θ	f_{tot}	Θ	f_{tot}
v1i0p0	0.0001	0.9999	7.66E-04	2.26E-14	1.65E-02	1.36E-10
v1i0p1	0.0001	0.9999	1.75E-03	3.18E-14	1.37E-02	4.33E-10
v1i0p3	0.0001	0.9999	1.60E-02	2.60E-13	3.41E-02	4.02E-10
v2i0p0	0.0001	0.9999	3.32E-03	6.61E-14	8.00E-03	3.69E-10
v2i0p1	0.0001	0.9999	3.20E-03	7.16E-14	7.30E-03	2.89E-09
v2i0p3	0.0001	0.9999	3.31E-02	4.41E-13	4.61E-02	6.39E-08
v2i0p6	0.0001	0.9999	5.31E-02	9.20E-13	5.68E-02	4.06E-09

table 4 – Obtained results for optimal shape parameters for the 3D analysis.

In the 3D study it was observed that the v1i0p0, v1i0p3, v2i0p3 and v2i0p6 NNRPIM formulations produce a stress field very irregular, which should be constant. In the table is also perceptible that the referred last three formulations compared with the others NNRPIM formulations present a higher medium displacement error. As so, in further examples using a 3D analysis the used NNRPIM formulations are the v1i0p1, v2i0p0 and the v2i0p1. Due to the significant computational cost, in this work, for the three-dimensional analysis the used integration scheme is the order 0. To conclude this section, in table 5 are displayed the optimal parameters for the distinct formulation.

Parameters	RPIM 2D	v1p1 2D	v2p0 2D	v2p1 2D	v1p1 3D	v2p0 3D	v2p1 3D
γ	0.0001	0.0001	0.0001	0.0001	0.0001	0.0001	0.0001
p	0.9999	0.9999	0.9999	0.9999	0.9999	0.9999	0.9999
Integration Scheme	3x3	order 2	order 1	order 1	order 0	order 0	order 0

table 5 – Selected optimal shape parameters.

3.8 NNRPIM Shell-Like Formulation – Optimization

The objective of the shell-like approach is to reduce the number of integration points used in the analysis of thin structures and the number of nodes discretizing the problem domain. Generally numerical methods, meshless or not, behave better in the presence of regular meshes. In the 3D analysis, if no special consideration regarding the spatial directions is made in the construction of the interpolation or approximation functions, the best

node mesh discretizing the problem domain is always a regular mesh which distributes nodes in unit ratio cubes. If the distance between two closest nodes in the zz direction is $d_{zz} = \xi$, then the distance between two closest nodes in the yy direction will be also $d_{yy} = \xi$ and in the xx direction it will be $d_{xx} = \xi$ again, i.e., $d_{xx}/d_{yy} = d_{yy}/d_{zz} = d_{xx}/d_{zz} = 1$. One of the purposes of this work is to achieve a formulation that permits the use of a higher ratio between the nodal distance in the distinct spatial coordinates. Therefore, nodes are distributed only in the upper face and bottom face of the problem domain, figure 28, and the following ratio is the objective, $d_{xx}/d_{yy} = 1 \wedge (d_{yy}/d_{zz} = d_{xx}/d_{zz}) \gg 1$. Consider a square isotropic plate with all edges simply supported, $L = D = 10m$ and $h = 0.5m$. The material properties of the problem are $E = 1.0MPa$ and $\nu = 0.3$. The obtained results are compared with the Navier Solution, a transverse bending plate analytical solution (2D-AS). In order to simulate the nearest like deformation conditions of the 2D analytical solution, only the superior mid-half of the plate is considered and the nodes at $z = 0$ have the displacements in the x and y directions constrained. In the surface $z = 0$ it is applied a uniformly distributed load, $f = 1.0 kN/m^2$. Also considering the natural symmetry only a quarter of the plate is considered in the analysis. These features can be seen in figure 45. In this example $d_{xx}/d_{yy} = 1$ and only the geometrical parameter λ varies, being $\lambda = d_{yy}/d_{zz} = d_{xx}/d_{zz} \geq 1$. In this stage the problem domain is discretized only in regular meshes, figure 45.

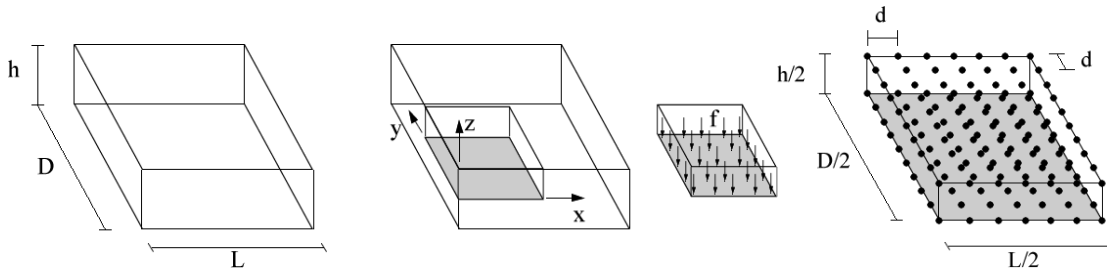


figure 45 – Geometric and load properties of the square plate and regular nodal mesh distribution.

Notice that maintaining the dimensions of the plate and increasing the discretization along x and y the value of λ decreases. Maintaining the discretization along x and y and reducing the thickness of the plate, the value of λ increases.

Firstly the integration scheme is studied. The problem domain is discretized in several regular node meshes, varying the parameter λ . For each one of these studied meshes the 2D integration mesh obtained, as in figure 28, is reproduced according to the Gaussian distribution rule along the thickness of the plate, from 2 to 10, in order to obtain the best integration distribution. However it was observed that for Gaussian distributions higher than 5 the results are the same, as so only the results from 2 to 5 are presented. In table 6 are presented the centre transverse displacement results regarding the NNRPIM 3D Shell-Like formulation (3DSL), the results obtained with the NNRPIM using a 3D classical formulation (3DCF) and the FEM results obtained with the 8 node brick element (FEM) with a integration Gauss distribution $2 \times 2 \times 2$.

It is perceptible that the NNRPIM 3DSL formulation converges faster to the final solution in comparison with the FEM and also that the final converged result is closer to 2D-AS than the FEM solution. About the best integration scheme it can be said that using a Gaussian distribution GD=2 is enough, however it is only with GD=3 that the solution truly stabilizes. Thus, in the next plate examples it is always used the Gaussian distribution GD=3. The computational time spend in the analysis for the distinct NNRPIM formulations in shown in table 7. It is visible that the 3DSL formulation can be three times faster than the 3DCF NNRPIM.

The purpose is now to study the optimal range values for λ . The same square plate problem is analysed. The domain discretization is fixed with the regular mesh $25 \times 25 \times 2$ nodes, as so for the λ parameter to vary it is necessary to change the thickness of the plate. Note that as it is a shell-like formulation it is only considered a top and bottom face mesh discretization, there is no nodes along thickness. The thickness h varies between 0.2 and 0.004, and the other two dimensions are fixed, $L = D = 10m$. For each value of λ the plate centre transverse displacement is obtained and then the error in relation to the analytical solution of Navier is calculated,

$$\Theta_w = \frac{|w_{Navier} - w_{NNRPIM}|}{w_{Navier}} \quad (221)$$

The results are presented in figure 46, which shows that the minimum error is obtained for values of λ between 2 and 5.

node mesh	nodes	λ	NNRPIM				3D-CF	FEM	2D-AS
			3D-SL						
			GD = 2	GD = 3	GD = 4	GD = 5			
5x5x2	50	5.00	3.1235	3.1266	3.1266	3.1266	3.2786	1.7005	3.5872
7x7x2	98	3.33	3.4858	3.4910	3.4909	3.4909	3.6958	2.3769	3.5872
9x9x2	162	2.50	3.5965	3.6032	3.6031	3.6031	3.8237	2.7660	3.5872
11x11x2	242	2.00	3.6444	3.6522	3.6522	3.6522	3.8778	2.9957	3.5872
13x13x2	338	1.67	3.6712	3.6800	3.6800	3.6800	3.9068	3.1388	3.5872
15x15x2	450	1.43	3.6888	3.6984	3.6984	3.6984	3.9248	3.2329	3.5872
17x17x2	578	1.25	3.7011	3.7116	3.7116	3.7116	3.9368	3.2976	3.5872
19x19x2	722	1.11	3.7099	3.7212	3.7213	3.7213	3.9450	3.3438	3.5872
21x21x2	882	1.00	3.7160	3.7282	3.7283	3.7283	3.9505	3.3779	3.5872

table 6 – Centre transverse displacement for distinct formulations and increasing nodal discretization.

node mesh	nodes	λ	3D-SL				3D-CF
			GD = 2	GD = 3	GD = 4	GD = 5	
5x5x2	50	5.00	1.93	2.81	3.65	4.36	9.70
7x7x2	98	3.33	5.57	7.66	9.86	12.14	17.84
9x9x2	162	2.50	10.83	15.38	19.44	24.21	35.05
11x11x2	242	2.00	19.27	26.67	33.53	40.60	59.20
13x13x2	338	1.67	28.47	41.01	51.34	63.64	90.59
15x15x2	450	1.43	41.05	56.95	73.79	90.53	132.36
17x17x2	578	1.25	56.72	77.80	102.09	126.06	185.23
19x19x2	722	1.11	79.79	106.13	139.51	168.00	242.46
21x21x2	882	1.00	108.52	143.75	184.75	221.70	314.35

table 7 – Computational time in seconds for the distinct NNRPIM formulations and increasing nodal discretization.

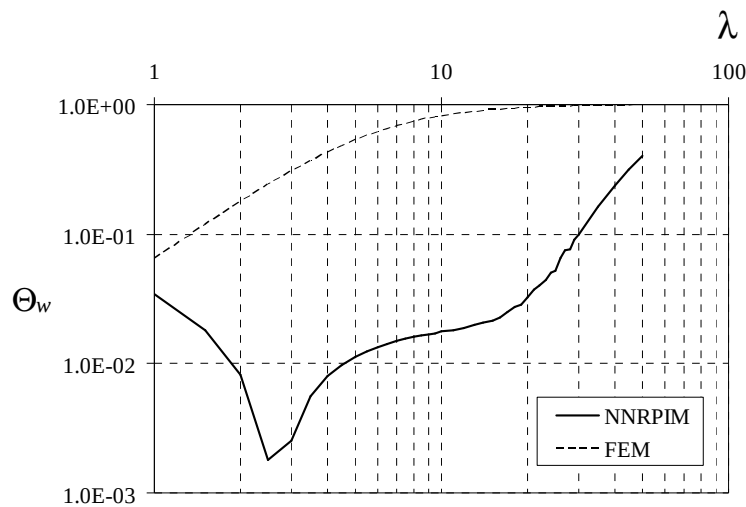


figure 46 – Relative error obtained in the analysis when the parameter λ varies.

Notice that even for values of λ near 10 the error is small. It is also visible that the FEM does not behave very well when one of the dimensions is very small compared with the other two.

3.9 Conclusions and Remarks

In this chapter the Natural Neighbour Radial Point Interpolation Method, NNRPIM, was presented. The NNRPIM is a truly meshless method, since the interpolation function construction and the integration scheme depends uniquely on the nodal mesh used to discretize the problem domain.

The NNRPIM uses geometric concepts such as the Voronoï diagrams to enforce the nodal connectivity, building ‘influence-cells’ for that matter. There are two types of ‘influence-cells’, the first degree influence-cell, which imposes a lower connectivity, and the first degree influence-cell, where a much higher connectivity is imposed. It was shown, despite the relative higher computational cost, that second degree influence cells leads to more stable and accurate results. The integration scheme used in the NNRPIM is totally dependent on the nodal mesh. Using the Delaunay construction the integration mesh is constructed. Several integrations orders were suggested, however it was found, after comparing the computational cost with the achieved accuracy, that the integration scheme of order 0 was sufficient. The Radial Point Interpolators (RPI), used to construct the interpolation functions in the NNRPIM, are typically build with two blocks, a Radial Basis Function (RBF) and polynomial basis. The multiquadrics RBF depends on two shape parameter, c and p . The variation of such shape parameters changes the RPI behaviour. The shape parameters wrong choice leads to RPI functions that do not possess the delta Kronecker property. In this work optimal shape parameter, c and p , were found, granting to the RPI the delta Kronecker property, permitting an easier imposition of boundary conditions. It was also found that the use of a polynomial basis is not important, and that the RPI can pass the standard patch without such polynomial basis.

To conclude, a stable and accurate NNRPIM formulation is the one that uses:

- i.* a second-degree influence cell
- ii.* an integration scheme of order 0
- iii.* a constant (or a null) polynomial basis
- iv.* the RBF shape parameter, $c = 0.0001$ and $p = 1.0001$

This is the typical NNRPIM formulation used in this work. Although these achieved optimal parameters are valid for the 2D, the 3D and the plate and laminate analysis, [29, 32], for the 3D analysis it was found that the first-order influence cell is sufficient. Within the 3D Shell-Like formulation it was obtained that respecting $\lambda \in [2.0, 5.0]$ for the geometric parameter λ , the solution improves and the computational cost decreases.

Chapter 4

4 Linear Applications

In this chapter several linear benchmark examples are presented. Firstly, it are analysed simple static examples, where the method high convergence and accuracy are shown. Afterwards, more complex examples are solved. Then, the 3D shell-like formulation is presented and its application field is explained. The dynamic analysis follows the same process, first simple benchmark examples and in the end more complex studies.

4.1 Static Analysis

The static analysis of structures generally is the most important study within the structural project. Accurate strain and stress fields are very important to optimize the structure design. This section presents a series of benchmark static examples, which are successfully solved by the NNRPIM.

4.1.1 The Unit Square Plate under Uniaxial Stress

An unit square plate is submitted to an uniform uniaxial stress, $\sigma = 10 \text{ kPa}$, in the x direction. The material, loading, geometry and boundary conditions are illustrated in figure 47(a). The analytical solution to this problem is given by,

$$\begin{aligned} u &= \frac{\sigma}{E} \cdot \frac{x}{L} \\ v &= -\frac{\nu \sigma}{E} \cdot \frac{y}{L} \end{aligned} \tag{222}$$

The problem is discretized considering irregular and regular meshes. In figure 47(b) and (c) an irregular and a regular mesh of 169 nodes are presented, which will be used in the problem analysis.

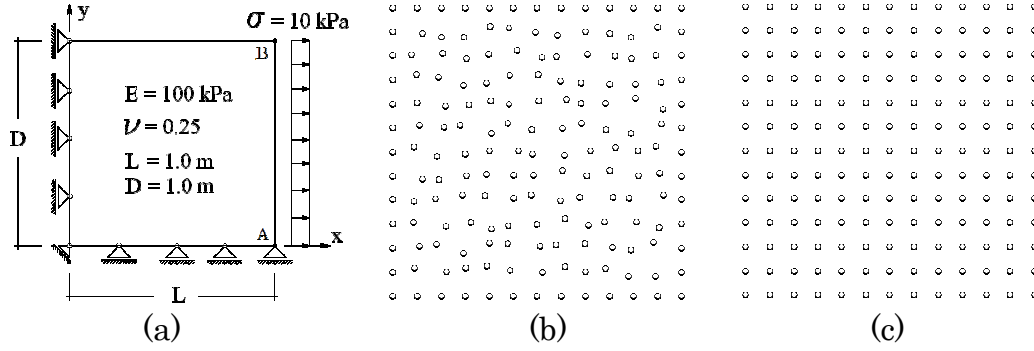


figure 47 – (a) Considered material and geometric conditions of the unit square plate problem. (b) Irregular 169 node mesh. (c) Regular 169 node mesh.

In figure 48 is presented the convergence study for the displacement values of points A and B, represented in figure 47. It is possible to observe in figure 48 that the NNRPIM formulation convergence rate in comparison with the RPIM formulation is higher and the converged result is very near the exact solution ($u_A = 0.1, u_B = 0.1$ and $u_A = 0.025$).

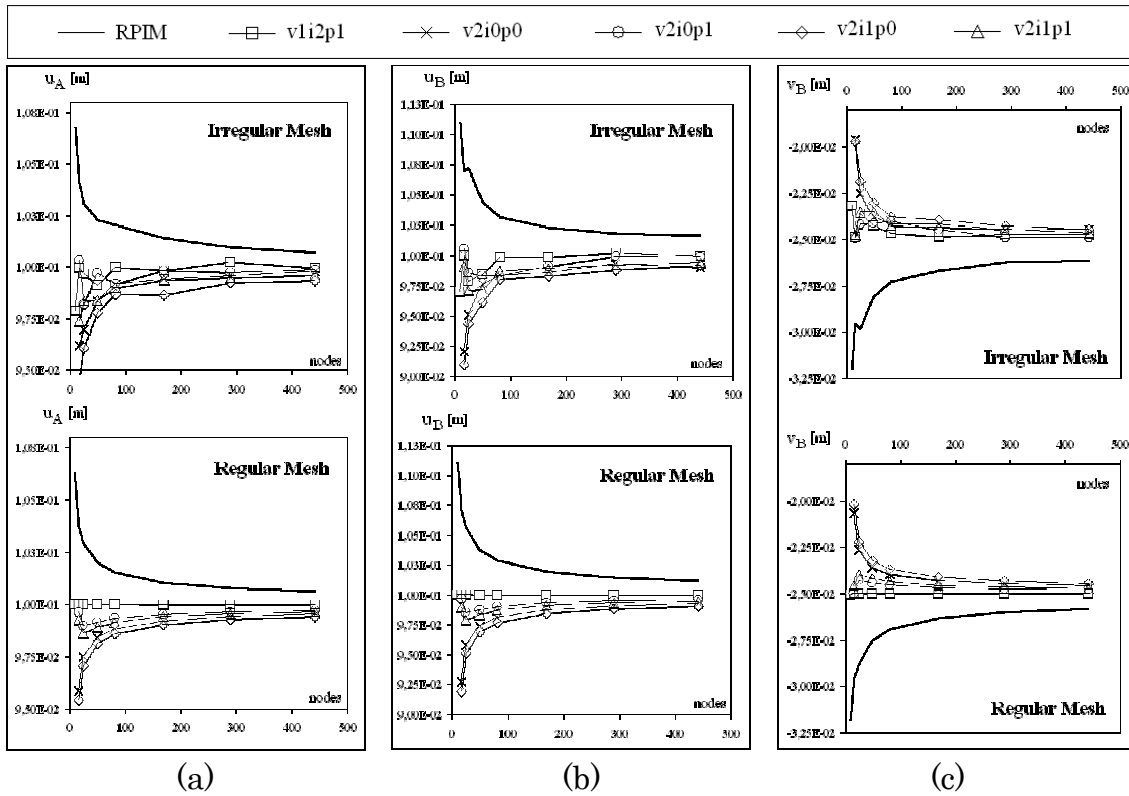


figure 48 – Displacement values for the interest points.

The results, for the irregular and regular meshes respectively, of the convergence study for the medium displacement error, equation (214), are shown in figure 49(a) and (b). It is visible that using an irregular mesh the NNRPIM v1i2p1 formulation presents a high medium displacement error and when the regular mesh is considered the error is very low (10^{-4}). Both

figure 49(a) and (b) enforces the former statement, the NNRPIM formulations have a high convergence rate in comparison with the RPIM formulation. The present problem should generate a constant stress field,

$$\boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{xx} & \sigma_{yy} & \sigma_{xy} \end{Bmatrix} = \begin{Bmatrix} \sigma & 0 & 0 \end{Bmatrix} \quad (223)$$

where $\sigma = 10 \text{ kPa}$. Thus the medium stress error $\Theta_{\sigma_{xx}}$ is calculated using the expression,

$$\Theta_{\sigma_{xx}} = \frac{1}{n_Q} \sum_{i=1}^{n_Q} \frac{|\sigma_{xx i} - \sigma_{xx exact}|}{|\sigma_{xx exact}|} \quad (224)$$

Being n_Q the number of integration points. In figure 49(c) and (d) the results for the irregular and regular meshes are shown.

It is visible in figure 49(d) that the NNRPIM formulations for regular meshes converge faster to the exact solution than the RPIM formulation. It is also visible in figure 49(c) that when an irregular mesh is used the v2i1p0 and the v2i1p1 are the only NNRPIM formulations that produce similar results with the RPIM formulation.

Another interesting issue is the computational effort of the method. The medium displacement errors in relation to the time spent in the analysis are presented in figure 50.

As figure 50 shows, using a regular mesh, the NNRPIM, with the lower integration scheme, presents a lower computational cost when compared with the RPIM and that the time spent in the analysis is even lower when irregular meshes are considered, which is explained with the lower number of nodes in each influence-cell in the case of the irregular meshes. It must be noted that, the computational cost of the regular meshes is about the double of the irregular meshes. The NNRPIM with the v1i2p1 formulation presents a very high computational cost, which is explained by the number of integration nodes generated with the second degree integration scheme. In the second degree influence-cell NNRPIM formulations no significant differences are observed using an integration scheme of order 0 or of order 1. As so, in further examples only the v2i0p0 and the v2i0p1 are used once these formulations present a much lower computational cost.

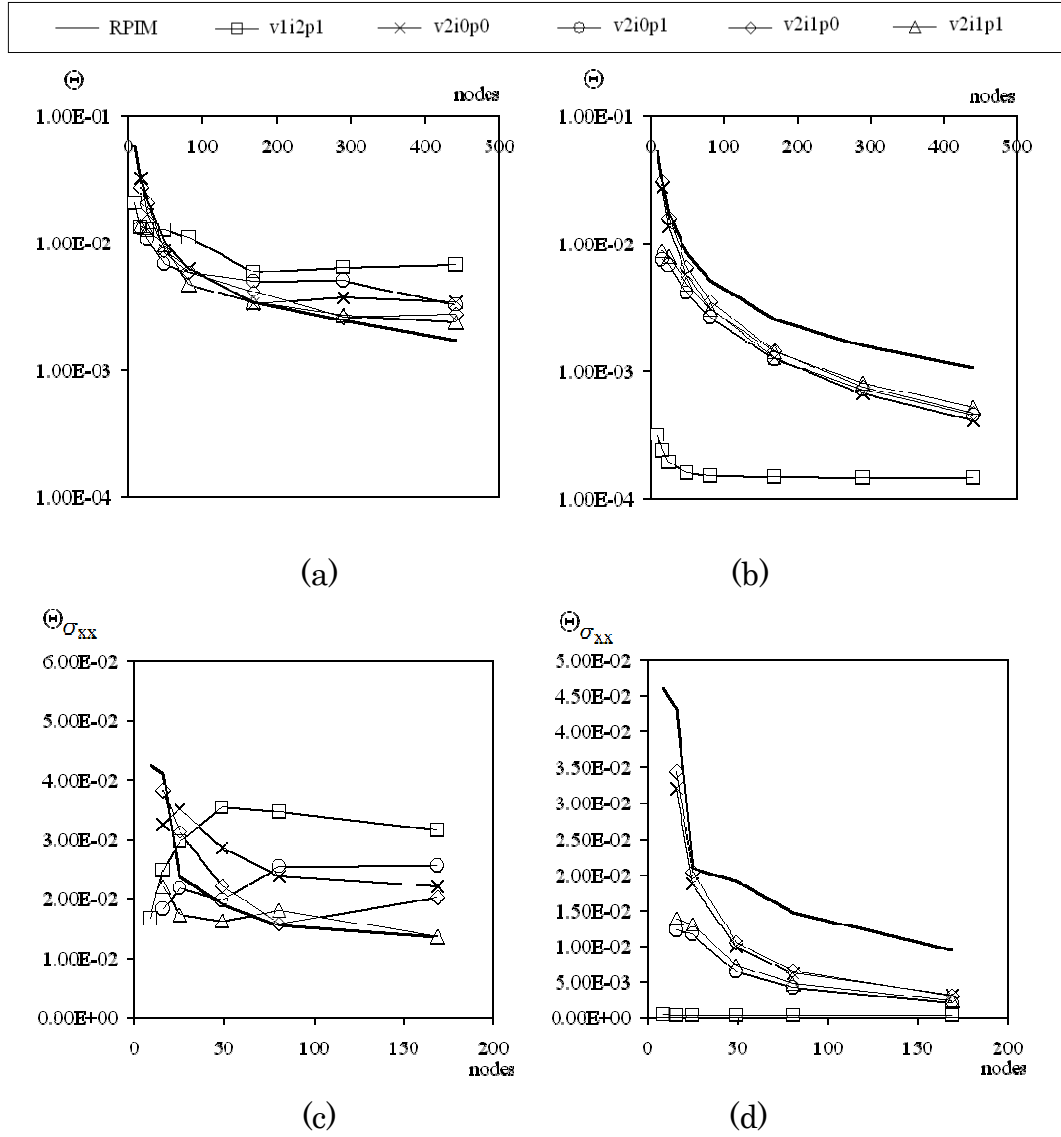


figure 49 – Medium displacement and stress errors in relation to the number of nodes obtained with the: irregular mesh (a) and (c) and with the regular mesh (b) and (d). yy axis in logarithmic scale.

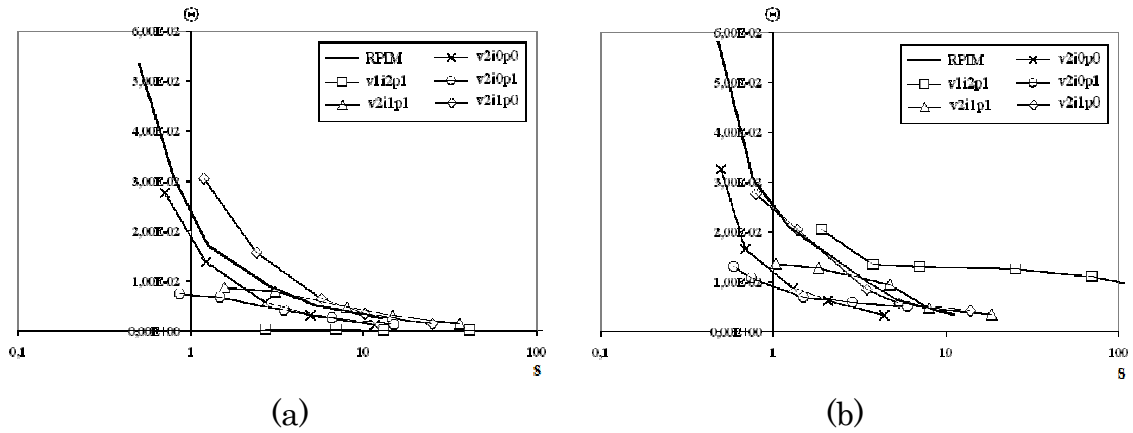


figure 50 – Medium displacement error, in relation to the computational cost, in a logarithmic scale, obtained with the (a) regular mesh and with the (b) irregular mesh.

4.1.2 The Cantilever Beam

The cantilever beam problem is also used to study the accuracy of the NNRPIM. The schematic illustration of the problem is presented in figure 51.

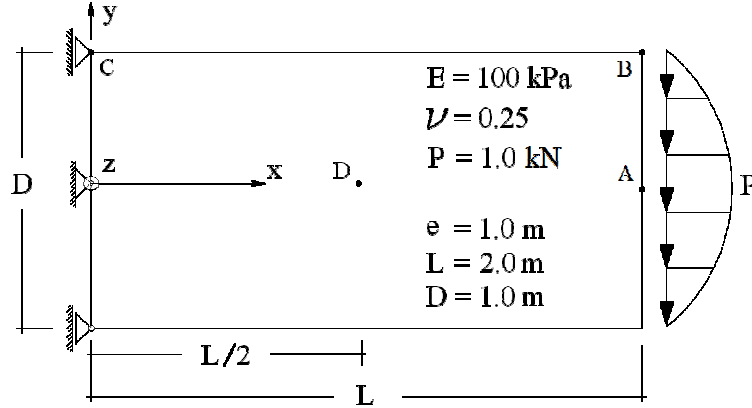


figure 51 – Considered material and geometric conditions of the cantilever beam problem.

The exact solution for the displacement field of the 2D problem is given by,

$$\begin{aligned} u &= -\frac{P y}{6 E I} \left[(2 L - x) 3 x + (2 + \nu) \left(y^2 - \frac{D^2}{4} \right) \right] \\ v &= \frac{P}{6 E I} \left[x^2 (3 L - x) + 3 \nu (L - x) y^2 + \frac{4 + 5 \nu}{4} D^2 x \right] \end{aligned} \quad (225)$$

where I is the moment of inertia, $I = D^3/12$. The stress field for the same dimensional problem is obtained with,

$$\begin{aligned} \sigma_{xx} &= -\frac{P (L - x) y}{I} \\ \sigma_{yy} &= 0 \\ \sigma_{xy} &= \frac{P}{2 I} \left(\frac{D^2}{4} - y^2 \right) \end{aligned} \quad (226)$$

The problem is solved considering the 2D plane stress analysis and the three-dimensional analysis. The domain is discretized in regular and irregular meshes. In figure 52(a) and (b) is given an example of a regular and an irregular mesh of 231 nodes used in the 2D problem analysis. In figure 52(c) and (d) is given an example of a regular and an irregular mesh of 637 nodes used in the 3D problem analysis.

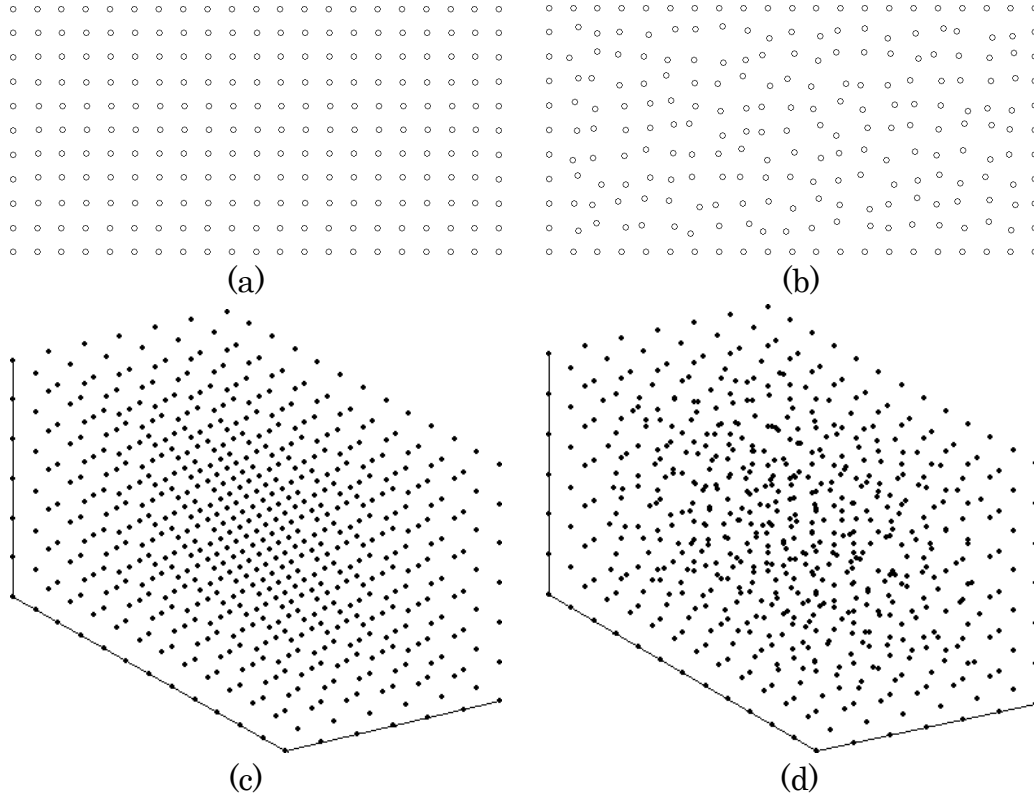


figure 52 – (a) 2D Regular 231 node mesh. (b) 2D Irregular 231 node mesh. (c) 3D Regular 637 node mesh. (d) 3D Irregular 637 node mesh.

In order to compare the obtained results between the 3D analysis and the 2D analysis, in the following diagrams only the number of nodes of the face Oxy of the 3D mesh are represented. In figure 53 are shown the medium displacement errors, Θ , and the medium stress errors, $\Theta_{\sigma_{xx}}$ and $\Theta_{\sigma_{xy}}$. These results were obtained using regular meshes and irregular meshes. In figure 54 are presented the displacement values of interest points A and B, in relation to the number of nodes used to discretize the problem, for the regular and the irregular meshes.

It is visible in figure 53 and figure 54 that once again, within the NNRPIM formulation, when an irregular mesh is considered the vli2p1 generates the worst result. It is also visible in figure 53 that the NNRPIM formulation has a faster convergence to the exact solution when compared with the RPIM formulation. In general the medium errors are lower when second degree influence-cells are used, in the 2D and in the 3D analysis. The results for the 3D analysis seems to be worst than the ones produced by the 2D analysis. This discrepancy is mainly due to the difficulty of comparing the different discretization of the problem for the distinct analysis, 2D and 3D.

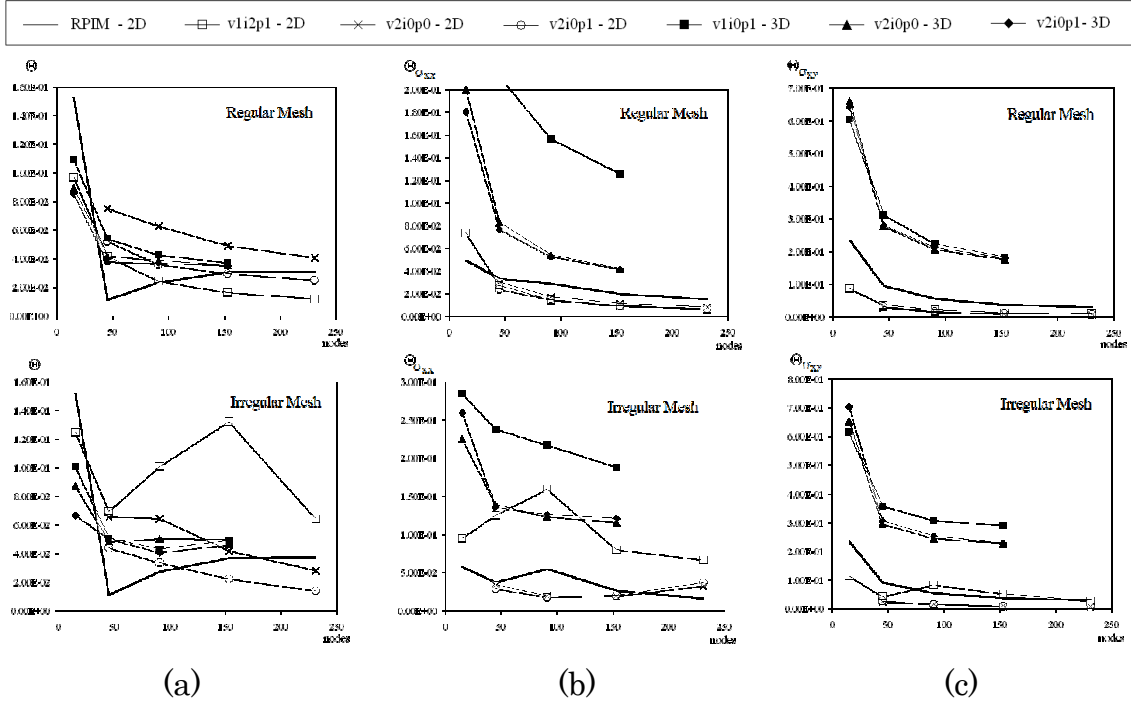


figure 53 – (a)Medium displacement error Θ . (b)Medium stress error $\Theta_{\sigma_{xx}}$. (c)Medium stress error $\Theta_{\sigma_{xy}}$.

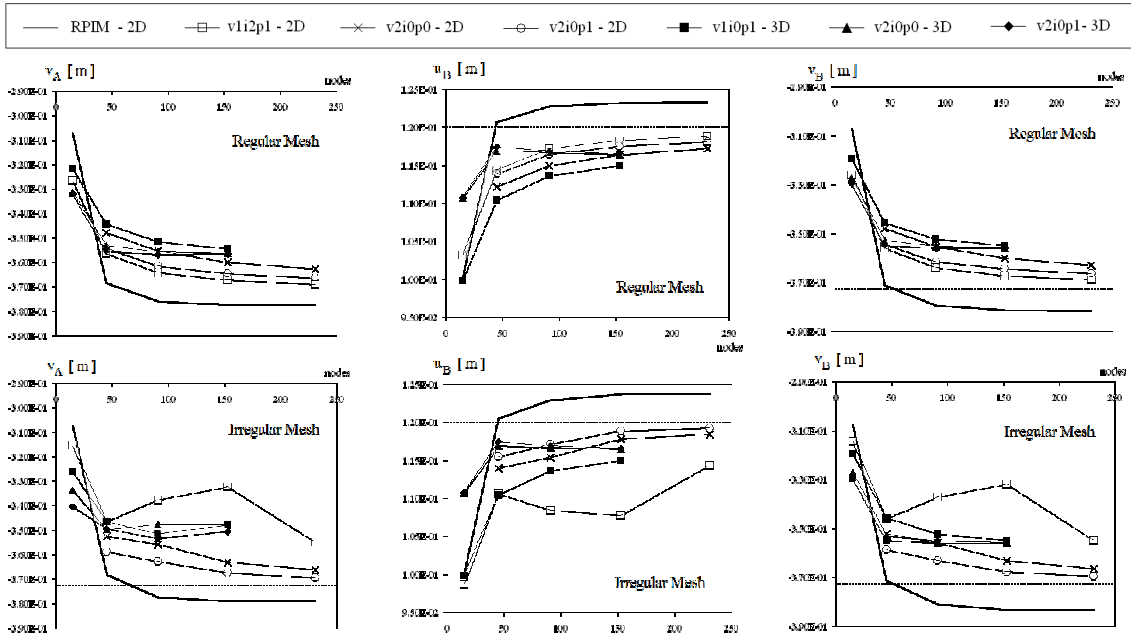


figure 54 – Displacement values obtain with irregular and regular meshes in interest points A and B.

In figure 54, for the 2D analysis, it is visible that the NNRPIM formulations converges with more accuracy to the exact solution when compared with the RPIM formulation. It was observed that the horizontal component of the displacement on interest point A on neutral axis, u_A , for all the NNRPIM

formulations is equal to the machine precision, 10^{-16} , and for the RPIM formulation is about 10^{-6} . In figure 55 the normal stress σ_{xx} on interest point C and the shear stress σ_{xy} on interest point D, for the regular and the irregular mesh, are shown.

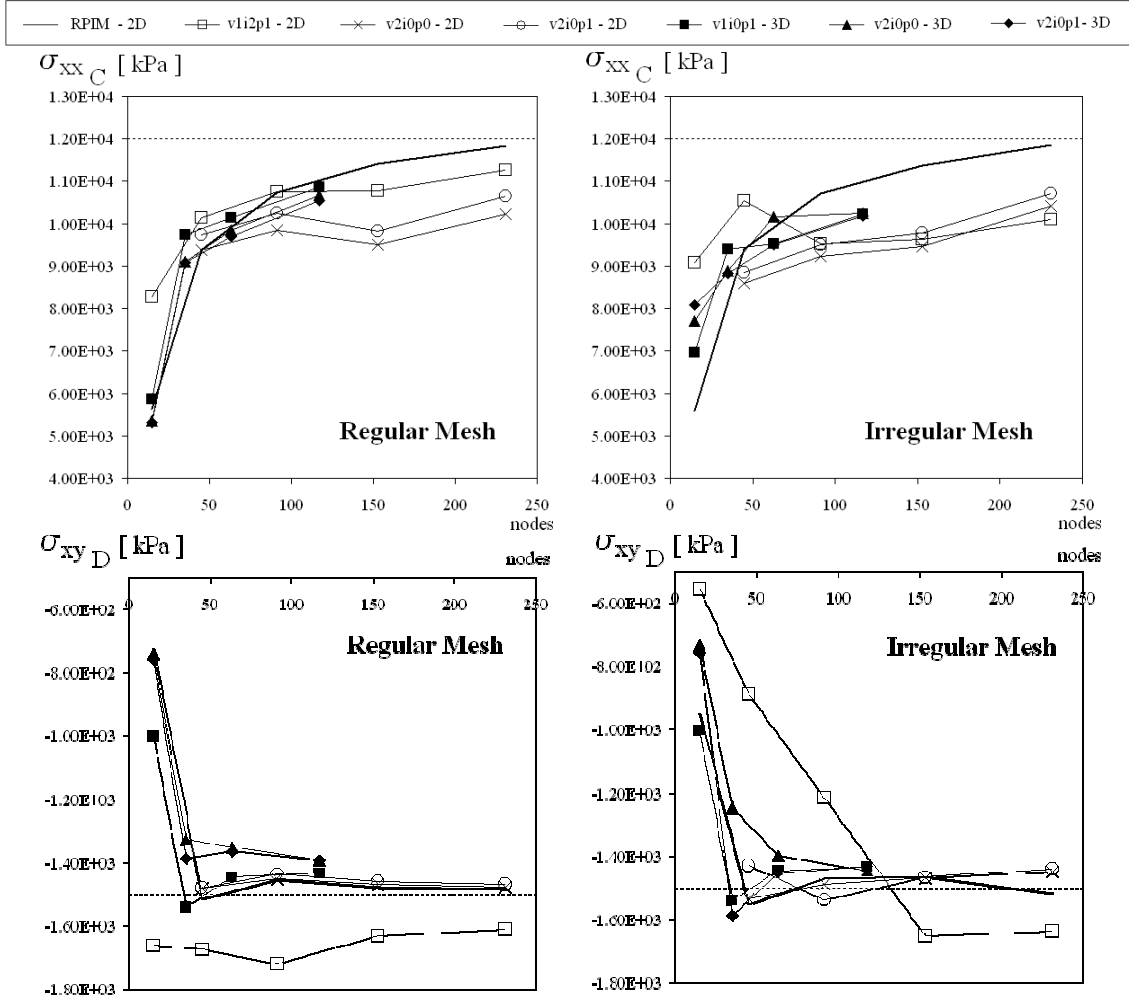


figure 55 – Stress values on the interest points C and D for irregular meshes and for regular meshes.

The results obtained for the shear stress σ_{xy} in the interest point D show a good concordance between the two formulations. However the results for the normal stress σ_{xx} in the interest point C show that all NNRPIM formulations are less accurate when compared with the RPIM formulation. Such results can be explained by the fact that the integration point where the stress is obtained for the RPIM is always much close to point C compared with the closest integration point in the NNRPIM. It must be noted as well, that for the stress results for the solution obtained with the 3D analysis approaches the solution obtained with a 2D analysis.

4.1.3 Infinite Plate with a Circular Hole

An infinite plate with a circular hole is considered, due to the existing symmetry only the upper right quadrant of the plate is analysed, as represented in figure 56. The traction illustrated on the natural boundary conditions is given by the exact solution,

$$\begin{aligned}\sigma_{xx} &= 1 - \frac{a^2}{r^2} \left(\frac{3}{2} \cos(2\theta) + \cos(4\theta) \right) + \frac{3a^4}{2r^4} \cos(4\theta) \\ \sigma_{yy} &= -\frac{a^2}{r^2} \left(\frac{1}{2} \cos(2\theta) - \cos(4\theta) \right) - \frac{3a^4}{2r^4} \cos(4\theta) \\ \sigma_{xy} &= -\frac{a^2}{r^2} \left(\frac{1}{2} \sin(2\theta) + \sin(4\theta) \right) + \frac{3a^4}{2r^4} \sin(4\theta)\end{aligned}\quad (227)$$

In this problem only regular meshes are used. Mesh examples are presented in figure 57.

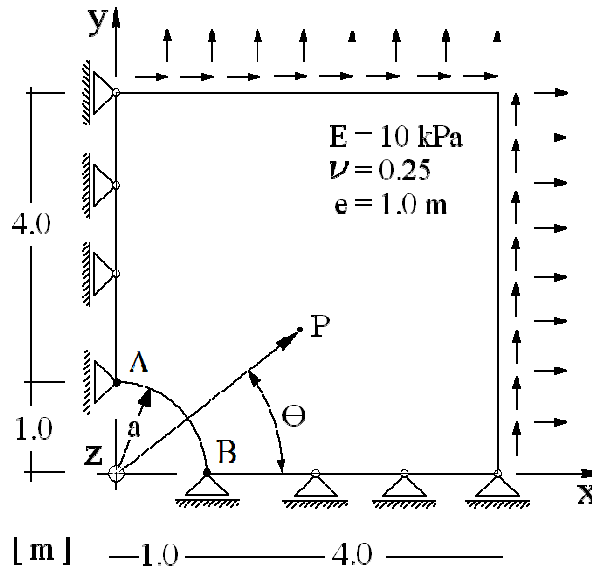


figure 56 – Material and geometrical characteristics and boundary and load conditions of the plate with a circular hole.

In figure 58 it is presented the displacement values on interest points A and B, indicated in figure 56. It is visible that the NNRPIM converges with a higher rate when compared with the FEM and the RPIM. It is also seen that the performance of the NNRPIM formulation is not affected by the used integration scheme, once both integration schemes produce similar results. It is visible that the NNRPIM 3D analysis produces results very close to the FEM 2D analysis.

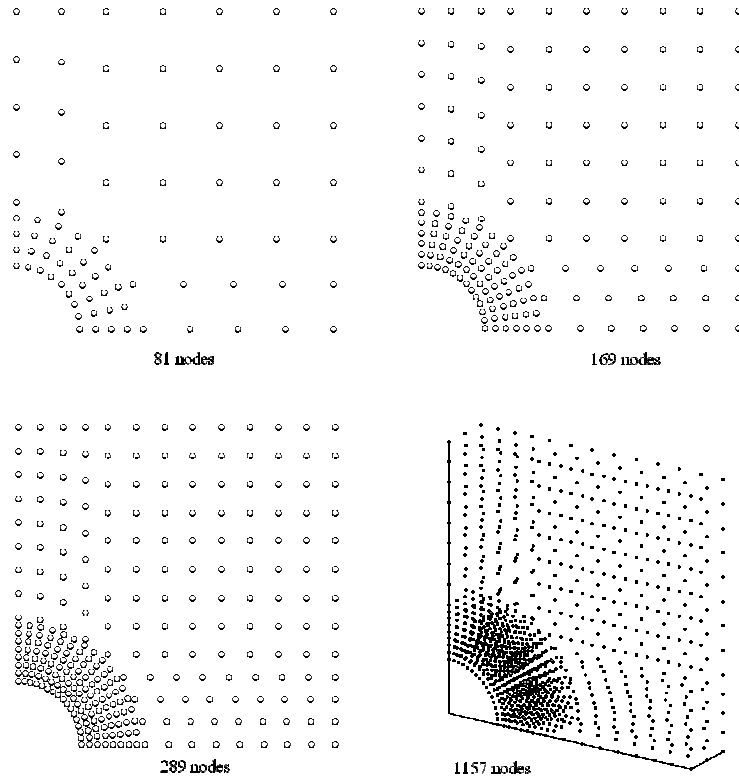


figure 57 – Examples of the regular nodal discretization for the plate with a circular hole.

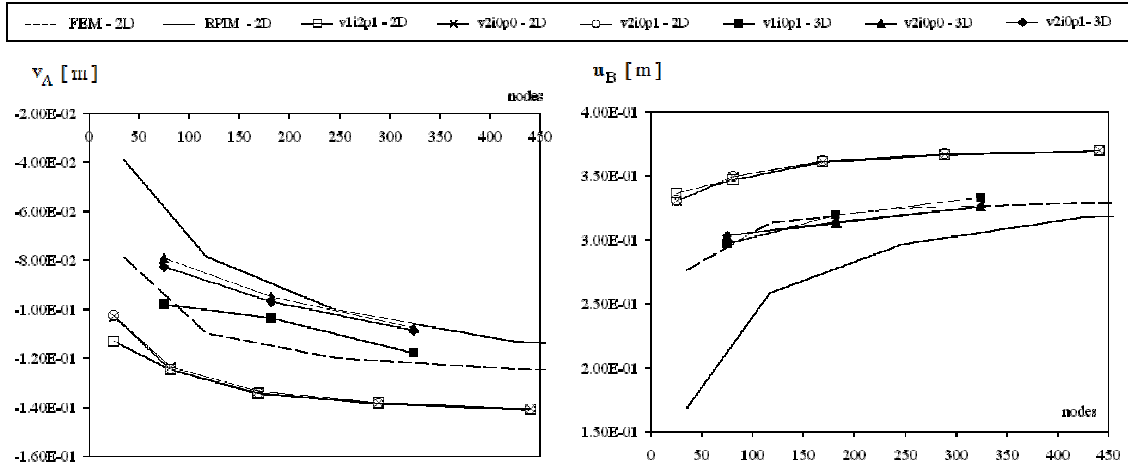


figure 58 – Displacement values in interest points A and B.

In figure 59 are presented the medium stress errors, $\Theta_{\sigma_{xx}}$, $\Theta_{\sigma_{yy}}$ and $\Theta_{\sigma_{xy}}$, and in figure 60 are presented the obtained normal stress σ_{xx} and σ_{yy} , respectively for interest points A and B.

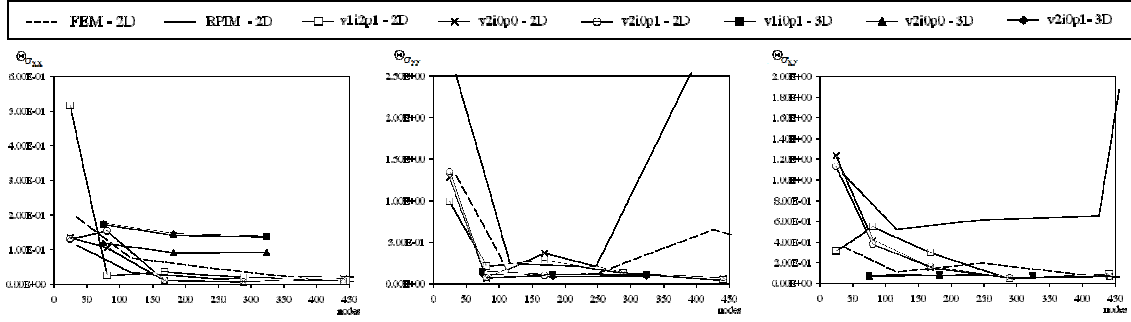


figure 59 – Medium stress errors.

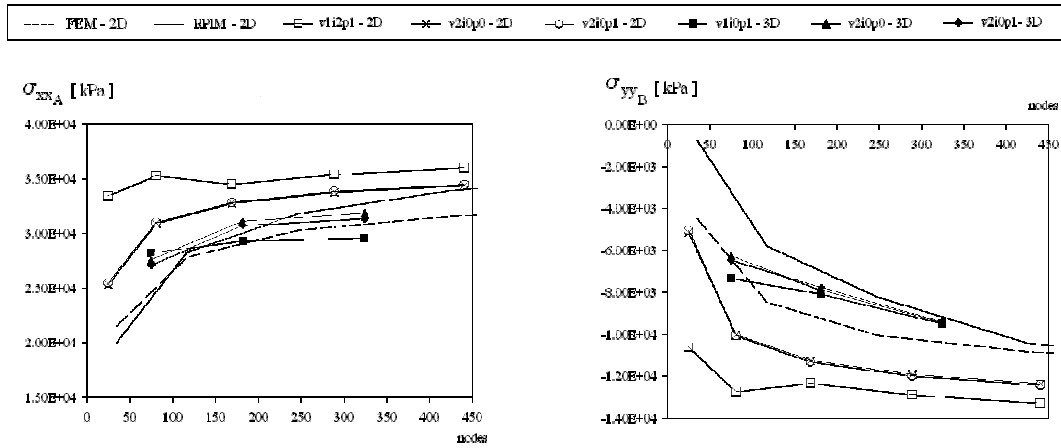


figure 60 – Normal stresses in interest points A and B.

It can be seen that the NNRPIM has, once again, a convergence rate considerably higher than the FEM or the RPIM. It is also observed that the stress field produced by the NNRPIM is very smooth. Once again the results obtained with the NNRPIM 3D analysis approaches the FEM solution. In figure 61 is presented the displacement field and the stress distribution obtained with the NNRPIM and with the exact solution.

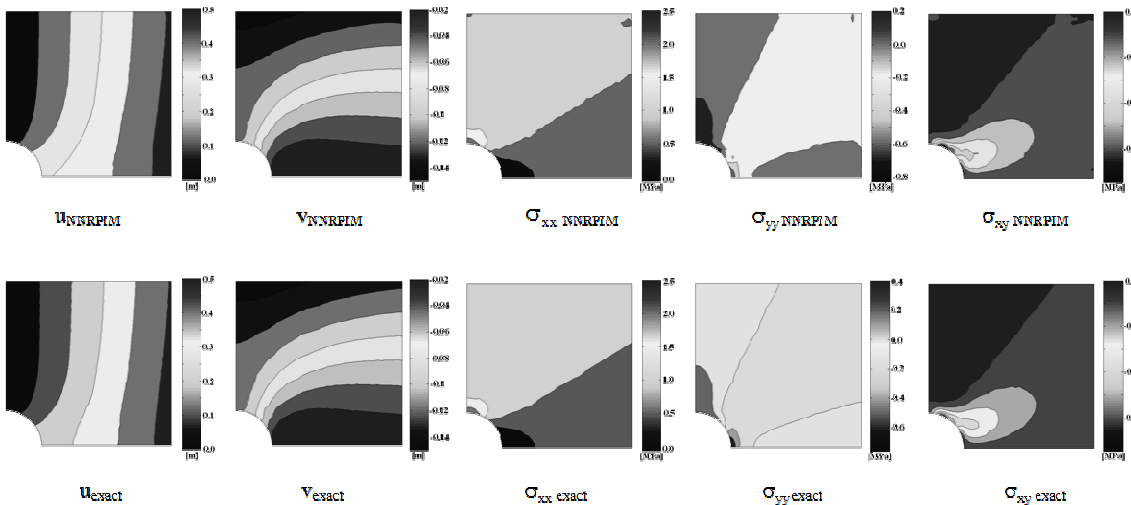


figure 61 – Displacement field and the stress distribution obtained with the NNRPIM and with the exact solution.

The results obtained show an almost perfect similarity between the NNRPIM solution and the exact solution and that the NNRPIM is capable of producing smooth displacement and smooth stress fields.

4.1.4 Plates subjected to Transversal Loads Convergence Study

The goal of the examples presented in this subsection is to study the convergence of the NNRPIM when the 3D shell-like (3DSL) formulation is used. The 3DSL formulation is useful when the analysed problem is a thin structure, such as plates or shells. As it was found in section 3.8 the geometric parameter λ must be $\lambda \in [2.0, 5.0]$. In this example the geometric parameter λ is fixed, $\lambda = 4$, thus the thickness of the plate must vary with the nodal discretization in order to respect $\lambda = 4$, i.e., a more dense mesh correspond to a thinner plate.

Consider a square isotropic plate $L = D = 10\text{ m}$, with all edges simply supported and subjected to a uniformly distributed load $q_0 = 1.0\text{ kN} / \text{m}^2$. The material properties of the problem are $E = 1.0\text{ MPa}$ and $\nu = 0.3$. The plate thickness h changes with the number of nodes discretizing the problem domain. Regular meshes (RM) and irregular meshes (IM) are considered in the analysis, figure 62. Along with the NNRPIM 3DSL formulation this example was also solved with the classical 3D NNRPIM approach (3DCF).

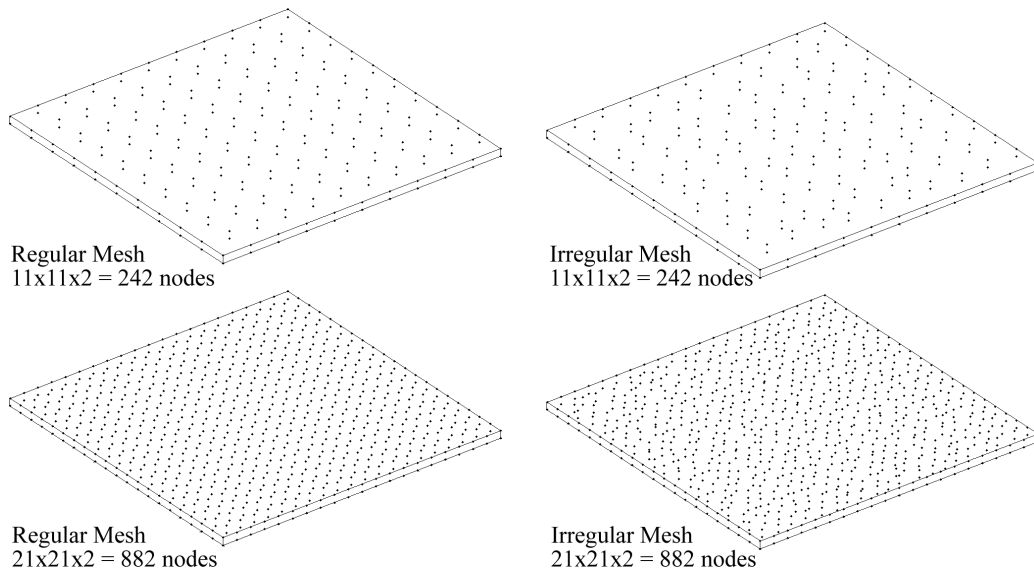


figure 62 – Square plate regular and irregular meshes.

In figure 63(a) the results regarding the transverse centre displacement for increasing nodal discretization and decreasing thickness are presented. The same plate problem was again analysed except for the boundary conditions,

the plate is now considered to be clamped in all edges. The results of the convergence study are presented in figure 63(b).

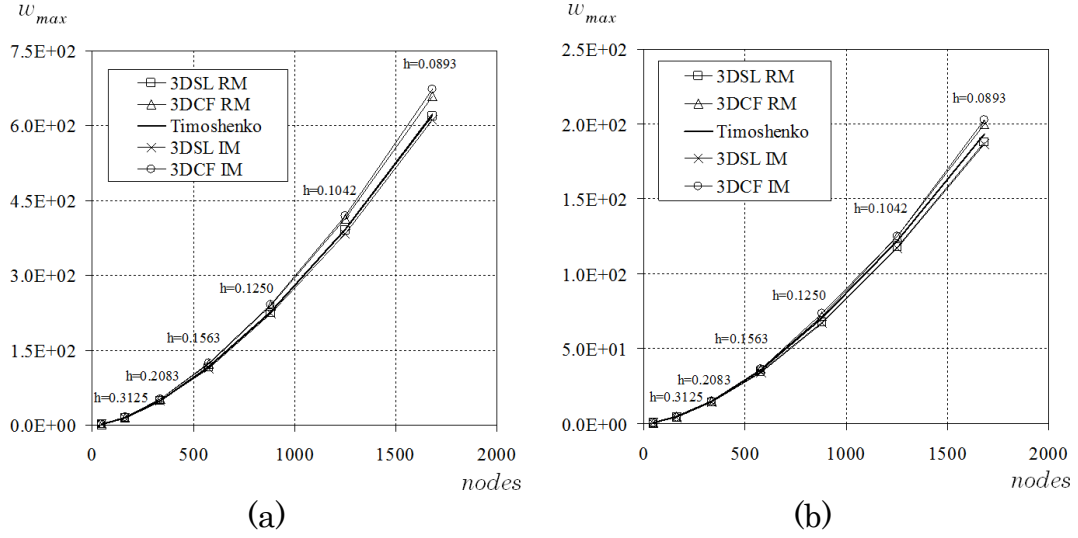


figure 63 – Centre transverse displacement versus number of used nodes for the (a) simply supported square plate and (b) clamped square plate.

It is visible in both plates, simply supported and clamped, that the 3DSL formulation approaches more the analytical solution proposed by Timoshenko [68].

Consider now a circular plate with radius $R = 5.0\text{ m}$ and variable thickness, h . The material properties are the same as in previous plate example. Due the symmetry of the plate only one quarter of the plate is analysed, figure 64. All the NNRPIM 3D Shell-Like formulation (3DSL) results are compared with the results obtained with the NNRPIM using a 3D classical formulation (3DCF) and analytical solution proposed by Timoshenko [68].

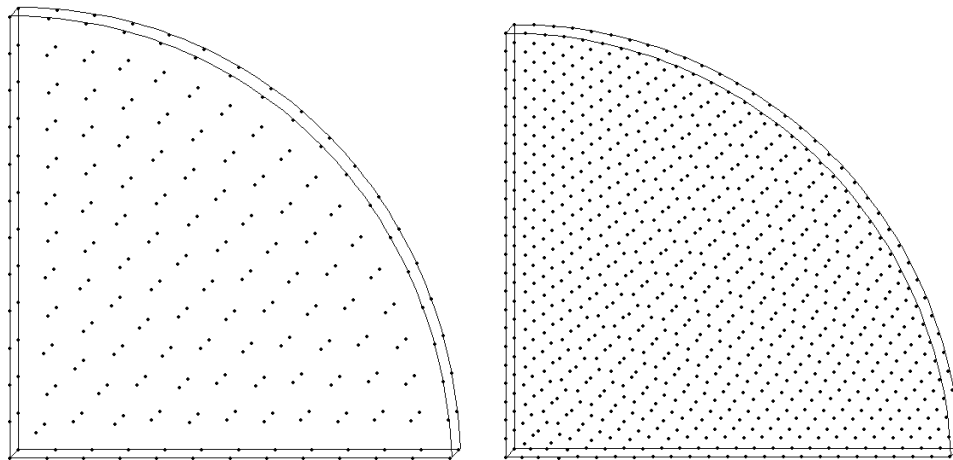


figure 64 – Circular plate meshes.

It is presented in figure 65(a) the convergence study for the simply supported circular plate and in figure 65(b) the convergence study for the clamped circular plate.

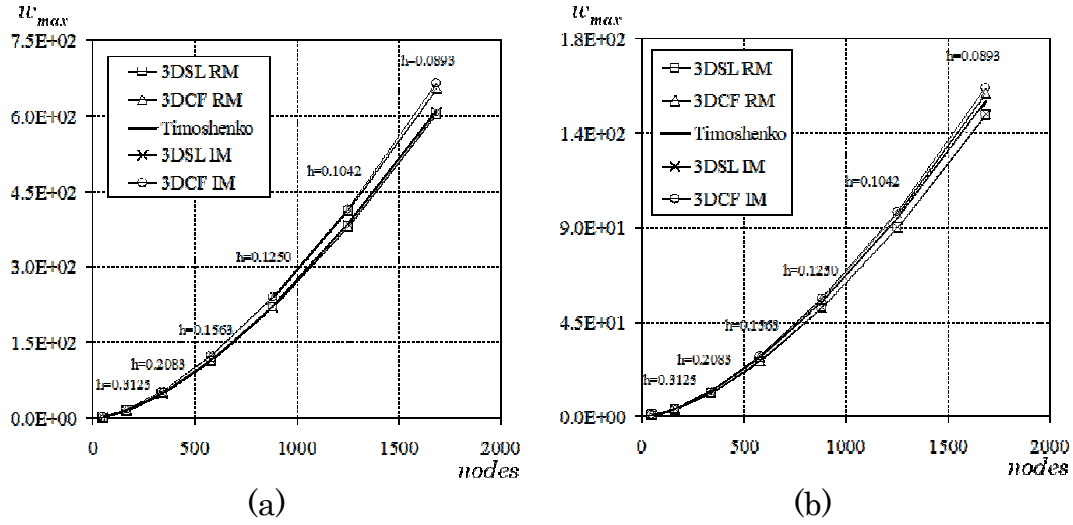


figure 65 – Centre transverse displacement versus number of used nodes for the (a) simply supported circular plate and (b) clamped circular plate.

It is visible that for the simply supported circular plate the 3DSL formulation achieves closer results to the analytical solution once compared with the 3DCF, despite the used mesh type. For the clamped circular plate the results obtained with both formulations are very close to the analytical solution.

Consider a skewed plate, simply supported in all edges, with the geometric properties described in figure 66(a), where $L = 20.2m$ and $D = 10m$, and the same material properties as in previous example. The plate is subjected to an uniformly distributed load $q_0 = 1.0 \text{ kN} / m^2$. In figure 66(a) it is presented an example of the regular mesh used in the analysis. The NNRPIM 3D Shell-Like formulation (3DSL) results are compared with the analytical solution proposed by Timoshenko [68]. In figure 66(b) it is presented the convergence study of the skewed plate.

