

THESIS SUBMITTED TO FACULDADE DE ENGENHARIA DA  
UNIVERSIDADE DO PORTO AS A REQUIREMENT TO OBTAIN THE  
MSc DEGREE IN MECHANICAL ENGINEERING

---

# Elastoplastic Analysis Of 2D And 3D Structural Elements Using Meshless Methods

---

**Author:**

Miguel Couto Aires

**Under the supervision of:**

Professor Jorge Américo Oliveira Pinto Belinha

**Under the co-supervision of:**

Professor Renato Manuel Natal Jorge

Professor Lúcia Maria de Jesus Simas Dinis

Porto, June 2019



---

”The only way to escape  
the corruptible effect of praise  
is to go on working.”  
*Albert Einstein*



# Agradecimentos

Como autor deste trabalho, gostaria de agradecer primeiramente ao meu orientador, Professor Doutor Jorge Belinha. Desde a escolha do tema da dissertação, cujo seu apoio foi uma ajuda fundamental, por todo o auxílio e disponibilidade durante a execução deste projeto, pelo conhecimento que me transmitiu não só sobre mecânica computacional mas também sobre temas basilares da minha formação. Um sincero obrigado pela oportunidade de poder trabalhar consigo, foi uma experiência que nunca esquecerei.

Aos meus amigos, os meus sinceros agradecimentos, pelo constante apoio e crença nas minhas capacidades quando eu mesmo duvidei delas. Sem o vosso apoio, esta dissertação e todo o meu percurso, passado, presente e futuro, não teria este sucesso.

Agradeço aos meus colegas de curso e de dissertação, que fizeram comigo este percurso que agora termina. Torna-se, então, num adeus, ou num até já, à vida académica mas a introdução numa nova fase, fora dela, pelo que podem contar sempre com o meu apoio. Desejo-vos as maiores felicidades.

Por último, e apenas porque em memória viva permanecem as últimas palavras, uma sincera palavra de apreço ao meu pai, Sérgio Aires. Ao meu avô e avó, Zeferino Nobre e Judite Couto. Os vossos nomes escrevo em extenso, pela extensão da vossa influência em tudo o que sou e fui capaz de fazer.

Termino com a minha mãe, de nome completo, Maria da Graça Pinto da Costa e Couto. Tudo o que sou hoje e o que me irei tornar no futuro, a ti te devo. Foste tu que me permitiste não só academicamente mas durante a minha breve existência, ter todas as condições possíveis para chegar a este ponto. Reconheço todos os sacrifícios por ti feitos em meu nome, e estarei eternamente grato. Não existe forma mensurável para evidenciar o quanto valor tens, pelo que espero que estas palavras se aproximem desse ideal, de alguma forma. Foste, és e serás a pedra basilar na qual eu me apoio, e a quem eu dedico, na íntegra, esta dissertação.



# Institutional Acknowledgments

The author truly acknowledges the work conditions provided by the Applied Mechanics Division (SMAp) of the department of mechanical engineering (DEMec) of Faculty of Engineering of the University of Porto (FEUP), and by the MIT-Portugal project “MIT-EXPL/ISF/0084/2017”, funded by Massachusetts Institute of Technology (USA) and “Ministério da Ciência, Tecnologia e Ensino Superior - Fundação para a Ciência e a Tecnologia” (Portugal).

Additionally, the author gratefully acknowledges the funding of Project NORTE-01-0145-FEDER-000022 - SciTech - Science and Technology for Competitive and Sustainable Industries, cofinanced by Programa Operacional Regional do Norte (NORTE2020), through Fundo Europeu de Desenvolvimento Regional (FEDER).

Finally, the author acknowledges the synergetic collaboration with the collaborators of “Computational Mechanics Research Laboratory CMech-Lab” (ISEP/FEUP/INEGI), and its director, Prof.Dr. Jorge Belinha, and its senior advisors, Prof.Dr. Renato Natal Jorge and Prof.Dr. Lúcia Dinis.



# Abstract

The need to design structures and structural elements that are most efficient in terms of performance is a key aspect of engineering. In order for a given material to be used to its maximum capacity, the consideration of non-linear characteristics is mandatory. The nonlinear regime is a subject of extreme interest for this same reason, and an area with intense research activity.

For both linear and non-linear analysis, finite element method (FEM) *software* packages, whether commercial or *in-house*, have greatly contributed to facilitate the analysis of simple and complex structures with various work conditions.

However, specifying for the non-linear analysis, the FEM, being the most common tool in the analysis of linear problems, presents not only certain difficulties in non-linear problems but also some problems in the element discretization of more complex geometries, especially with geometric discontinuities. In order to solve these problems, meshless methods, such as the Radial Point Interpolation Method (RPIM) and the Natural Neighbor Radial Point Interpolation Method (NNRPIM) are used. The capability to analyze non-linear problems and the discretization flexibility of physical domains makes these methods valuable tools for this type of analysis and more, being currently used as a viable solution to the FEM.

In this thesis, the RPIM and NNRPIM are used for the analysis of several cases of structural elements submitted to various working conditions, both linear and non-linear (elastoplastic).

**Keywords** - Meshless Methods, RPIM, NNRPIM, FEM, Elastoplasticity, Structural Elements, Structures.



# Resumo

A necessidade de projectar estruturas e elementos estruturais que, em termos de desempenho, tenham o máximo de eficiência é um aspecto fundamental da engenharia. De forma a que um dado material consiga ser utilizado à sua máxima capacidade, a consideração de características não lineares é mandatória. O regime não linear é um tópico de elevado interesse por esta mesma razão, e uma área com intensa actividade de pesquisa.

Tanto para análise linear e não linear, os pacotes de *software* do método dos elementos finitos (MEF), sejam eles comerciais ou *in-house*, muito contribuíram para facilitar a análise de estruturas simples e complexas com as mais diversas condições de carregamento.

Contudo, e especificando para a análise não-linear, o MEF, sendo a ferramenta mais comum na análise de problemas lineares, apresenta não só certas dificuldades em problemas não lineares como também alguns problemas na discretização de geometrias mais complexas, especialmente com descontinuidades geométricas. De forma a solucionar estes problemas, os métodos sem malha, como o Radial Point Interpolation Method (RPIM) e o Natural Neighbour Radial Point Interpolation Method (NNRPIM) são utilizados. A boa capacidade de análise de problemas não lineares como também a flexibilidade de discretização de domínios tornam estes métodos ferramentas valiosas para este tipo de análise e não só, sendo actualmente utilizados como uma solução viável ao MEF.

Nesta tese, o RPIM e o NNRPIM são usados para a análise de vários casos de elementos estruturais submetidos a diversas condições de carregamento, tanto em regime linear como não-linear (elastoplástico).

***Palavras-Chave*** - Métodos Sem Malha, RPIM, NNRPIM, FEM, Elastoplasticidade, Elementos Estruturais, Estruturas.



# Contents

|          |                                                                 |           |
|----------|-----------------------------------------------------------------|-----------|
| <b>1</b> | <b>Introduction</b>                                             | <b>1</b>  |
| 1.1      | Meshless Method . . . . .                                       | 1         |
| 1.2      | Plasticity . . . . .                                            | 2         |
| 1.3      | Purpose of this Thesis . . . . .                                | 2         |
| 1.4      | Thesis Structure . . . . .                                      | 2         |
| <b>2</b> | <b>Scientific Review</b>                                        | <b>4</b>  |
| 2.1      | Finite Element Method . . . . .                                 | 4         |
| 2.2      | Finite Element Method in Elastoplasticity . . . . .             | 5         |
| 2.3      | Meshless Method . . . . .                                       | 7         |
| 2.4      | Meshless Method in Elastoplasticity . . . . .                   | 8         |
| <b>3</b> | <b>Meshless Method</b>                                          | <b>10</b> |
| 3.1      | Standard Procedure of a Meshless Method . . . . .               | 10        |
| 3.2      | Radial Point Interpolation Method . . . . .                     | 12        |
| 3.2.1    | Influence-Domains and Nodal Connectivity . . . . .              | 12        |
| 3.2.2    | Numerical Integration . . . . .                                 | 13        |
| 3.2.3    | Approximation or Interpolation Functions . . . . .              | 14        |
| 3.3      | Natural Neighbour Radial Point Interpolation Method . . . . .   | 17        |
| 3.3.1    | Natural Neighbours . . . . .                                    | 18        |
| 3.3.2    | Influence-Cells and Nodal Connectivity . . . . .                | 20        |
| 3.3.3    | Numerical Integration . . . . .                                 | 20        |
| <b>4</b> | <b>Fundamentals of Solid Mechanics</b>                          | <b>23</b> |
| 4.1      | Strain Field, Stress Tensor and Equilibrium Equations . . . . . | 23        |
| 4.2      | Galerkin Strong and Weak Form Formulations . . . . .            | 25        |
| 4.2.1    | Galerkin Weak Form . . . . .                                    | 25        |
| 4.3      | Discrete System of Equations . . . . .                          | 28        |
| 4.4      | Boundary Conditions . . . . .                                   | 29        |
| <b>5</b> | <b>Continuum Mechanics of Plasticity</b>                        | <b>31</b> |
| 5.1      | Overview on Plasticity . . . . .                                | 31        |
| 5.2      | Yield Criterion . . . . .                                       | 33        |
| 5.3      | Hardening Rule . . . . .                                        | 35        |
| 5.3.1    | Isotropic Hardening . . . . .                                   | 35        |
| 5.3.2    | Kinematic Hardening . . . . .                                   | 37        |

---

|          |                                                                                                                         |           |
|----------|-------------------------------------------------------------------------------------------------------------------------|-----------|
| 5.3.2.1  | Bauschinger Effect . . . . .                                                                                            | 37        |
| 5.3.3    | Independent Hardening . . . . .                                                                                         | 38        |
| 5.4      | Flow Rule . . . . .                                                                                                     | 39        |
| 5.5      | Matrix Formulation . . . . .                                                                                            | 40        |
| <b>6</b> | <b>Non-Linear Solution Algorithms</b>                                                                                   | <b>43</b> |
| 6.1      | Incremental-Iterative Methods . . . . .                                                                                 | 43        |
| 6.1.1    | Combined Incremental And Newton-Raphson Method (KT-ALL)                                                                 | 44        |
| 6.1.2    | Combination Of Incremental Predictors With Modified Newton-Raphson Iterations (KT-1) . . . . .                          | 45        |
| 6.1.3    | Initial Stress Method Combined With An Incremental Solution (KT-0) . . . . .                                            | 46        |
| 6.2      | Stress-Returning Algorithm . . . . .                                                                                    | 47        |
| 6.3      | KT-ALL Algorithm - Mathematical Formulation . . . . .                                                                   | 49        |
| <b>7</b> | <b>Numerical Analysis</b>                                                                                               | <b>51</b> |
| 7.1      | Cook's Membrane . . . . .                                                                                               | 51        |
| 7.1.1    | Geometry, Boundary and Load Conditions . . . . .                                                                        | 51        |
| 7.1.2    | Mechanical Properties . . . . .                                                                                         | 52        |
| 7.1.3    | Numerical Methods And Nodal Discretization . . . . .                                                                    | 52        |
| 7.1.4    | Results . . . . .                                                                                                       | 54        |
| 7.1.4.1  | Vertical Displacement Of Point <b>C</b> . . . . .                                                                       | 55        |
| 7.1.4.2  | Principal Stress $\sigma_1$ Of Point <b>A</b> . . . . .                                                                 | 56        |
| 7.1.4.3  | Calculation Of The Difference Between The Values Obtained By The Numerical Methods And The Reference Solution . . . . . | 56        |
| 7.1.4.4  | $\sigma_x$ Stress Along The $y$ Axis Of The Fixed Edge Of The Membrane . . . . .                                        | 59        |
| 7.1.4.5  | Von Mises Effective Stress . . . . .                                                                                    | 60        |
| 7.2      | Elastoplastic Analysis Of Uniformly Loaded Plates . . . . .                                                             | 62        |
| 7.2.1    | Geometry, Boundary And Load Conditions . . . . .                                                                        | 62        |
| 7.2.2    | Mechanical Properties . . . . .                                                                                         | 63        |
| 7.2.3    | Reference Sections And Points . . . . .                                                                                 | 63        |
| 7.2.4    | Load/Normalized Displacement Of Point <b>P</b> . . . . .                                                                | 63        |
| 7.2.4.1  | Load/Normalized Displacement Of Point <b>P</b> For The Clamped Plate . . . . .                                          | 65        |
| 7.2.4.2  | Load/Normalized Displacement Of Point <b>P</b> For The Simply Supported Plate . . . . .                                 | 67        |
| 7.2.5    | Stress Diagrams Along The Clamped Plate Thickness . . . . .                                                             | 69        |
| 7.2.5.1  | $\sigma_x, \sigma_y, \tau_{xz}, \tau_{yz}$ stress diagrams along section A-A' . . .                                     | 70        |
| 7.2.5.2  | $\sigma_x, \sigma_y, \tau_{xy}, \tau_{yz}$ stress diagrams along section B-B' . . .                                     | 73        |
| 7.2.5.3  | $\tau_{xy}$ stress diagrams along section C-C' . . . . .                                                                | 75        |
| 7.2.5.4  | $\sigma_x, \sigma_y, \tau_{xz}, \tau_{yz}$ stress diagrams along section D-D' . . .                                     | 76        |
| 7.2.6    | Stress Diagrams Along The Simply Supported Plate Thickness . . .                                                        | 77        |
| 7.2.6.1  | $\sigma_x, \sigma_y, \tau_{xz}, \tau_{yz}$ stress diagrams along section A-A' . . .                                     | 78        |
| 7.2.7    | Effective Stress Distribution Maps . . . . .                                                                            | 79        |

---

|         |                                                                                          |     |
|---------|------------------------------------------------------------------------------------------|-----|
| 7.2.7.1 | Effective Stress Distribution Maps For The Clamped Plate . . . . .                       | 80  |
| 7.2.7.2 | Effective Stress Distribution Maps For The Simply Supported Plate . . . . .              | 84  |
| 7.3     | Elastoplastic Analysis Of 2D Frames . . . . .                                            | 88  |
| 7.3.1   | Two Bay Asymmetric Frame . . . . .                                                       | 88  |
| 7.3.1.1 | Geometry, Boundary and Load Conditions . . . . .                                         | 88  |
| 7.3.1.2 | Mechanical Properties . . . . .                                                          | 88  |
| 7.3.1.3 | Horizontal Displacement of Point <b>A</b> . . . . .                                      | 89  |
| 7.3.1.4 | Effective Plastic Strain Distribution Maps - Development Of The Plastic Hinges . . . . . | 89  |
| 7.3.1.5 | Evolution Of The Effective Stress On The Plastic Hinges                                  | 91  |
| 7.3.2   | Bowstring Bridge . . . . .                                                               | 97  |
| 7.3.2.1 | Geometry, Boundary and Load Conditions . . . . .                                         | 97  |
| 7.3.2.2 | Mechanical Properties . . . . .                                                          | 97  |
| 7.3.2.3 | Vertical Displacement Of Point <b>A</b> . . . . .                                        | 98  |
| 7.3.2.4 | Effective Plastic Strain Maps - Development . . . . .                                    | 98  |
| 7.4     | Elastoplastic Analysis Of 3D Structures . . . . .                                        | 103 |
| 7.4.1   | Two Bay Asymmetric Frame . . . . .                                                       | 103 |
| 7.4.1.1 | Horizontal Displacement Of Point <b>A</b> . . . . .                                      | 103 |
| 7.4.1.2 | Effective Plastic Strain Distribution Maps - Development Of The Plastic Hinges . . . . . | 103 |
| 7.4.1.3 | Evolution Of The Effective Stress On The Plastic Hinges                                  | 105 |
| 7.4.1.4 | Comparison Between The 2D And 3D Results . . . . .                                       | 105 |
| 7.4.2   | Bowstring Bridge . . . . .                                                               | 111 |
| 7.4.2.1 | Vertical Displacement Of Point <b>A</b> . . . . .                                        | 111 |
| 7.4.2.2 | Effective Plastic Strain Maps - Development . . . . .                                    | 111 |
| 7.4.2.3 | Comparison Between The 2D and 3D Results . . . . .                                       | 112 |
| 7.5     | Numerical Analysis of Three-Point Bending Test On 3D Printed PLA Specimens . . . . .     | 116 |
| 7.5.1   | Three-Point Bending Test . . . . .                                                       | 116 |
| 7.5.2   | Manufacturing and Geometry . . . . .                                                     | 116 |
| 7.5.3   | Load and Boundary Conditions . . . . .                                                   | 117 |
| 7.5.4   | Mechanical properties of PLA . . . . .                                                   | 117 |
| 7.5.5   | Numerical Results . . . . .                                                              | 119 |
| 7.5.5.1 | Effective Stress/Effective Strain . . . . .                                              | 120 |

## 8 Conclusions 121



# List of Figures

|     |                                                                                                                                                                                                 |    |
|-----|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| 3.1 | <b>a)</b> Solid Domain. <b>b)</b> Regular discretization. <b>c)</b> Irregular discretization                                                                                                    | 11 |
| 3.2 | <b>a)</b> Gaussian integration mesh fitted to the problem domain. <b>b)</b> General Gaussian integration mesh. . . . .                                                                          | 11 |
| 3.3 | <b>a)</b> Fixed rectangular influence-domains. <b>b)</b> Fixed circular influence-domain. <b>c)</b> Flexible circular shaped influence-domains . . . . .                                        | 13 |
| 3.4 | Problem domain discretization in randomly distributed nodes . . . . .                                                                                                                           | 18 |
| 3.5 | <b>a)</b> Initial nodal set of potential neighbour nodes. <b>b)</b> Construction of the cell containing only neighbour nodes. <b>c)</b> Voronoï cell. <b>d)</b> Voronoï diagram. [38] . . . . . | 19 |
| 3.6 | <b>a)</b> First degree influence-cell. <b>b)</b> Second degree influence cell. [38] . . .                                                                                                       | 20 |
| 3.7 | <b>a)</b> Initial Voronoï diagram. <b>b)</b> Delaunay tessellation. . . . .                                                                                                                     | 21 |
| 3.8 | <b>a)</b> Sub-cell obtained through the overlapping of the Voronoï diagram and the Delaunay tessellation. <b>b)</b> Quadrature points following the Gauss-Legendre integration scheme. . . . .  | 22 |
| 4.1 | Continuous solid subjected to different kind of forces . . . . .                                                                                                                                | 26 |
| 5.1 | Elastoplastic material behaviour. <b>a)</b> Elastic-perfectly plastic model. <b>b)</b> Bilinear elastoplastic model. <b>c)</b> Non-linear elastoplastic model. . . . .                          | 31 |
| 5.2 | Uniaxial stress-strain diagram for arbitrary hardening . . . . .                                                                                                                                | 32 |
| 5.3 | Yield surface . . . . .                                                                                                                                                                         | 33 |
| 5.4 | Yield surface for the von Mises and Tresca yield criterion . . . . .                                                                                                                            | 34 |
| 5.5 | Hardening rules. <b>a)</b> Isotropic Hardening. <b>b)</b> Kinematic Hardening. <b>c)</b> Independent Hardening. . . . .                                                                         | 36 |
| 5.6 | Idealized stress-strain curve with Bauschinger effect for uniaxial strain hardening. . . . .                                                                                                    | 38 |
| 5.7 | Representation of the flow rule (orthogonality principle). . . . .                                                                                                                              | 39 |
| 6.1 | Incremental methods. <b>a)</b> Incremental (Euler) scheme. <b>b)</b> Newton-Raphson method . . . . .                                                                                            | 44 |
| 6.2 | Combined incremental and Newton-Raphson method (KT-ALL) . . . .                                                                                                                                 | 45 |
| 6.3 | Combination of incremental predictors with modified Newton-Raphson iterations (KT-1) . . . . .                                                                                                  | 46 |
| 6.4 | Initial stress method combined with an incremental solution (KT-0) . .                                                                                                                          | 47 |
| 6.5 | Backward-Euler scheme . . . . .                                                                                                                                                                 | 48 |
| 7.1 | <b>a)</b> Cook's Membrane 2D geometry. <b>b)</b> Cook's Membrane 3D Geometry.                                                                                                                   | 52 |

|      |                                                                                                                                                                                                                       |    |
|------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| 7.2  | Nodal discretization for the 2D case. <b>a)</b> Discretization using 25 nodes (coarsest mesh used). <b>b)</b> Discretization using 81 nodes. <b>c)</b> Discretization using 4225 nodes (finest mesh used). . . . .    | 54 |
| 7.3  | Nodal discretization for the 3D case. <b>a)</b> Discretization using 75 nodes (coarsest mesh used). <b>b)</b> Discretization using 243 nodes. <b>c)</b> Discretization using 3267 nodes (finest mesh used). . . . .   | 54 |
| 7.4  | Vertical displacement of point <b>C</b> . <b>a)</b> 2D. <b>b)</b> 3D. . . . .                                                                                                                                         | 55 |
| 7.5  | Principal stress of point <b>A</b> . . . . .                                                                                                                                                                          | 57 |
| 7.6  | $\sigma_x$ stress along the fixed edge of the membrane. <b>a)</b> 2D. <b>b)</b> 3D . . . . .                                                                                                                          | 59 |
| 7.7  | $\sigma_{eff}$ stress distribution along the Cook's Membrane for the 2D case. <b>a), b)</b> and <b>c)</b> FEM, RPIM and NNRPIM for 25 nodes. <b>d), e)</b> and <b>f)</b> FEM, RPIM and NNRPIM for 4225 nodes. . . . . | 60 |
| 7.8  | $\sigma_{eff}$ stress distribution along the Cook's Membrane for the 3D case. <b>a), b)</b> and <b>c)</b> FEM, RPIM and NNRPIM for 75 nodes. <b>d), e)</b> and <b>f)</b> FEM, RPIM and NNRPIM for 3267 nodes. . . . . | 61 |
| 7.9  | Square plate representation . . . . .                                                                                                                                                                                 | 62 |
| 7.10 | Boundary conditions for the square plate considering symmetry. <b>a)</b> Simply Supported Plate. <b>b)</b> Clamped Plate. . . . .                                                                                     | 63 |
| 7.11 | Reference point and sections for both boundary conditions. <b>a)</b> Simply Supported Plate. <b>b)</b> Clamped Plate. . . . .                                                                                         | 64 |
| 7.12 | Vertical displacement of point <b>P</b> for the three non-linear solution algorithms using the FEM . . . . .                                                                                                          | 65 |
| 7.13 | Vertical displacement of point <b>P</b> for the three non-linear solution algorithms using the RPIM . . . . .                                                                                                         | 66 |
| 7.14 | Vertical displacement of point <b>P</b> for the three non-linear solution algorithms using the NNRPIM . . . . .                                                                                                       | 66 |
| 7.15 | Vertical displacement of point <b>P</b> for the three non-linear solution algorithms using the FEM . . . . .                                                                                                          | 67 |
| 7.16 | Vertical displacement of point <b>P</b> for the three non-linear solution algorithms using the RPIM . . . . .                                                                                                         | 68 |
| 7.17 | Vertical displacement of point <b>P</b> for the three non-linear solution algorithms using the NNRPIM . . . . .                                                                                                       | 68 |
| 7.18 | $\sigma_x$ and $\sigma_y$ stress diagrams along section A-A' . . . . .                                                                                                                                                | 70 |
| 7.19 | $\tau_{xz}$ and $\tau_{yz}$ stress diagrams along section A-A' . . . . .                                                                                                                                              | 71 |
| 7.20 | $\sigma_x$ and $\sigma_y$ stress diagrams along section B-B' . . . . .                                                                                                                                                | 73 |
| 7.21 | $\tau_{xy}$ and $\tau_{yz}$ stress diagrams along section B-B' . . . . .                                                                                                                                              | 74 |
| 7.22 | $\tau_{xy}$ stress diagrams along section C-C'. . . . .                                                                                                                                                               | 75 |
| 7.23 | $\sigma_x$ and $\sigma_y$ stress diagrams along section D-D' . . . . .                                                                                                                                                | 76 |
| 7.24 | $\tau_{xz}$ and $\tau_{yz}$ stress diagrams along section D-D' . . . . .                                                                                                                                              | 77 |
| 7.25 | $\sigma_x$ and $\sigma_y$ stress diagrams along section A-A' . . . . .                                                                                                                                                | 78 |
| 7.26 | $\tau_{xz}$ and $\tau_{yz}$ stress diagrams along section A-A' . . . . .                                                                                                                                              | 79 |
| 7.27 | Effective stress distribution maps obtained by the FEM. . . . .                                                                                                                                                       | 80 |
| 7.28 | Effective stress distribution maps obtained by the RPIM. . . . .                                                                                                                                                      | 81 |
| 7.29 | Effective stress distribution maps obtained by the NNRPIM. . . . .                                                                                                                                                    | 82 |
| 7.30 | Effective stress distribution maps obtained by the FEM. . . . .                                                                                                                                                       | 84 |

---

---

|      |                                                                                                  |     |
|------|--------------------------------------------------------------------------------------------------|-----|
| 7.31 | Effective stress distribution maps obtained by the RPIM. . . . .                                 | 85  |
| 7.32 | Effective stress distribution maps obtained by the NNRPIM. . . . .                               | 86  |
| 7.33 | Geometry, boundary and load conditions of the two bay asymmetric frame. . . . .                  | 88  |
| 7.34 | Cross-section dimensions. . . . .                                                                | 89  |
| 7.35 | Horizontal displacement of point <b>A</b> given by the FEM . . . . .                             | 90  |
| 7.36 | Horizontal displacement of point <b>A</b> given by the RPIM . . . . .                            | 90  |
| 7.37 | Horizontal displacement of point <b>A</b> given by the NNRPIM . . . . .                          | 91  |
| 7.38 | Plastic hinges in chronological order. . . . .                                                   | 91  |
| 7.39 | Development of the plastic hinges for the FEM . . . . .                                          | 92  |
| 7.40 | Development of the plastic hinges for the RPIM . . . . .                                         | 93  |
| 7.41 | Development of the plastic hinges for the NNRPIM . . . . .                                       | 94  |
| 7.42 | Evolution of the effective stress on the plastic hinges given by the FEM                         | 95  |
| 7.43 | Evolution of the effective stress on the plastic hinges given by the RPIM                        | 95  |
| 7.44 | Evolution of the effective stress on the plastic hinges given by the NNR-PIM . . . . .           | 96  |
| 7.45 | Dimensions of the bowstring bridge in $[mm]$ . . . . .                                           | 97  |
| 7.46 | Boundary and load conditions of the bowstring bridge . . . . .                                   | 98  |
| 7.47 | Vertical displacement of point <b>A</b> of the 2D bowstring bridge . . . . .                     | 99  |
| 7.48 | Development of the effective plastic strain for the FEM . . . . .                                | 100 |
| 7.49 | Development of the effective plastic strain for the RPIM . . . . .                               | 101 |
| 7.50 | Development of the effective plastic strain for the NNRPIM . . . . .                             | 102 |
| 7.51 | Thickness variation and additional boundary condition for the 3D model - Upper view. . . . .     | 103 |
| 7.52 | Horizontal displacement of point <b>A</b> given by the FEM . . . . .                             | 104 |
| 7.53 | Horizontal displacement of point <b>A</b> given by the RPIM . . . . .                            | 104 |
| 7.54 | Horizontal displacement of point <b>A</b> given by the NNRPIM . . . . .                          | 105 |
| 7.55 | Development of the plastic hinges for the FEM - 3D case . . . . .                                | 106 |
| 7.56 | Development of the plastic hinges for the RPIM - 3D case . . . . .                               | 107 |
| 7.57 | Development of the plastic hinges for the NNRPIM - 3D case . . . . .                             | 108 |
| 7.58 | Evolution of the effective stress on the plastic hinges given by the FEM - 3D case . . . . .     | 109 |
| 7.59 | Evolution of the effective stress on the plastic hinges given by the RPIM - 3D case . . . . .    | 109 |
| 7.60 | Evolution of the effective stress on the plastic hinges given by the NNR-PIM - 3D case . . . . . | 110 |
| 7.61 | 2D and 3D results for the horizontal displacement of point <b>A</b> . . . . .                    | 110 |
| 7.62 | Vertical displacement of point <b>A</b> of the 3D bowstring bridge . . . . .                     | 111 |
| 7.63 | Development of the effective plastic strain for the FEM - 3D case . . . . .                      | 112 |
| 7.64 | Development of the effective plastic strain for the RPIM - 3D case . . . . .                     | 113 |
| 7.65 | Development of the effective plastic strain for the NNRPIM . . . . .                             | 114 |
| 7.66 | 2D and 3D results for the vertical displacement of point <b>A</b> . . . . .                      | 115 |
| 7.67 | Three-point flexural test representation . . . . .                                               | 116 |
| 7.68 | Printing direction of the specimen . . . . .                                                     | 117 |
| 7.69 | Geometry and dimensions of the specimen . . . . .                                                | 117 |

---

7.70 Boundary and load conditions for the beam . . . . . 118

7.71 Dimensions of the specimens used in compressive tests . . . . . 118

7.72 Results obtained from the compressive tests . . . . . 119

7.73 2D and 3D results for the effective stress/effective strain . . . . . 120

# List of Tables

|      |                                                                                                                                       |     |
|------|---------------------------------------------------------------------------------------------------------------------------------------|-----|
| 3.1  | Polynomial basis functions for 2D . . . . .                                                                                           | 15  |
| 3.2  | Polynomial basis functions for 3D . . . . .                                                                                           | 16  |
| 5.1  | Possible situations resulting from the plastic multiplier and stress point                                                            | 40  |
| 7.1  | Mechanical Properties of both 2D and 3D Cook's Membrane . . . . .                                                                     | 52  |
| 7.2  | 2D meshless method parameters . . . . .                                                                                               | 53  |
| 7.3  | 3D meshless method parameters . . . . .                                                                                               | 53  |
| 7.4  | Relationship between the number of elements along the edge of the geometry and the thickness of the model . . . . .                   | 54  |
| 7.5  | Vertical displacement values obtained for the different numerical methods used for the 2D case. . . . .                               | 58  |
| 7.6  | Vertical displacement values obtained for the different numerical methods used for the 3D case. . . . .                               | 58  |
| 7.7  | Value of $\delta$ for the FEM, RPIM and NNRPIM considering the reference value for the vertical displacement for the 2D case. . . . . | 58  |
| 7.8  | Value of $\delta$ for the FEM, RPIM and NNRPIM considering the reference value for the vertical displacement for the 3D case. . . . . | 58  |
| 7.9  | Mechanical properties of the plate. . . . .                                                                                           | 63  |
| 7.10 | Mechanical properties of the two bay asymmetric frame. . . . .                                                                        | 89  |
| 7.11 | Mechanical properties of the bowstring bridge . . . . .                                                                               | 98  |
| 7.12 | Elastic modulus values obtained via the compressive tests . . . . .                                                                   | 119 |
| 7.13 | Mechanical properties of PLA . . . . .                                                                                                | 119 |



# Chapter 1

## Introduction

In this work, meshless methods are applied in the elastoplastic analysis of 2D and 3D structural elements. The Radial Point Interpolation Method (RPIM) and the Natural Neighbour Radial Point Interpolation Method (NNRPIM), are used for the analysis of such elements.

### 1.1 Meshless Method

In recent years, meshless methods had a significant development, being considered a viable alternative to the finite element method (FEM), in engineering and science as a whole. Computational mechanics is the discipline concerned with the use of computational methods to study phenomena governed by the principles of mechanics. It is used every day to help predict, develop and improve not only structural and fluid applications in the industrial sense, but also in medicine and other important areas of science [1]. With the constant evolution of engineering, the challenges for computational mechanics are numerous. For example, when analyzing the simulation of manufacturing processes such as extrusion or molding, it is necessary to deal with large displacements of the mesh, while in simulations of failure processes, the modeling of crack propagation is highly complex.

These and other problems are not quite suited for conventional methods such as the FEM, due to the dependency of the mesh and the treatment of discontinuities. Also, the treatment of complex structures can be a challenge, due to, again, the need of a element mesh for the analysis [2].