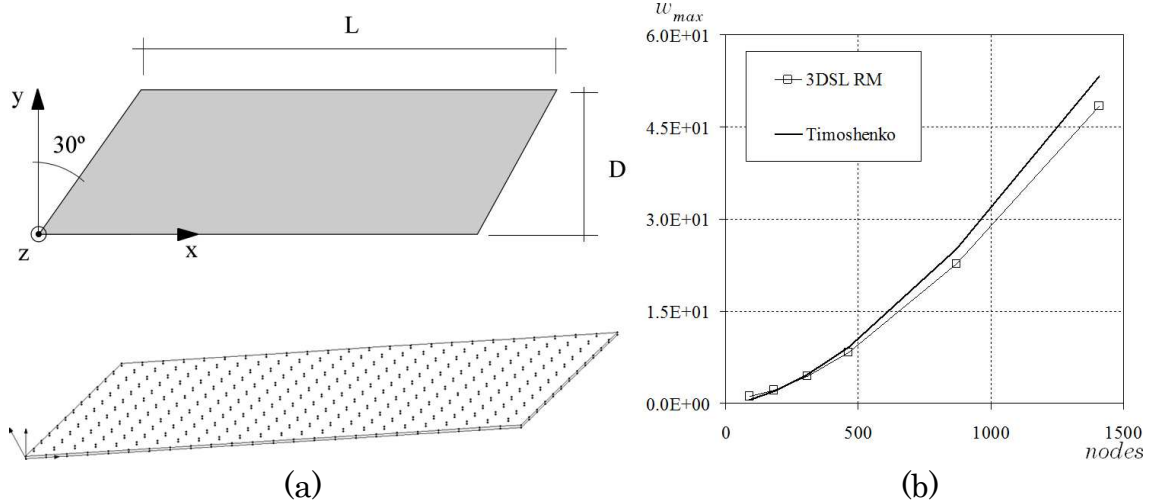


figure 66 – (a) Geometric properties of the skewed plate and typical regular mesh. (b) Centre transverse displacement versus number of used nodes.

It is visible the good agreement between the NNRPIM solution and the analytical solution.

4.1.5 Scordelis-Lo Cylindrical Roof

This example was originally proposed by Scordelis and Lo [69] and has become a shell benchmark example. A cylindrical roof, as the one presented in figure 67, is submitted to its self-weight and the two extremities, $y = 0$ and $y = D$, are supported by rigid diaphragms, the other two edges are free. Due the natural symmetry only one quarter of the roof is analysed. The obtained displacement along the zz axis on point A is compared with the analytical solution, $w_A = 0.3024\text{ m}$, from [70]. The material and geometrical properties of the shell are presented in figure 67.

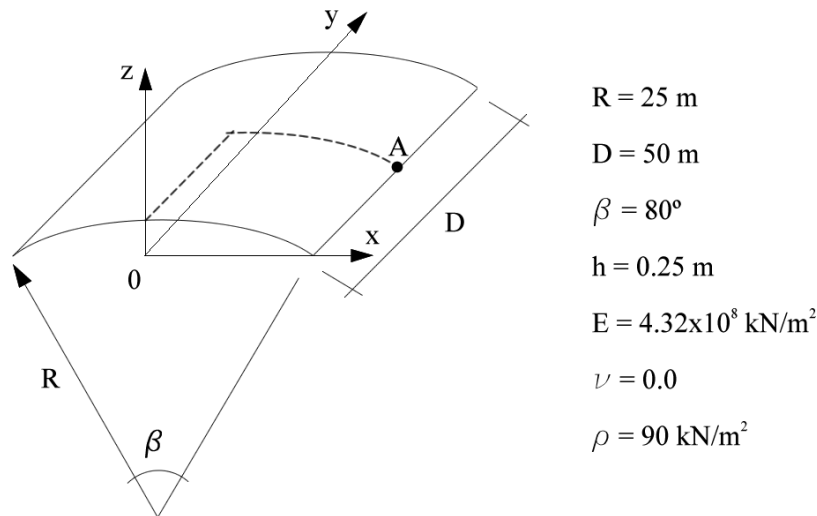


figure 67 – Geometric and material properties of the Scordelis-Lo cylindrical roof.

Three types of meshes discretizing the problem domain were used in the analysis, a regular mesh (RM), an irregular mesh (IM) and a parabolic mesh (PM), where the node density follows a parabolic distribution being higher nearby the boundaries and lower in the centre of the discretization, figure 68.

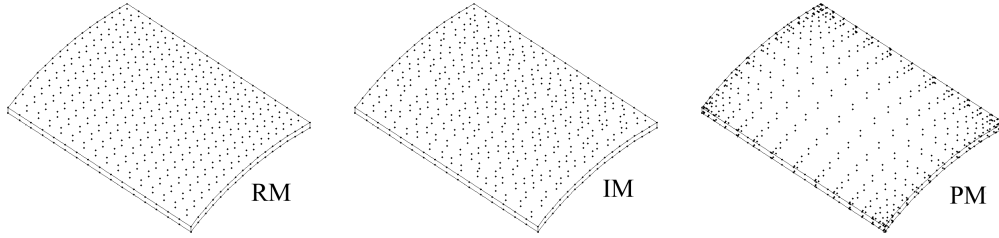


figure 68 – Used meshes in the analysis of the Scordelis-Lo cylindrical roof problem.

The obtained results are presented in figure 69 for the distinct meshes. The final converged values of each mesh are: $w_{RM} = 0.2842 m$, $w_{IM} = 0.2761 m$ and $w_{PM} = 0.2989 m$.

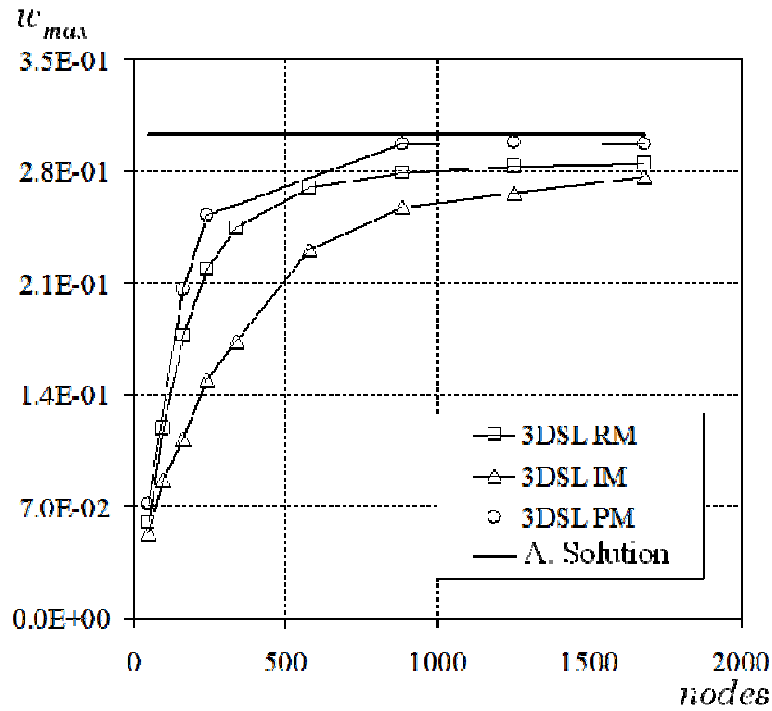


figure 69 – Transverse displacement on point A versus number of used nodes.

It is visible that the NNRPIM solution converges to a final value very close to the analytical solution. If a parabolic mesh (PM) is used the convergence rate increases significantly and the obtained error compared with the analytical solution is just 1%.

4.1.6 Cylindrical Shell

In figure 70 it is presented one of the most used benchmark shell example. One cylinder with the two opposite edges supported by rigid diaphragms is symmetrically pinched in the middle section. The geometrical and material properties of the problem are shown in figure 70. Due to symmetry only 1/8 of the problem is analysed.

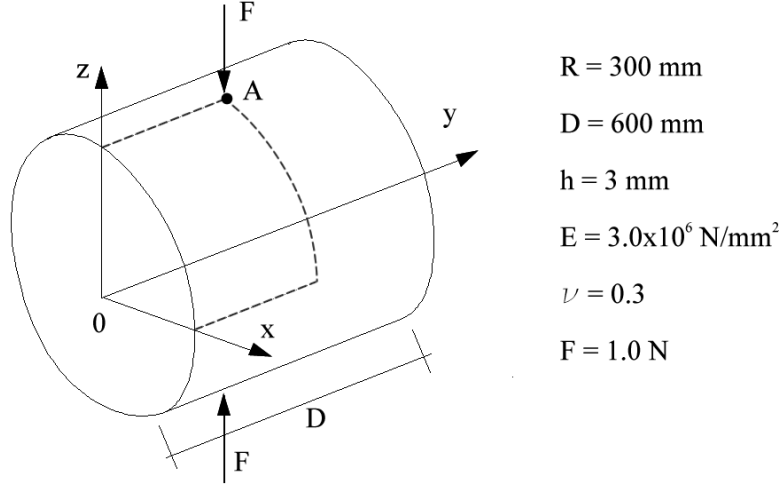


figure 70 – Geometric and material properties of the cylindrical shell.

The analytical solution in the point A is $w_A = 1.82488 \times 10^{-5} \text{ mm}$, from [71]. Again three types of meshes discretizing the problem domain are used, a regular mesh (RM), an irregular mesh (IM) and a parabolic mesh (PM), figure 71.

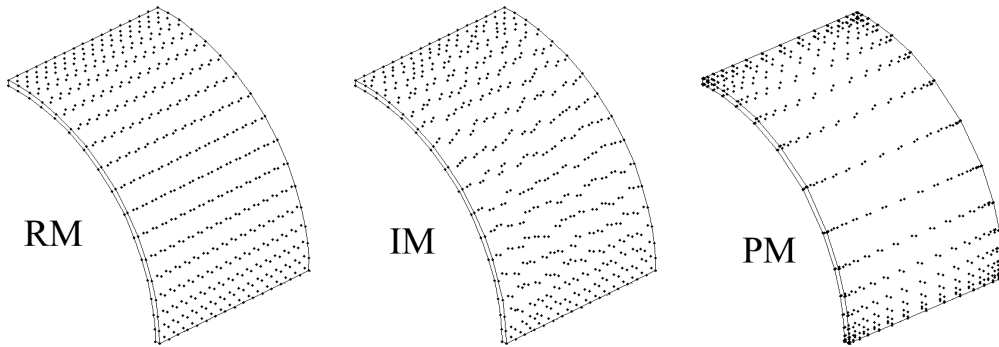


figure 71 – Used meshes in the analysis of cylindrical shell problem.

The obtained results are presented in figure 72 for the distinct meshes. The final converged values of each mesh are: $w_{RM} = 1.70683 \times 10^{-5} \text{ mm}$, $w_{IM} = 1.51948 \times 10^{-5} \text{ mm}$ and $w_{PM} = 1.74938 \times 10^{-5} \text{ mm}$.

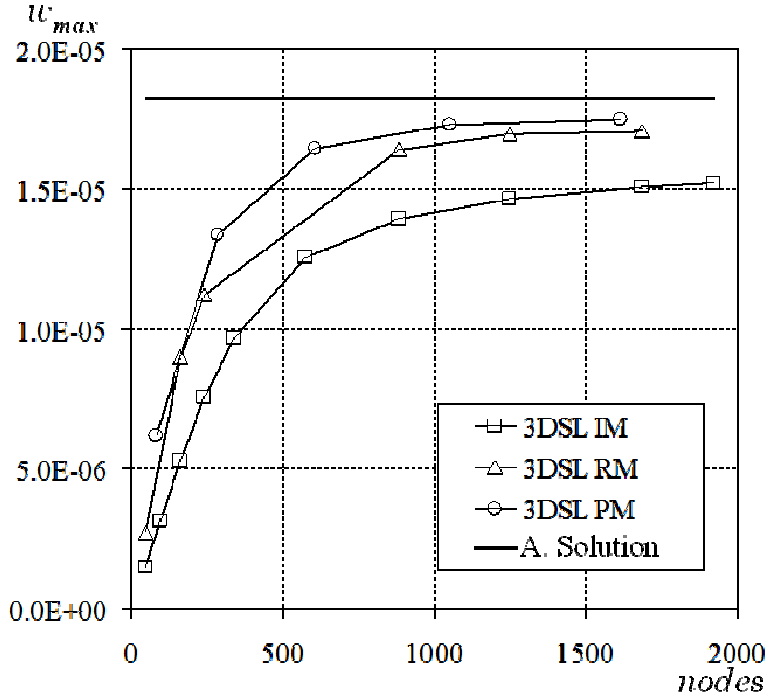


figure 72 – Transverse displacement on point A versus number of used nodes.

In figure 72 it is visible if a parabolic mesh (PM) is used the solution converges to a final value very close to the analytical solution and with a fast convergence rate.

4.1.7 Orthotropic Plate and Shell Examples

In this subsection, using the NNRPIM 3D Shell-Like formulation, laminate plates and shells examples are studied. For all examples the repetition number of the integration mesh surface along the thickness is 2 for each laminate layer, in order to achieve more accuracy and precision. It is also respected the proper geometric parameter λ range values, $\lambda \in [2.0, 5.0]$. The obtained results are compared with the two-dimensional plate transverse deformation theories analytical solution (FSDT and TSDT) [72] and the three-dimensional analytical elasticity solution of Pagano (ELP-3D) [73, 74]. All the examples are compared with the analytical solution when available. Two generic loads are applied, a uniformly distributed load (UDL) $q = q_0$ and a sinusoidal distributed load (SSL) defined as,

$$q = q_0 \sin\left(\frac{\pi x}{L}\right) \sin\left(\frac{\pi y}{D}\right) \quad (228)$$

The magnitude of the load factor q_0 is 1.0 kN/m^2 in all examples and the load is numerically applied in the midplane of the laminate. The geometric and material characteristics of the generic laminate are defined in figure 73.

In the examples two typical graphite-epoxy materials are used, the lamina properties are also presented in figure 73. The material property in material direction 2 is defined as $E_2 = 2.1 \times 10^6 \text{ kN/m}^2$.

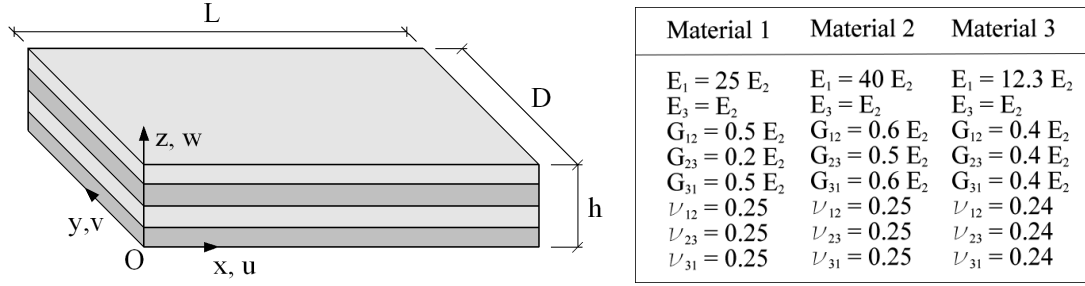


figure 73 – Laminate geometric properties and lamina material properties.

In the presented studies the transverse displacement is obtained in the centre of the laminate, for $\{x \ y \ z\} = \{L/2 \ D/2 \ h/2\}$, and is normalized with the expression $w_{norm} = 100 \times w \times (E_2 h^3) / (L^4 q_0)$, the normal stresses σ_{xx} and σ_{yy} are also obtained in the centre of the laminate, in the upper layer, for $\{x \ y \ z\} = \{L/2 \ D/2 \ h\}$ and are normalized with the expression $\sigma_{xx}^{norm} = \sigma_{xx} \times h^2 / (L^2 q_0)$. The shear stresses σ_{xz} and σ_{yz} are obtained in the edge layer with higher stress, as it is indicated in [72], and are both normalized with the expression $\sigma_{ij}^{norm} = \sigma_{ij} \times h / (L q_0)$. The lamina thickness is considered constant, all the laminates are regular hexahedrons, and the total thickness of the laminate is divided equally by each layer.

4.1.7.1 Square Symmetric Laminates

In order to show the accuracy and the high convergence rate of the NNRPIM a square symmetric cross-ply laminate simply supported in all edges is considered. The geometrical properties are $L = D = 10m$ and the total thickness is $h = 0.1m$. The material properties of the layers are the same as Material 1. Two distinct laminates are studied, one has four layers, oriented in 0/90/90/0 and the other has three layers, oriented in 0/90/0. Two types of load were considered, a uniform distributed load (UDL) and a sinusoidal load (SSL). Due to the natural symmetry of the plate only one quarter of the laminate is studied. The domain was discretized in several regular meshes and irregular meshes, figure 62. The NNRPIM 3DSL results and the NNRPIM 3DCF results are compared with the analytical solution of the 2D bending deformation theories, the classical plate theory (CLPT), the first order shear deformation theory (FSDT) and the third order shear deformation theory (TSDT), which can be found in [72]. The results were also compared with the 3D solution of Pagano [74]. Firstly it is presented a

convergence study. In figure 74 it is shown the normalized transverse centre displacement for the 0/90/90/0 laminate. The results regarding the 0/90/0 laminate are presented in figure 75.

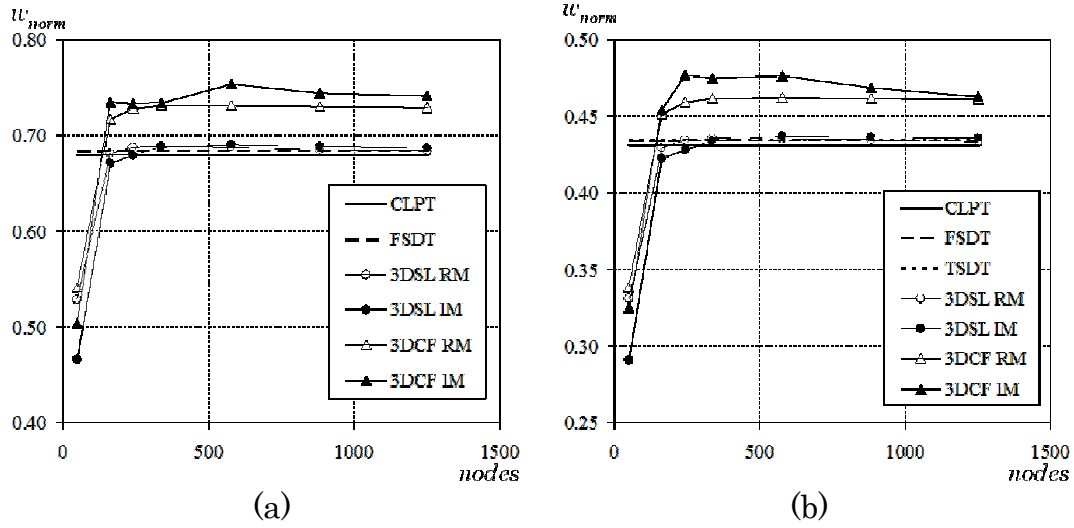


figure 74 – Normalized centre transverse displacement versus number of used nodes discretizing the 0/90/90/0 laminate. (a) ULD. (b) SSL.

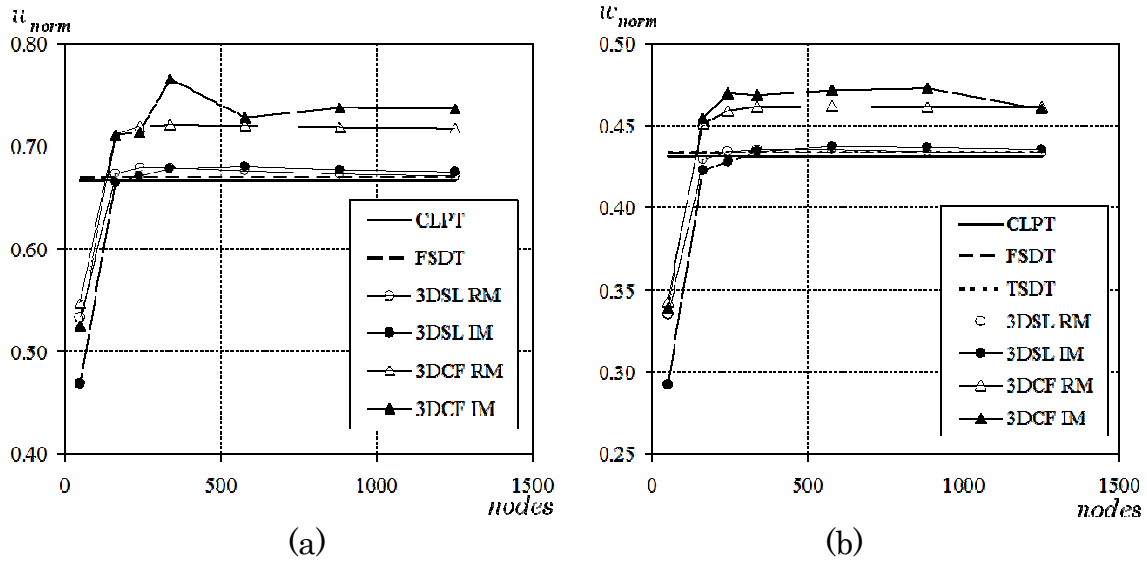


figure 75 – Normalized centre transverse displacement versus number of used nodes discretizing the 0/90/0 laminate. (a) ULD. (b) SSL.

Both figures show that the 3DSL formulation has a better behaviour than the 3DCF, once it converges faster to the final solution and the converged solution is closer to the analytical solutions. This can be confirmed in table 8, where the normalized displacements obtained with the NNRPIM formulations, for $25 \times 25 \times 2$ nodes, are compared with the analytical solutions.

Laminate	Load	Analytical Solutions				NNRPIM			
		ELS	FSDT	TSDT	CLPT	3DSL RM	3DSL IM	3DCF RM	3DCF IM
0/90/90/0	UDL	-	0.6833	-	0.6796	0.6830	0.6863	0.7283	0.7414
	SSL	0.4380	0.4337	0.4340	0.4312	0.4331	0.4353	0.4609	0.4628
0/90/0	UDL	-	0.6697	-	0.6660	0.6708	0.6745	0.7174	0.7367
	SSL	-	0.4337	0.4342	0.4313	0.4334	0.4355	0.4612	0.4601

table 8 – Normalized displacement in the centre of the laminates for distinct formulations and nodal meshes.

The obtained stress field is smooth and accurate. Along thickness the stress field approaches very well the 2D deformation theories. It is shown in figure 76 the stress along thickness of the 0/90/90/0 laminate for the 3DSL formulation, subjected to a sinusoidal load. The normal stresses σ_{xx} and σ_{yy} are obtained in the centre of the laminate, the shear stress σ_{yz} is obtained in $\{x \ y\} = \{L/2 \ 0\}$ and the shear stress σ_{zx} is obtained in $\{x \ y\} = \{0 \ D/2\}$.

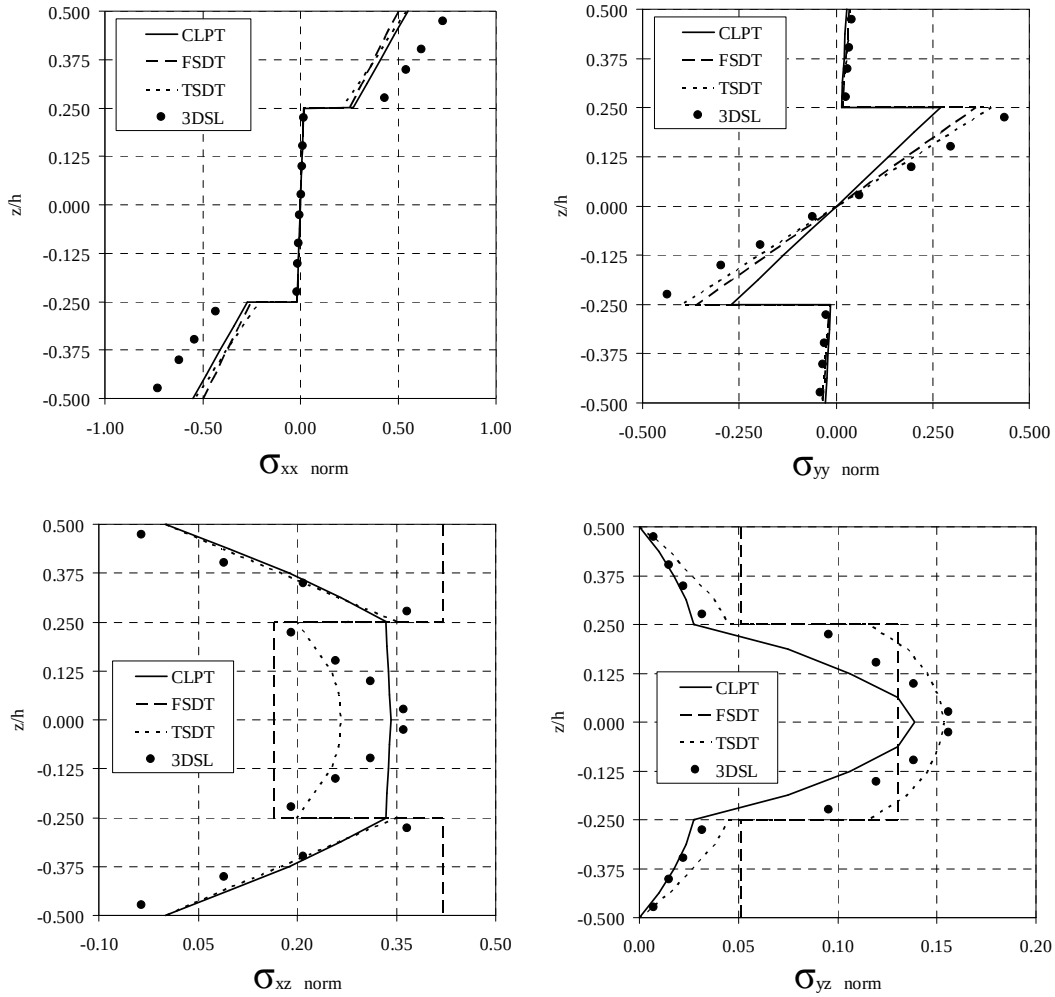


figure 76 – Non-dimensional stresses versus thickness for a simply supported 0/90/90/0 laminate under a sinusoidal load.

In order to verify the behaviour of the 3DSL formulation to the variation of the essential boundary conditions several boundary conditions were tested. The same three layered (0/90/0) symmetric square laminate is studied. The geometrical properties are $L = D = 10m$ and the total thickness is $h = 0.1m$. The material properties of the layers are the same as Material 3 and the applied load is the uniformly distributed load (UDL). Several boundary conditions are considered, for $x = 0$ and $x = L$ the edges are always simply supported and for $y = 0$ and $y = D$ the boundary condition changes. The six studied cases are: S-S-S-S, S-S-S-C, S-C-S-C, S-F-S-F, S-F-S-S and S-F-S-C. Where “S” stands for simply supported edge, “C” for clamped edge and “F” for free edge. A regular mesh and an irregular mesh of $25 \times 25 \times 2$ nodes were used in the analysis. The displacement is obtained in the centre of the laminate and it is normalized. The results are compared with the analytical solution (AS) presented in [72]. The results are presented in table 9.

	SSSS	SSSC	SCSC	SFSF	SFSS	SFSC
3DSL RM	1.227	0.553	0.284	10.115	5.898	2.384
3DSL IM	1.215	0.553	0.287	9.907	5.505	2.302
3DCF RM	1.307	0.593	0.306	9.738	5.833	2.450
3DCF IM	1.343	0.606	0.307	9.822	5.870	2.450
AS	1.206	0.544	0.280	10.920	5.992	2.376

table 9 – Normalized displacement in the centre of the laminate for distinct formulations and nodal meshes, varying the boundary conditions.

It is visible that the obtained solution is always very close to the analytical solution, and that the 3DSL formulation presents better results once compared with the classical NNRPIM formulation.

4.1.7.2 Square Anti-Symmetric Laminates

In this section several square anti-symmetric laminates are considered. The study starts with the analysis of two square cross-ply anti-symmetric laminates, 0/90 and (0/90)₄, simply supported in all edges. Two load types are considered the UDL and the SSL. The geometric properties are $L = D = 10m$ and the total thickness is $h = 0.1m$. The material properties of the layers are the same as Material 1. The results are compared with the FSDT analytical solution and with the CLPT analytical solution [72]. Two meshes are used in the analysis, a regular mesh and an irregular mesh,

both with $25 \times 25 \times 2$ nodes. The obtained normalized displacements in the centre of the laminate are presented in table 10.

		FSDT	CLPT	3DSL RM	3DSL IM	3DCF RM	3DCF IM
0/90	UDL	1.6980	1.6955	1.6283	1.6239	2.3678	2.3821
	SSL	1.0653	1.0636	1.0217	1.0141	1.4861	1.5014
(0/90) ₄	UDL	0.7175	0.7150	0.7035	0.7104	0.9649	0.9815
	SSL	0.4496	0.4479	0.4407	0.4437	0.6045	0.6105

table 10 – Normalized displacement in the centre of the laminate for distinct formulations and nodal meshes varying the boundary conditions.

As table 10 shows the 3DSL formulation results are very close to the analytical solutions, regardless the used mesh.

Several square angle-ply anti-symmetric laminates with two or eight layers are now considered. The studied laminates are $\theta/-\theta$ and $(\theta/-\theta)_4$, with $\theta = \{5^\circ \ 30^\circ \ 45^\circ\}$. The load type considered is the SSL. The geometric properties are $L = D = 10m$ and the total thickness varies $h = \{0.4 \ 0.2 \ 0.1\}m$. The material properties are the same as material 2 and the laminates are simply supported in all edges. Two meshes are used in the analysis a regular mesh and an irregular mesh, both with $25 \times 25 \times 2$ nodes. The results are compared with the FSDT analytical solution [72]. The obtained normalized displacements in the centre of the laminate are shown in table 11.

		FSDT		SL3D RM	
L/h		(-θ/θ)	(-θ/θ) ₄	(-θ/θ)	(-θ/θ) ₄
θ=5°	25	3.427	3.105	3.9246	3.1281
	50	3.215	2.806	3.5928	2.8233
	100	3.162	2.754	3.5078	2.7489
θ=30°	25	5.119	1.9	5.7077	2.1977
	50	4.979	1.761	5.2062	1.8999
	100	4.944	1.726	5.0201	1.7992
θ=45°	25	4.844	1.693	5.3438	2.0243
	50	4.711	1.56	4.8234	1.7179
	100	4.678	1.527	4.6251	1.6081

table 11 – Normalized displacement in the centre of the angle-ply laminates for distinct thickness.

As it is perceptible in table 11 once more the results obtained with the 3DSL formulation are accurate and very close to the analytical solution.

4.1.7.3 Long Cylindrical Laminated Shell

In this example long cylindrical laminated shells are studied. Consider a laminate with the geometric properties showed in figure 77(a), where the dimensions are: $R = 10\text{ m}$, $D = 30\text{ m}$, $h = \{0.2\ 0.1\}\text{ m}$ and $\beta = 60^\circ$. Point A is in the centre of the laminate shell. The material properties of the layers that compose the laminate are the same as material 1. The laminate is simply supported in edges bc2 and bc4 and free in the others. Three laminates are studied, a 0° , $0/90$ and $0/90/0$. The applied load is a sinusoidal load $q = q_0 \sin(\pi x/L)$, where $q_0 = 1.0\text{ kN/m}^2$. The displacement obtained in point A is normalized with the expression: $w_{norm} = w \times 10 \times E_2 / (q_0 \times h \times S^4)$, being $S = R/h$. Due to the symmetry of the plate only one quarter of the problem is analysed, figure 77(a). Since the laminate is very long another modulation was assumed. It was considered the analysis of only a unit strip of the laminate, as figure 77(b) indicates.

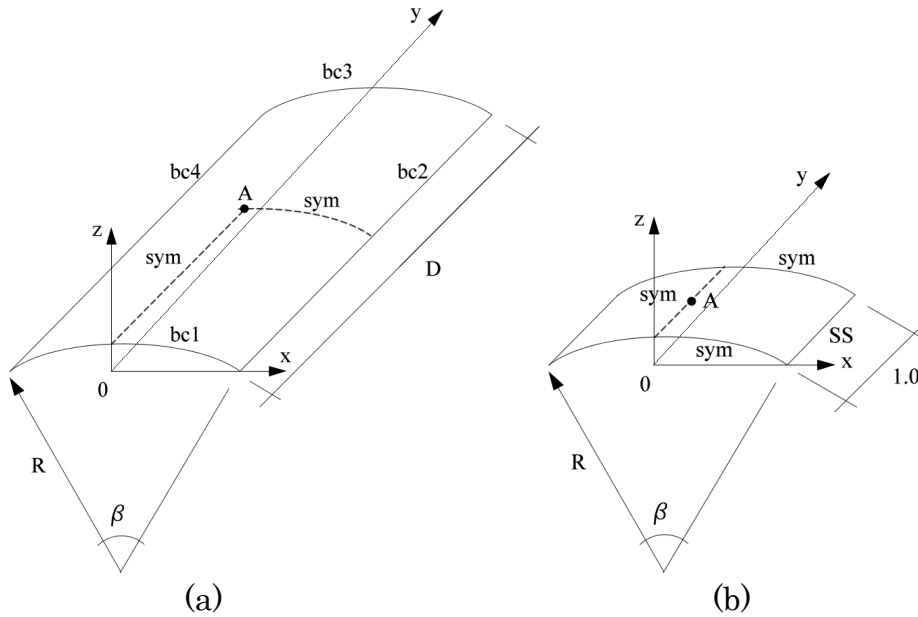


figure 77 – Geometric properties of the cylindrical laminated shell (a) Complete shell. (b) Unit strip.

For the strip unit approach a regular (RM1) and an irregular meshes (IM1) with $61 \times 13 \times 2$ nodes were considered, figure 78(a) and (b). For the one quarter approach only a regular mesh was considered (RM2) with $29 \times 29 \times 2$ nodes, figure 78(c).

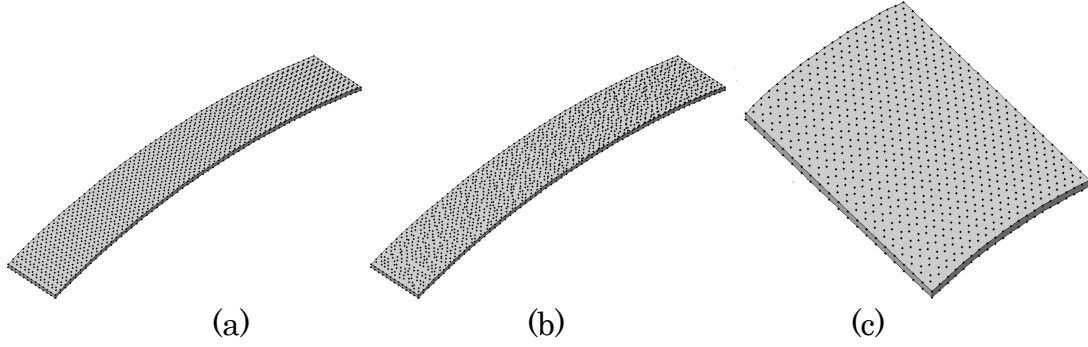


figure 78 – Meshes used in the analysis. (a) Strip mesh RM1. (b) Strip mesh IM1. (c) Quarter mesh RM2.

The results are compared with the finite element solution (HSDT FEM and FSDT FEM) [75] and an elasticity solution ELS [76]. In table 12 it is presented the normalized centre displacement obtained with the NNRPIM 3DSL formulation.

θ	R/h	HSDT-FEM	FSDT-FEM	ELS	SL3D RM1	SL3D IM1	SL3D RM2
0	50	0.0748	0.0749	0.077	0.0709	0.0663	0.0761
	100	0.0736	0.0736	0.0755	0.0694	0.0649	0.0748
0/90	50	0.3946	0.3951	0.409	0.3342	0.3155	0.3376
	100	0.3549	0.3999	0.403	0.3384	0.3207	0.3623
0/90/0	50	0.0782	0.0779	0.0808	0.0731	0.0683	0.0755
	100	0.0766	0.0767	0.0787	0.0723	0.0679	0.0794

table 12 – Normalized displacement in the centre of the cross-ply shell laminates for distinct formulations and nodal meshes.

In table 12 it is visible the good agreement between the NNRPIM 3DSL formulation and the reference solutions.

4.1.7.4 Simply Supported Cylindrical Laminated Shell

Consider two cylindrical laminated shells, 0/90 and 0/90/90/0, simply supported in all edges, as the one presented in figure 77(a). The dimensions are: $R = 5m$, $D = 30m$, $h = 0.05m$ and $\beta = 60^\circ$. The material properties of the layers that compose the laminate are the same as material 1. The applied load is a sinusoidal load $q = q_0 \sin(\pi x/L) \sin(\pi y/D)$, where $q_0 = 1.0 kN/m^2$. The displacement obtained in point A, in the centre of the laminate shell, is normalized as in previous example: $w_{norm} = w \times 10 \times E_2 / (q_0 \times h \times S^4)$, being $S = R/h$. Only one quarter of the problem is analysed due to the natural symmetry of the structure, figure 77(a). Two regular meshes are considered,

M1 and M2. M1 is a regular mesh where the nodes are distributed along the shell domain with the rule: $div \times (5div) \times 2$ and M2 is another regular mesh where the nodal distribution follows $div \times (10div) \times 2$. Both mesh examples can be seen in figure 79.

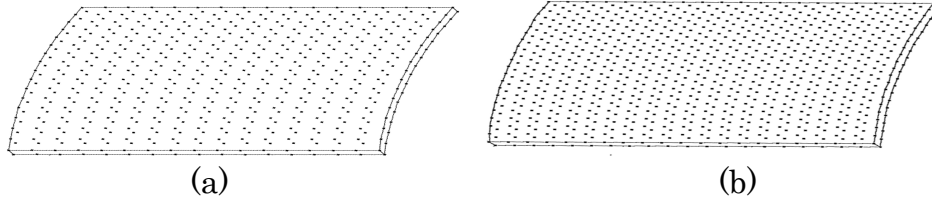


figure 79 – Meshes used in the analysis. (a) strip mesh M1. (b) Strip mesh M2.

The results are compared with the finite element solution (HSDT FEM and FSDT FEM) [75] and a elasticity solution ELS [77]. The convergence results are presented in figure 80.

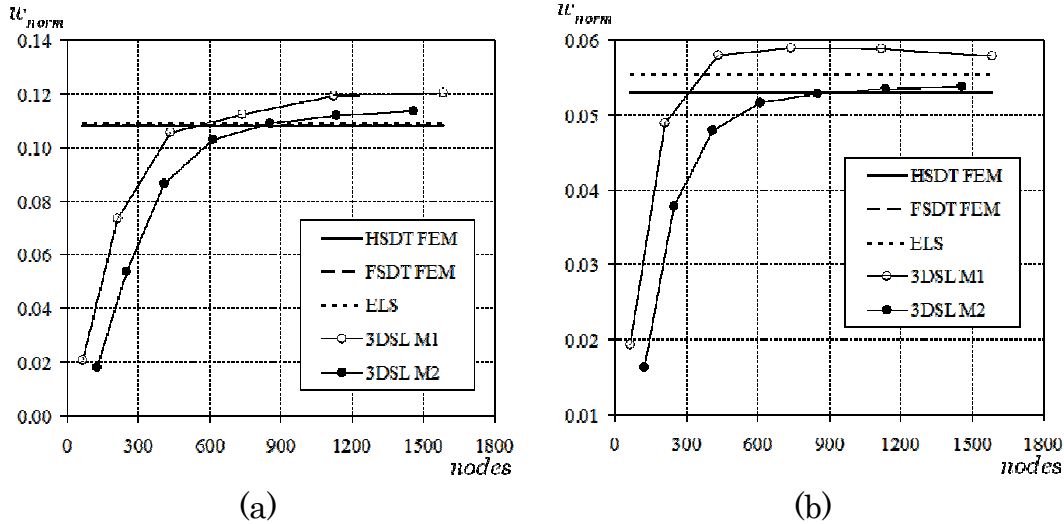


figure 80 – Normalized centre transverse displacement versus number of used nodes discretizing the (a) 0/90 laminate and (b) 0/90/90/0 laminate.

As figure 80 shows, the NNRPIM 3DSL formulation converges fast to the final solution and the converged solution is close to the reference solutions.

4.2 Dynamic Analysis

The assemblage of spatial thin shells structures permits the construction of several engineering structures, such as roof structures, boat hulls and aeroplane fuselages, among many others. Nowadays, shell structures are design to be light, being the shells themselves the main supporting structure, reducing the number of structure stiffeners. On the other hand this structural material optimization has a design disadvantage, it leads to

lower fundamental frequencies, increasing the risk of collapse by resonance. Thus the dynamic analysis became an important part in shell structures design. In this section the dynamic behaviour of several thin structures will be analysed using the NNRPIM.

4.2.1 Cantilever Beam

The NNRPIM is applied to analyse the free vibration of a cantilever beam, figure 81. The geometrical parameters of the beam are, $L=48.0m$, $D=12.0m$ and thickness $h=1.0m$. The material properties are Young's modulus $E=3 \times 10^7 N/m^2$, Poisson ratio $\nu=0.3$ and mass density $\rho=1.0kg/m^3$. A convergence study was performed. The problem was analysed in 2D and 3D, with several numerical methods. Considering the problem as a two-dimensional problem (2D plain stress deformation theory), it was used the FEM, 9 node element, the Radial Point Interpolation Method [21] and the NNRPIM. Considering the problem as a three-dimensional problem it was used in the analysis the NNRPIM 3D classic formulation [29] and the NNRPIM 3D Shell-Like formulation. The problem domain was discretized in two different meshes, regular meshes and irregular meshes, figure 81.

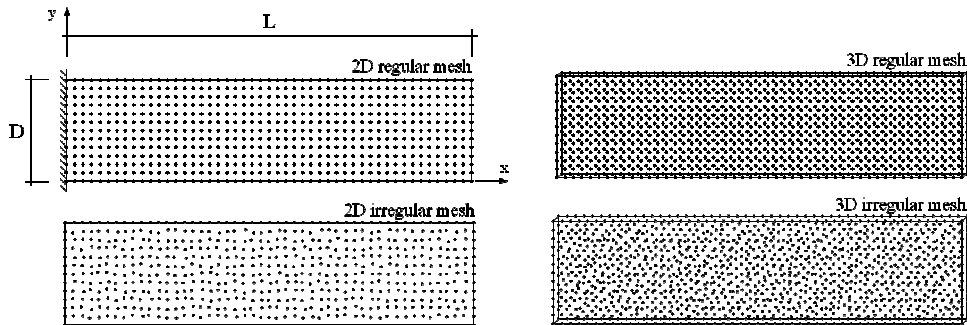


figure 81 – Cantilever beam. Examples of 2D and 3D, regular and irregular meshes.

2D	Regular Mesh			3D	Regular Mesh		Irregular Mesh	
	RPIM $p=1.03; c=1$	FEM 9 nodes	NNRPIM classic		NNRPIM classic	NNRPIM shell-like	NNRPIM classic	NNRPIM shell-like
27	29.725	167.830	29.599	27 x 2	29.501	29.635	28.294	28.422
85	28.054	83.301	28.050	85 x 2	28.085	28.109	28.097	28.211
175	27.845	55.478	27.840	175 x 2	27.892	27.904	27.868	27.968
297	27.785	41.594	27.775	297 x 2	27.833	27.840	27.735	27.825
451	27.759	33.270	27.747	451 x 2	27.807	27.812	27.719	27.808
637	27.746	27.722	27.732	637 x 2	27.793	27.797	27.701	27.787

table 13 – Convergence of the first natural frequency ω (rad/s) varying the number of nodes discretizing the cantilever beam domain.

In table 13 are presented the obtained results regarding the first natural frequency. It is perceptible that the NNRPIM 3D Shell-Like formulation produces results very close to the others formulations and present a fast convergence to the final solution. The use of irregular meshes does not seem to disturb the method performance. In table 14 are presented the first ten natural frequencies obtained for the finest mesh indicated in table 13.

Vibration mode	Regular Mesh			Regular Mesh		Irregular Mesh	
	RPIM	FEM	NNRPIM	NNRPIM	NNRPIM	NNRPIM	NNRPIM
	p=1.03; c=1 2D	9 nodes 2D	classic 2D	classic 3D	shell-like 3D	classic 3D	shell-like 3D
1	27.746	27.722	27.732	27.793	27.797	27.701	27.787
2	141.285	140.838	141.257	141.429	141.478	140.933	141.373
3	179.746	179.714	179.737	179.961	179.967	179.246	179.806
4	323.371	323.724	323.214	323.538	323.722	324.435	323.469
5	526.705	522.970	526.269	526.807	527.238	524.940	526.581
6	536.799	536.534	536.741	537.396	537.438	535.459	537.133
7	735.818	729.172	734.954	735.873	736.666	732.426	734.716
8	881.841	881.058	881.650	882.648	882.776	879.683	882.432
9	905.421	898.567	904.427	905.415	906.243	901.604	904.423
10	1006.030	999.162	1005.051	1006.148	1006.969	1002.522	1005.656

table 14 – First ten natural frequencies ω (rad/s) obtained for the cantilever beam.

In order to determine the NNRPIM 3D Shell-Like formulation efficiency, the same problem was analysed. The time spend in the analysis was registered and compared with the time spend with the NNRPIM classic formulation. The result was categorical, the NNRPIM 3D Shell-Like formulation is three times faster than the NNRPIM classic formulation [37].

4.2.2 Variable Cross Section Beams

In this example two distinct cantilever beams with a variable cross section are presented, figure 82(a) and (b). The geometrical parameters of beam A are, $L = 10.0\text{ m}$, $D_1 = 5.0\text{ m}$ and $D_2 = 3.0\text{ m}$. For the beam B, $L_1 = 8.0\text{ m}$, $L_2 = 2.0\text{ m}$, $D_1 = 3.0\text{ m}$ and $D_2 = 1.0\text{ m}$. For both beams the thickness is $h = 1.0\text{ m}$ and the material properties are Young's modulus $E = 3 \times 10^7\text{ N/m}^2$, Poisson ratio $\nu = 0.3$ and mass density $\rho = 1.0\text{ kg/m}^3$. The problem is analysed considering a 2D and a 3D approach. The nodal arrangements for both beams are presented in figure 82. For the 2D analysis the beam A is discretized with 231 nodes and beam B with 287 nodes. In the 3D analysis 1122 nodes are used for beam A and 918 nodes for beam B,

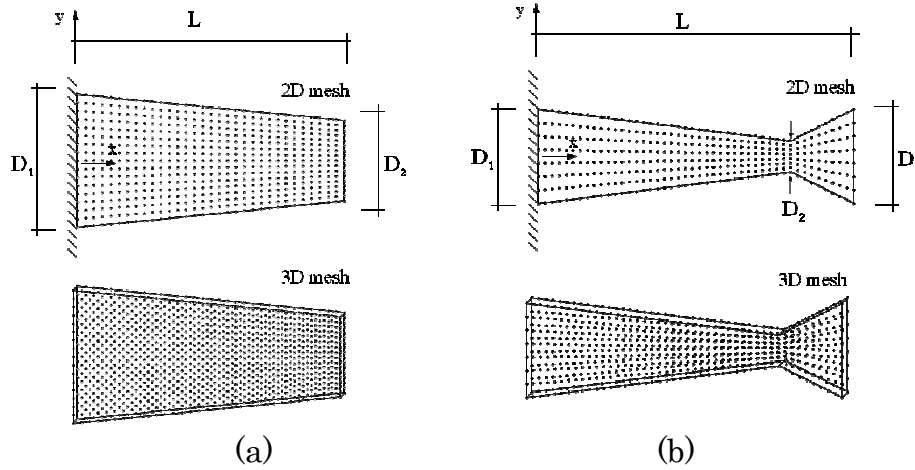


figure 82 – Cross section of the variable beams and respective nodal arrangement for the 2D and 3D modulation. (a) Beam A and (b) Beam B.

The first five natural frequencies (rad/s) obtained for beam A with the 2D-NNRPIM formulation, with the 3D-NNRPIM classical formulation, with the 3D-NNRPIM shell-like formulation, with the MLPG [78], with the LRPIM [79] and with the FEM-ABAQUS are presented in table 15. The results obtained for beam B are presented in table 16.

Vibration mode	MLPG	LRPIM $p=1.03; c=1$	NNRPIM classic	NNRPIM classic	NNRPIM shell-like	FEM Abaqus
	2D	2D	2D	3D	3D	2D
1	263.21	262.13	262.58	262.20	262.39	262.09
2	923.03	920.81	925.21	921.32	921.65	918.93
3	953.45	952.06	952.43	953.42	953.39	951.86
4	1855.14	1854.32	1874.89	1863.06	1863.43	1850.92
5	2589.78	2589.87	2589.76	2590.23	2590.37	2578.63

table 15 – Obtained natural frequencies ω (rad/s) with the meshless solutions and the FEM for beam A.

The results of the distinct analyses are obtained for the same nodal discretization. It is visible that the results obtained with the NNRPIM are in a good agreement with both meshless methods and with the FEM.

Vibration mode	NNRPIM classic 2D	NNRPIM classic 3D	NNRPIM shell-like 3D	FEM 9 nodes 2D
1	132.21	134.01	138.14	131.59
2	484.63	476.90	518.53	469.88
3	878.63	891.45	943.94	878.31
4	1117.20	1075.85	1168.69	1062.42
5	1942.27	1883.92	2002.34	1863.21

table 16 – Obtained natural frequencies ω (rad/s) with the meshless solutions and the FEM for beam B.

4.2.3 Shear-Wall

A shear wall with four openings is studied in this example. The geometrical parameters of the shear wall are presented in figure 83, as well as the 2D nodal mesh of 559 nodes and 3D nodal mesh of 1118 nodes. The material properties are Young's modulus $E = 1000 \text{ N/m}^2$, Poisson ratio $\nu = 0.2$ and mass density $\rho = 1.0 \text{ kg/m}^3$.

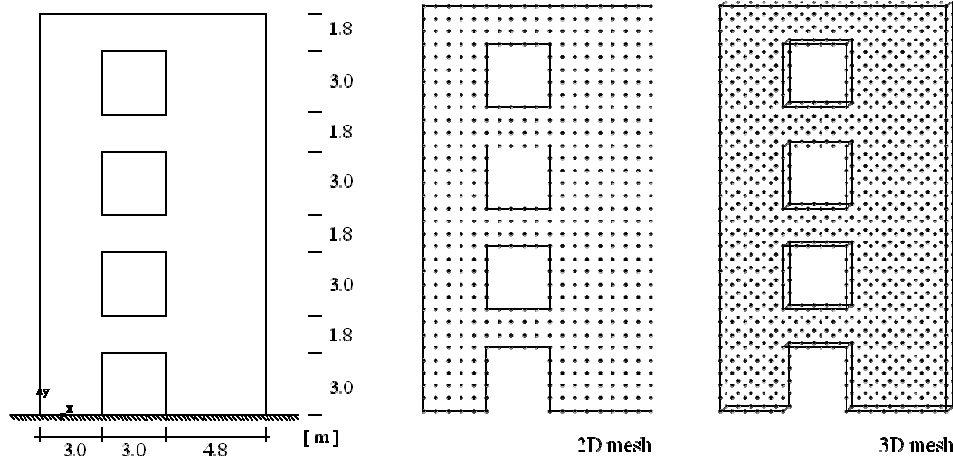


figure 83 – Shear wall with four openings and respective mesh discretization for the 2D and the 3D modulation.

The problem is analysed with the 2D NNRPIM formulation, with the 3D-NNRPIM classical and shell-like formulations, and the obtained results are compared with the MLPG [78], with the LRPIM [79], with the Boundary Element Method (BEM) [80], and with the FEM-ABAQUS for the same nodal arrangement. The obtained results of the first five frequencies are presented in table 17.

Vibration mode	MLPG 2D	BEM 2D	LRPIM p=1.03; c=1 2D	NNRPIM classic 2D	NNRPIM classic 3D	NNRPIM shell-like 3D	FEM Abaqus 2D
1	2.069	2.079	2.086	2.098	2.276	2.280	2.073
2	7.154	7.181	7.152	7.110	7.107	7.106	7.096
3	7.742	7.644	7.647	7.647	7.520	7.532	7.625
4	12.163	11.833	12.019	12.353	12.297	12.351	11.938
5	15.587	15.947	15.628	15.418	15.790	15.789	15.341

table 17 – First five natural frequencies ω (rad/s) obtained for the shear-wall.

4.2.4 Free Vibrations of Plates

Several plates with various shapes are studied in this subsection. In all examples the material properties are the same: Young modulus $E=1000\text{ N/m}^2$, Poisson ratio $\nu=0.3$ and mass density $\rho=100\text{ kg/m}^3$. A square plate with the dimensions $L=D=10\text{ m}$ and $h=0.1\text{ m}$ was considered in first place. Several boundary conditions are considered in the analysis. The boundary $x=0$ is identified as bc1 and $x=L$ corresponds to bc3. The bc2 and the bc4 are respectively the boundaries $y=0$ and $y=D$. Respecting the order bc1–bc2–bc3–bc4 four types were considered: SSSS, CCCC, SCSC and SCSS, where S stands for simply supported and C for clamped edge.

In the convergence study of the square plate two types of meshes are used in the analysis, nodal meshes with regular distribution and nodal meshes with irregular distribution. Examples of both are shown in figure 62. As it was explained in section 3.8 for the NNRPIM 3D Shell-Like formulation there is no nodes along the plate or shell thickness, there is only one up-layer and bottom-layer of nodes.

In figure 84 the convergence studies for the SSSS, CCCC, SCSC and SCSS square plates are presented. The NNRPIM 3DSL formulation is compared with the Reissner-Mindlin analytical solution (RMAS) [81]. The figure shows the first natural frequencies obtained with the 3DSL formulation for increasing nodal meshes. The natural frequencies are normalised with the expression $\omega_{\text{norm}} = \omega \left(\frac{L^2}{\pi^2} \right) \sqrt{\rho h / \kappa}$ where $\kappa = E h^3 / (12(1-\nu^2))$.

From table 18 to table 21 are presented the normalized natural frequencies of the square plate for the referred boundary conditions. It was used in the

analysis a regular (RM) and an irregular mesh (IM) with $25 \times 25 \times 2 = 1250$ nodes. The NNRPIM 3DSL formulation results are compared with the NNRPIM using the Reissner-Mindlin plate theory (NNRPIM-P), with FEM results [82] and with the analytical solution [81] (RMA5).

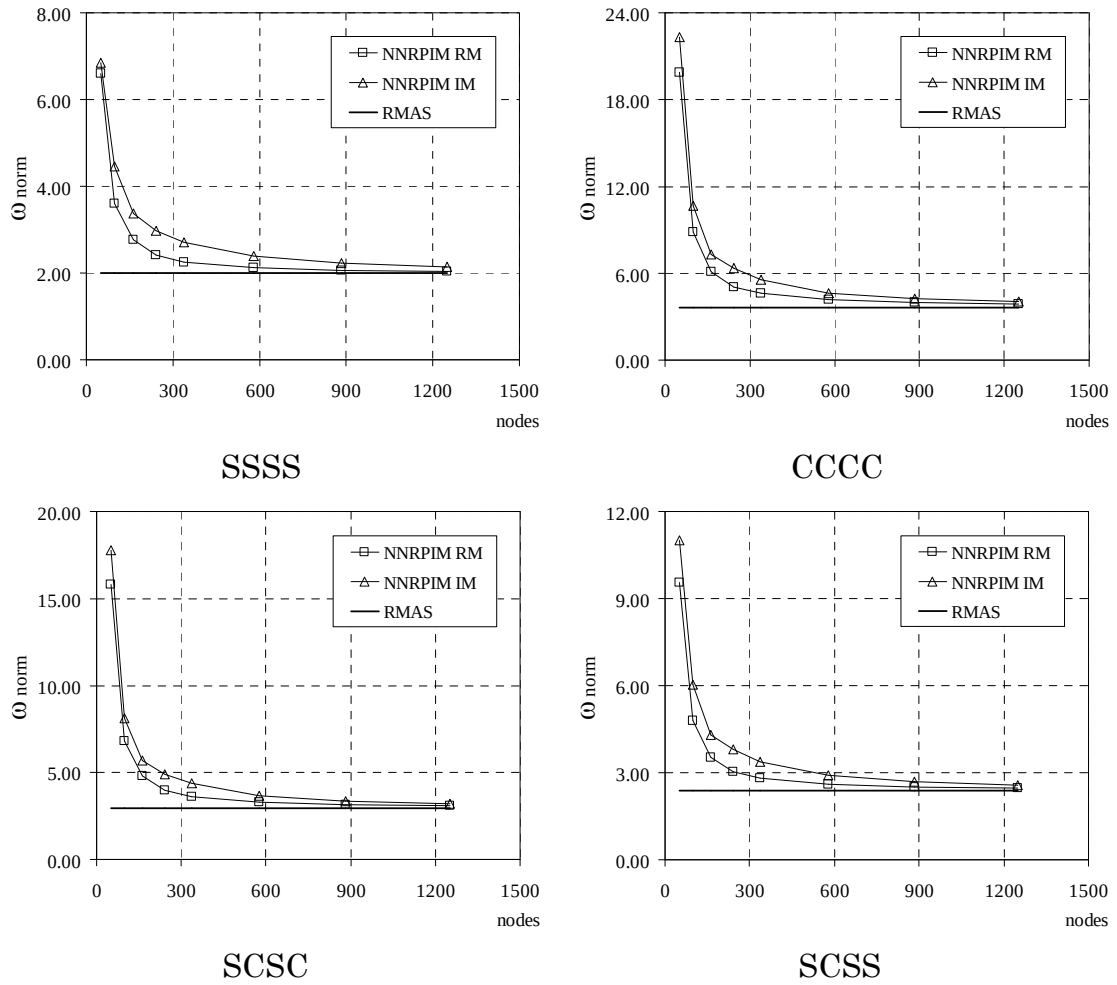


figure 84. – Square plate convergence studies for distinct boundary conditions.

SSSS		h/L = 0.01				
vibration mode (m,n)		RMAS	FEM	NNRPIM-P	SL3D - RM	SL3D - IM
1	1,1	2.000	2.006	2.025	2.044	2.136
2	2,1	4.996	5.046	5.077	5.153	5.280
3	1,2	4.996	5.046	5.077	5.153	5.324
4	2,2	7.990	8.077	8.084	8.323	8.706
5	3,1	9.986	10.233	10.203	10.469	10.612
6	1,3	9.986	10.233	10.222	10.493	10.649
7	3,2	12.976	13.248	13.159	13.734	14.174
8	2,3	12.976	13.248	13.159	13.734	14.284
9	4,1	16.959	17.756	17.486	18.272	18.433
10	1,4	16.959	17.756	17.486	18.272	18.459

table 18 – Normalized natural frequencies of the SSSS square plate.

CCCC		h/L = 0.01				
vibration mode (m,n)		RMAS	FEM	NNRPIM-P	SL3D - RM	SL3D - IM
1	1,1	3.642	3.665	3.872	3.879	4.026
2	2,1	7.425	7.548	7.911	8.016	8.287
3	1,2	7.425	7.548	7.911	8.016	8.326
4	2,2	10.951	11.128	11.527	11.827	12.528
5	3,1	13.294	13.775	14.275	14.722	15.094
6	1,3	13.356	13.775	14.357	14.800	15.190
7	3,2	-	17.166	17.585	18.367	19.296
8	2,3	-	17.166	17.585	18.367	19.385
9	4,1	-	22.584	23.030	24.323	24.794
10	1,4	-	22.949	23.030	24.323	24.866

table 19 – Normalized natural frequencies of the CCCC square plate.

SCSC		h/L = 0.01				
vibration mode (m,n)		RMAS	FEM	NNRPIM-P	SL3D - RM	SL3D - IM
1	1,1	2.930	2.947	3.080	3.091	3.193
2	2,1	5.540	5.594	5.692	5.769	5.949
3	1,2	7.012	7.138	7.461	7.570	7.772
4	2,2	9.568	9.709	9.941	10.213	10.676
5	3,1	10.339	10.586	10.603	10.879	11.096
6	1,3	13.038	13.533	14.017	14.463	14.744
7	3,2	-	14.478	14.536	15.160	15.820
8	2,3	-	16.120	16.450	17.198	17.906
9	4,1	-	18.009	17.755	18.556	18.802
10	1,4	-	20.782	20.828	22.162	23.279

table 20 – Normalized natural frequencies of the SCSC square plate.

SCSS		h/L = 0.01				
vibration mode (m,n)		RMAS	FEM	NNRPIM-P	SL3D - RM	SL3D - IM
1	1,1	2.394	2.405	2.463	2.480	2.578
2	2,1	5.235	5.283	5.336	5.415	5.567
3	1,2	5.935	6.016	6.163	6.252	6.444
4	2,2	8.719	8.823	8.919	9.174	9.611
5	3,1	10.152	10.391	10.383	10.658	10.828
6	1,3	11.489	11.801	11.994	12.338	12.551
7	3,2	-	13.815	13.781	14.383	14.944
8	2,3	-	14.602	14.686	15.338	15.941
9	4,1	-	17.872	17.607	18.402	18.618
10	1,4	-	19.515	19.382	20.618	21.276

table 21 – Normalized natural frequencies of the SCSS square plate.

As it can be seen the 3DSL results are very close to the NNRPIM and to the FEM results using the 2D plate bending theory, regardless the considered boundary condition.

Consider now a thin circular clamped plate, with radius $R=5m$ and thickness $h=0.1m$. The material properties are the same as in previous example. Regular and irregular meshes are considered, figure 85. The convergence study is presented in figure 86.

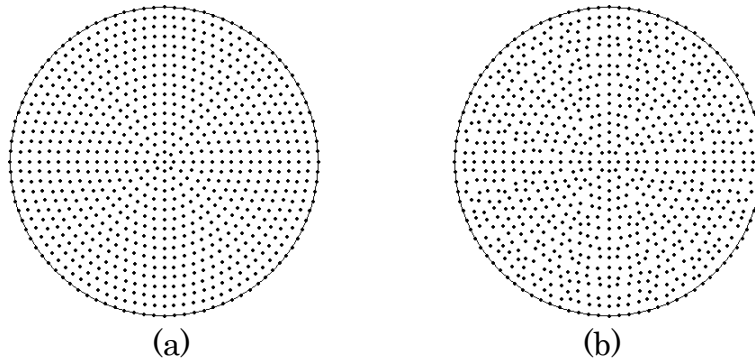


figure 85 – (a) Circular regular mesh. (b) Circular irregular mesh.

The results are normalized with the expression $\omega_{\text{norm}} = \omega R^2 \sqrt{\rho h / \kappa}$ and are compared with the Reissner-Mindlin analytical solution (RMAS) [81]. It is visible that regardless the used mesh the NNRPIM solution converges very fast and that the converged solution is very near the analytical solution.

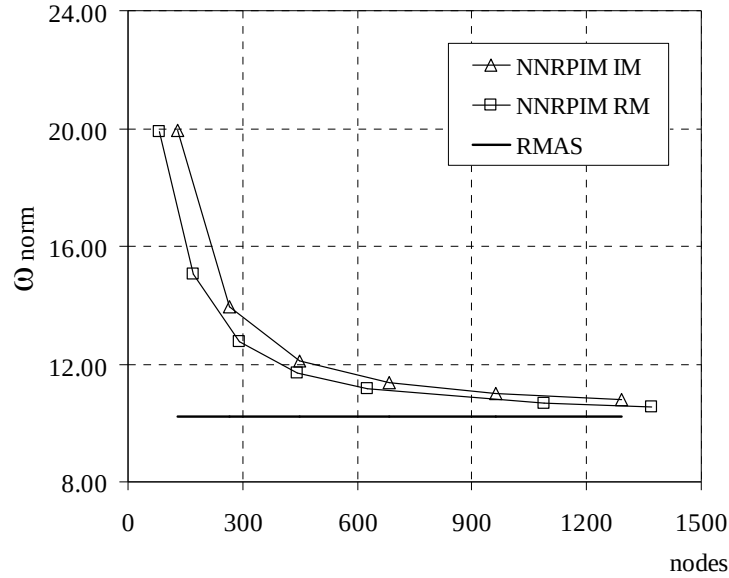


figure 86 – Convergence study of the circular clamped plate.

In table 22 are presented the first ten normalized natural frequencies of the circular clamped plate obtained with the analysis of a 1370 nodes regular mesh and with the analysis of a 1290 nodes irregular mesh. Again the NNRPIM 3DSL formulation results are compared with the NNRPIM-P, with the FEM [82] and with the analytical solution [81] (RMAS).

h/2R = 0.01					
mode	RMPT	FEM	NNRPIM-P	SL3D - RM	SL3D - IM
1	10.216	10.257	10.621	10.537	10.772
2	21.260	21.498	22.091	22.066	22.669
3	21.260	21.498	22.091	22.066	22.669
4	34.877	35.394	36.187	36.664	37.647
5	34.877	35.517	36.237	36.862	37.698
6	39.771	40.898	41.443	41.840	43.155
7	51.030	52.205	52.930	54.666	55.903
8	51.030	52.205	52.930	54.666	55.903
9	60.829	63.240	63.421	65.014	67.344
10	60.829	63.240	63.421	65.014	67.344

table 22 – Normalized natural frequencies of the circular clamped plate.

The next example is a skewed plate example, consider a plate with the geometric properties and essential boundary conditions as represented in figure 87. The angle θ varies between 0° and 50° degrees. The material properties are again the same as in previous example. The same mesh discretization is used in every analysis, a $25 \times 25 \times 2 = 1250$ nodes regular mesh.

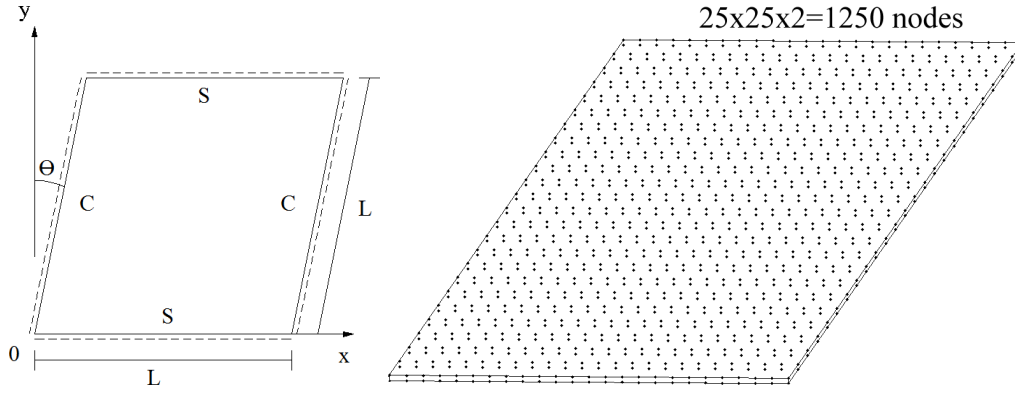


figure 87 – Generic skewed plate geometric properties and used regular mesh with $\theta=30^\circ$.

For each value of θ considered the first ten natural frequencies are obtained. The normalized natural frequencies are presented in table 23 and obtained with the expression $\omega_{\text{norm}} = \omega \left(L^2 / \pi^2 \right) \sqrt{\rho h / \kappa}$. The 3DSL NNRPIM results are compared with the FEM solution [82] and with the analytical solution [83] (RMAS).

θ		vibration modes									
		1	2	3	4	5	6	7	8	9	10
0°	RMAS	2.93	5.55	7.02	9.59	10.36	13.08	14.21	15.69	-	-
	FEM	2.95	5.59	7.14	9.71	10.59	13.53	14.48	16.12	-	-
	3DSL	3.09	5.77	7.57	10.21	10.88	14.46	15.16	17.20	18.56	22.16
5°	RMAS	2.95	5.57	7.08	9.56	10.50	13.18	14.14	15.87	-	-
	FEM	2.97	5.62	7.20	9.68	10.72	13.64	14.42	16.30	-	-
	3DSL	3.17	5.91	7.72	10.45	11.15	14.68	15.52	17.70	18.85	22.38
10°	RMAS	3.01	5.64	7.27	9.53	10.86	13.50	14.03	16.34	-	-
	FEM	3.03	5.68	7.38	9.66	11.06	13.96	14.31	16.75	-	-
	3DSL	3.20	5.93	7.85	10.25	11.51	14.94	15.23	18.03	19.32	21.61
15°	RMAS	3.12	5.77	7.58	9.55	11.39	13.98	14.08	16.96	-	-
	FEM	3.13	5.80	7.68	9.69	11.58	14.26	14.51	17.38	-	-
	3DSL	3.29	6.02	8.15	10.19	12.06	15.03	15.50	18.55	20.26	21.11
20°	RMAS	3.28	5.96	8.04	9.70	12.10	14.05	14.91	17.75	-	-
	FEM	3.28	5.98	8.13	9.82	12.27	14.32	15.34	18.19	-	-
	3DSL	3.44	6.20	8.61	10.26	12.79	15.01	16.36	19.28	20.88	21.62
25°	RMAS	3.50	6.23	8.68	9.95	13.00	14.30	16.10	18.80	-	-
	FEM	3.48	6.24	8.74	10.06	13.16	14.53	16.48	19.21	-	-
	3DSL	3.65	6.45	9.24	10.47	13.71	15.16	17.56	20.24	20.93	23.37
30°	RMAS	3.80	6.60	9.54	10.30	14.10	14.70	17.80	19.60	-	-
	FEM	3.77	6.59	9.57	10.44	14.30	14.92	17.99	20.42	-	-
	3DSL	3.94	6.81	10.09	10.83	14.87	15.51	19.13	21.29	21.50	25.43
35°	RMAS	4.20	7.10	10.70	10.90	15.40	15.70	19.60	20.90	-	-
	FEM	4.14	7.07	10.66	10.99	15.54	15.76	19.83	21.36	-	-
	3DSL	4.33	7.29	11.21	11.36	16.07	16.33	20.90	22.28	23.16	27.27
40°	RMAS	4.75	7.79	11.80	12.20	16.40	17.60	21.30	23.30	-	-
	FEM	4.65	7.72	11.77	12.12	16.44	17.68	21.49	23.56	-	-
	3DSL	4.84	7.93	12.11	12.69	16.91	18.23	22.23	24.80	25.35	28.54
45°	RMAS	5.51	8.74	12.90	14.40	17.80	20.30	23.00	27.30	-	-
	FEM	5.34	8.61	12.86	14.09	17.75	20.23	23.13	27.31	-	-
	3DSL	5.53	8.80	13.14	14.68	18.11	20.74	23.62	28.25	28.74	30.05
50°	RMAS	6.59	10.10	14.60	17.40	19.70	24.00	25.30	32.00	-	-
	FEM	6.31	9.84	14.39	16.80	19.64	23.72	25.34	31.45	-	-
	3DSL	6.49	10.00	14.60	17.41	19.87	24.13	25.56	31.73	32.68	34.31

table 23 – Skewed plates normalized natural frequencies.

Despite the plate distortion angle used the 3DSL formulation produces results very near the analytical solution.

4.2.5 Free Vibration of Shells

In this subsection shell structures are considered. The first example analysed is the thin clamped cylindrical shell panel. This example was experimented in Southampton University [84] and then it was verified

numerically with the FEM [85]. The geometric and material properties of the problem are presented in figure 88. Taking advantage of the symmetry, only one quarter of the cylindrical shell panel was considered and the necessary symmetric or anti-symmetric boundary conditions were introduced along the lines of symmetry. The problem domain was discretized in several regular and irregular meshes in order to study the convergence of the numerical method. Examples of the used meshes in the analysis are presented as well in figure 88.

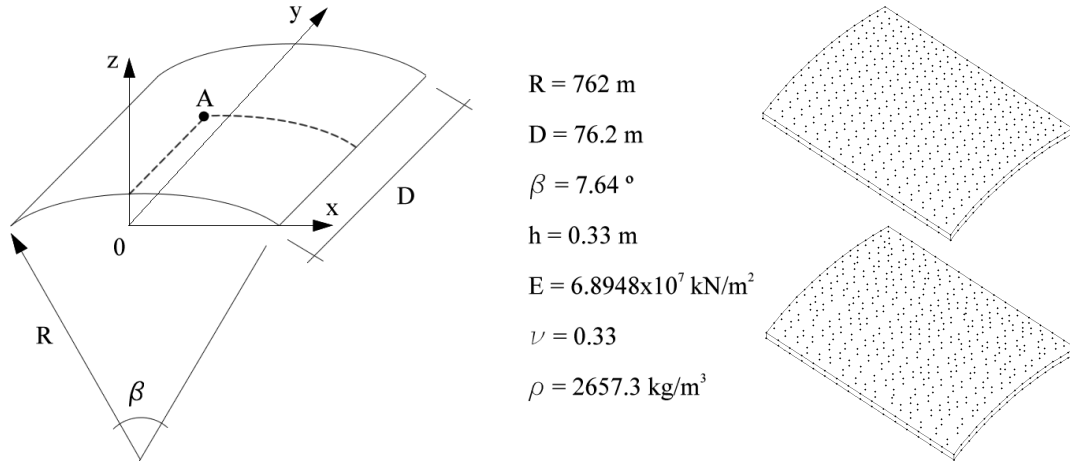


figure 88 – Thin clamped cylindrical shell panel geometric and material properties and regular and irregular meshes examples used in the analysis.

In figure 89 is presented a diagram with the evolution of the first vibration frequencies (Hz) with increasing number of nodes discretizing the problem domain. The results are compared with the experimental results [84] and the FEM solution [85]. For future comparison the final converged first vibration frequencies obtained were $f_{3\text{DSL-RM}} = 811.19 \text{ Hz}$ and $f_{3\text{DSL-IM}} = 825.53 \text{ Hz}$. The reference solutions are $f_{\text{exp}} = 814 \text{ Hz}$ and $f_{\text{FEM}} = 869 \text{ Hz}$. Notice that the NNRPIM solution, in comparison with the FEM solution, is much closer to the experimental solution.

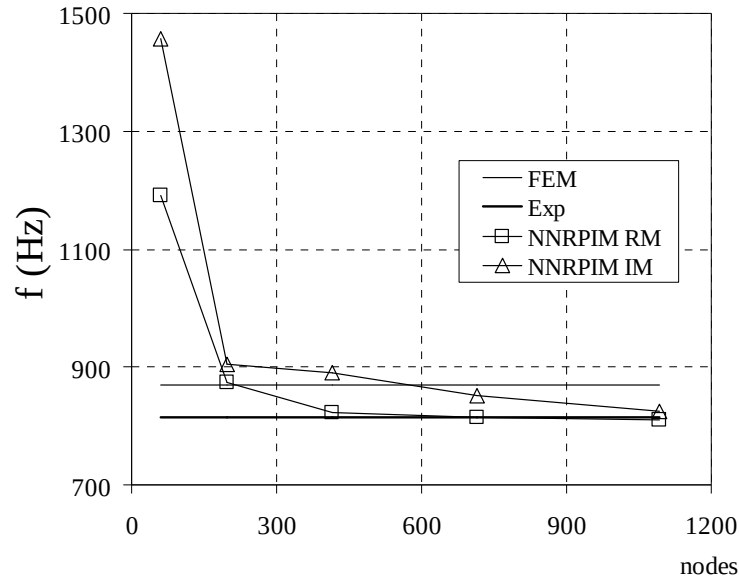


figure 89 – Convergence study of the thin clamped cylindrical shell panel.

The next shell example is the clamped-free cylindrical shell. This example was analysed with the High Precision Finite Element Method (HPFEM) in a reference work [86]. The geometric and material properties of the shell problem are shown in figure 90. The shell is clamped in the $y=0$ edge and free in the $y=D$ edge. Only one quarter of the cylindrical shell panel was considered in the analysis. The necessary symmetric or anti-symmetric boundary conditions were introduced along the lines of symmetry. In the convergence study are used regular and irregular meshes, examples of such meshes are presented in figure 90.

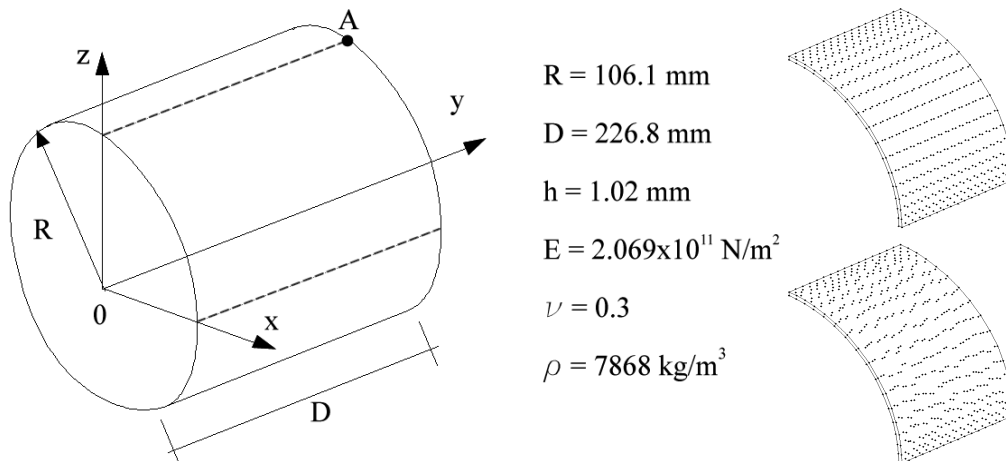


figure 90 – Thin clamped-free cylindrical shell panel geometric and material properties and regular and irregular meshes examples used in the analysis.

The results are presented in figure 91. It can be seen that the 3DSL NNRPIM approaches the HPFEM solution. The final converged values

presented are: $f_{3DSL-RM} = 485.52 \text{ Hz}$, $f_{3DSL-IM} = 517.74 \text{ Hz}$ and $f_{HPFEM} = 487 \text{ Hz}$, which are in fact very near each other.

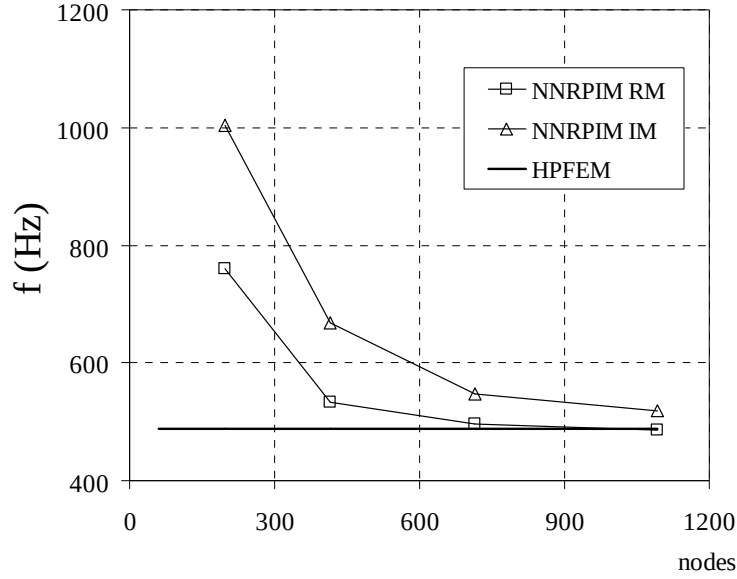


figure 91 – Convergence study of the thin clamped-free cylindrical shell panel.

4.2.6 Plates Transient Response

In this subsection square plates are subjected to forced vibration. The material properties, the geometrical properties and the boundary conditions are the same as in subsection 4.2.4, with the exception in the thickness-span ratio, considered in this analysis as $h/L = 0.05$. The plate is subjected to a uniform distributed load, $f(t) = g(t)$, where $g(t)$ is a time function. The three different time-dependent loading conditions presented in subsection 2.3.6.1, equations (122), (123) and (124), are considered and for the load case B, in this example, t_i changes with the plate distinct boundary conditions. For SSSS, $t_i = 0.01050 \text{ s}$, for CCCC, $t_i = 0.00625 \text{ s}$, for SCSC, $t_i = 0.00785 \text{ s}$ and for SCSS, $t_i = 0.00945 \text{ s}$.

Displacements for all nodes are obtained. Detailed results are presented for the transverse displacement in the point where the transverse displacement is maximum for a static load. For comparison are presented the results obtained with the nine node finite element using the Reissner-Mindlin plate theory (FEM-BPT) and the NNRPIM using the same plate deformation theory (NNRPIM-BPT) [32, 37]. Both approaches use a $25 \times 25 = 625$ nodes regular mesh. The NNRPIM 3D Shell-Like approach (3DSL) approach uses a $21 \times 21 \times 2 = 882$ nodes regular mesh. In figure 92 the responses to the dynamic load case A for the distinct analysed plates are presented. The time

is normalized by the expression $t_n = L^2/4\sqrt{\rho h/\kappa}$. As it is expected the plate amplitude to due the dynamic load is the double of the vertical displacement of the static solution. The 3DSL NNRPIM results are in a very good agreement with the results obtained with the FEM and the NNRPIM using a plate bending theory.

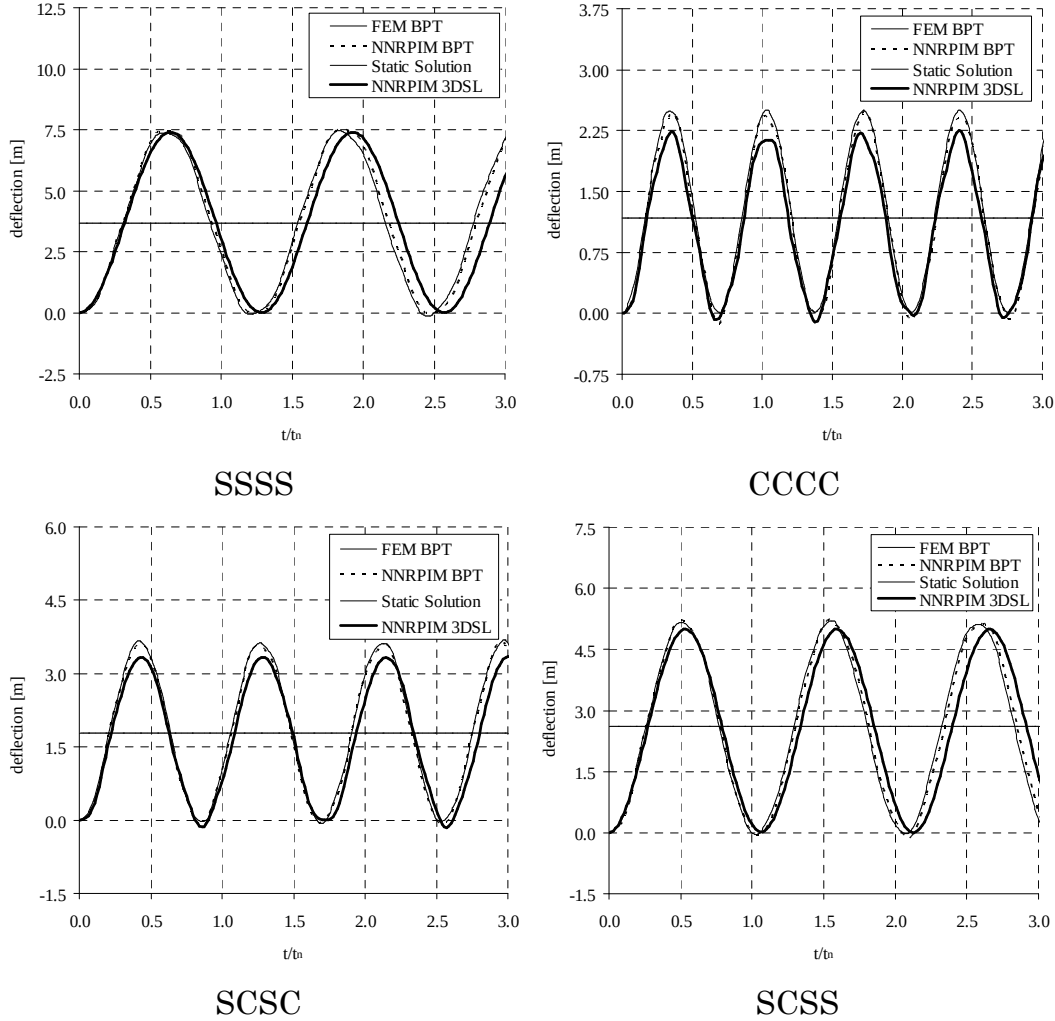


figure 92 – Square plate maximum deflection varying along time for a suddenly applied load.

The plate transient response when subjected to the load case B is presented in figure 93. In this case the plate is subjected to a load which suddenly vanish at $t = t_i$. It is visible that in the period of time between $t = 0$ and $t = t_i$ the plate response is identical to the load case A, as it should be, after $t = t_i$ the plate vibrates freely around the initial position. Once more the 3DSL NNRPIM results are similar with the FEM-BPT and NNRPIM-BPT.

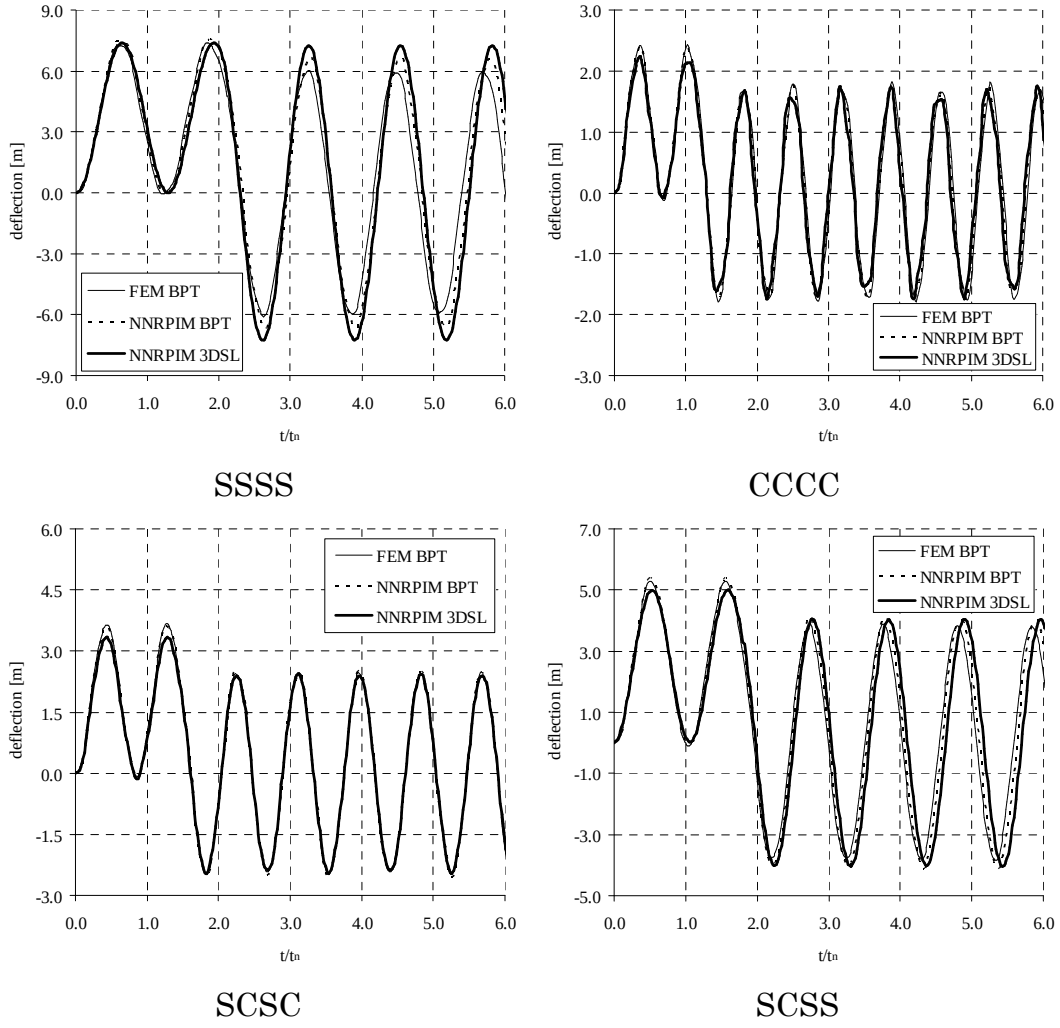


figure 93 – Square plate maximum deflection varying along time for an applied load that suddenly vanishes.

In load case C a simple harmonic dynamic load is applied to the plate. The frequency γ of the dynamic load is varied and the maximum amplitudes of the plate transverse deflection are obtained for each one of the load frequencies applied. In figure 94 the maximum value of the transverse displacement versus the normalized frequency of the dynamic load, $\gamma_{\text{norm}} = \gamma(L^2/\pi^2)\sqrt{\rho h/\kappa}$, is presented.

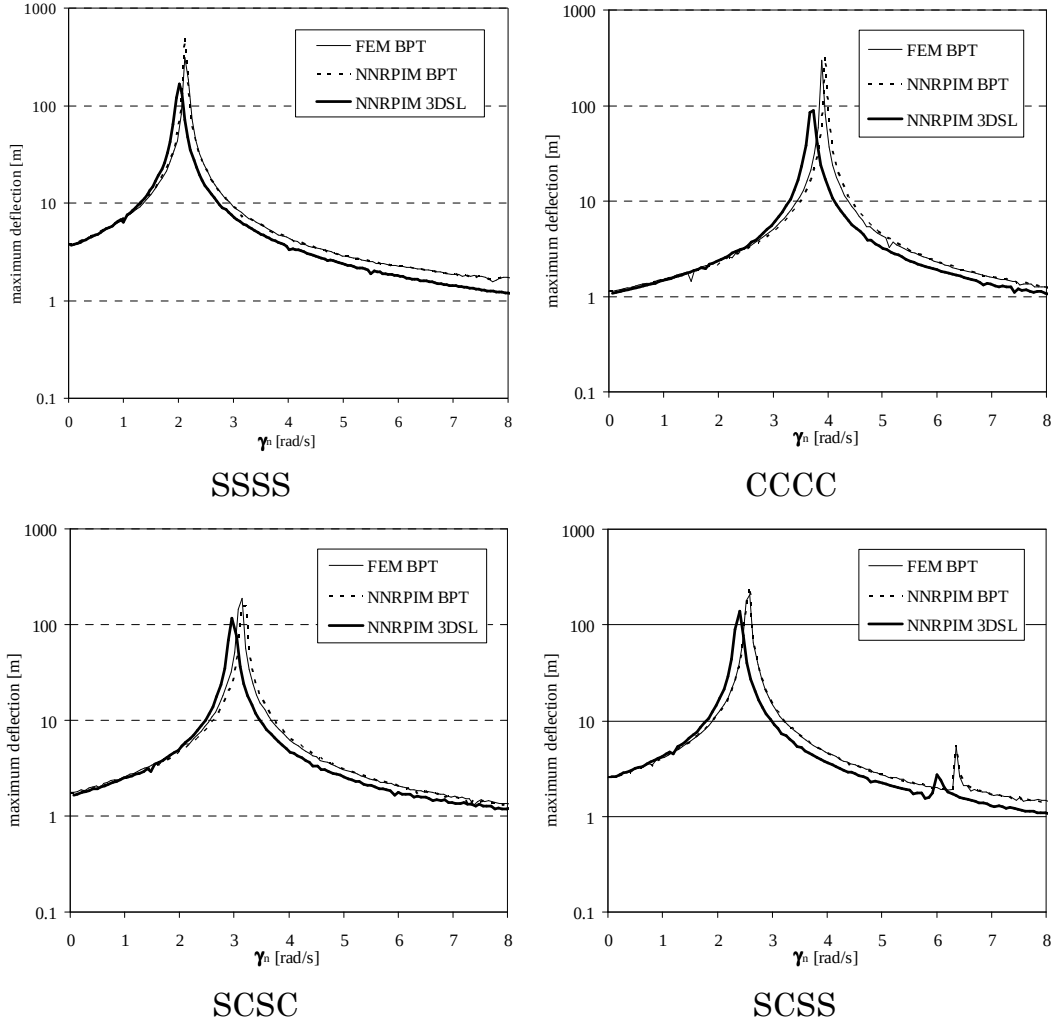


figure 94 – Square plate absolute value of the amplitude versus the load normalized frequency.

It is visible that 3DSL NNRPIM formulation present results almost coincident with the FEM-BPT and NNRPIM-BPT results.

4.2.7 Shells Transient Response

The same two shells analysed in subsection 4.2.5 are now considered for forced vibration analysis. The geometric and material properties are the same as in figure 88 and figure 90. The thin clamped cylindrical shell panel is subjected to a uniform distributed load, $f(t) = g(t)$, considering the time-dependent load presented in subsection 2.3.6.1, equations (122), (123) and (124). The clamped-free cylindrical shell is subjected to a pair of opposite loads, $f(t) = \bar{v} g(t)$, applied in point A, $\bar{v} = (0, 0, -1)$, and in the symmetric location in relation to Oxy plane, $\bar{v} = (0, 0, 1)$. In the load case B t_i changes also with the shell considered. For the thin clamped cylindrical shell panel $t_i = 0.003s$ and for the clamped-free cylindrical shell $t_i = 0.005s$. In this

analysis the domain discretization respect the regular nodal arrangement presented in figure 88 and figure 90, and for both shells it was used a regular mesh of $13 \times 16 \times 2 = 416$ nodes. In figure 95 the responses to the dynamic load case A for the two analysed shells are presented.

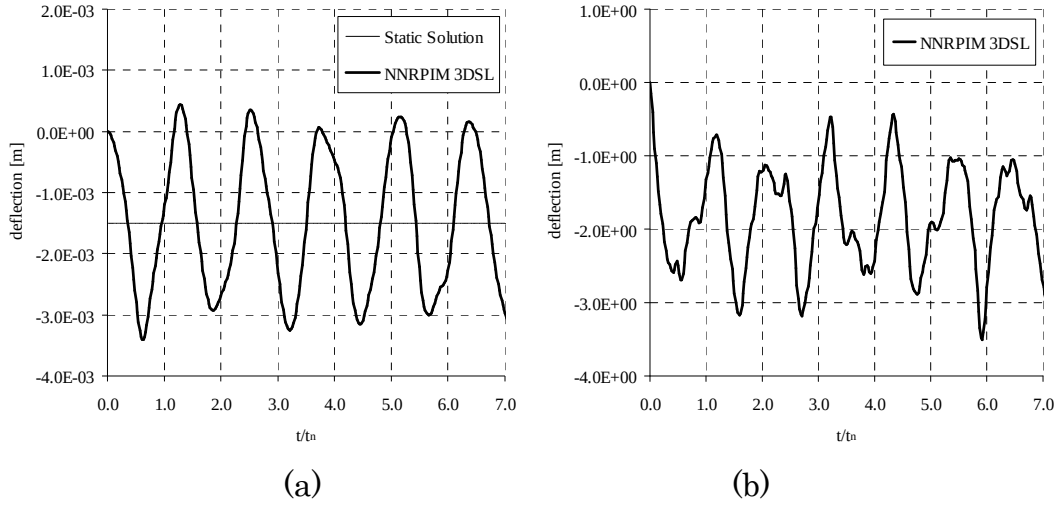


figure 95 – Maximum deflection varying along time for a suddenly applied load for (a) the thin clamped cylindrical shell panel and for (b) the clamped-free cylindrical shell.

The shell transient response when subjected to the load case B is presented in figure 96. The shell is subjected to a suddenly vanishing load at $t = t_i$, until this instant the shell response is identical to the load case A and after $t = t_i$ the shell vibrates freely around the initial position.

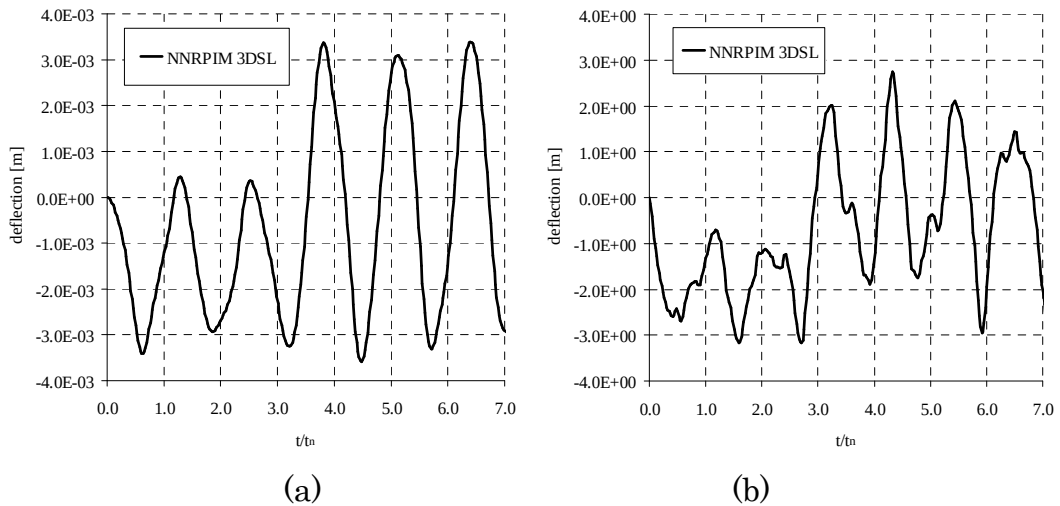


figure 96 – Maximum deflection varying along time for an applied load that suddenly vanishes for (a) the thin clamped cylindrical shell panel and for (b) the clamped-free cylindrical shell.

Finally a simple harmonic dynamic load is applied to the shell, load case C. As in previous example, the frequency γ of the dynamic load is varied and

the maximum amplitudes of the shell transverse deflection are obtained for each one of the load frequencies applied. In figure 97 it is presented the maximum value of the transverse displacement versus the normalized frequency of the dynamic load, $\gamma_{\text{norm}} = \gamma(S^2/\pi^2)\sqrt{\rho h/\kappa}$, being $S = R \sin(\beta)$ for the thin clamped cylindrical shell panel and $S = R$ for the clamped-free cylindrical shell.

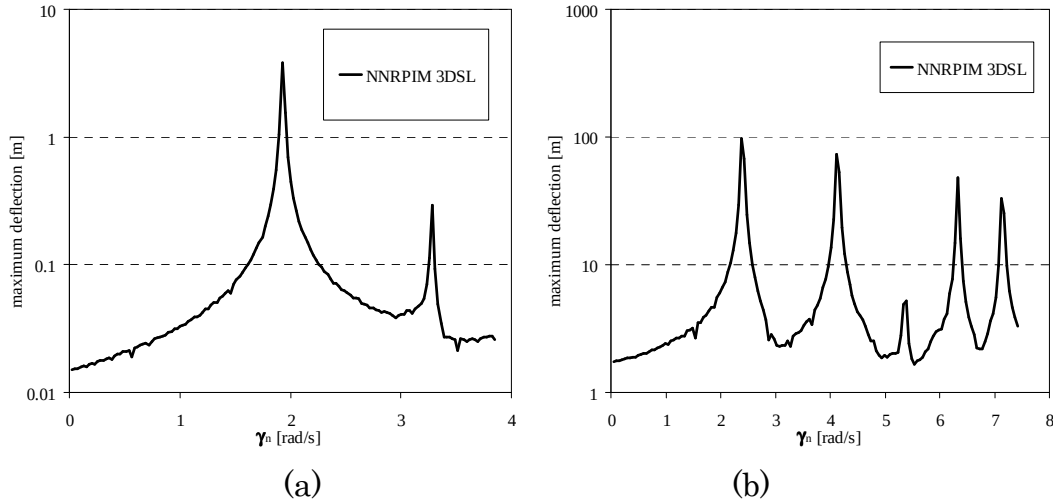


figure 97 – Absolute value of the amplitude versus the load normalized frequency for (a) the thin clamped cylindrical shell panel and for (b) the clamped-free cylindrical shell.

4.3 Conclusions and Remarks

In this chapter the NNRPIM was applied to linear solid mechanical problems. Benchmark static and dynamic examples were studied and the results permit to gauge some conclusions:

- i.* The NNRPIM is a stable and accurate interpolator meshless method, which can be applied to diverse solid mechanical problems.
- ii.* Regardless the used NNRPIM formulation (2D, 3D or 3DSL), generally the convergence rate is high and the final converged solution is always very close to the considered problem analytical solution.
- iii.* The obtained displacement field and stress field is smooth and accurate.
- iv.* The variation of the boundary conditions and the irregularity of the mesh do not disturb significantly the method performance.
- v.* The interpolation functions of the NNRPIM, which are very simple to construct, permit an easy imposition of the boundary conditions.

- vi.** The 3DSL prove to be more efficient than the classical NNRPIM approach for the 3D analysis of thin structures, presenting a lower computational cost, higher convergence rates and more accurate results.

With the results obtained and the experience acquired along the development of this work [29, 32-38] one can conclude that the NNRPIM is an efficient and alternative numerical method in the static and dynamic analysis of solid mechanics.

Chapter 5

5 Nonlinear Analysis

The nonlinear numerical analysis is an essential component of the structural design. The use of nonlinear numerical simulation instead prototype testing provides a faster and cheaper way to evaluate design concepts and details. In many manufacturing fields simulation is speeding the design process by allowing the numerical recreation of nonlinear processes, such as sheet-metal forming and extrusion. The nonlinear analysis is a complex numerical analysis, however permits to obtain optimized and efficient structures. Nowadays, the nonlinear analysis is becoming more common due the stringent design requirements and because the ever-growth computation capacity make it possible.

5.1 Material Nonlinearity

In this section several benchmark examples are presented to show that the nonlinear algorithms were successfully implemented. First well-known two-dimensional examples are solved with the NNRPIM and the obtained solution is compared with another meshless method solution and the FEM solution. Then three-dimensional examples are solved and the results are compared with the available FEM solutions.

5.1.1 Cantilever Beam

A clamped beam with the geometric and material characteristics presented in figure 98(a) is analysed with the NNRPIM, considering it as a two-dimensional Plane Stress problem.

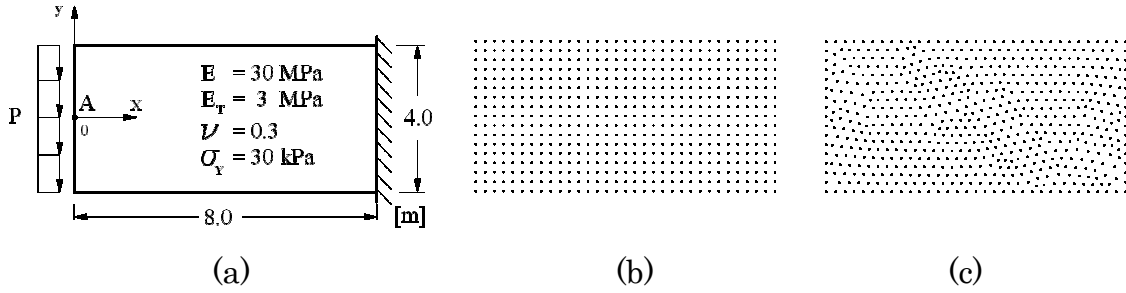


figure 98 – (a) Cantilever Beam. (b) Regular Mesh. (c) Irregular Mesh.

The problem domain is discretized in two distinct meshes a regular mesh of 561 nodes, figure 98 (b), and an irregular mesh of the same mesh density, figure 98 (c). The obtained results are compared with the FEM [87] and with the EFGM [42]. The vertical displacement of point A in relation to the tip force applied is presented in figure 99. It is perceptible by the figure that the NNRPIM results are closer to the solution obtained with the FEM than the EFGM solution. It is visible that the analysis with the irregular mesh (NNRPIM-IM) produces very similar results to the analysis with the regular mesh (NNRPIM-RM).

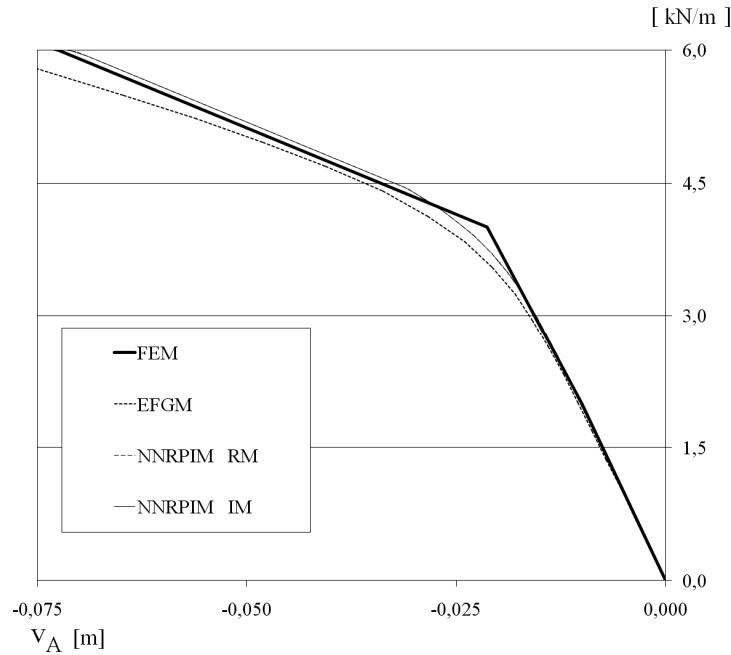


figure 99 – Load/Displacement diagram for the vertical displacement in point A.

In figure 100 it is presented the normal stress σ_{xx} distribution along the essential boundary for increasing load levels. Both analysis, with the regular and the irregular mesh, adjust well, even for high load levels.

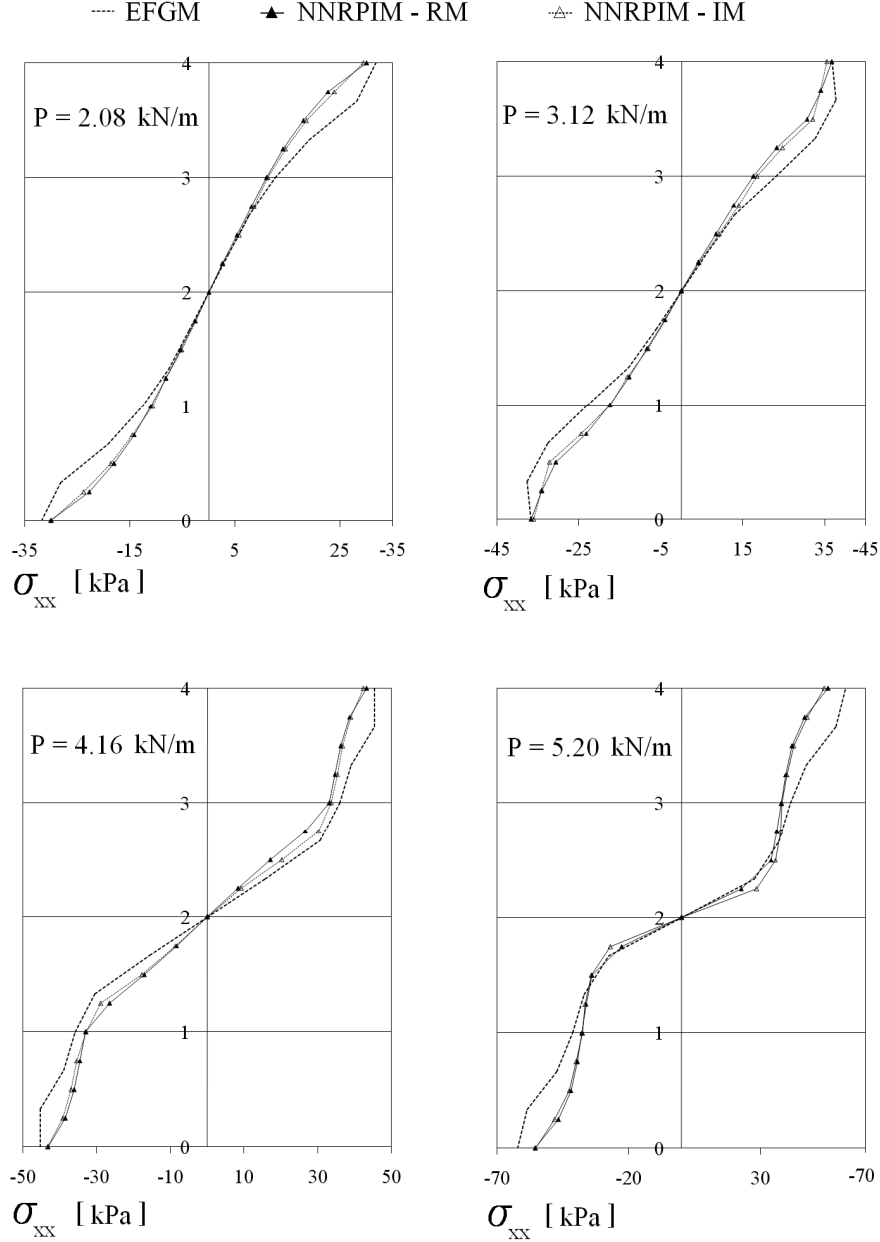


figure 100 – Normal stress distribution along the essential boundary for increasing load levels.

The difference between the EFGM results and the NNRPIM results can be explained by the difficulty of the imposition of the boundary conditions in the EFGM. The boundary conditions in the EFGM are imposed by the penalty method [42], which is not a totally accurate method to impose these conditions in approximation methods.

5.1.2 Cook Membrane

The Cook membrane presented in figure 101(a) is studied. The irregular mesh of 462 nodes on figure 101(b) is used in the NNRPIM analysis. The

two-dimensional Plane Stress deformation theory is considered. The NNRPIM solution is compared with the FEM results and the EFGM results [42]. In figure 102 it is presented the displacement on point A for an increasing load. As it is visible the NNRPIM adjust very well to the results obtained with the FEM analysis.

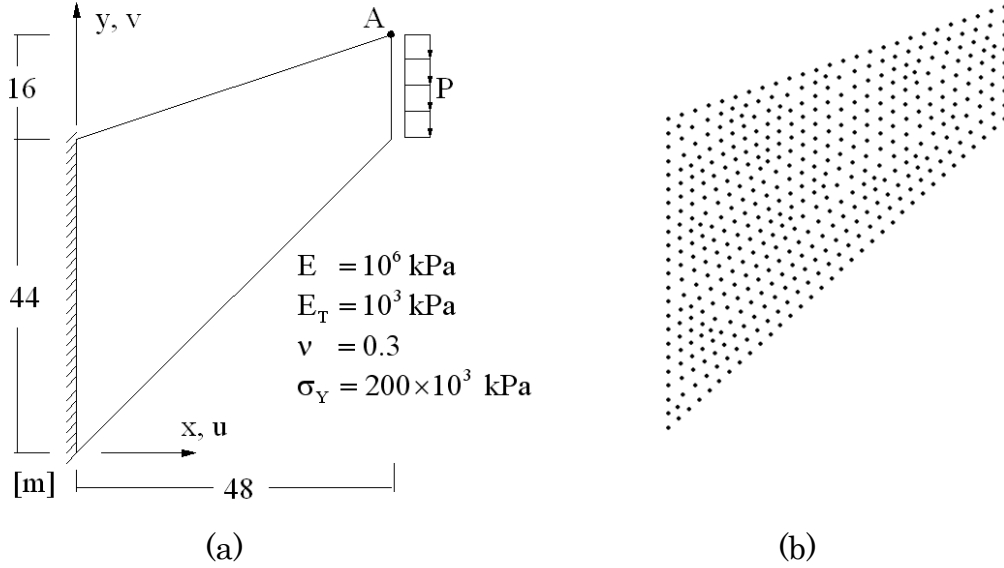


figure 101 – (a) Cook Membrane. (b) Irregular 462 node Mesh.

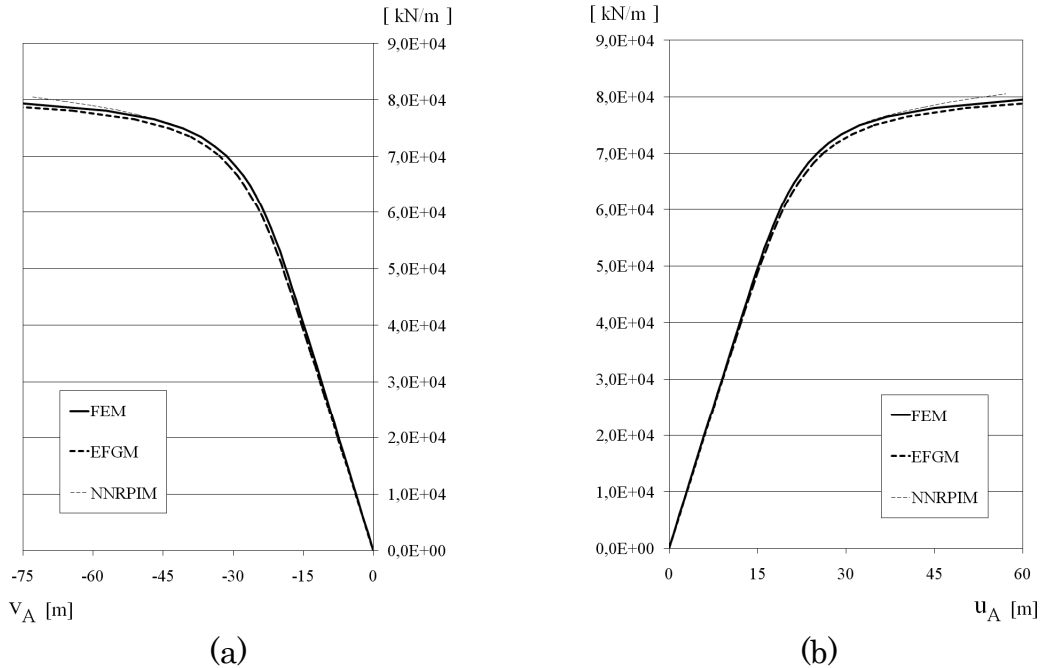


figure 102 – (a) v_A and (b) u_A displacement components in relation to the applied load.

In figure 103 and figure 104 are presented respectively the normal stress σ_{xx} and the shear stress τ_{xy} distribution along the membrane essential boundary condition, for increasing load levels.

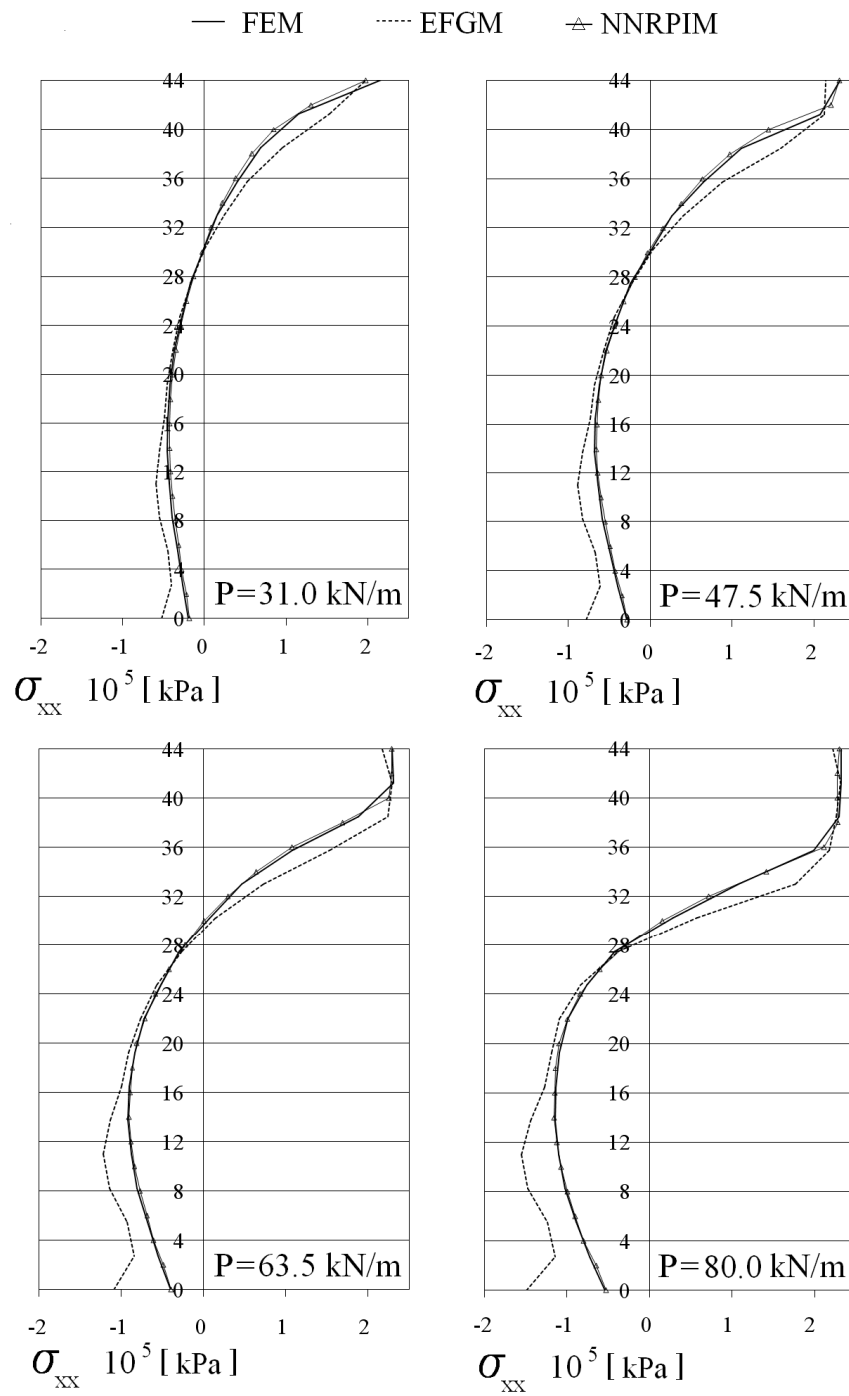


figure 103 – Normal stress distribution along the membrane essential boundary for increasing load levels.

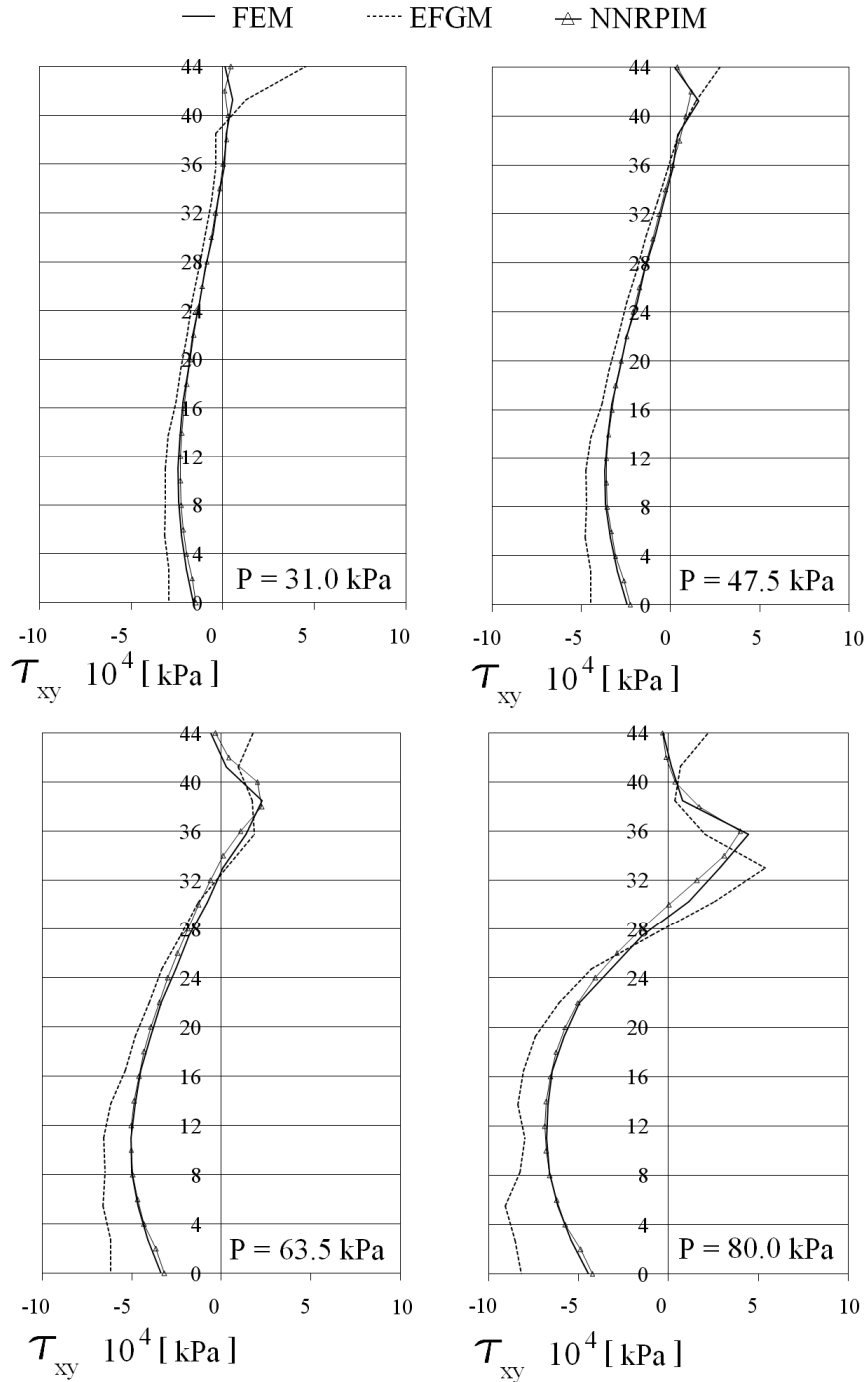


figure 104 – Shear stress distribution along the membrane essential boundary for increasing load levels.

It is visible on both figures that the NNRPIM has a high correlation with the FEM, much better than the EFGM. Once again the reason is the imposition of the essential boundary conditions, which in the NNRPIM are imposed with the direct imposition method, as in the FEM, and in the EFGM the used imposition method is the Penalty Method. The good adjustment between the NNRPIM formulation and the FEM is maintained even for high load levels.

5.1.3 Infinite Plate With a Circular Hole

Consider an infinite plate with a central hole subjected to a traction pressure in the extremities. Due the symmetry of the problem only a quarter is analysed. The geometrical and material properties of the problem are presented in figure 105(a). The problem is discretized in an irregular mesh of 660 nodes, figure 105(b), and analysed with the NNRPIM. The obtained results are compared with the nine node finite element (FEM) and with the EFGM [41]. In figure 106 are presented the results regarding to the displacements of the interest points of the plate, marked in figure 105(a), in relation to the applied load.

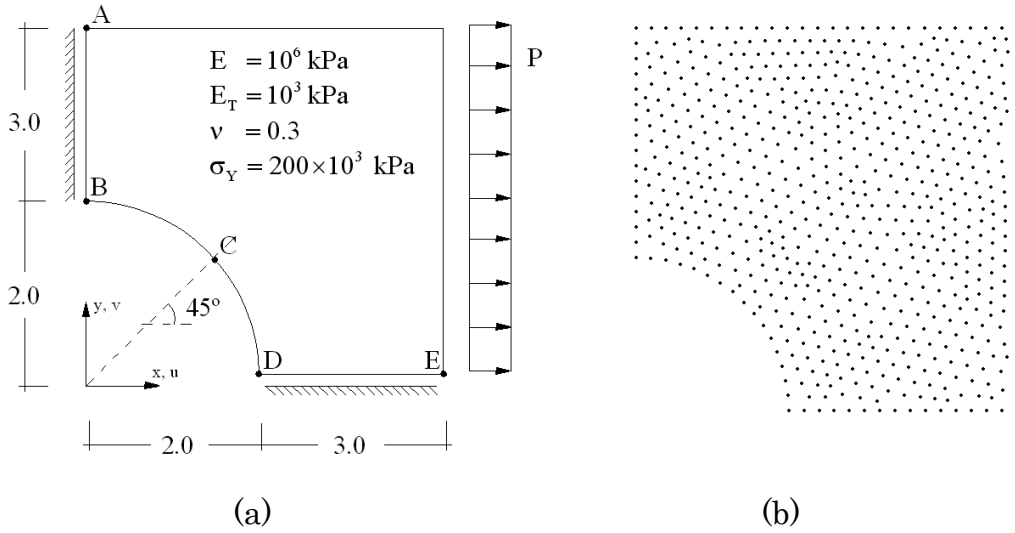


figure 105 – (a) Quarter of the plate with the circular hole. (b) 660 node mesh.

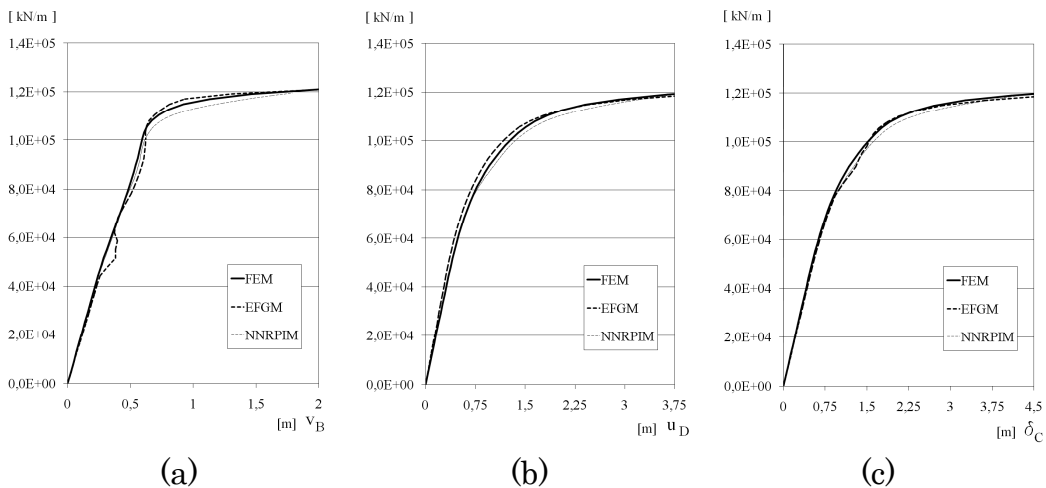


figure 106 – (a) v_B , (b) u_D and (c) δ_C displacement components in relation to the applied load.

In figure 107 it is presented the normal stress σ_{xx} distribution along the essential boundary defined by $\overline{AB}$ for increasing load levels.

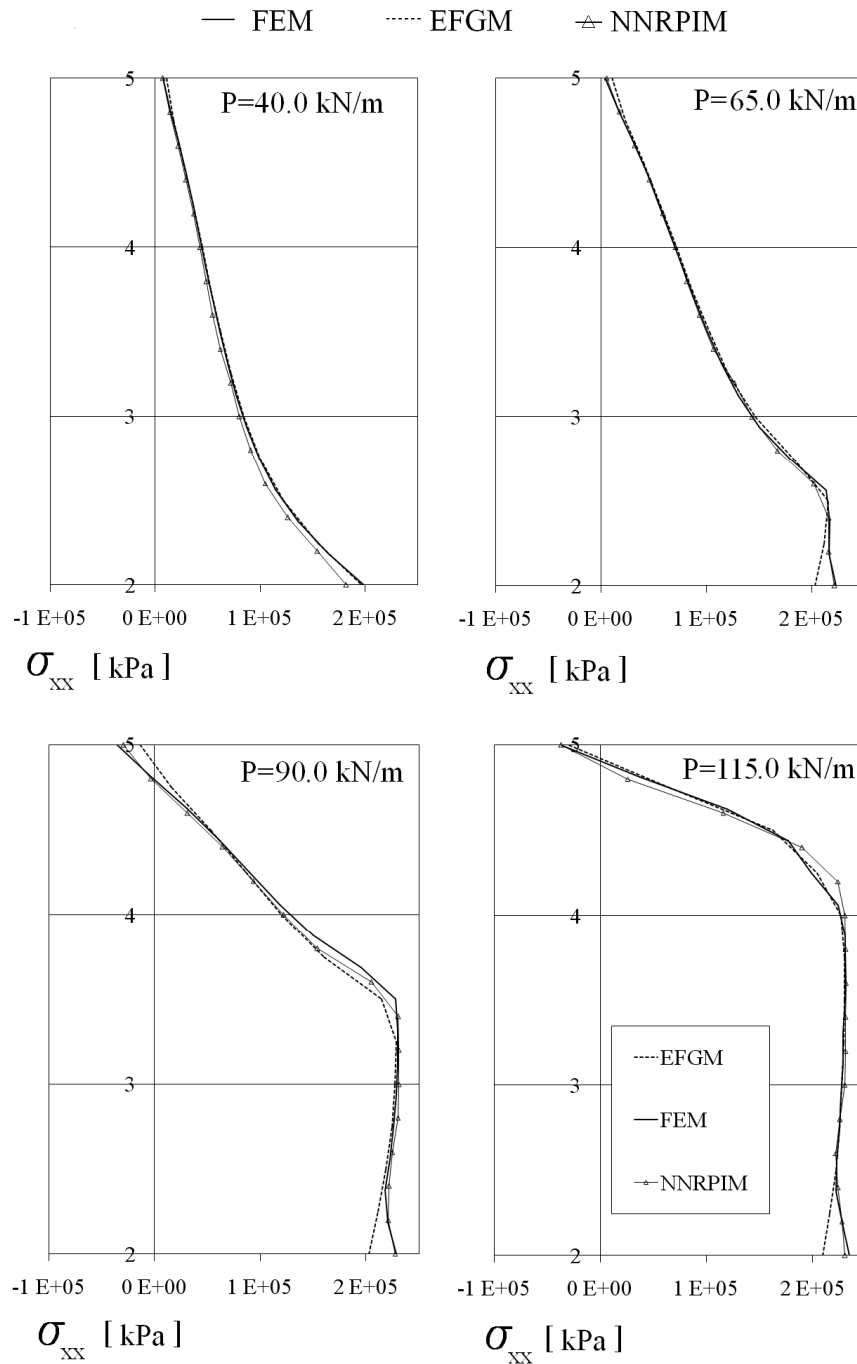


figure 107 – Normal stress distribution along essential boundary define by $\overline{AB}$ for increasing load levels.