Meshless methods are an alternative to the classical methods. The problem domain is modeled by a random set of nodes rather than an element mesh. The concept of influence-domain is used instead of elements, and the nodes can be distributed in a random fashion [3, 4].

These and other characteristics make meshless methods ideal for the resolution of problems involving complex geometries [5].

## 1.2 Plasticity

The mechanical behaviour of structures is often analyzed using a linear analysis. However, the existence of non-linear phenomena applied to structures during their working conditions is a reality and they need to be accounted for.

Non-linear behaviour can bring certain economic and safety benefits to structural projects, being that after the material surpasses its yield limit, it is not necessarily true that the structure composed of such material is on its way to collapse.

In reality, that same structure can still withstand load. Accounting for elastoplastic behaviour leads to a more complete project, not only in safety terms but also in economic terms, with dimensions of profiles and structural elements being inferior to the ones used in a all-elastic domain dimensioning.

The need for thinner, lighter and more resistant mechanical and structural elements lead to the development and research in elastoplastic theory, being that it is commonly used today in engineering endeavours as a way to reduce costs and increase the safety of structural elements.

## 1.3 Purpose of this Thesis

The purpose of this thesis is the elastoplastic analysis of 2D and 3D structural elements using not only the **Finite Element Method [FEM]**, but also 2 promising meshless methods, the **Radial Point Interpolation Method [RPIM]** and **Natural Neighbour Radial Point Interpolation Method [NNRPIM]**.

Although several research papers and thesis explore elastoplastic analysis of structural elements using the FEM, meshless methods have been less used for this purpose. Therefore, the analysis of these elements, including plates, frames and 3D printed elements, using meshless methods and the results that can be achieved using such methods is considered an interesting research topic and one that adds important knowledge to the computational mechanics community and to the scientific community as a whole.

## 1.4 Thesis Structure

This work is organized in the following manner:

- In Chapter 2, a scientific review on the FEM and the application of it to elastoplasticity is made. In addition, a review on meshless methods and, also, the application on elastoplasticity is presented.
- Chapter 3 presents a detailed description of the meshless methods used throughout this work, including the mathematical formulation, pre and pos-processing.
- In Chapter 4, the fundamentals of solid mechanics are presented, including a detailed mathematical explanation of the associated calculations that these methods perform in order to return the desired results.

- Chapter 5 exposes fundamental concepts of plasticity, including the yield criterion, flow and hardening rule.
- The non-linear solution algorithms necessary to solve elastoplastic problems are presented in Chapter 6, including the stress-returning algorithm.
- In Chapter 7, the numerical analysis of several selected problems are presented and analyzed. A convergence test is made, and the elastoplastic analysis of plates and frames, in 2D and 3D, is presented. Moreover, experimental work involving additive manufacturing is shown.
- Chapter 8 contains the main conclusions of the developed work and future work is suggested.



# Chapter 2

## Scientific Review

In this Chapter, a literature survey concerning the topics being developed in this thesis is performed. There is a concern to perform the survey by chronological order of technique development. Therefore, first, an approach to the finite element method is made. In sequence, works combining the FEM and plasticity are referenced. At last, a review in meshless methods is made, and the Chapter ends with the recent developments combining the meshless methods with non-linear analysis of structures.

### 2.1 Finite Element Method

The finite element method can be interpreted as an algorithm for numerically solving the differential and integral equations governing the problem under study. The FEM allows the analysis of the variables that represent a physical system, being that in structural analysis, it is used to obtain the displacements, stresses and strains of a structure [6].

The first discrete variational method and formulation of the approximate solution within the hole domain can be attributed to W.Ritz in 1909 [7].

In 1915, B.G. Galerkin [8] introduced the concepts of orthogonality of the residuum and the weight function, including the formulation of the approximate solution and the weight function within the whole domain.

The basis of the finite element method is the matrix analysis of structures. William Weaver and James M. Gere [9] introduced the concepts of flexibility and stiffness methods in structural analysis, being given an emphasis on the latter, due to a more adequate application for computer programming.

The work of J.H. Argyris and S. Kelsey [10] is a fundamental basis of the development of the finite element method, in which the generalization and extension of the energy principles on the elastic analysis of structures is made. These principles are a pillar of the FEM, such as the principle of virtual work or the minimal potential energy principle. Approximately at the same time, M.J. Turner, R.W. Clough, H.C. Martin and L.J. Topp [11] formulated for the first time the FEM based on the principle of virtual work, combined with the derivation of a triangular plane stress element with six degrees of freedom.

R.W. Clough [12], in 1960 introduces for the first time the term 'finite element'.

The first book in which the FEM is comprehensively defined and studied as a whole can be attributed to Zienkiewicz and Taylor [13]. The Galerkin method of approximation [14] is studied and used as a way to calculate the stiffness matrix of a structure. Topics such as the solution of non-linear algebraic equations are developed and inelastic and non-linear materials are studied. Also, discrete element methods are described and a classical set of chapters including the analysis of rods and plate bending considering the theories of Kirchhoff [15] and Reissner-Mindlin [16] for plates and shells is presented. Non-linear structural analysis is also approached.

From this point onward, the expansion and generalization of the finite element method and variations was truly made. Books such as [17, 18, 19] are considered classics in this field, all of them with an approach to computer programming and computational mechanics, more specifically the work of Young [19] is a solid reference.

Currently, the FEM is used in the analysis of various materials like composites [20]. It is also used in other areas besides engineering. Computational modeling is used in several medical applications such as orthopedics, cardiovascular, surgery and dentist [21].

## 2.2 Finite Element Method in Elastoplasticity

The finite element method using isoparametric elements applied to elastoplastic behaviour of 3D structures was analyzed for the first time by Ajaya K. Gupta, Bijan Mohraz and William C. Schnobrich [22] in 1972. A three-dimensional isoparametric element was formulated and used in an "initial" stress plastic behaviour. The element was applied to an infinitely long hollow circular cylinder under uniform internal pressure and to a simply supported beam subjected to a uniform lateral load. At the time, investigators avoided the analysis and application to three-dimensional structures due to the fact that a two-dimensional approach simplified significantly the calculation and was more economical to solve.

The results obtained for the cylinder were in an excellent agreement with the analytic values existent at the time. For the beam however, a discrepancy existed between the values, due to a "loss" in stresses in critical regions which did not allow the plastification of the material to occur. A possible solution for this issue presented by the author would be the use of a finer grid in the direction of the stress gradient.

In 1981, J.H. Argyris, B. Boni, U. Hindenlang and M. Kleiber [23] analyzed with the finite element method large deformations in elasto-plastic frames. Attention was focused on the step by step analysis of the nonlinear behaviour of such structures and a natural formulation was used in terms of the generalized cross-sectional forces. The analysis was applied to various cases, from a simple case such as a rectangular cross-section beam clamped at both ends and subjected to a concentrated load, to a more elaborate case as the three-dimensional frame dome. The different plastic behaviour of the structures analyzed were recorded with stress-deformation graphs.

Wang Xucheng and Wang Xiaoning [24], in 1993 developed a simplified method for elastoplastic finite element analysis of hardening materials. Based on the deformation theory of linear hardening materials, a finite element analysis is proposed to calculate elastoplastic responses of general hardening materials. This method comes

as a alternative to the general elastoplastic incremental finite element method analysis. It avoids incremental iteration of nodal displacements and the constitutive equation integration at each Gauss integral point.

The elastoplastic analysis of a beam with rectangular cross-section is made, using eight 8-noded solid elements with internal freedom degrees. The results obtained, comparing with the incremental method commonly used showed good agreement, being that the method developed in the research paper is much faster to compute.

A T-joint subjected to internal pressure is also analyzed, with the results being similar to the ones obtained in the first example.

In 2006, Bahattin Kanber and O. Yavuz Bozkurt [25] used transition rectangular plate elements for the analysis of elastoplastic plate problems. As a way to verify the the developed elements, various elastoplastic problems are solved and compared with ANSYS results. Three different elastoplastic problems are solved. Making a reference only to the first example, a simply supported plate with a vertical downward force applied is analyzed. The result obtained with the transition element and the ANSYS solution have good convergence, including the fact that the solution time for the transition element is 60% lower than the regular 8-node rectangular element.

Ehsan Hashemi and Behrooz Farhsi [26], in 2014, investigated the elastoplastic behaviour of of classical and first order Timoshenko beams by a finite element formulation under two major kinematic hardening models.

The objective was the proposition of a finite element formulation with a variable stiffness matrix and a optimized solution for the Armstrong-Fredrick theory with the Zigler-Prager model for cyclic flexural loading conditions considering a controlled deformation. For symmetrical cyclic deformation and flexural controlled states, the conclusion that was reached was that the total stress-strain curve tended to be coincident.

Furthermore, it also corroborated that the anisotropic characteristics cases with symmetric loading exhibit a ratcheting response for both beam models.

In 2016, Fabio De Angelis and Robert L. Taylor [27] proposed a new computational procedure for the finite element solution of elastoplastic boundary value problems in the structural analysis of ductile materials. Local constitutive equations lead to the solution of a nonlinear scalar equation which ultimately reduces to a single variable algebraic equation in the plastic rate parameter. An integration scheme was proposed, allowing to find the analytical solution of the algebraic equation in closed form.

A important characteristic of this method is the lack of necessity for a iterative method to solve the local constitutive problem. The algorithmic scheme uses an alternative formulation of the algorithmic counterpart of the plastic consistency condition. A consistent linearization of elasto-plastic boundary value problems was used.

Numerical examples are shown, considering a tension strip with cyclic loading in tension-compression with increasing, decreasing, and same levels of loading. A Cook's membrane problem is also studied, with increasing mean value of loading. The results show that the computational procedure is well suited for applications to elasto-plastic boundary value problems subject to elaborate loading conditions.

## 2.3 Meshless Method

Meshless methods and their origin can be traced to the Smooth Particle Hydrodynamics Method (SPH) [28]. It was applied to polytropic stellar models and used statistical techniques to recover analytical expressions for the physical variables from a known distribution of fluid elements, starting with a non-axisymmetric distribution of approximately 80 particles in three dimensions.

This method was applied to solid mechanics by Larry D. Libersky and A.G. Petschek [29], in 1991, where a elastic-perfectly-plastic constitutive model is formulated. It was found that excellent features of the SPH such as robustness, simplicity and ease of adding new physics could be transferred to the analysis of solid mechanics problems [30].

The above method uses a strong formulation of the physical problem in order to solve it. An effort was made to develop meshless methods that depended on the weak form in order to find a solution. Therefore, the Element Free Galerkin Method (EFGM) [2], one of the most popular meshless methods and one of the first to depend on a global weak form was developed. Using approximation functions to produce smooth solutions, the EFGM was originated from the Diffuse Element Method (DEM) [31], the first mature meshless method, which uses moving least-squares approximants [32] to construct the shape functions.

At the same time, the Reproducing Kernel Particle Method (RKPM) [33] was developed, which maintained the attractive features of the SPH, MLS, DEM and EFGM methods. With the addition of a correction function, it gives much better results.

Another popular meshless method based on a local weak form formulation is the Meshless Local Petrov-Galerkin method [34], a truly meshless method, based on a local symmetric weak form (LSWF). The method does not need a element mesh for the approximation of the solution variables. Integrals are solved over a regular shaped domain and their boundaries.

The methods that were referenced and explained above employ approximation functions, which has a direct impact on the Kronecker delta property. Due to the dependency of approximation function to find a solution to a given problem, the aforesaid property is not satisfied. Without the satisfaction of the Kronecker delta property, the treatment of essential and boundary conditions to a physical problem is more complex.

To overcome this issue, interpolator meshless methods were developed.

The Point Interpolator Method (PIM) [35] represents the problem domain by scattered points and uses a technique to construct the polynomial interpolants with the Kronecker delta property.

The Radial Point Interpolation Method (RPIM) [36] is based on combining radial and polynomial basis functions. The use of radial based functions overcomes some possible cases of singularity that occur on meshless methods based only on polynomial functions.

The Natural Neighbour Radial Point Interpolation Method (NNRPIM) [37, 38] combines the natural neighbour finite element method with the radial point interpolation method. The nodal connectivity and the node dependent integration background mesh are constructed resorting to the Voronoï tessellation and to the Delaunay tri-

angulation. This method guarantees that the shape functions developed have the Kronecker delta property, and results show high accuracy and convergence rate.

## 2.4 Meshless Method in Elastoplasticity

Somnath Ghosh and Suresh Moorthy [39], in 1994, developed a Voronoi Cell finite element method to solve small deformation and elastoplastic problems for arbitrary heterogeneous materials. Furthermore, a composite Voronoi Cell finite element method is also developed.

Numerical examples are presented, considering plane strain analysis of a thick homogeneous cylindrical pressure vessel subjected to internal pressure and a thin rectangular plate with a hole analyzed in plane stress. The results show excellent agreement with the theoretical and experimental data.

The introduction of the geometrical concept of Voronoi Cells and tessellation in a numerical model is of great importance since the Natural Neighbour Radial Point Interpolation Method (NNRPIM) uses these concepts in its meshless form. Taking into account the importance of these geometrical ideas for the method referred to above, and its application to elastoplastic theory, a scientific review of the paper was considered relevant in this section.

In 1999, William Barry and Sunil Saigal [40] developed a Element Free Galerkin formulation for small strain, three-dimensional, elastic and elastoplastic regime. The Moving Least-Squares were used for the determination of the shape functions and a variable domain of influence EFGM is introduced. The elastoplastic formulation was based on the incremental formulations for non-linear analysis and the consistent tangent operator.

For elastoplastic examples, a thick walled pressure vessel with internal pressure and a extension of a strip with a circular hole were studied. Comparing with reference solutions [41], the method showed fast convergence with the data, comparable to that obtained in the FEM.

In 2004, Belinha [42] used the Element Free Galerkin Method (EFGM) in the analysis of two-dimensional problems, plates and laminates in elasto-plastic regime. Various modified Newton-Rapson algorithms were used for the non-linear solution. A Cook membrane, made of isotropic and sandwich material under distributed load is studied, with the results being compared to FEM solutions, showing excellent convergence.

In the elastoplastic domain, plates subjected to axial forces and bending forces were studied, including laminated plates with alternating materials. The influence of the non-linear solution algorithms in the computational cost is also analyzed. The results showed that the EFGM was applied with success to the problems proposed, obtaining similar results to the FEM analysis.

Dinis et al. [43] in 2011, applied the NNRPIM to non-linear analysis, considering large deformation of elastic and elasto-plastic structures. Validating the method in the large deformation elastoplastic analysis, two non-linear solution algorithm are used: the Newton-Rapson initial stiffness method and the efficient "forward-Euler" procedure.

A 3D frame dome is analyzed using the NNRPIM, with two distinct meshes. The obtained results were precise and very close to the reference solutions. The stress field

obtained was smooth and accurate and it was found that the solution obtained with the method was not significantly affected by the mesh discretization.

Cheng et al. [44], in 2014, developed a novel interpolating EFG method for two-dimensional elasto-plasticity. Using interpolating moving least-squares method (IMLS) in order to obtain the shape functions, the Kronecker delta property is satisfied, and the essential boundary conditions can be applied directly. The method has some advantages such as a simpler formulae and the direct application of the boundary conditions, as referred to above.

Numerical examples are studied, such as a cantilever beam subjected to a concentrated and distributed load, where the results show that the IEFMG is in excellent agreement with those of ANSYS software, concerning the displacements of the beam. It is also shown that the IEFMG has a higher precision than the EFG method.

Malgorzata A. Jankowska [45], in 2018, developed a numerical method based on the method of fundamental solutions (MFS) and the method of particular solutions (MPS). The material hardening due to plastic deformation is characterized by the Chakrabarty model [46] and the meshless method developed has its basis on the radial basis function collocation method [47].

Numerical examples such as a plate with a narrowing subjected to uniaxial extension and a perforated plate subjected to uniaxial extension are contemplated. The results showed very good agreement with the FEM solutions also presented in the paper, including the stress distribution throughout the geometries considered.

Recently, in 2019, Belinha et al [48] extended the radial point interpolating meshless method to the elastoplastic analysis of aluminium alloys. Two efficient meshless methods were used, the RPIM and NNRPIM. In order to obtain comparable results, due to the lack of a suitable elastoplastic solution, an experimental test was performed in order to obtain the force/displacement curve and the corresponding mechanical properties. A dog-bone shaped specimen was elasto-plastically analyzed, under uniaxial tensile state condition.

Two different influence-cell degrees were considered, first degree and second degree, with the results showing that the second degree influence cell returned a smooth result, while the first degree cell had some discontinuities. Moreover, the results obtained with either the RPIM and NNRPIM and both influence-cells showed good convergence with the experimental data.

The major advantage of the NNRPIM is highlighted: The only requirement necessary to analyze any numerical problem is the computational nodal distribution, making it a truly "meshless" method. However, the major disadvantage of the method is also pointed: the pre-processing phase presents a higher computational cost when compared to the FEM and RPIM, due to the time spent in the determination of the natural neighbours and the construction of the integration mesh.

# Chapter 3

## Meshless Method

In this chapter, the meshless methods used throughout this work, the Radial Point Interpolation Method (RPIM) and the Natural Neighbour Radial Point Interpolation Method (NNRPIM), are presented. Firstly, the standard procedure of a meshless method is presented. Subsequently, the formulation for the RPIM and NNRPIM is introduced. Concepts such as influence-domain and nodal connectivity are explained, including the integration scheme and the interpolation function that are used in both methods. Moreover, for the NNRPIM, the notion of influence-cells and how they are constructed using mathematical concepts such as Voronoï diagrams is referred.

### 3.1 Standard Procedure of a Meshless Method

The majority of meshless methods follow a generic procedure. After the problem geometry and the natural and essential boundary conditions are described and established, the problem domain and the respective boundary are discretized using a nodal set that can follow a regular or irregular distribution. Similarly to mesh dependent methods, a higher density of nodes generally returns more accurate results. Generally, irregular distributions have a lower accuracy when compared to regular distributions (Figure 3.1(b)). Nonetheless, in specific problems where a particular geometrical detail can have a considerable effect on the stress distribution (fracture mechanics, orifices, specific boundary conditions), a higher nodal density in those locations is required to obtain better results (Figure 3.1(c)).

The referred nodal distribution does not form a mesh. Truly meshless methods do not require any kind of previous information about the relationship between each node to construct the interpolation functions. The only information necessary is the location of each node used.

After the nodal discretization of the problem, the background integration mesh is constructed. This mesh can be nodal dependent, as in the NNRPIM, or nodal independent, as in the RPIM. The size of the integration mesh can vary, as it can have the size of the problem domain (Figure 3.2(a)) or even a larger one (Figure 3.2(b)).

Nodal independent integration meshes generally use the Gaussian integration method, as applied to the FEM, while nodal dependent integration meshes, as used in the NNRPIM, use nodal integration and concepts such as natural neighbours and Voronoï dia-

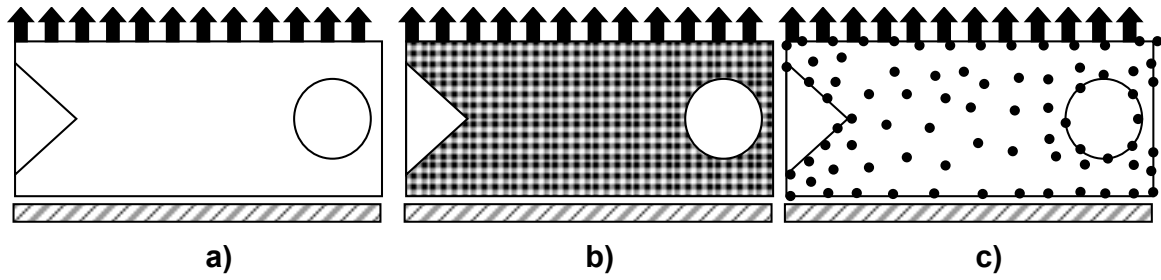


**Figure 3.1:** a) Solid Domain. b) Regular discretization. c) Irregular discretization

grams to integrate the weak form equations. In nodal integration schemes, the nodal distribution of the problem domain will also become the integration mesh.

With the nodal distribution and integration mesh defined and constructed, the nodal connectivity can be imposed. In the FEM, a mesh dependent method using finite elements, the relationship between nodes is automatically imposed by the interaction between neighbour elements (no-overlap rule between elements), and also at the finite element level (nodes that belong to the same element interact with each other).

In most meshless methods, the nodal connectivity is defined by the overlap of influence domains. Therefore, for a certain interest point  $\mathbf{x}_I$  of the problem domain, areas or volumes centered at said point are defined, with the nodes inside these areas or volumes belonging to the influence-domain of node  $\mathbf{x}_i$ .

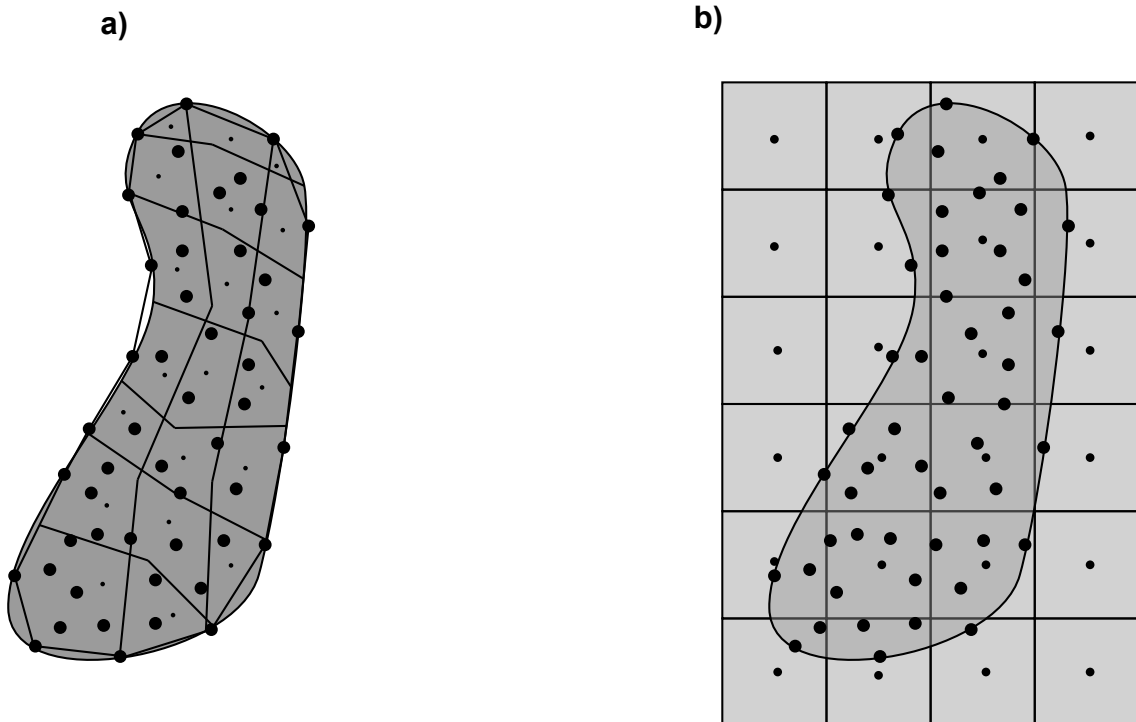


**Figure 3.2:** a) Gaussian integration mesh fitted to the problem domain. b) General Gaussian integration mesh.

Regarding the NNRPIM, areas (for 2D problems) and volumes (for 3D prob-

lems) are defined using the mathematical concept of Voronoï diagrams, and are called Voronoï cells.

The cells are created based on the nodal distribution alone, making the NNRPIM a truly meshless method, only requiring the nodal distribution of the problem domain. For most meshless methods, the interest points are the integration points from the background integration mesh. With that being said, there are meshless methods in which the interest points are the nodes of the nodal distribution. As a recommendation, all influence-domains should possess, in approximation, the same number of nodes inside. Therefore, the size of the influence-domain is dependent of the nodal density in the premise of the interest point  $\mathbf{x}_I$ .

The next phase of the procedure is obtaining the field variables using approximation or interpolation functions. The two meshless methods used in this work use the same approximation functions.

Consider the displacement field  $\mathbf{u}$ . The displacement components  $\mathbf{u}_I = (u, v, w)$ , at a determined interest point  $\mathbf{x}_I$  that belongs to the problem domain, are approximated using the nodal displacement of the nodes inside the influence-domain or influence-cell of said interest point. Therefore, the following equation can be established for the  $O_x$  component of the displacement,

$$u(\mathbf{x}_I) = \sum_{j=1}^n \varphi_j(\mathbf{x}_I) u(\mathbf{x}_i) \quad (3.1)$$

where  $n$  is the number of nodes inside the influence-domain or influence-cell of the interest point  $\mathbf{x}_I$ ,  $\mathbf{u}_i$  can be obtained by repeating Eq. 3.1 for each degree of freedom:  $\mathbf{u}_I$ ,  $\mathbf{v}_I$  and  $\mathbf{w}_I$ , being  $\mathbf{u}_i$  a vector considering the displacement components of each node that is included in the influence-domain or influence-cell and  $\varphi_j(\mathbf{x}_I)$  is the approximation or interpolation function value at the  $j$ th node, considering the  $n$  nodes of the influence-domain or influence cell.

The next step is to set the equation system. The discrete system of equations can be determined using the interpolation or approximation functions applied to the weak or strong form. These equations are then arranged in a local nodal matrix form and assembled into a global equation system matrix. For static problems, the displacement field is obtained.

## 3.2 Radial Point Interpolation Method

### 3.2.1 Influence-Domains and Nodal Connectivity

For the RPIM, nodal connectivity is obtained by the overlapping of the influence-domain of each node. These influence-domains are defined after the nodal discretization and integration mesh, as mentioned earlier.

In this work, 2D and 3D influence-domains are considered. For each interest point, the influence-domain can assume various shapes. For the two-dimensional case, influence-domains can be circular or rectangular areas. As for the three-dimensional case, the same applies, however we are considering volumes instead of areas.

The size and the shape variation of the influence-domain can have effects on the performance and the final solution of the method. All influence-domains should have the same or approximately the same number of nodes inside. In irregular nodal distributions, influence-domains hardly have the same number of nodes inside each one of them, which affects the final solution to the problem, as explained earlier in the chapter.

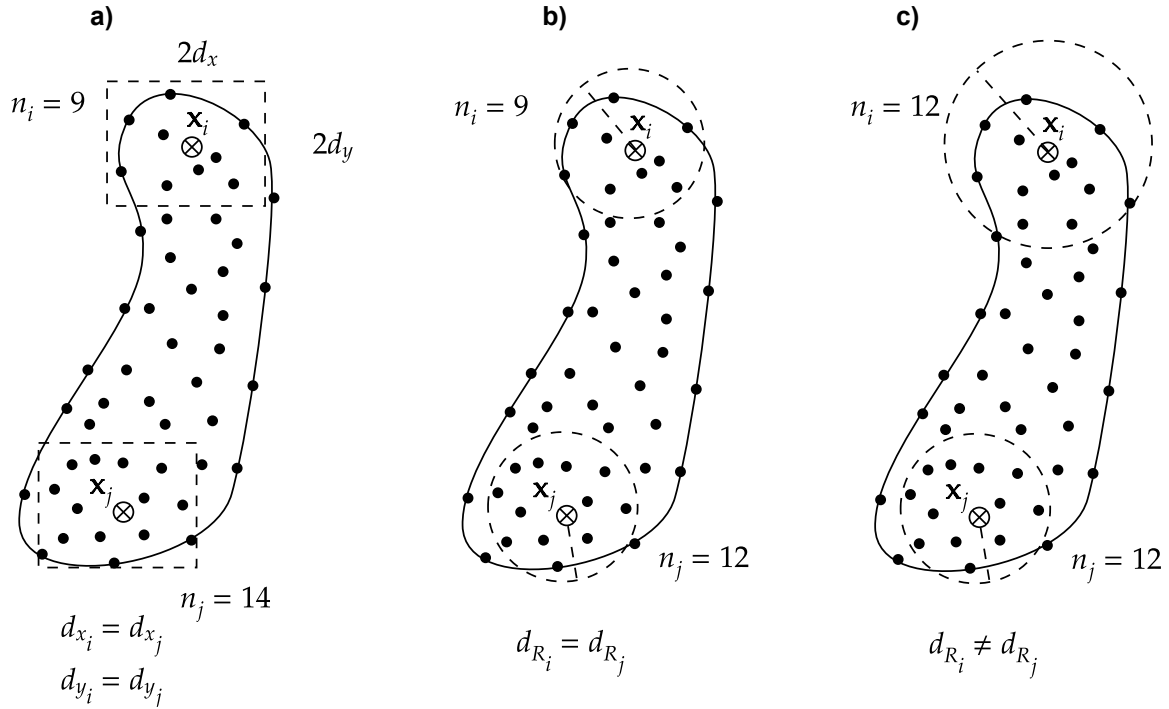


**Figure 3.3:** a) Fixed rectangular influence-domains. b) Fixed circular influence-domain. c) Flexible circular shaped influence-domains

Observing Figure 3.3(a) and (b), we can see that fixed size influence-domains in a irregular distribution may result in a different number of nodes inside each one of them. On the other hand, flexible shaped influence-domains (Figure 3.3(c)), although their size is not constant throughout the problem domain, all of them have the same number of nodes inside, depending on the interest point. According to [36], each influence-domain should possess between  $n = 9$  and  $n = 16$  nodes. The uniformity in the number of nodes inside each influence-domain permits the construction of shape-functions with the same level of complexity.

### 3.2.2 Numerical Integration

As explained earlier in the chapter, the RPIM uses the Gauss-Legendre quadrature integration method to integrate the Galerkin weak form differential equations. Therefore, it is necessary to create a background integration mesh.

Using Gauss-Legendre integration, the problem domain is divided in a regular grid, as Figure 3.2(a) and (b) shows. Each grid division has integration points within it, using the Gauss-Legendre integration rule.

The Gauss-Legendre integration scheme is ruled by the following equation,

$$I = \int_{-1}^1 \int_{-1}^1 f(\xi, \eta) d\xi d\eta = \sum_{i=1}^{n\xi} \sum_{j=1}^{n\eta} f(\xi_i, \eta_j) w_i w_j \quad (3.2)$$

where  $w_i$  and  $w_j$  are the weights of the integration point,  $f(\xi, \eta)$  the function defined in the problem domain and  $f(\xi_i, \eta_j)$  the value of said function at the integration point.

### 3.2.3 Approximation or Interpolation Functions

The RPIM and NNRPIM use the same interpolation functions, therefore the formulation for the radial point interpolation (RPI) functions will be presented in the present section and not repeated in a following section for the NNRPIM.

Consider a function  $u(x)$  defined in the domain  $\omega$ , domain which is discretized by a set of  $N$  nodes. It is assumed that only the nodes that are inside the influence-domain of the interest point  $\mathbf{x}_I$  have effect on the function  $u(x)$ . For the RPIM, said function passes through all nodes using a radial basis function (RBF). Considering the value of the function at the interest point,  $u(\mathbf{x}_I)$ , we can establish,

$$u(\mathbf{x}_I) = \sum_{i=1}^n R_i(\mathbf{x}_I) a_i(\mathbf{x}_I) + \sum_{j=1}^m p_j(\mathbf{x}_I) b_j(\mathbf{x}_I) = \mathbf{R}^T(\mathbf{x}_I) \mathbf{a}(\mathbf{x}_I) + \mathbf{p}^T(\mathbf{x}_I) \mathbf{b}^T(\mathbf{x}_I) \quad (3.3)$$

where  $n$  is the number of nodes within the influence-domain of interest point  $\mathbf{x}_I$ ,  $R_i(\mathbf{x}_I)$  is the radial basis function and  $a_i(\mathbf{x}_I)$  and  $b_j(\mathbf{x}_I)$  are non-constant coefficients of  $R_i(\mathbf{x}_I)$  and  $p_j(\mathbf{x}_I)$ , respectively, where  $p_j(\mathbf{x}_I)$  defines the monomials of the polynomial basis, with  $m$  being the basis monomial number.