As it is perceptible, the NNRPIM adjust well to the FEM and the EFGM solutions. In this example there is no perturbation in the essential boundary in the EFGM results. This is explained by the fact that the result of the EFGM is regarding a complete study of this problem, no symmetry was

considered. As so the line $\overline{AB}$ is not in fact an essential boundary in the EFGM analysis [41]. Notice that the NNRPIM stress field in the boundary is as smooth as the EFGM stress field in an inner zone of the domain.

5.1.4 Infinite Cylinder

An infinite thick pipe subjected to an internal pressure is studied. Due the symmetry and the infinite length of the problem only a quarter is analysed. The geometric characteristics are presented in figure 108(a). The material properties are: $E = 21 \text{ MPa}$, $E_T = 21 \text{ kPa}$, $\nu = 0.3$ and $\sigma_Y = 24 \text{ kPa}$. The problem is analysed considering the two-dimensional Plane Strain theory (NNRPIM-2D) and the three-dimensional Strain Theory (NNRPIM-3D). In the three-dimensional analysis is considered, along the zz axis, a thick of 50 mm and the restriction of displacement w in the faces at $z = 0 \text{ mm}$ and $z = 50 \text{ mm}$. The two-dimensional mesh used is presented in figure 108(b). In the three-dimensional analysis two different meshes are used, a regular mesh (RM) of 2205 nodes and a irregular mesh (IM) of the same node density, figure 108(c).

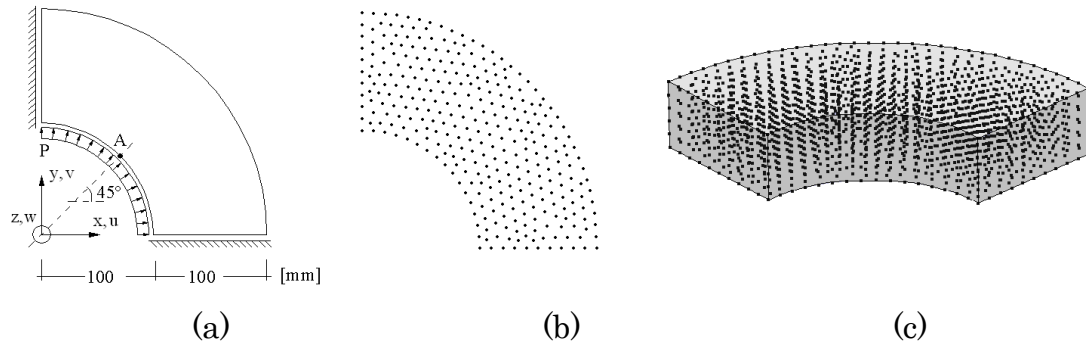


figure 108 – (a) Cylinder geometric characteristics. (b) two-dimensional mesh with 315 nodes and (c) three-dimensional regular mesh with 2205 nodes.

The displacement on point A, δ_A , for increasing loads is shown in figure 109. The NNRPIM results are compared with the FEM solution presented in [4].

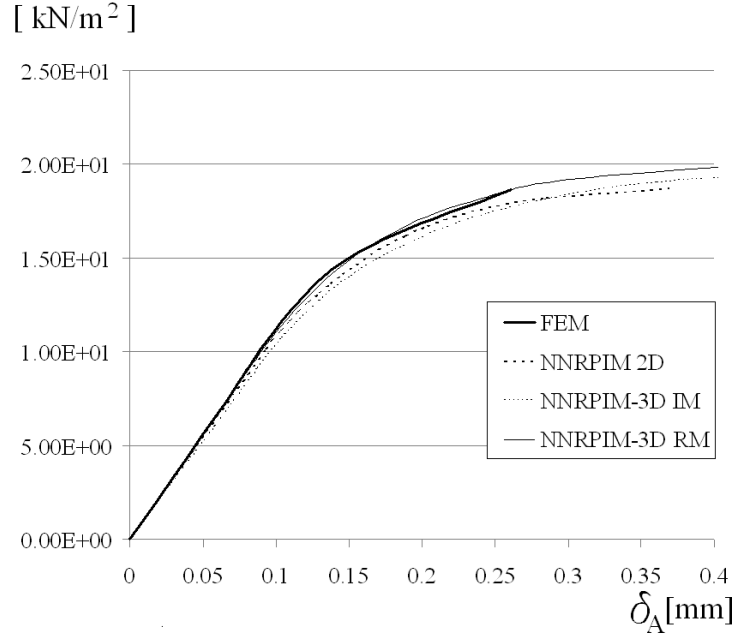


figure 109 – Load/displacement diagram.

It is visible the good correlation between all the NNRPIM formulations and the FEM solution. Once more there is not a significant difference between the results obtained with a regular mesh and the irregular mesh.

5.1.5 Block Under Concentrated Pressure

Consider the block under a centre concentrated pressure, as figure 110(a) indicates. Due to the symmetry only a quarter of the block is studied. The material properties are: $E = 210 \text{ MPa}$, $\nu = 0.3$, $\sigma_y = 250 \text{ kPa}$ and the hardening parameter $A = 250$. The problem is discretized in a $11 \times 11 \times 11$ nodes regular mesh, figure 110(b).

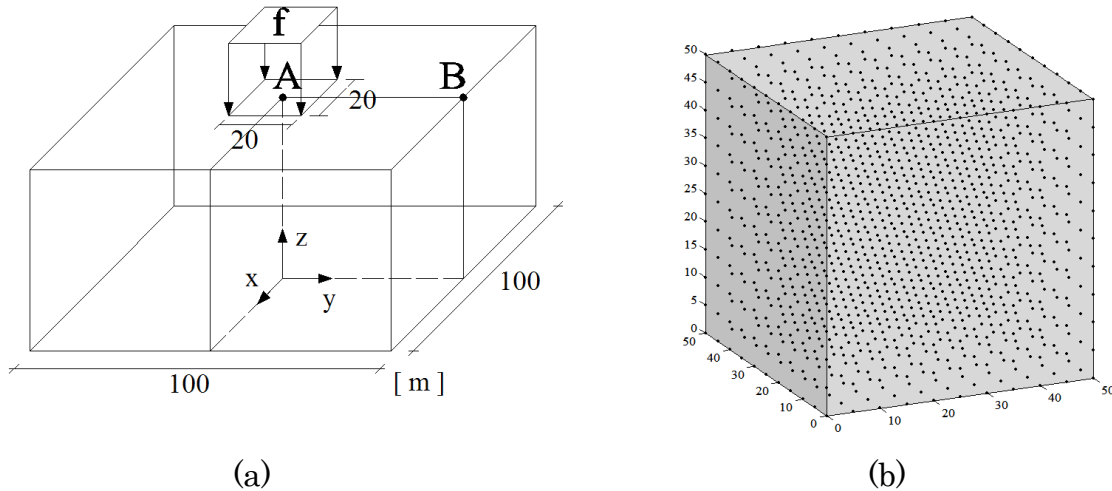


figure 110 – (a) Block under pressure. (b) Regular mesh of 1331 nodes.

The zz component of the displacement on point A, w_A , for increasing loads is shown in figure 111(a). It is perceptible the good correlation between the NNRPIM formulation and the FEM solution (8-nodes brick finite element) [88]. In figure 111(b) it is presented the evolution of the zz component of the displacement, w , on line $\overline{AB}$ for increasing load levels.

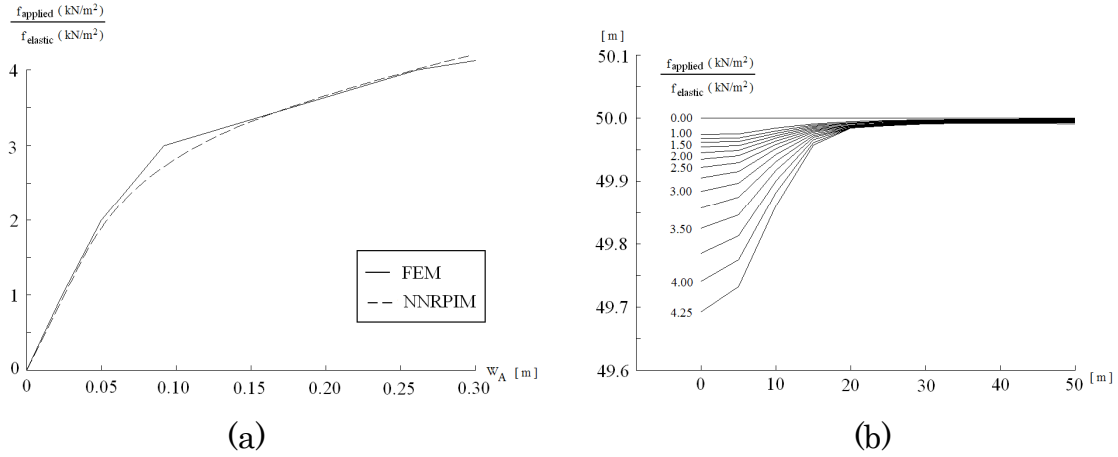


figure 111 – (a) Load/Displacement w_A diagram. (b) Load/displacement w diagram for line $\overline{AB}$.

The effective stress map for different load levels is shown in figure 112. It is visible that the stress distribution is smooth.

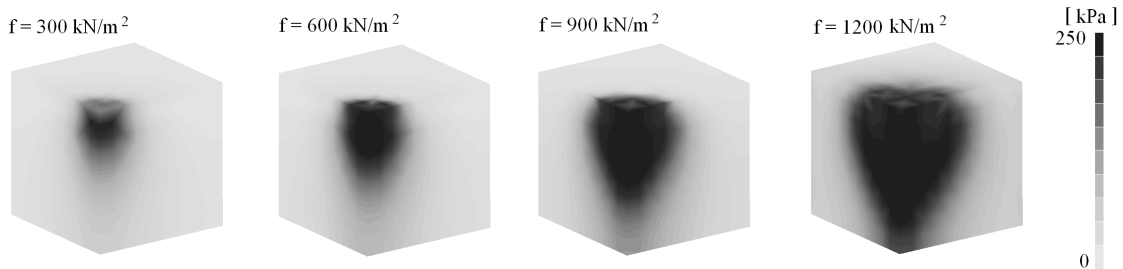


figure 112 – Effective stress maps.

5.1.6 Square Plate Under Transversal Load

A three-dimensional square plate under a distributed uniform transverse load is considered, figure 113(a). The material properties are: $E = 206 \text{ MPa}$, $E_T = 2.06 \text{ MPa}$, $\nu = 0.3$ and $\sigma_Y = 206 \text{ kPa}$. Due to the existing symmetry only one quarter of the plate is analysed. In this example two distinct cases are studied. The simple supported plate in all edges (SSSS) and the clamped plate in all edges (CCCC). In the SSSS NNRPIM analysis two different meshes are used, a three-dimensional regular mesh with three node levels (M2) and a three-dimensional regular mesh with five node levels (M4), figure 113(b).

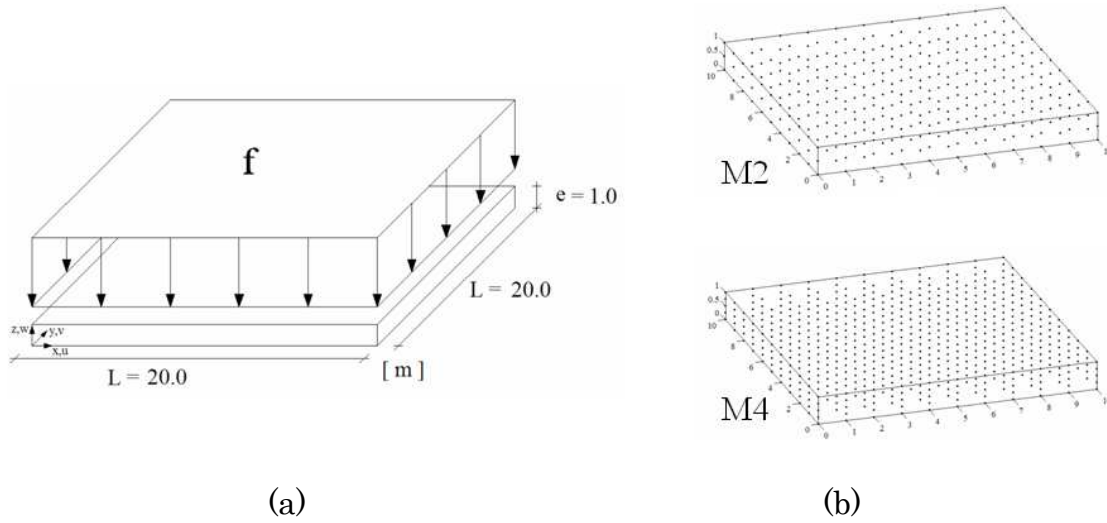


figure 113 – (a) Studied square plate. (b) Regular meshes with 363 nodes (M2) and regular mesh with 605 nodes (M4).

In figure 114(a) it is presented the SSSS load/central displacement w_c diagram. The results are compared with the FEM solution [44] for plates with the First Order Shear Deformation Theory (FOSDT). As it is visible by figure 114(a) the analysis that uses more nodes along the thickness of the plate (M4) produces a solution which approaches more the FEM solution. As so, only the M4 mesh is used in the CCCC analysis. In figure 114(b) it is presented the CCCC load/central displacement w_c diagram. As it is visible the NNRPIM formulation approaches the FEM solution for plates, with the FOSDT, proposed by Barbosa [89] and Figueiras [90].

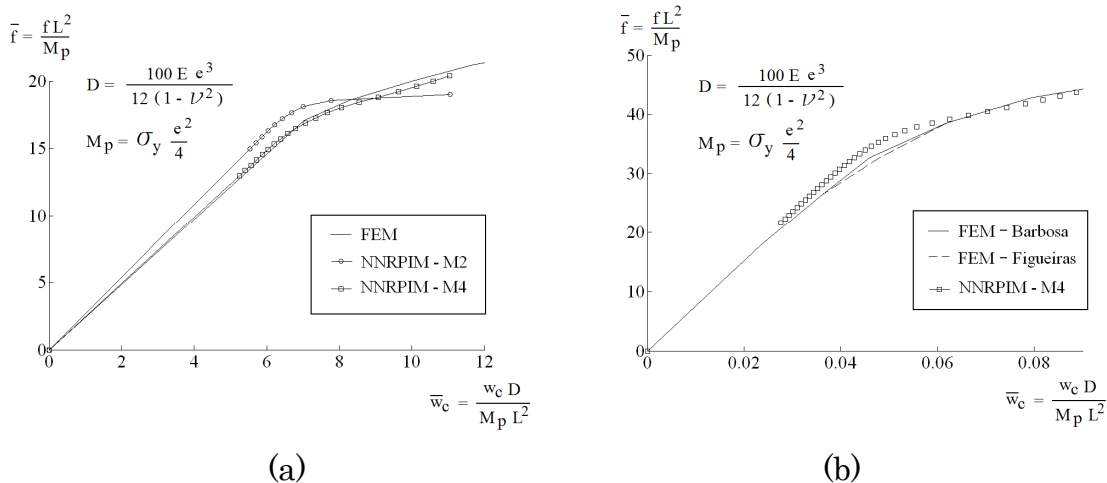


figure 114 – Normalized load/vertical central displacement w_c diagram for the (a) SSSS plate and the (b) CCCC plate.

In figure 115 are respectively presented the effective stress maps for different load levels for the SSSS plate and the CCCC plate.

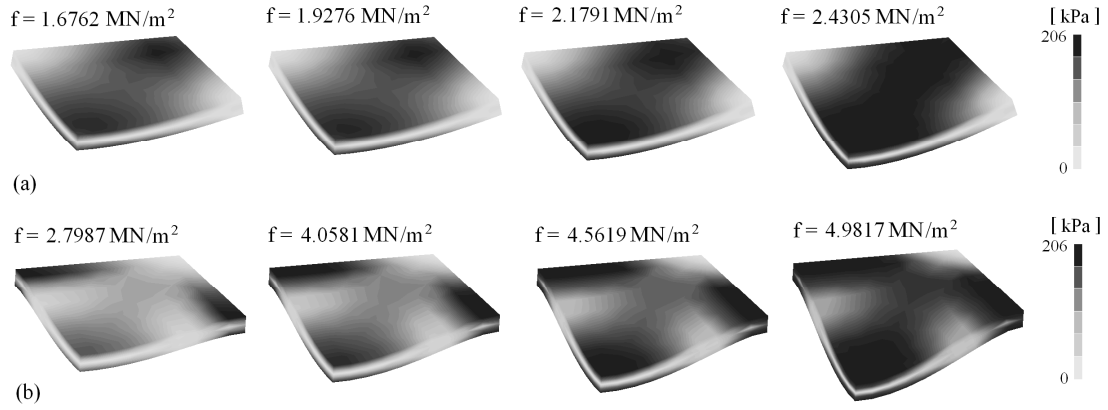


figure 115 – Effective stress map for (a) the SSSS plate and (b) the CCCC plate.

In both cases it is clear the formation of the characteristic plate rupture lines.

5.2 Large Deformation Analysis

In this section several examples are presented in order to show the good behaviour of the NNRPIM in large deformation analysis. Firstly the application of the orthogonal actualized Ramm's method to the NNRPIM is validated by comparison with the Newton-Raphson nonlinear solution method. Then the efficiency of the arch-length method applied to the NNRPIM is tested with a problem that shows structural instability. Afterwards the NNRPIM solutions for large deformation benchmark examples are presented. The NNRPIM solution is compared with analytical solution and FEM solutions, whenever available.

5.2.1 Clamped Thick Arch

In this example a thick arch is studied, clamped in both ends and subjected to a concentrated load in the mid span, as figure 116 shows. The material properties of the arch are $E = 4.8 \text{ MPa}$ and several values for Poisson ratios are considered, $\nu = \{0.00 \ 0.10 \ 0.20 \ 0.30 \ 0.40 \ 0.49\}$, these values are used along the analysis. Using the symmetry of the solid only half of the problem is studied. The problem is analysed using two distinct approaches, a 2D approach and a 3D approach. The nodal meshes used in the analysis are presented in figure 117. The NNRPIM solution is compared with the analytical solution presented in [91].

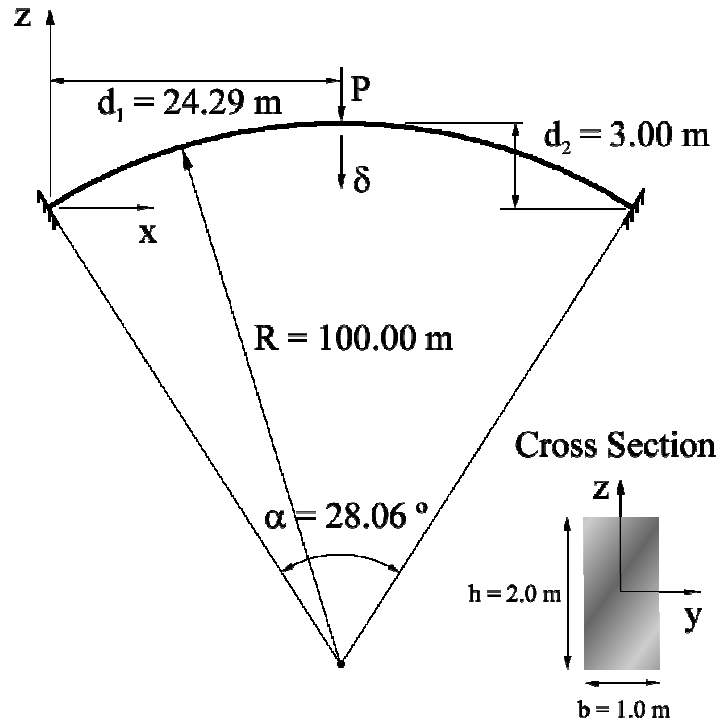


figure 116 – Geometrical model of proposed thick arch.

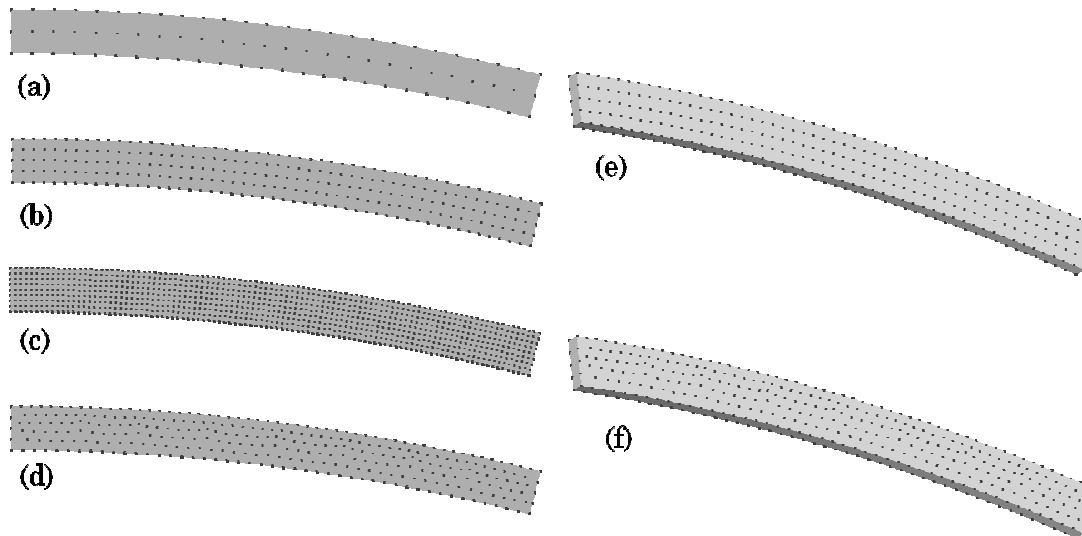


figure 117 – Meshes used in the analysis. (a) 2D regular 73 nodal mesh. (b) 2D regular 255 nodal mesh. (c) 2D regular 909 nodal mesh. (d) 2D irregular 303 nodal mesh. (e) 3D regular 510 nodal mesh. (f) 3D irregular 606 nodal mesh.

In order to study the influence of the nodal mesh discretization in the accuracy of the solution, it was considered the three regular meshes presented in figure 117(a), (b) and (c). The load applied versus the displacement along z direction in the mid-span of the arch is presented in figure 118. The nonlinear solution method used to solve the large deformation problem is the initial stiffness Newton-Raphson method.

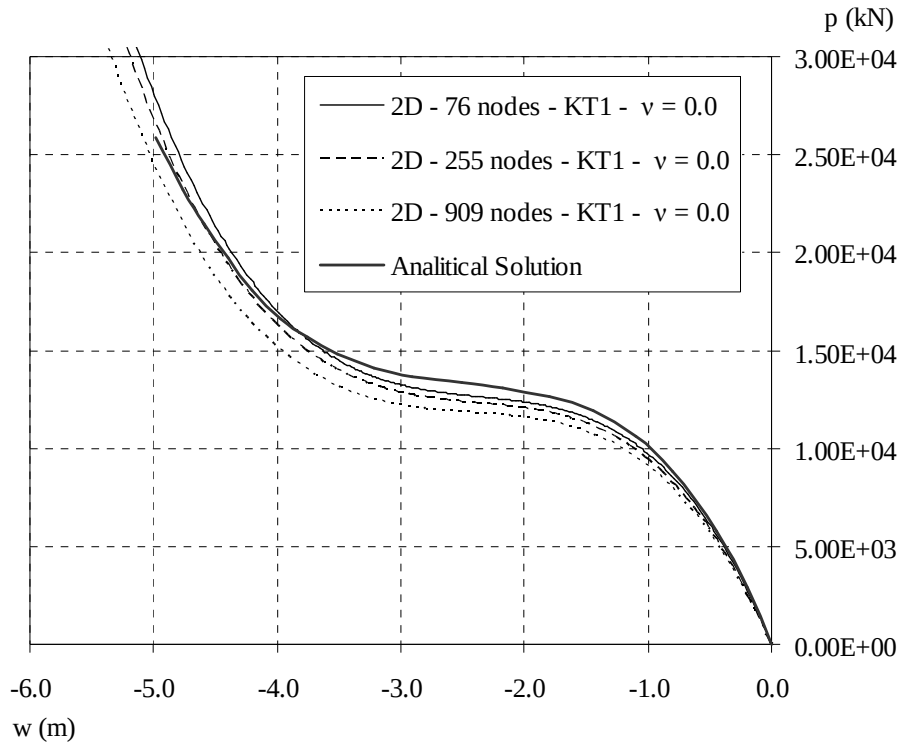


figure 118 – Evolution of the displacement on the mid span of the arch for distinct meshes.

As figure 118 indicates the solution converges. It also shows that there is a good concordance between the analytical solution and the 2D solution obtained with the NNRPIM.

The numerical implementation of the nonlinear solution method (KT1) was successfully applied to the NNRPIM [39], thus it is important to validate the orthogonal actualized Ramm's method, or arch-length method, comparing the results obtained with the KT1 with the results obtained with the arch-length method (ArcM). In figure 119 both nonlinear solution methods are compared. As it is visible on figure 119, both nonlinear solution methods produce coincident results, as it should be. As so, from this example forward, in this work all the next examples are solved using the orthogonal actualized Ramm's method.

The same problem is now considered but the Poisson ratio is varied between 0.00 and 0.49. The mesh used in the analysis is the 255 nodes regular mesh. The load applied versus the displacement along z direction in the mid-span of the arch is presented in figure 120. It is visible that the Poisson ratio does not significantly influence the obtained solution.

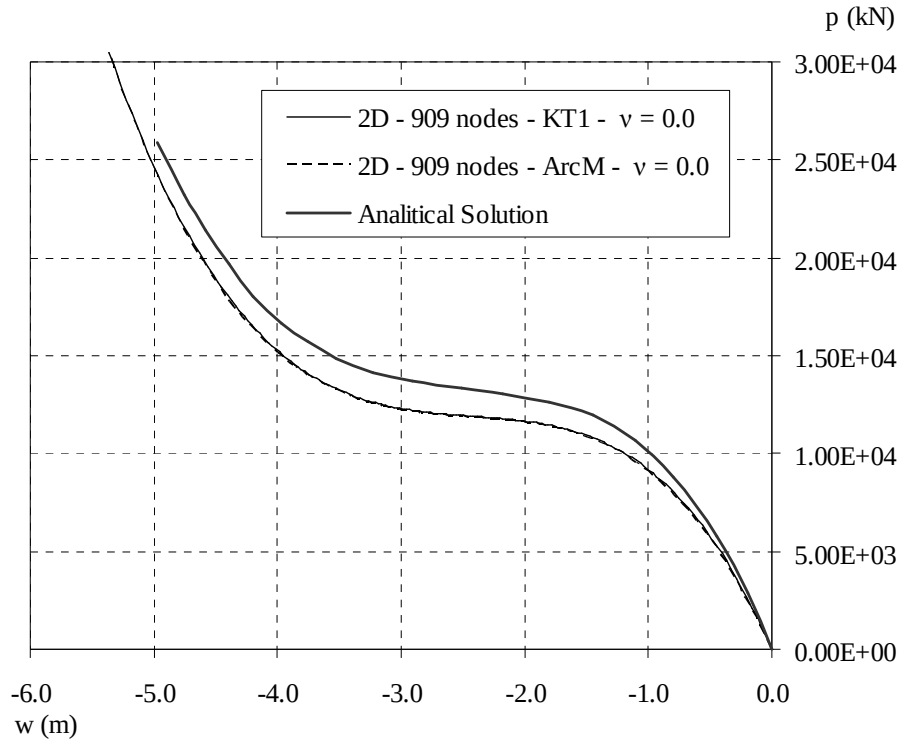


figure 119 – Evolution of the displacement on the mid span of the arch when two distinct nonlinear solution methods are used.

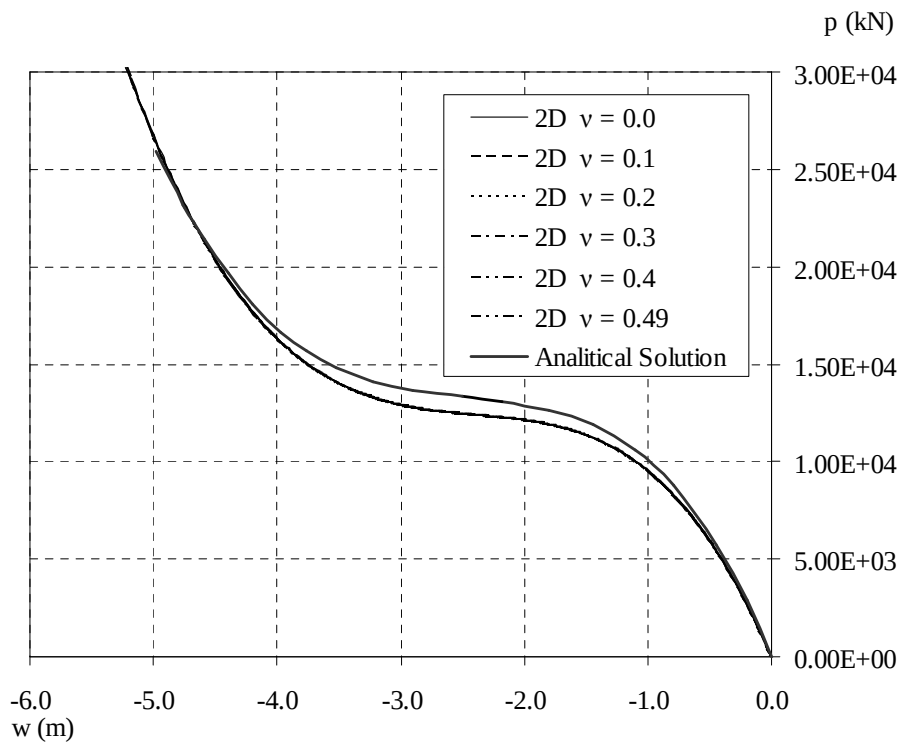


figure 120 – Evolution of the displacement on the mid span of the arch for several values of the Poisson ratio.

The NNRPIM 3D formulation is now applied to solve the problem. The 3D regular and irregular meshes presented respectively in figure 117(e) and (f) are used in the analysis. A 2D irregular mesh of 303 nodes, figure 117 (d), is also used in the analysis. The results are compared with the 2D solution (909 nodes regular mesh) and the analytical solution. The results are presented in figure 121 and it is visible the good concordance between the 3D solution and the analytical solution, regardless if it is a 3D regular mesh or a 3D irregular mesh.

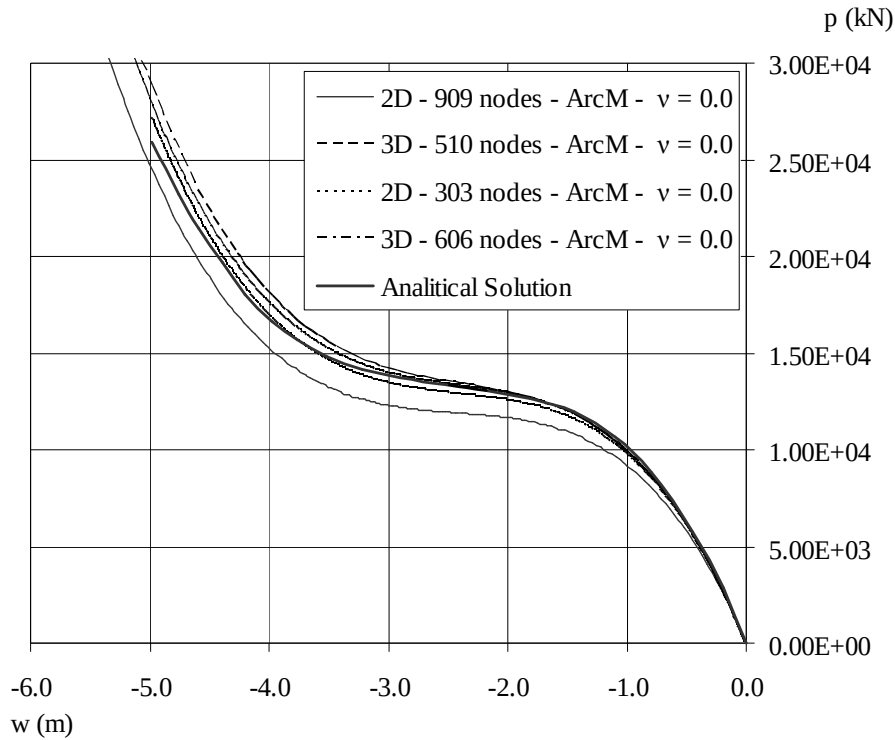


figure 121 – Evolution of the displacement on the mid span of the arch obtained with a 3D formulation and with irregular meshes.

Notice from figure 121 that the irregularity of the mesh does not disturb the method performance or accuracy.

5.2.2 Simply Supported Shallow Arch

A simply supported shallow arch with snap-through buckling behaviour was analyzed. The arch is loaded by a point load at the top centre and it is simply supported in both ends in the centre of the section. The geometry and dimensions are shown in figure 122. The material properties of the arch are $E = 700\text{ MPa}$ and $\nu = 0.0$. Using the symmetry of the solid only half of the problem is studied. The results are compared with a FEM solution for a 1D analysis using the Timoshenko beam element solution [92].

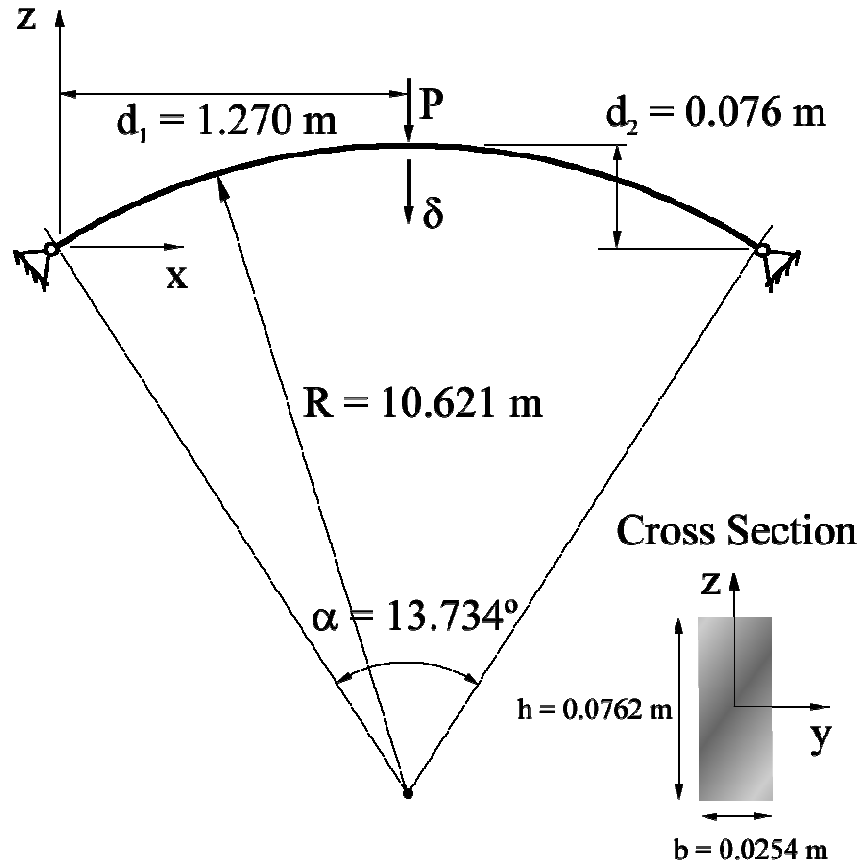


figure 122 – Geometrical model of the proposed shallow arch.

The problem is solved considering the 2D and the 3D approach. The regular and irregular meshes, 2D and 3D, used in the analysis are shown in figure 123.

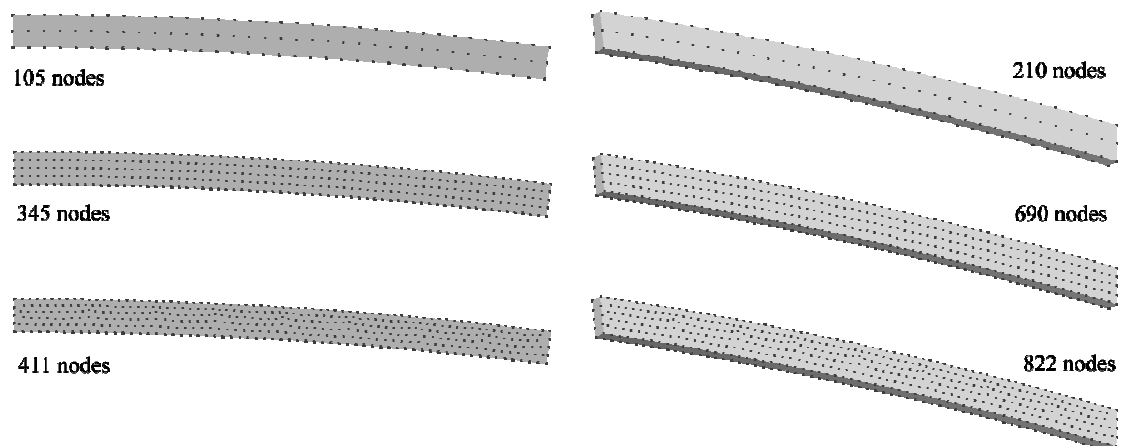


figure 123 – Meshes used in the analysis.

The vertical displacement on the mid span of the arch in relation to the applied load is present in figure 124.

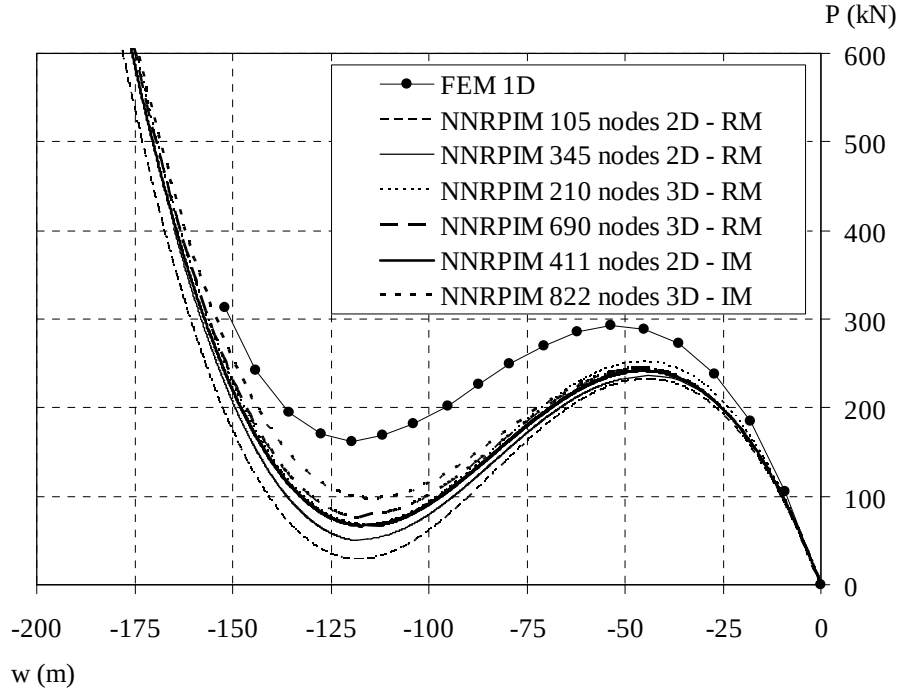


figure 124 – Displacement on the mid span of the arch.

It is visible in figure 124 that the orthogonal actualized Ramm's method algorithm predicts well the snap-through buckling phenomenon. The NNRPIM 2D and 3D solutions are all very close to each other, regardless if the used mesh is regular or irregular. The NNRPIM solution is not very close with the FEM solution, probably because the FEM solution is regarding a 1D approach and the NNRPIM solution are for 2D and 3D analyses.

5.2.3 Pure Bending of a Cantilever Beam

Described in [93] and solved numerically in several papers [94, 95], this classical geometrically nonlinear solid mechanics problem consists of an elastic beam subjected to a bending moment at the tip end and clamped at the other end, figure 125. The intensity of the applied moment is gradually increased until a perfect cylinder configuration is reached. The dimensions used are length $L=10.0\text{ m}$, width $B=1.0\text{ m}$ and thickness $H=0.1\text{ m}$. The Young modulus and the Poisson ratio considered are $E=12\text{ MPa}$ and $\nu=0.0$.

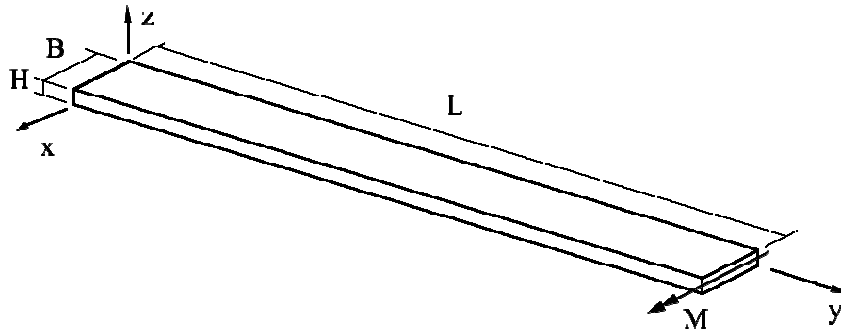


figure 125 – Geometrical model of the roll-beam proposed.

This problem was solved considering two distinct approaches, a 2D and a 3D approach. The 2D analysis uses a regular mesh of 3×101 nodes and the 3D analysis considers a regular mesh of 126 nodes, as it is shown in figure 126.

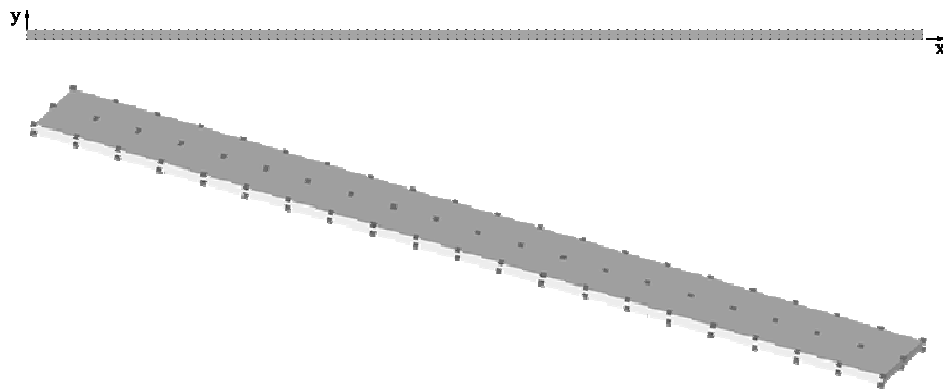


figure 126 – 2D and 3D meshes used in the analysis.

The numerical results obtained with the NNRPIM are presented in figure 127 for the 2D analysis and in figure 128 for the 3D analysis.

Deformed meshes are presented for increasing tip moments and it is visible that the final deformed mesh is representing a nearly perfected circular ring, both for the 2D and the 3D analysis.

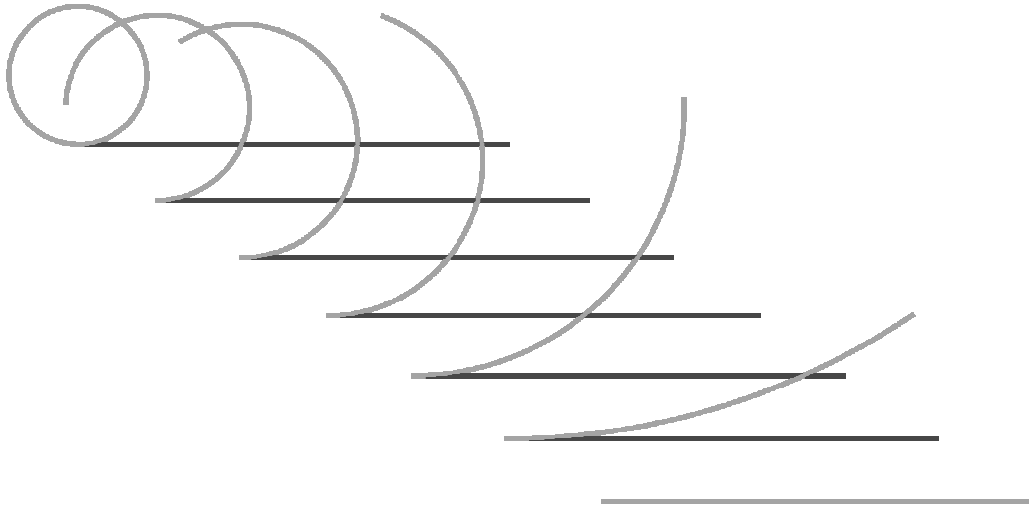


figure 127 – Deformed meshes for increasing tip moments for the 2D analysis.

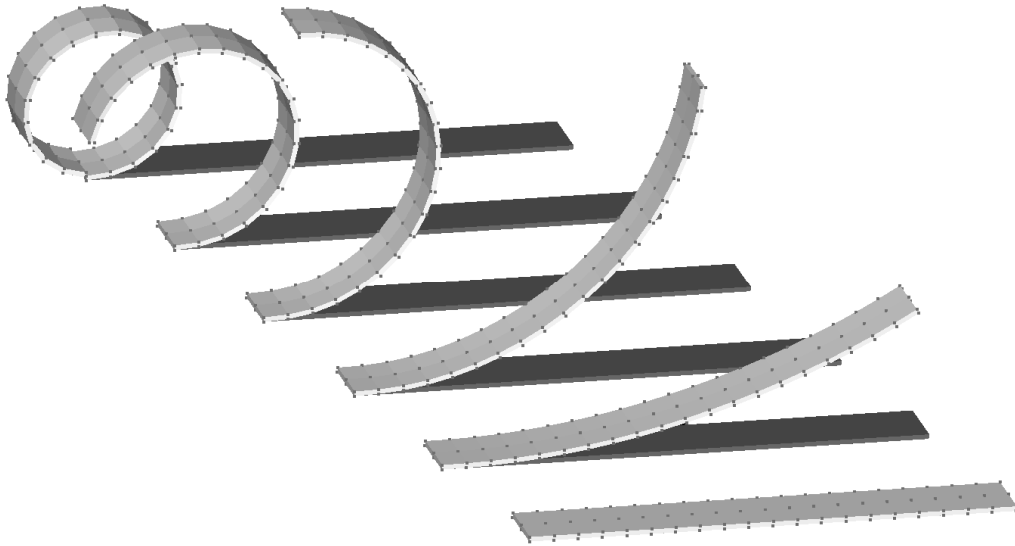


figure 128 – Deformed meshes for increasing tip moments for the 3D analysis.

5.2.4 Cantilever Beam

Consider now an elastic cantilever beam with the same length $L = 10.0\text{ m}$ but with a cross section $1.0 \times 1.0\text{ m}^2$, the material properties are $E = 10\text{ MPa}$ and $\nu = 0.3$. The load f is applied uniformly distributed at $x = L$ and the beam is clamped at $x = 0$, as figure 129 indicates. In this example two meshes for the 2D analysis and other two meshes for the 3D analysis are considered, the meshes and the number of nodes of each one are presented in figure 129.

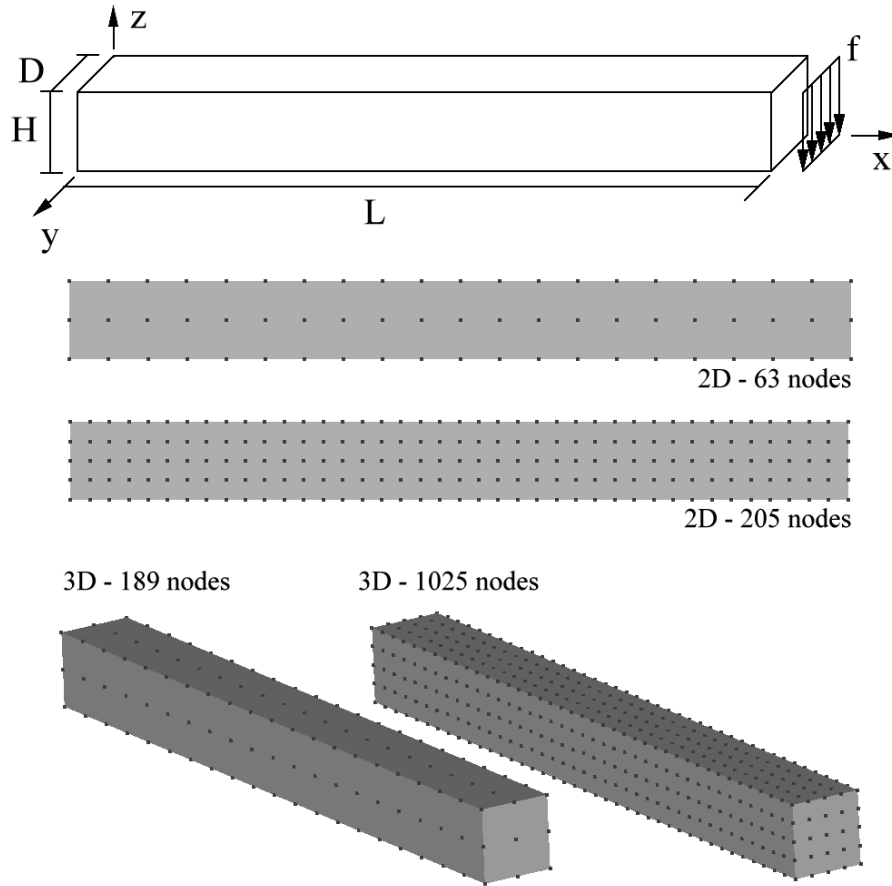


figure 129 – Geometrical model of the cantilever beam proposed and meshes considered in the analysis.

Considering a large deformation analysis the load is applied gradually. The results obtained with the NNRPIM, for the 2D and the 3D analysis, are compared with a theoretical solution which relies on inextensional elastica [96]. The point A, $A=[10 \ 0,5 \ 0,5]$, displacement in the z direction is presented in figure 130 for the several meshes considered

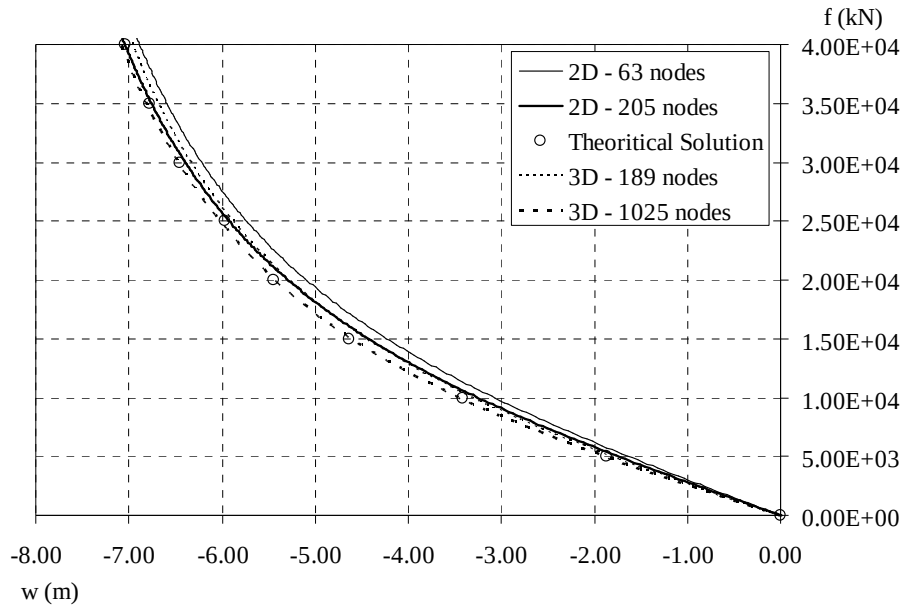


figure 130 – Evolution of the displacement on point A with the load level for distinct meshes, for the 2D and the 3D analysis.

As it is visible on figure 130 when the mesh is refined the solution approaches the theoretical solution, for the NNRPIM-2D analysis and the NNRPIM-3D analysis. Both 2D solution and 3D solution are very near the theoretical solution. In figure 131 and figure 132 are presented respectively the 2D and the 3D deformation of the cantilever beam for several increasing load levels.

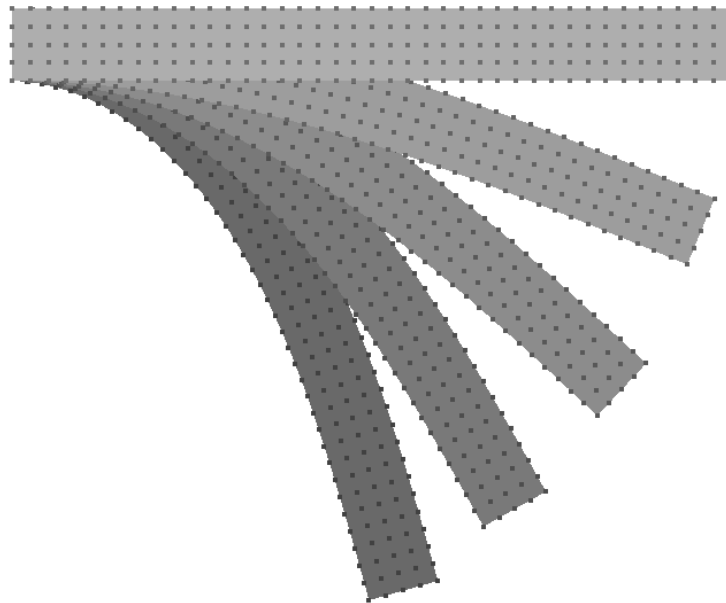


figure 131 – Deformation of the cantilever beam for several increasing load levels for the 2D analysis.

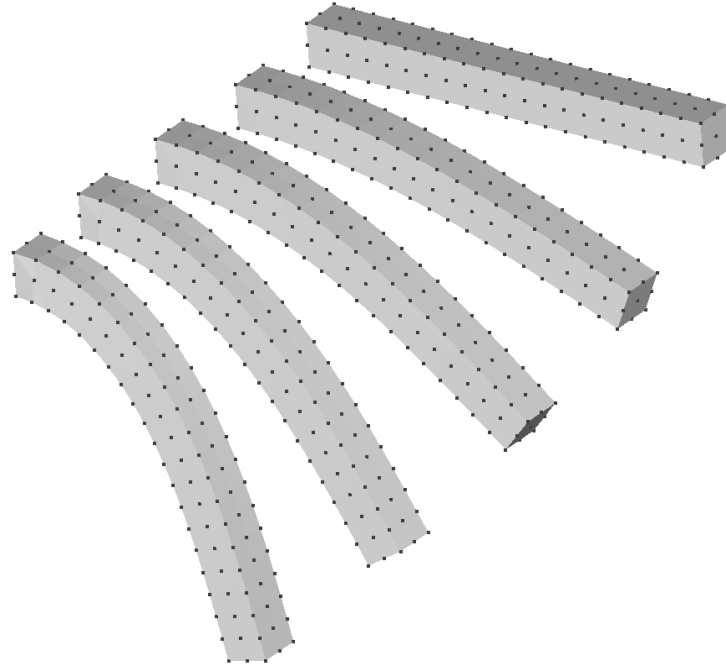


figure 132 – Deformation of the cantilever beam for several increasing load levels for the 3D analysis.

As it is visible, the displacement is smooth and there is no distortion on the mesh for largest deformations.

5.2.5 Square Plate Under Transversal Load

In this example is considered a square plate subjected to a uniformly distributed load, as figure 133 indicates, where $L = D = 10.0m$ and $H = 0.5m$.

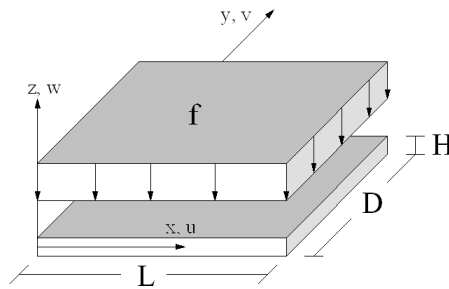


figure 133 – Geometrical model of the plate and applied load.

Two cases for the boundary conditions are considered in this study. The first case is a simply supported plate in all edges for $z = 0$. Using the natural symmetry of the solid only a quarter of the plate is analysed. The considered material properties are, $E = 1000kPa$ and $\nu = 0.316$. The bending of the plate is analysed using the NNRPIM 3D formulation and the results are compared with the analytical solution presented in [97]. The quarter of the plate is discretized in two distinct meshes, figure 134. A regular mesh with

363 nodes and in an irregular mesh with 474 nodes are used with the 3D NNRPIM formulation, considering large deformations analysis.

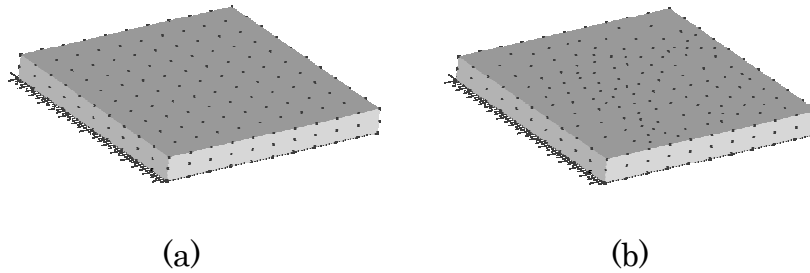


figure 134 – (a)Regular mesh with 363 nodes and (d) Irregular mesh with 474 nodes.

In figure 135 it is represented the displacement along the z direction on the centre of the plate for increasing load levels. The NNRPIM results are close to the analytical solution, regardless if a regular or an irregular mesh is used in the analysis.

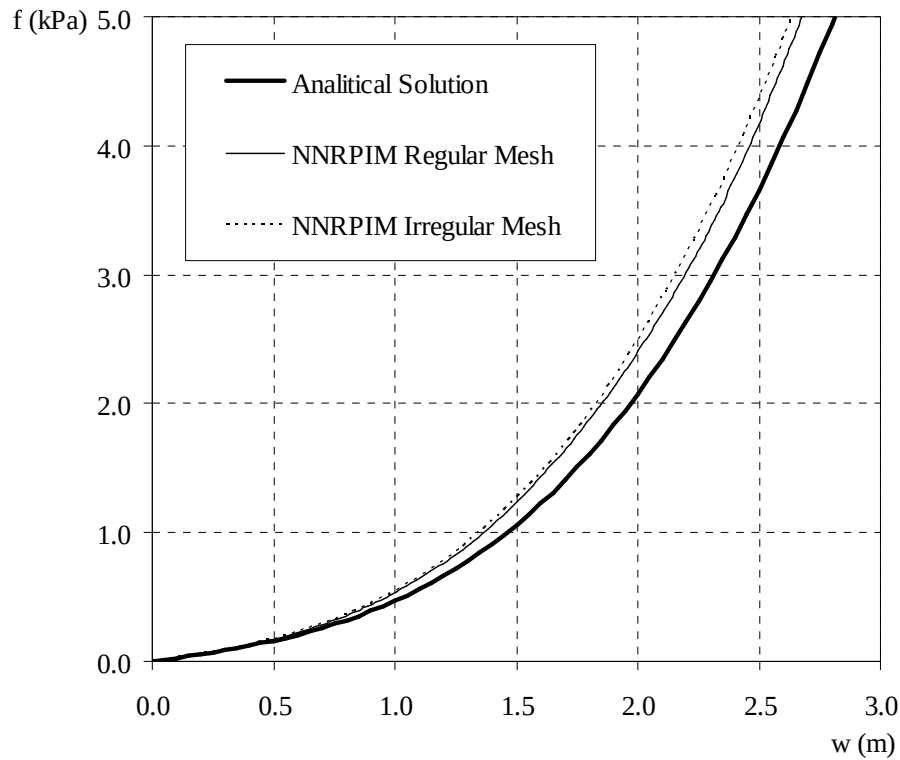


figure 135 – Evolution of the central displacement of the simply supported plate with the load level, for the 3D analysis.

Another case for the boundary conditions was studied. A square plate clamped in all edges with the same material and geometric properties as the previous one was analysed. Also the two distinct meshes presented in figure 134 are used, the 363 node regular mesh and the 474 node irregular mesh.

The results obtained applying the NNRPIM 3D formulation and using large deformation analysis are compared with the analytical solution presented in [97]. The displacement along the z direction on the centre of the plate for increasing load levels is presented in figure 136.

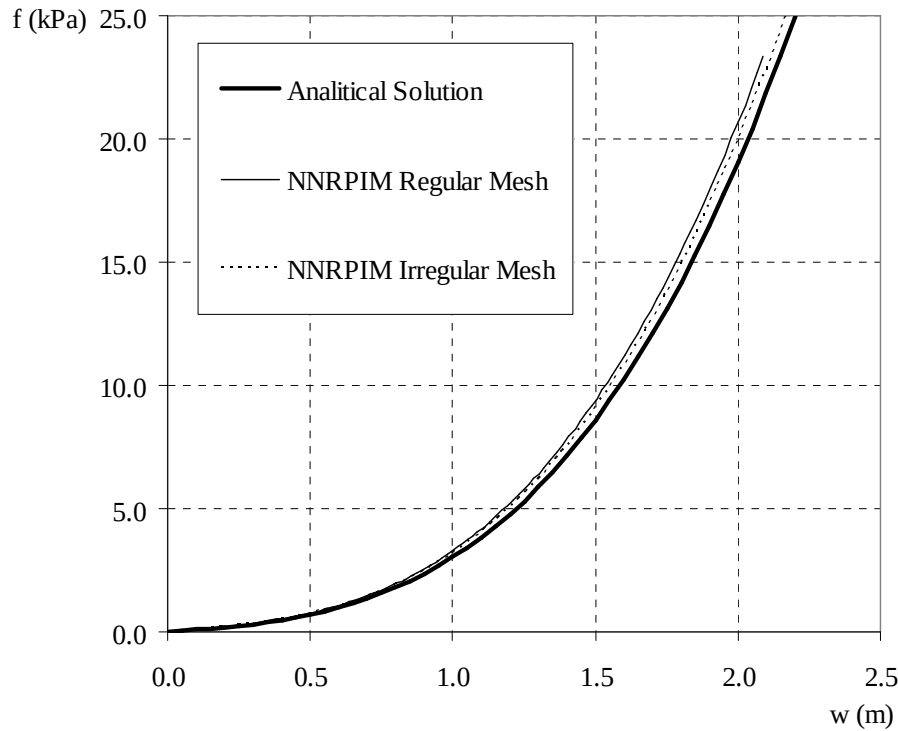


figure 136 – Evolution of the central displacement of the clamped plate with the load level, for the 3D analysis.

It is perceptible that the NNRPIM solution is almost coincident with the analytical solution. In figure 137 it is presented the plates deformation for distinct load levels and it is perceptible the good behaviour of the NNRPIM for high levels of the applied load.

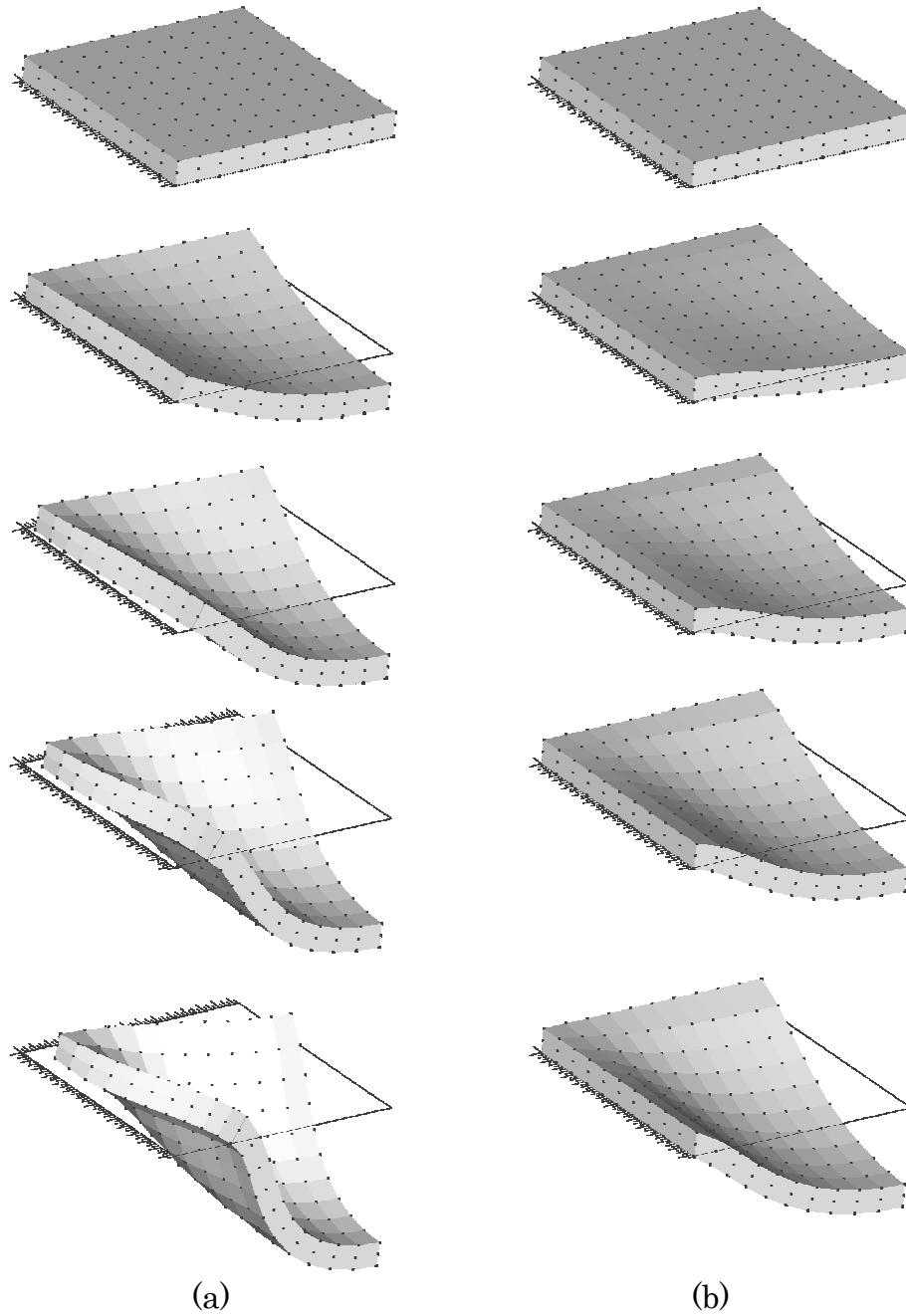


figure 137 – Deformation of the clamped square plate for several increasing load levels. (a) Simply supported plate. (b) Clamped plate.

The 3D displacement field obtained in the analysis, for distinct load levels, is always smooth and accurate.

5.2.6 Curved Cantilever Beam

The following example shows the applicability of the NNRPIM for thin 3D-beams. The large displacement response of a three-dimensional curvilinear cantilever beam subject to a concentrated end load is calculated [98]. Due to the geometry and the applied load the curved cantilever beam undertakes

both bending and twisting deformations. The beam lies in the Oxy plane and its cross-sectional area is $0.01 \times 0.01 \text{ m}^2$. The cantilever beam has a radius of $R=1.00 \text{ m}$ and an angle on the Oxy plane of 45° , as figure 138 indicates.

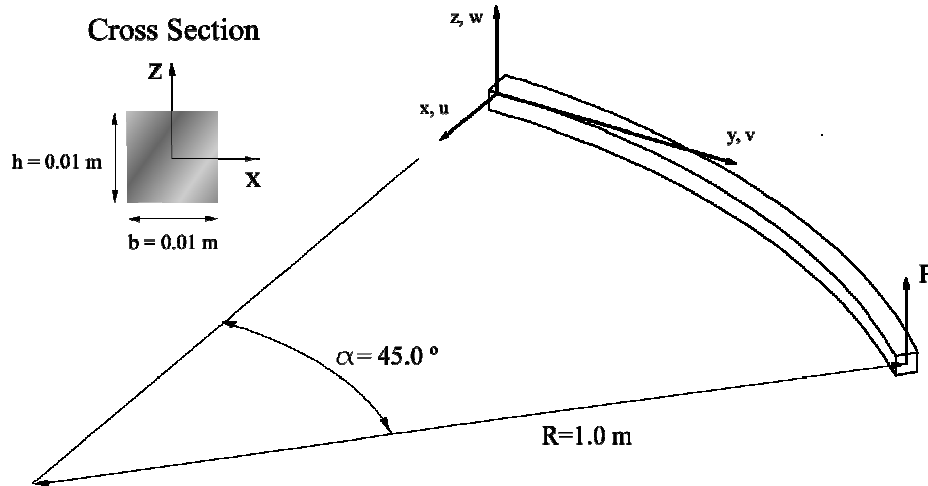


figure 138 – Geometrical model of the curved cantilever beam and applied load.

The concentrated load is applied at one end of the beam and the opposite end is kept fixed. The analysis is carried out using a Young's modulus of $E = 10 \text{ MPa}$ and a Poisson ratio of $\nu = 0.0$. Several three-dimensional meshes are used in the analysis, as figure 139 shows. The NNRPIM results are compared with an optimized FEM solution [98].

The NNRPIM results for the end tip of the curve beam, regarding the three variables of the displacement, are presented from figure 140 to figure 142. The applied load is normalized with the expression $P_{norm} = P \times R^2 / (E \times I)$ being I the inertia of the beam cross section. As it is visible, there is a path of convergence for the NNRPIM. With the refinement of the mesh the NNRPIM solution approaches the reference FEM solution. Using mesh 5, the most refined mesh, the results approach even more the FEM solution.

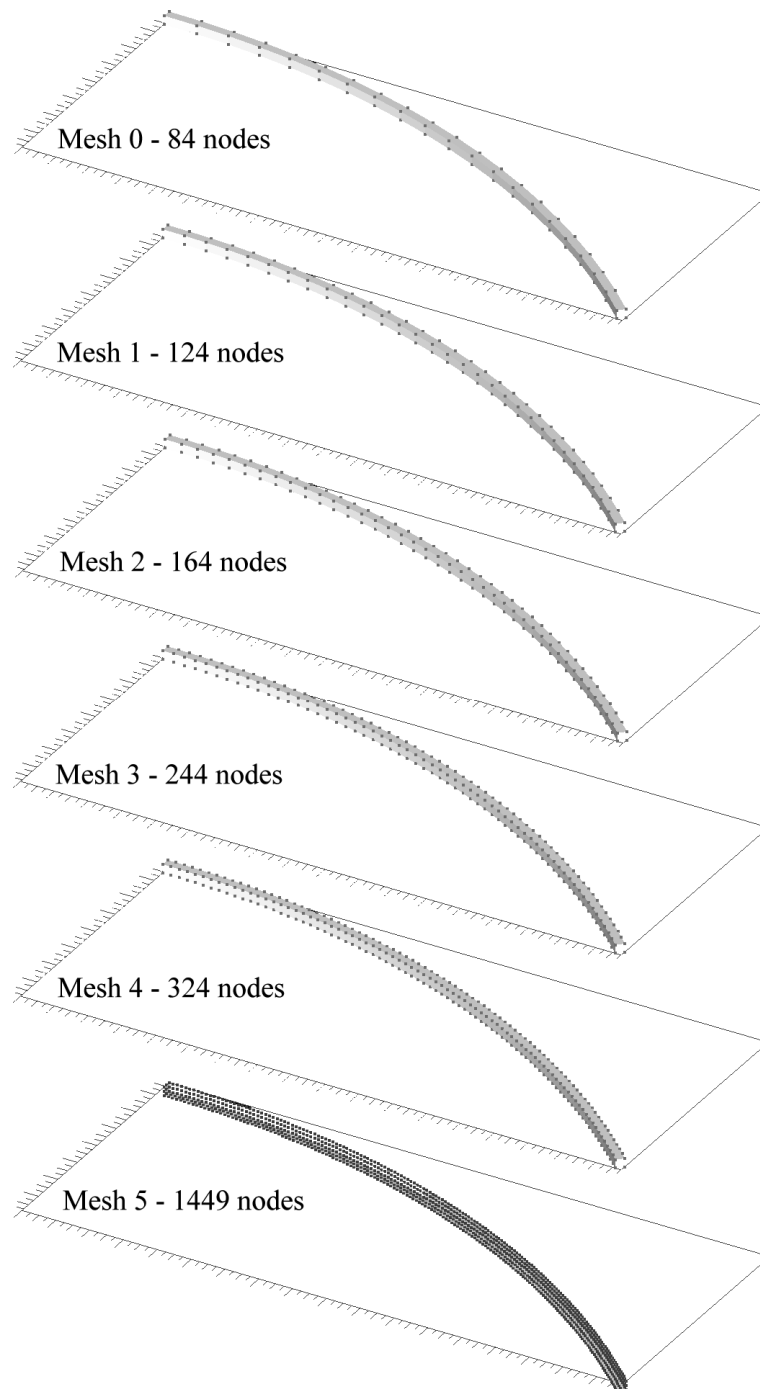


figure 139 – Meshes used in the analysis.

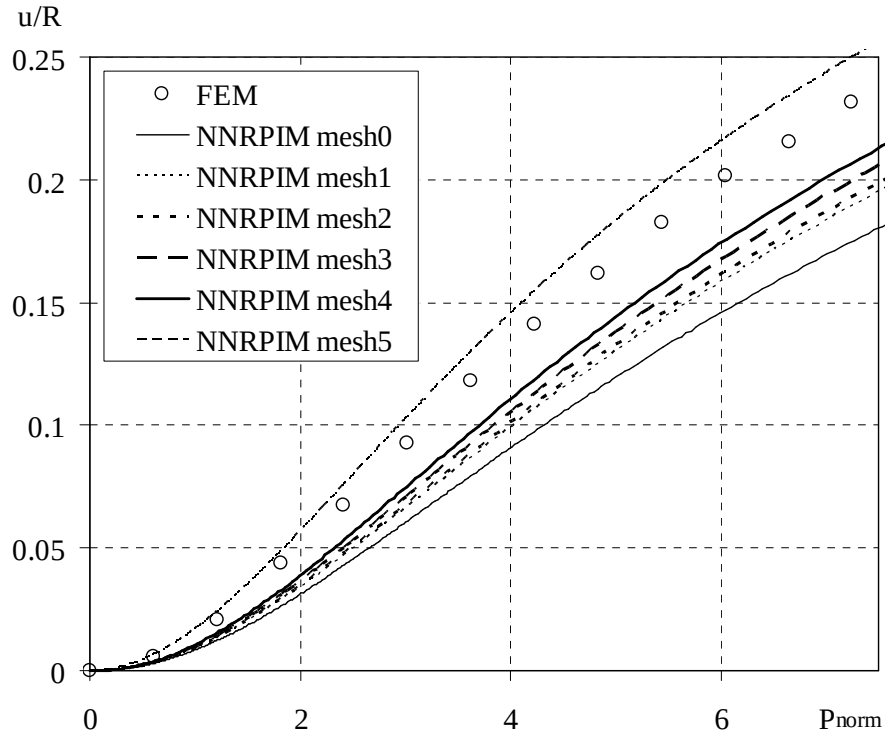


figure 140 – Evolution of the displacement along x direction on the end tip of the curved cantilever beam with the load level, for distinct meshes.

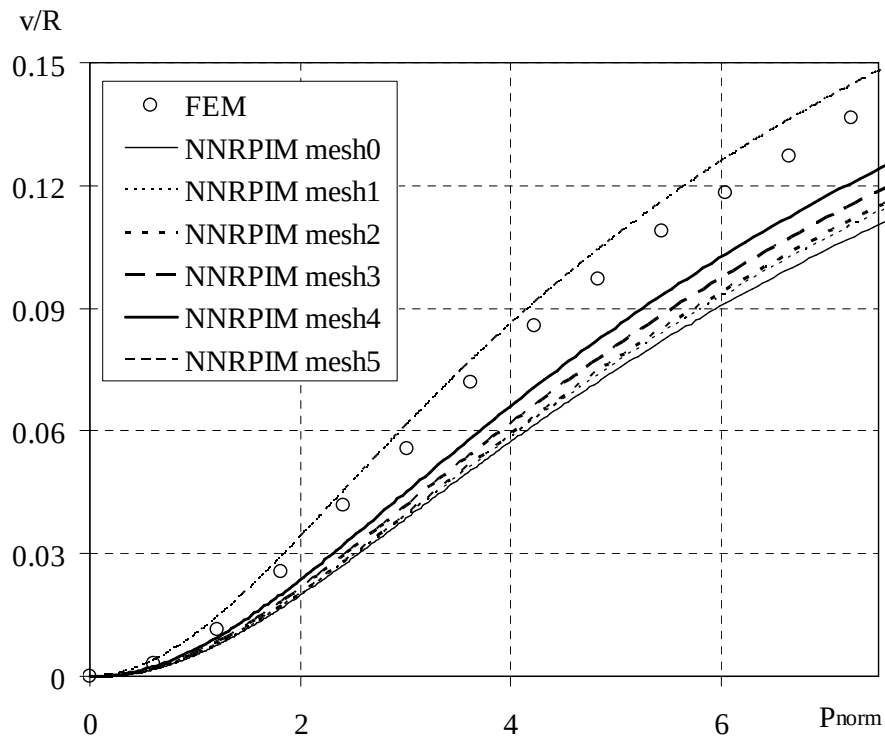


figure 141 – Evolution of the displacement along y direction on the end tip of the curved cantilever beam with the load level, for distinct meshes.

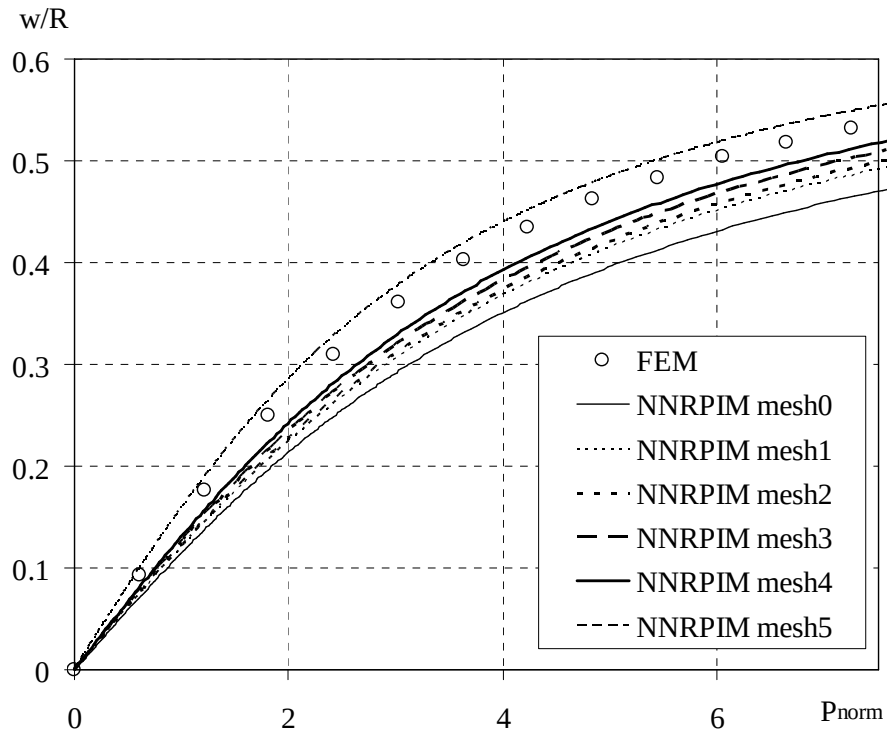


figure 142 – Evolution of the displacement along z direction on the end tip of the curved cantilever beam with the load level, for distinct meshes.

In figure 143 it is presented the deformation of the curved beam for distinct load levels and it is perceptible the good behaviour of the NNRPIM for high levels of the applied load.

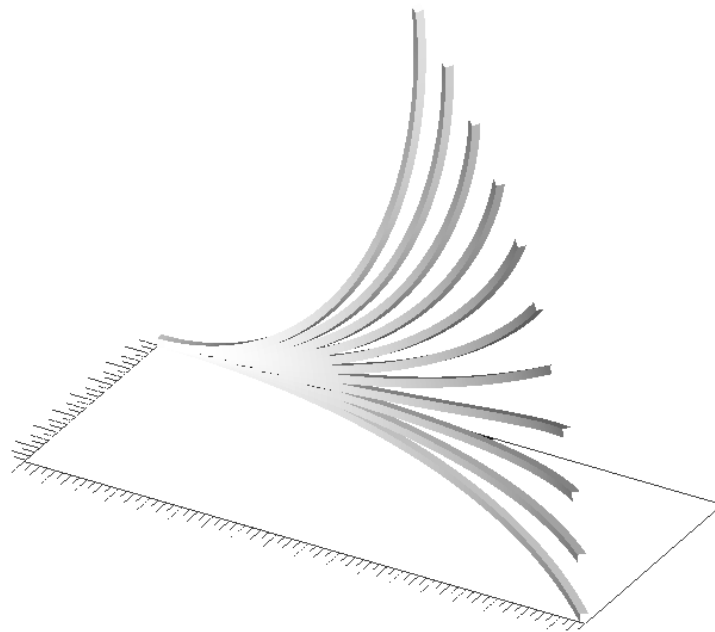


figure 143 – Deformation of the curved beam for several increasing load levels.

5.3 Geometric and Material Nonlinearity

The three-dimensional nonlinear analysis of structures has captured the interest of many researchers during the past decades. The frame dome is a benchmark example that permits to verify if all the nonlinear algorithms applied to the NNRPIM were effectively implemented.

Several works have studied the elastic and plastic mechanical behaviour of framed domes [99-102] using the FEM. The nonlinear mechanical behavior of the framed dome shown in figure 144 is studied with the objective of validating the proposed meshless formulation and the applied nonlinear algorithms. The linear elastic properties of the material are: $E = 20700 \text{ MPa}$ and $\nu = 0.17$. For the elastoplastic analysis with large strain it is considered a perfectly plastic behavior and a yield stress $\sigma_y = 80 \text{ Pa}$. The problem was analysed with two distinct meshes; a more coarse mesh with a 3×4 nodes per beam section (Mesh 1) and a more fine mesh with 5×5 nodes per beam section (Mesh 2), figure 144. Due the problem symmetry only one quarter of the model was analysed, considering the required displacement constraints in the structure symmetry surfaces.

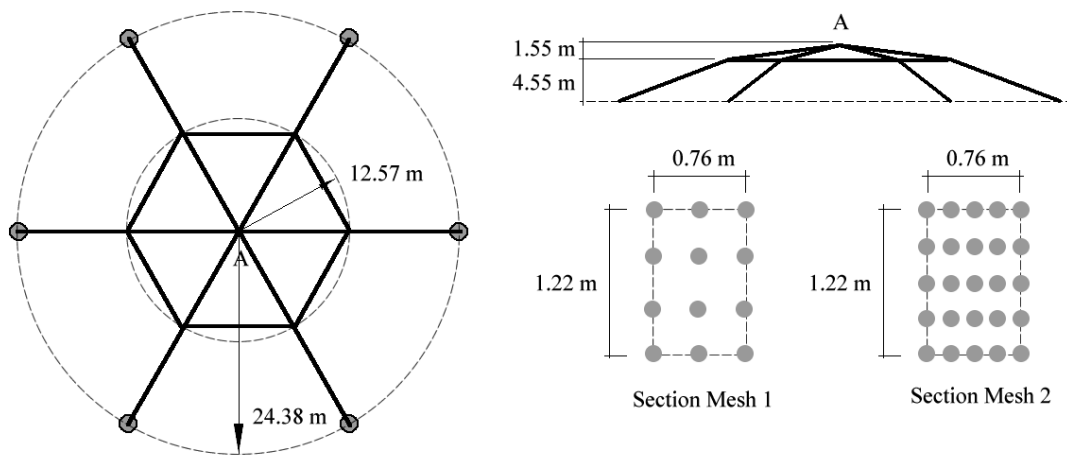


figure 144 – Frame dome geometric properties and section meshes used in the analysis.

In figure 145 it is possible to visualize the constructed meshes using the 3×4 nodal discretization per beam section (Mesh1) and the 5×5 nodal discretization per beam section (Mesh 2). In figure 146 it is presented the example of a complete mesh (not used in the analysis) just for a global understanding of the analysed problem.

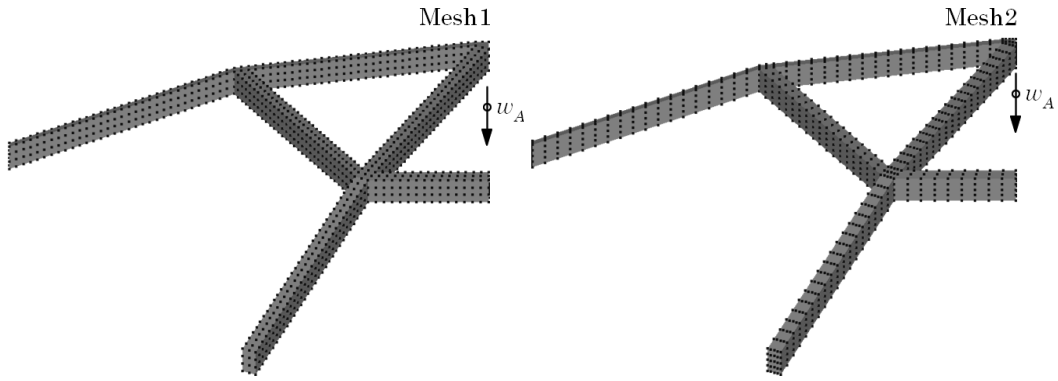


figure 145 – 3D meshes used in the analysis.

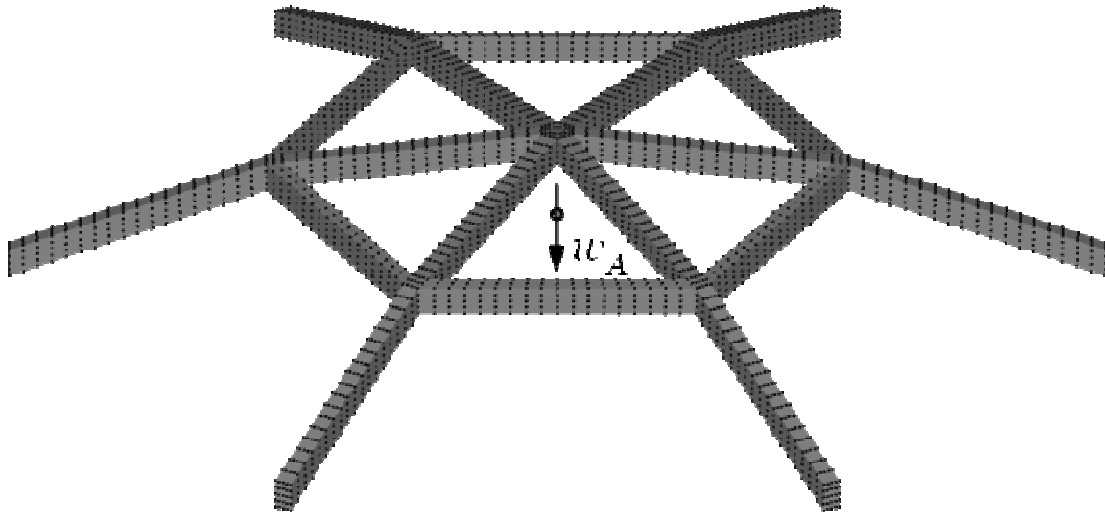


figure 146 – Global 3D mesh based on Mesh2.

In figure 147 it is presented the vertical displacement on point A, w_A , versus the applied load for the several cases considered in the analysis. The problem was analysed considering always large deformations, however the material behaviour was consider as elastic in a first instance and elastoplastic in a following study. As so, NNRPIM-LD acronym stands for the large deformations elastic analysis and the acronym NNRPIM-EP+LD stands for the large deformations elastic-plastic analysis.

In Park work [99] it was observed that the refinement of the mesh influences the result. The “FEM-LD sparse section mesh” results were obtained with a mesh similar with Mesh1 used in this work. The “FEM-LD dense section mesh” results were obtained with a 17×17 nodal discretization per beam section, [99]. With the NNRPIM the same phenomenon is observed.

It is clear that the 5×5 nodal discretization per beam section is not enough to obtain sufficient accurate results, however due to computational limitation it was not possible to evolve to denser meshes.

The NNRPIM results for the elastoplastic analysis with large deformations (NNRPIM-EP+LD) also show that the density of the used mesh influences the result. Nevertheless it is visible that the NNRPIM results corroborate the FEM solution [102]. In both mesh cases, when inelastic constitutive relations are employed, the curve of the global structural response shows a snap-through which couples constitutive and geometric effects, and it is in accordance with the results available in the literature [102].

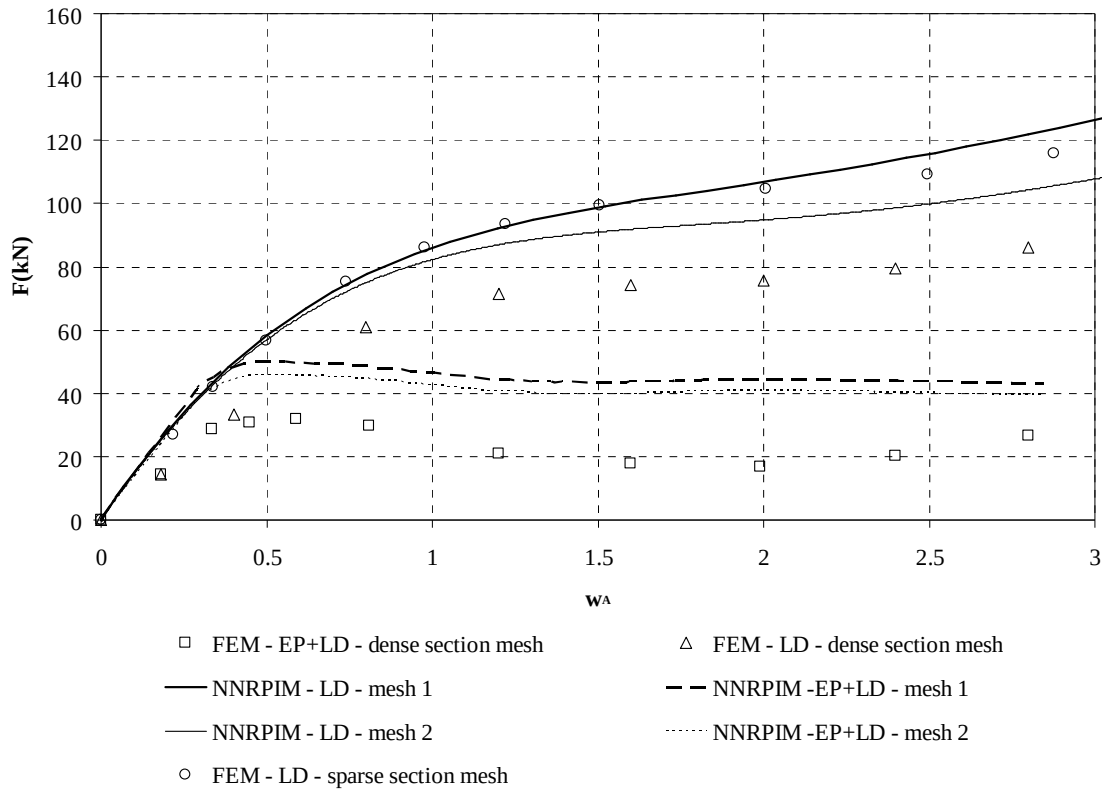


figure 147 – Evolution of the displacement w_A on the top of the dome with the load level, for distinct meshes and distinct numerical methods.

In figure 148 it is shown the deformation and stress field of the dome for several increasing load levels considering the two analysed cases; the elastic and the elastoplastic material behaviour.

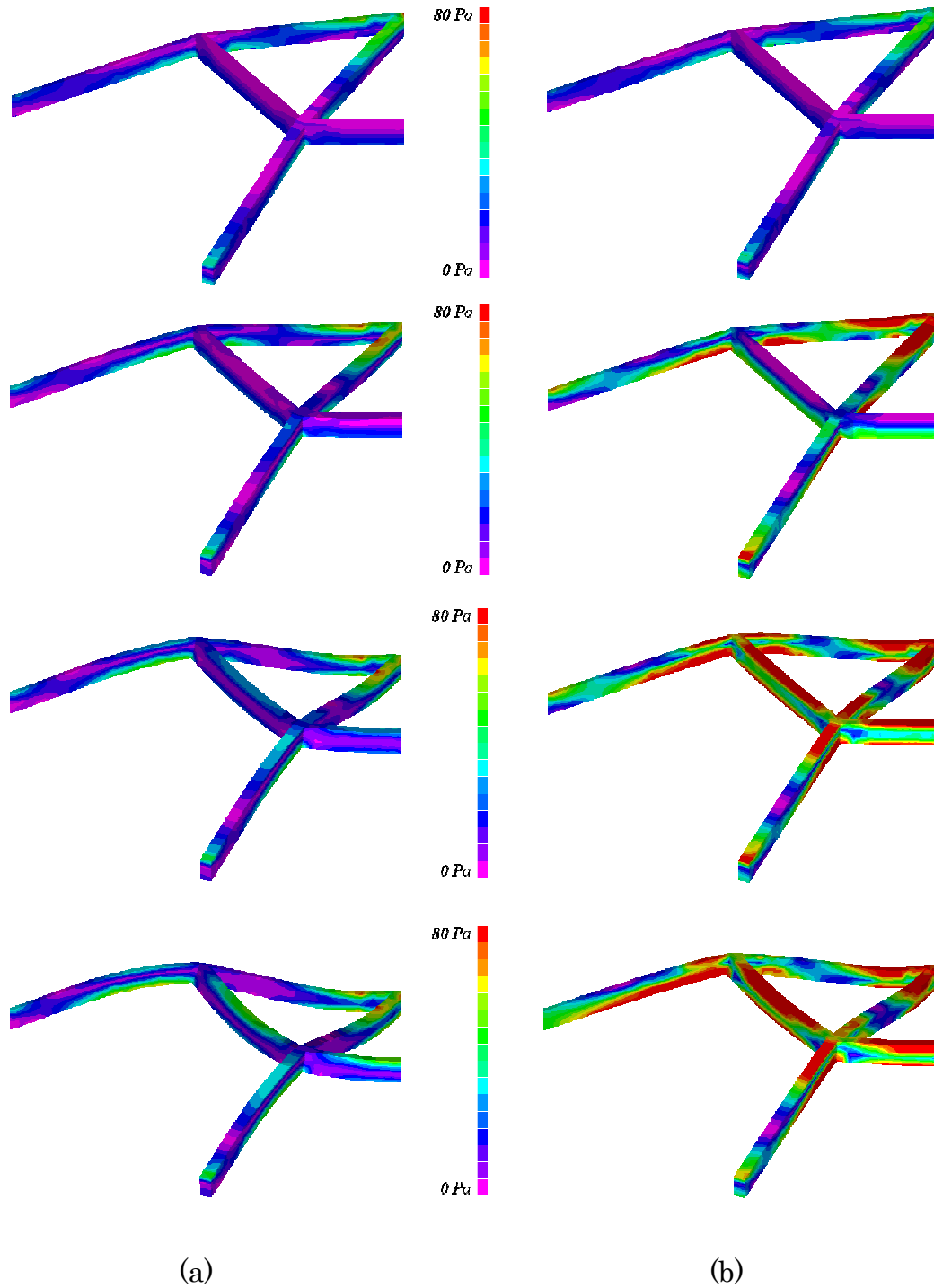


figure 148 – Deformation and stress field of the dome for several increasing load levels. (a) Large deformations with elastic material behaviour. (b) Large deformations with elastoplastic material behaviour.

The quality of the obtained results for this complex example proved that the nonlinear algorithms were successfully applied to the NNRPIM.

5.4 Conclusions and Remarks

In this chapter the NNRPIM was applied to the nonlinear solid mechanical analysis. Benchmark examples were studied and the results permit to conclude:

- i.* The NNRPIM nonlinear analysis produces a displacement field and a stress field smooth, accurate and very close to the FEM solution.
- ii.* With the NNRPIM the imposition of the boundary conditions is straightforward, reducing the computational time comparing with the approximation methods, where that does not happen. The NNRPIM results near the boundary proved to be more accurate than the EFGM.
- iii.* Nonlinear solution algorithms, Newton-Raphson algorithm and the Orthogonal Actualized Ramm's method (arch-length method), were successfully implemented.
- iv.* The stress returning algorithm used in the elastoplastic analysis confers the analysis accuracy, stability and speed.
- v.* The NNRPIM handles large material distortion effectively and provides an accurate solution under large deformation.

The experience acquired along the development of this work and the obtained results [39, 40] permit to conclude that the NNRPIM is an efficient and alternative numerical method for the static nonlinear analysis of solid mechanics, considering large deformations and elastoplastic material behaviour.

Chapter 6

6 Bone Tissue

The bone, one of the hardest tissues on the human body, is a living-dynamic biomaterial tissue active during its life time. The bone tissue possesses many interesting structural properties, comparing with the steel the traction strength is similar, it is three times lighter and ten times more flexible. These properties are essentially due to its heterogeneous microstructure, composed of an organic part (mostly collagen, providing traction capacity) and a mineral part (providing stiffness and strength under compression). It possesses a remarkable remodelling capacity. Under permanent change in response to different signals, such as external loads or hormonal influence, it adapts, auto-repairs and changes the form and the internal biomaterial properties distribution.

6.1 Bone biology: Basic Concepts

The bone tissue is a connective tissue formed by cells, blood vessels, fibbers and organic (collagen) and inorganic substances (carbonated hydroxyapatite). However, in opposition with other connective tissues the extracellular components suffer calcification, which confers hardness. The bone material properties are a consequence of the combination between the organic and inorganic substances. Due to collagen fibbers the bone present high values for the elastic modulus and the ultimate strength in tension. The high value for the ultimate strength in compression is given by the mineral components. In the end, the product is a light material with a microscopic and macroscopic layout, which maximizes the resistance and minimizes the weight. Bones perform several bio functions [103], such as:

Mechanical Functions

- i. Support* – The skeleton provide a frame that keeps the body supported.
- ii. Protection* - Bones serve to protect internal organs and soft tissues.

- iii. Movement* - Bones, muscles, tendons, ligaments and joints function are combined together to generate and transfer forces so that located body parts, or the entire body, may be manipulated.

Metabolic Functions:

- i. Mineral repository* – Bones are an important mineral repository (calcium and phosphorus).
- ii. Blood cells producer* - In the red marrow (hematopoietic) erythrocytes (red blood cells), leucocytes (with blood cells) and platelets are produced.
- iii. Energy repository* - In the yellow marrow fat cells can be found, an energy depot.

Besides all this, the bone is a living tissue, in constant mutation, experimenting a continuous reconstruction and reformulation along its life span, due to mechanical, nutritional and endocrine stimulus [103].

This chapter describes the structure, the composition and material and the mechanical properties of the bone tissue. Bone biologic material laws are presented, as well as a new developed law. Afterwards the bone remodelling algorithm is presented.

6.1.1 Bone Morphology

The human skeleton structure has more than 200 bones classified, by shape, in four major types: long bone, short bone, flat bone and irregular bone, [103]. These bone types are represented in figure 149.

Long bones can be found in the upper and lower members of the body. This type of bone has three primary structures: the diaphysis, the epiphyses and the epiphyseal plate (in a growing phase) or epiphyseal line (in adulthood). Long bones have the particularity of being longer than wider, growing in an initial phase by the elongation of the diaphysis (the body of the bone) towards the two heads in the extremity of the bone, the epiphyses. This longitudinal growth of long bones is the result of the endochondral ossification at the epiphyseal plate. When the growth stops the epiphyseal plate gives origin to the epiphyseal line. The two epiphyses are covered with articular cartilage, which reduces the articular friction, and the rest of the bone body is wrapped with the periosteum, a peripheral layer of connective tissue glued to the bone by collagen fibres (Sharpey fibbers) which has osteogenic functions.

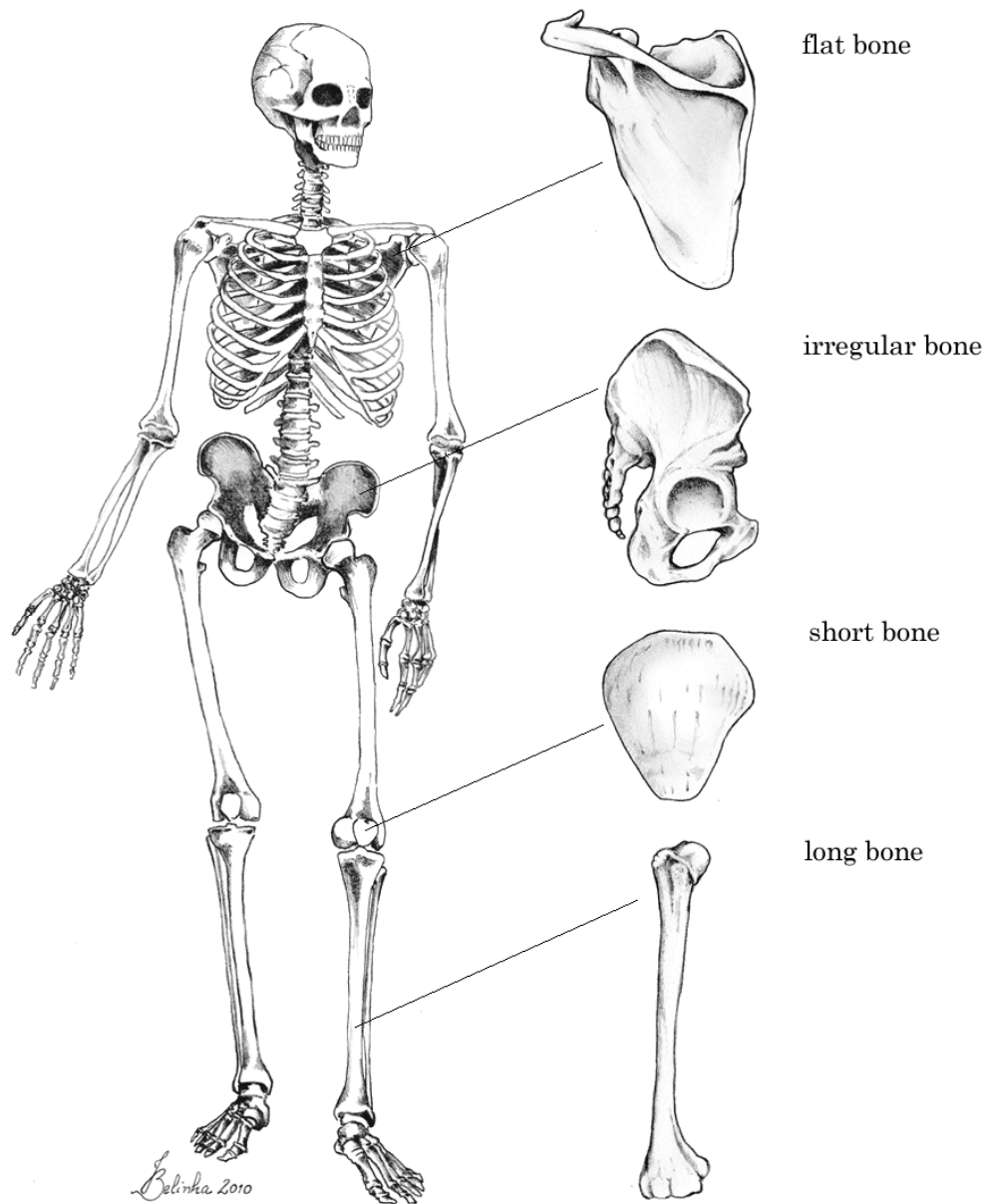


figure 149. – Bone shape major groups.

In the diaphysis zone, the outer shell of the long bone is cortical bone (compact bone) and in the centre it can be found the medullar cavity, which contains in adulthood yellow marrow (mainly fat cells). In the epiphyses the outer shell of cortical bone is thin and in the core it can be found trabecular bone (spongy bone) and red marrow. The red marrow, unlike the yellow marrow which is just a soft tissue with energy depot functions, is a hematopoietic tissue and it is the site of production of red cells, platelets and most of the white blood cells. These features can be found in figure 150.

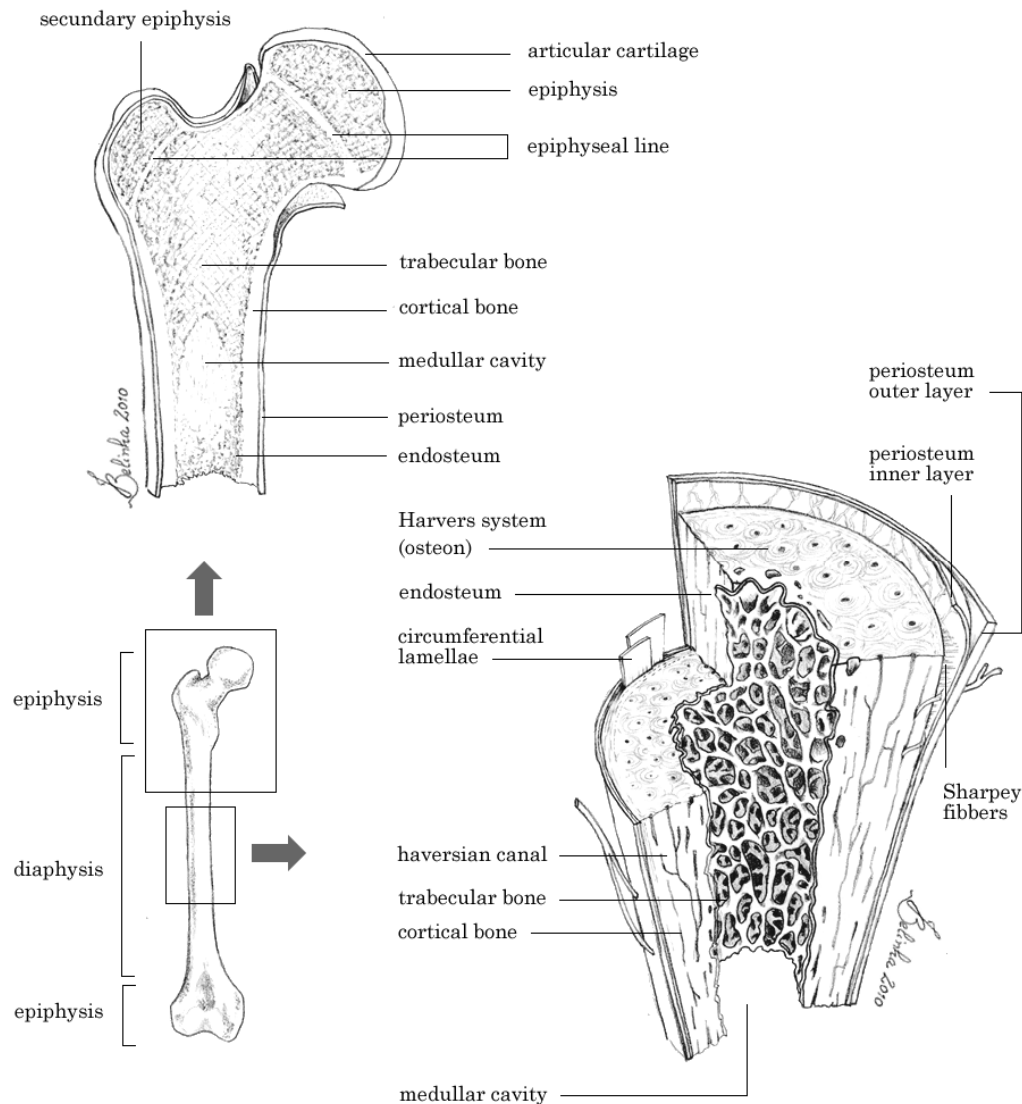


figure 150 – Long bone. Human femur with the identification of the primary structures and inner features of a diaphysis section.

Short bones are characterized to be small and to have no preferential dimension. These bones consist of cancellous tissue (trabecular bone) enclosed within a thin layer of cortical bone. Examples are the carpals, the tarsals and the bones in the ankle and in the wrist.

Flat bones do not have diaphysis or epiphyses. These bones are composed of two thin layers of cortical bone sandwiching trabecular bone tissue, which is the location of red bone marrow. In an adult, most red blood cells are formed in flat bones. These bones have generally a dimension much smaller than the other two. Examples are the cranial bones, sternum or the scapula.

Like the flat bones, irregular bones do not possess diaphysis or epiphyses and are assembled by two thin layers of cortical bone enclosing trabecular bone tissue. In these bones there is no preferential dimension or shape. Irregular bones serve various purposes in the body, such as protection of nervous tissue (the spinal cord is protected by the vertebrae), multiple muscle-skeletal anchor (the sacrum) or support of articulated structures (the hip bone or the zygomatic bone).

6.1.2 Composition and Structure of the Bone

6.1.2.1 Bone matrix

In relation to the bone body dry weight, the bone matrix is proximally composed by 30% of organic material and 70% of inorganic material [103]. The bone organic part is composed majority by collagen (about 94%) and small parts of noncollagenous proteins (about 4%) and bone cells (about 2%). The collagen present in the bone matrix is of type I, although small quantities of type V collagen can be found. Collagen fibbers are arranged along lines of mechanical stress according to Wolff's Law [104]. It is the orientation and the quantity of these fibbers that confer to the bone mechanical properties such as elasticity and high ultimate traction and shear stress. The collagen fibbers are synthesized by the osteoblasts (bone cells). For instance, the cortical bone and the periosteum are bonded through collagen fibbers, the Sharpey fibbers.

Although it represents only 5% of the organic bone matrix, the noncollagenous proteins play an important role in the bone metabolism and in the mineralization of the bone matrix. Only 2% of the organic matrix are bone cells, but without such living elements the bone could not repair, rebuild, remodel, reshape itself or even grow.

The bone inorganic part consists mainly of a carbonate-rich hydroxyapatite, also called bone apatite, which is smaller and less perfect in crystal arrangement than pure hydroxyapatite, $[Ca_{10}(PO_4)_6(OH)_2]$. However it is due to this crystalline imperfections that bone apatite is biological more appropriate than pure hydroxyapatite, once it is more readily available for metabolic activity and for body fluid exchanges. Other minerals are also present in the bone mineral matrix, but in much lower percentage: sodium, magnesium, potassium, fluoride and chloride. The inorganic components

that confer to the bone structure the hardness, the stiffness and the high ultimate compression stress.

6.1.2.2 Bone cells

The bone cells are divided, generally, in five cell-types: osteoprogenitors, osteoblasts, osteocytes, bone-lining cells and osteoclasts. The role of these cells in the bone reabsorption process is presented in figure 151.

The *osteoprogenitors* are immature non-specialized cells which differentiate to origin the osteoblasts [105]. These cells are in the genesis of all bones during the growth process, since the conception, and are generated in the bone marrow, or in other connective tissues. The osteoprogenitors can be found in the interior layer of the periosteum, in the endosteum and in the Haversian's and Volkmann's canals.

The *osteoblasts* are mononuclear cells responsible for the bone formation and for the mineralization of the osteoid matrix. These cells are in the origin of the osteocytes. The osteoblasts, produced by the osteoprogenitors in the periosteum, the endosteum and the bone marrow, can be found in the bone inner surface (endosteum) or outer surface (periosteum) in a laminar distribution. When active the osteoblasts are cuboidal or polygonal shaped [103]. The number of osteoblast cells tends to decrease as individuals become elder, therefore decreasing the natural renovation of the bone tissue [106].

The *bone-lining cells* are inactive elongated and thin osteoblasts. These cells remain inactive in the bone surface once the bone formation, or the bone remodelling, is finished. However the bone-lining cells can be reactivated in response to mechanical and/or chemical stimulation [107].

The *osteocytes* are mature cells derived from the osteoblasts. These cells can be found inside the mineralized bone matrix occupying small chambers, called lacunas. It are connected to each other by long cytoplasmic extensions, through thin canals called canaliculi, which are used in the exchange of nutrients, forming a complete and vast network. The osteocytes, through the canaliculi, conserve the connection with the outer osteoblasts and have as primary purpose the preservation of the surrounding mineralized bone matrix vitality. The osteocytes, using the connections with the outer cells, act as local sensors of the mechanical and chemical state of the bone and, in case of necessity, initiate the bone reabsorption process from the surface [108, 109]. These cells are affected directly by the damage

in the mineralized bone matrix, once it destroys the cytoplasmic extensions, stopping the cellular metabolic process and the cellular fluid exchange. In the particular case of the bone micro damage, the destruction of the cytoplasmic extensions is the trigger button to activate the bone remodelling process. This process is also activated if the osteocytes are unused [110-112].

The *osteoclasts* are plurinuclear cells responsible for the reabsorption and/or elimination of the bone structure. These cells cause the demineralization of the mineralized bone matrix and destroy the bone organic matrix. The osteoclasts cells are formed by the fusion of precursor cells from the bone marrow or from other haematopoietic tissue. These cells are located in superficial depressions (Howship lacunas) which identify the bone reabsorption zones.

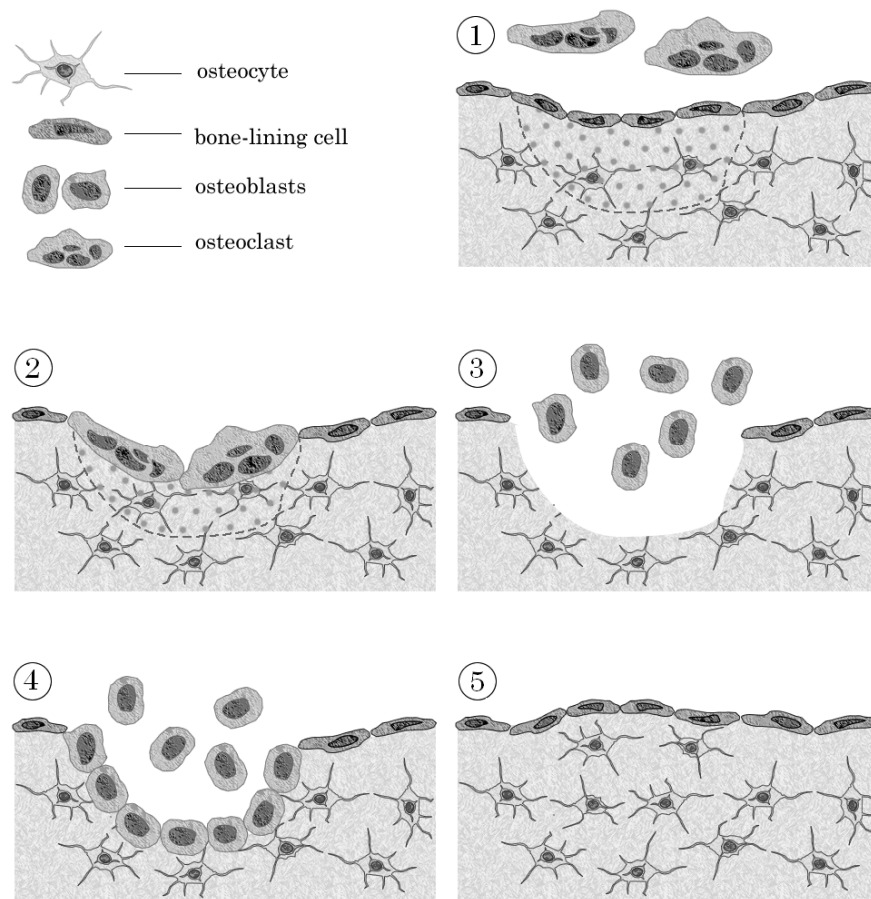


figure 151 – Bone reabsorption process.

6.1.2.3 Reticular and Lamellar Bone

Bone tissue can be classified in reticular or lamellar bone tissue according with the collagen fiber organization inside the bone matrix. The reticular

bone tissue is a neo-formed or immature bone, commonly referred as woven bone. In this bone tissue the collagen fiber arrangement have anisotropic layout, since all are oriented in different directions (randomly), with no preferential orientation. In the fetal development or in a fracture repair, the reticular bone is the first to be formed. In this bone tissue active osteoblasts, excited by growing factors or stimulated by fracture damage, build more solid bone structures.

In a posterior phase, by the remodelling process, the reticular bone gives place to the lamellar bone. The lamellar bone is a mature bone tissue organized in thin layers glued by collagen fibers oriented in perfect alignment with each other. The osteocytes in the interior of the lamellar bone are displayed in sandwich layers between the lamellae. The lamellar bone has the same chemical composition and material properties throughout the skeleton, regardless its mechanism of formation – intramembranous or endochondral – or its structural organization – cortical (compact) or trabecular (soft) bone [103], figure 152.

6.1.3 Cortical and Trabecular Bone

In adulthood the skeleton contains only two types of bone, cortical bone (compact bone) and trabecular bone (soft or cancellous bone) [103].

6.1.3.1 Cortical Bone

The basic functional unit of cortical bone is the osteon (also known as the Havers system), which consists of concentric layers of bony lamellae surrounding a central haversian canal. The haversian canal contains the blood capillaries that supplies the bone tissue. Cross branches, called Volkmann's canals, connect the osteons with each other, conducting blood vessels. Each cylindrical lamella forming the osteon is shaped by osteocytes regularly distributed. The osteocytes communicate with each other by projecting throughout the lamellae very thin and long cytoplasmic extensions along the tiny canals called canaliculi. Is through the canaliculi that electro-chemical information flows and the nutrients and minerals reach all points of the bone matrix. Besides the osteons, long bones contain in the periphery subperiosteal circumferential lamellae, spawned by the inner layer of the periosteum.

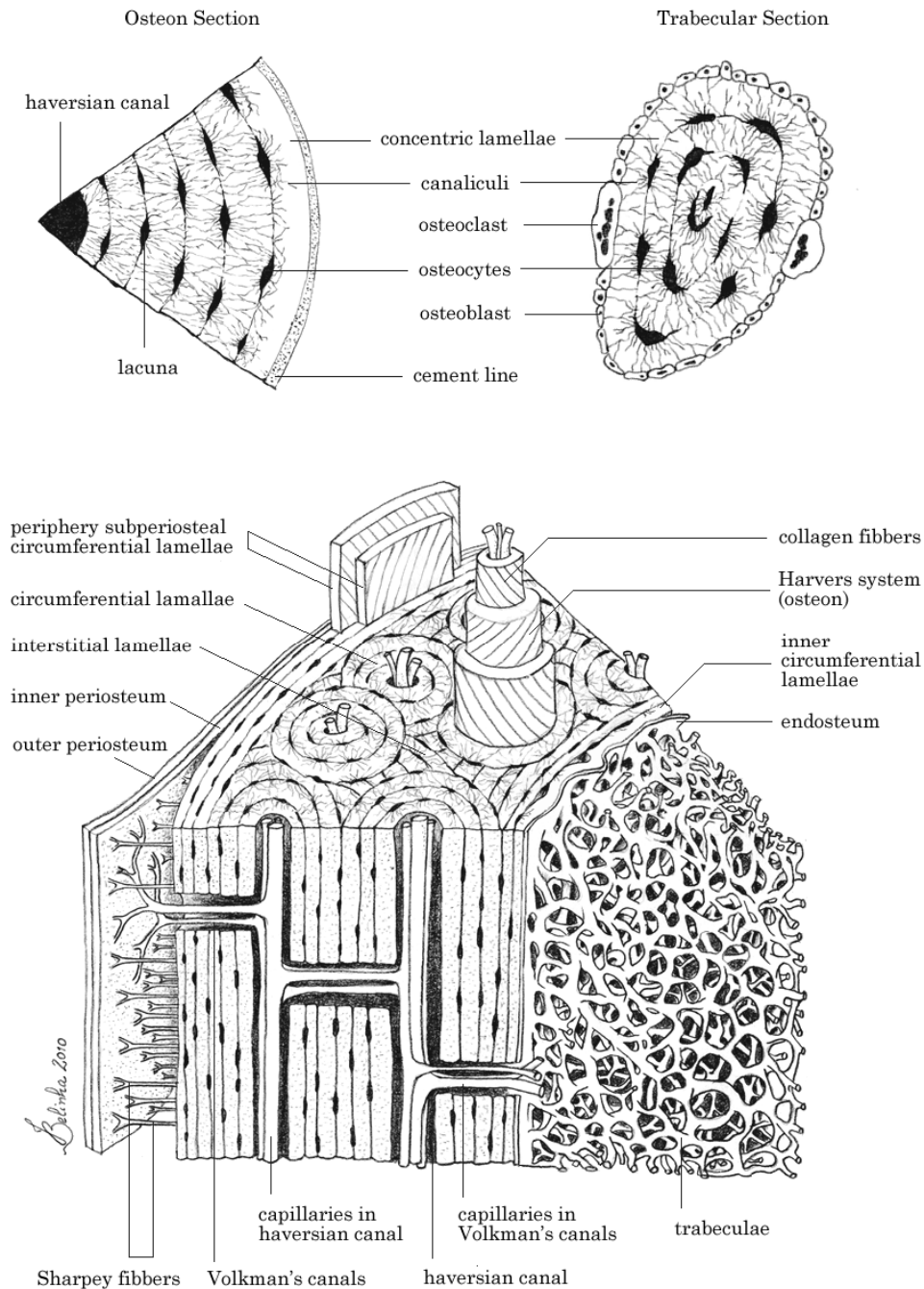


figure 152 – Transversal cross-section pie-shaped of an osteon and transversal cross-section of a trabecular branch. Detailed view of a typical bone cortex.

6.1.3.2 Cortical Bone Remodelling

The remodelling process in cortical bone is activated by growing factors, or changing of load patterns or due to micro or macro damage of the bone tissue. The initial step is the removal of the inactive osteons, by osteoclast activity. Afterwards new lamellae are deposited in new concentric layers, from exterior to interior, until a new osteon is formed. In mature bone the

recurring remodelling process causes the formation and destruction of many osteon generations. The new osteons are organized by a complete circumferential layers, a complete haversian system, this causes that some older osteons present portions of the outer border occupied by the outer border of a new osteon. In some old osteons, this process have occurred so many times that all it is left from the original lamellae are small portions of the circumferential lamellae. These small portions are called interstitial lamellae. Cortical bone is remodelled by bone cells lining the periosteal, the endosteal and the haversian canal surfaces. The periosteal surface is responsible for the bone width growth. The endosteal activity determines the diameter of the medullary canal. The combined activity of the periosteal and endosteal structures determines the thickness of the bone cortex. The haversian canal surface plays an important role in the remodelling process of the bone and it is responsible for the cortex density.

6.1.3.3 Trabecular Bone

The trabecular bone is composed of mature lamellar bone and, in contrast with the compact structure of cortical bone, it is a complex network of intersecting curved plates and tubes. The osteocytes within each trabecula are concentrically oriented and have a well-developed canalicular set of connections. In the case of long bones the trabecular bone is typically located at the proximal ends, here the arrangement of the trabeculae is relatively regular, reflecting the direction of the principal mechanical stresses to which this bone is subjected. Trabecular bone constitutes 20% of the skeletal bone mass, being the remaining 80% occupied by cortical bone, nevertheless due to the trabeculae vast surface the surface-to-volume ratio is ten times higher than the cortical bone. It is because this high specific surface that the metabolic activity of trabecular bone is nearly eight times higher when compared with the cortical bone, which may help to explain why metabolic bone diseases have a greater effect on trabecular bone than on cortical bone. Regardless of its relatively small volume and apparent porosity, trabecular bone is well adapted to resist and conduct compressive loads, as can be seen by the vertebrae, handing over to the cortical bone other structural functions, as resisting bending and torsional stresses.

6.1.3.4 Trabecular Bone Remodelling

The border of the trabecula is filled with osteoblasts and osteoclasts, ready and active. During the growth or remodelling of the trabecula the deposition of new bone by osteoblast activity is counterbalanced by the removal of bone by osteoclast activity in the opposite surface of the trabecula, remodelling

and reshaping the trabeculae configuration to a better stress path configuration or simply to repair micro-factures. The diminish of the bone mass with age is the result of the change in the balance between the rate of bone formation and the rate of bone reabsorption. Pathologic fractures occur when the bone density becomes so low that the skeleton can no longer withstand the mechanical stresses of everyday life. In a experimental study [113] it was showed that the compressive strength of bone is proportional to the square of its apparent density, i.e., if the bone density decreases by a factor of 3, its compressive strength decreases by a factor of 9. Both cortical and trabecular bone present analogous microscopic physical properties and biochemical composition. However, in order to suit local physical requirements, due to loads and to stresses distribution, the bone macroscopic structure is organized in order to maximize the strength and stiffness. For example, at the ends of long bones the thin cortical shell, inner supported by trabecular bone, is the optimal configuration to hold and distribute in the bone structure the concentrated loads applied. While the tubular cortical diaphysis is optimized to support the large torsional and bending loads. Dispite being cortical or a trabecular structure, all normal adult bone is lamellar bone. In adults, only due to normal fracture healing, or in pathologic conditions, immature woven bone, or fibber bone, is seen.

6.2 Bone Tissue Mechanical Properties

As previously referred, bone tissue can be classified as cortical bone, highly densified bone tissue, and trabecular bone, which shows a considerably smaller apparent density. Although both types show the same molecular arrangement the mechanical behaviour is different. Many experimental studies show that the bone mechanical properties depend on the bone composition and on the bone porosity (directly related with the bone density) [114-119].

In the work of Carter and Hayes [120] it is stated that for trabecular and cortical bone the elastic modulus is closely related to the cube of the bone apparent density and that the strength is closely related to the square of the bone apparent density. The apparent density, ρ_{app} , is defined as the wet mineralised mass of bone of the sample tissue, w_{sample} , over the volume occupied by the same sample tissue, V_{sample} .

$$\rho_{app} = w_{sample}/V_{sample} \quad (229)$$

The apparent density is dependent on the bone porosity, p , which can be obtained by the expression,

$$p = \frac{V_{holes}}{V_{sample}} \quad (230)$$

Where the volume of bone holes, V_{holes} , is obtained by the Archimedes principle [121]. Thus, the apparent density can also be achieved using the porosity,

$$\rho_{app} = \rho_0 \cdot (1 - p) \quad (231)$$

Being ρ_0 the compact bone density, $\rho_0 = 2.1 \text{ g/cm}^3$. The apparent density was the only variable accounted in these first models, therefore several authors started to study the influence of other variables in the mechanical properties of the cortical and trabecular bone. Thus, the apparent density combined with the bone mineral content, condensed in the bone ash density definition, was studied and correlation curves for the bone elasticity modulus and the bone ultimate compression stress were obtained [122-126]. However these models have limitations, one of which is that none of these models take into account the influence of splitting the bone volume fraction from the ash fraction, therefore Hernandez [127] express the apparent density as a function of the bone volume fraction and the ash fraction, achieving a mathematical model that determine the elastic modulus and compressive strength with a 97% correlation.