The vectors present in Eq.(3.3) are defined by,

$$\begin{aligned} \mathbf{R}^T(\mathbf{x}_I) &= \{R_1(\mathbf{x}_I), R_2(\mathbf{x}_I), \dots, R_n(\mathbf{x}_I)\} \\ \mathbf{p}^T(\mathbf{x}_I) &= \{p_1(\mathbf{x}_I), p_2(\mathbf{x}_I), \dots, p_m(\mathbf{x}_I)\} \\ \mathbf{a}^T(\mathbf{x}_I) &= \{a_1(\mathbf{x}_I), a_2(\mathbf{x}_I), \dots, a_n(\mathbf{x}_I)\} \\ \mathbf{b}^T(\mathbf{x}_I) &= \{b_1(\mathbf{x}_I), b_2(\mathbf{x}_I), \dots, b_m(\mathbf{x}_I)\} \end{aligned} \quad (3.4)$$

Eq.(3.3) can be rewritten,

$$u(\mathbf{x}_I) = \{\mathbf{R}^T(\mathbf{x}_I), \mathbf{p}^T(\mathbf{x}_I)\} \begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} \quad (3.5)$$

In order to obtain a more stable function, the number of monomial  $m$  of the polynomial basis should be  $m < n$ . The purpose of adding the polynomials into the basis function is to ensure the consistency of the RPI functions, moreover adding to the RBF a linear polynomial ensures  $C^1$  consistency and helps the RPI pass the standard patch test.

However, Belinha et al [49] determined that that the polynomial basis function is not necessary for the RPI to pass the standard patch test. This fact leads to reduced computational cost and results in shape functions only dependant on the RBF.

A Multiquadratic Radial Basis Function (MQ-RBF), proposed initially by Hardy [50] is used in this work,

$$R(r_{ij}) = (r_{ij}^2 + c^2)^p \quad (3.6)$$

with  $c$  and  $p$  being the shape parameters that need to be optimized in a effort to maximize the performance of the method.

In earlier works on the RPI [36, 51], optimal values for the shape functions are proposed,  $c = 1,42$  and  $p = 1,03$ . In fact, the proposed values are inaccurate and are poor in stability and precision [37, 49]. One of the most relevant problems of meshless methods using RBF is that the variation of the shape parameters affects the performance of the RBF in a major way, and there are no standard or precise theoretical methods for the optimal values of  $c$  and  $p$ . Belinha et al [38] performed a optimization study for the shape parameters and found that  $c = 0.0001$  and  $p = 0.9999$  are more suitable values, and these values are the ones that will be used in this work.

The Euclidean distance between the interest point  $\mathbf{x}_I$  and a node  $\mathbf{x}_i$  belonging to the influence-domain (RPIM) or influence-cell (NNRPIM), has the general form for the 2D case,

$$r_{ij} = \sqrt{(\mathbf{x}_I - \mathbf{x}_i)^2 + (\mathbf{y}_I - \mathbf{y}_i)^2} \quad (3.7)$$

As for the 3D case, we can establish,

$$r_{ij} = \sqrt{(\mathbf{x}_I - \mathbf{x}_i)^2 + (\mathbf{y}_I - \mathbf{y}_i)^2 + (\mathbf{z}_I - \mathbf{z}_i)^2} \quad (3.8)$$

The polynomial basis functions present the following sequence of terms, for 2D,

$$\mathbf{p}^T(\mathbf{x}_I) = \{1, x, y, x^2, xy, y^2, \dots\} \quad (3.9)$$

and 3D,

$$\mathbf{p}^T(\mathbf{x}_I) = \{1, x, y, z, x^2, xy, y^2, yz, z^2, \dots\} \quad (3.10)$$

The sequence of terms presented in Eq.(3.9) and Eq.(3.23) depends on the chosen monomial number  $m$ . For the 2D case, we can establish the following polynomial basis functions,

| Basis                 | Polynomial Function                                    |
|-----------------------|--------------------------------------------------------|
| Null ( $m = 0$ )      | $\mathbf{p}^T(\mathbf{x}) = \{0\}$                     |
| Constant ( $m = 1$ )  | $\mathbf{p}^T(\mathbf{x}) = \{1\}$                     |
| Linear ( $m = 3$ )    | $\mathbf{p}^T(\mathbf{x}) = \{1, x, y\}$               |
| Quadratic ( $m = 6$ ) | $\mathbf{p}^T(\mathbf{x}) = \{1, x, y, x^2, xy, y^2\}$ |

**Table 3.1:** Polynomial basis functions for 2D

As for the 3D case,

| Basis                  | Polynomial Function                                                    |
|------------------------|------------------------------------------------------------------------|
| Constant ( $m = 1$ )   | $\mathbf{p}^T(\mathbf{x}) = \{1\}$                                     |
| Linear ( $m = 4$ )     | $\mathbf{p}^T(\mathbf{x}) = \{1, x, y, z\}$                            |
| Quadratic ( $m = 10$ ) | $\mathbf{p}^T(\mathbf{x}) = \{1, x, y, z, x^2, xy, y^2, yz, z^2, zx\}$ |

**Table 3.2:** Polynomial basis functions for 3D

In this work, a constant polynomial basis is applied, in order to reduce computational cost and increase the NNRPIM accuracy. The polynomial basis has to satisfy an extra requirement in order to obtain a unique solution,

$$\sum_{i=1}^n p_j(\mathbf{x}_i) a_i(\mathbf{x}_i) = 0 \quad (3.11)$$

with  $j = \{1, 2, \dots, m\}$ .

Therefore, combining Eq.(3.3) with Eq.(3.11), a new equation matrix can be written,

$$\begin{Bmatrix} \mathbf{u}_s \\ 0 \end{Bmatrix} = \begin{bmatrix} \mathbf{R} & \mathbf{p} \\ \mathbf{p}^T & 0 \end{bmatrix} \begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} = \mathbf{G} \begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} \quad (3.12)$$

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

$$\mathbf{u}_s = \{u_1, u_2, \dots, u_n\}^T \quad (3.13)$$

with the matrix  $\mathbf{R}$ , of size  $[n \times n]$ , containing all interest points of the RBF calculated by Eq.(3.6),

$$\mathbf{R} = \begin{bmatrix} R(r_{11}) & R(r_{12}) & \dots & R(r_{1n}) \\ R(r_{21}) & R(r_{22}) & \dots & R(r_{2n}) \\ \vdots & \vdots & \ddots & \vdots \\ R(r_{n1}) & R(r_{n2}) & \dots & R(r_{nn}) \end{bmatrix} \quad (3.14)$$

and the matrix  $\mathbf{p}$ , of size  $[n \times m]$ , of the polynomial basis being given by,

$$\mathbf{p} = \begin{bmatrix} p_1(\mathbf{x}_1) & p_2(\mathbf{x}_1) & \dots & p_m(\mathbf{x}_1) \\ p_1(\mathbf{x}_2) & p_2(\mathbf{x}_2) & \dots & p_m(\mathbf{x}_2) \\ \vdots & \vdots & \ddots & \vdots \\ p_1(\mathbf{x}_n) & p_2(\mathbf{x}_n) & \dots & p_m(\mathbf{x}_n) \end{bmatrix} \quad (3.15)$$

The  $\mathbf{G}$  matrix is symmetric since the distance between the interest points and the nodes that belong to a certain influence-domain are independent in directional terms. Solving Eq.(3.12),

$$\begin{Bmatrix} \mathbf{a} \\ \mathbf{b} \end{Bmatrix} = \mathbf{G}^{-1} \begin{Bmatrix} \mathbf{u}_s \\ 0 \end{Bmatrix} \quad (3.16)$$

We can now substitute Eq.(3.16) in Eq.(3.5), obtaining,

$$u(\mathbf{x}_I) = \{\mathbf{R}^T(\mathbf{x}_I), \mathbf{p}^T(\mathbf{x}_I)\} \mathbf{G}^{-1} \begin{Bmatrix} \mathbf{u}_s \\ 0 \end{Bmatrix} = \varphi(\mathbf{x}_I) \mathbf{u}_s \quad (3.17)$$

where  $\varphi(\mathbf{x}_I)$  is the interpolation or approximation function of the interest point  $\mathbf{x}_I$ ,

$$\varphi(\mathbf{x}_I) = \{\mathbf{R}^T(\mathbf{x}_I), \mathbf{p}^T(\mathbf{x}_I)\} \mathbf{G}^{-1} = \{\varphi_1(\mathbf{x}_I), \varphi_2(\mathbf{x}_I), \dots, \varphi_n(\mathbf{x}_I)\} \quad (3.18)$$

The Galerkin weak form depends on the partial derivative of  $\varphi(\mathbf{x}_I)$ . It can be established that the partial derivative in order to a variable  $\xi$  as,

$$\varphi_\xi(\mathbf{x}_I) = \{\mathbf{R}_\xi^T(\mathbf{x}_I), \mathbf{p}_\xi^T(\mathbf{x}_I)\} \mathbf{G}^{-1} \quad (3.19)$$

with the partial derivatives of the MQ-RBF in order of the same variable  $\xi$  being,

$$R_\xi(r_{ij}) = 2p(r_{ij}^2 + c^2)^{p-1}(\xi_j - \xi_i) \quad (3.20)$$

As the RPI functions are interpolation functions that possess the Kronecker delta property, they pass through all nodes, which in turn makes the imposition of the essential and natural boundary conditions easier.

Also, they satisfy the partition of unity,

$$\sum_{i=1}^n \varphi_i(\mathbf{x}_i) = 1 \quad (3.21)$$

If the interpolation function  $\varphi(\mathbf{x}_i)$  possesses reproducing properties and the constant of the polynomial basis is included, it can be established,

$$\sum_{i=1}^n \varphi_i(\mathbf{x}_i) \mathbf{x}_i = \mathbf{x} \quad (3.22)$$

on the condition that the first order monomial is included in the polynomial basis.

### 3.3 Natural Neighbour Radial Point Interpolation Method

As referred earlier in the chapter, while the RPIM uses influence-domains in order to impose the nodal connectivity, the NNRPIM uses mathematical concepts such as Voronoï diagrams and Delaunay tessellations in order to construct the influence-cells, the fundamental structure for nodal connectivity and the background integration mesh. Again, the NNRPIM's necessary information is the nodal distribution of the problem domain, making it a truly "meshless" method.

In the forthcoming subsections, the NNRPIM formulation will be approached and these concepts will be explained.

### 3.3.1 Natural Neighbours

Sibson [52] presented the concept of natural neighbours, which is utilized in the NNRPIM to impose the nodal connectivity, combined with Voronoï diagrams for the determination of the discretized domain.

Voronoï diagrams [53] consist of a set of Voronoï cells, to which every cells has but only one node associated to it, adding to all points closer to that node than any other points in the problem domain. Only 2D procedures will be presented for the sake of simplicity. However, this procedure can be applied to a  $\mathbb{R}^d$  space.

Consider the problem domain  $\Omega$ , bounded by a physical boundary  $\Gamma$ , with a random discretization of nodes  $\mathbf{N} = \{n_0, n_1, \dots, n_n\} \in \mathbb{R}^2$ , as it is represented in the following figure.

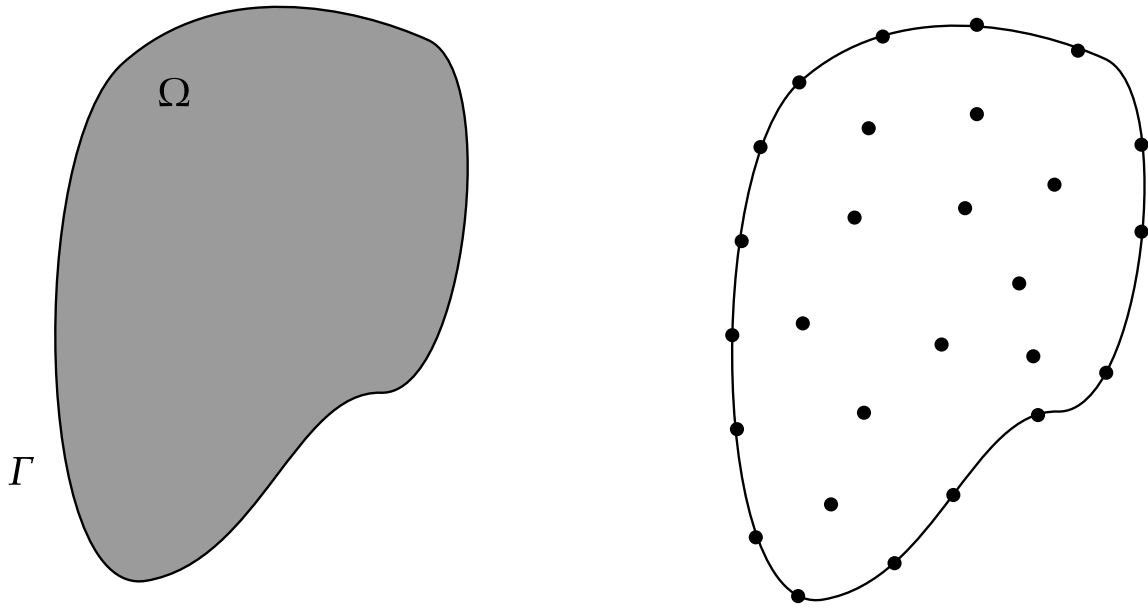


**Figure 3.4:** Problem domain discretization in randomly distributed nodes

As it can be seen in Figure 3.5(d)), the Voronoï diagram of  $\mathbf{N}$  results of the partition of the domain in sub-regions  $V_I$ , closed and convex, while each sub-region  $V_I$  is associated to node  $I$ ,  $n_I$ , in such a way that whatever point in the interior of the  $V_I$  is closer to  $n_I$  than any other node  $n_J$ , where,

$$n_J \in \mathbf{N} \wedge J \neq I \quad (3.23)$$

Therefore,  $V_I$  is the geometric place where all points are closer to  $n_I$  than any other node, with the sub-regions  $V_k$  being the "Voronoï cells" which form the Voronoï diagram, for  $\mathbf{k} = \{1, 2, \dots, \mathbf{N}\}$ .

Mathematically, the Voronoï cell can be defined,

$$V_I = \{\mathbf{x} \in \mathbb{R}^2 : E_n(\mathbf{x}, \mathbf{x}_I) < E_n(\mathbf{x}, \mathbf{x}_n) \forall J \neq I\} \quad (3.24)$$

with  $E_n(\mathbf{x}, \mathbf{x}_I)$  being the Euclidean distance between an interest point  $\mathbf{x}$  and a node  $\mathbf{x}_I$ , and  $E_n(\mathbf{x}, \mathbf{x}_J)$  the same but regarding an outside node,  $J \neq I$ .

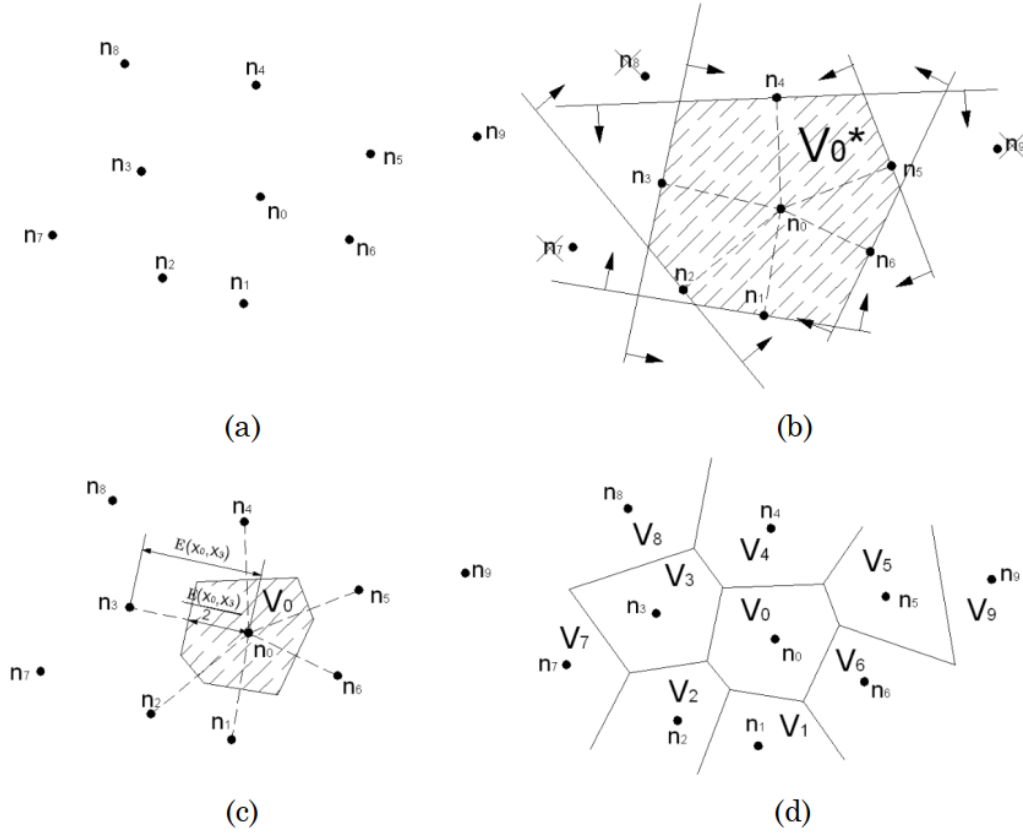


**Figure 3.5:** **a)** Initial nodal set of potential neighbour nodes. **b)** Construction of the cell containing only neighbour nodes. **c)** Voronoi cell. **d)** Voronoi diagram. [38]

The construction of the sub-region  $V_I$  is represented in Figure 3.5. For an interest node  $n_0$ , a set of potential neighbour nodes is identified (Figure 3.5(a)). A set of dashed lines, with their origin at the interest node, ending at potential neighbour nodes is drawn. At the same time, straight and full lines, which are perpendicular to the dashed ones, are drawn. Evidently, each of these lines contains a neighbour node. These neighbour nodes are obtained by the intersection of domains whose limits are defined by the lines that intersect the nodes  $J$  and are normal to the dashed lines, as referred earlier.

A provisional Voronoi cell,  $V_0^*$  is created (Figure 3.5(b)) and all nodes located outside of this cell are discarded. All nodes inside the cell are the neighbour nodes of node  $n_0$ .

The final Voronoi cell,  $V_0$ , is obtained (Figure 3.5(c)) by reducing the size of the provisional cell,  $V_0^*$ . The reduction is made by defining the distance between node  $n_0$  and the neighbour nodes as half of the Euclidean distance between node  $n_0$  and the neighbour node in question. The cell  $V_0$  is the homothetic form of  $V_0^*$ .

The same is made for every other node in the problem domain, resulting in a total Voronoi cell discretization of the geometry, as it can be seen in Figure 3.5(d)).

### 3.3.2 Influence-Cells and Nodal Connectivity

As seen earlier for the case of the RPIM, the size and shape of the "influence-domains" can and will affect the performance and accuracy of the method. In turn, for the NNRPIM, the nodal connectivity is imposed by the overlapping of the "influence-cells".

In Figure 3.6, two different types of influence-cells are represented: first degree influence-cell and second degree influence-cell. The first degree influence-cell, Figure 3.6(a)), is composed by the first natural neighbours of interest point  $\mathbf{x}_I$ , while the second degree influence-cell, Figure 3.6(b)), in addition to the natural neighbours of interest point  $\mathbf{x}_I$ , also contains the natural neighbours of the interest points' natural neighbours.

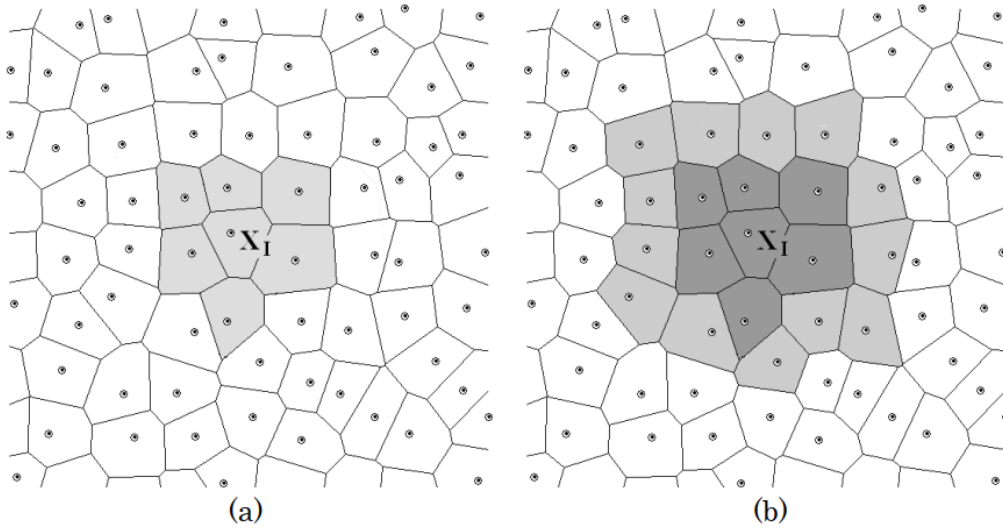


**Figure 3.6:** a) First degree influence-cell. b) Second degree influence cell. [38]

It is visible that the second degree influence-cell has a larger area and, consequently, larger nodal connectivity than the first degree influence-cell. Using higher degree influence-cells permits to achieve more accurate and precise results. Therefore, throughout this work, only second-degree influence-cells will be used.

### 3.3.3 Numerical Integration

After the Voronoi diagram is defined, Figure 3.5(d)), it is necessary to create a integration mesh in order to numerically integrate the differential equation that rules the physical problem being studied. This is possible using a particular geometric construction, the Delaunay tessellation. This geometric construct is the geometrical dual of the Voronoi diagram and it is constructed by connecting the nodes whose Voronoi cells have common boundaries, which can be seen in Figure 3.7.

Belinha et. al [38] proposed an integration scheme based on the Voronoi diagram and the Delaunay tessellation. Only the 2D procedure is explained below for the sake of simplicity, with the 3D procedure following the same steps with an extra dimension.

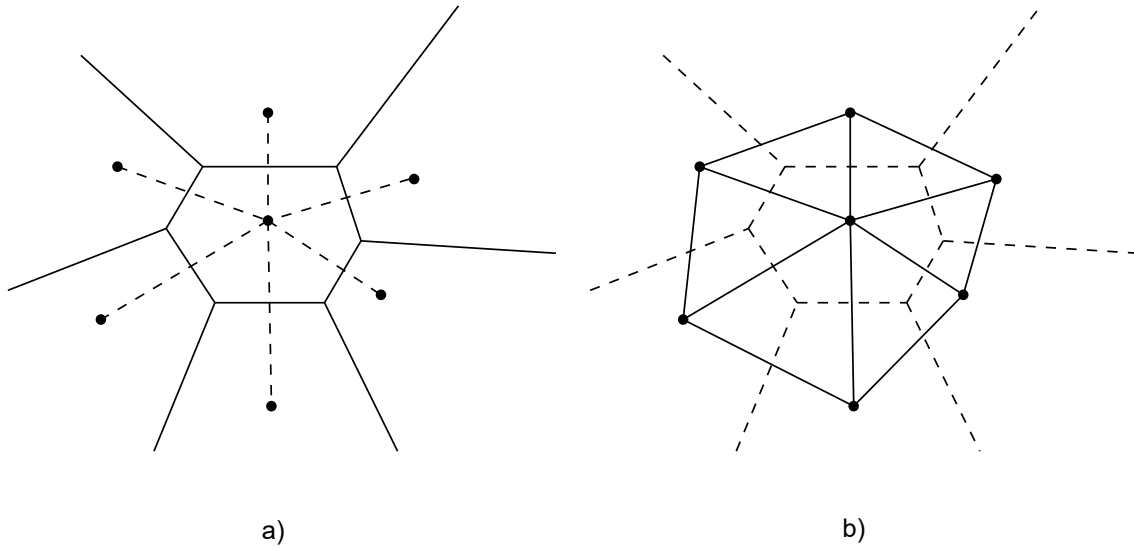


**Figure 3.7:** a) Initial Voronoi diagram. b) Delaunay tessellation.

The Delaunay tessellation allows the original Voronoi cell area,  $A^{V_j}$ , of a certain node  $\mathbf{x}_j$  to be divided in  $k$  sub-areas,  $A_i^{V_j}$ , with,

$$A^{V_j} = \sum_{i=1}^k A_i^{V_j} \quad (3.25)$$

as it can be seen in Figure 3.8(a) below.

In order to obtain the integration mesh, integration points are distributed inside each sub-area  $A_i^{V_j}$ , following the Gauss-Legendre quadrature rule, Figure 3.8(b)) for each Voronoi cell discretizing the problem domain. For this example, only one quadrature point was used. With this integration scheme, it is possible to obtain a numerical integration mesh dependent only on the nodal distribution of the problem domain.

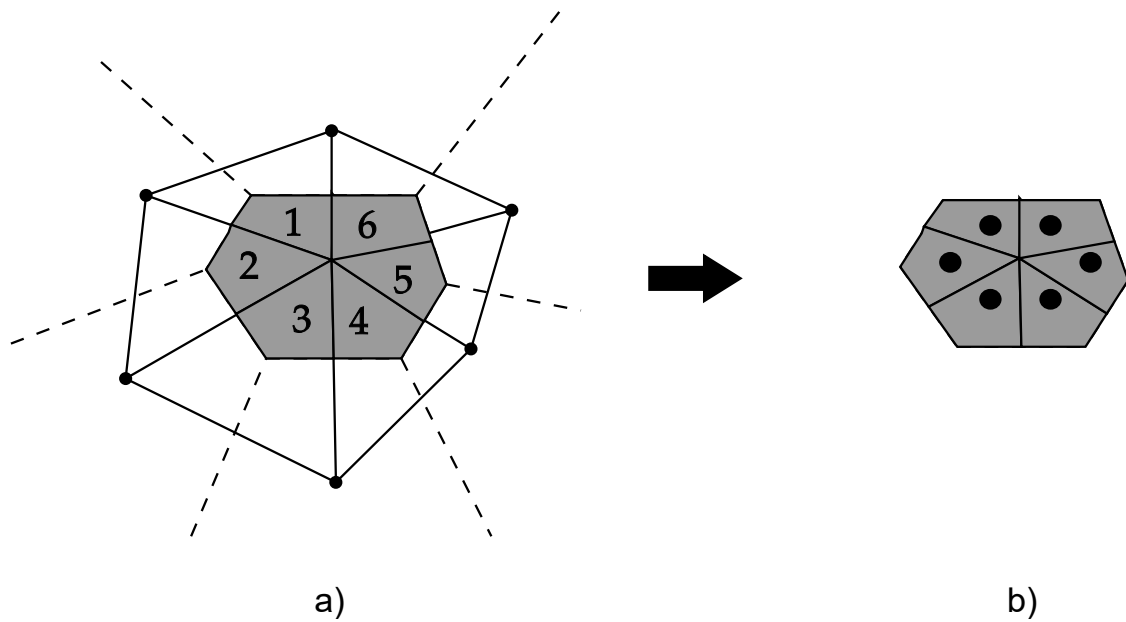


**Figure 3.8:** a) Sub-cell obtained through the overlapping of the Voronoï diagram and the Delaunay tessellation. b) Quadrature points following the Gauss-Legendre integration scheme.



# Chapter 4

## Fundamentals of Solid Mechanics

In this chapter, the fundamentals of solid mechanics behind the numerical analysis made in this work are presented. Firstly, general concepts of strain and stress fields are approached with the equilibrium equations being presented. Thereafter, the strong and weak form formulation is explained and the discrete equation system based on the Principal of Virtual Work (PVW) is deduced, with the shape functions presented in Chapter 3 being used as trial functions.

Since non-linear and linear mechanics have a common ground in subjects as kinematics, stress and equilibrium equations, a review on these subjects is crucial to the understanding of the numerical analysis made throughout this work.

### 4.1 Strain Field, Stress Tensor and Equilibrium Equations

For a 3D problem, the Cauchy stress tensor is defined as,

$$\mathbf{\Lambda} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} \quad (4.1)$$

The Cauchy tensor  $\mathbf{\Lambda}$  is a symmetric tensor and it represents the stresses of a body in a certain state.

The Voigt notation, for numerical implementation, can be a useful technique since it allows a reduced representation of the stress tensor. Applying the aforesaid notation to Eq.(4.1),

$$\boldsymbol{\sigma} = \{\sigma_{xx} \quad \sigma_{yy} \quad \sigma_{zz} \quad \sigma_{xy} \quad \sigma_{yz} \quad \sigma_{zx}\}^T \quad (4.2)$$

The constitutive matrix  $\mathbf{c}$  contains the mechanical properties of the material. For a isotropic material,

$$\mathbf{c} = \frac{E}{(1-2\nu)(1+\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \quad (4.3)$$

with  $\nu$  being the Poisson's coefficient and  $G$  being the shear modulus which can be obtain with the following equation,

$$G = \frac{E}{2(1+\nu)} \quad (4.4)$$

The strain tensor can be obtained by multiplying the constitutive matrix  $\mathbf{c}$  by the strain tensor, which is given by,

$$\boldsymbol{\varepsilon} = \{\varepsilon_{xx} \quad \varepsilon_{yy} \quad \varepsilon_{zz} \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}\}^T \quad (4.5)$$

$$\boldsymbol{\sigma} = \mathbf{c}\boldsymbol{\varepsilon} \quad (4.6)$$

The strain components can be calculated from the displacement field, which is composed of the displacements  $u$ ,  $v$  and  $w$  along the Cartesian coordinate system axis  $x$ ,  $y$  and  $z$ , respectively,

$$\begin{aligned} \varepsilon_{xx} &= \frac{\partial u}{\partial x} \\ \varepsilon_{yy} &= \frac{\partial v}{\partial y} \\ \varepsilon_{zz} &= \frac{\partial w}{\partial z} \\ \gamma_{xy} &= \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \\ \gamma_{xz} &= \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \\ \gamma_{yz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \end{aligned} \quad (4.7)$$

For an infinitesimal element, the equilibrium equations can be established,

$$\begin{aligned} \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \sigma_{xz}}{\partial z} + F_x &= 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + F_y &= 0 \\ \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} + F_z &= 0 \end{aligned} \quad (4.8)$$

where  $F_x$ ,  $F_y$  and  $F_z$  are the body forces applied to the element along the Cartesian axis.

## 4.2 Galerkin Strong and Weak Form Formulations

The differential equations that govern a physical system must obey the equilibrium equations or another kind of equilibrium equations that directly result from simplifications of the original equilibrium equations. Therefore, the strong form consists of the governing equations and the boundary conditions for physical system. This means that the aforesaid equations must be satisfied at any mathematical point of the domain. It is difficult to solve these equations when the physical system is complex, due to its form or boundary conditions.

To solve this issue, the weak form formulation is used. Instead of solving the differential equations that govern the system, the weak form produces stable algebraic systems of equations and is able to give a discretized system of equations in order to obtain the approximate solution of the problem.

At every integration point, the weak form is established, which in turn means that the precision and accuracy of the obtained solution depends on the number of nodes in which the problem domain is discretized. Thus, generally, a higher nodal density returns more accurate results than a lower one. Additionally, the imposition of the boundary conditions is much easier, since they can be applied directly at the nodes.

### 4.2.1 Galerkin Weak Form

Both the RPIM and NNRPIM use the Galerkin weak form formulation to convert the differential equations into a discrete equation system, in order to obtain approximated solutions to the strong form formulation. For this purpose, all of the admissible displacements must satisfy the following requirements:

- All of the displacements are compatible with the problem domain
- All of the displacements constraints are satisfied
- The displacement history must satisfy the constraints at the initial and final times

In order to obtain the Galerkin weak form, we must use a energy principle - the Hamilton's principle.

The Hamilton's principle allows the derivation of the partial differential equations and can be stated as follows: "Of all possible time histories of displacement states that satisfy the compatibility equations and the constraints or the kinematic boundary conditions and that also satisfy the conditions at initial and final times ( $t_1$  and  $t_2$ ), the history corresponding to the actual solution makes the Lagrangian functional a minimum." [54].

$$L = T - U + W_f \quad (4.9)$$

with  $T$  being the kinetic energy,  $U$  being the strain energy and  $W_f$  the work performed by external forces. The Lagrangian functional  $L$  contains all physical information concerning the system and the forces acting on it.