With these mathematical models one is capable to predict the bone main mechanical properties, however these correlations do not consider the influence of the bone micro-structure (cortical versus trabecular) or even the bone mechanical behaviour in distinct directions.

6.2.1 Lotz material law

The work of Lotz [128] was one of the firsts to consider the bone orthotropic behaviour. Lotz was able to determine the elasticity modulus and the ultimate compressive stress mathematical laws for both cortical and trabecular bone in the axial and transversal direction using as variable only the apparent density. The bone mechanical properties are approximated with the expression,

$$\begin{aligned} E_i &= a_1 \cdot (\rho_{app})^{a_2} \\ \sigma_i^c &= a_3 \cdot (\rho_{app})^{a_4} \end{aligned} \quad (232)$$

Being E_i the elasticity modulus and σ_i^c the ultimate compression stress in direction i , both are expressed in MPa and the apparent density ρ_{app} in g/cm^3 . The coefficients a_j are presented in table 24 for the axial and transversal direction for both cortical and trabecular bone.

Bone Tissue	Direction	a1	a2	a3	a4
Cortical	Axial	2065	3.09	72.4	1.88
	Transversal	2314	1.57	37	1.51
Trabecular	Axial	1904	1.64	40.8	1.89
	Transversal	1157	1.78	21.4	1.37

table 24 – Lotz law coefficients.

Thus, Lotz considering the bone anisotropy propose distinct material laws for the cortical and the trabecular bone.

6.2.2 Proposed material law

A recent experimental study by Zioupos [129], reinforces the idea that density is a salient property of bone and plays a crucial role in determining the mechanical properties of both its trabecular and cortical structural forms. The study, using the measured apparent density, was able to objectively isolate the bone in trabecular and compact forms. The results from the work of Zioupos show that the relation between E_i and ρ_{app} is not an increasing monotonic function, as it is Lotz law, but instead it is a 'boomerang'-like pattern. The experimental work of Zioupos also shows that the law governing the mechanical behaviour of the bone tissue is the same for cortical bone and trabecular bone. This experiment permits to identify the trabecular bone from the cortical bone only with the apparent density measure,

$$\begin{aligned} \text{trabecular bone} &\rightarrow \rho_{app} \leq 1.3 g / cm^3 \\ \text{cortical bone} &\rightarrow \rho_{app} > 1.3 g / cm^3 \end{aligned} \quad (233)$$

The identified bone minimum and maximum apparent density in Zioupos work was: $\rho_{app}^{min} = 0.1 g/cm^3$ and $\rho_{app}^{max} = 2.1 g/cm^3$. In figure 153 it is presented the experimental data shown in Zioupos work [129]. Thus, following the conclusions of Zioupos [129], this work proposes a unifying law for cortical and trabecular bone. The bone elasticity modulus for the axial direction is obtained using the approximation curve,

$$\begin{aligned} E_{axial} &= \sum_{j=0}^3 a_j \cdot (\rho_{app})^j \quad \text{if } \rho_{app} \leq 1.3 g/cm^3 \\ E_{axial} &= \sum_{j=0}^3 b_j \cdot (\rho_{app})^j \quad \text{if } \rho_{app} > 1.3 g/cm^3 \end{aligned} \quad (234)$$

being the coefficients a_j and b_j presented in table 25. This curve presents a 95% correlation with the experimental data, figure 153. Since in Zioupos work only the bone elasticity modulus in the axial direction is analysed, the other curves related with the elasticity modulus in the transversal direction and the ultimate compression stress in the axial and transversal direction were obtained based in the values suggested in [128]. As in the curve for the elasticity modulus in the axial direction, the other curves suggested in this work are unifying cortical and trabecular bone curves,

$$E_{trans} = \sum_{j=0}^3 c_j \cdot (\rho_{app})^j \quad (235)$$

$$\sigma_{axial}^c = \sum_{j=0}^3 d_j \cdot (\rho_{app})^j \quad (236)$$

$$\sigma_{trans}^c = \sum_{j=0}^3 e_j \cdot (\rho_{app})^j \quad (237)$$

where the coefficients c_j , d_j and e_j are presented in table 25. The elasticity modulus and the ultimate compression stress approximated by the curves presented in equations (234), (235), (236) and (237) are expressed in *GPa* and the apparent density ρ_{app} in g/cm^3 .

Coefficients	$j = 0$	$j = 1$	$j = 2$	$j = 3$
a_j	0.00	721.61	805.86	0.00
b_j	-177044.29	386136.43	-279771.43	68357.14
c_j	0.00	0.00	2003.56	-144.23
d_j	0.00	0.00	26.80	20.35
e_j	0.00	0.00	25.01	1.25

table 25 – Coefficients of the propose bone model

The plot of the approximation for the elasticity modulus in the transversal direction is presented in figure 154. For the compression stress in the axial direction and in the transversal direction the approximation curves are presented respectively in figure 155 and in figure 156. The ultimate tension stress of the bone tissue is usually defined as a proportion of the ultimate compression stress, $\sigma_i^t = \alpha \cdot \sigma_i^c$. There are several studies on this subject, experimental [130-132] and numerical [133]. The value of α to be used is not consensual, in the various works available in the literature α varies between 0.33 and 1.00, as so in this work it will be considered $\alpha = 0.5$, once it is a conservative value. Although it will be not considered in the analyses on this work, the bone shear ultimate stress varies between 49 MPa and 69 MPa [132].

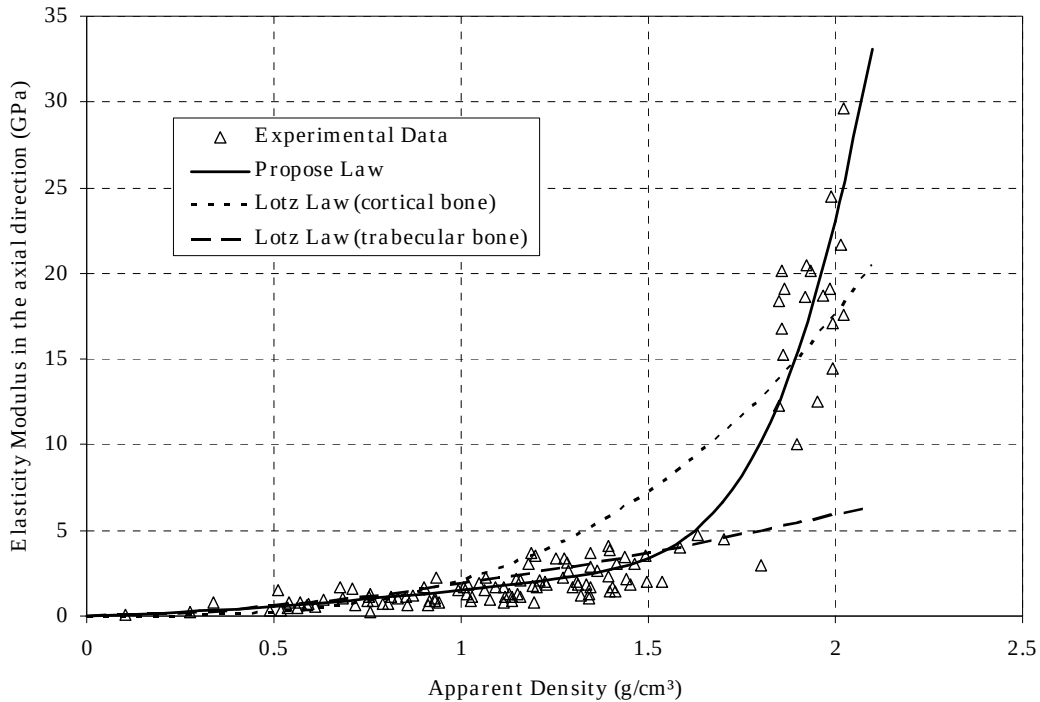


figure 153 – Elasticity modulus in the axial direction. Experimental data obtain in Zioupos work compared with Lotz law for cortical and trabecular bone and with the mathematical model proposed in this work.

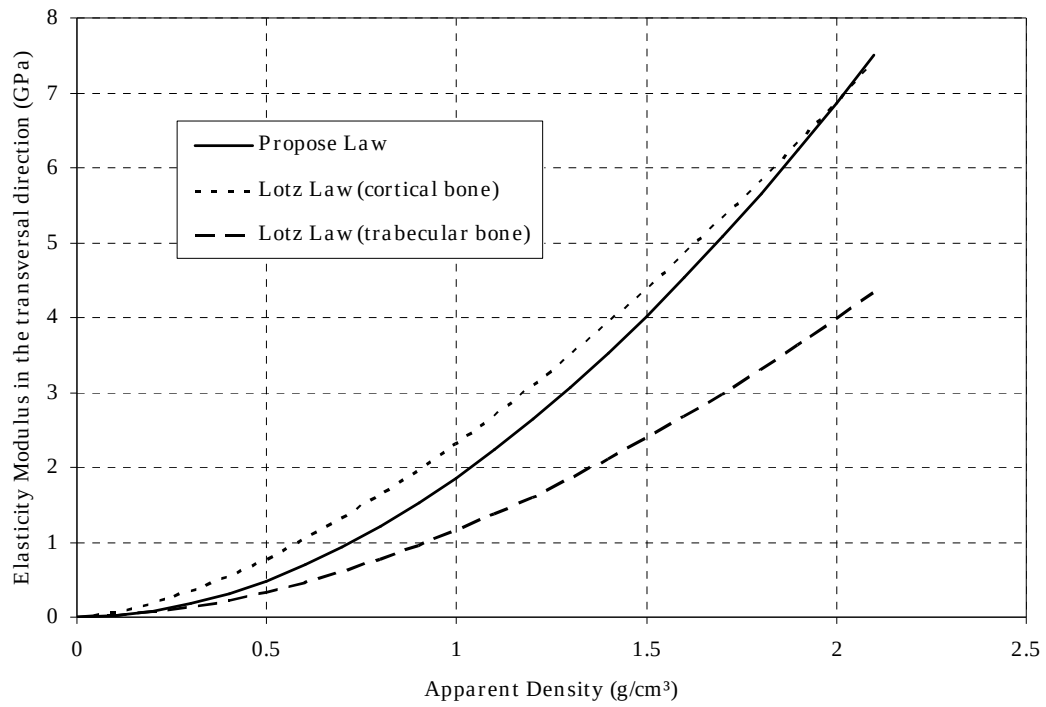


figure 154 - Elasticity modulus in the transversal direction. Lotz law for cortical and trabecular bone compared with the mathematical model proposed in this work.

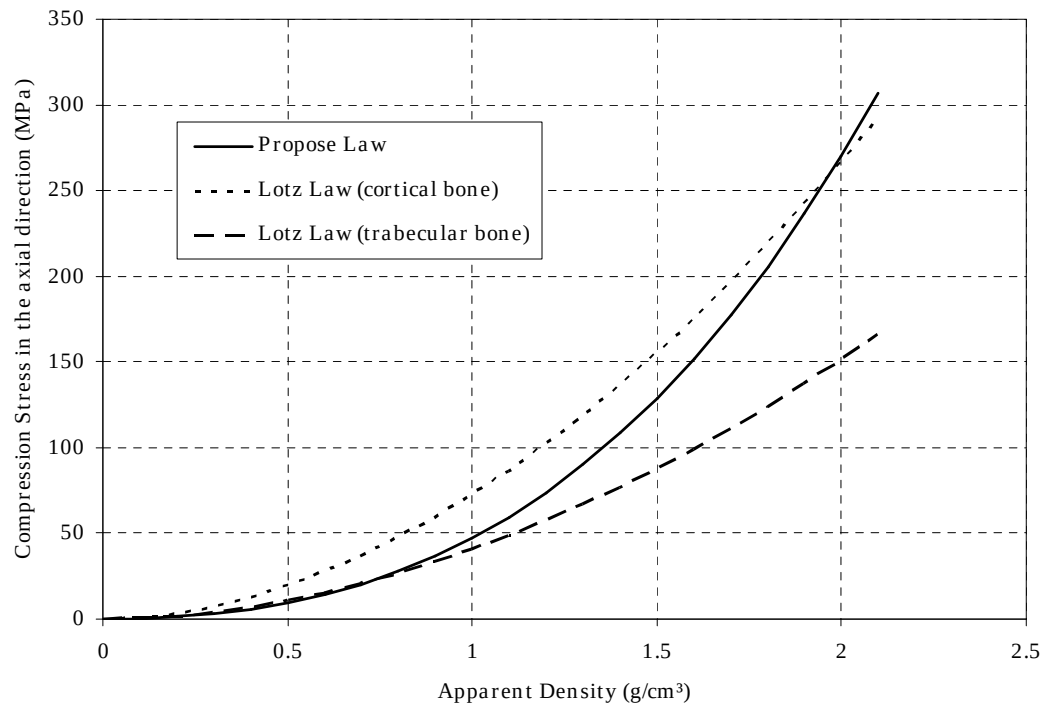


figure 155 – Compression stress in the axial direction. Lotz law for cortical and trabecular bone compared with the mathematical model proposed in this work.

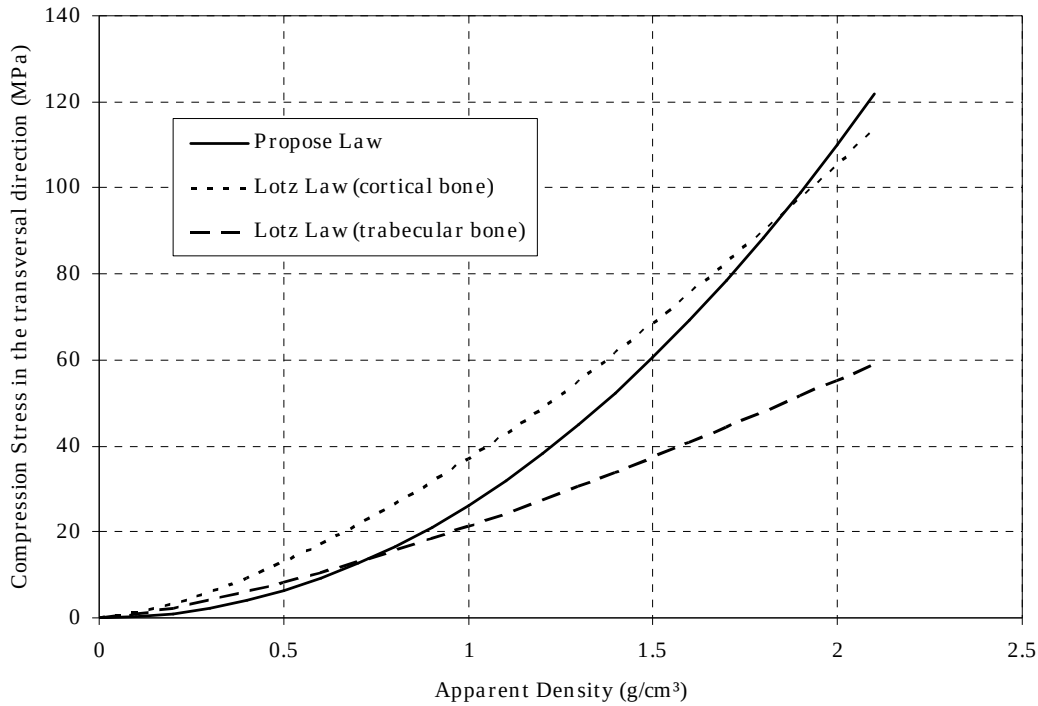


figure 156 · Compression stress in the transversal direction. Lotz law for cortical and trabecular bone compared with the mathematical model proposed in this work.

6.3 Bone Remodelling Algorithms

Recent experimental investigations shown the existence, in the bone remodelling process, of a strong correlation between the bone functional adaptations and the induced stress (or the strain). The strain distribution, the dynamic nature of the loads and the number of loading cycles seem to be the most significant external stimuli in the bone remodelling process. However, experimental research has shown that the bone remodelling and functional adaptation are quite complex and so far cannot be readily described in detail.

In the pursue of explaining the nature of bone remodelling phenomena, numerous semi empirical laws, phenomenological based mathematical descriptions of the remodelling processes, were developed and proposed. With these semi empirical laws is possible to predict and simulate the actual stress distribution and path related with bone remodelling process. These mathematical formulations consider bone tissue to be a local adaptive material, directly dependent on the mechanical loading, mostly characterized by the strain or stress tensors, with the local remodelling reactions as observed in experiments, through appropriate bone growth laws. The basis of the computational tools simulating the natural bone

adaptation, occurring under known stress states and changes in geometry and stiffness, are these semi empirical bone growth laws. Next, a brief overview over bone remodelling theories proposed by several authors is presented.

6.3.1 Pauwels's Model

Pauwels [134] was one of the firsts to suggest a mathematical formulation for the “Wolffs Laws”. In order to ensure a balanced state of bone reabsorption and deposition, the existence of an optimal mechanical stimulus S_n was considered. Pauwels was only interested in the surface remodelling of long bones, in which the stress state can be approximately described as uniaxial, thus the remodelling mechanical stimulus S_n was assumed to be the axial stress. Therefore the optimal value for S_n corresponds to an optimal axial stress value, σ_n . Stress values exceeding σ_n lead to bone hypertrophy and values below σ_n indicate bone atrophy. This material law can be used iteratively, forcing the bone stress state to move in the direction of the optimal homeostatic value, σ_n . In Pauwels law one has to consider, as bone material property, a maximum admissible stress, σ_{max} , and a minimum admissible stress, σ_{min} , being $\sigma_n = (\sigma_{max} - \sigma_{min})/2$. The actual stress must be limited between the interval: $\sigma_{min} \leq \sigma \leq \sigma_{max}$. Kummer [135] summarized this model in a simple cubic expression.

$$\frac{\partial m}{\partial t} = \beta (\sigma - \sigma_{min})(\sigma - \sigma_n)(\sigma - \sigma_{max}) \quad (238)$$

In this expression $\partial m / \partial t$ is the variation of mass bone through time. The coefficient β is an empiric model parameter, which must be obtained experimentally [135].

6.3.2 Cowin's Model

The bone internal remodelling process (the growth and the reabsorption of bone material) was described as the result of chemical reactions between bone matrix and the extracellular fluids in a sophisticated continuum theory developed by Cowin [136], named adaptive elasticity model of Cowin. In order to account the reorientation and the changes in the anisotropic material behaviour of the trabecular design, Cowin [137] used the fabric

tensor to develop an anisotropic material model for trabecular bone. The fabric tensor is a symmetric second order tensor, which describes the stereological behaviour of the microstructural arrangement of trabeculae and pores [138]. The fabric tensor is directly related with the material elasticity tensor [139]. In this model it is assumed that the state of equilibrium remodelling is achieved when the stress (and strain) principal directions are coincident with the fabric tensor principal axes. This model presents a significant disadvantage, being necessary a high number of bone remodelling parameters in order to describe the remodelling behaviour.

In the pursue to overcome this parameter identification problem it was suggested [140] that, in the case of bone surface and core remodelling, the strain energy density (SED) could serve as a mechanical stimulus. In Huiskes work [140] it was proposed a SED 'lazy zone' in which no bone remodelling take place. This modified version of the 'Adaptive Elasticity' approach was used to predict the proximal femur density distribution [140]. The obtained results converge to a solution very close with the actual density patterns observed in the real sample femur.

This model was improved by many authors. The overstrain necrosis, with anisotropic material behaviour, was considered along with a SED approach by Harrigan and Hamilton [141-143]. Based in the idea that the remodelling process is not just only influenced by the momentary strain state, Bucháček [144] developed an extended version of the adaptive elasticity model of Cowin where bone density and material orientation are dependent on the whole bone strain history. It was assumed that the difference between the actual and optimal density and the variation between the actual and optimal material orientation, with both depending on the strain state, serve mutually as the adaptive stimuli and have a time-fading effect on the remodelling process. In this manner, at a given time t any past strain event will have some effect on the remodelling process, however because it is considered an exponential time-fading function the bone reactions are dominated mainly by the recent strain state.

6.3.3 *Carter's Model*

A mathematical formulation for the functional adaptation of trabecular bone based on a self-optimization concept was proposed by Carter [145-147]. Also in this model, as in Pauwels's model, it was assumed, within the bone tissue, that a mechanical stimulus S_n must be present in order to maintain a quasi-stationary state in the bone remodelling process. Carter suggested

that this stimulus should be constant for the entire bone and proportional to the effective stress.

$$S_n \propto \sum_{i=1}^l m_i \bar{\sigma}_i^{\kappa} \quad (239)$$

In this formulation distinct load cases can be considered, $i = 1, \dots, l$, and each one is weighted by the corresponding number of load cycles, m_i . The influence of the magnitude of the correspondent stress state is considered by the exponent κ . The effective stress $\bar{\sigma}_i^{\kappa}(\sigma_i, \rho_{app})$ depends on the local stress state σ_i of load case i and on the local apparent density ρ_{app} . Regardless the considered bone material law or the biological basis of bone remodelling, with this model it is assumed that functional adaptation provides the bone with the ability to maximize its structural integrity with the least amount of bone mass. Which is the same as assuming that the induced stress from bone remodelling acts as an optimization tool, minimizing an objective function [148]. Instead of using the local effective stress, it is possible to use the strain energy density approach (SED). The use of the strain energy is connected with the idea that the bone tissue is attempting to maximize its stiffness while the use of a failure stress criterion is related with the material strength optimization. The SED approaches lead to a correlation between the apparent density ρ_{app} and the local strain energy, U .

$$\rho_{app} \propto \left(\sum_{i=1}^l m_i U_i^{\kappa} \right)^{\frac{1}{\kappa}} \quad (240)$$

This expression permits to estimate the apparent density as a function of the strain energy in an equilibrium state of the bone remodelling process. It can also be used as an optimality criterion in an iterative optimization procedure. The finite element method (FEM) was used, combined with this formulation, to predict the apparent density distribution in the actual bone [145]. The proximal femur was the presented example, and the FEM was used to obtain the stress and strain distributions for each distinct typical loading cases. With the proposed algorithm, starting from a homogenous density distribution, it was possible to predict density distributions similar with those found in the real femur within only a few iterations. However, the convergence could not be obtained, and the iteration process led to non-physiologic states, such as a complete bone with zero density or a complete bone with cortical density.

Carter assumed that the trabeculae are oriented in the direction of the principal stresses, respecting the trajectorial hypothesis accepted by many authors. For a single load case it was shown that an alignment between the material principal directions and the stress principal directions (and consequently the strain principal directions) results in an optimal configuration with respect to the local stiffness maximization [148]. If multiple load cases are considered, a weighted combination of the normal stress components, with respect to a normal-vector $\bar{n}$, was suggested to serve as a stimulus for trabecular growth in the corresponding direction. The effective global normal stress $\bar{\sigma}_n$ is calculated in analogy with equation (240).

$$\bar{\sigma}_n(\bar{n}) = \left(\sum_{i=1}^l \frac{m_i}{\sum_{j=1}^l m_j} \bar{\sigma}_{n_i}^{\kappa}(\bar{n}) \right)^{\frac{1}{\kappa}} \quad (241)$$

It was suggested [146] that the material stiffness in any direction $\bar{n}$ was directly dependent on the magnitude of the corresponding global normal stress $\bar{\sigma}_n$, however no practical implementation of this trabecular orientation approach was shown.

A modified version of Carter's algorithm was proposed [149] in a research on the adaptive growth reactions of bone following total hip joint replacement. In order to reduce the number of independent material properties of the orthotropic case, the bone material was assumed to be transversely isotropic. This modified version respects the trajectorial hypotheses of Wolff law, the directions of the material axes were aligned with the stress principal direction. The femur density distribution obtained numerically, in the case of the pre-surgery state, was very close with the actual density distribution.

Pettermann [150] suggested an improved version of Carter's algorithm. It is based in the assumption that the adaptation of bony tissue can be described appropriately on the continuum level by using overall tissue material parameters and stress/strain measures. The specific mechanical stimuli act as driving forces in the remodelling processes. Any material parameter actually contributing to the local bone stiffness will be subjected to a specific remodelling process which tries to adapt the effective stiffness behaviour at the particular site under consideration according to the local stresses and strains. In this approach the essential material parameters governing the elastic material behaviour of bone tissue are given by the apparent density,

ρ_{app} , the structural anisotropy is described by the orthotropic parameters and the orientation of the principal material axes with respect to the global coordinate system, as described by a rotational vector. Negative values of the stimulus lead to bone reabsorption whereas positive ones give rise to local bone hypertrophy. Bone hypertrophy in the case of internal remodelling is interpreted as an increase in bone apparent density. Each remodelling stimulus has to exceed a specific threshold level to cause any actual adaptive changes at all, which means that bone material is assumed to show a ‘lazy-zone’ behaviour in the vicinity of its homeostatic state.

Within Carter’s model, the effects of the nonlinearity in the equation of bone remodelling was also studied [151] and developed [152]. This nonlinear development of Carter’s model has in the origin the work of Huiskes [140, 153], Weinans [154] and Mullender [155]. The Carter model was widely applied with the FEM [156], with diverse variations in order to obtain the most accurate density distribution [157, 158]. An interesting approach was proposed by Chen [159], in which it is presented an alternative way to obtain the density, very useful for the case of meshless methods. The density is iteratively obtained in the nodes, and not in the integration points, reducing in this manner the computational time and also the accuracy of the process. More recently the Carter’s model was extended to the micro-finite element method [160, 161], with stunning results. The bone adaptation and the inner remodelling processes were studied and the results show a perfect correlation with the actual density in the human proximal femur. Another conclusion was that a bone abnormality which has little contribution to bone structural integrity tends to disappear.

6.3.4 *Rodrigues’ Models*

Rodrigues and co-workers presented a model [162] that considers a global-local hierarchical approach in which a global model of an entire bone supplies strain and density information to a series of local models characterizing the trabecular microstructure at each global model location. The process of bone adaptation is described for two levels of the bone structure: the macroscopic level where the bone apparent density is determined and a microscopic level characterizing the trabecular structure. The law of bone remodelling is obtained assuming that bone adapts to functional demands in order to satisfy a multicriteria for structural stiffness and metabolic cost of bone formation [163]. More recently the model was successfully extended to the three-dimensional analysis [164], being the distribution of the bone apparent density as well as the microstructural

designs characterizing both anisotropy and bone surface area density were consistent with the real bone apparent density distribution.

6.3.5 *Proposed Adaptation of Carter's Model Models*

In this section it is presented the remodelling algorithm considered in the adaptation of bony tissue analysis with the NNRPIM. This approach is an adaptation of Carter's remodelling algorithm [145]. It is based on the assumption that the adaptation of bony tissue responds mainly to mechanical stimulus, acting as driving forces in the remodelling process, which can be suitably described by stress and/or strain measures. The local density and material orientation is dependent on the stress/strain field caused by the mechanical load. A forward Euler scheme is implemented, resulting in an iterative remodelling process described next.

In a preliminary phase it is considered the bone model as a homogenous body. The material properties are the same as the cortical bone obtained for the maximum apparent density $\rho_{app}^{max} = 2.1 g/cm^3$, only considering the axial direction, obtained from table 24 and table 25, respectively for Lotz law and for the bone material law proposed in this work. Thus, initially the mechanical load is applied to an isotropic model, no preferential directions are known. With the applied load case i the strain field, $\boldsymbol{\epsilon}^i$, and the stress field, $\boldsymbol{\sigma}^i$, are obtained. The principal stresses and directions can be obtained using the stress field $\boldsymbol{\sigma}^i$. Afterwards the Von Mises effective stress field, $\bar{\sigma}^i$, is obtained using the second deviatoric stress invariant, J_2 .

$$\bar{\sigma} = \sqrt{3J_2} = \sqrt{\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \quad (242)$$

In equation (242) σ_1 , σ_2 and σ_3 are the principal stresses, where $\sigma_1 \geq \sigma_2 \geq \sigma_3$. Next, the strain energy density (SED) field for the considered load case is determined using the stress field and the strain field.

$$U^i = \frac{1}{2} \boldsymbol{\sigma}^T \boldsymbol{\epsilon} \quad (243)$$

Following Carter's model [145], after obtaining the variable fields for each load cycle, the final generic variable field ξ is calculated by an appropriate superposition of a number of relevant discrete load cases, l , weighted according to the corresponding number of load cycles, m .

$$\xi = \sum_{i=1}^l \frac{m_i \xi^i}{\sum_{j=1}^l m_j} \quad (244)$$

The entire required variable fields, which are the effective stress field, the principal stress field, the principal direction field, the strain field and the SED field, are weighted with equation (244). After considering all load cycles, global variable fields are obtained. In this phase, due to several factors, there is the possibility that the bone model is being subjected to loads for which it is not capable to sustain. The wrong prevision of the applied load magnitude or the incorrect location of the applied loads can increase the stress inside the bone structure, conducting to stress values much higher than the ultimate stress considered in the bone material laws in section 6.2. Therefore, in order to level the stress field with the bone ultimate stress, the effective stress field is then compared with the admissible effective stress field $\bar{\sigma}^{ad}$, which depends on the apparent density in each interest point j .

$$\bar{\sigma}_j^{ad} = \sqrt{\frac{1}{2} \left[\left(\sigma_1(\rho_{app_j}) - \sigma_2(\rho_{app_j}) \right)^2 + \left(\sigma_2(\rho_{app_j}) - \sigma_3(\rho_{app_j}) \right)^2 + \left(\sigma_3(\rho_{app_j}) - \sigma_1(\rho_{app_j}) \right)^2 \right]} \quad (245)$$

In this work the bone material is considered an orthotropic material, with two preferential directions: the axial and the transversal direction. The idea is to orient the trabeculae in the direction of the principal stresses, respecting the trajectorial hypothesis proposed initially by Wolff, as so $\sigma_1 = \sigma_{axial}$ and $\sigma_2 = \sigma_3 = \sigma_{trans}$. Using these stress relations and developing equation (245) the admissible effective stress $\bar{\sigma}^{ad}$ is obtained for each interest point j .

$$\bar{\sigma}_j^{ad} = \left| \sigma_{axial}(\rho_{app_j}) - \sigma_{trans}(\rho_{app_j}) \right| \quad (246)$$

Then the ratio between the effective stress and the admissible effective stress is obtained for each interesting point j .

$$\lambda_j = \frac{\bar{\sigma}_j^{ad}}{\bar{\sigma}_j} \quad (247)$$

By definition, to obtain the maximum admissible effective stress supported by the cortical bone it is only necessary to substitute in equation (246) the

value $\rho_{app}^{max} = 2.1 g / cm^3$, which is the maximum effective stress permitted in this model, once the bone density cannot be higher than $\rho_{app}^{max} = 2.1 g / cm^3$. The global fields variables ξ are then levelled using the maximum value of λ_j , $\lambda = \max(\lambda_j)$.

$$\xi^{new} = \lambda \xi \quad (248)$$

This process insures that the stress in the model is not superior to the maximum admissible stress.

With the principal stresses levelled the next action is to determine the correspondent density for each interest point j . Although it is possible to be considered as anisotropic, in this work the density is considered isotropic, equal in the axial and transversal direction. As so, the following relations are assumed,

$$(\sigma_{axial})_j = (\sigma_1)_j \quad \text{and} \quad (\sigma_{trans})_j = \max((\sigma_2)_j, (\sigma_3)_j) \quad (249)$$

and by back substitution in equations (236) and (237) it is possible to obtain the correspondent density in the axial direction ρ_{app}^{axial} and in the transversal direction ρ_{app}^{trans} . For the interest point j the new density to be used in the next iteration step is obtained with $\rho_{app} = \max(\rho_{app}^{axial}, \rho_{app}^{trans})$. Also in the next iteration step the original material matrix is oriented with the principal directions obtained in the present iteration, orienting each time the material with the actualized load path,

$$\mathbf{c}^{current} = [\mathbf{T}_{oz}]_{\theta}^T \left[[\mathbf{T}_{oy}]_{\omega}^T \mathbf{c}^{initial} [\mathbf{T}_{oy}]_{\omega} \right] [\mathbf{T}_{oz}]_{\theta} \quad (250)$$

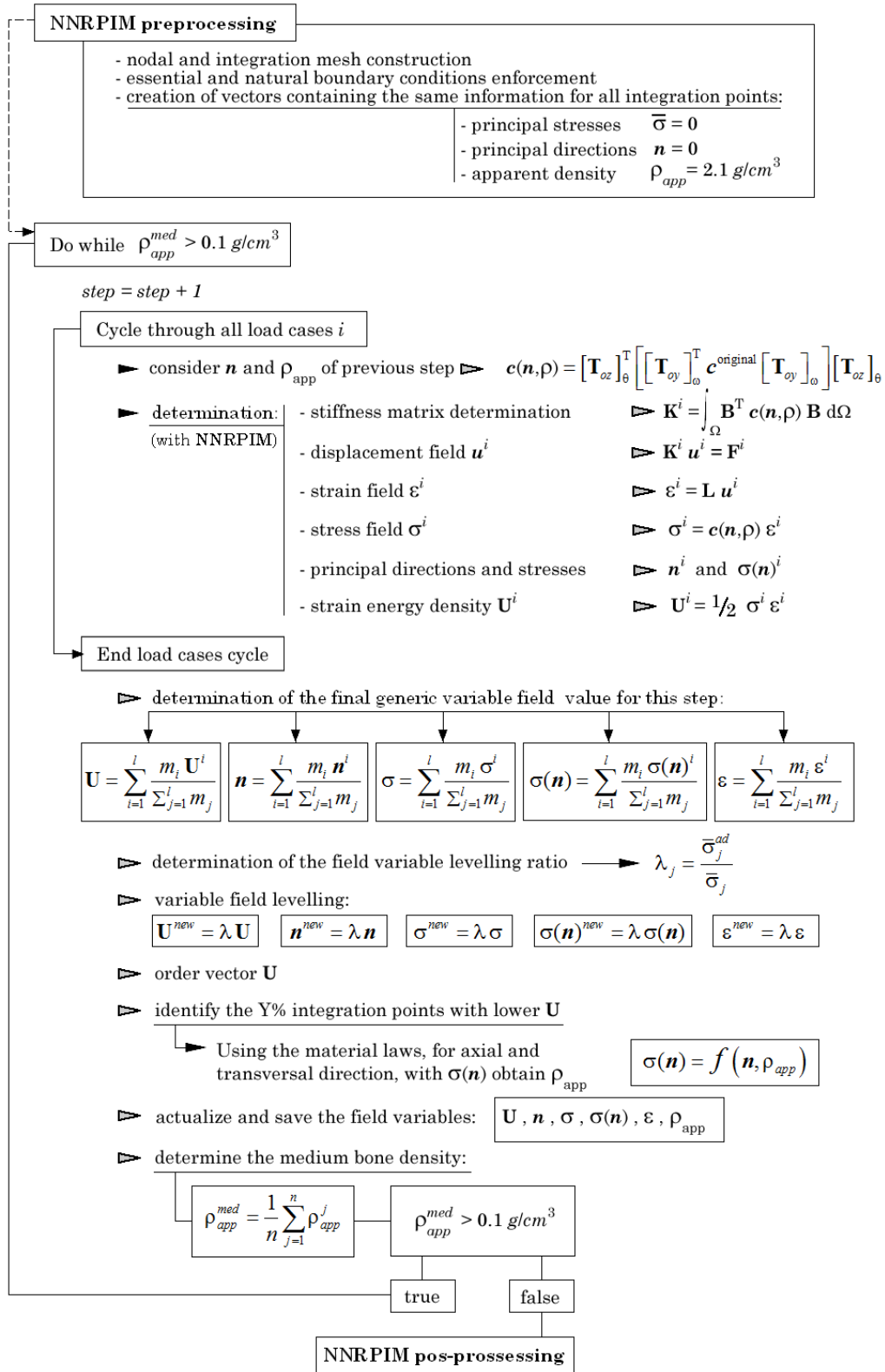
where $\mathbf{T}$ is the rotational transformation matrix defined in equations (40), (41) and (42). The angles θ and ω are related with the principal direction vector for the maximum principal stress, as it is shown in figure 4.

In this work, the presented remodelling algorithm is in fact a topology optimization based model for bone adaptation. In each step not all the interest points optimize the density in relation to the mechanical stimulus. Although for every interest point the material is oriented with the principal directions obtained in each iteration step, the density on each interest point

is not. Only the interest points with lower SED are subject to the density remodelling process, all the others maintain the previous density. With this approach the material properties orientation is continuously optimized and only a small fraction of bone material have its density actualized each time. The process stops when the medium bone density reaches a controlled value of $\rho_{app}^{med} = 0.1 g / cm^3$. The medium bone density is calculated with equation (251).

$$\rho_{app}^{med} = \frac{1}{n} \sum_{j=1}^n \rho_{app}^j \quad (251)$$

In this equation n represents the total number of interest points and ρ_{app}^j is the apparent density on interest point j . The used iterative process is the forward Euler scheme with the particular adaptations to suit the bone internal remodelling analysis. A description of the algorithm is presented in figure 157.



note: the symbol (►) represents a cycle through all integration points j

figure 157 – Proposed bone remodelling algorithm.

Chapter 7

7 Mechanobiology Examples

In this chapter the proposed bone remodelling algorithm, using the NNRPIM, is applied to several problems. First, a two-dimensional benchmark example is used to validate the bone trabecular remodelling. In this example distinct material laws are studied, as well as the influence of the model mesh discretization and material anisotropy. Next, the study is extended to the three-dimensional field, where a test problem based in another benchmark example is presented. Afterwards, the calcaneus bone is simulated using a two-dimensional approach, in this example the obtained trabecular bone architecture was in very good agreement with the ones found in calcaneus bone X-ray images. The same quality results were found in the two-dimensional approach of the femur example. Ending this section a three-dimensional analysis of the femur is presented.

7.1 The Plate Model

In this example a unit square plate with the dimensions indicated in figure 158(a) is studied. This is a benchmark example [151, 155, 159] used to validate the bone trabecular remodelling algorithms. Two types of meshes were used, a regular mesh (RM) and an irregular mesh (IM), both indicated in figure 158. The plate is subjected to a compressive stress distribution, decreasing linearly over the top edge. The node displacement is constrained for $y=0$ only in the y direction and in the origin is constrained in both x and y directions. The natural and essential boundary conditions are presented in figure 158. In the calculations, an uniform initial density distribution $\rho_{app}^{max} = 2.1g/cm^3$ is considered. In all examples the value of $\nu = 0.3$ for the Poisson ration, regardless the direction, is used.

Firstly it was studied the influence of the percentage of interest points being remodelled in each iteration. As so, using the regular mesh indicated in

figure 158 and considering an isotropic material the bone square patch was analysed. The material law used to obtain the bone material properties was the one proposed in section 6.2.2. The material properties in the transverse direction assume the correspondent axial values in order to respect the isotropic material assumption.

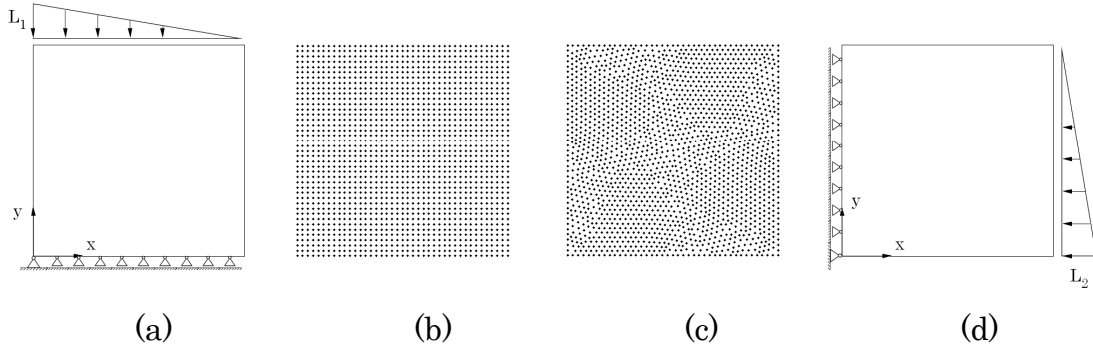


figure 158 – (a) Plate model geometry and essential and natural boundary conditions. (b) Regular nodal mesh (1681 nodes). (c) Irregular nodal mesh (1952 nodes). (d) Second load case considered in the analysis.

In figure 159 it is presented the evolution of the bone apparent density when 10% of interest points are being remodelled in each iteration. In this figure, and in all other figures of the same type, the white colour represents the considered maximum apparent density $\rho_{app}^{med} = 2.1 \text{ g/cm}^3$ and the dark-gray colour represent the minimum apparent density $\rho_{app}^{med} = 0.1 \text{ g/cm}^3$ admitted. All the other gray tones in the middle represent transitional apparent densities.

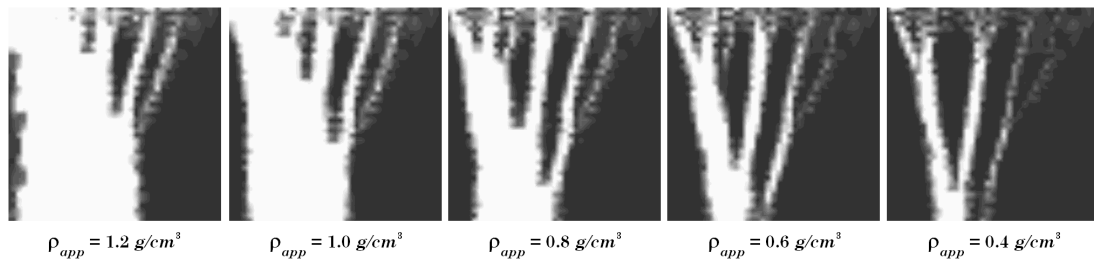


figure 159 – Evolution of the apparent density in the square bone patch when 10% of interest points are remodelled in each iteration. (proposed material law, regular mesh and isotropic material properties)

In figure 160 it is presented the evolution of trabecular architecture if a 5% of interest points are remodelled in each iteration. The results concerning the 2% and 1% percentages are shown in figure 161 and figure 162. The results shown in figure 162 are in very good agreement with the ones shown in [151, 155, 159], also obtained using isotropic materials. Nevertheless the results obtained with remodelling 10% of interest points in each iteration are not faraway from what should be the expected trabecular

rearrangement, for this load case. Therefore, in further studies these four percentages cases will be considered until a consistent conclusion can be made.

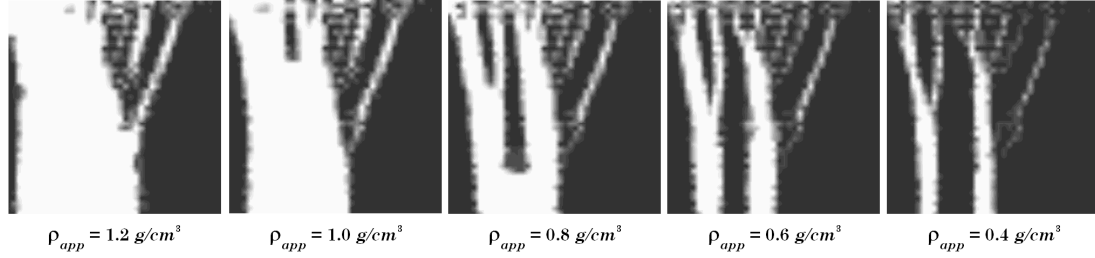


figure 160 – Evolution of the apparent density in the square bone patch when 5% of interest points are remodelled in each iteration. (proposed material law, regular mesh and isotropic material properties)

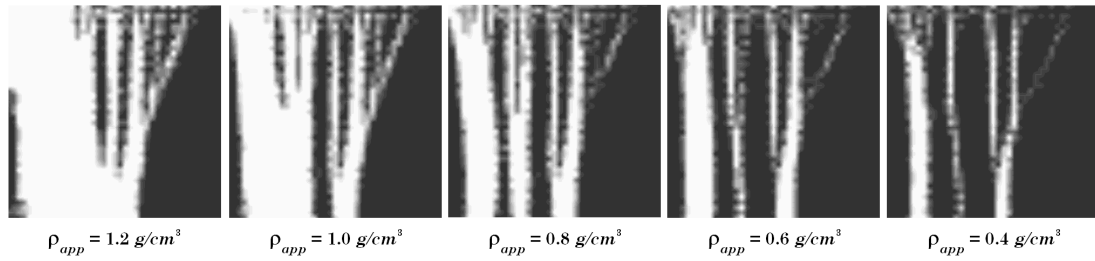


figure 161 – Evolution of the apparent density in the square bone patch when 2% of interest points are remodelled in each iteration. (proposed material law, regular mesh and isotropic material properties)

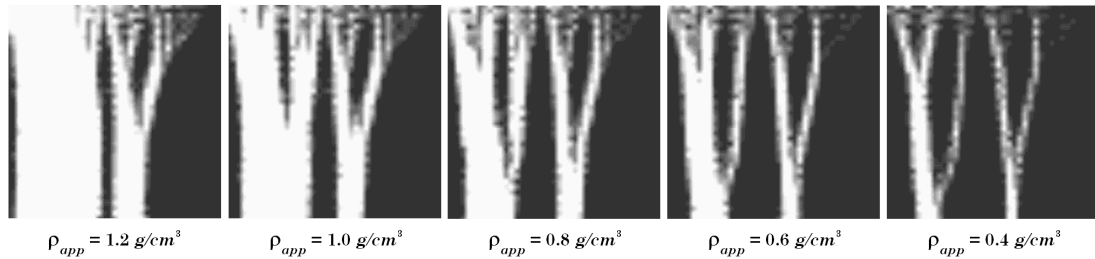


figure 162 – Evolution of the apparent density in the square bone patch when 1% of interest points are remodelled in each iteration. (proposed material law, regular mesh and isotropic material properties)

The next study is related with the introduction of anisotropy in bone material properties. The material law proposed in section 6.2.2 is fully tested, the axial and the transverse directions are distinctly considered. The results considering the distinct percentage of interest points remodelled in each iteration are presented from figure 163 to figure 166. From these results it is possible to observe that the evolution of trabecular architecture is different from the trabecular evolution when an isotropic material is considered. However it seems to be a convergent process. Diminishing the

percentage of interest points remodelled in each iteration approaches solutions.

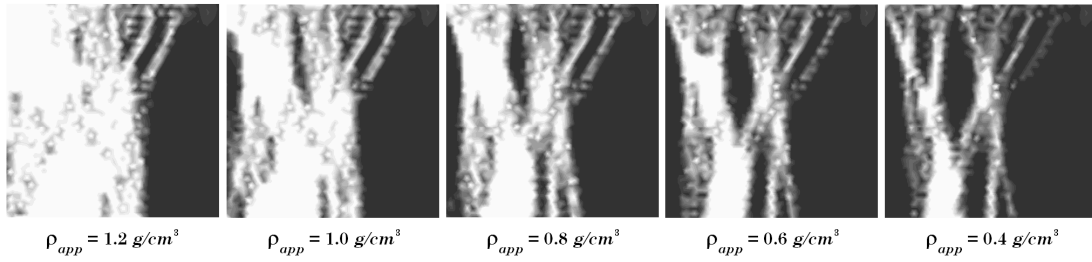


figure 163 – Evolution of the apparent density in the square bone patch when 10% of interest points are remodelled in each iteration. (proposed material law, regular mesh and anisotropic material properties)

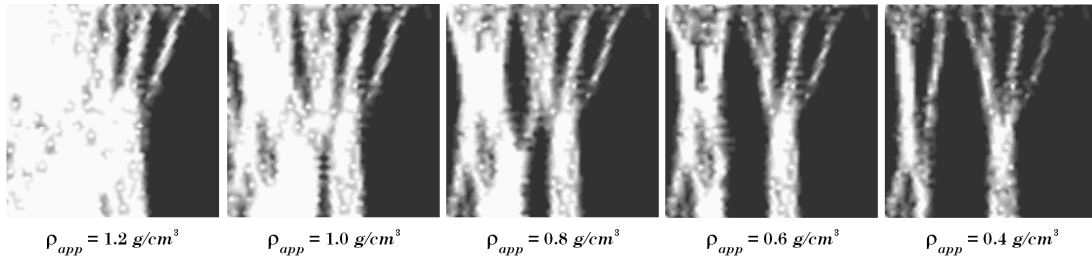


figure 164 – Evolution of the apparent density in the square bone patch when 5% of interest points are remodelled in each iteration. (proposed material law, regular mesh and anisotropic material properties)

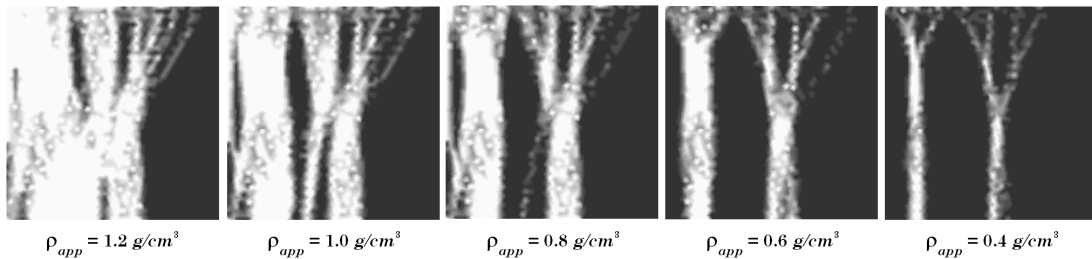


figure 165 – Evolution of the apparent density in the square bone patch when 2% of interest points are remodelled in each iteration. (proposed material law, regular mesh and anisotropic material properties)

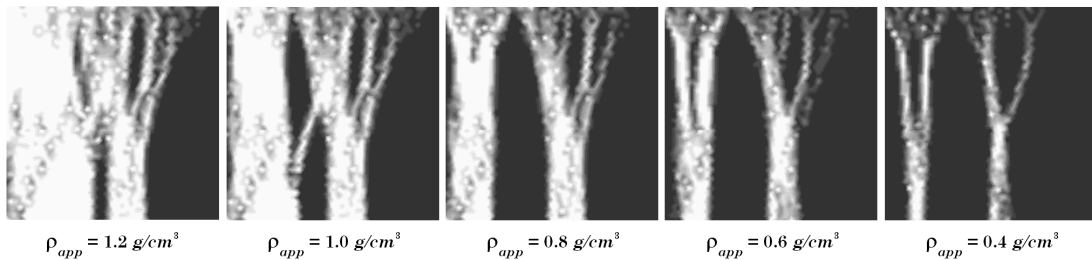


figure 166 – Evolution of the apparent density in the square bone patch when 1% of interest points are remodelled in each iteration. (proposed material law, regular mesh and anisotropic material properties)

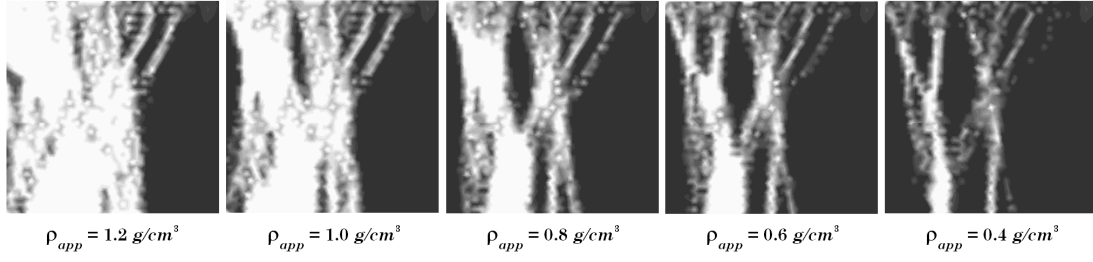


figure 167 – Evolution of the apparent density in the square bone patch when 10% of interest points are remodelled in each iteration. (proposed material law, irregular mesh and anisotropic material properties)

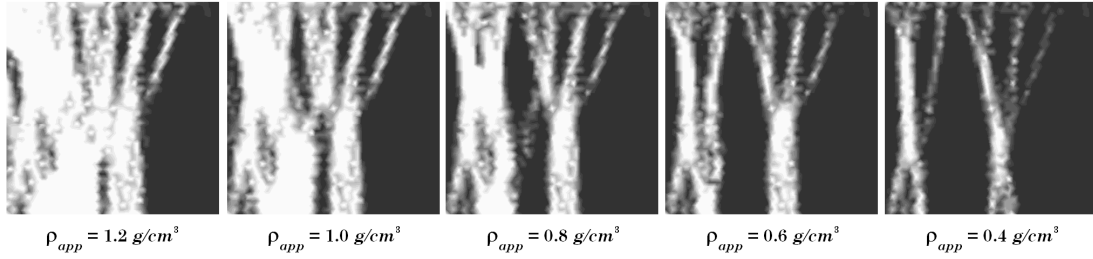


figure 168 – Evolution of the apparent density in the square bone patch when 5% of interest points are remodelled in each iteration. (proposed material law, irregular mesh and anisotropic material properties)

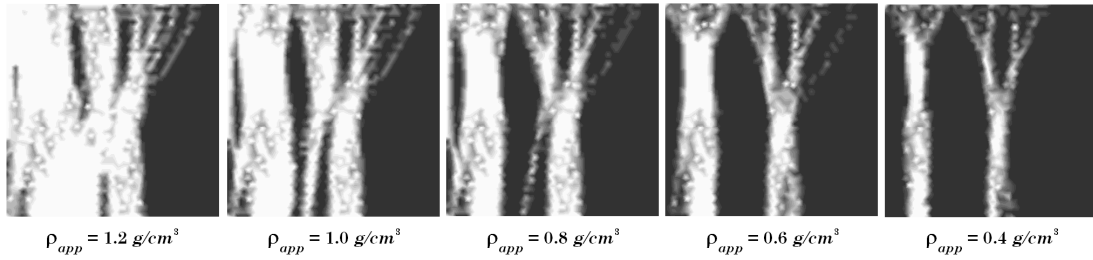


figure 169 – Evolution of the apparent density in the square bone patch when 2% of interest points are remodelled in each iteration. (proposed material law, irregular mesh and anisotropic material properties)

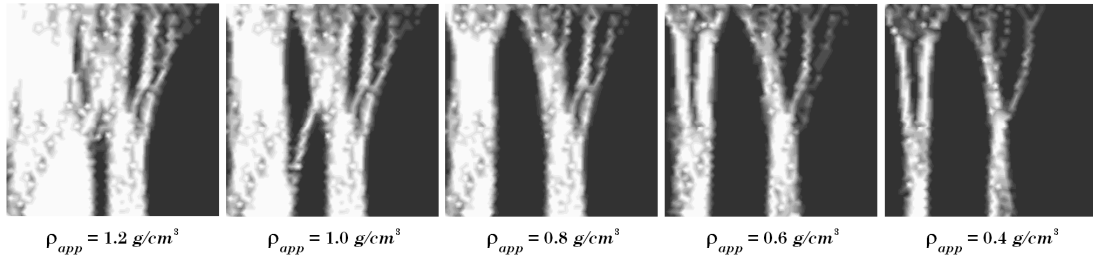


figure 170 – Evolution of the apparent density in the square bone patch when 1% of interest points are remodelled in each iteration. (proposed material law, irregular mesh and anisotropic material properties)

The influence of the used nodal mesh was also studied. The irregular nodal mesh shown in figure 158 is considered in the analysis. The material is

considered anisotropic following the law proposed in section 6.2.2. The results are presented from figure 167 to figure 170.

A closer look shows a remarkable similarity between the results obtained with the regular mesh and the irregular mesh, proving that the nodal arrangement of the used node mesh does not influence the bone remodelling process. In order to validate the material law proposed in this work, the same problem was analysed using the Lotz material law considering the bone as an anisotropic material, section 6.2.1. The results using the Lotz anisotropic material law, considering the regular nodal mesh presented in figure 158, are shown from figure 171 to figure 174.

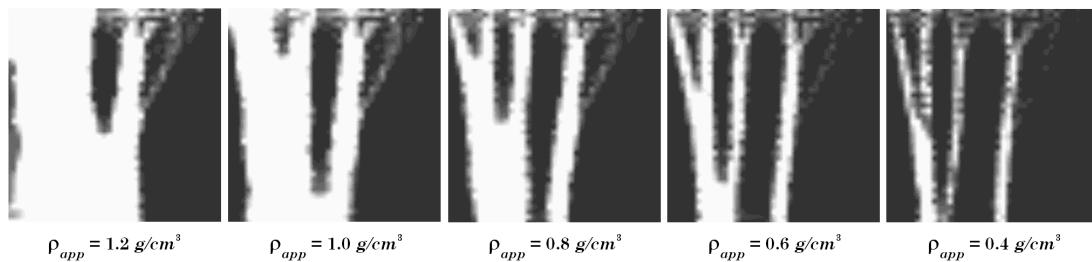


figure 171 – Evolution of the apparent density in the square bone patch when 10% of interest points are remodelled in each iteration. (Lotz material law, regular mesh and anisotropic material properties)

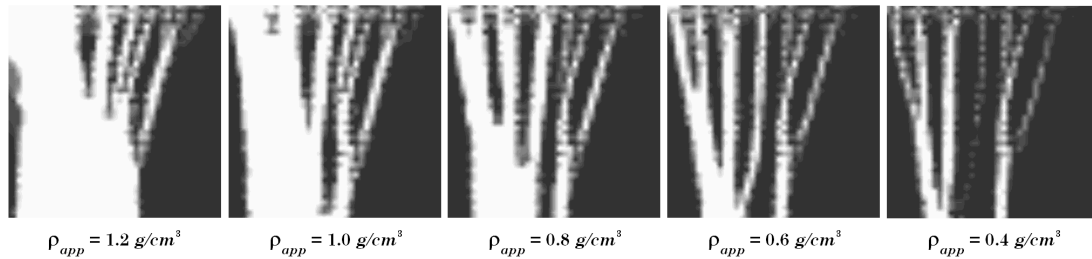


figure 172 – Evolution of the apparent density in the square bone patch when 5% of interest points are remodelled in each iteration. (Lotz material law, regular mesh and anisotropic material properties)

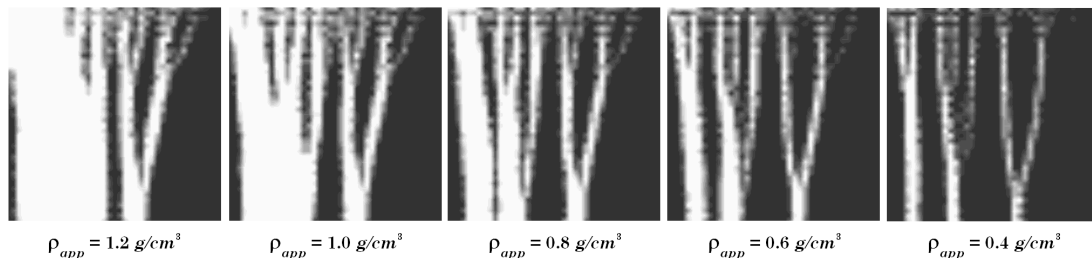


figure 173 – Evolution of the apparent density in the square bone patch when 2% of interest points are remodelled in each iteration. (Lotz material law, regular mesh and anisotropic material properties)

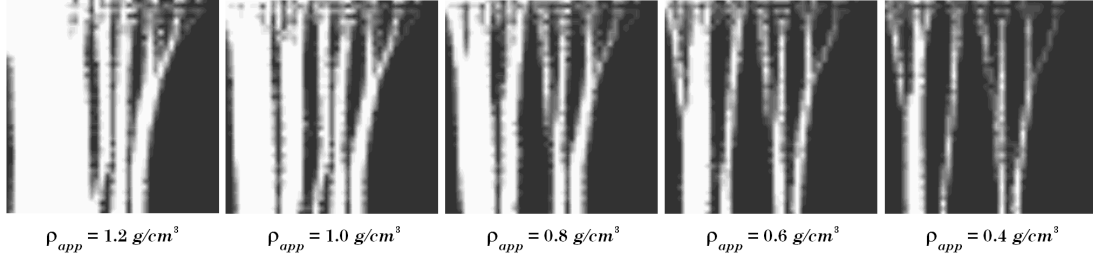


figure 174 – Evolution of the apparent density in the square bone patch when 1% of interest points are remodelled in each iteration. (Lotz material law, regular mesh and anisotropic material properties)

The results regarding the irregular mesh, figure 158, for the same material conditions are shown from figure 175 to figure 178.

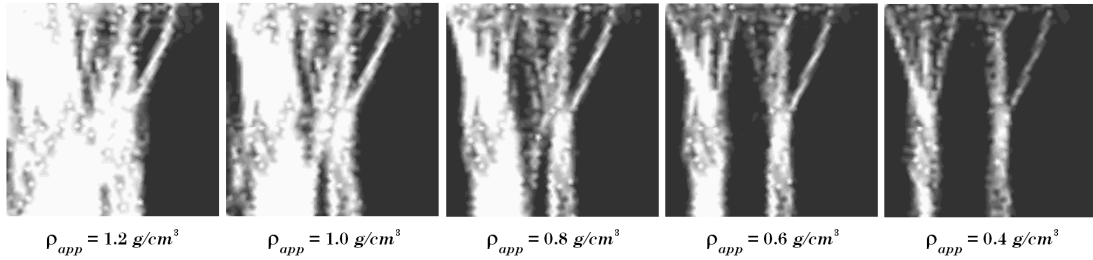


figure 175 – Evolution of the apparent density in the square bone patch when 10% of interest points are remodelled in each iteration. (Lotz material law, irregular mesh and anisotropic material properties)

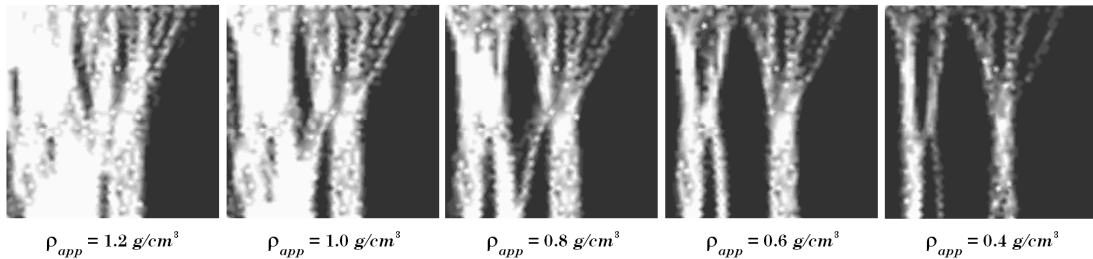


figure 176 – Evolution of the apparent density in the square bone patch when 5% of interest points are remodelled in each iteration. (Lotz material law, irregular mesh and anisotropic material properties)

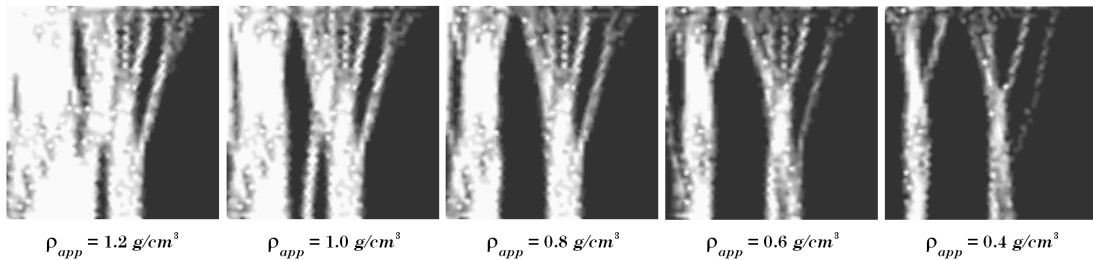


figure 177 – Evolution of the apparent density in the square bone patch when 2% of interest points are remodelled in each iteration. (Lotz material law, irregular mesh and anisotropic material properties)

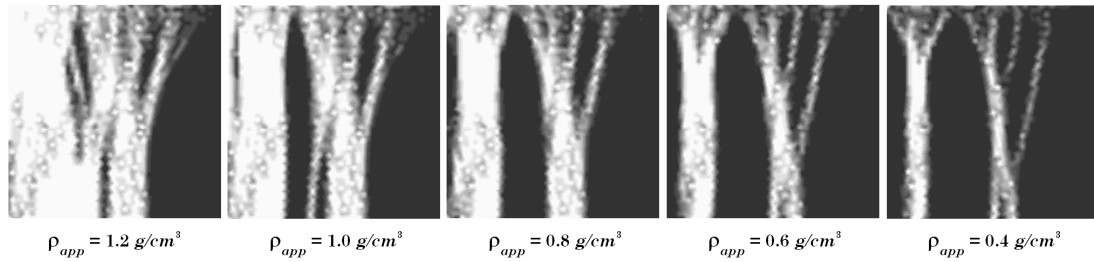


figure 178 – Evolution of the apparent density in the square bone patch when 1% of interest points are remodelled in each iteration. (Lotz material law, irregular mesh and anisotropic material properties)

In the case of the Lotz material law, it is clear the difference between the results of the analysis with the regular mesh and the obtained solution with the use of an irregular mesh. Only the material law was changed, everything else remains equal. In Lotz law it was postulated that whenever an interest point shows an apparent density $\rho_{app} > 1.3 \text{ g/cm}^3$ it is considered in the analysis the cortical material law, and for $\rho_{app} \leq 1.3 \text{ g/cm}^3$ it is considered the trabecular material law. It is not a continuous process, as it is in the proposed material law. In Lotz material law when an interest point leaves the cortical law and passes to the trabecular law a huge leap in the material laws occurs (notice in figure 153 to figure 156 for the Lotz material law the difference between the material properties of the cortical bone and the trabecular bone). This maybe in the origin of the difference between the results obtained with the regular and the irregular mesh.