With the Lagrangian defined and invoking the Hamilton's principle presented earlier, it is possible to establish,

$$\int_{t_1}^{t_2} [\delta T - \delta U + \delta W_f] dt = 0 \quad (4.10)$$

Substituting Eq.(4.9) in Eq.(4.10),

$$\delta \int_{t_1}^{t_2} L dt = 0 \quad (4.11)$$

Consider now the solid represented in Figure 4.1, where the domain  $\Omega$  is bounded by  $\Gamma$ . It is possible to define the surface on which the external forces  $\bar{t}$  are applied and the natural boundary,  $\Gamma_t$ . Also, the essential boundary,  $\Gamma_u$ , where the displacements are constrained. Besides these, a body force  $\mathbf{b}$  is applied.

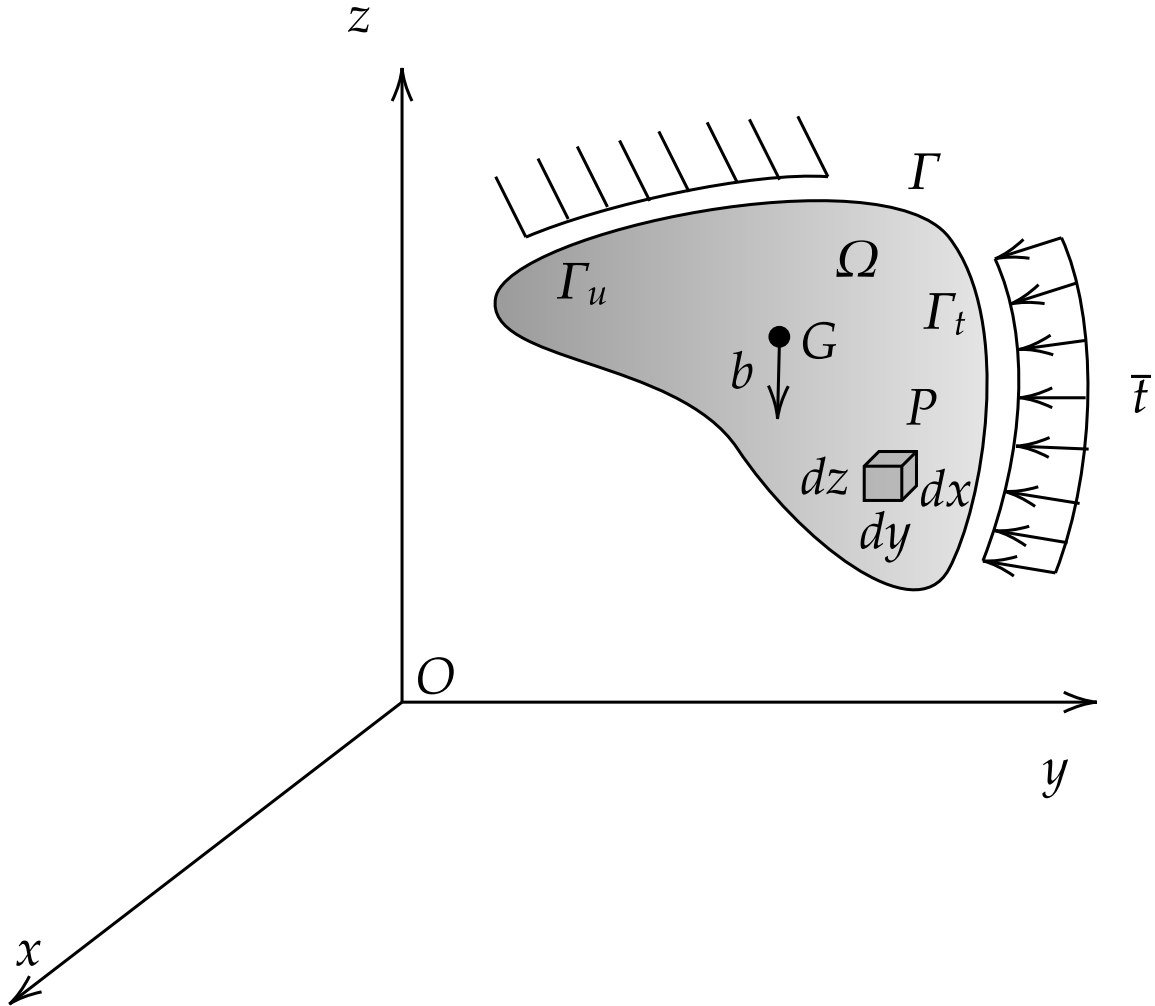


**Figure 4.1:** Continuous solid subjected to different kind of forces

The energy terms of Eq.(4.9) can be defined,

$$T = \frac{1}{2} \int_{\Omega} \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} d\Omega \quad (4.12)$$

$$U = \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega \quad (4.13)$$

$$W_f = \int_{\Omega} \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \quad (4.14)$$

with  $\Omega$  defining the solid volume,  $\dot{\mathbf{u}}$  is the velocity,  $\rho$  is the solid mass density,  $\boldsymbol{\varepsilon}$  being the strain vector,  $\boldsymbol{\sigma}$  the stress,  $\mathbf{u}$  is the displacement and  $\mathbf{b}$  the body forces.

It is possible to rewrite the Lagrangian functional  $L$  substituting Eqs.(4.10),(4.13) and (4.14) in Eq.(4.9),

$$L = \frac{1}{2} \int_{\Omega} \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} d\Omega - \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \quad (4.15)$$

Applying the Hamilton's principle, it is possible to establish,

$$\delta \int_{t_1}^{t_2} \left[ \frac{1}{2} \int_{\Omega} \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} d\Omega - \frac{1}{2} \int_{\Omega} \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \right] dt = 0 \quad (4.16)$$

Considering that the work presented concerns only static problems, the kinetic energy term can be discarded. Therefore, simplifying the integral and moving the variation operator  $\delta$  inside each integral, it is obtained,

$$\int_{t_1}^{t_2} \left[ -\frac{1}{2} \int_{\Omega} \delta(\boldsymbol{\varepsilon}^T \boldsymbol{\sigma}) d\Omega + \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \right] dt = 0 \quad (4.17)$$

The first term integrand from Eq.(4.17) can be rewritten as,

$$\delta(\boldsymbol{\varepsilon}^T \boldsymbol{\sigma}) = \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} + \boldsymbol{\varepsilon}^T \delta \boldsymbol{\sigma} \quad (4.18)$$

Since the terms from Eq.(4.18) are scalar, they can be transposed,

$$\boldsymbol{\varepsilon}^T \delta \boldsymbol{\sigma} = (\boldsymbol{\varepsilon}^T \delta \boldsymbol{\sigma})^T = \delta \boldsymbol{\sigma}^T \boldsymbol{\varepsilon} \quad (4.19)$$

Considering that  $\mathbf{c}^T = \mathbf{c}$ , using Eq.(4.6),

$$\delta \boldsymbol{\sigma}^T \boldsymbol{\varepsilon} = \delta(\mathbf{c} \boldsymbol{\varepsilon})^T \boldsymbol{\varepsilon} = \delta \boldsymbol{\varepsilon}^T \mathbf{c}^T \boldsymbol{\varepsilon} = \delta \boldsymbol{\varepsilon}^T \mathbf{c} \boldsymbol{\varepsilon} = \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} \quad (4.20)$$

Introducing Eq.(4.20) into Eq.(4.18),

$$\delta(\boldsymbol{\varepsilon}^T \boldsymbol{\sigma}) = 2\delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} \quad (4.21)$$

Substituting the second term of Eq.(4.21) in full Eq.(4.17),

$$\int_{t_1}^{t_2} \left[ -\int_{\Omega} \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma \right] dt = 0 \quad (4.22)$$

In order for Eq.(4.22) to be satisfied for all possible  $\mathbf{u}$ , for any initial time  $t_1$  and final time  $t_2$ , the integrand must be null. This leads to the Galerkin weak form equation,

$$-\int_{\Omega} \delta \boldsymbol{\varepsilon}^T \boldsymbol{\sigma} d\Omega + \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega + \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma = 0 \quad (4.23)$$

Considering the relationship presented in Eq.(4.6) and the fact the strain field can be calculated as a multiplication of a differential operator  $\mathbf{L}$  by the displacement field  $\mathbf{u}$ ,

$$\boldsymbol{\varepsilon} = \mathbf{L} \cdot \mathbf{u} \quad (4.24)$$

Eq.(4.23) can be rewritten as,

$$\int_{\Omega} \delta(\mathbf{L}\mathbf{u})^T \mathbf{c}(\mathbf{L}\mathbf{u}) d\Omega - \int_{\Omega} \delta \mathbf{u}^T \mathbf{b} d\Omega - \int_{\Gamma_t} \delta \mathbf{u}^T \bar{\mathbf{t}} d\Gamma = 0 \quad (4.25)$$

### 4.3 Discrete System of Equations

The discrete system of equations for the RPIM and NNRPIM can be obtained using the Galerkin weak form, which is established in Eq.(4.25), combined with the interpolation functions presented in Chapter 3.

For a solid in equilibrium, the Principle of Virtual Work (PVW) dictates that the total virtual work done by the body's internal stresses and the applied external forces on a system in static equilibrium is zero for a set of infinitesimal virtual displacements from equilibrium. The Galerkin weak form, combined with this principle allows the establishment of the discrete system of equations in meshless methods.

Considering Eq.(3.1),

$$u(\mathbf{x}_I) = \sum_{j=1}^n \varphi_j(\mathbf{x}_I) u(\mathbf{x}_j) \quad (4.26)$$

we can establish that, by the PVW, the virtual displacements of the nodes within the "Influence-Cells" (NNRPIM) or "Influence-Domains" (RPIM) of an interest point  $\mathbf{x}_I$ , will cause a virtual displacement of the integration point, which can be interpolated or approximated considering,

$$\delta u(\mathbf{x}_I) = \sum_{i=1}^n \varphi_i(\mathbf{x}_I) \delta u_i \quad (4.27)$$

Combining Eq.(3.1) and Eq.(4.25) with Eq.(4.27), we can establish,

$$\begin{aligned} - \int_{\Omega} \left( \sum_{i=1}^n \varphi_i(\mathbf{x}_I) \delta u_i \right)^T \mathbf{L}^T \mathbf{c} \mathbf{L} \left( \sum_{j=1}^n \varphi_j(\mathbf{x}_j) u_j \right) d\Omega + \int_{\Omega} \left( \sum_{i=1}^n \varphi_i(\mathbf{x}_I) \delta u_i \right)^T \mathbf{b} d\Omega \\ + \int_{\Gamma_t} \left( \sum_{i=1}^n \varphi_i(\mathbf{x}_I) \delta u_i \right)^T \bar{\mathbf{t}} d\Gamma = 0 \end{aligned} \quad (4.28)$$

The equation explicit above can be written in matrix form by eliminating the summations. For the 3D case,

$$\boldsymbol{\varphi}(\mathbf{x}) = \begin{bmatrix} \varphi_1(x) & 0 & 0 & \varphi_2(x) & 0 & 0 & \dots & \varphi_n(x) & 0 & 0 \\ 0 & \varphi_1(x) & 0 & 0 & \varphi_2(x) & 0 & \dots & 0 & \varphi_n(x) & 0 \\ 0 & 0 & \varphi_1(x) & 0 & 0 & \varphi_2(x) & \dots & 0 & 0 & \varphi_n(x) \end{bmatrix} \quad (4.29)$$

We can also define the general differential operator  $\mathbf{L}$  for a 3D problem,

$$\mathbf{L} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \quad (4.30)$$

The deformability matrix  $\mathbf{B}$ , which can be defined as,

$$\mathbf{B}^T = \begin{bmatrix} \frac{\partial \varphi_1}{\partial x} & 0 & 0 & \frac{\partial \varphi_1}{\partial y} & 0 & \frac{\partial \varphi_1}{\partial z} \\ 0 & \frac{\partial \varphi_1}{\partial y} & 0 & \frac{\partial \varphi_1}{\partial x} & \frac{\partial \varphi_1}{\partial z} & 0 \\ 0 & 0 & \frac{\partial \varphi_1}{\partial z} & 0 & \frac{\partial \varphi_1}{\partial y} & \frac{\partial \varphi_1}{\partial x} \end{bmatrix}^T \quad (4.31)$$

can be introduced in Eq.(4.28), obtaining,

$$\begin{aligned} - \sum_{i=1}^n \sum_{j=1}^n \delta u_i^T \int_{\Omega} \mathbf{B}^T \mathbf{c} \mathbf{B} d\Omega \cdot u + \sum_{i=1}^n \delta u_i^T \int_{\Omega} \varphi_i^T(\mathbf{x}_i) \mathbf{b} d\Omega \\ + \sum_{i=1}^n \delta u_i^T \int_{\Gamma} \varphi_i^T(\mathbf{x}_i) \bar{\mathbf{t}} d\Gamma = 0 \end{aligned} \quad (4.32)$$

which can be rewritten as,

$$-\delta \mathbf{U}^T [\mathbf{K} \mathbf{U} - \mathbf{F}] = 0 \quad (4.33)$$

with  $\mathbf{U} = \mathbf{u}$  and  $\mathbf{F} = \mathbf{f}_b + \mathbf{f}_t$ .

Solving the previous equation, the linear system of equations is obtained,

$$\mathbf{K} \mathbf{U} = \mathbf{F} \quad (4.34)$$

## 4.4 Boundary Conditions

The interpolation or approximation functions used in the RPIM and NNRPIM posses the Kronecker delta property,

$$\varphi_i(\mathbf{x}_i) = \delta_{ij} \quad (4.35)$$

where,

$$\begin{aligned}\delta_{ij} &= 1 & \text{if } i &= j \\ \delta_{ij} &= 0 & \text{if } i &\neq j\end{aligned}\tag{4.36}$$

Therefore, boundary conditions can be applied directly to the stiffness matrix  $\mathbf{K}$ , present in Eq.(4.34). These boundary conditions refer to:

- Essential boundary conditions (related with displacements)

# Chapter 5

## Continuum Mechanics of Plasticity

In this chapter, the fundamental theory of plasticity is presented. The yield criterion, the flow rule and the hardening rule are introduced. In addition, the theory will also be presented in matrix form, as to accompany the application of said calculations in computational mechanics.

### 5.1 Overview on Plasticity

The object of the mathematical theory of plasticity is to provide a theoretical description of the relationship between stress and strain for a material which exhibits an elastoplastic response [55].

Plastic behaviour is characterized by an irreversible deformation to a body under stress, which can only be achieved after a certain level of stress is applied to the body. Different types of materials exhibit different behaviours when stress is applied to them. In Figure 5.1, it is possible to observe different types of elastoplastic material behaviours.

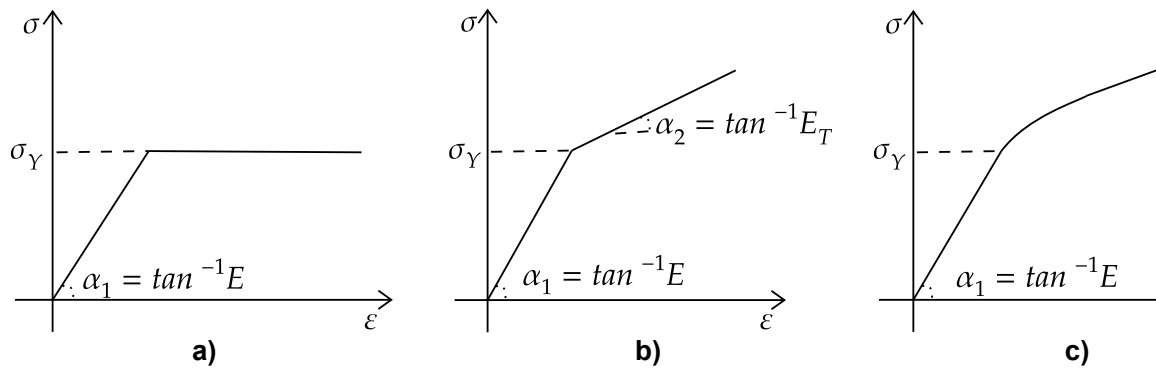


**Figure 5.1:** Elastoplastic material behaviour. **a)** Elastic-perfectly plastic model. **b)** Bilinear elastoplastic model. **c)** Non-linear elastoplastic model.

When a certain level of stress is passed, the elastic limit,  $\sigma_Y$ , the material will yield and any deformation beyond that point will be permanent. In Figure 5.1(a), it is possible to observe that the elastic part is linear, characterized by the Young

Modulus  $E$  and by  $\alpha_1 = \tan^{-1}E$ . When the yield stress,  $\sigma_Y$  is reached, the stress level is maintained while the deformation, of plastic nature only, increases.

In Figure 5.1(c)), two slopes are observed. The first slope, characterized by  $\alpha_1$  is the elastic part, while the second slope, when the yield stress is reached, characterized by  $\alpha_2 = \tan^{-1}E_T$  is the plastic part, with  $E_T$  being the tangent modulus. In case of Figure 5.1(c)), the stress relation after yield is no longer linear.

For a theoretical formulation of plasticity theory, three fundamental concepts have to be approached:

- A yield criterion, indicating the stress level at which plastic flow commences must be formulated. The existence of a initial yield surface which defines the elastic limit in a multi-axial stress state;
- A hardening law, which describes the evolution of the subsequent yield surfaces that are developed;
- A flow rule, which relates the plastic deformation to the stresses currently in the material and the stress increments outside the yield surface.

Hook's law can be defined as,

$$\sigma = E\varepsilon \quad (5.1)$$

As soon as the yield stress  $\sigma_Y$  is reached, plastic behaviour occurs, as it can be seen in Figure 5.2.

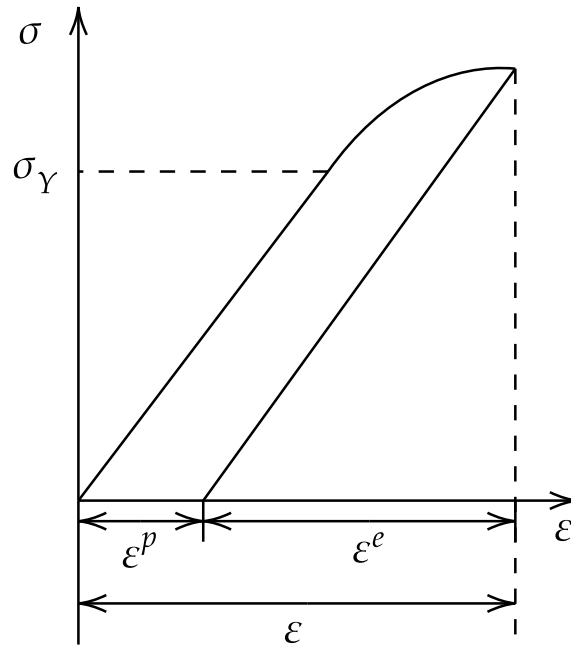


**Figure 5.2:** Uniaxial stress-strain diagram for arbitrary hardening

The characteristic feature of plastic material behaviour is that a remaining plastic strain  $\varepsilon^p$  occurs after complete unloading, Figure 5.2. Only the elastic strains  $\varepsilon^e$  returns to zero at complete unloading [56].

The total strain  $\boldsymbol{\varepsilon}$  can be decomposed in to two components,

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^p + \boldsymbol{\varepsilon}^e \quad (5.2)$$

where  $\boldsymbol{\varepsilon}^p$  is the plastic component of the total strain and  $\boldsymbol{\varepsilon}^e$  the elastic component.

Before the plastic flow occurs, the material has an elastic behaviour. This means that the relationship between the stress and strain field follows the generalized Hook's law,

$$\boldsymbol{\sigma} = \mathbf{c}\boldsymbol{\varepsilon}^e \quad (5.3)$$

where  $\mathbf{c}$  is the constitutive matrix that contains the mechanical properties of the material, already presented in Chapter 4.

Plastic flow begins when the yield stress  $\sigma_Y$ , established by the yield criterion, is reached. At that moment, Hook's linear law is no longer valid, and the stress field is now developed following a flow rule.

## 5.2 Yield Criterion

In order to characterize the yield for a certain material, a yield surface, the material elastic limit and the surface in which the stresses occur after the elastic limit is attained, is used, as Figure 5.3 represents.

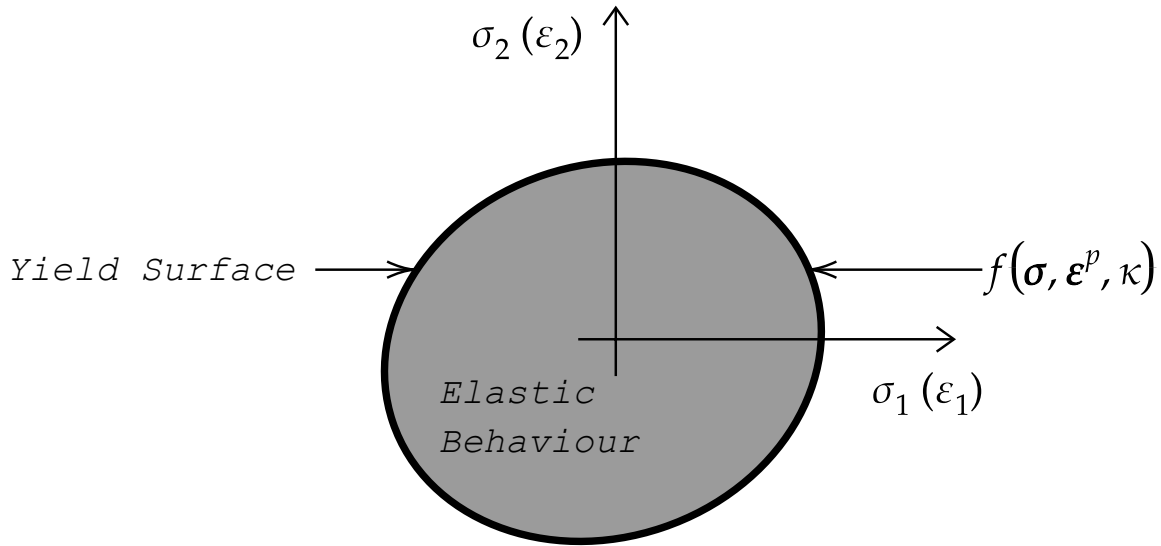


**Figure 5.3:** Yield surface

For a certain stress state, the limit for elastic behaviour is defined, mathematically, by a yield criterion. The yield criterion is used to assess if the materials suffers only elastic or also plastic strains. The yield criterion is a function of the stress,  $\boldsymbol{\sigma}$  and the load history, being defined as,

$$F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa) = f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa) - \sigma_Y(\kappa) = 0 \quad (5.4)$$

where  $f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa)$  is a yield function dependent on the stress state  $\boldsymbol{\sigma}$ , the plastic strain  $\boldsymbol{\varepsilon}^p$ , a hardening parameter  $\kappa$  and the yield stress of the material  $\sigma_Y(\kappa)$ , which can also be independent of the hardening parameter  $\kappa$ .

For an isotropic material, the yield stress  $\sigma_Y$  is independent of the hardening parameter  $\kappa$ , and Eq.(5.4) can be simplified,

$$F(\boldsymbol{\sigma}) = f(\boldsymbol{\sigma}) - \sigma_Y = 0 \quad (5.5)$$

In terms of a yield criterion, the most commonly used are the von Mises and Tresca criterion. In this work, only the von Mises yield criterion was considered, due to the higher accuracy when compared with the Tresca criterion (for metals), which has some edges in the surface, which can lead to singularities, Figure 5.4.

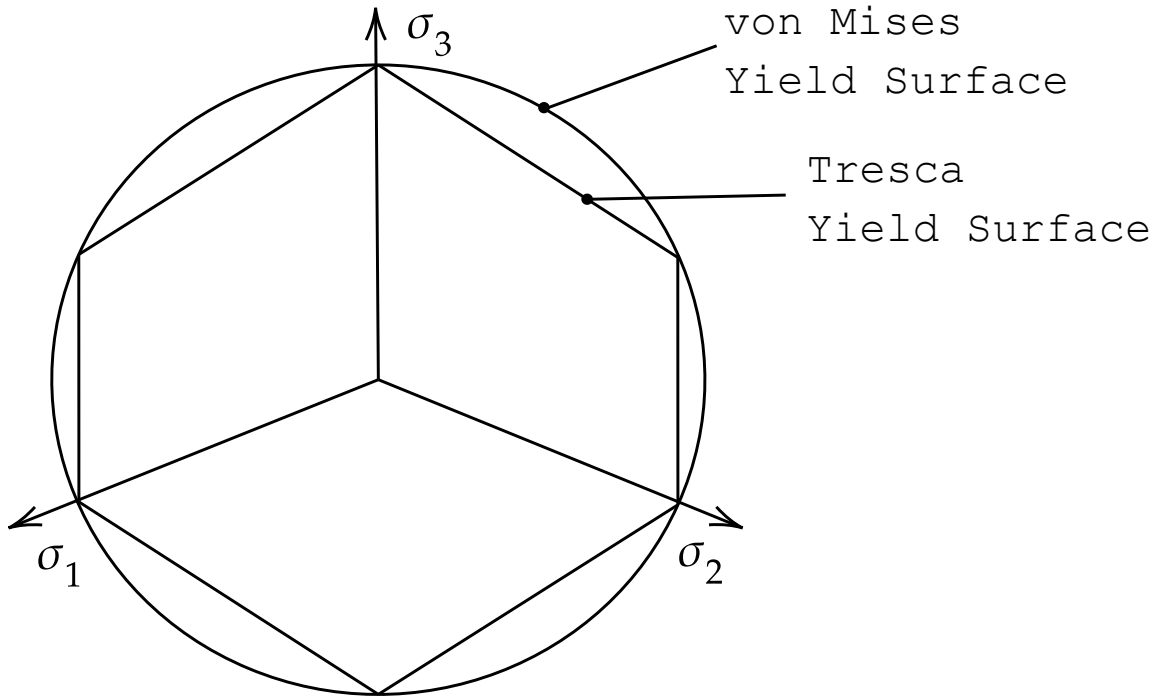


**Figure 5.4:** Yield surface for the von Mises and Tresca yield criterion

Therefore, using the von Mises yield criterion, yield occurs when second deviatoric stress invariant,  $J_2$ , reaches a critical value,

$$J_2 = \left( \frac{\sigma_Y}{\sqrt{3}} \right)^2 \quad (5.6)$$

The yield criterion established in Eq.(5.6) can be written in terms of the effective stress  $\bar{\sigma}$

$$\bar{\sigma}^2 = \sigma_Y^2 = 3J_2 \quad (5.7)$$

Defining  $J_2$  in terms of the stress tensor, it can be established,

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_{xx} - \sigma_{yy})^2 + (\sigma_{yy} - \sigma_{zz})^2 + (\sigma_{zz} - \sigma_{xx})^2 + 6(\sigma_{yz}^2 + \sigma_{zx}^2 + \sigma_{xy}^2)]} \quad (5.8)$$

With the above considerations, Eq.(5.5) can be written in the following form,

$$F(\boldsymbol{\sigma}) = \sqrt{\frac{1}{2}[(\sigma_{xx} - \sigma_{yy})^2 + (\sigma_{yy} - \sigma_{zz})^2 + (\sigma_{zz} - \sigma_{xx})^2 + 6(\sigma_{yz}^2 + \sigma_{zx}^2 + \sigma_{xy}^2)]} - \sigma_Y(\kappa) = 0 \quad (5.9)$$

While the Tresca yield criterion predicts a yield stress  $\sigma_Y$ , the von Mises yield criterion predicts a yield stress that is  $\frac{2}{\sqrt{3}}$  times higher.

It is important to note that, besides these two criteria, there is a large variety of criteria to choose from. For example, the Mohr-Columb and Drucker-Prager criteria are commonly used in solid mechanics, with the latter used to analyze concrete structures.

## 5.3 Hardening Rule

After initial yielding, the stress level at which further plastic deformation occurs may be dependent on the current degree of plastic straining. Such a phenomenon is termed work hardening or strain hardening [41].

The new conditions for the plastic flow after initial yielding are dependent on the alterations that the yield surface suffers. This phenomenon is dependent on the hardening parameter  $\kappa$ .

Considering the large variety of hardening rules existent, the most commonly used models are the Isotropic, Kinematic and Independent Hardening Rules, presented in Figure 5.5.

As a consequence of hardening, the elastic domain can change either its size, position, or both. The Elastic-Perfectly Plastic Model, represented in Figure 5.1(a), is an exception since the elastic domain remains constant. Therefore, the stress state moves along the surface while the yield surface remains unchanged.

Therefore, it is possible to establish the yield criterion as,

$$F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}, \kappa) = f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}) - \kappa^2(\boldsymbol{\varepsilon}^p) = 0 \quad (5.10)$$

where  $\kappa^2$  represents the size of the yield surface,  $F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}, \kappa)$  represents the shape of the yield surface,  $\boldsymbol{\varepsilon}^p$  is the plastic strain and  $\boldsymbol{\alpha}$  defines the center of the yield surface.

In this work, only the Isotropic Hardening Rule was used.

### 5.3.1 Isotropic Hardening

Of all hardening rules, the Isotropic Hardening Rule is the most simple, either in terms of formulation or implementation.

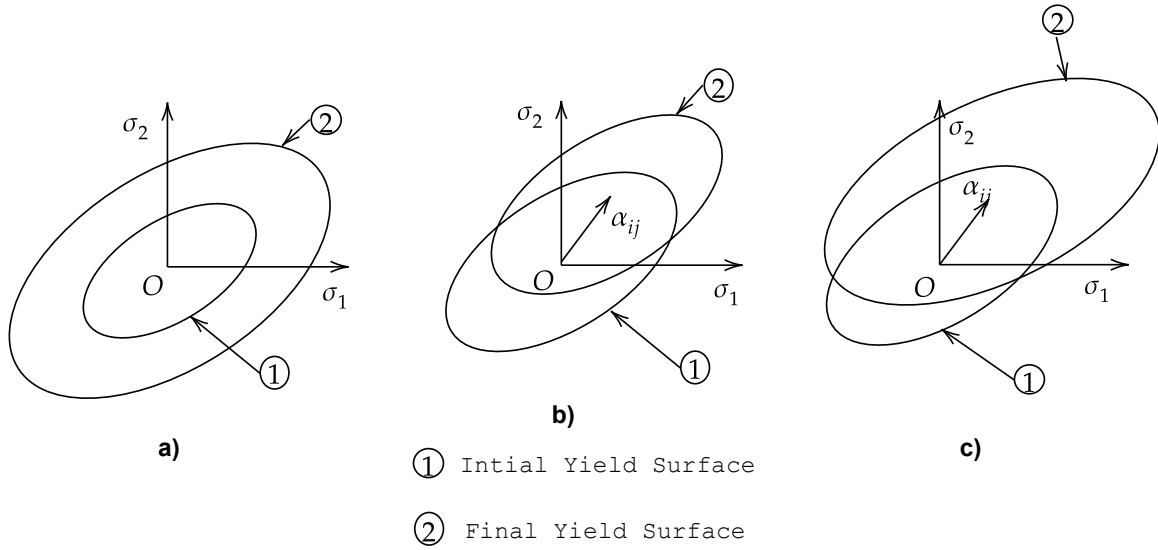


**Figure 5.5:** Hardening rules. **a)** Isotropic Hardening. **b)** Kinematic Hardening. **c)** Independent Hardening.

For an isotropic model, the evolution of the yield surface consists in an expansion of the initial surface, with the same origin. Therefore, the yield surface, during plastic flow, only changes its size, expanding uniformly.

For this rule, the yield criterion can be defined as,

$$f(\boldsymbol{\sigma}) = \kappa^2(\boldsymbol{\varepsilon}^p) \quad (5.11)$$

The variation in the yield stress,  $\sigma_Y$ , as a function of the accumulated plastic strain can be defined in two possible ways: using either the total plastic work  $W_p$  or the total plastic strain  $\bar{\boldsymbol{\varepsilon}}^p$ .

For the first case, the hardening parameter  $\kappa$  is considered equal to the total plastic work,

$$\kappa = W_p \quad (5.12)$$

where,

$$W_p = \int \boldsymbol{\sigma} d\boldsymbol{\varepsilon}^p \quad (5.13)$$

in which  $d\boldsymbol{\varepsilon}^p$  is the plastic component of strain occurring during a strain increment. In alternative,  $\kappa$  can be defined as,

$$\kappa = \bar{\boldsymbol{\varepsilon}}^p \quad (5.14)$$

with the effective, generalized or equivalent plastic strain being established as,

$$\bar{\boldsymbol{\varepsilon}}^p = \int d\bar{\boldsymbol{\varepsilon}}^p = \int \left[ \frac{2}{3} d\boldsymbol{\varepsilon}_{ij}^p d\boldsymbol{\varepsilon}_{ij}^p \right]^{\frac{1}{2}} \quad (5.15)$$

In cases when the material is assumed to have yielding that is independent of any hydrostatic stress,

$$d\bar{\epsilon}^p = 0 \quad (5.16)$$

from which we can establish that,

$$d\mathbf{e}_{ij}^p = d\bar{\epsilon}^p \quad (5.17)$$

where  $d\mathbf{e}^p$  is the plastic component of the variations of the deviation strains,  $d\mathbf{e}$ , that can be defined as,

$$d\mathbf{e}_{ij} = d\bar{\epsilon}_{ij} - \frac{1}{3}(\delta_{ij}d\bar{\epsilon}_{ij})\delta_{ij} \quad (5.18)$$

We can now write Eq.(5.15) as,

$$\bar{\epsilon}^p = \int d\bar{\epsilon}^p = \int \left[ \frac{2}{3} d\mathbf{e}_{ij}^p d\mathbf{e}_{ij}^p \right]^{\frac{1}{2}} \quad (5.19)$$

as a function of the plastic component of the variations of the deviation strains.

The Isotropic Hardening Rule does not consider the Bauschinger effect [57], while the Kinematic and Independent Hardening Rules do. The Bauschinger effect will be detailed in the same section as the Kinematic Hardening Rule as this rule does, in fact, consider this effect.

### 5.3.2 Kinematic Hardening

For the case of Kinematic Hardening, the yield surface suffers a translation from its original position only. This means that the size of the yield surface remains constant due to the unchanged elastic part during hardening, but the center of the yield surface changes position, with a new center defined by  $\alpha_{ij}$ .

For this rule, the yield criterion can be defined as,

$$F(\boldsymbol{\sigma}, \boldsymbol{\epsilon}^p, \boldsymbol{\alpha}) = f(\boldsymbol{\sigma} - \boldsymbol{\alpha}) - \kappa^2 = 0 \quad (5.20)$$

As referred earlier, this rule considers the Bauschinger effect, which will be explained immediately.

#### 5.3.2.1 Bauschinger Effect

Let us consider pure monotonic loading, i.e. pure tensile or pure compression. For the cases of the Isotropic and Kinematic Hardening Rules, it is not possible to distinguish between one or the other from the stress-strain diagram [56].

Consider the uniaxial test with plastic deformation and stress reversal as shown in Figure 5.6.

In the beginning, there is no initial stress or strain in the origin of the diagram (point 'O'), and a tensile load is constantly increased. As long as the stress is below the yield stress,  $\sigma_Y$ , the path  $\overline{O1}$  is in the pure elastic range and Hooke's law is valid.

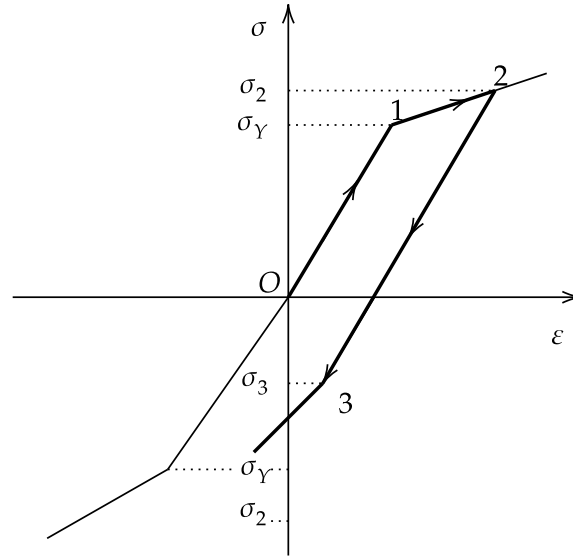


**Figure 5.6:** Idealized stress-strain curve with Bauschinger effect for uniaxial strain hardening.

Reaching the yield stress, point 1, plastic deformation occurs and the slope of the diagram changes. While the load increases, the plastic deformation and strain increases.

Reaching point 2, the load is reversed. The unloading is completely elastic and compressive stress develops as the load path crosses the strain axis. The interesting part of the diagram to study is the subsequent plastic deformations in the compressive regime. This deformation occurs, for the kinematic hardening, at a stress level  $\sigma_3$ , lower than the yield stress  $\sigma_Y$  or the subsequent stress  $\sigma_2$ .

This phenomenon is known as the Bauschinger effect [57], and requires plastic pre-straining with subsequent load reversal. It is a kind of directional anisotropy created by the plastic deformation, very common in material that are subjected to cyclic loading.

### 5.3.3 Independent Hardening

For the Independent Hardening Rule, Figure 5.5(c)), the yield surface is the combination of both rules explained earlier, as the yield surface not only suffers an expansion but also translation and/or rotation. The material is assumed to be hardened independently in tension and in compression.

For this rule, the yield criterion is more general and can be written as,

$$F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \boldsymbol{\alpha}) = f(\boldsymbol{\sigma} - \boldsymbol{\alpha}) - \kappa^2(\boldsymbol{\varepsilon}^p) = 0 \quad (5.21)$$

This rule also takes in account the Bauschinger effect.

## 5.4 Flow Rule

The flow rule serves as a mathematical description of the evolution of the infinitesimal increments of the plastic strain  $d\boldsymbol{\varepsilon}^p$  in the course of the load history of the body [56].

As the incremental load increases, and the yield function  $f$  reaches a value equal to the yield stress of the material,  $\sigma_Y$ , the frontier of elasticity is attained and plastic regime begins. As such, when, for a elastic-perfectly plastic material, the yield surface is reached, the material is in a state of plastic flow. In the case of this particular material, the yield stress,  $\sigma_Y$ , is not dependent on the degree of plastification and maintains its initial value,  $\sigma_Y^0(\kappa = 0)$ .

For a material without elastic-perfectly plastic properties, i.e. materials that present hardening, the behaviour of the material will be dependent of the variation of the yield function  $f$ , with respect to the stress state  $\boldsymbol{\sigma}$ , as it can be established,

$$df = \left( \frac{\partial f}{\partial \boldsymbol{\sigma}} \right)^T d\boldsymbol{\sigma} \quad (5.22)$$

where  $\frac{\partial f}{\partial \boldsymbol{\sigma}}$  is the  $f$  gradient, and therefore a orthogonal vector to the yield surface for a certain stress change  $d\boldsymbol{\sigma}$ , which moves tangentially to the surface, being orthogonal to the  $f$  gradient.

Applying the Prandtl-Reuss flow rule, which has been extensively employed in theoretical work, we reach,

$$d\boldsymbol{\varepsilon}^p = d\lambda \frac{\partial f}{\partial \boldsymbol{\sigma}} \quad (5.23)$$

where  $d\lambda$  is a proportional constant termed: plastic multiplier. The vector  $\frac{\partial f}{\partial \boldsymbol{\sigma}}$  is a vector directed normal to the yield surface at the stress point under consideration, as shown in Figure 5.7.

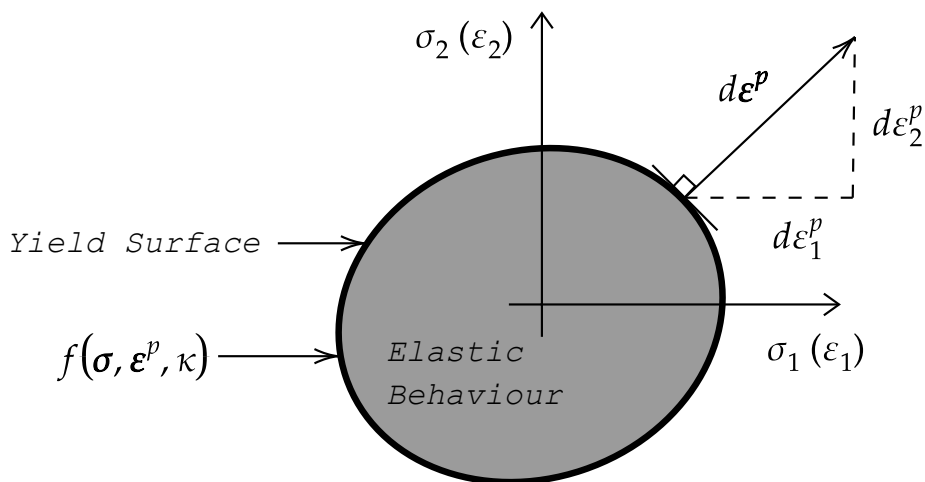


**Figure 5.7:** Representation of the flow rule (orthogonality principle).

The following conclusions, explicit in Table 5.1, are withdrawn from the analysis of Figure 5.7 [41, 56]:

|                |                                                                                                                                                                                                                   |
|----------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| $d\lambda < 0$ | Elastic unloading occurs (elastic behaviour) and the stress point returns inside the yield surface. Hook's law defined in Eq.(5.1) is still valid.                                                                |
| $d\lambda = 0$ | Stress point is on the yield surface. Neutral loading (plastic behaviour for a elastic-perfectly plastic material). If the material has hardening, this situation corresponds to the beginning of the plasticity. |
| $d\lambda > 0$ | Plastic loading. The stress point is on a constantly expanding yield surface (plastic behaviour for a strain hardening material).                                                                                 |

**Table 5.1:** Possible situations resulting from the plastic multiplier and stress point

## 5.5 Matrix Formulation

As this work has its basis on the numerical analysis of elastoplastic problems, a matrix formulation for the elastoplastic theory presented earlier is convenient since its in this form that all mathematical aspects of elastoplasticity are implemented in computational mechanics [41].

The von Mises yield criterion, first defined in Eq. 5.4, is,

$$F(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa) = f(\boldsymbol{\sigma}, \boldsymbol{\varepsilon}^p, \kappa) - \sigma_Y(\kappa) = 0 \quad (5.24)$$

By differentiating Eq. (5.24), it is possible to write,

$$dF = \left( \frac{\partial F}{\partial \boldsymbol{\sigma}} \right) d\boldsymbol{\sigma} + \frac{\partial F}{\partial \kappa} d\kappa = 0 \quad (5.25)$$

Eq. (5.25) can be written as,

$$\mathbf{a}^T d\boldsymbol{\sigma} - A d\lambda = 0 \quad (5.26)$$

where the vector  $\mathbf{a}$  is termed the flow vector,  $\frac{\partial f}{\partial \boldsymbol{\sigma}}$ ,  $d\lambda$  is a proportional constant termed plastic multiplier, and  $A$  can be written as,

$$A = -\frac{1}{d\lambda} \frac{\partial F}{\partial \kappa} d\kappa \quad (5.27)$$

The complete incremental relationship between stress and strain for elasto-plastic deformation,  $d\boldsymbol{\varepsilon}$ , can be presented as,

$$d\boldsymbol{\varepsilon} = \mathbf{c}^{-1} d\boldsymbol{\sigma} + d\lambda \frac{\partial F}{\partial \boldsymbol{\sigma}} \quad (5.28)$$

with  $\mathbf{c}$  being the constitutive matrix of material properties. With some mathematical manipulation, it is possible to obtain the plastic multiplier,

$$d\lambda = \frac{1}{[A + \mathbf{a}^T \mathbf{c} \cdot \mathbf{a}]} \mathbf{a}^T \mathbf{c} d\boldsymbol{\varepsilon} \quad (5.29)$$

Substituting Eq. (5.29) in Eq. (5.25) permits to obtain,

$$d\boldsymbol{\sigma} = \mathbf{c} d\boldsymbol{\varepsilon} - \frac{\mathbf{a}^T \mathbf{c} d\boldsymbol{\varepsilon}}{\mathbf{a}^T \mathbf{c} \cdot \mathbf{a} + A} \mathbf{c} \cdot \mathbf{a} = \left( \mathbf{c} - \frac{\mathbf{c} \cdot \mathbf{a} \cdot \mathbf{a}^T \mathbf{c}}{\mathbf{a}^T \mathbf{c} \cdot \mathbf{a} + A} \right) d\boldsymbol{\varepsilon} = \mathbf{c}^p d\boldsymbol{\varepsilon} \quad (5.30)$$

with  $\mathbf{c}^p$  being the elastoplastic tangential modular matrix.

The last mathematical parameter left to determine is  $A$ , which is dependent on the considered hardening rule.

For the isotropic hardening rule, in Eq. (5.24), the yield stress becomes a function of the plastic work,

$$\sigma_Y(\kappa = W_p) \quad (5.31)$$

where  $W_p$  is given by,

$$\kappa = W_p = \int \sigma_Y d\varepsilon_p = \int \boldsymbol{\sigma} d\boldsymbol{\varepsilon}_p = \int d\lambda \boldsymbol{\sigma}^T \mathbf{a} \quad (5.32)$$

Therefore, the plastic work rate is given by,

$$d\kappa = dW_p = \sigma_Y d\varepsilon_p = \boldsymbol{\sigma}^T d\boldsymbol{\varepsilon}_p = d\lambda \boldsymbol{\sigma}^T \mathbf{a} \quad (5.33)$$

Substituting Eq. (5.34) in Eq. (5.27), we obtain the hardening constant  $A$ ,

$$A = \boldsymbol{\sigma}^T \mathbf{a} \frac{\partial \sigma_Y}{\partial \kappa} \quad (5.34)$$

Defining the relationship between the yield stress,  $\sigma_Y$ , and the hardening parameter,  $\kappa$ , lead to the total definition of the hardening constant  $A$ .

Imposing the relationship to the plastic strain  $\boldsymbol{\varepsilon}_p$ , we obtain,

$$\frac{\partial \sigma_Y}{\partial \kappa} = \frac{\partial \sigma_Y}{\partial \varepsilon_p} \frac{\varepsilon_p}{\partial \kappa} \quad (5.35)$$

Substituting the plastic work rate,  $d\kappa$ , in Eq. (5.35), we can establish,

$$\frac{\partial \sigma_Y}{\partial \kappa} = \frac{\partial \sigma_Y}{\partial \varepsilon_p} \frac{1}{\sigma_Y} \quad (5.36)$$

The relation,  $\frac{\sigma_Y}{\partial \varepsilon_p}$ , can be achieved from the stress/plastic strain relationship. Therefore,

$$\frac{\partial \sigma_Y}{\partial \varepsilon_p} = \frac{d\sigma}{d\varepsilon - d\varepsilon_e} = \frac{1}{\frac{d\varepsilon}{d\sigma} - \frac{d\varepsilon_e}{d\sigma}} = \frac{1}{\frac{1}{E_T} - \frac{1}{E}} = \frac{E_T}{1 - \frac{E_T}{E}} = H' \quad (5.37)$$

Replacing Eq. (5.37) in Eq. (5.36), we obtain,

$$\frac{\partial \sigma_Y}{\partial \kappa} = \frac{H'}{\sigma_Y} \quad (5.38)$$

We can now define the hardening constant  $A$  as,

$$A = \frac{H' \boldsymbol{\sigma}^T \mathbf{a}}{\sigma_Y} \quad (5.39)$$

Considering the yield function as a homogeneous function of first order, the Euler's theorem shows that,

$$\frac{\partial f^T}{\partial \boldsymbol{\sigma}} \boldsymbol{\sigma} = \mathbf{a}^T \boldsymbol{\sigma} = \sigma_Y \quad (5.40)$$

Therefore,  $A = H'$ .

# Chapter 6

## Non-Linear Solution Algorithms

In this chapter, the non-linear solution algorithms that are used in this thesis are explained.

The elastoplastic analysis of structures involves a set of non-linear equations that need to be solved in order to obtain a solution for a certain problem. However, being the set of these equations of non-linear nature, a direct solution methodology is not applicable. Therefore, it is mandatory the use of incremental methods to obtain a solution.

For this purpose, the Newton-Rapson method is the basis of the three non-linear solution algorithms used. In order to better understand how these algorithms perform, the Newton-Rapson will be detailed. Thereafter, the three non-linear solution algorithms used, KT-0, KT-1 and KT-ALL will be explained.

The stress-returning algorithm will also be approached, as it computes how the stress evolves with plastic deformation and what consequence it has for the yield surface.

### 6.1 Incremental-Iterative Methods

The use of incremental methods [58], has the intent of achieving an approximate solution to a certain problem with the application of successive load increments, as it is shown in Figure 6.1(a)).

The displacement,  $u$ , for each load stage  $\delta f$  is given by,

$$\delta u = \left( \frac{df}{du} \right)^{-1} \delta f = K_T^{-1} \delta f \quad (6.1)$$

The most significant issue with the incremental method is the accumulation of errors as the load increments are applied. As it can be seen in Figure 6.1(a)), considering point 2 of the diagram, the solution given by the method drifts from the equilibrium curve of the solution. As the load increases, these errors will accumulate. As the non-linearity of the solution increases,  $\frac{df}{du} \rightarrow 0$ , the amount of cumulative errors will increase as well.

As a way to overcome this fact, an iterative method is required. One of the most commonly used is the Newton-Raphson incremental scheme. In short, the Newton-

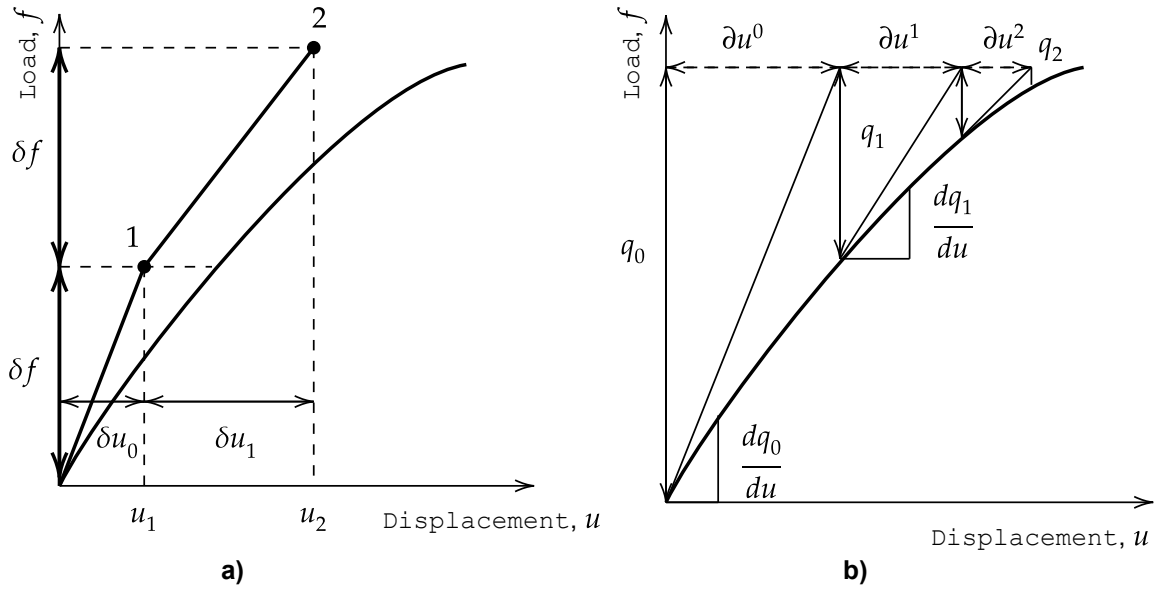


**Figure 6.1:** Incremental methods. **a)** Incremental (Euler) scheme. **b)** Newton-Raphson method

Raphson method only provides the displacement,  $u$ , of a pre-defined load,  $f$ , as it is shown in Figure 6.1(b)).

The iterative changes,  $j$ , are defined,

$$\partial u^j = \left( \frac{dq^j}{du} \right)^{-1} q^j(u^j) \quad (6.2)$$

and the displacements are given by,

$$u^j = u^{j-1} + \partial u^{j-1} \quad (6.3)$$

### 6.1.1 Combined Incremental And Newton-Raphson Method (KT-ALL)

The incremental and iterative methods can be combined, Figure 6.2, allowing to obtain the problem solution and a reduction of the lack of equilibrium presented in the incremental process.

Analyzing Figure 6.2, the incremental methods acts as a predictor agent. It provides a starting solution,  $u_i^0$ , in order for the iterative process to have a starting point to begin the convergence process.  $u_i^j$  represents the displacement of iteration  $j$  for a load increment  $i$ .

It is evident that the use of a predictor as a starting solution to the problem allows the improvement of the convergence process regarding the iterative procedure, allowing less iterations to be used.

In the KT-ALL non-linear solution algorithm, the stiffness matrix of the problem is calculated at the beginning of each increment, and throughout all iterations of all

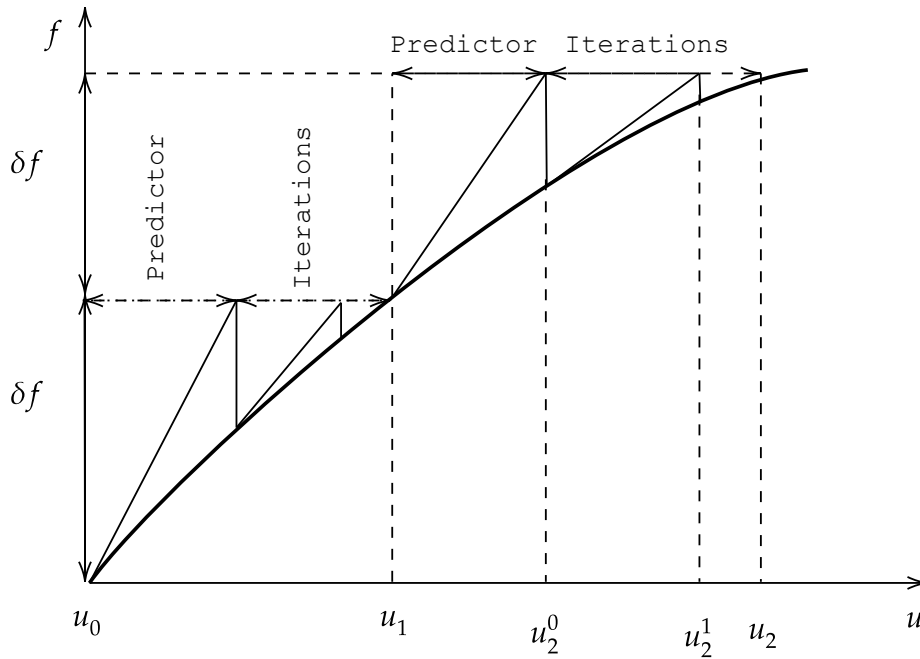


**Figure 6.2:** Combined incremental and Newton-Raphson method (KT-ALL)

increments. Therefore, for every single increment, the algorithm calculates the stiffness matrix of the problem, considering the initial conditions and previous increments. This fact reduces the amount of iterations needed for the convergence of the solution, since the stiffness matrix is updated regularly. However, this fact may lead to a slow and costly computational process.

### 6.1.2 Combination Of Incremental Predictors With Modified Newton-Raphson Iterations (KT-1)

In order to reduce the computational cost of calculating the stiffness matrix at the beginning of each iteration, variants of the KT-ALL method were studied.

One of these variants is the combination of the incremental predictors with the modified Newton-Raphson iterations, or the KT-1 algorithm, represented in Figure 6.3

In the KT-1 algorithm, the stiffness matrix of the problem is calculated at the beginning of the increment, and that stiffness matrix is used for all iterations regarding the corresponding increment. Therefore, in contrast with the KT-ALL algorithm, throughout the number of iterations needed for each increment, every single one of them uses the same stiffness matrix.

Due to the fact that the stiffness matrix is not required to be calculated at each iteration but at the beginning of each increment, such solution scheme reduces the computational cost. On the other hand, without the constant update of the stiffness matrix, the KT-1 algorithm needs more iterations for each increment to achieve the solution, as it can be seen comparing Figure 6.2 and 6.3.

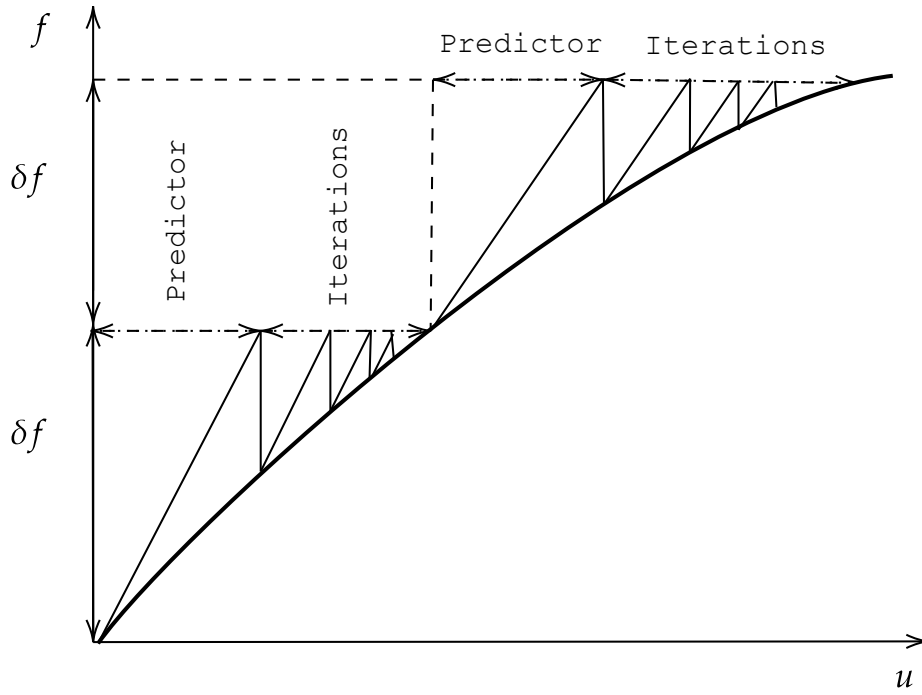


**Figure 6.3:** Combination of incremental predictors with modified Newton-Raphson iterations (KT-1)

### 6.1.3 Initial Stress Method Combined With An Incremental Solution (KT-0)

The KT-1 algorithm can be simplified. If, rather than calculating the stiffness matrix at the beginning of each increment, the stiffness matrix is calculated only once, at the beginning of the first increment, the same stiffness matrix is considered for all increments and iterations.

Represented in Figure 6.4, the KT-0 algorithm will require far more iterations for the convergence of the solution, when compared with the KT-ALL and KT-1 algorithms, precisely because there is absolutely no update to the stiffness matrix.

Comparing the three algorithms, the KT-ALL will take more time calculating the stiffness matrix and its inverse at every increment and iteration..

The KT-1 algorithm is at a middle point of the three, updating the stiffness matrix but only at the beginning of each increment.

At last, the KT-0 saves time by calculating only one stiffness matrix, at the beginning of the very first increment, for the very first iteration. However, the number of iterations needed for the solution to converge due to this fact is much higher than the other algorithms considered.

In this work, the behaviour of all three algorithms will be studied, with the KT-ALL being used for the rest of the numerical analysis.

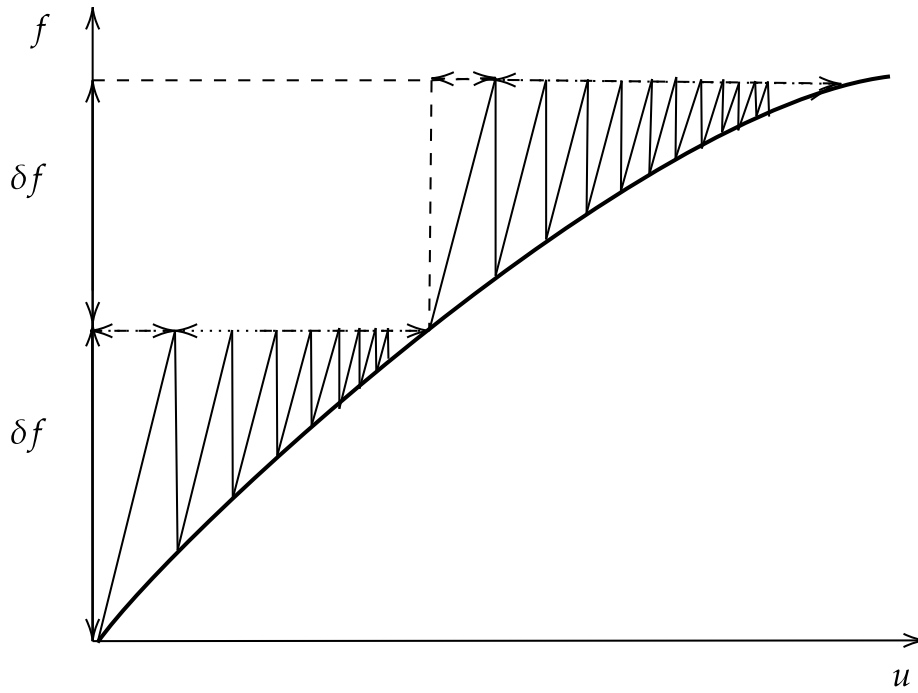


**Figure 6.4:** Initial stress method combined with an incremental solution (KT-0)

## 6.2 Stress-Returning Algorithm

As it has been presented earlier in this chapter, the main purpose of using incremental-iterative methods is to avoid the accumulation of errors during the solving of the non-linear equations that govern elastoplastic problems. Therefore, the non-linear algorithms that were explained earlier must include a stress-returning algorithm, necessary to ensure that the stresses that are developed remain inside the yield surface or, at least, very close to it.

For this effect, some procedures can be applied when the stresses, eventually, are located outside the yield surface [58]. For this work, the backward-Euler scheme was adopted, Figure 6.5.

This method acts at the Gauss point level and has no need to know the intersection point of the incremental load with the yield surface in order to solve the non-linear equations.

After every incremental load and for every single Gauss point, the stress state is analyzed and accessed if it is inside or outside the yield surface. Therefore, the yield criterion,

$$f(\boldsymbol{\sigma}) \leq \sigma_Y^* \quad (6.4)$$

where  $f(\boldsymbol{\sigma})$  is the yield surface,  $\sigma_Y^*$  is the updated yield stress, given by,

$$\sigma_Y^* = \sigma + [E_T \cdot \overline{\varepsilon}_{i-1}^p] \quad (6.5)$$

$\overline{\varepsilon}_{i-1}^p$  is the accumulated effective plastic strain from the previous increment,  $(i-1)$ .