In order to validate the remodelling algorithm when distinct load cases are considered, the plate was subjected to two individual loads. A load case L1, figure 158(a), and a load case L2, figure 158(d). In both cases the sample load has the same magnitude. In a first approach load case L1 was applied with 1000 cycles and load case L2 with also 1000 cycles. Only the proposed anisotropic material law in section 6.2.2 was considered and 1% of interest points were remodelled in each iteration. The problem was analysed with a regular mesh, figure 158(b), and an irregular mesh, figure 158(c). The results are shown in figure 179 and figure 180.

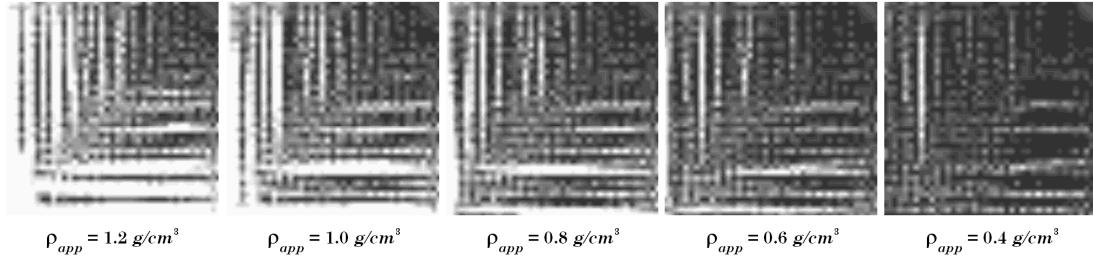


figure 179 – Evolution of the apparent density in the square bone patch with a regular nodal mesh for the two load cases: L1 with 1000 cycles and L2 with 1000 cycles.

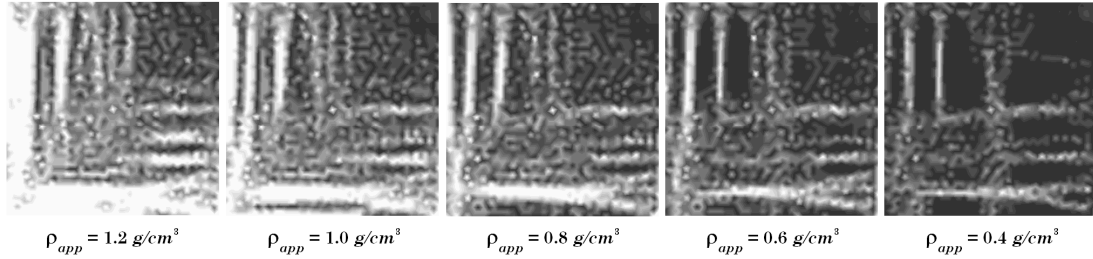


figure 180 – Evolution of the apparent density in the square bone patch with a irregular nodal mesh for the two load cases: L1 with 1000 cycles and L2 with 1000 cycles.

In figure 179, the regular mesh results, the expected symmetry is clear, however the results from the analysis using an irregular mesh, figure 180, do not show a so evident symmetry. Nonetheless, for the irregular mesh analysis it is perceptible the existence of a similar bone apparent density in both directions (x and y).

A final test regarding the square plate example was conducted. The two load cases already referred were now applied with the following condition: load case L1 was applied with 1000 cycles and load case L2 with 5000 cycles. All the other variables remain the same as in previous study. The results are shown in figure 181 and figure 182.

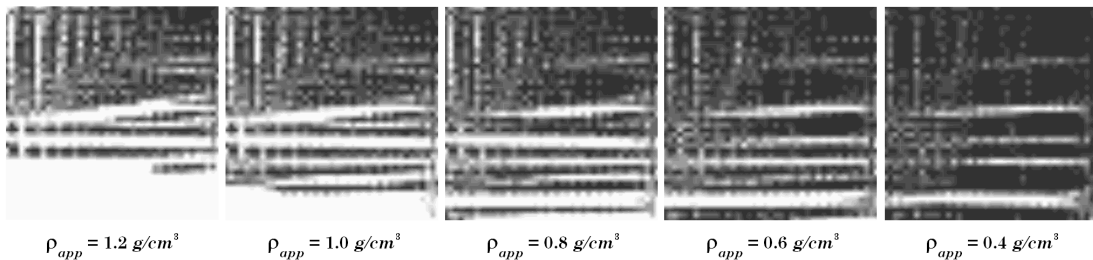


figure 181 – Evolution of the apparent density in the square bone patch with a regular nodal mesh for the two load cases: L1 with 1000 cycles and L2 with 5000 cycles.

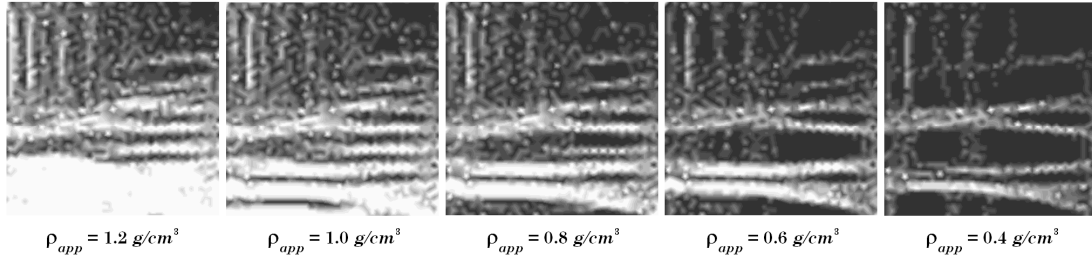


figure 182 – Evolution of the apparent density in the square bone patch with a irregular nodal mesh for the two load cases: L1 with 1000 cycles and L2 with 5000 cycles.

In both figures it is obvious the trabeculae structural design preference. The trabeculae developed towards the higher load, in the y direction. The results on this benchmark example have shown the capacity of the proposed remodelling algorithm, combined with the proposed new material law and the NNRPIM, to predict the internal bone trabecular architecture.

7.2 The Cubic Model

In [165] it is studied the trabecular regeneration process and cubic bone patches are analysed to verify the proposed fuzzy-logic algorithm. In the present work similar cubic patches are studied. Consider the patch presented in figure 183(a), with a $2 \times 1 \times 2 \text{ mm}^2$. A surface load $F = 1.0 \text{ N/m}^2$, with the direction indicated in the figure, is applied in two square areas on the top of the cubic patch. On the patch bottom another two square areas locally constrain the patch movement in all directions. The problem is analysed considering the regular mesh, with 2681 nodes, presented in figure 183(c). For all studied examples in this section, an initial density distribution $\rho_{app}^{max} = 2.1 \text{ g/cm}^3$ is assumed, with a Poisson ration $\nu = 0.3$, regardless the material direction, and 1% of interest points are remodelled in each iteration. The anisotropic material law presented in section 6.2.2 is considered. In order to present the results, the analysed cubic patch is cut by the section presented in figure 183(d).

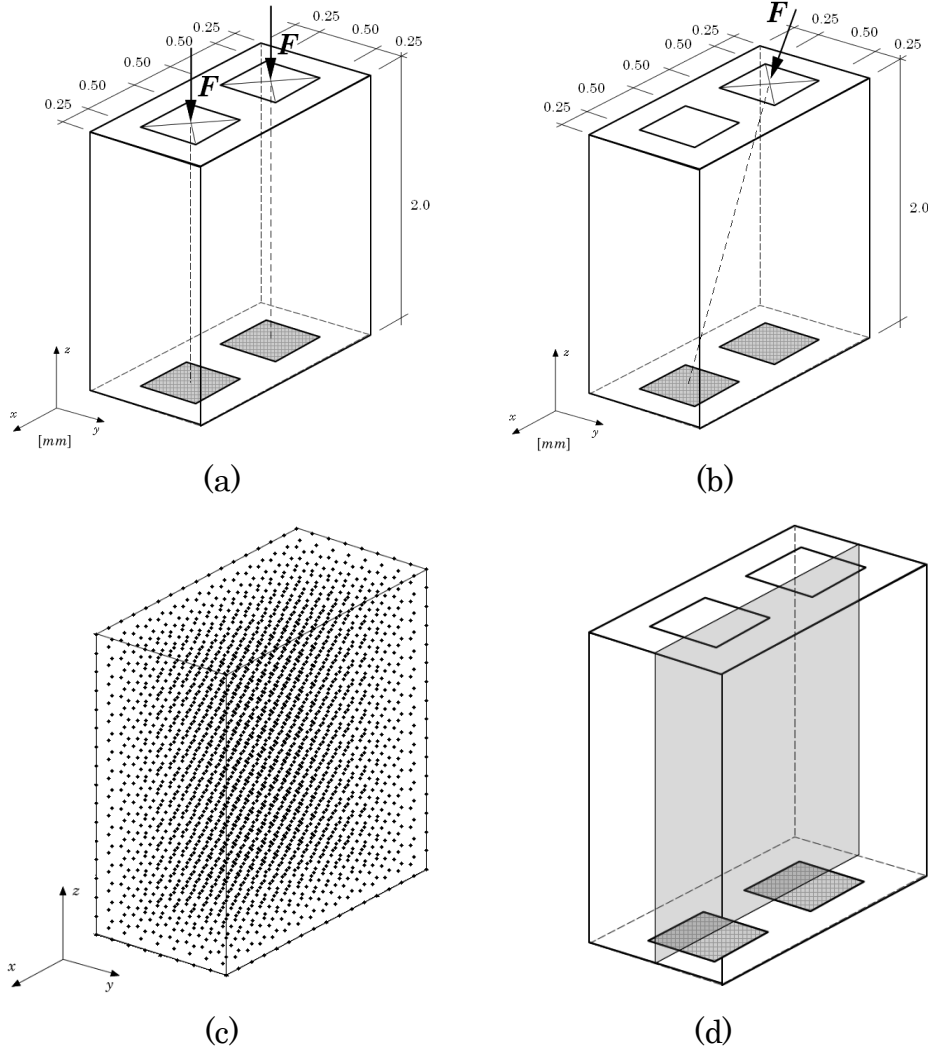


figure 183 – (a) Cubic patch model submitted to vertical loads. (b) Cubic patch model submitted to a diagonal load. (c) Regular nodal mesh (2681 nodes). (d) Patch section cut.

In figure 184 is presented the evolution of the trabecular bone remodelling process, where the white colour represents the considered maximum apparent density $\rho_{app}^{med} = 2.1g/cm^3$ and the dark-gray colour represent the minimum apparent density $\rho_{app}^{med} = 0.1g/cm^3$ admitted. All the other gray tones in the middle represent transitional apparent densities. As expected, the applied loads lead the bone to build vertical trabeculae.

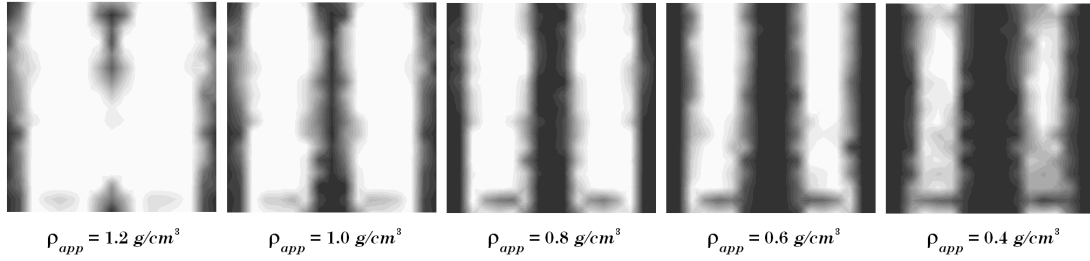


figure 184 – Evolution of the apparent density in the bone cubic patch for vertical loads.

In order to verify the influence of the nodal mesh discretization, a single diagonal load is considered, as figure 183(b) illustrate. The same essential boundary conditions and material properties are assumed. The results are presented in figure 185. It is visible that evolution of the trabecular bone remodelling process leads to a single diagonal trabecula.

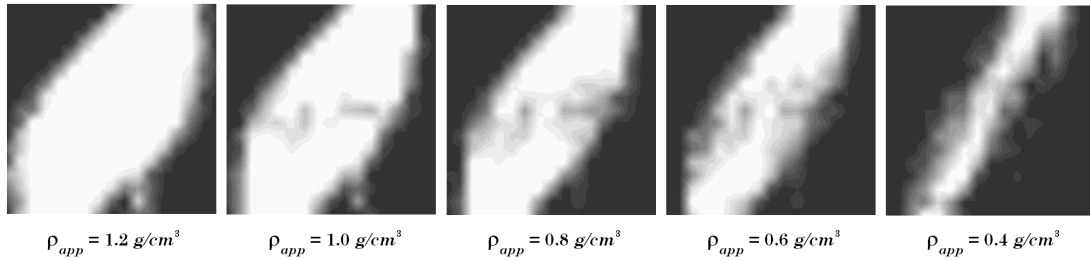


figure 185 – Evolution of the apparent density in the bone cubic patch for diagonal load.

In figure 186 it are presented the three-dimensional section views for both analyses when the apparent density $\rho_{app} = 0.4 \text{ g/cm}^3$ is achieved.

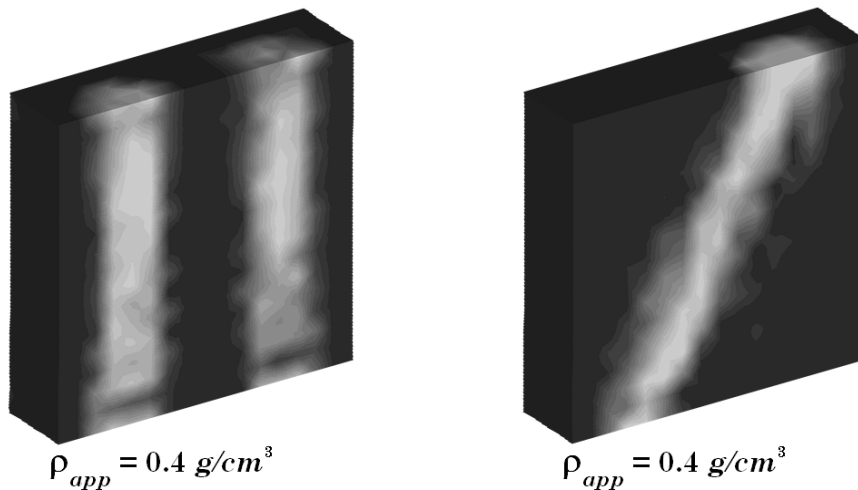


figure 186 – Cubic patch apparent density three-dimensional view.

Notice that the bone seems to form cylindrical trabeculae in the direction of the applied load.

Another cubic example is studied. Consider the patch presented in figure 187(a), with a $2 \times 2 \times 2 \text{ mm}^3$. A surface load $F = 1.0 \text{ N/m}^2$, with the direction indicated in the figure, is diagonal applied in a square area on the top of the cubic patch. On the patch bottom four square areas locally constrain the patch movement in all directions. The problem is analysed considering a regular mesh with 2744 nodes, figure 187(b). The same essential boundary conditions and material properties are assumed. In order to present the results, the analysed cubic patch is cut by the section presented in figure 187(c). The results regarding the evolution of the trabecular architecture are presented in figure 188.

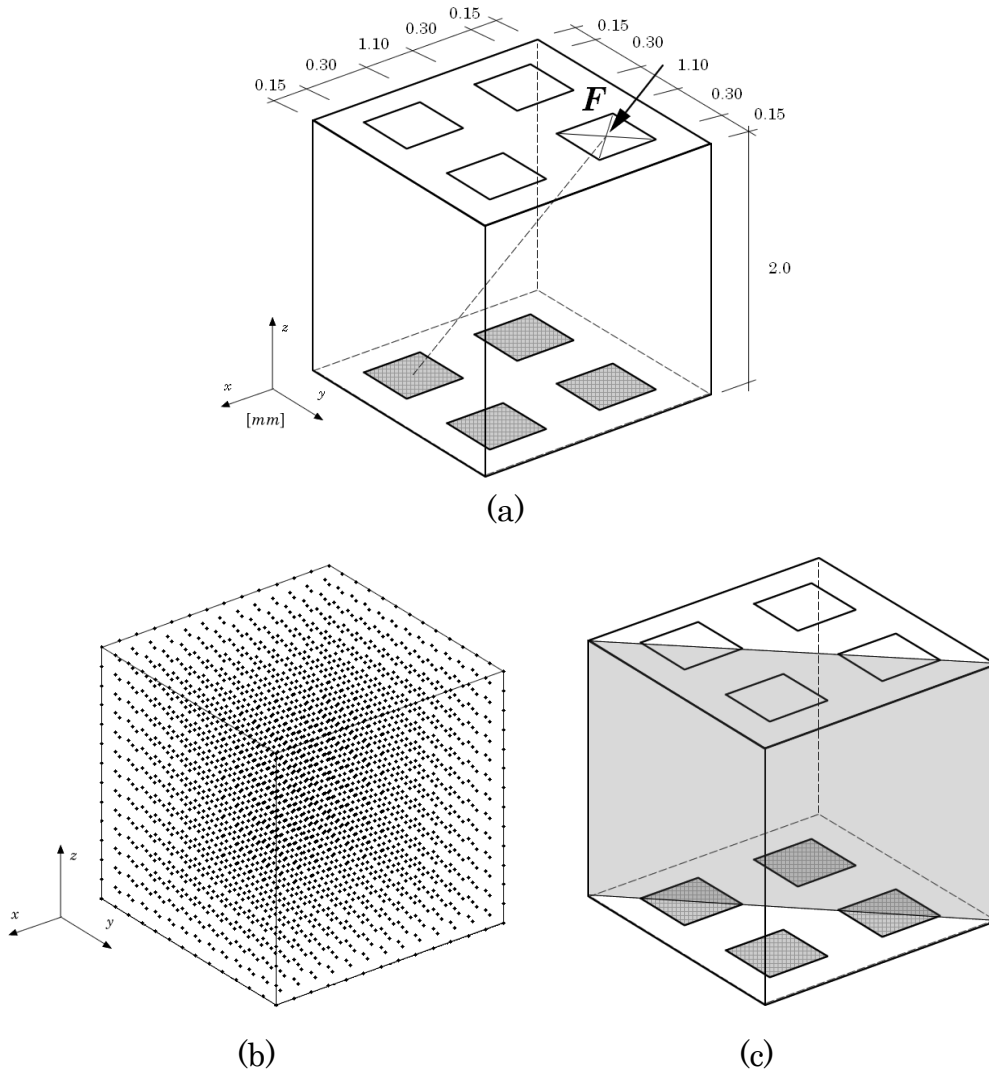


figure 187 – (a) Cubic patch model submitted to a diagonal load. (b) Regular nodal mesh (2744 nodes). (c) Patch section cut.

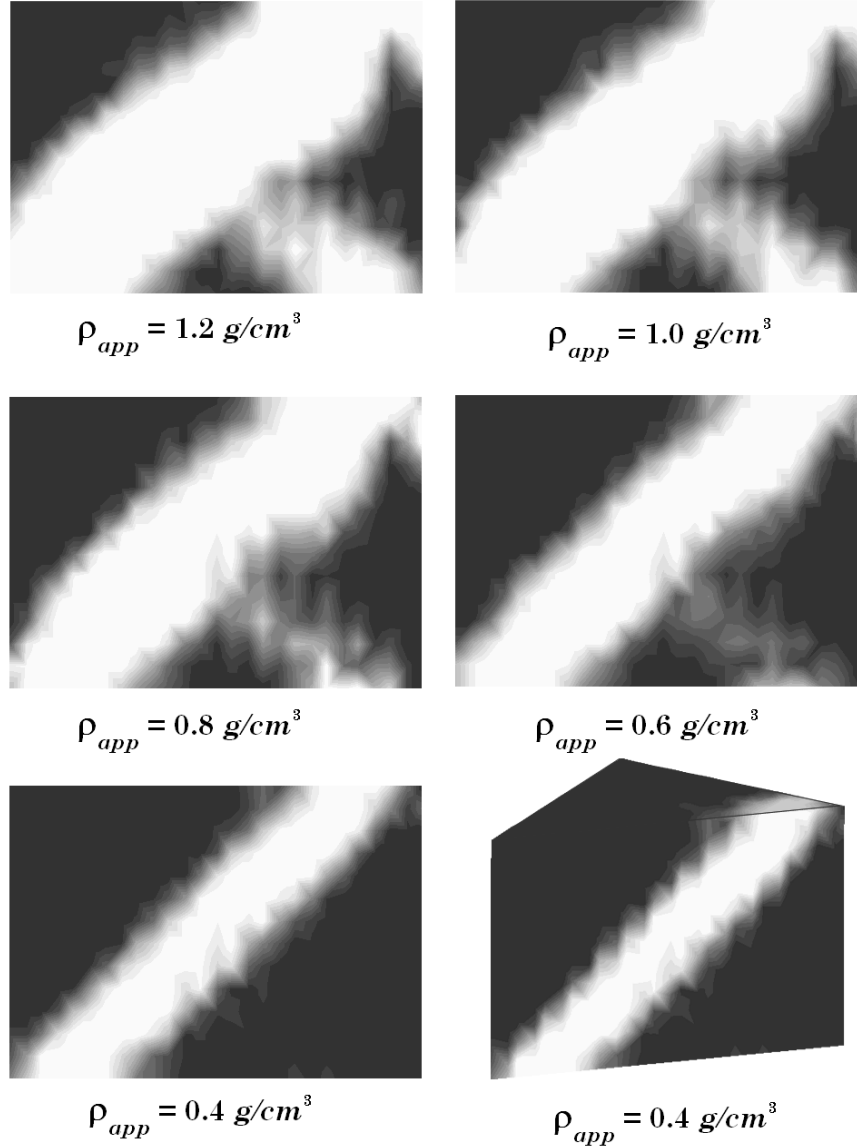


figure 188 – Evolution of the apparent density in the bone cubic patch for diagonal load.

Once again, despite the applied load being not collinear with the mesh distribution, the formed trabecula is perfectly oriented in the load direction. Notice that for apparent densities $\rho_{app} > 0.6 \text{ g/cm}^3$ a secondary trabecula remains in the bone patch. The structural function of this secondary trabecula is to stabilize the principal diagonal trabecula, stopping a possible buckling phenomenon. The manifestation of the secondary trabecula proves that the remodelling algorithm and the proposed material law can predict secondary structures in the trabecular bone medium.

To end this section dedicated to the three-dimensional bone patch analysis, a cubic bone patch with $2 \times 2 \times 2 \text{ mm}^3$ presented in figure 189(a) is studied. Surface loads $F = 1.0 \text{ N/m}^2$ are cross diagonal applied in square areas on the

top of the cubic patch. On the patch bottom four square areas locally constrain the patch movement in all directions. The problem is analysed considering the same regular mesh presented in figure 187(b). This example stimulates in the bone patch torsion effects, which the bone must resist with a suitable trabecular structure. In order to observe the internal bone reorganization four cuts are made in the analysed cubic patch, figure 189(b).

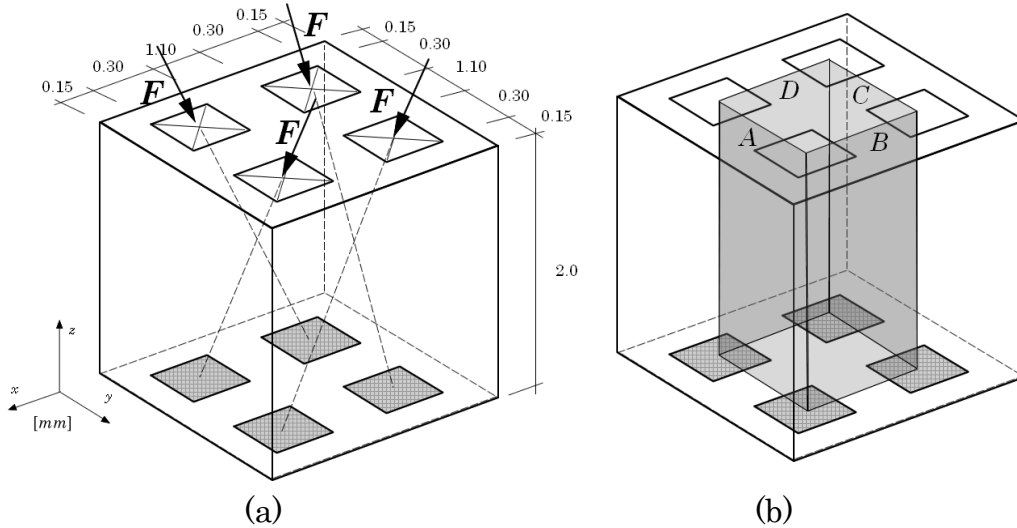


figure 189 – (a) Cubic patch model submitted to cross diagonal loads. (b) Patch sections cuts.

First the patch was analysed considering the bone as an isotropic material. The material law used to obtain the bone material properties was the one proposed in section 6.2.2. The material properties in the transverse direction assume the correspondent axial values in order to respect the isotropic material assumption. The results on the evolution of the parent density in the bone cubic patch for the cross load are presented in figure 190. The results of figure 190 show that the bone forms symmetric diagonal trabeculae to resist the torsion effect produced by the applied load. Secondary trabeculae do not appear as evident structures. Therefore the study continues and the cubic patch was analysed considering the bone as an anisotropic material using the material law proposed in section 6.2.2. The results are shown in figure 191. Notice that in the vertical direction the appearance of vertical secondary trabecular structures. These results are corroborated with the solution obtained when the Lotz anisotropic material law is considered, figure 192.

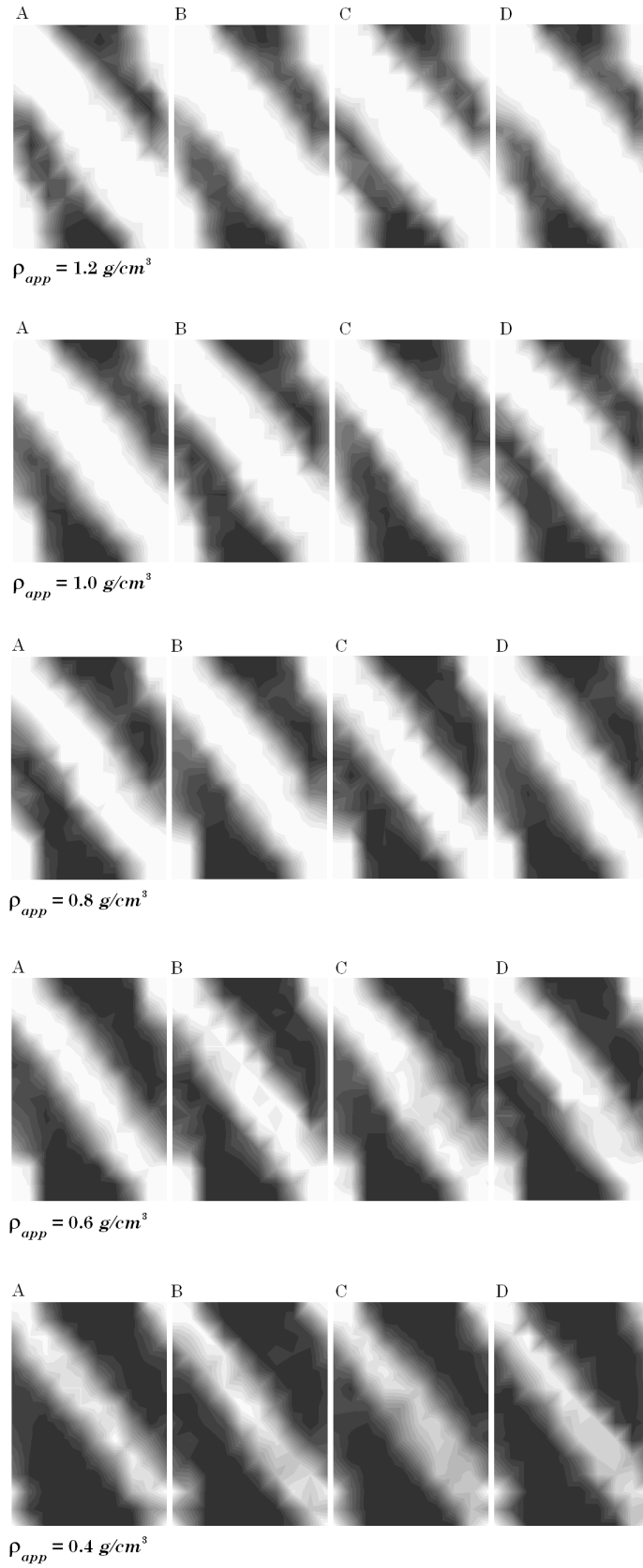


figure 190 – Evolution of the apparent density in the bone cubic patch for the cross diagonal loads. (isotropic material considering the proposed material law).

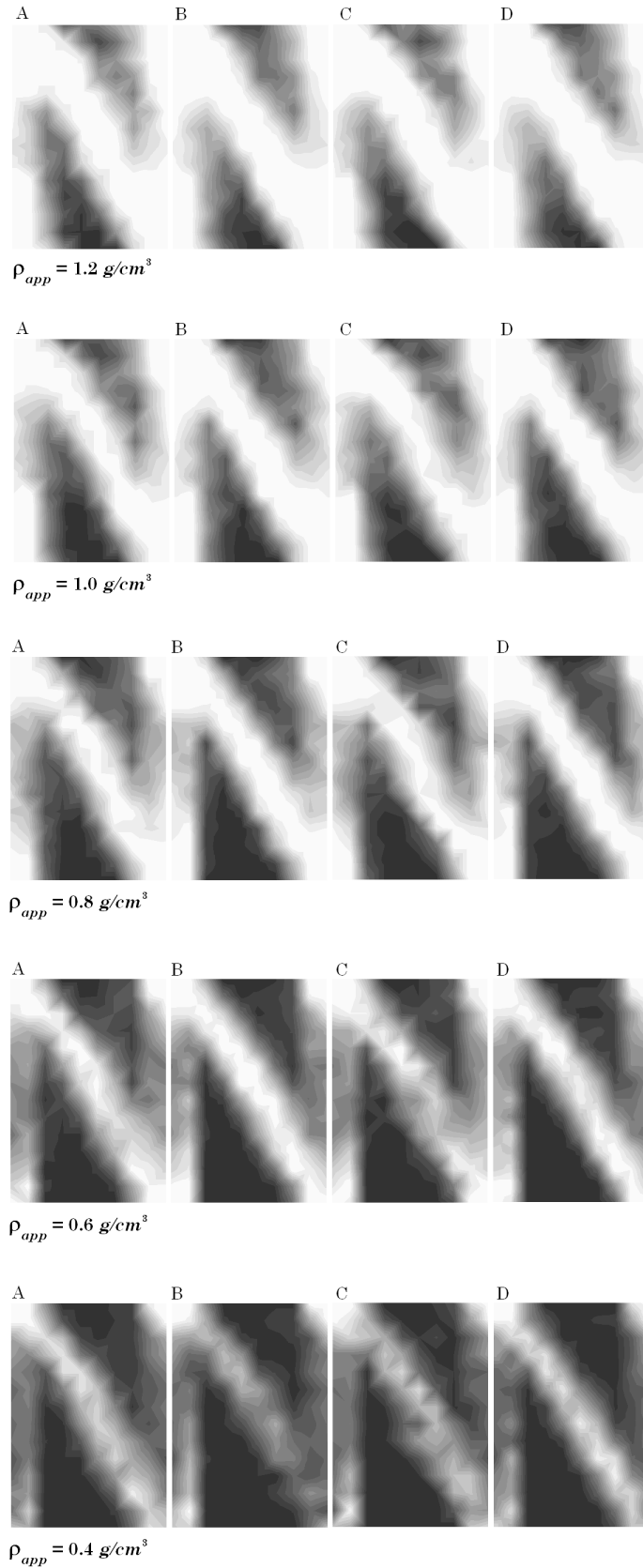


figure 191 – Evolution of the apparent density in the bone cubic patch for the cross diagonal loads. (anisotropic material considering the proposed material law).

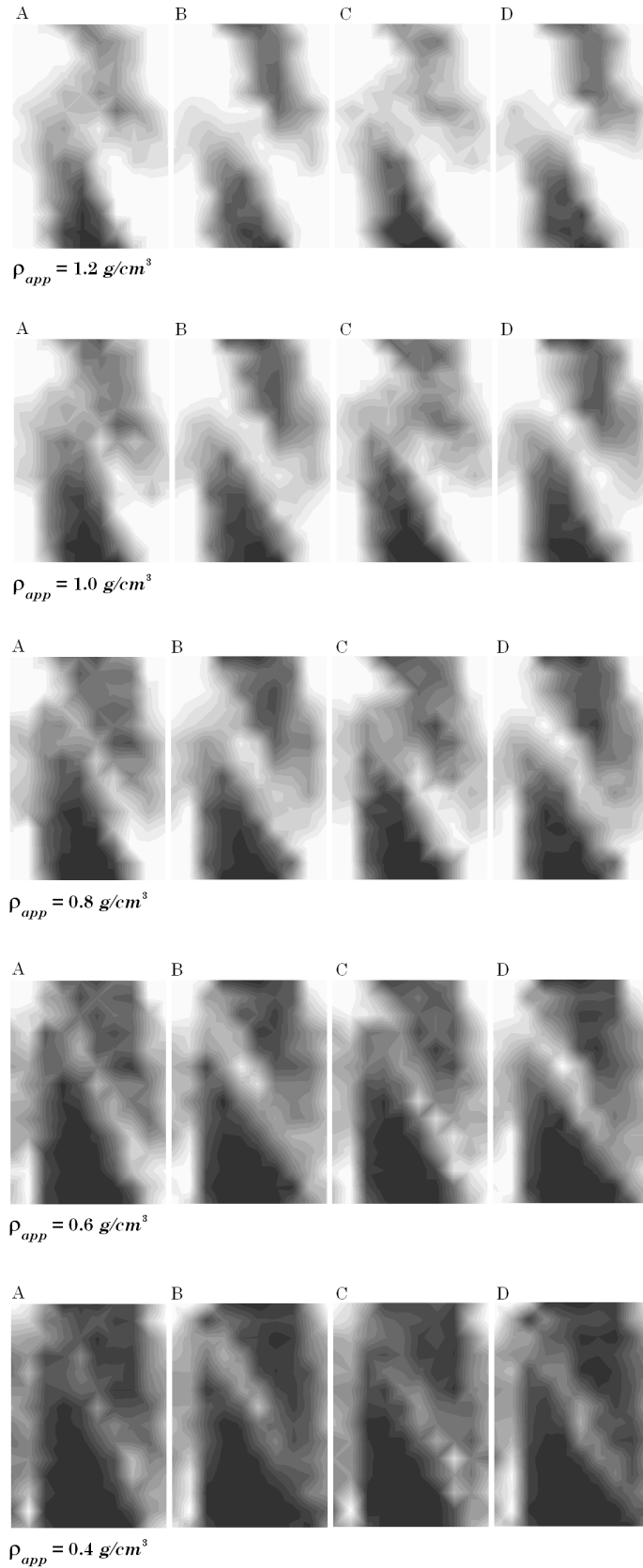


figure 192 – Evolution of the apparent density in the bone cubic patch for the cross diagonal loads. (anisotropic material considering Lotz material law).

The same vertical secondary structures appear when the Lotz anisotropic material law is considered. This example proved that in the bone trabecular remodelling process it is important to consider the bone as an anisotropic material. The secondary trabecular structures confer to the trabecular bone mesh a higher stability and resistance. The manifestation of such structures indicates that the present study on bone remodelling is moving forward on the right path.

7.3 The Calcaneus Bone

In the human species the foot is the first mechanical contact with the ground. The calcaneus is one of the many bones composing the foot structure. Its function is to distribute the load from above. The body weight is transmitted by the talus bone, which is loaded directly by the tibia bone. The cuboid bone articulates with its anterior side, the Achilles tendon is inserted into a roughened area on its superior side and the plantar fascia fan-tendon is inserted on its bottom. In figure 193(a) it is possible to visualize a foot x-ray plate where foot bones are indicated. A closer look on the calcaneus bone permit to empirically determine the compressive and tensile lines indicated in figure 193(b). The foot is a dynamic structure where the several bones composing it suffer a constant shift of load cases. Therefore the foot bones are forced to achieve an internal trabecular architecture which permits to resist the several applied load cases. The compressive and tensile lines empirically obtained, and indicated in figure 193(b), are the product of the remodelling process in the calcaneus bone.

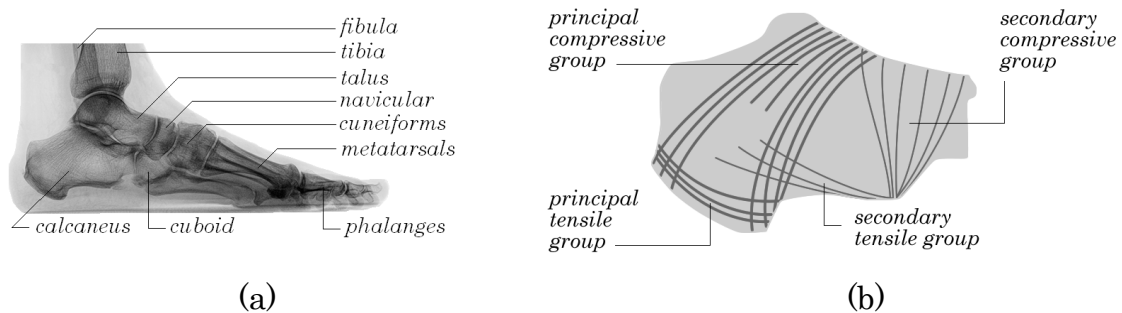


figure 193 – (a) Foot x-ray plate. (b) Calcaneus internal principal trabecular structures.

In the literature there is no sufficient information regarding the several loading conditions to which the foot is submitted. Nevertheless there are some works that indirectly permit to obtain the most important load cases for the calcaneus bone. In [166] a simple biomechanical model of the foot is proposed to analyse the load bearing mechanism, from which is possible to

obtain the principal load contact points on the calcaneus bone. In [167] a finite element model for analysis of the structural behaviour of the human foot during standing was utilized to investigate the biomechanical effects of releasing the plantar fascia. In this work several numerical models are analysed and the principal tensile and compressive loads on the calcaneus bone are proposed. In [168] a three-dimensional finite element model of the human foot and ankle, incorporating geometrical and material nonlinearity, was employed to investigate the loading response of the plantar fascia in the standing foot with different magnitudes of Achilles tendon loading. This work supplies important information regarding the relationship between the Achilles tendon tensile force and the plantar fascia tensile force. The same relationship is obtained in [169] but recreating the position of the foot when stretch is introduced on the plantar fascia (foot movement). All these authors agree that the principal tendons inserted in the calcaneus bone are the Achilles tendon, the plantar fascia fan-tendon and the calcaneus-metatarsal tendon. In figure 194 it is possible to schematically visualize the respective tendon tensile forces.

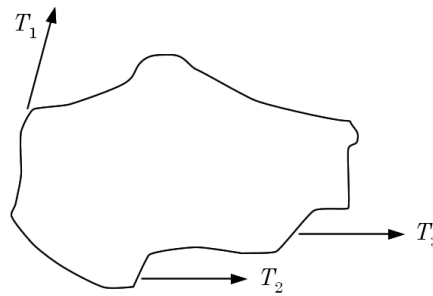


figure 194 – Tendon forces. Achilles tendon force (T_1), plantar fascia fan-tendon force (T_2) and calcaneus-metatarsal tendon force (T_3).

Based on the x-ray plate presented in figure 193(a) a two-dimensional model of the calcaneus bone was constructed. The geometry and the nodal mesh discretizing the problem domain are presented in figure 195.

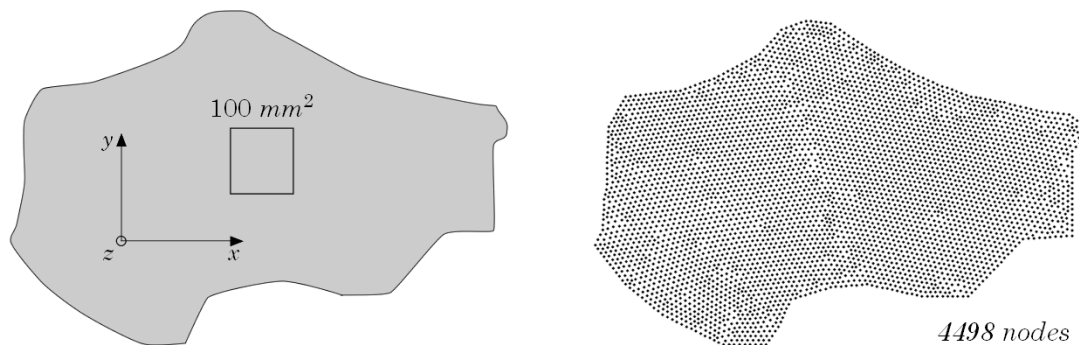


figure 195 – Geometry and nodal mesh used in the analysis of the calcaneus bone.

The human foot is a structure which possesses a robust and relative wide angle articulation, permitting this way diverse load situations. However, two structural mechanical conditions are recurrent: the standing foot and the walking foot. In this work three mechanical cases, based in the previous referred articles [166-169], are proposed. Although the human species possesses two feet it is common to stand only in one of them, and during normal walking merely one foot is standing in the ground. Therefore in this work it is considered a human with a body weight of $P=1000N$ standing always in one foot. In figure 196 it is presented the first mechanical case. This case is due to the arrival of the foot to the ground during the walking movement. When the foot arrives to the floor the first contact is with the calcaneus bone. In this mechanical case the compressive forces are the most relevant. The pressures applied in the distinct natural boundaries have the magnitude, distribution and orientation indicated. The essential boundary condition indicated as ebc_1 constrains the movement on the direction normal to the surface and the ebc_2 essential boundary condition constrains the movement on the x direction.

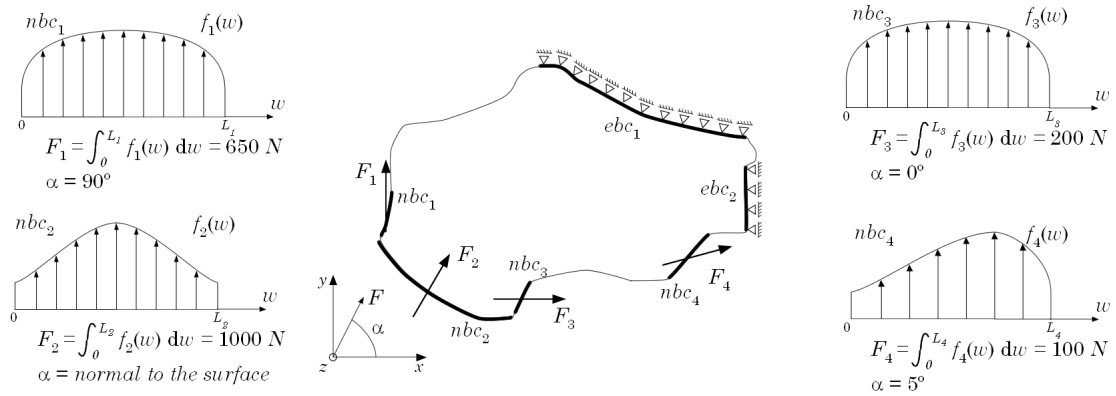


figure 196 – Loads and constrains of the first mechanical case of the calcaneus bone.

The second mechanical case is presented in figure 197. This case corresponds to the foot lifting movement. This movement is used to jump or simply to walk. In this work the magnitudes of the forces involved correspond to the walking movement. The calcaneus bone is submitted to high tensile forces from the Achilles tendon and the plantar fascia tendon. The talus bone distributes most of the compressive forces to the navicular bone and the cuboide bone, constraining the calcaneus bone movements. The pressures applied in the distinct natural boundaries have the magnitude, distribution and orientation indicated in figure 197. The essential boundary condition ebc_1 constrains the movement on the direction normal to the surface, the ebc_2 essential boundary condition constrains the movement on the x direction and ebc_3 on the y direction.

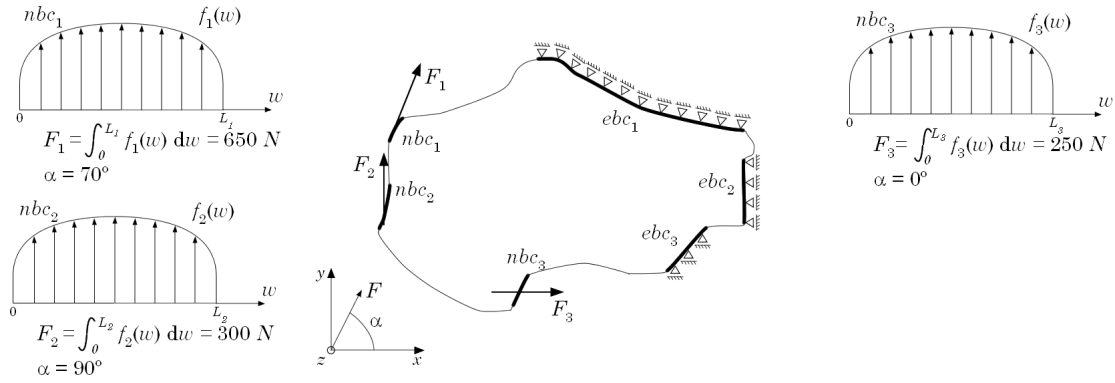


figure 197 – Loads and constraints of the second mechanical case of the calcaneus bone.

In figure 198 the third mechanical case is presented. This case corresponds to the standing position. During the standing position the Achilles tendon is always in tension, providing a dynamic equilibrium to the body. In this mechanical case there is a balance between the compressive forces and the tensile forces applied in the calcaneus. In figure 198 are presented the pressures applied in the distinct natural boundaries and the respective magnitudes, distributions and orientations. The essential boundary conditions ebc_1 and ebc_3 constrains the movement on the direction normal to the surface and the ebc_2 essential boundary condition constrains the movement on the y direction.

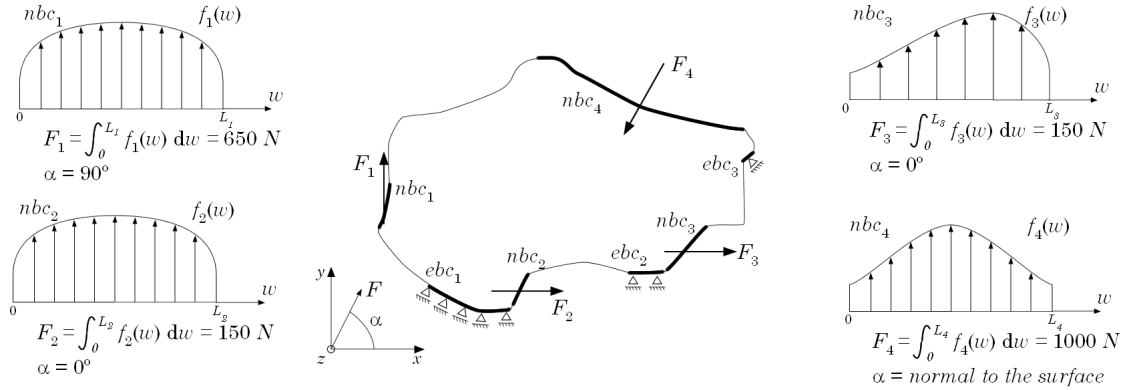


figure 198 – Loads and constraints of the third mechanical case of the calcaneus bone.

In all calcaneus bone mechanical cases studied, an initial density distribution $\rho_{app}^{max} = 2.1 g/cm^3$ is assumed, with a Poisson ration $\nu = 0.3$, regardless the material direction, and 1% of interest points are remodelled in each iteration. The anisotropic material law presented in section 6.2.2 is considered. The evolution of the trabecular architecture in the calcaneus bone is presented in figure 199 for the distinct mechanical cases. The white colour represents the considered maximum apparent density $\rho_{app}^{med} = 2.1 g/cm^3$

and the dark-gray colour represents the minimum apparent density $\rho_{app}^{med} = 0.1 g / cm^3$ admitted. All the other gray tones in the middle represent transitional apparent densities.

The internal trabecular structures indicated in figure 193(b) can be found in the plates of figure 199. These results indicate that it is possible to individually achieve the internal trabecular calcaneus bone structure.

The next phase is to mix the three mechanical cases in order to achieve a closer solution to the x-ray plate initially presented. Therefore it was considered two mix situations. The first consists on applying the three mechanical cases simultaneous with 6000 cycles per day for the first mechanical case, 6000 cycles per day for the second mechanical case and 4200 cycles per day for the third mechanical case. The results are presented in figure 200(a). The second mix situation corresponds to the simultaneous application of the three mechanical cases with 6000 cycles per day for all the three mechanical cases. The results are shown in figure 200(b).

The results show a clear similitude with the x-ray plate presented in figure 193(a) and corroborate the empirical trabecular architecture suggested in figure 193(b). These results indicate that it is possible to achieve the internal trabecular bone structure of a bone if the applied loads are known.

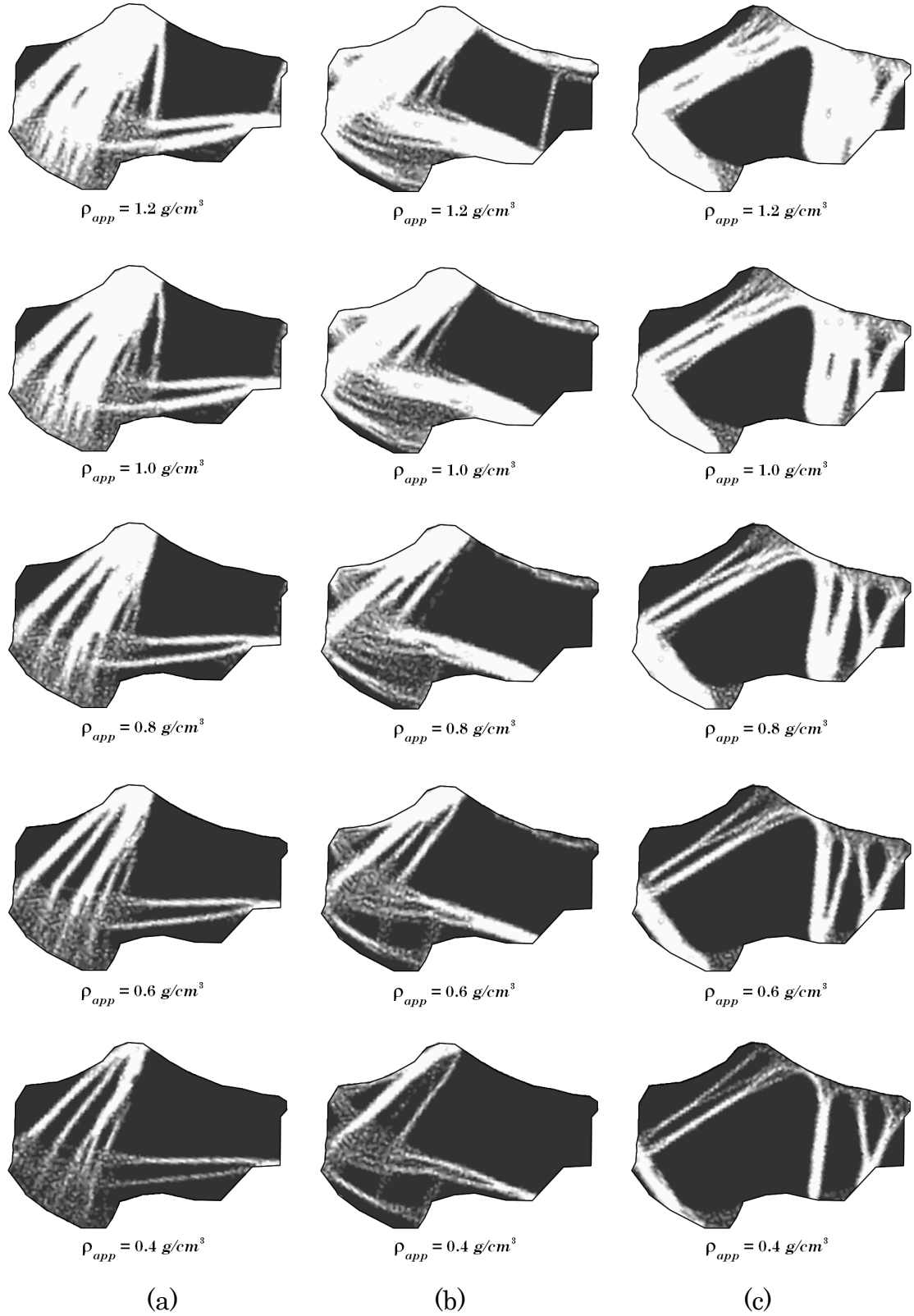


figure 199 – Evolution of the apparent density in the calcaneus bone for the (a) first mechanical case, (b) second mechanical case and (c) third mechanical case.

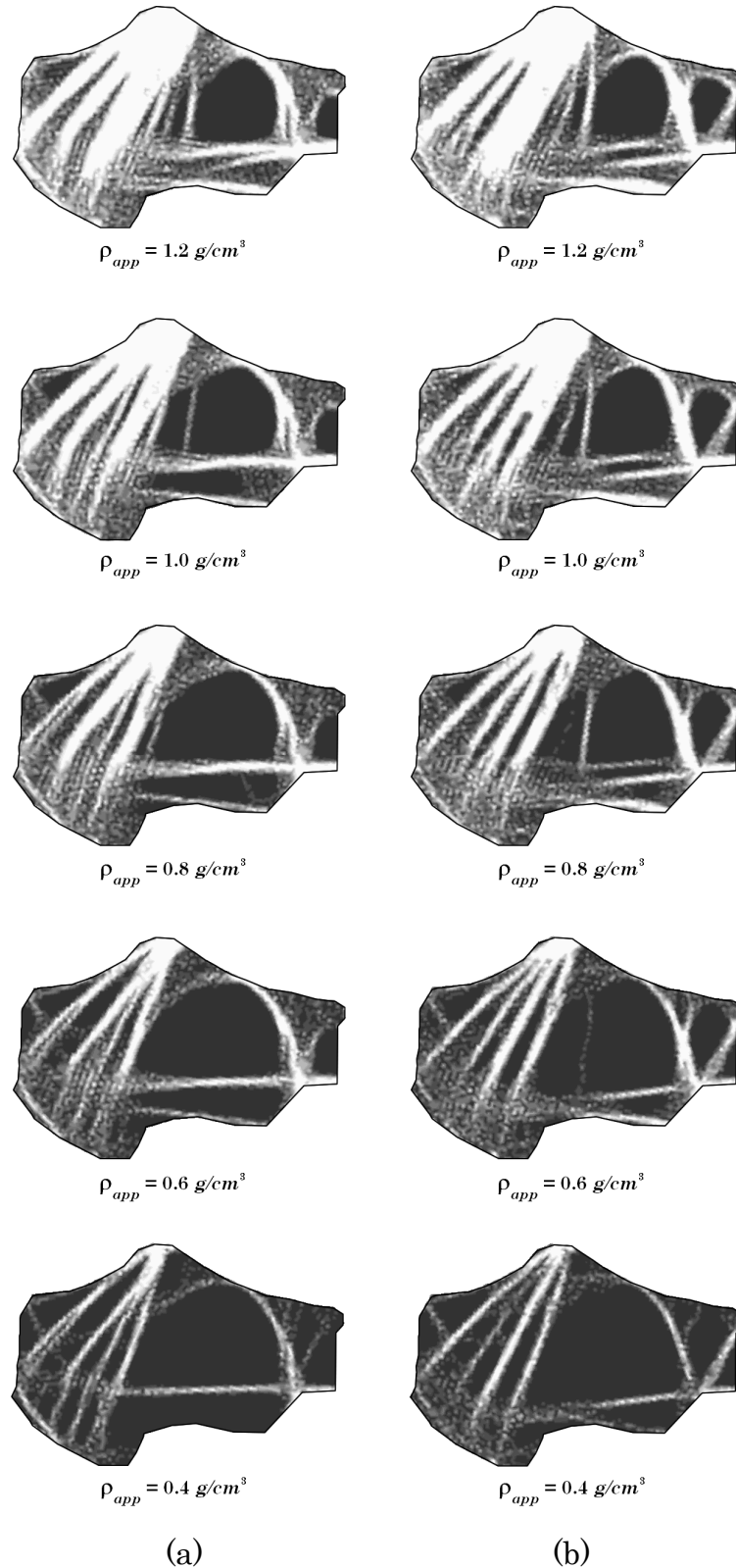


figure 200 – Evolution of the apparent density in the calcaneus bone for the mixing cases.
(a) mechanical load 1 - 6000 cycles; mechanical load 2 - 6000 cycles; mechanical load 3 - 4200 cycles.
(b) mechanical load 1 - 6000 cycles; mechanical load 2 - 6000 cycles; mechanical load 3 - 6000 cycles.

7.4 The Femur Bone

The femur bone is probably the most studied bone example available in the literature. The remodelling process of this long bone was analysed by several authors using the two-dimensional analysis [150, 157, 158, 160, 164, 170-173] and the three-dimensional analysis [174-176]. In this section the bone material law proposed in section 6.2.2 and the bone remodelling algorithm presented in figure 157 are combined with the NNRPIM to achieve a femoral internal trabecular bone architecture similar with the real density distribution found in the femur bone.

7.4.1 The Femoral Two-Dimensional Analysis

The femur is a long bone articulated in the hip-bone. The body weight is directly applied in the femur head. This long bone is a natural choice to validate an anisotropic remodelling algorithm, since it is a well-studied bone in biomechanics and the trabecular structure in the proximal femur is relatively well oriented. An example of a x-ray plate of the proximal femur is presented in figure 201(a). From figure 201(a) it is possible to empirically obtain the compressive and tensile lines indicated in figure 201(b). It was used the geometry of the two-dimensional proximal femur model presented in [160]. The domain was discretized in the nodal mesh presented in figure 201(c).

The femur loading history was approximated by the three-load cases used by Beaupré [170, 171], each consisting of one parabolic distributed load over the joint surface and another parabolic distributed load on the trochanter, representing the abductor muscle attachment. In figure 202 it is possible to observe the resultant of each applied parabolic distributed load and the correspondent direction. For the three considered mechanical cases, all degrees of freedom are constrained in the basis.

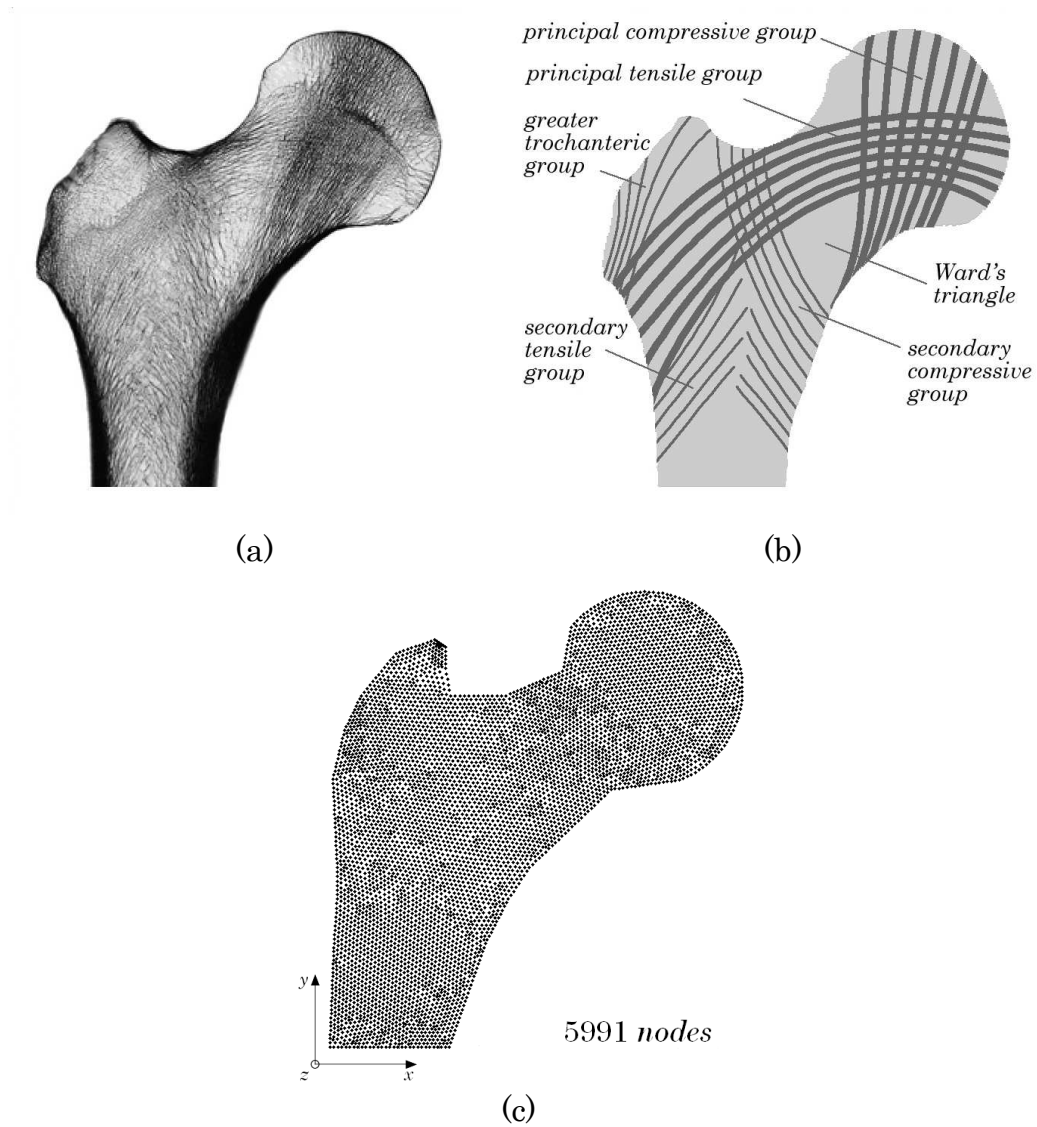


figure 201 – (a) Femoral x-ray plate. (b) Femoral internal principal trabecular structures. (c) Nodal mesh.

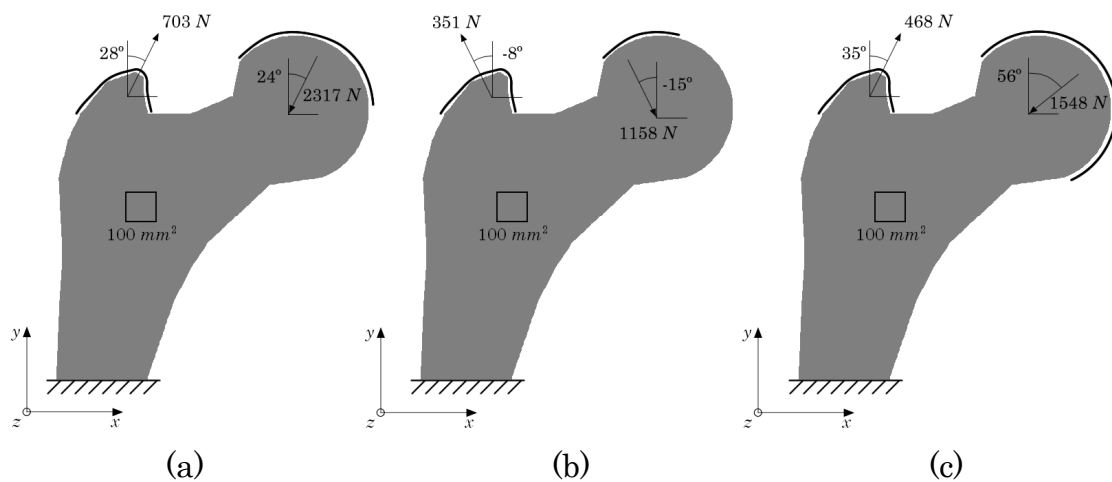


figure 202 – Loads and constraints of the various mechanical case of the femur bone. (a) first mechanical case, (b) second mechanical case and (c) third mechanical case.