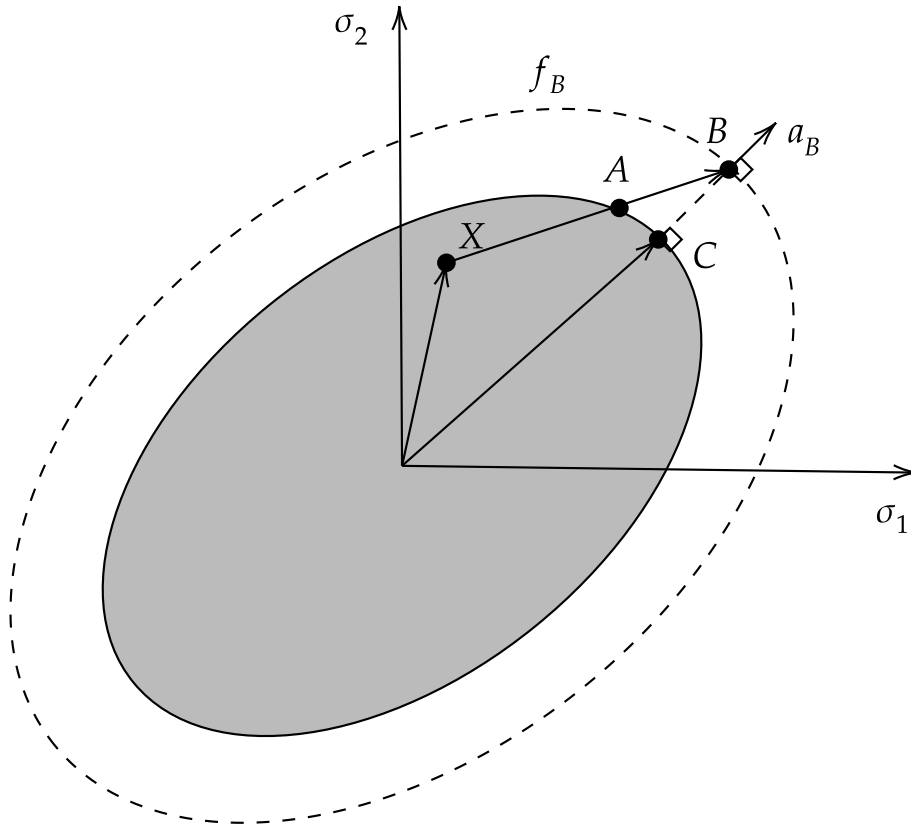


**Figure 6.5:** Backward-Euler scheme

Therefore, if the Gauss point in question has not entered plastic regime yet, the second term of Eq.(6.5) will be equal to 0, and as such, the updated yield stress equals the reference yield stress.

The stress-returning algorithm is called every time Eq.(6.4) is not verified. To analyze this case, consider Figure 6.5. The stress point  $B$  is outside the yield surface and needs to be pushed back to point  $C$ . The algorithm starts with a predictor, that considers a yield function  $f_B$  on which point  $B$  is located, avoiding the calculation of the intersection point  $A$ .

The flow vector,  $\mathbf{a}_B$ , is calculated from point  $B$ . The yield function,  $f_B$  is defined by,

$$f_B = \overline{\sigma}_B - \sigma_Y^* \quad (6.6)$$

In sequence, the plastic multiplier,  $d\lambda$  is calculated, given by,

$$d\lambda = \frac{f_B}{\mathbf{a}_B^T \mathbf{c} \cdot \mathbf{a}_B + H'} \quad (6.7)$$

With  $d\lambda$  calculated, it is possible to estimate the stress in point  $C$ ,

$$\boldsymbol{\sigma}_C = \boldsymbol{\sigma}_B - d\lambda \mathbf{c} \cdot \mathbf{a}_B \quad (6.8)$$

This value is only an estimate. The correct calculation to achieve the stress in point  $C$  would be using the flow vector calculated directly in point  $C$ ,

$$\boldsymbol{\sigma}_C = \boldsymbol{\sigma}_B - d\lambda \mathbf{c} \cdot \mathbf{a}_C \quad (6.9)$$

The flow vector in point  $C$  cannot be calculated only with data from point  $X$  and point  $B$ . Therefore, an iterative process using data from point  $B$  must be executed in order to reach point  $C$  from point  $B$ .

For each iteration, the effective plastic strain is updated in order to apply Eq.(6.5) and repeat the entirety of the process. At the end of the process, point  $B$  should be very close to point  $C$ , inside a given tolerance that is considered acceptable for the proximity. The whole process involves several iterations until such tolerance is reached and the next Gauss point is analyzed.

### 6.3 KT-ALL Algorithm - Mathematical Formulation

Since the KT-ALL will be the preferred non-linear solution algorithm used in this work, it is important to detail the mathematical formulation behind it.

In order to begin the elastoplastic analysis, information about the problem is needed. Problem dimensions, the mechanical properties of the material, nodal mesh, essential and natural boundary conditions, the elastoplastic ratio that is going to be used, selected tolerance for the solution and finally the number of increments in the analysis.

For all numerical methods being used, the pre-processing must occur before the algorithm initializes. Therefore, for the FEM, RPIM and NNRPIM, the creation of the nodal mesh, integration mesh and shape functions is done beforehand. While for the FEM, a specific type of element is used, for the RPIM and NNRPIM, certain parameters must be specified, which have already been addressed in previous chapters of this work.

After the pre-processing is done, the algorithm can begin. Taking into account the mechanical properties of the material and boundary conditions, the first stiffness matrix,  $\mathbf{K}_{11}$  is calculated. This stiffness matrix will be updated for every iteration of every increment. As a way of clarifying terminology,  $\mathbf{K}_{ij}$  is the stiffness matrix of increment  $i$  and iteration  $j$ .

At this point, the incremental load process can begin. The incremental load,  $\mathbf{f}_i$ , is given by,

$$\mathbf{f}_i = \frac{\mathbf{F}}{inc} \quad (6.10)$$

where  $\mathbf{F}$  is the maximum load being applied and  $inc$  is the number of increments considered.

The displacement field can be obtained by,

$$\mathbf{u}_{ij} = \mathbf{K}_{ij}^{-1} \mathbf{f}_i \quad (6.11)$$

and the stress field is given by,

$$\boldsymbol{\sigma}_{ij} = \mathbf{c} \cdot \mathbf{B} \cdot \mathbf{u}_{ij} \quad (6.12)$$

The stress on each Gauss point is verified. If the stress calculated at said Gauss point surpasses the yield surface, the stress-returning algorithm is invoked. At the end of this algorithm, the stress vector of the iteration,  $\delta\boldsymbol{\sigma}_{ij}$ , is calculated.

The total stress vector,  $\boldsymbol{\sigma}_{i(j-1)}$  can be obtained,

$$\boldsymbol{\sigma}_{ij} = \boldsymbol{\sigma}_{i(j-1)} + \delta\boldsymbol{\sigma}_{ij} \quad (6.13)$$

Therefore, if the present iteration of the present increment on a specific Gauss point has passed beyond the yield surface, the residual forces are calculated,

$$\mathbf{f}_{ij}^{res} = \mathbf{f}_i - \int_{\Omega} \mathbf{B}^T \delta\boldsymbol{\sigma} d\Omega \quad (6.14)$$

After the residual forces are calculated, an evaluation of the significance of the value calculated is necessary. The tolerance verification serves, also, as a stop criterion for the algorithm. If the residual forces are under the given tolerance, the iteration stops and the next increment is applied. If not, the next iteration begins.

After all the load increments are applied, the load-displacement curves can be obtained, along with other information about the problem.

# Chapter 7

## Numerical Analysis

In this chapter, the numerical analysis of some selected solid mechanics problems and examples is made.

For the purpose of developing the geometries and creation of the meshes, the software *FEMAP* (**F**inite **E**lement **M**odelling **A**nd **P**ostprocessing), developed by Siemens, was used.

The software that was used for all the numerical simulations runned in this work was provided and developed by Prof. Jorge Belinha. The *FEMAS* (**F**inite **E**lement and **M**eshless **A**nalysis **S**oftware) was used in order to impose all natural and essential boundary conditions, including load conditions. All of the pre-process, calculation and post-process of the FEM, RPIM and NNRPIM was also made using the *FEMAS*.

As for the numerical examples being presented, firstly, a Cook's Membrane is studied using a linear analysis in 2D and 3D. This example is followed by the elastoplastic analysis of uniformly loaded plates, both clamped and simply supported. Next, the elastoplastic analysis of frames, specifically a two bay asymmetric frame and a bow-string bridge, for 2D and 3D geometry is presented. Lastly, a bar, produced by 3D printing, is submitted to the three-point bending test experimentally. Then, the problem is analyzed considering FEMAS and the analytical results are compared with the experimental results.

### 7.1 Cook's Membrane

The first example being studied in this work is a classical problem in computational mechanics and used extensively in convergence analysis. It is commonly used to test the performance of a particular element under combined bending and shear. The primary focus of this example is not only to use the FEM, RPIM and NNRPIM to analyze the geometry in linear regime but also to test the convergence of the numerical methods being used against literature solutions.

#### 7.1.1 Geometry, Boundary and Load Conditions

Both 2D and 3D geometry of the Cook's Membrane studied in this section are represented in Figure 7.1.

For the 2D case, both  $u$  and  $v$  displacements are blocked in the left side of the geometry, as represented in Figure 7.1(a), while in the 3D case, represented in Figure 7.1(b), the  $u$ ,  $v$  and  $w$  displacements are blocked in the same section as the 2D geometry, adding a symmetry boundary condition for  $z = 0$ , where the displacement  $w$  is blocked. The thickness for the 2D case is of constant value and equal to  $e = 1$ , while in the 3D case, that same thickness will vary as the mesh is refined. This aspect will be explained further ahead.

As for the load conditions, a total force of 1 [N], uniformly distributed, is applied to the 2D and 3D case in the same location.

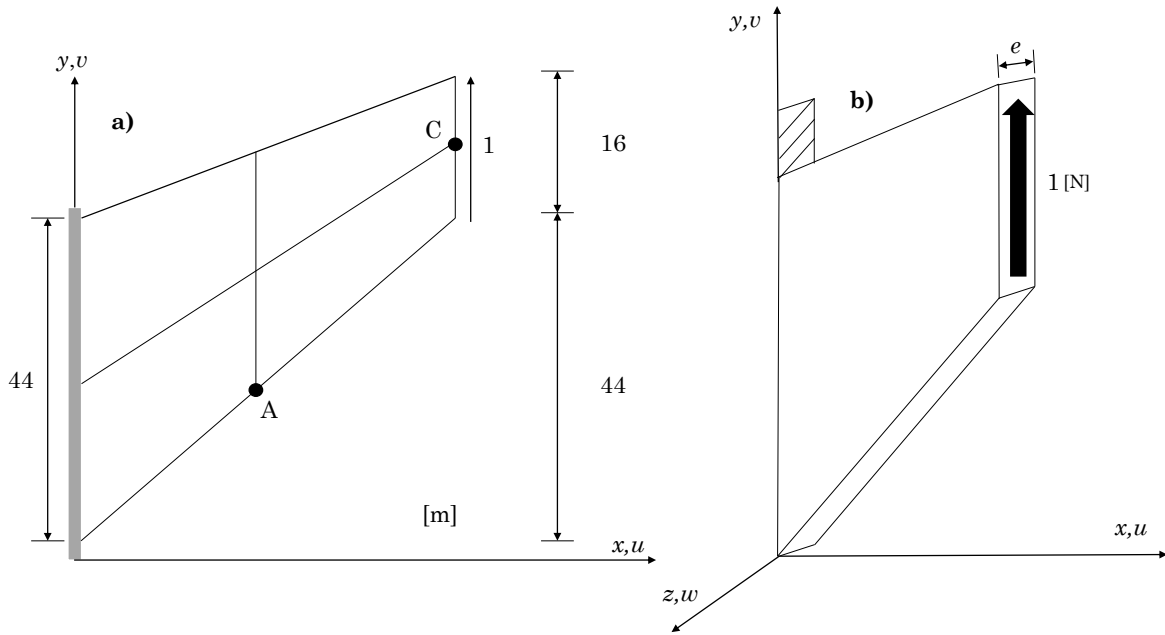


**Figure 7.1:** a) Cook's Membrane 2D geometry. b) Cook's Membrane 3D Geometry.

### 7.1.2 Mechanical Properties

The mechanical properties for both 2D and 3D cases are as follows,

| Mechanical Property             | Value  |
|---------------------------------|--------|
| Young's Modulus [E]             | 1 [Pa] |
| Poisson's Coefficient [ $\nu$ ] | 0,33   |

**Table 7.1:** Mechanical Properties of both 2D and 3D Cook's Membrane

### 7.1.3 Numerical Methods And Nodal Discretization

As explained previously in earlier chapters of this work, all numerical examples performed will be analyzed using three distinct numerical method: FEM, RPIM, NNR-PIM. The formulation for the FEM can be found in various reference works [6, 17],

while the formulation for the two meshless methods being used, RPIM and NNRPIM, can be found in Chapter 3.

As for the meshless method parameters being used throughout not only this example but all examples being performed, the following tables detail those same parameters,

| Meshless Parameters (2D)  | RPIM                                            | NNRPIM                                          |
|---------------------------|-------------------------------------------------|-------------------------------------------------|
| Nodes In Influence Domain | 16                                              | -                                               |
| Influence Cell Order      | -                                               | Second                                          |
| Parameter C               | 0,0001                                          | 0,0001                                          |
| Parameter P               | 0,9999                                          | 0,9999                                          |
| Polynomial Basis          | Constant ( $\mathbf{p}^T(\mathbf{x}) = \{1\}$ ) | Constant ( $\mathbf{p}^T(\mathbf{x}) = \{1\}$ ) |
| Integration Scheme        | 2                                               | 1 (Full Integration)                            |

**Table 7.2:** 2D meshless method parameters

| Meshless Parameters (3D)  | RPIM                                            | NNRPIM                                          |
|---------------------------|-------------------------------------------------|-------------------------------------------------|
| Nodes In Influence Domain | 27                                              | -                                               |
| Influence Cell Order      | -                                               | Second                                          |
| Parameter C               | 0,0001                                          | 0,0001                                          |
| Parameter P               | 0,9999                                          | 0,9999                                          |
| Polynomial Basis          | Constant ( $\mathbf{p}^T(\mathbf{x}) = \{1\}$ ) | Constant ( $\mathbf{p}^T(\mathbf{x}) = \{1\}$ ) |
| Integration Scheme        | 2                                               | 1 (Full Integration)                            |

**Table 7.3:** 3D meshless method parameters

As for the nodal discretization, for the 2D (Figure 7.2) and 3D (Figure 7.3) case, a quadratic discretization rule was used in order to test the convergence of the numerical methods.

As it can be observed in Figure 7.3, the thickness of the 3D model varies as the mesh is refined. This fact has a purpose, as to avoid corrupt elements in the interior of the model and to maintain a constant number of elements along the thickness of the geometry. For both 2D and 3D models, every edge has the same number of elements, and along the thickness of the 3D model, there is a constant number of 2 elements.

The thickness of the 3D for different meshes is obtained by the following equation,

$$e = \frac{44}{n} \cdot 2 \quad (7.1)$$

in which  $n$  is the number of elements each edge will be discretized in.

The following table summarizes the various thickness values used in the analysis,

As it can be observed in Table 7.4, the thickness of the model will assume 4 different values, for each element number along the edge.

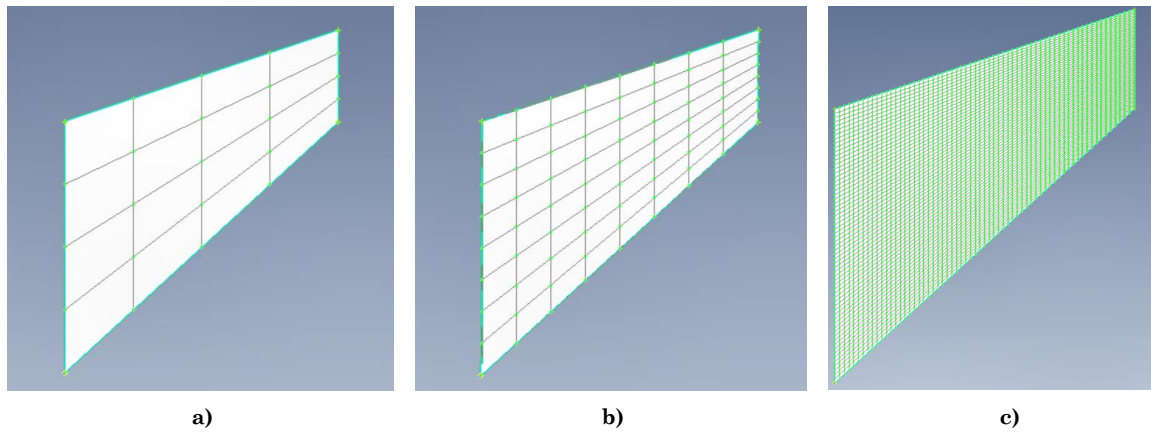


**Figure 7.2:** Nodal discretization for the 2D case. **a)** Discretization using 25 nodes (coarsest mesh used). **b)** Discretization using 81 nodes. **c)** Discretization using 4225 nodes (finest mesh used).

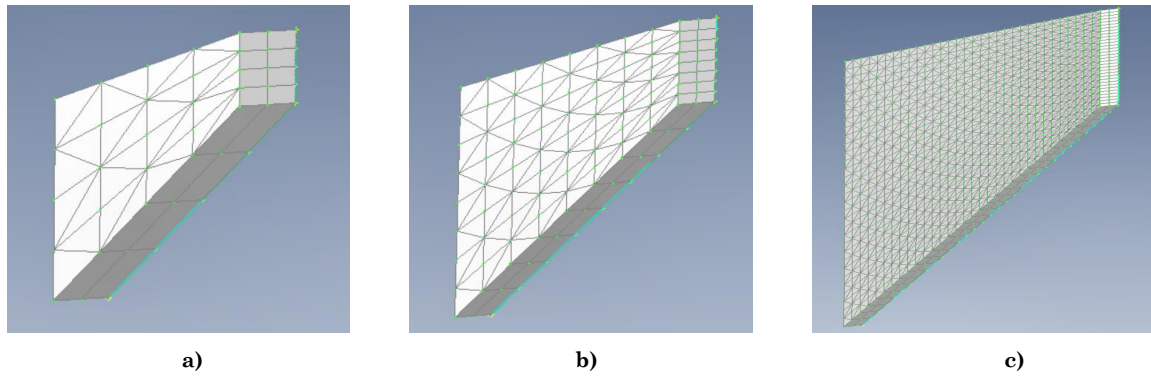


**Figure 7.3:** Nodal discretization for the 3D case. **a)** Discretization using 75 nodes (coarsest mesh used). **b)** Discretization using 243 nodes. **c)** Discretization using 3267 nodes (finest mesh used).

| N° Elements | Thickness [e] |
|-------------|---------------|
| 4           | 22            |
| 8           | 11            |
| 16          | 5,5           |
| 32          | 2,75          |

**Table 7.4:** Relationship between the number of elements along the edge of the geometry and the thickness of the model

### 7.1.4 Results

In this section the results obtained for the example proposed are shown and discussed.

The analysis is based on the article [59]. In the "Numerical Examples" section of the article, the Cook's Membrane is analyzed, where various solutions using different

elements are presented, including a reference solution to the problem, which is the value that is used for comparison in this work.

The vertical displacement,  $v$ , of point **C**, represented in Figure 7.1(a)) and the principal stress  $\sigma_1$  of point **A**, represented in the same figure, are used for the analysis of the convergence of the methods used.

#### 7.1.4.1 Vertical Displacement Of Point C

The following graphs show the results obtained for the 2D and 3D case with respect to the vertical displacement of point **C**, compared to the reference solution in [59].

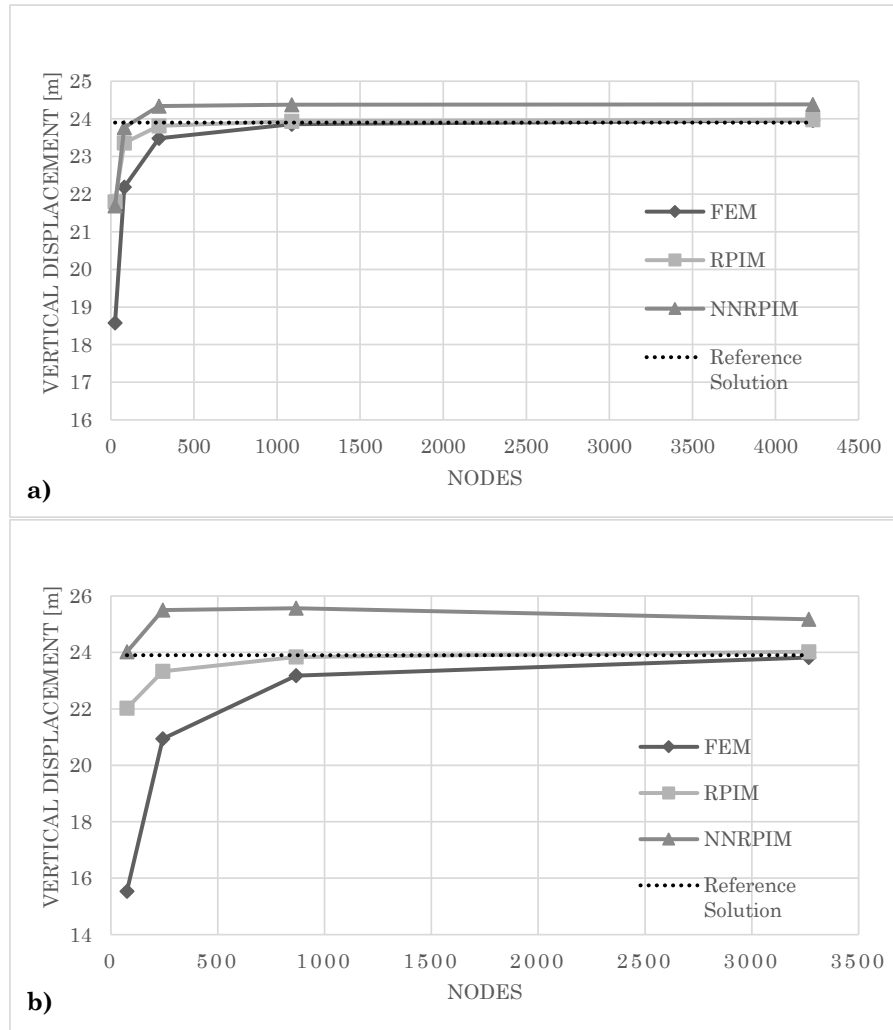


**Figure 7.4:** Vertical displacement of point **C**. a) 2D. b) 3D.

Analyzing Figure 7.4(a), it is clear that the RPIM and NNRPIM, for a discretization of 25 nodes, return a value which is closer to the reference solution than the FEM does. As the nodal density increases, the RPIM and NNRPIM converge faster than the FEM, with the RPIM for a value of 289 nodes approximating very closely to the reference solution, and maintaining an almost constant value afterwards, while the

NNRPIM follows the same tendency of constant value but the value in itself does not converge so well as the RPIM. The FEM takes more time to converge, however, for 1089 nodes, it maintains a constant value a very close to the reference solution.

As for Figure 7.4(b), both the RPIM and NNRPIM start at a higher value for the vertical displacement than the FEM, and closer to the reference solution. The RPIM, for a nodal density of 867 nodes comes very close to the reference solution and maintains almost a constant value as the nodal density increases. The NNRPIM reaches a higher value than the reference solution, and as the nodal density increases, tends to come closer to the reference value but with a higher difference than the RPIM or even the FEM. Discussing the FEM results, even though the value that the method returns for the coarsest mesh used is the farthest apart for the three methods used, it is evident that the FEM converges to the reference solution. Analyzing the graph shown for a nodal discretization of 3267 nodes, the FEM and RPIM return very similar values, while the NNRPIM is the farthest apart from the proposed solution.

For both graphs, it is perceptible that the numerical methods converge towards the reference solution, and demonstrate that the higher the nodal density is, the more accurate the results are, until a certain value at which the vertical displacement maintains a almost constant value.

#### 7.1.4.2 Principal Stress $\sigma_1$ Of Point A

The following graphs show the results obtained for the 2D and 3D with respect to the principal stress  $\sigma_1$  of point **A**, compared to the reference solution in [59].

Observing Figure 7.5(a), it can recognize the same trend as in the vertical displacement analysis done before. Both the RPIM and NNRPIM start at a higher and closer value to the reference solution than the FEM, for the lowest nodal density used. For this case, both the RPIM and the FEM for 1089 nodes come very close to the reference solution, while the NNRPIM, although close to the comparative value, again reaches a higher value and maintains it throughout the rising nodal density.

As for Figure 7.5(b), apart from the lowest nodal density in which the three methods return different starting values and have distinct evolutions, at a nodal density value of 867, all methods used show the same behaviour, with a clear tendency to converge towards the reference solution, with the RPIM, again, coming very close to the comparative value, while in this case the NNRPIM returns a value that is under the reference solution and not over, as it had occur previously, while the FEM comes under the reference solution for the highest nodal density used.

Again, the graphs analyzing the principle stress of point **A** point out the clear tendency of the numerical methods used to converge towards the solution proposed.

#### 7.1.4.3 Calculation Of The Difference Between The Values Obtained By The Numerical Methods And The Reference Solution

To better analyze the accuracy of the values obtained in the numerical simulations, the difference between the numerical values and the comparative values will be calculated.

For this purpose, a simple calculation is made using the following equation,

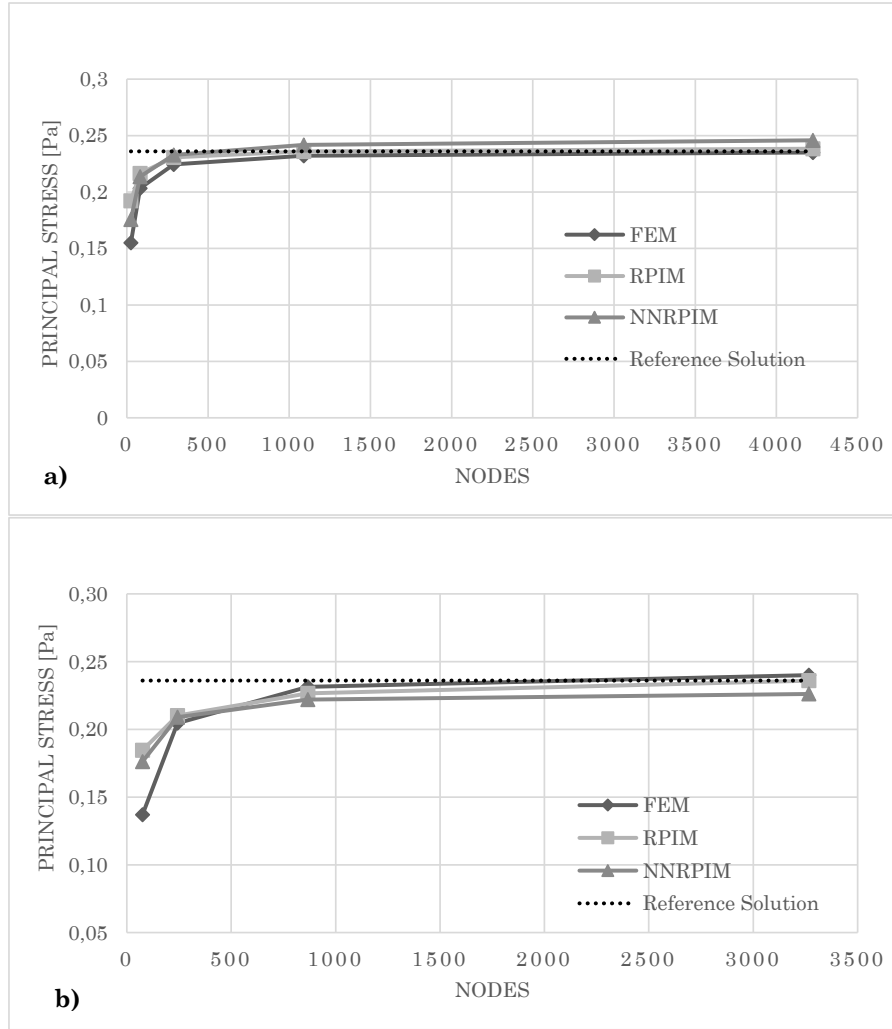


**Figure 7.5:** Principal stress of point A  
a) 2D. b) 3D.

$$\delta = \frac{N.V - R.S}{R.S} \quad (7.2)$$

where  $N.V$  is the numerical value obtained by the FEM, RPIM and NNRPIM in the different simulations runned,  $R.S$  is the reference solution for each case (vertical displacement and principal stress) and  $\delta$  is the difference between the two values.

The following tables organize this calculation by different meshes, nodes and numerical methods, as shown below,

Observing Table 7.7, it is evident that the FEM, for the higher nodal density, achieves the lowest difference between the numeric value and the reference solution. The FEM and RPIM, as the nodal density rises, converge towards the reference solution, pointing out the fact that the RPIM converges faster than the FEM for this case, even though the final value is less accurate. The NNRPIM has the highest difference between the three methods used.

| Mesh  | Elements Along Edge | Nodes | FEM [m] | RPIM [m] | NNRPIM [m] | Ref. Sol. [m] |
|-------|---------------------|-------|---------|----------|------------|---------------|
| $M_1$ | 4                   | 25    | 18,58   | 21,80    | 21,69      | 23,90         |
| $M_2$ | 8                   | 81    | 22,19   | 23,36    | 23,77      | 23,90         |
| $M_3$ | 16                  | 289   | 23,48   | 23,81    | 24,34      | 23,90         |
| $M_4$ | 32                  | 1089  | 23,86   | 23,94    | 24,37      | 23,90         |
| $M_5$ | 64                  | 4225  | 23,96   | 23,98    | 24,38      | 23,90         |

**Table 7.5:** Vertical displacement values obtained for the different numerical methods used for the 2D case.

| Mesh  | Elements Along Edge | Nodes | FEM [m] | RPIM [m] | NNRPIM [m] | Ref. Sol. [m] |
|-------|---------------------|-------|---------|----------|------------|---------------|
| $M_1$ | 4                   | 75    | 15,54   | 22,02    | 24,02      | 23,90         |
| $M_2$ | 8                   | 243   | 20,94   | 23,33    | 25,50      | 23,90         |
| $M_3$ | 16                  | 867   | 23,18   | 23,83    | 25,56      | 23,90         |
| $M_4$ | 32                  | 3267  | 23,82   | 24,02    | 25,18      | 23,90         |

**Table 7.6:** Vertical displacement values obtained for the different numerical methods used for the 3D case.

| Mesh  | FEM [ $\frac{m}{m}$ ] | RPIM [ $\frac{m}{m}$ ] | NNRPIM [ $\frac{m}{m}$ ] |
|-------|-----------------------|------------------------|--------------------------|
| $M_1$ | -2,23E-01             | -8,79E-02              | -9,25E-02                |
| $M_2$ | -7,16E-02             | -2,25E-02              | -5,54E-03                |
| $M_3$ | -1,74E-02             | -3,74E-03              | 1,83E-02                 |
| $M_4$ | -1,83E-03             | 1,54E-03               | 1,99E-02                 |
| $M_5$ | 2,47E-03              | 3,36E-03               | 2,02E-02                 |

**Table 7.7:** Value of  $\delta$  for the FEM, RPIM and NNRPIM considering the reference value for the vertical displacement for the 2D case.

| Mesh  | FEM [ $\frac{m}{m}$ ] | RPIM [ $\frac{m}{m}$ ] | NNRPIM [ $\frac{m}{m}$ ] |
|-------|-----------------------|------------------------|--------------------------|
| $M_1$ | -3,50E-01             | -7,86E-02              | 4,97E-03                 |
| $M_2$ | -1,24E-01             | -2,38E-02              | 6,69E-02                 |
| $M_3$ | -3,03E-02             | -2,73E-03              | 6,96E-02                 |
| $M_4$ | -3,47E-03             | 5,15E-03               | 5,34E-02                 |

**Table 7.8:** Value of  $\delta$  for the FEM, RPIM and NNRPIM considering the reference value for the vertical displacement for the 3D case.

For the 3D case, analyzing Table 7.8, it is possible to observe the same trend as before regarding the rate of the convergence. The RPIM and NNRPIM both have a lower difference than the FEM does for the lowest nodal density. However, for the highest nodal density, the FEM achieves the lowest value of difference of the numerical methods tested, mirroring the results obtained for the 2D case.

#### 7.1.4.4 $\sigma_x$ Stress Along The $y$ Axis Of The Fixed Edge Of The Membrane

The following graphs represent the  $\sigma_x$  value obtained by the three numerical methods along the fixed edge of the membrane, for 2D and 3D geometry. The values represented are for the highest nodal density tested (3267 nodes).

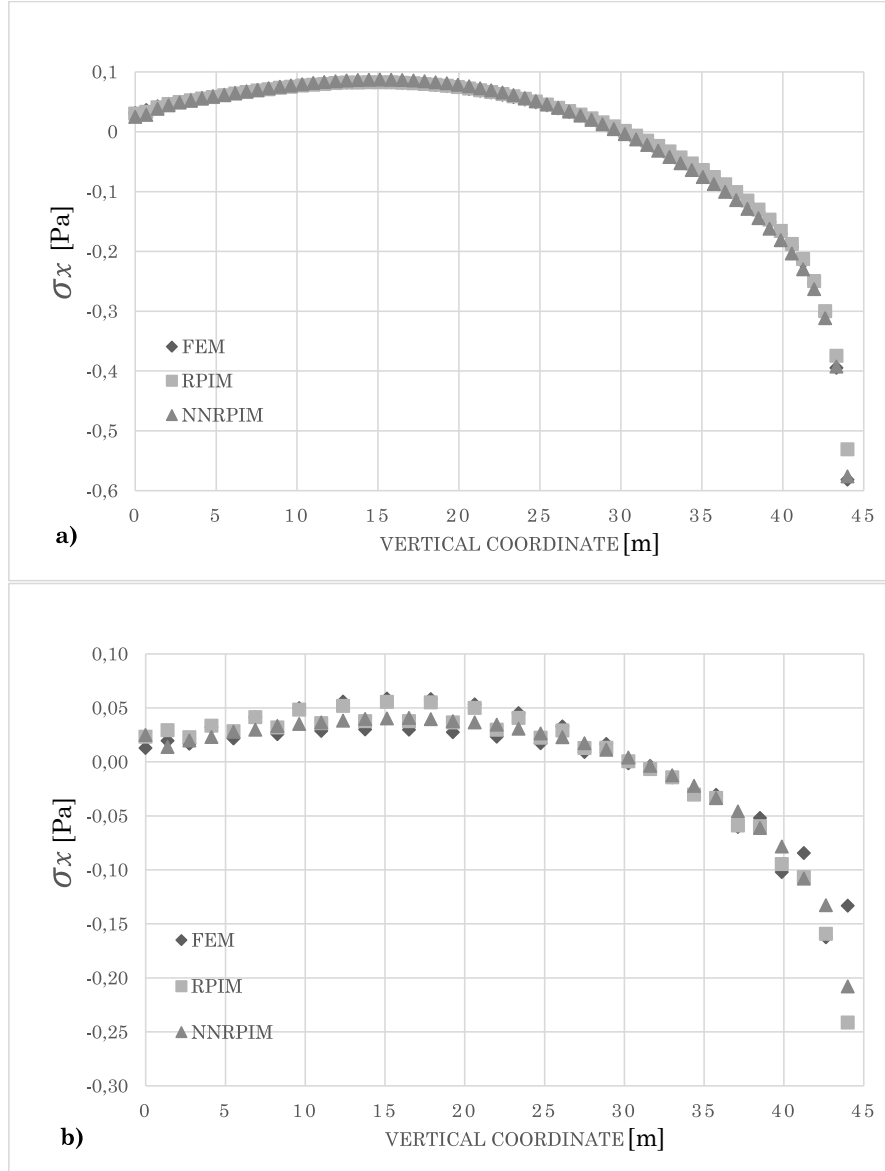


**Figure 7.6:**  $\sigma_x$  stress along the fixed edge of the membrane. a) 2D. b) 3D

Analyzing Figure 7.6, for both 2D and 3D geometries, all three methods return almost equal results. Comparing the 2D case with the 3D, there is good agreement between them, although the 3D case returns stress values lower than the 2D case. This issue can be resolved by increasing the nodal density of the 3D case, although the computational cost, specifically with three-dimensional geometries, is very high.

#### 7.1.4.5 Von Mises Effective Stress

The following figures show the Von Mises effective stress distribution along the Cook's Membrane for the different numerical methods used and the different nodal density's tested, both for 2D and 3D.

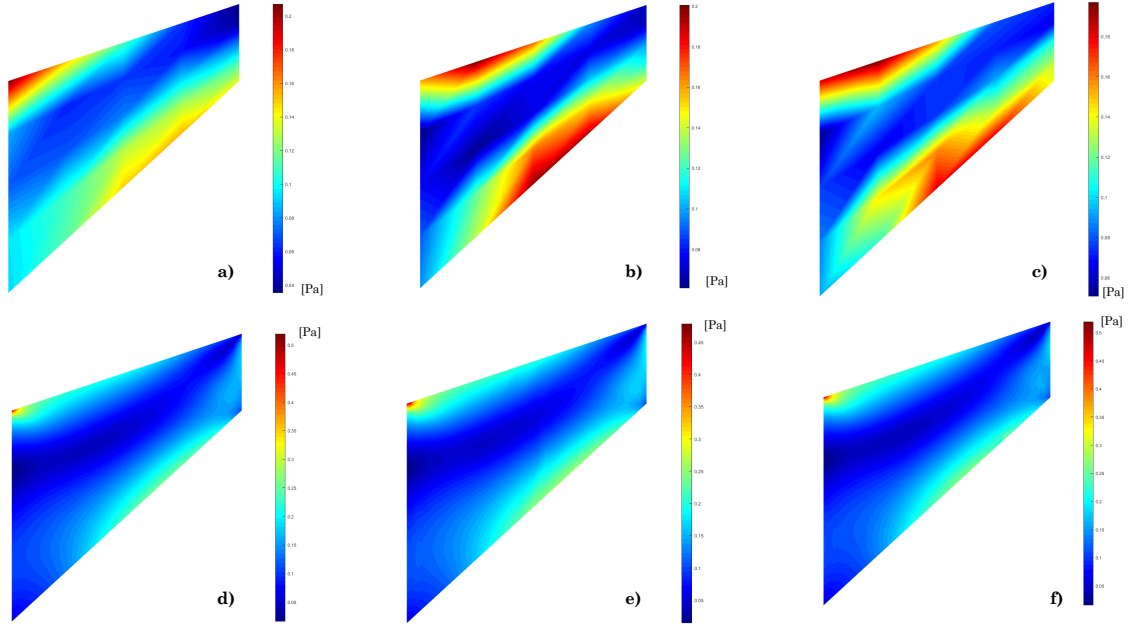


**Figure 7.7:**  $\sigma_{eff}$  stress distribution along the Cook's Membrane for the 2D case. **a)**, **b)** and **c)** FEM, RPIM and NNRPIM for 25 nodes. **d)**, **e)** and **f)** FEM, RPIM and NNRPIM for 4225 nodes.

Firstly, globally analyzing Figure 7.7, it can be observed that the three numerical methods, for the highest nodal density return almost identical stress distributions. Observing the top left corner of the membrane, for all numerical methods used, the stress value and its distribution on that location is the same. Transitioning to the location where point **A** is located and its surrounding area, the stress distribution for the FEM, RPIM and NNRPIM is approximately the same, where the RPIM extends the stress area further than the other methods, with the same stress value. We can conclude, therefore, that the three numerical values lead to approximately the same result, with small variations between them.

Comparing the evolution from the lowest nodal density used to the highest, it can be determined some interesting characteristics about the three methods. For the coarsest mesh used, the stress distribution obtained by the FEM is not only smoother but the closest to the final distribution, which can be assessed observing the area where point **A** is located and the top left corner. The FEM returns a reduced stress area for these locations comparing with the RPIM and NNRPIM. However, the stress values are of the same magnitude for the three numerical methods. The RPIM and NNRPIM stress distributions are not as smooth as the FEM, but the RPIM stress distribution is smoother than the NNRPIM's distribution.

Analyzing Figure 7.8, for the coarsest mesh, the stress distribution for the RPIM

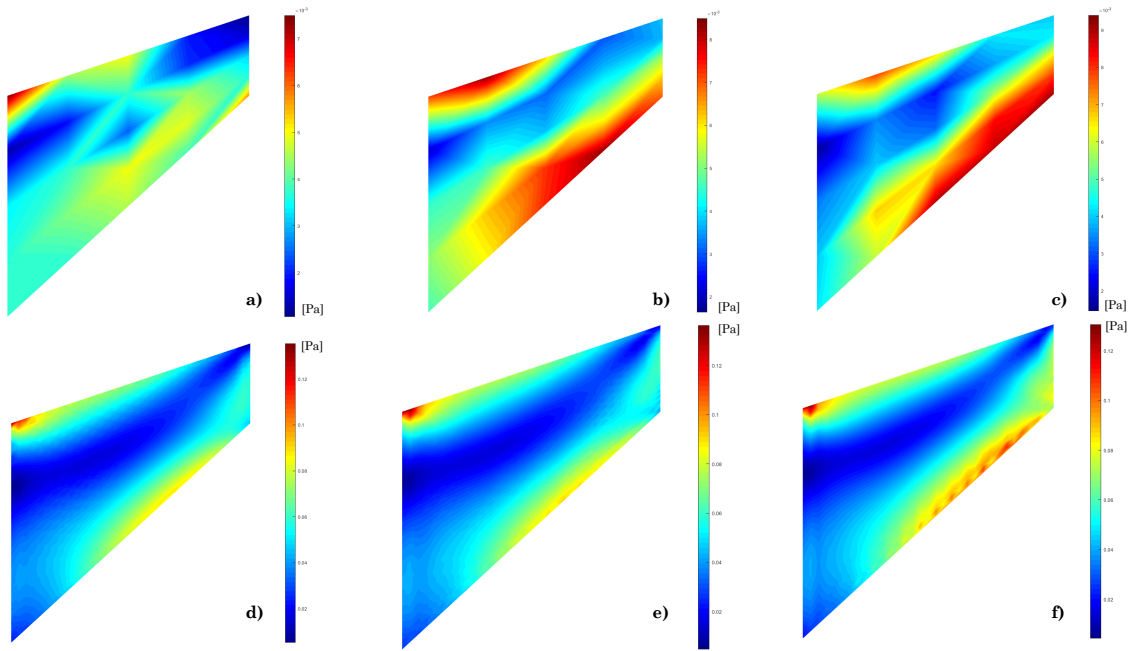


**Figure 7.8:**  $\sigma_{eff}$  stress distribution along the Cook's Membrane for the 3D case. **a)**, **b)** and **c)** FEM, RPIM and NNRPIM for 75 nodes. **d)**, **e)** and **f)** FEM, RPIM and NNRPIM for 3267 nodes.

and NNRPIM are similar, with some small deviation in the stress values and distribution, especially at the lower left corner. The FEM distribution is not as similar. Making a comparison with the 2D case for the same nodal density, it can be confirmed that the stress distribution for the RPIM and NNRPIM are quite similar, while the FEM distribution is not, specially in the middle section of the membrane.

Analyzing the results for the denser mesh used, it can be observed that the stress distribution is quite similar to the 2D analyzed before (Figure 7.7). The stress values obtained by the three numerical methods are identical within the 3D case, and the location of those values are very similar throughout them. There is a small variation of the results for the NNRPIM, specifically at the surrounding area of the location of point **A**.

## 7.2 Elastoplastic Analysis Of Uniformly Loaded Plates

In this section, simply-supported and clamped 3D plates in elastoplastic regime are analyzed. The three non-linear solution algorithms (KT-0, KT-1 and KT-ALL) are used and compared for the three numerical methods and for both plate configurations. The results are compared with Belinha et al [60]

### 7.2.1 Geometry, Boundary And Load Conditions

The 3D plate, represented in Figure 7.9, is studied.

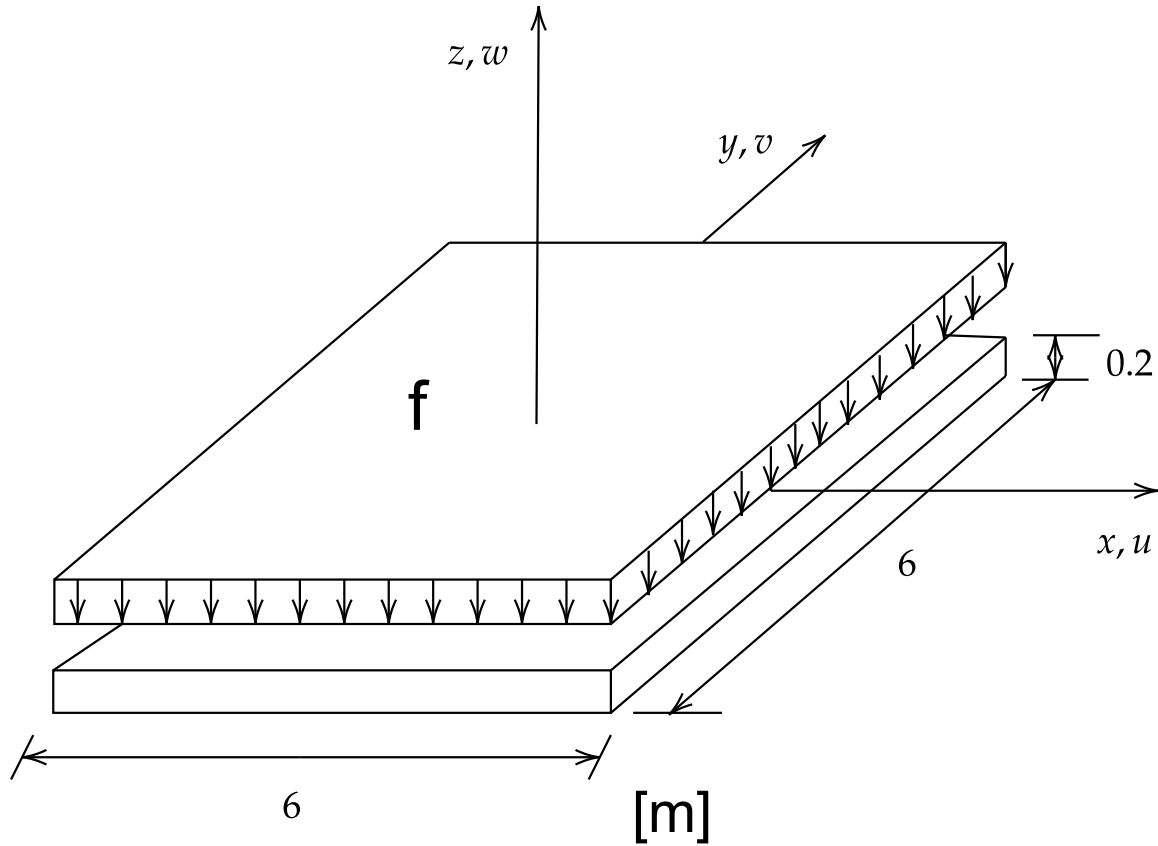


**Figure 7.9:** Square plate representation

As it can be seen, a uniform load is applied to the plate.

As for the boundary conditions, both clamped and simply supported plates will be studied, with the same load conditions and geometry. Taking into account symmetry, only a quarter of the plate will be analyzed, as it can be seen in Figure 7.10.

For the simply supported plate, only the center line is completely blocked for all possible displacements. As for the clamped plate, the whole exterior planes are blocked for all displacements. For the interior planes, the boundary conditions result from the applied symmetry to the solid.

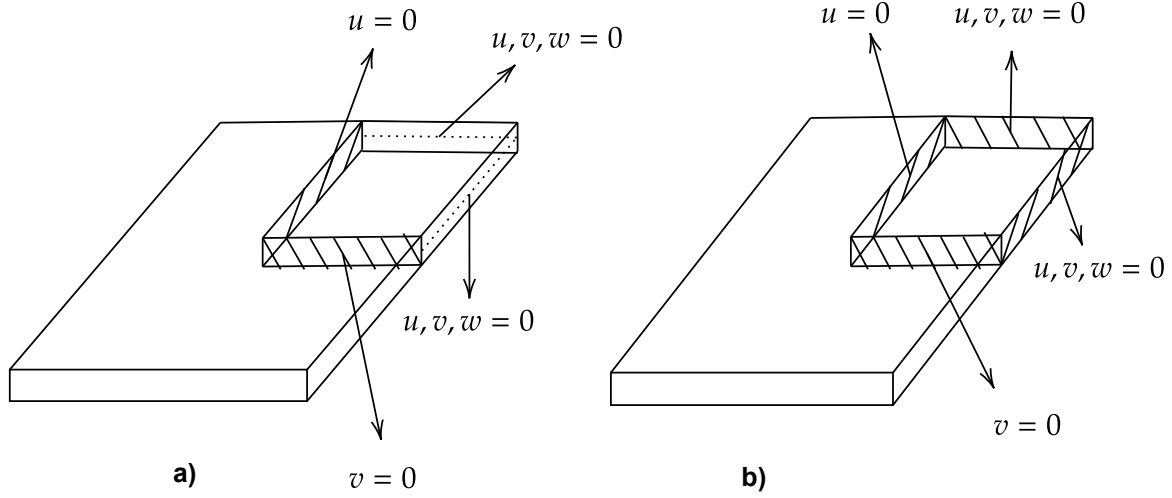


**Figure 7.10:** Boundary conditions for the square plate considering symmetry. **a)** Simply Supported Plate. **b)** Clamped Plate.

### 7.2.2 Mechanical Properties

The mechanical properties of the material are as follows,

| Mechanical Properties | Value     |
|-----------------------|-----------|
| $E$ [Pa]              | 206,7E+09 |
| $E_T$ [Pa]            | 2,067E+09 |
| $\nu$                 | 0,3       |
| $\sigma_Y$ [Pa]       | 206,7E+06 |

**Table 7.9:** Mechanical properties of the plate.

### 7.2.3 Reference Sections And Points

The results of the analysis will be focused on the point and sections represented in Figure 7.11.

### 7.2.4 Load/Normalized Displacement Of Point P

In this section, the results obtained for the vertical displacement of point **P** will be presented.

The displacement in point **P** was calculated for both plate configurations and using the three algorithms for each numerical method.

In order to obtain normalized curves, the displacement and load are calculated and presented with the following calculation, respectively,

$$w_{norm} = \frac{wD}{M_p L^2} \quad (7.3)$$

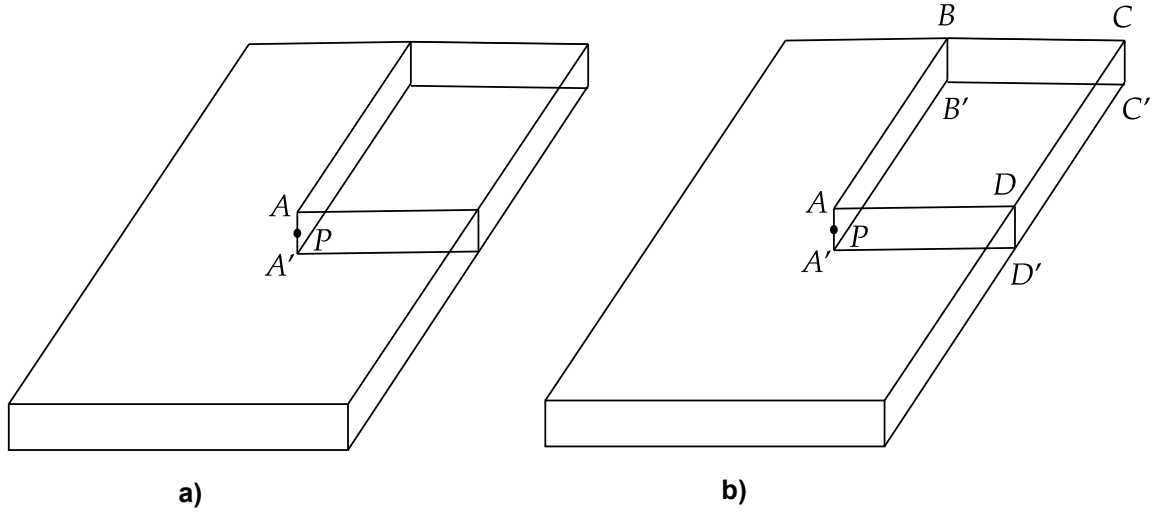


**Figure 7.11:** Reference point and sections for both boundary conditions. **a)** Simply Supported Plate. **b)** Clamped Plate.

$$f_{norm} = \frac{fL^2}{M_p} \quad (7.4)$$

with,

$$D = \frac{100Eh^3}{12(1 - \nu^2)} \quad (7.5)$$

and,

$$M_p = \sigma_Y \frac{h^2}{4} \quad (7.6)$$

where  $h$  is the thickness of the plate,  $E$  the Young Modulus,  $\nu$  the Poisson's coefficient,  $M_p$  the plastic moment and  $\sigma_Y$  the yield stress of the material being used.

## 7.2.4.1 Load/Normalized Displacement Of Point P For The Clamped Plate

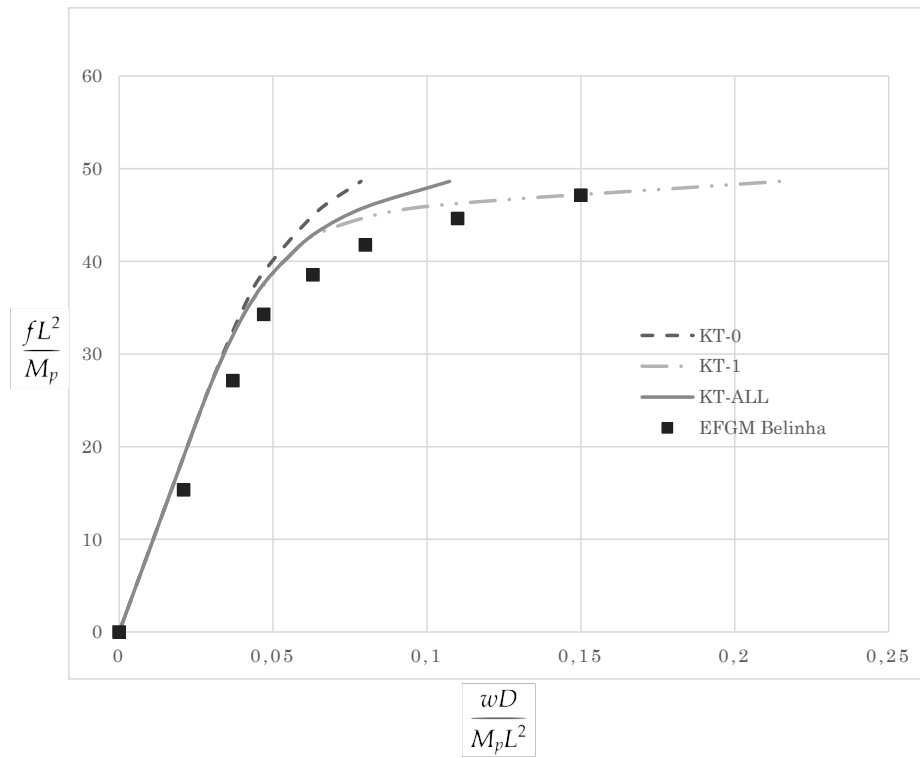


**Figure 7.12:** Vertical displacement of point **P** for the three non-linear solution algorithms using the FEM

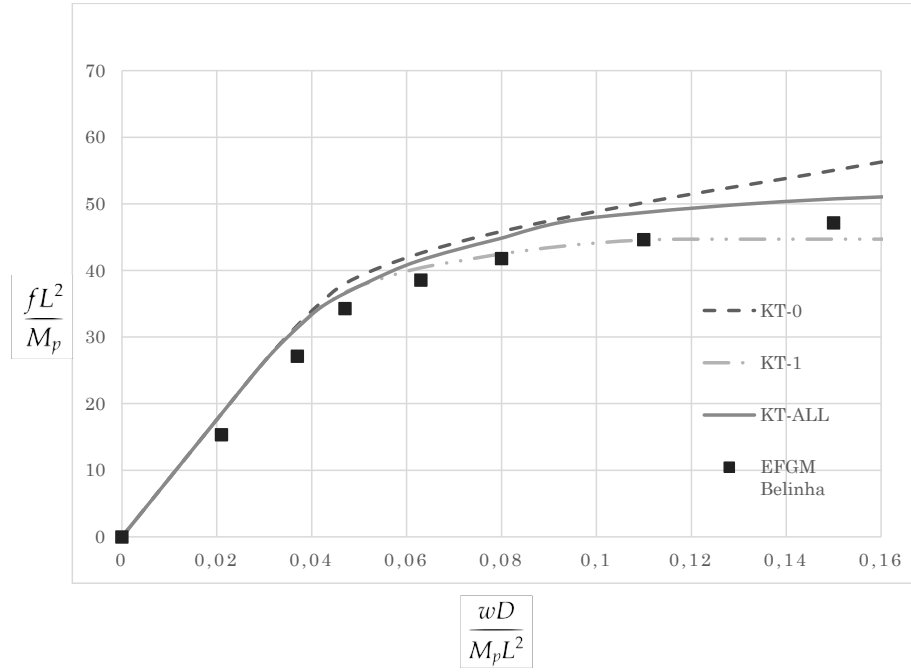


**Figure 7.13:** Vertical displacement of point **P** for the three non-linear solution algorithms using the RPIM

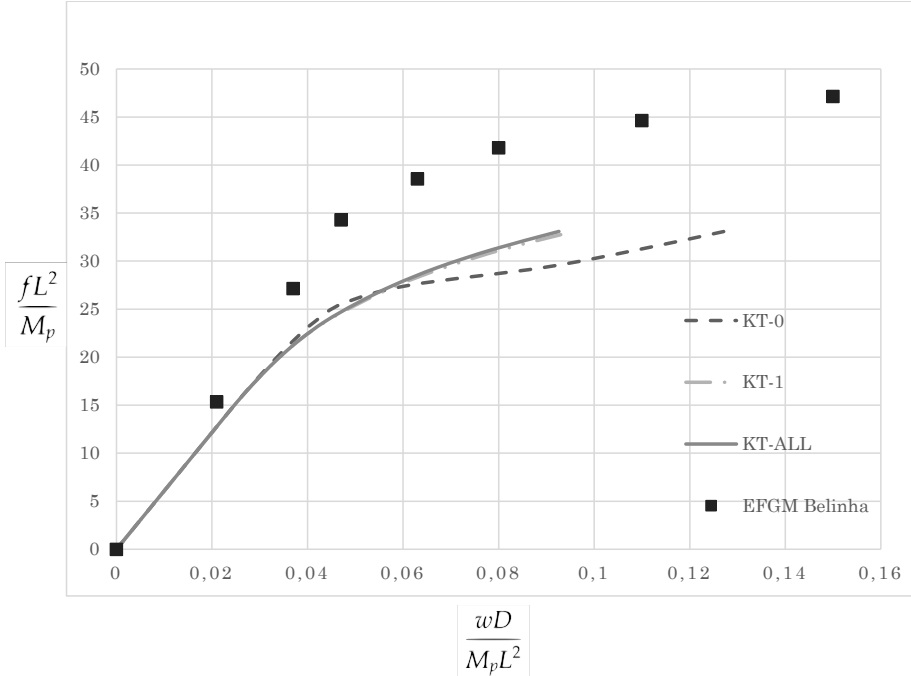


**Figure 7.14:** Vertical displacement of point **P** for the three non-linear solution algorithms using the NRPIM

Analyzing Figure 7.12 and 7.13, it can be observed that the results allow to obtain a good approximation with the ones documented in Belinha [60], and by consequence

with the ones obtained by Figueiras [61] and Barbosa [62].

As for Figure 7.14, the results obtained by the NNRPIM using the three non-linear algorithms deviate from the reference solution, with displacements values below the ones expected. The cause for this deviation is the number of nodes used in the simulation. While the FEM and RPIM, for the same number of nodes as the NNRPIM, obtain very valid and close solutions, the NNRPIM need a higher number of nodes to reach the reference solution. This fact shows that the NNRPIM, in general, needs a much higher number of nodes to reach a solution that can be compared with other numerical methods and reference solutions.

On the other hand, the results obtained by the NNRPIM follow the same tendency as the ones obtained by the FEM and RPIM, with the shape of the curves being similar to what is expected from a plasticity solution.

In more general terms, in all three figures, the plasticity behaviour is clearly depicted. After yielding, the load/displacement curve is clearly similar to the one represented in Figure 5.1(c)), considering the non-linear elastoplastic model.

#### 7.2.4.2 Load/Normalized Displacement Of Point P For The Simply Supported Plate

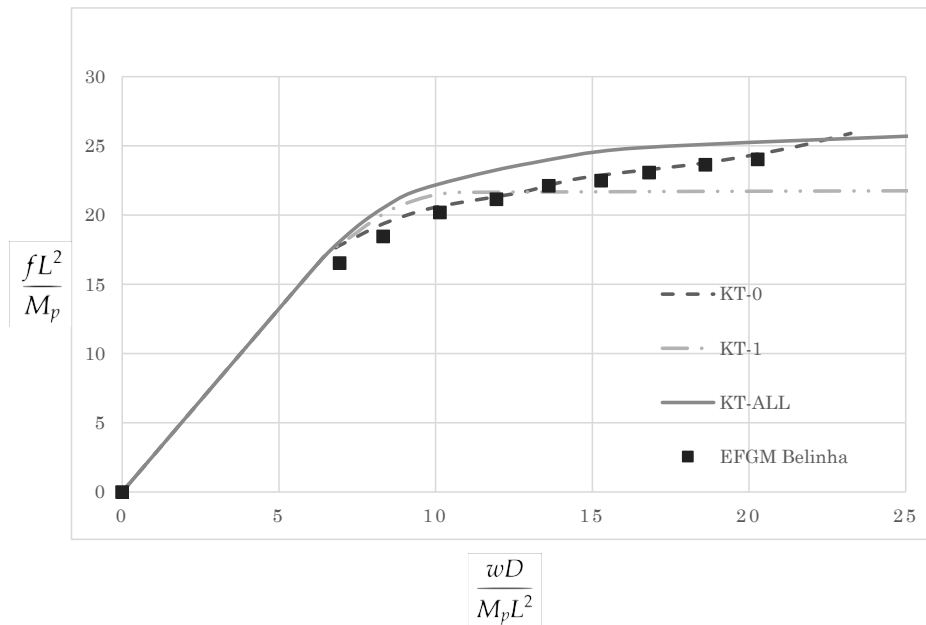


**Figure 7.15:** Vertical displacement of point **P** for the three non-linear solution algorithms using the FEM

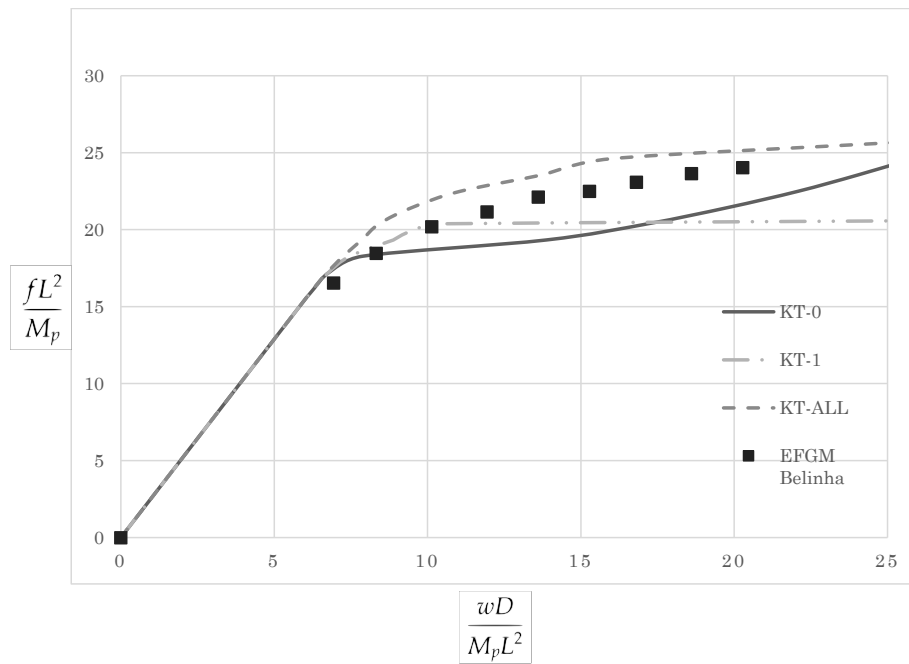


**Figure 7.16:** Vertical displacement of point  $P$  for the three non-linear solution algorithms using the RPIM

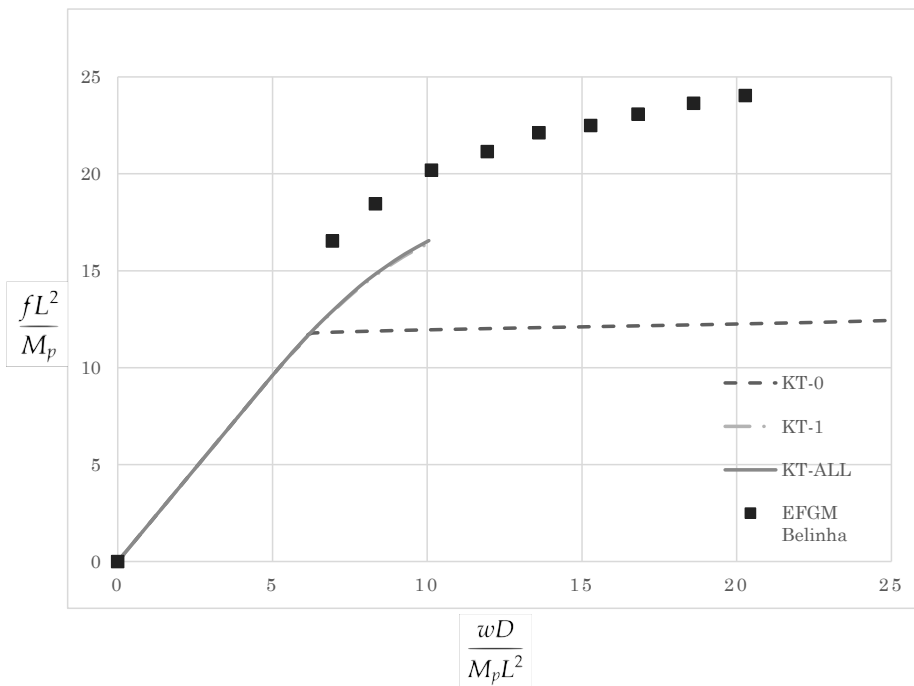


**Figure 7.17:** Vertical displacement of point  $P$  for the three non-linear solution algorithms using the NNRPIM

For the simply supported plate, the same type of results are obtained. The FEM and RPIM reach a good approximation with the reference solutions, while the NNR-

PIM stays below the reference values. Once again it is evident the need for the NNR-PIM to use a higher number of nodes.