Each one of the load cases was independently analysed. An initial density distribution $\rho_{app}^{max} = 2.1 g/cm^3$ is assumed, with a Poisson ratio $\nu = 0.3$, regardless the material direction, and 1% of interest points are remodelled in each iteration. The anisotropic material law presented in section 6.2.2 is considered. The evolution of the trabecular bone architecture in the femur is presented in figure 203 for the distinct mechanical cases. The white colour represents the considered maximum apparent density $\rho_{app}^{med} = 2.1 g/cm^3$ and the dark-gray colour represents the minimum apparent density $\rho_{app}^{med} = 0.1 g/cm^3$ admitted. All the other gray tones in the middle represent transitional apparent densities.

In the plates of figure 203 it are presented the individual study of each mechanical case. In order to obtain a closer result to the real proximal femur x-plate one have to simultaneously apply all mechanical cases. In the work of Beaupré [170, 171] was originally suggested that the first mechanical case should be applied 6000 cycles per day and 2000 cycles per day for the second and the third mechanical cases. The results are presented in figure 204.

In figure 204 it is possible to observe the internal trabecular structures indicated in figure 201(b) and the remaining internal formations, such the Ward's triangle and the greater trochanteric group. Notice that also the secondary structures can be accurately predicted with the proposed remodelling algorithm and bone anisotropic material law.

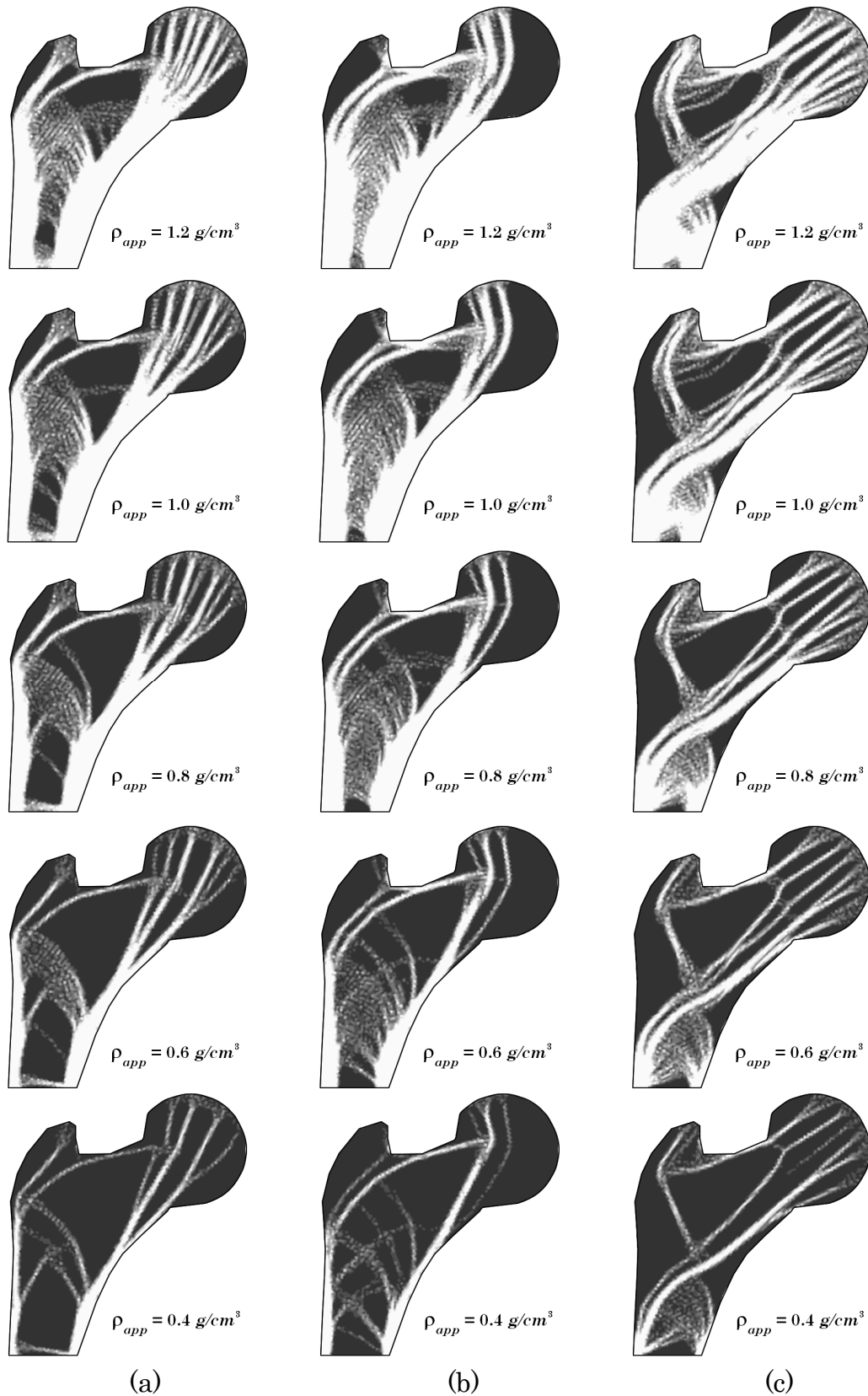


figure 203 – Evolution of the apparent density in the femoral bone for the (a) first mechanical case, (b) second mechanical case and (c) third mechanical case.

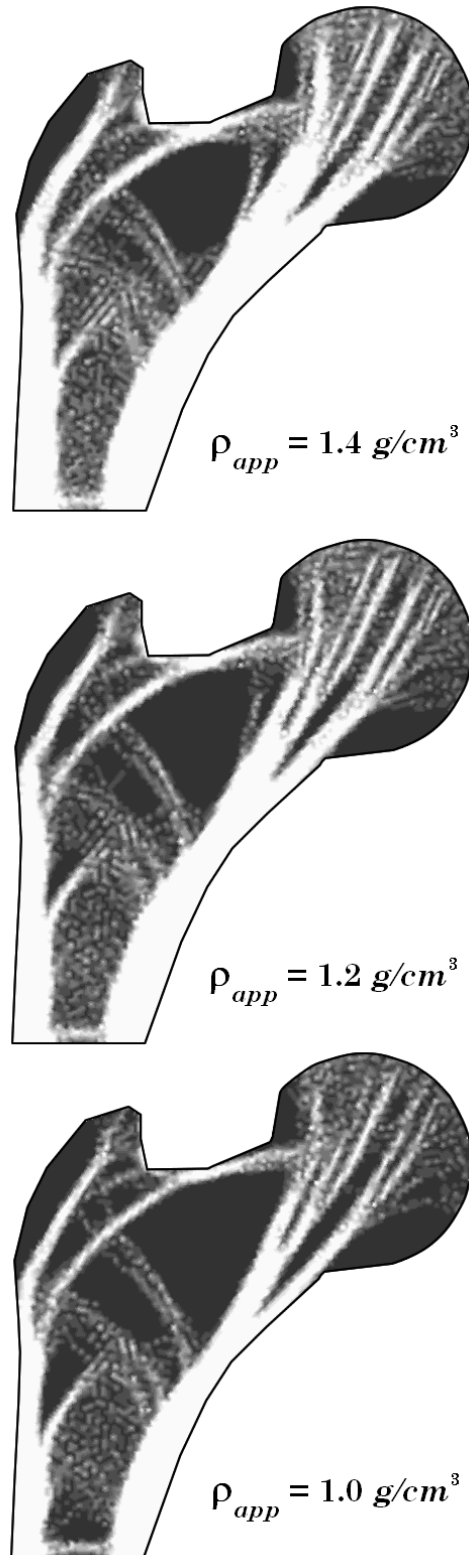


figure 204 – Evolution of the apparent density in the femoral bone when the three mechanical cases are applied simultaneously.

7.4.2 The Femoral Three-Dimensional Analysis

The three-dimensional analysis of the femur bone is similar with the two-dimensional analysis. For this analysis a three-dimensional model of the femur was constructed, figure 205. The model was discretized in 3318 nodes, which is a very coarse three-dimensional nodal mesh. However, for the three-dimensional analysis computer limitations impose a limit of 3500 nodes. In order to have a three-dimensional mesh with the same density as the one used in the two-dimensional analysis, the problem domain should be discretized with nearly 300.000 nodes.

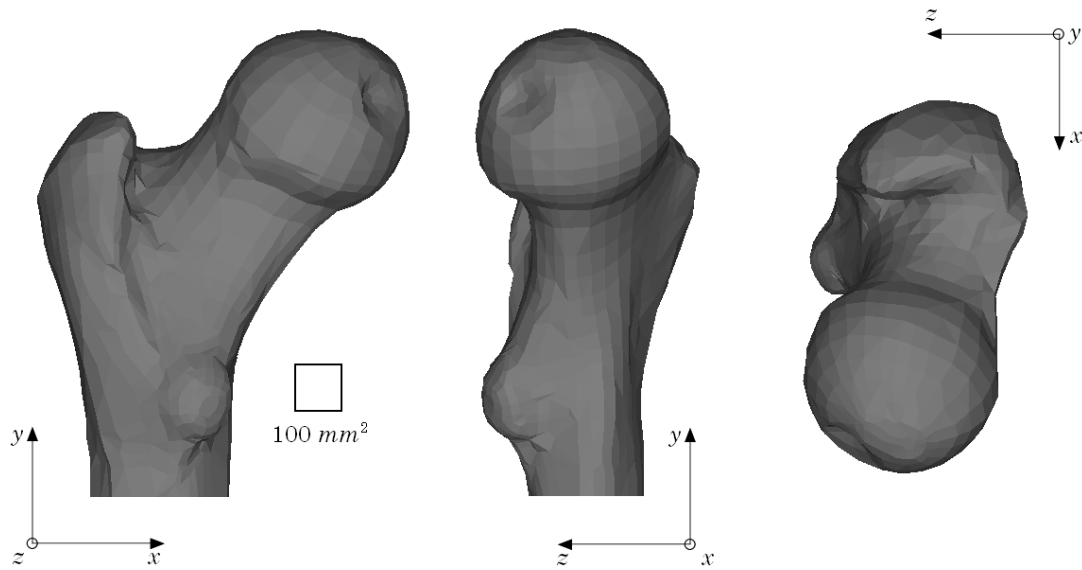


figure 205 – Three-dimensional model of the femur bone.

For the three-dimensional analysis three-load cases were considered, adapting the load cases suggested by Beaupré [170, 171], figure 206. The loads respect the direction and magnitude of the ones presented in figure 202.

By option, the load vectors possess a null z component, $F_{ij} = \{f_x \ f_y \ 0\}$. The loads are, as in the two-dimensional analysis, parabolic distributed along the surfaces indicated in figure 206. In all three cases the proximal femur model is clamped in the basis. In order to visualize the apparent density evolution in the bone interior a sectional cut is made in the model, figure 207.

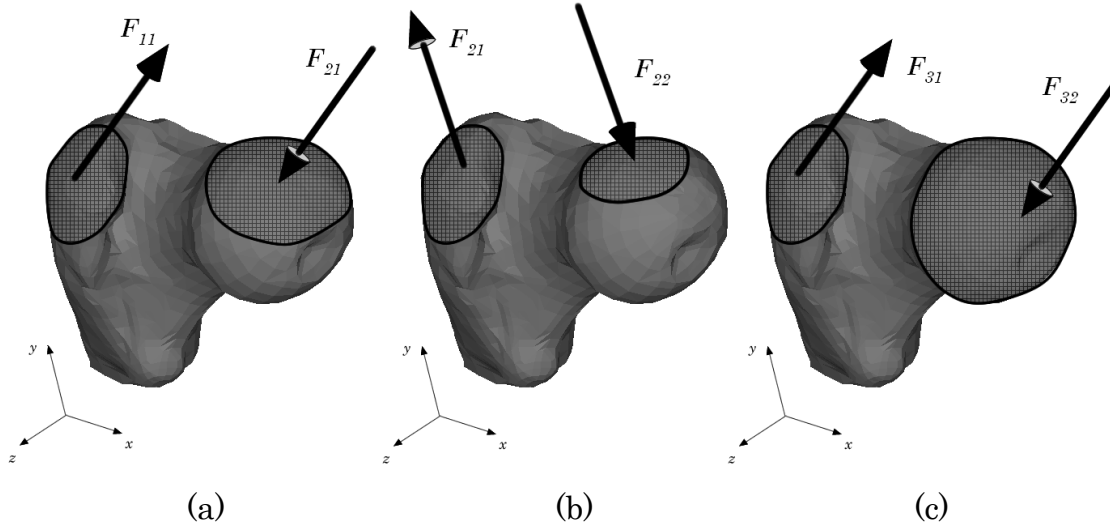


figure 206 – (a) First mechanical case. (b) Second mechanical case. (c) Third mechanical case.

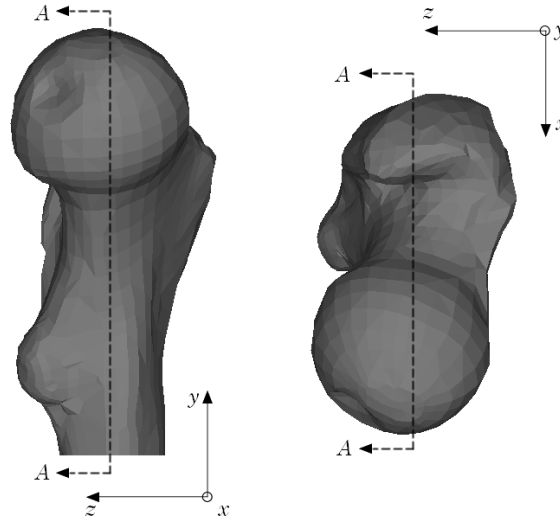


figure 207 – Three-Dimensional model sectional cut AA.

In a first stage each one of the load cases was independently analysed. As in previous example an initial density distribution $\rho_{app}^{max} = 2.1g/cm^3$ is assumed, with a Poisson ration $\nu = 0.3$, regardless the material direction, and 1% of interest points are remodelled in each iteration. It is considered the anisotropic material law presented in section 6.2.2. In figure 208 it is presented the evolution of the trabecular bone architecture in the femur for the distinct mechanical cases. The white colour represents the considered maximum apparent density $\rho_{app}^{med} = 2.1g/cm^3$ and the dark-gray colour represents the minimum apparent density $\rho_{app}^{med} = 0.1g/cm^3$ admitted. All the other gray tones in the middle represent transitional apparent densities. Notice that it is not possible to visualize clearly the trabecular structure of the inner bone. The used three-dimensional model was created with a very

coarse nodal mesh, which does not permit the formation of trabecular structures as in the two-dimensional example. However, it is possible to observe clear density areas, which are in agreement with the two-dimensional results.

In this following analysis the three-load cases were also applied simultaneously. As in the two-dimensional example, the first mechanical was applied with 6000 cycles per day and the remaining mechanical cases with 2000 cycles per day. The results are presented in figure 208(d).

In figure 208(d) it is possible to observe the principal internal structures empirically suggested in figure 201(b). For medium apparent densities $\rho_{app}^{med} > 0.8 g/cm^3$ the principal compressive and tensile group are detectable and also the Ward's triangle and the greater trochanteric group.

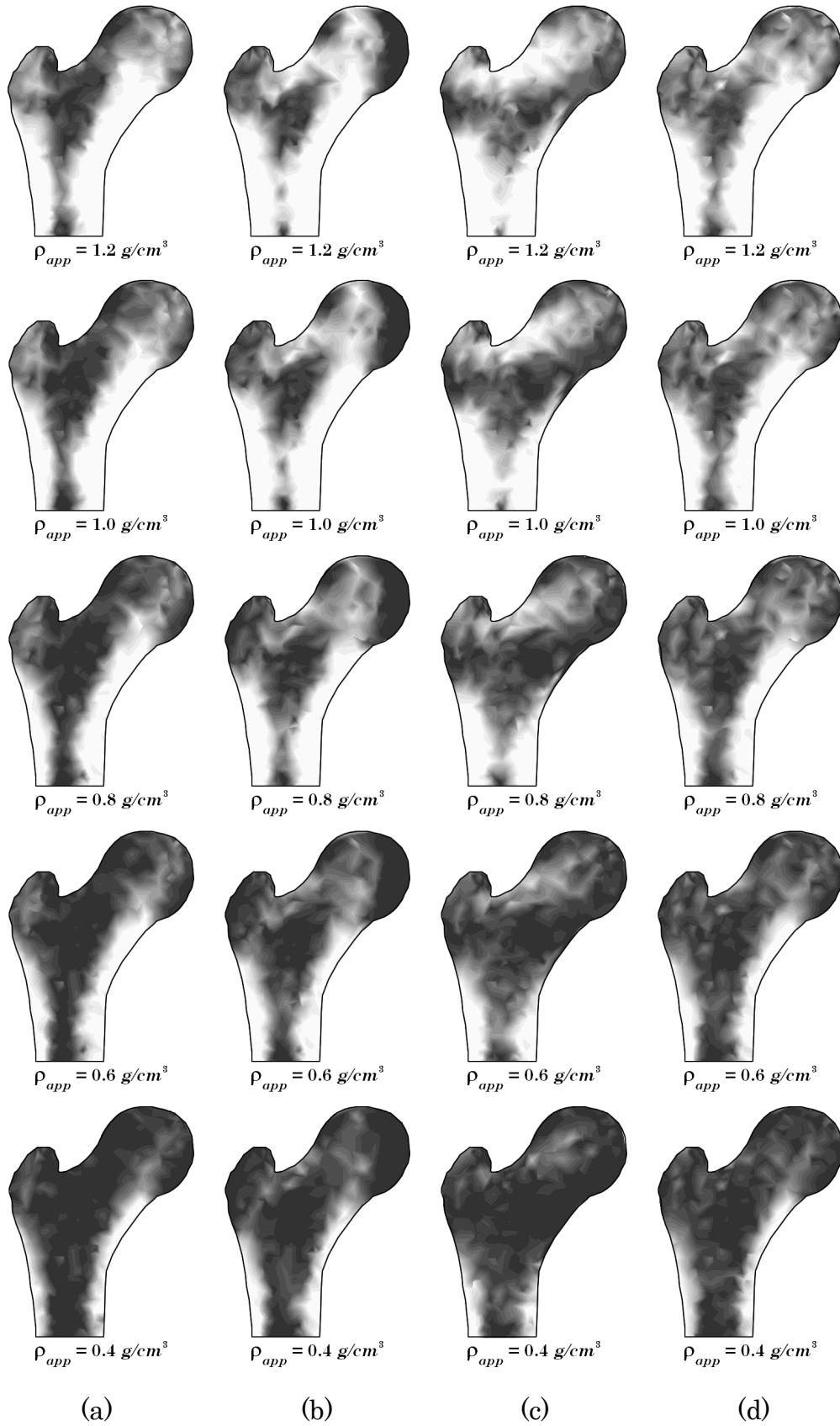


figure 208 – Evolution of the apparent density in the femoral bone for the (a) first mechanical case, (b) second mechanical case and (c) third mechanical case. (d) Three mechanical cases applied simultaneously.

7.5 Conclusions and Remarks

In this chapter the NNRPIM was extended to the biomechanical analysis using a new bone material law, section 6.2.2, and the proposed bone remodelling algorithm, figure 157. Several examples well-documented in the literature were studied and solved, and the following conclusions may be established:

- i.* The proposed bone anisotropic material law permits a smooth transition between the cortical bone stage and the trabecular bone condition, which can be observed in the totality of the presented analysis.
- ii.* The level of the domain discretization plays an important role in the analysis. However, even for more course meshes the remodelling algorithm is able to predict accurately the major internal bone trabecular structures, as it was shown in the proximal femur three-dimensional example.
- iii.* The developed remodelling algorithm combined with the NNRPIM accuracy permits to predict correctly the secondary trabecular structures, which are very important in the stability of the principal structures.
- iv.* The results obtained in the calcaneus bone example prove that the proposed mechanical cases can be used as numerical reference.

The developed work and the obtained results permit to conclude that the NNRPIM, being an accurate and flexible meshless method, has the potential to progress on the immediate future in several biomechanics fields.

Chapter 8

8 Conclusions

The main purpose of this work was to demonstrate the potential of the new developed meshless method, the Natural Neighbour Radial Point Interpolation Method (NNRPIM), in the mechanical and biomechanical engineering fields. In the author opinion this purpose was successfully achieved.

8.1 Conclusions and Remarks

In this work the NNRPIM was presented and developed. In the basis of the present meshless method lays the Natural Neighbour concept combined with the Radial Point Interpolators. Within the NNRPIM, the Natural Neighbour concept inclusion warrants that the nodal connectivity imposition and the integration mesh, used to integrate the weak form, are exclusively dependent on the nodal mesh used to discretize the problem domain, which permits the use of “truly meshless method” denomination for the NNRPIM.

In other meshless methods the nodal connectivity is imposed by rigid geometric shapes, called influence domains, which can lead to unbalance node sets. In the NNRPIM the nodal connectivity is imposed by influence-cells, which are organic structures dependent on the nodal mesh arrangement. Two distinct influence-cell are proposed: a first degree influence cell and a second degree influence-cell. The first degree influence-cell is form by a lower number of neighbour nodes, imposing a lower nodal connectivity. However this small influence-cell is very useful on the three-dimensional analysis, since it represents a lower computational cost. Alternatively, the second degree influence-cell imposes a higher nodal connectivity, since it is form by a large number of neighbour nodes. This larger influence-cell leads to more stable and accurate results, despite the relative higher computational cost.

The integration scheme used in the NNRPIM is almost a direct nodal integration, since the integration mesh is totally dependent on the nodal mesh. Using the Delaunay triangulation each interest node gives origin to a set of integration points. The sum of the individual weight of each integration points on the set defines the weight of the interest node influence area or influence volume. It was found that the simplest nodal scheme proposed is the more efficient, leading to faster and accurate results. The flexibility permitted by the influence-cells and the integration scheme allows the NNRPIM to efficiently solve problems discretized with highly irregular meshes and various boundary conditions, in shape and form.

With the NNRPIM the Radial Point Interpolators were improved. It was found that the polynomial basis is not important, and only contributes to increase the computational cost. Within this work only the Multiquadrics Radial Basis Function was studied. The conducted optimization studies permit to obtain more efficient shape parameters, which confer finally to the constructed interpolation functions the delta Kronecker property, announced but not achieved in earlier works on the Radial Point Interpolators. This important breakthrough permits an easier and faster essential and natural boundary condition imposition, which represents an advantage in nonlinear problems, where the essential boundary matrices and natural boundary vectors have to be computed in each iteration. The optimized shape parameters also lead the NNRPIM to more accurate solutions and smother displacements and stress fields.

For thin three-dimensional structures, such as plates, laminates and shells, a new approach was proposed, the NNRPIM 3D Shell-Like formulation. Improvements on the integration level were made and the results are promising: accurate solutions and lower computational costs.

In the nonlinear analysis the nonlinear solution algorithms were successfully implemented. The Newton-Raphson method and the Orthogonal Actualized Ramm's method (arch-length method), produced similar results, validating both methods. In the geometric nonlinearity problems the snap-through phenomenon was accurately predicted by the NNRPIM combined with the arch-length method. Within the elastoplastic problems the implemented stress returning algorithm confers the nonlinear analysis accuracy, stability and speed.

Within the NNRPIM extension to the biomechanical analysis a new bone anisotropic material law was proposed. In the totality of the presented examples it is perceptible that the proposed material law permits a smooth

transition between the cortical bone and the trabecular bone stage. Along with a new material law a bone remodelling algorithm was developed, which combined with the NNRPIM accuracy permits to predict correctly all the trabecular structures. Primary and secondary trabecular structures, which are very important in the stability of the principal structures, are accurately obtained, predicting a promising future for the NNRPIM in several biomechanics fields

With the results obtained and the experience acquired along the development of this work [29, 32-40] it can be concluded that the NNRPIM is a flexible efficient and alternative numerical method that can be used in several fields of solid mechanics and biomechanical analysis.

8.2 Future Developments

Being the NNRPIM a new meshless method, numerous engineering fields lack to be explored and experimented. The first steps on crack opening path analysis were already made, however a more deep and coherent study need to be done. Belytschko work on arbitrary evolving cracks for three-dimensional large deformation analysis using meshfree method [177] can be reproduced and then developed with the NNRPIM as well as the extension to the mechanics of nanofracture [178, 179] using the multiscale analysis.

The topological study of biomechanical structures can be developed and improved. In this work the proposed bone remodelling algorithm does not includes a 'lazy-zone' [150]. However the existence of this 'lazy-zone' is crucial since whenever the analysis reaches this zone the process is considered in the homeostatic state and stops, achieving a converged solution. In this work only the mechanical stimulus was considered, however much more stimulus could be considered.

Cell biologic models could be easily introduced. These cell biology based models consider a number of bone-cell activities functions to be involved in the bone remodelling process. Therefore in this models the remodelling process is summarizes in a mathematical formulation that take in account pure mechanical stimulus and also biochemical stimulus. Beaupré [170, 171] developed a time dependent description for the bone remodelling process where the bone remodelling reaction was a function of a mechanical stimulus and an additional stimulus that depends on the local apparent density. In this same study it was also considered the bone surface area available for osteoblastic and osteoclastic activity. Hart and Davy [180] used

a similar mathematical formulation with the one used in the model of Carter and developed a cell biology based remodelling theory that depends on biologic parameters. These parameters quantify the cell differentiation and the cell-function processes, based on the assumption that the remodelling process on the bone surface can be expressed as the sum of the osteoclast and osteoblast activity per unit area. Mullender [155, 181] studied the influence of the biologic mechano-sensors in the bone remodelling process. As a result a mathematical formulation that considered biologic parameters was developed, enriching the model of Carter.

Doblaré and co-workers developed an approach that considers the directional activity of the basic multicellular units (BMUs) [176, 182]. This model takes in account the all cell-activity: osteoclast and osteoblast activity and matrix mineralization. Changes in the porosity, damage level (density of microcracks accumulated by cyclic loads) and mineral content are predicted in this model. In a first approach the bone constitutive behaviour was considered isotropic [182] and afterwards the anisotropic nature of the bone was introduced in the model [176].

The preliminary work developed in cracks and nanocracks in solid mechanics could be useful in the biomechanics. The NNRPIM extention to damage models in biomechanics does not represent a difficulty. It is consensual amongst several authors the idea that bone remodelling avoids the accumulation of microdamage caused by daily activities, by means of an effective self-healing process. The continuous process of damage and repair controls the bone adaptation. Prendergast and Taylor [112, 183] proposed a mathematical formulation, ruling both bone growth and bone reabsorption, which assumes that in the remodelling equilibrium exists damage in the form of microcracks, and these microcracks are repaired in an equal rate to the one that it are generated. García and Doblaré [174, 184] proposed a model for internal anisotropic bone remodelling. Following the principles of continuum damage mechanics this model considers bone remodelling under the extent of a general ‘damage-repair’ theory. A ‘damage-repair’ tensor is defined in terms of the apparent density and Cowin’s ‘fabric tensor’, respectively, associated with porosity and directionality of the trabeculae. The different elements of a thermodynamically consistent damage theory are established, including reabsorption and apposition criteria, evolution law and rate of remodelling. Afterwards the same model was improved [173] and extended to the mehsless analysis [175].

8.3 Embracing the NNRPIM

The work of Belytschko [1] is a good starting point for those who want to initiate themselves in the meshless methods world. Afterwards the reading of Liu's book it is advised [60] here one can find the description and the mathematical development of the most important meshless methods. Besides the construction description of the approximation and interpolation functions of several meshless methods, this book contains the results of many benchmark examples (static and dynamic analysis). In parallel it is recommended the reading of Dolbow work [185] and Nguyen work [3], in order to begin, from a safe starting point, the meshless programming. There are many meshless methods, each one with its own variations, and the NNRPIM is just one of them. However the NNRPIM have a huge advantage to the others. The NNRPIM is a recent meshless method, giving the first steps in the computational mechanics. Numerous engineering fields are waiting to be explored with the NNRPIM. In order to start the study of the NNRPIM, the first step is to get familiar with the work of Wang [21, 22]. Then, the first NNRPIM paper is mandatory [29]. In this paper the interpolation functions are presented, as well as the integration scheme (complete nodal-dependent) and the original studies regarding the shape parameters optimization. Next, it can be embraced the study of the static analysis of isotropic and orthotropic plates [32] and the functionally graded material plate analysis [33], the 3D shell-like approach [34] for laminated plates and shells [35]. The dynamic analysis of several solid-mechanic problems can be found in [36-38]. Finally more demanding applications can be studied and reproduced, such as the material nonlinearity [39] and the large deformation analysis [40].

9 References

1. Belytschko T., Krongauz Y., Organ D., Fleming M., and Krysl P., *Meshless Methods: an overview and recent developments*. Computer Methods in Applied Mechanics and Engineering, 1996. **139**(1): p. 3-47.
2. Gu Y.T., *Meshfree methods and their comparisons*. International Journal of Computational Methods, 2005. **2**(4): p. 477-515.
3. Nguyen V.P., Rabczuk T., Bordas S., and Duflot M., *Meshless methods: A review and computer implementation aspects*. Mathematics and Computers in Simulation, 2008. **79**(3): p. 763-813.
4. Zienkiewicz O.C., *The Finite Element Method*. 4th ed. 1989: McGraw-Hill.
5. Bathe K.J., *Finite Element Procedures*. 1996: Prentice-Hall: Englewood Cliffs, NJ.
6. Nayroles B., Touzot G., and Villon P., *Generalizing the Finite Element Method: Diffuse Approximation and Diffuse Elements*. Computational Mechanics, 1992. **10**: p. 307-318.
7. Lancaster P. and Salkauskas K., *Surfaces Generation by Moving Least Squares Methods*. Mathematics of Computation, 1981. **37**: p. 141-158.
8. Belytschko T., Lu Y.Y., and Gu L., *Element-Free Galerkin Method*. International Journal for Numerical Methods in Engineering, 1994. **37**: p. 229-256.
9. Monaghan J.J., *Smoothed Particle Hydrodynamics: Theory and Applications to Non-Spherical Stars*. Monthly Notices of the Astronomical Society, 1977. **181**: p. 375-389.
10. Liu W.K., Jun S., and Zhang Y.F., *Reproducing Kernel Particle Methods*. International Journal for Numerical Methods in Fluids, 1995. **20**(6): p. 1081-1106.
11. Atluri S.N. and Zhu T., *A new meshless local Petrov-Galerkin (MLPG) approach in computational mechanics*. Computational Mechanics, 1998. **22**(2): p. 117-127.
12. De S. and Bathe K.J., *The method of finite spheres*. Computational Mechanics, 2000. **25**(4): p. 329-345.
13. Oñate E., Idelsohn S., Zienkiewicz O.C., Taylor R.L., and Sacco C.A., *A stabilized finite point method for analysis of fluid mechanics problems*. Computer Methods in Applied Mechanics and Engineering, 1996. **139**(1-4): p. 315-346.

14. Oñate E., Idelsohn S., Zienkiewicz O.C., and Taylor R.L., *A Finite Point Method in Computational Mechanics - Applications to Convective Transport and Fluid Flow*. International Journal for Numerical Methods in Engineering, 1996. **39**: p. 3839-3866.
15. Kansa E.J., *A scattered data approximation scheme with applications to computational fluid-dynamics - I & II*. Computers and Mathematics with Applications, 1990. **19**: p. 127-161.
16. Hardy R.L., *Theory and applications of the multiquadrics - Biharmonic method (20 years of discovery 1968-1988)*. Computers and Mathematics with Applications, 1990. **19**: p. 127-161.
17. Ferreira A.J.M., Roque C.M.C., and Martins P.A.L.S., *Radial basis functions and higher-order shear deformation theories in the analysis of laminated composite beams and plates*. Composite Structures, 2004. **66**(1-4): p. 287-293.
18. Tiago C.M. and Leitão V.M.A., *Application of radial basis functions to linear and nonlinear structural analysis problems*. Computers & Mathematics with Applications, 2006. **51**(8): p. 1311-1334.
19. Liu G.R. and Gu Y.T., *A Point Interpolation Method for Two-Dimensional Solids*. International Journal for Numerical Methods in Engineering, 2001. **50**: p. 937-951.
20. Liu G.R., *A Point Assembly Method for Stress Analysis for Two-Dimensional Solids*. International Journal of Solid and Structures, 2002. **39**: p. 261-276.
21. Wang J.G. and Liu G.R., *A Point Interpolation Meshless Method based on Radial Basis Functions*. International Journal for Numerical Methods in Engineering, 2002. **54**: p. 1623-1648.
22. Wang J.G. and Liu G.R., *On the Optimal Shape Parameters of Radial Basis Functions used for 2-D Meshless Methods*. Computer Methods in Applied Mechanics and Engineering, 2002. **191**: p. 2611-2630.
23. Sergio R., Idelsohn S., Oñate E., Calvo N., and Del Pin F., *The Meshless Finite Element Method*. International Journal for Numerical Methods in Engineering, 2003. **58**(6): p. 893-912.
24. Traversoni L., *Natural Neighbour Finite Elements*. Int. Conf. on Hydraulic Engineering Software, Hydrossoft Proc., Computational Mechanics Publications, 1994. **2**: p. 291-297.
25. Sukumar N., Moran B., Semenov A.Y., and Belikov V.V., *Natural neighbour Galerkin methods*. International Journal for Numerical Methods in Engineering, 2001. **50**(1): p. 1-27.
26. Braun J. and Sambridge M., *A numerical method for solving partial differential equations on highly irregular evolving grids*. Nature, 1995. **376**: p. 655-660.

27. Sukumar N., Moran B., and Belytschko T., *The natural element method in solid mechanics*. International Journal for Numerical Methods in Engineering, 1998. **43**(5): p. 839–887.
28. Cueto E., Doblaré M., and Gracia L., *Imposing essential boundary conditions in the natural element method by means of density-scaled - shapes*. International Journal for Numerical Methods in Engineering, 2000. **49**(4): p. 519-546.
29. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *Analysis of 3D solids using the natural neighbour radial point interpolation method*. Computer Methods in Applied Mechanics and Engineering, 2007. **196**(13-16): p. 2009-2028.
30. Voronoï G.M., *Nouvelles applications des paramètres continus à la théorie des formes quadratiques*. Deuxième Mémoire: Recherches sur les paralléloèdres primitifs, J. Reine Angew. Math., 1908. **134**: p. 198-287.
31. Delaunay B., *Sur la sphère vide. A la memoire de Georges Voronoï*. Izv. Akad. Nauk SSSR, Otdelenie Matematicheskikh i Estestvennykh Nauk, 1934. **7**: p. 793-800.
32. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *Analysis of plates and laminates using the natural neighbour radial point interpolation method*. Engineering Analysis with Boundary Elements, 2008. **32**(3): p. 267-279.
33. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *An Unconstrained Third-Order Plate Theory applied to Functionally Graded Plates using a Meshless Method*. Mechanics of Advanced Materials and Structures, 2010. **17**: p. 1-26.
34. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *Composite Laminated Plates: A 3D natural neighbour radial point interpolation method approach*. Journal of Sandwich Structures and Materials, 2010. **12**(2): p. 119-138.
35. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *A 3D Shell-Like approach using a Natural Neighbour Meshless Method: isotropic and orthotropic thin structures*. Composite Structures, 2010. **92**(5): p. 1132-1142.
36. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *The Natural Neighbour Radial Point Interpolation Method: Dynamic Applications*. Engineering Computations, 2009. **26**(8): p. 911-949.
37. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *The dynamic analysis of thin structures using a radial interpolator meshless method*, in *Vibration and Structural Acoustics Analysis*, Ferreira A.J.M., et al., Editors. 2011, Springer. p. (in press).

38. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *Static and Dynamic Analysis of Laminated Plates Based on an Unconstrained Third Order Theory and Using a Radial Point Interpolator Meshless Method*. Computers and Structures, 2011: p. (in press).
39. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *Radial Natural Neighbours Interpolators: 2D and 3D Elastic and Elastoplastic Applications*, in *Progress on Meshless Methods*, Ferreira A.J.M., et al., Editors. 2008, Springer.
40. Dinis L.M.J.S., Jorge R.M.N., and Belinha J., *Large Deformation Applications with the Radial Natural Neighbours Interpolators*. Computer Modelling in Engineering and Sciences, 2009. **44**(1): p. 1-34.
41. Belinha J., *Elasto-Plastic Analysis Considering the Element Free Galerkin Method*. 2004, MSc Thesis, Faculdade de Engenharia da Universidade do Porto.
42. Belinha J. and Dinis L.M.J.S., *Análise Elasto-plástica de Problemas Anisotrópicos considerando o Método Livre de Elementos de Galerkin*. Revista Internacional de Metodos Numericos para calculo y diseño en Ingenieria, 2006. **22**(2).
43. Belinha J. and Dinis L.M.J.S., *Analysis of Plates and Laminates using the Element Free Galerkin Method*. Computers and Structures, 2006. **84**: p. 1547-1559.
44. Belinha J. and Dinis L.M.J.S., *Elasto-Plastic Analysis of Plates using the Element Free Galerkin Method*. Engineering Computations, 2006. **23**(3): p. 525-551.
45. Belinha J. and Dinis L.M.J.S., *O Método Livre de Elementos de Galerkin na Análise Não-Linear de Estruturas Anisotrópicas*. Revista Portuguesa de Engenharia de Estruturas, 2007. **2**(Série II): p. 19-35.
46. Belinha J. and Dinis L.M.J.S., *Non Linear Analysis of Plates and Laminates using the Element Free Galerkin Method*. Composite Structures, 2007. **78**(3): p. 337-350.
47. Fung Y.C., *Foundations of Solid Mechanics*. 1965, New Jersey, U.S.A.: Englewood Cliffs, Prentice-Hall.
48. Hodge P.G., *Continuum Mechanics*. 1970, New York: Mc Graw-Hill.
49. Malvern L.E., *Introduction of the Mechanics of a Continuous Medium*. 1969, New Jersey, U.S.A.: Englewood Cliffs, Prentice-Hall.
50. Lekhnitskii S.G., *Anisotropic Plates*. 2nd ed. 1968, New York, U.S.A.: Gordon and Breach Science Publishers.
51. Timoshenko S. and Goodier J.N., *Theory of Elasticity*, ed. 3^a. 1970: McGraw Hill.
52. Reddy J.N., *Applied Functional Analysis and Variational Methods in Engineering*. 1986, Singapore: McGraw-Hill.

53. Hill R., *The Mathematical Theory of Plasticity*. Oxford Engineering Sciences Series. 1950, U.K.: Oxford University Press.
54. Chakrabarty J., *Theory of Plasticity*. 1987: Mc Graw-Hill International Edition.
55. Von Mises R., *Göttinger Nachrichten Math. Phys. Klasse*. 1913: p. 582.
56. Owen D. and Hinton E., *Finite Element in Plasticity*. 1 ed. 1980: Pineridge press.
57. Tresca H., *Sur l'écoulement des corps solides soumis á des portes pressions*. Comptes Rendus Academie des Sciences Paris, 1864. **59**: p. 754.
58. Crisfield M.A., *Non-Linear Finite Element Analysis of Solids and Structures*. 1 ed. 1991: Jonh Wiley and Sons, Ltd, Baffins Lane.
59. Anderson E., *LAPACK Users' Guide*. 2 ed. 1995, Philadelphia: Society for Industrial and Applied Mathematics.
60. Liu G.R., *Mesh Free Methods, Moving beyond the Finite Element Method*. 2002: CRC Press.
61. Sibson R., *A brief description of natural neighbor interpolation*. Interpreting Multivariate Data, in V. Barnett (ed.), (Wiley, Chichester), 1981: p. 21-36.
62. Lawson C.L., *Software for C1 surface interpolation*. in J. R. Rice (ed.), Mathematical Software III, Vol. 3, Academic Press, New York, N.Y., 1977.
63. Watson D.F., *Contouring: A Guide to the Analysis and Display of Spatial Data*. Pergamon Press, Oxford, 1992.
64. Wang J.G., Liu G.R., and Wu Y.G., *A Point Interpolation Method for Simulating Dissipation Process of Consolidation*. Computer Methods in Applied Mechanics and Engineering, 2001. **190**: p. 5907-5922.
65. Dai K.Y., Liu G.R., Han X., and Li Y., *Inelastic analysis of 2D solids using a weak-form RPIM based on deformation theory*. Computer Methods in Applied Mechanics and Engineering, 2006. **195**: p. 4179-4193.
66. Liu G.R., Gu Y.T., and Dai K.Y., *Assessment and applications of interpolation methods for computational mechanics*. International Journal for Numerical Methods in Engineering, 2004. **59**: p. 1373-1379.
67. Irons B.M. and Razzaque A., *Experience with the patch test for convergence of nite elements*. in A. K. Aziz (ed.), The Mathematical Foundations of the Finite Element Method with Applications to Partial Dierential Equations, Academic Press, New York, 1972.
68. Timoshenko S. and Woinowsky-Krieger S., *Theory of Plates and Shells*, ed. 2^a. 1959: McGraw Hill.

69. Scordelis A.C. and Lo K.H., *Computer analysis of cylindrical shells*. Journal of American Concrete Institute, (1969). **61**: p. 539-561.
70. Simo J.C., Fox D.D., and R. M.S., *On a stress resultant geometrically exact shell model*. Computer Methods in Applied Mechanics and Engineering, (1989). **73**: p. 53-92.
71. Lindberg G., Olson M.M.D., and C. G.R., *New developments in the Finite Element Analysis of Shells*, in *Quarterly Bulletin of the Division of Mechanical Engineering and the National Aeronautical Establishment*. (1969): National Research Council of Canada.
72. Reddy J.N., *Mechanics of Laminated Composite Plates*. 1996: CRC Press.
73. Pagano N.J., *Exact solutions for rectangular bidirectional composites and sandwich plates*. Journal of Composite Materials, 1970. **4**: p. 20-34.
74. Pagano N.J. and Hatfield S.J., *Elastic behavior of multilayered composite bi-directional composites*. American Institute of Aeronautics and Astronautics Journal, 1972. **10**: p. 931-933.
75. Kant T. and Khare R.K., *A high order facet quadrilateral composite shell element*. International Journal for Numerical Methods, (1997). **40**: p. 4477-4499.
76. Ren J.G., *Exact solutions for laminated cylindrical shells in cylindrical bending*. Composites Science and Technology, (1987). **29**: p. 169-187.
77. Ren J.G., *Analysis of simply supported laminated circular cylindrical shells*. Composite Structures, (1989). **11**: p. 277-292.
78. Gu Y.T. and Liu G.R., *A Meshless local Petrov-Galerkin (MLPG) method for free and forced vibration analyses for solids*. Computational Mechanics, 2001. **vol. 27**: p. 188-198.
79. Gu Y.T. and Liu G.R., *A Local Radial Point Interpolation Method (LRPIM) for free vibration analyses of 2-D solids*. Journal of Sound and Vibration, 2001. **vol. 246**(1): p. 29-46.
80. Brebbia C.A., Telles J.C., and Wrobel L.C., *Boundary Element Techniques*. 1984, Berlin: Springer Verlag.
81. Dawe, D.J. and O.L. Roufaeil, *Rayleigh-Ritz vibration analysis of Mindlin plates*. Journal of Sound and Vibrations, (1980). **69**: p. 345-359.
82. Lee, S.J., *Free vibrations analysis of plates by using a four-node finite element formulated with assumed natural transverse shear strain*. Journal of Sound and Vibrations, (2004). **278**: p. 657-684.
83. Nair P.S. and Durvasula S., *Vibration of skew plates*. Journal of Sound and Vibrations, (1973). **26**: p. 1-19.

84. Nath D., *Dynamics of rectangular curved plate*. (1969), PhD Thesis, Southampton University.
85. Au F.T.K. and Cheung Y.K., *Free Vibration and Stability Analysis of Shells by the Isoparametric Spline Finite Strip Method*. Thin-Walled Structures, (1996). **24**: p. 53-82.
86. Tou S.K. and Wong K.K., *High-precision finite element analysis of cylindrical shells*. Computers & Structures, (1987). **26**: p. 847-854.
87. Brünig M., *Nonlinear analysis and elasto-plastic behavior of anisotropic structures*. Finite Elements in Analysis and Design, 1995. **20**: p. 155-177.
88. Alves de Sousa R.J., Natal Jorge R.M., Fontes Valente R.A., and César de Sá J.M.A., *A new volumetric and shear locking-free 3d enhanced strain element*. Engineering Computations, 2003. **20**(7): p. 896-925.
89. Barbosa J.A.T., *Análise Não Linear Por Elementos Finitos de Placas e Cascas Reforçadas*. 1992, Faculdade de Engenharia da Universidade do Porto, PhD thesis.
90. Figueiras J.A., *Ultimate Load Analysis of Anisotropic and Reinforced Concrete Plates and Shells*. 1983, University of Wales, PhD Thesis, C/Ph/72/83, Swansea, U.K.
91. Schreyer H.L. and Masur E.F., *Buckling of shallow arches*. ASCE Journal of Engineering Mechanics Division, 1966. **EM4**: p. 1-19.
92. Gomes C.R., *Matrix Update Methods in Nonlinear Analysis of two Dimensional and thin Shell Problems*, in *Civil Engineering*. 1982, University of Wales: Swansea.
93. Love A.E.H., *A Treatise on the Mathematical Theory of Elasticity*. 1996: Dover Publications: New York.
94. Betsch P., Menzel A., and Stein E., *On parametrization of finite rotations in computational mechanics. A classification of concepts with application to smooth shells*. Computer Methods in Applied Mechanics and Engineering, 1998. **155**: p. 273-305.
95. Areias. P.M.A., César de Sá J. M. A., Conceição António C. A., and Fernandes A.A., *Analysis of 3D problems using a new enhanced strain hexahedral element*. International Journal for Numerical Methods in Engineering, 2003. **58**: p. 1637-1682.
96. Frisch-Fay R., *Flexible Bars*. 1962: Butterworths, London.
97. Chia C.Y., *Non Linear Analysis of Plates*. 1 ed. 1980: McGraw-Hill.
98. Bathe K.J. and Bolourchi S., *Large displacement analysis of three-dimensional beam structures*. International Journal for Numerical Methods in Engineering, 1979. **14**: p. 961-986.
99. Park M.S. and Lee B.C., *Geometrically non-linear and elastoplastic threedimensional shear flexible beam element of Von-Mises-type*

- hardening material*. International Journal for Numerical Methods in Engineering, 1996. **39**: p. 383–408.
100. Shugyo M., *Elastoplastic large deflection analysis of three-dimensional steel frames*. Journal of Structural Engineering, 2003. **129**: p. 1259–1267.
 101. Turkalj G., Brnic J., and Prpic-Orsic J., *ESA formulation for large displacement analysis of framed structures with elastic-plasticity*. Computers & Structures, 2004. **82**: p. 2001–2013.
 102. Mata P., Oller S., and Barbat A.H., *Static analysis of beam structures under nonlinear geometric and constitutive behavior*. Computer Methods in Applied Mechanics and Engineering, 2007. **196**: p. 4458–4478.
 103. Netter F.H., *Musculoskeletal System - Netter Collection of Medical Illustrations*. 1 ed. Vol. 8. 1994: Saunders.
 104. Wolff J., *The Law of Bone Remodeling (Das Gesetz der Transformation der Knochen, Hirschwald, 1892)*. 1986, Berlin Heidelberg New York: Springer.
 105. Aubin J.E. and Heersche J.N.M., *Osteoprogenitor cell differentiation to mature bone-forming osteoblasts*. Drug Development Research, 2000. **49**(3): p. 206-215.
 106. D'Ippolito G., Schiller P.C., Ricordi C., Roos B.A., and Howard G.A., *Age-related osteogenic potential of mesenchymal stromal stem cells from human vertebral bone marrow*. Journal of Bone and Mineral Research, 1999. **14**(7): p. 1115-1122.
 107. Miller S.C. and Jee W.S.S., *Bone lining cells*, in *Bone*. 1992, CRC Press: Boca Raton. p. 1-19.
 108. Lanyon L.E., *Osteocytes, strain detection, bone modeling and remodeling*. Calcified Tissue International, 1993. **53**: p. 102-107.
 109. Cowin S.C., Moss-Salentijn L., and Moss M.L., *Candidates for the mechanosensory system in bone*. Journal of Biomechanical Engineering, 1991. **113**(2): p. 191-197.
 110. Lee T.C., Staines A., and Taylor D., *Bone adaptation to load: microdamage as a stimulus for bone remodelling*. Journal of Anatomy, 2002. **201**(6): p. 437-446.
 111. Taylor D. and Kuiper J.H., *The prediction of stress fractures using a 'stressed volume' concept*. Journal of Orthopaedic Research, 2001. **19**(5): p. 919-926.
 112. Prendergast P.J. and Taylor D., *Prediction of bone adaptation using damage accumulation*. Journal of Biomechanics, 1994. **27**(8): p. 1067-1076.

113. Carter D.R. and Hayes W.C., *Bone compressive strength: the influence of density and strain rate*. Science, 1976. **194**(4270): p. 1174 - 1176.
114. Carter D.R. and Hayes W.C., *The compressive behavior of bone as a twophase porous structure*. Journal of Bone and Joint Surgery, (1977). **59**(A): p. 954–962.
115. Carter D.R. and Spengler D.M., *Mechanical properties and composition of cortical bone*. Clinical Orthopaedics and Related Research, 1978. **135**: p. 192-217.
116. Gibson L.J., *The mechanical behaviour of cancellous bone*. Journal of Biomechanics, 1985. **18**: p. 317–328.
117. Goldstein S.A., *The mechanical properties of trabecular bone: dependence on anatomic location and function*. Journal of Biomechanics, 1987. **20**: p. 1055–1061.
118. Rice J.C., Cowin S.C., and Bowman J.A., *On the dependence of the elasticity and strength of cancellous bone on apparent density*. Journal of Biomechanics, 1988. **21**(2): p. 155-168.
119. Martin R.B., *Determinants of the mechanical properties of bones*. Journal of Biomechanics, 1991. **24**(1): p. 79-88.
120. Carter D.R. and Hayes W.C., *The compressive behavior of bone as a two phase porous structure*. Journal of Bone and Joint Surgery, 1977. **59**(A): p. 954–962.
121. Pietruszczak S., Jiang J., and Mirza F.A., *An elastoplastic constitutive model for concrete*. International Journal of Solids and Structures, 1988. **24**(7): p. 705-722.
122. Currey J.D., *The mechanical consequences of variation in the mineral content of bone*. Journal of Biomechanics, 1969. **2**(1): p. 1-11.
123. Currey J.D., *The effect of porosity and mineral content on the Young's modulus of elasticity of compact bone*. Journal of Biomechanics, 1988. **21**: p. 131-139.
124. Schaffler M.B. and Burr D.B., *Stiffness of compact bone: effects of porosity and density*. Journal of Biomechanics, 1988. **21**: p. 13-16.
125. Keller T.S., *Predicting the compressive mechanical behavior of bone*. Journal of Biomechanics, 1994. **27**: p. 1159-1168.
126. Keyak J.H., Lee I., and Skinner H.B., *Correlations between orthogonal mechanical properties and density of trabecular bone: use of different densitometric measures*. Journal of Biomedical Materials Research, 1994. **28**(11): p. 1329-1336.
127. Hernández C.J., Beaupré G.S., Keller T.S., and Carter D.R., *The influence of bone volume fraction and ash fraction on bone strength and modulus*. Bone, 2001. **29**(1): p. 74-78.

128. Lotz J.C., Gerhart T.N., and Hayes W.C., *Mechanical properties of metaphyseal bone in the proximal femur*. Journal of Biomechanics, 1991. **24**(5): p. 317-329.
129. Zioupos P., Cook R.B., and Hutchinson J.R., *Some basic relationships between density values in cancellous and cortical bone*. Journal of Biomechanics, 2008. **41**: p. 1961–1968.
130. Keaveny T.M., Wachtel E.F., Ford C.M., and Hayes W.C., *Differences between the tensile and compressive strengths of bovine tibial trabecular bone depend on modulus*. Journal of Biomechanics, 1994. **27**(9): p. 1137-1146.
131. Stone J.L., Beaupre G.S., and Hayes W.C., *Multiaxial strength characteristics of trabecular bone*. Journal of Biomechanics, 1983. **16**(9): p. 743-752.
132. Reilly D.T. and Burstein A.H., *The elastic and ultimate properties of compact bone tissue*. Journal of Biomechanics, 1975. **8**(6): p. 393-405.
133. Keyak J.H. and Rossi S.A., *Prediction of femoral fracture load using finite element models: an examination of stress- and strain-based failure theories*. Journal of Biomechanics, 2000. **33**(2): p. 209-214.
134. Pauwels F., *Gesammelte Abhandlungen zur Funktionellen Anatomie des Bewegungsapparates*. 1965, Berlin: Springer-Verlag.
135. Kummer B.K.F., *Biomechanics of bone: Mechanical properties, functional structure, functional adaptation*, in *Biomechanics: Its Foundation and Objectives*, F. Y.C., Editor. 1972, Prentice-Hall, Englewood Cliffs: NJ. p. 237-271.
136. Cowin S.C. and Hegedus D.H., *Bone remodeling I: A theory of adaptive elasticity*. Journal of Elasticity, 1976. **6**: p. 313–326.
137. Cowin S.C., Sadegh A.M., and Luo G.M., *An evolutionary Wolff's law for trabecular architecture*. Journal of Biomechanical Engineering, 1992. **114**(1): p. 129-136.
138. Turner C.H., Cowin S.C., Rho J.Y., Ashman R.B., and Rice J.C., *The fabric dependence of the orthotropic elastic constants of cancellous bone*. Journal of Biomechanics, 1990. **23**(6): p. 549-561.
139. Cowin S.C., *The relationship between the elasticity tensor and the fabric tensor*. Mechanics of Materials, 1985. **4**(2): p. 137-147.
140. Huiskes R., Weinans H., Grootenboer H.J., et al., *Adaptive bone-remodeling theory applied to prosthetic-design analysis*. Journal of Biomechanics, 1987. **20**(11-12): p. 1135–1150.
141. Harrigan T.P. and Hamilton J.J., *An analytical and numerical study of the stability of bone remodelling theories: dependence on microstructural stimulus*. Journal of Biomechanics, 1992. **25**(5): p. 477-488.

142. Harrigan T.P. and Hamilton J.J., *Optimality conditions for finite element simulation of adaptive bone remodeling*. International Journal of Solids and Structures, 1992. **29**(3): p. 2897-2906.
143. Harrigan T.P. and Hamilton J.J., *Finite element simulation of adaptive bone remodelling: A stability criterion and a time stepping method*. International Journal for Numerical Methods in Engineering, 1993. **36**(5): p. 837-854.
144. Bucháček K., *Nonequilibrium bone remodelling: Changes of mass density and of the axes of anisotropy*. International Journal of Engineering Science, 1990. **28**(10): p. 1039-1044.
145. Carter D.R., Fyhrie D.P., and Whalen R.T., *Trabecular bone density and loading history: regulation of connective tissue biology by mechanical energy*. Journal of Biomechanics, 1987. **20**(8): p. 785-794.
146. Carter D.R., Orr T.E., and Fyhrie D.P., *Relationship between loading history and femoral cancellous bone architecture*. Journal of Biomechanics, 1989. **22**(3): p. 231-244.
147. Whalen R.T., Carter D.R., and Steele C.R., *Influence of physical activity on the regulation of bone density*. Journal of Biomechanics, 1988. **21**(10): p. 825-837.
148. Fyhrie D.P. and Carter D., *A unifying principle relating stress to trabecular bone morphology*. Journal of Orthopaedic Research, 1986. **4**: p. 304-317.
149. Starke, G.R., Mercer C.D., Spirakis A., Martin J.B., and Learmonth I.D. *Some aspects of adaptative bone growth as a result of total hip joint replacement*. in *ABACUS User's Conference*. 1992. Providence, RI.
150. Pettermann H., Reiter T., and Rammerstorfer F.G., *Computational simulation of internal bone remodeling*. Archives in Computational Methods in Engineering, 1997. **4**(4): p. 295-323.
151. Xinghua Z., He G., Dong Z., and Bingzhao G., *A study of the effect of non-linearities in the equation of bone remodeling*. Journal of Biomechanics, 2002. **35**: p. 951-960.
152. He G. and Xinghua Z., *The numerical simulation of osteophyte formation on the edge of the vertebral body using quantitative bone remodeling theory*. Joint Bone Spine, 2006. **73**: p. 95-101.
153. Huiskes R., Weinans H., and Dalstra M., *Adaptive bone remodeling and biomechanical design considerations for noncemented total hip arthroplasty*. Orthopedics, 1989. **12**: p. 1255-1267.
154. Weinans H., Huiskes R., and Grootenboer H.J., *The behaviour of adaptive bone-remodelling simulation models*. Journal of Biomechanics, 1992. **25**(12): p. 1425-1441.

155. Mullender M.G., Huiskes R., and Weinans H., *A physiological approach to the simulation of bone remodeling as a selforganizational control process*. Journal of Biomechanics, 1994. **27**(11): p. 1389–1394.
156. Prendergast P.J., *Finite element models in tissue mechanics and orthopaedic implant design*. Clinical Biomechanics, 1997. **12**(6): p. 343–366.
157. Jacobs C.R., Levenston M.E., Beaupre G.S., Simo J.C., and Carter D.R., *Numerical instabilities in bone remodelling simulations: the advantages of a node-based finite element approach*. Journal of Biomechanics, 1995. **28**(4): p. 449-459.
158. Jacobs C.R., Simo J.C., Beaupré G.S., and Carter D.R., *Adaptive bone remodeling incorporating simultaneous density and anisotropy considerations*. Journal of Biomechanics, 1997. **30**(6): p. 603–613.
159. Chen G., Pettet G., Pearcy M., and McElwain D.L.S., *Comparison of two numerical approaches for bone remodelling*. Medical Engineering & Physics, 2007. **29**: p. 134–139.
160. Jang I.G. and Kim I.Y., *Computational simulation of simultaneous cortical and trabecular bone change in human proximal femur during bone remodeling*. Journal of Biomechanics, 2010. **43**: p. 294–301.
161. Jang I.G. and Kim I.Y., *Application of design space optimization to bone remodeling simulation of trabecular architecture in human proximal femur for higher computational efficiency*. Finite Elements in Analysis and Design, 2010. **46**(4): p. 311-319.
162. Rodrigues H., Jacobs C., Guedes J., and Bendsøe M., *Global and local material optimization applied to anisotropic bone adaptation*, in *Synthesis in Bio Solid Mechanics*, Perdersen P. and Bendsøe M., Editors. 1999, Kluwer Academic Publishers. p. 209–220.
163. Coelho P.G., Fernandes P., Guedes J., and Rodrigues H., *A hierarchical model for concurrent material and topology optimization of three-dimensional structures*. Structural and Multidisciplinary Optimization, 2008. **35**: p. 107–115.
164. Coelho P.G., Fernandes P.R., Rodrigues H.C., Cardoso J.B., and Guedes J.M., *Numerical modeling of bone tissue adaptation - A hierarchical approach for bone apparent density and trabecular structure*. Journal of Biomechanics, 2009. **42**.
165. Shefelbine S.J., Augat P., Claes L., and Simon U., *Trabecular bone fracture healing simulation with finite element analysis and fuzzy logic*. Journal of Biomechanics, 2005. **38**: p. 2440–2450.
166. Kim W. and Voloshin A.S., *Role of plantar fascia in the load bearing capacity of the human foot*. Journal of Biomechanics, 1995. **28**(9): p. 1025-1033.

167. Gefen A., *Stress analysis of the standing foot following surgical plantar fascia release*. Journal of Biomechanics, 2002. **35**(5): p. 629-637.
168. Cheung J.T.M, Zhang M., and An K.N., *Effect of Achilles tendon loading on plantar fascia tension in the standing foot*. Clinical Biomechanics, 2006. **21**: p. 194–203.
169. Cheng H.Y.K., Lin C.L., Wang H.W., and Chou S.W., *Finite element analysis of plantar fascia under stretch—The relative contribution of windlass mechanism and Achilles tendon force*. Journal of Biomechanics, 2008. **41**(9): p. 1937-1944.
170. Beaupré G.S., Orr T.E., and Carter D.R., *An approach for timedependent bone modelling and remodelling. Theoretical development*. Journal of Orthopaedic Research, 1990. **8**(5): p. 651–661.
171. Beaupré G.S., Orr T.E., and Carter D.R., *An approach for timedependent bone modelling and remodelling. A preliminary remodelling simulation*. Journal of Orthopaedic Research, 1990. **8**(5): p. 662–670.
172. Rossi J.M. and Wendling-Mansuy S., *A topology optimization based model of bone adaptation*. Computer Methods in Biomechanics and Biomedical Engineering, 2007. **10**(6): p. 419–427.
173. Doblaré M. and García J.M., *Anisotropic bone remodelling model based on a continuum damage-repair theory*. Journal of Biomechanics, 2002. **35**(1): p. 1-17.
174. Doblare M. and Garcia J.M., *Application of an anisotropic bone-remodellingmodel based on a damage-repair theory to the analysis of the proximal femur before and after total hip replacement*. Journal of Biomechanics, 2001. **34**: p. 1157–1170.
175. Doblare M., Cueto E., Calvo B., et al., *On the employ of meshless methods in biomechanics*. Computer Methods in Applied Mechanics and Engineering, 2005. **194**: p. 801–821.
176. Martínez R.J., García J.M., Domínguez J., and Doblaré M., *A bone remodelling model including the directional activity of BMUs*. Biomechanics and Modeling in Mechanobiology, 2009. **8**: p. 111–127.
177. Rabczuk T. and Belytschko T., *A three-dimensional large deformation meshfree method for arbitrary evolving cracks*. Computer Methods in Applied Mechanics and Engineering, 2007. **196**(29-30): p. 2777-2799.
178. Huang S., Zhang S., Belytschko T., Terdalkar S.S., and Zhu T., *Mechanics of nanocrack: Fracture, dislocation emission, and amorphization*. Journal of the Mechanics and Physics of Solids, 2009. **57**(5): p. 840-850.

179. Song J.H. and Belytschko T., *Multiscale aggregating discontinuities method for micro–macro failure of composites*. Composites Part B: Engineering, 2009. **40**(6): p. 417-426.
180. Hart R.T. and Davy D.T., *Theories of Bone Modeling and Remodeling*, in *Bone Mechanics*, C. S.C., Editor. 1989, CRC Press: Boca Raton. p. 253-277.
181. Mullender M.G. and Huiskes R., *Osteocytes and bone lining cells: which are the best candidates for mechano-sensors in cancellous bone?* Bone, 1997. **20**(6): p. 527–532.
182. García J.M., Rueberg T., and Doblare M., *A bone remodelling model coupling microdamage growth and repair by 3D BMU-activity*. Biomechanics and Modeling in Mechanobiology, 2005. **4**(2-3): p. 147-167.
183. Prendergast P.J. and Taylor D., *Design of intramedullary prostheses to prevent bone loss: predictions based on damage-stimulated remodelling*. Journal of Biomechanical Engineering, 1992. **14**(6): p. 499-506.
184. Garcia J.M., Martinez M.A., and Doblare M., *An anisotropic internal–external bone adaptation model based on a combination of CAO and continuum damage mechanics technologies*. Computer Methods in Biomechanics and Biomedical Engineering, 2001. **4**(4): p. 355-378.
185. Dolbow J. and Belytschko T., *An Introduction to Programming the Meshless Element Free Galerkin Method*. Archives in Computational Mechanics, 1998. **5**(3): p. 207-241.