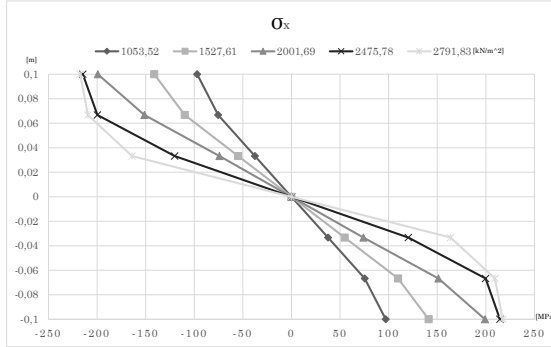
In general terms, once more, the curves obtained for the three numerical values are what is expected from a plasticity point of view.

### **7.2.5 Stress Diagrams Along The Clamped Plate Thickness**

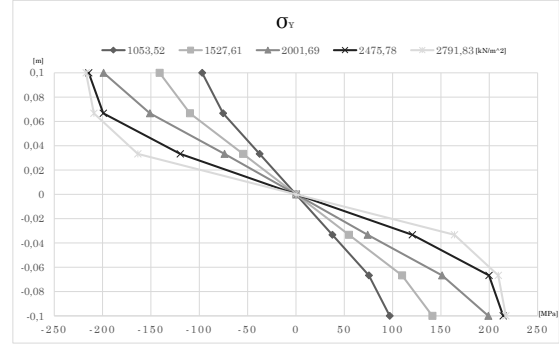
In this section, the stress diagrams along the clamped plate thickness, for the three numerical methods and for the KT-ALL algorithm will be presented.

As for units, the increments are in  $\frac{kN}{m^2}$ , the stress value ( $x$  axis) in  $MPa$  and the thickness coordinate ( $y$  axis) is in meters.

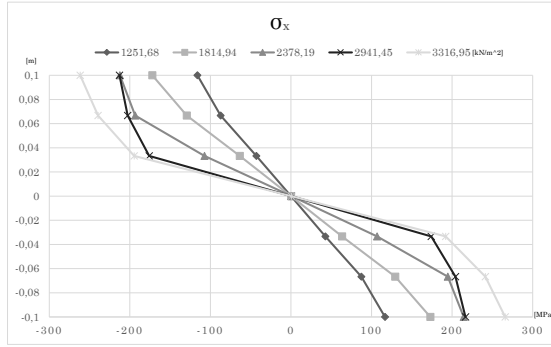
### 7.2.5.1 $\sigma_x$ , $\sigma_y$ , $\tau_{xz}$ , $\tau_{yz}$ stress diagrams along section A-A'



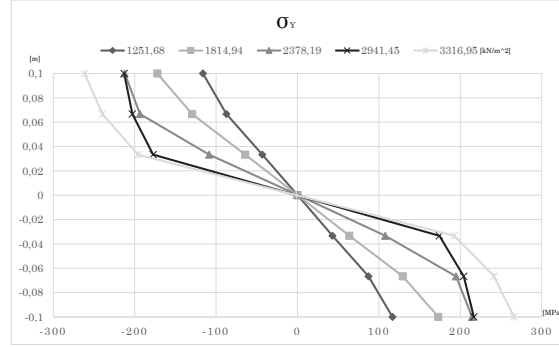
(a)  $\sigma_x$  stress for the FEM



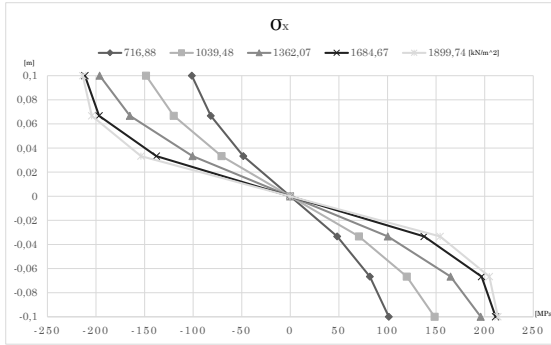
(b)  $\sigma_y$  stress for the FEM



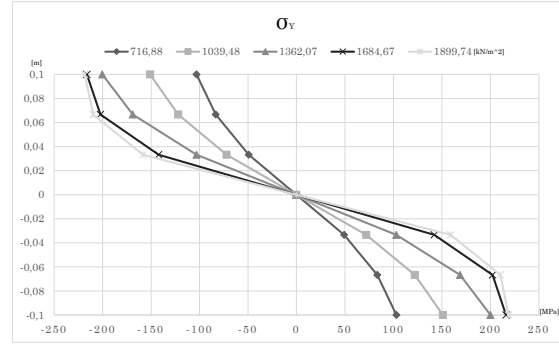
(c)  $\sigma_x$  stress for the RPIM



(d)  $\sigma_y$  stress for the RPIM

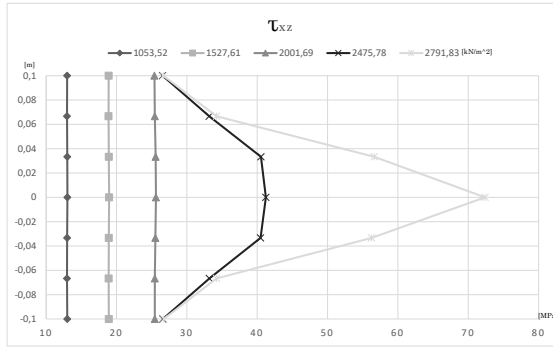
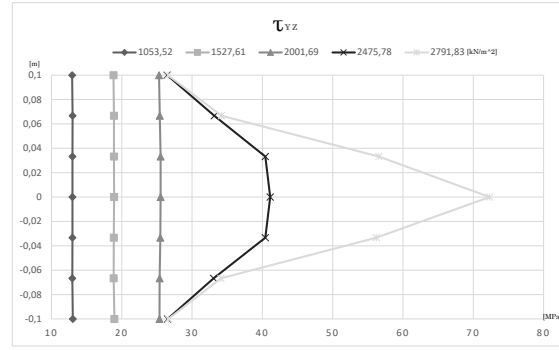
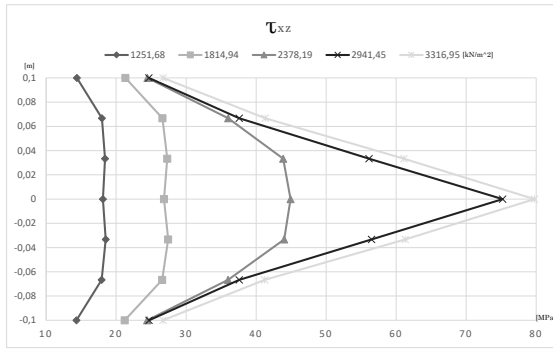
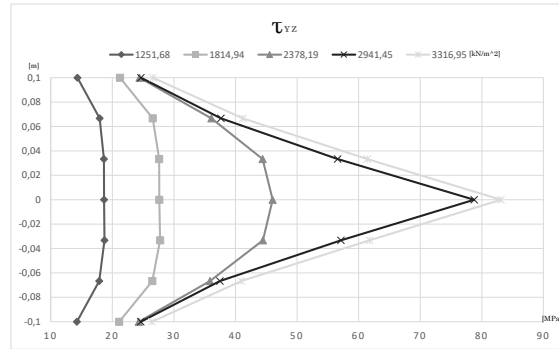
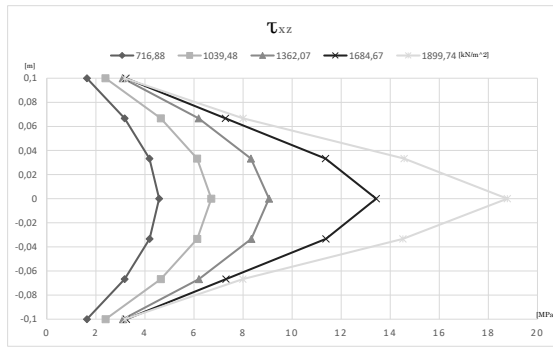
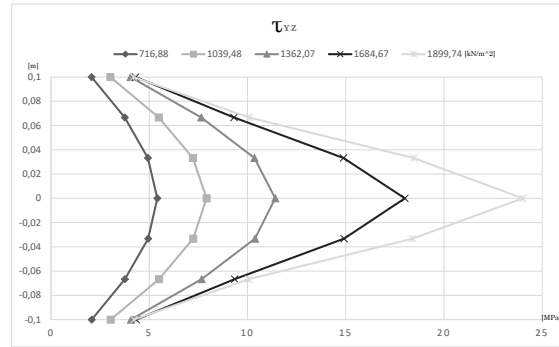


(e)  $\sigma_x$  stress for the NNRPIM



(f)  $\sigma_y$  stress for the NNRPIM

**Figure 7.18:**  $\sigma_x$  and  $\sigma_y$  stress diagrams along section A-A'

(a)  $\tau_{xz}$  stress for the FEM(b)  $\tau_{yz}$  stress for the FEM(c)  $\tau_{xz}$  stress for the RPIM(d)  $\tau_{yz}$  stress for the RPIM(e)  $\tau_{xz}$  stress for the NNRPIM(f)  $\tau_{yz}$  stress for the NNRPIM**Figure 7.19:**  $\tau_{xz}$  and  $\tau_{yz}$  stress diagrams along section A-A'

Analyzing Figure 7.18, it is visible that for all numerical methods, the  $\sigma_x$  and  $\sigma_y$  stress distribution along the plate thickness is very similar between them. As for the numerical values, for the FEM, RPIM and NNRPIM, the stress values achieved are very close between them.

For all diagrams, as the load value increases, it is visible the modification of the stress curve, from a straight line (concerning the yield stress), to a "approximate" straight line parallel to the  $x$  axis, as it is expected.

In terms of comparison between  $\sigma_x$  and  $\sigma_y$ , as expected, the stress values and distribution is equal, since we are analyzing the center section of the plate.

As for Figure 7.19, the  $\tau_{xz}$  and  $\tau_{yz}$  stress distribution approximates the shape of a parabola, as expected. Due to the fact that the nodal discretization of the thickness

consisted of 6 nodes, the shape of the diagrams illustrated, as the load increases, tends to a spike. If, on the other hand, the nodal discretization of the thickness was much higher, even for the highest load increment, the curve shape would tend to a parabola as well, as it does for lower values of the load.

Once again, for all three numerical methods, the stress distribution is similar between them. The FEM and RPIM achieve  $\tau_{xz}$  and  $\tau_{yz}$  approximately equal between them. As for the NNRPIM, although the stress distribution is equal, the values are not as close. This is due to, once again, the nodal discretization used. The NNRPIM needs a higher nodal density considering the thickness of the plate in order to achieve the load increment values and, therefore, stress values achieved by the FEM and RPIM.

As the load increases, the plasticity behaviour of the plate is clearly represented for all three methods, with a straight line or small parabola at the yield point and the development of the definitive parabola at the highest load increment. This is due to the continuous plastification of the material, as the load increases.

### 7.2.5.2 $\sigma_x$ , $\sigma_y$ , $\tau_{xy}$ , $\tau_{yz}$ stress diagrams along section B-B'

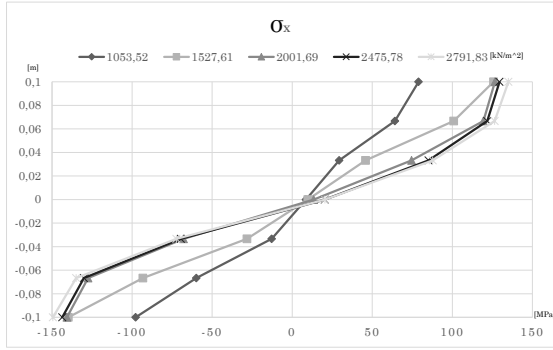
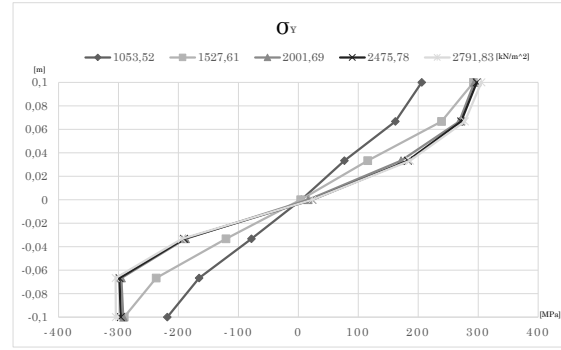
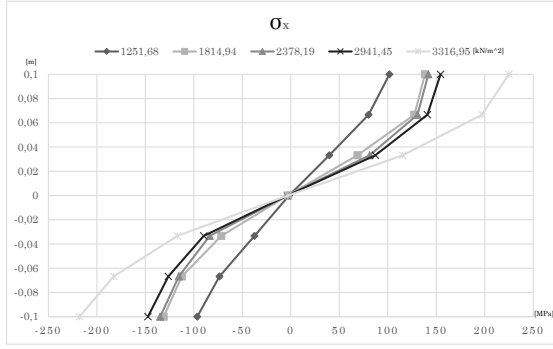
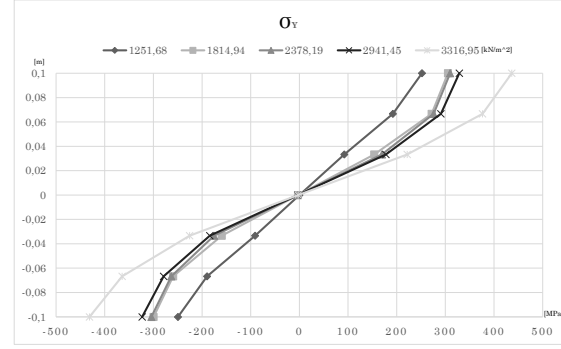
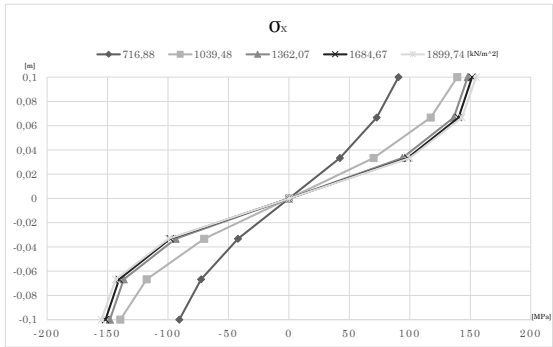
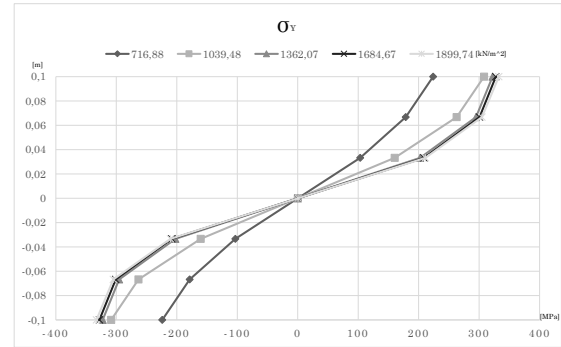
(a)  $\sigma_x$  stress for the FEM(b)  $\sigma_y$  stress for the FEM(c)  $\sigma_x$  stress for the RPIM(d)  $\sigma_y$  stress for the RPIM(e)  $\sigma_x$  stress for the NNRPIM(f)  $\sigma_y$  stress for the NNRPIM

**Figure 7.20:**  $\sigma_x$  and  $\sigma_y$  stress diagrams along section B-B'

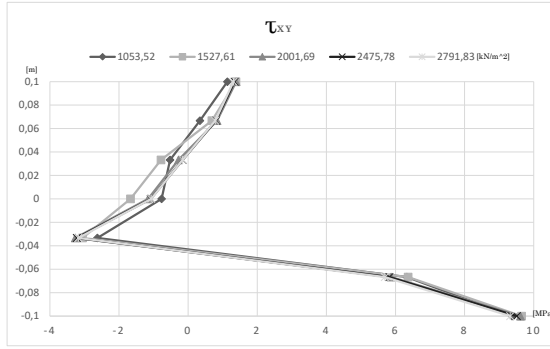
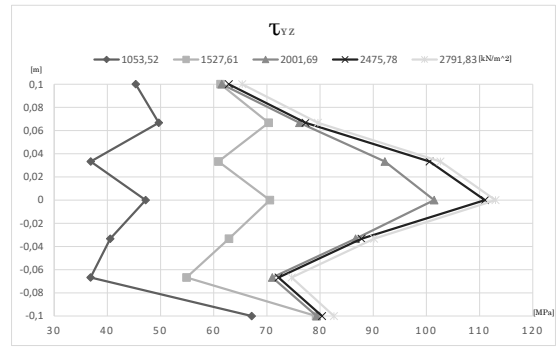
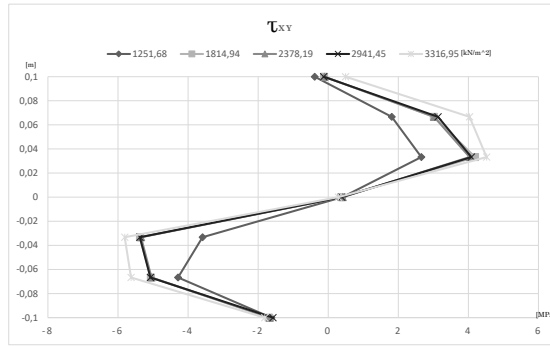
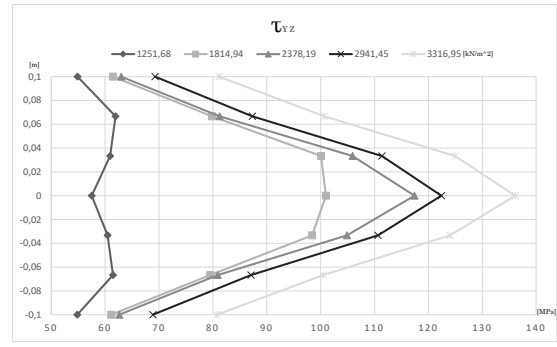
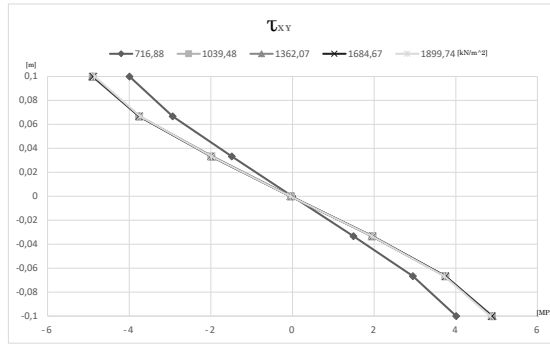
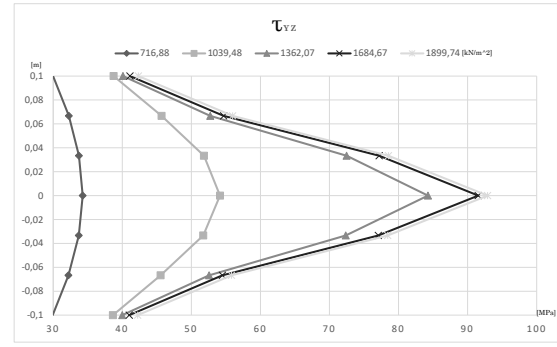

 (a)  $\tau_{xy}$  stress for the FEM

 (b)  $\tau_{yz}$  stress for the FEM

 (c)  $\tau_{xy}$  stress for the RPIM

 (d)  $\tau_{yz}$  stress for the RPIM

 (e)  $\tau_{xy}$  stress for the NNRPIM

 (f)  $\tau_{yz}$  stress for the NNRPIM

**Figure 7.21:**  $\tau_{xy}$  and  $\tau_{yz}$  stress diagrams along section B-B'

Once again, for Figure 7.20, the  $\sigma_x$  and  $\sigma_y$  stress distribution and values are very similar for the three methods, and the plasticity behaviour of the material is clearly represented.

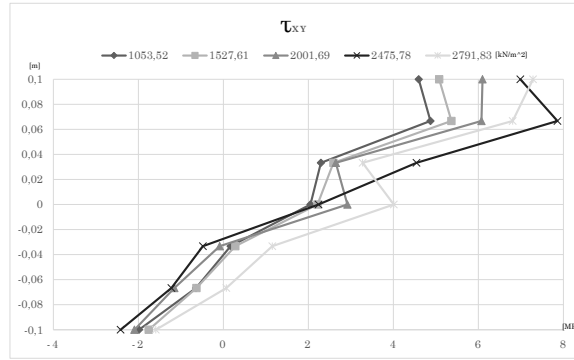
As for Figure 7.21, concerning the  $\tau_{xy}$  stress, the stress distribution is irregular between the three methods. It is important to note that, for the B-B' section, this stress value should be close to zero. Considering the magnitude of the values for  $\sigma_x$  and  $\sigma_y$ , the values achieved for the  $\tau_{xy}$  stress are much lower. Therefore, they are very close to zero considering this fact. As for the distribution variation, for such low values, some distortion is expected.

As for the  $\tau_{yz}$  stress distribution, the parabolic shape is obtained. Some distortion is achieved by the FEM, however, the general expected shape is achieved. The RPIM

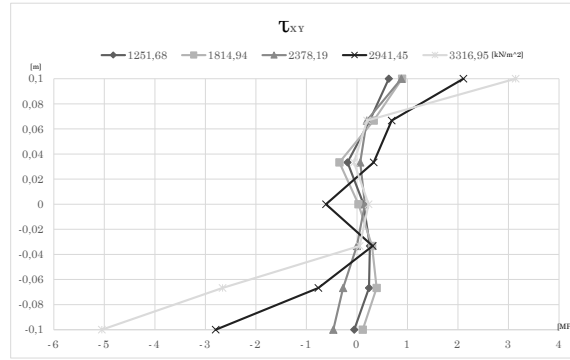
distribution, although closer to a parabola than the FEM, still has some distortion. The values between them are very close.

For the NNRPIM, although the existence of the value disparity between methods is still present, the solution is the smoothest of the three numerical methods.

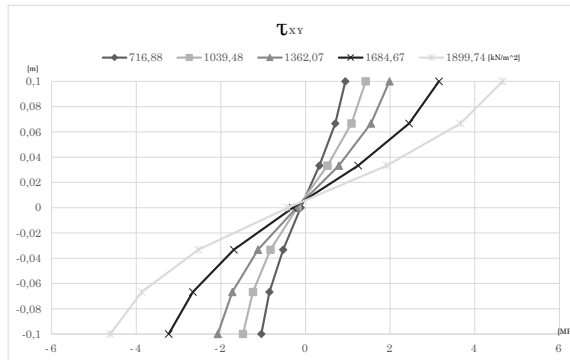
### 7.2.5.3 $\tau_{xy}$ stress diagrams along section C-C'



(a)  $\tau_{xy}$  stress for the FEM.



(b)  $\tau_{xy}$  stress for the RPIM.

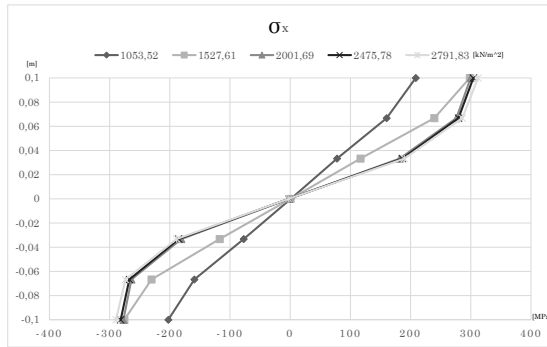


(c)  $\tau_{xy}$  stress for the NNRPIM.

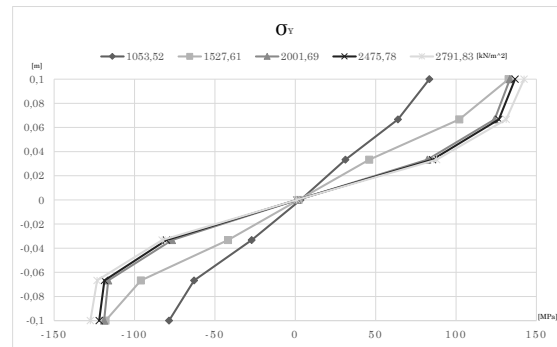
**Figure 7.22:**  $\tau_{xy}$  stress diagrams along section C-C'.

For the C-C' section, as it happens to the  $\tau_{xy}$  stress in the B-B' section, the  $\tau_{xy}$  stress should be, and in Figure 7.22 is, close to zero. Once again, for very low stress values, some stress distribution distortion is presented by the numerical methods.

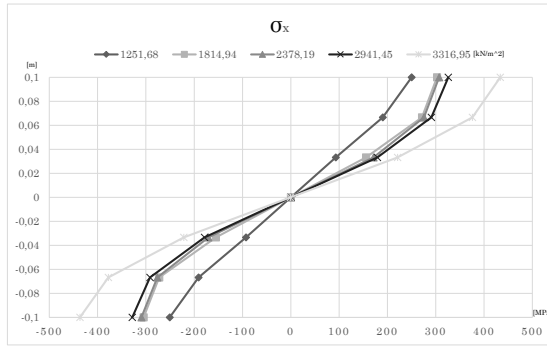
### 7.2.5.4 $\sigma_x$ , $\sigma_y$ , $\tau_{xz}$ , $\tau_{yz}$ stress diagrams along section D-D'



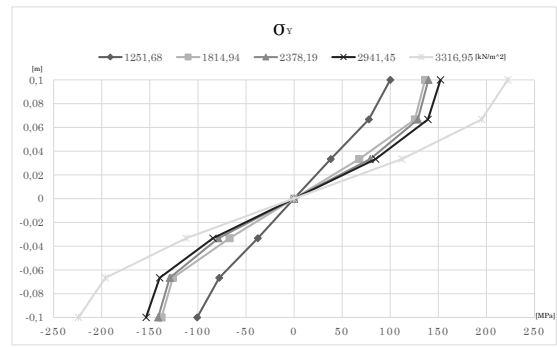
(a)  $\sigma_x$  stress for the FEM



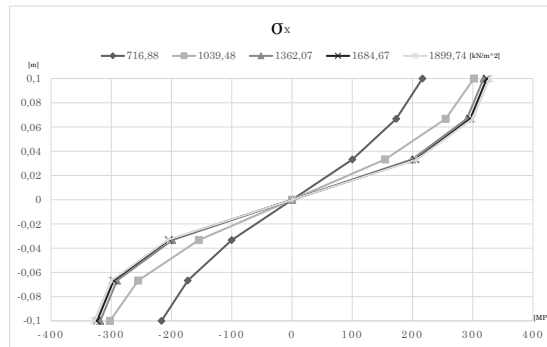
(b)  $\sigma_y$  stress for the FEM



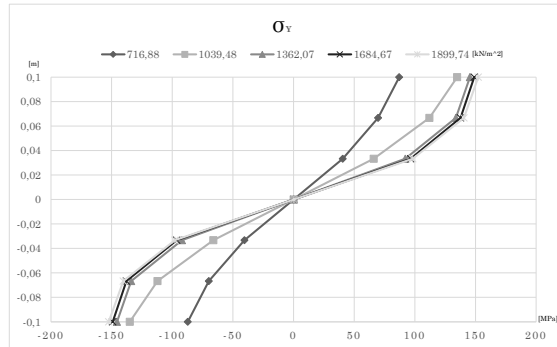
(c)  $\sigma_x$  stress for the RPIM



(d)  $\sigma_y$  stress for the RPIM

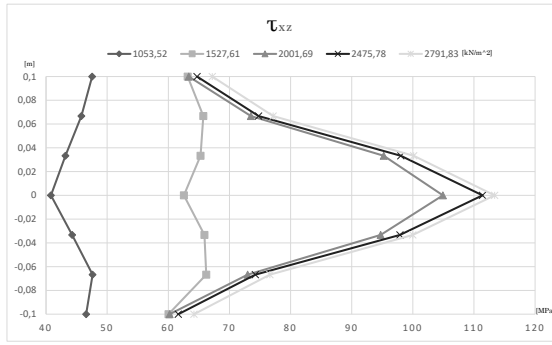
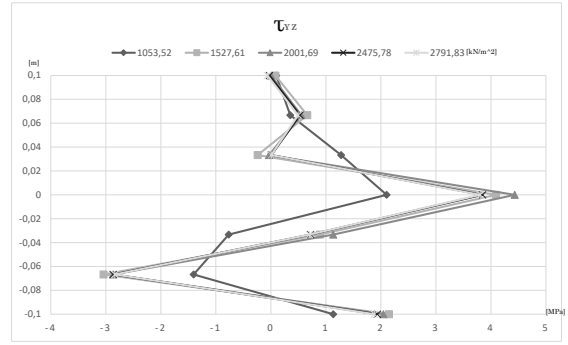
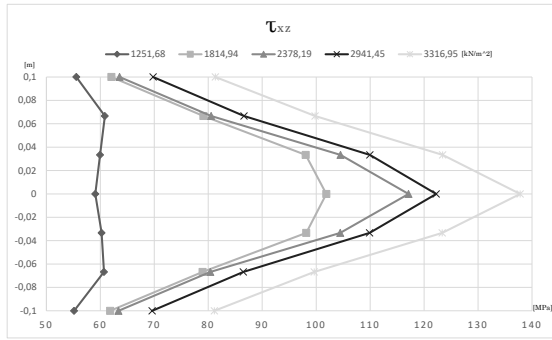
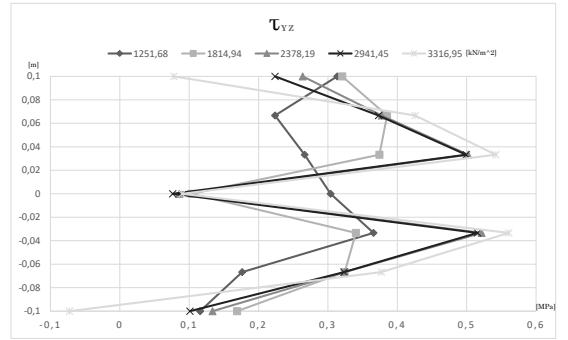
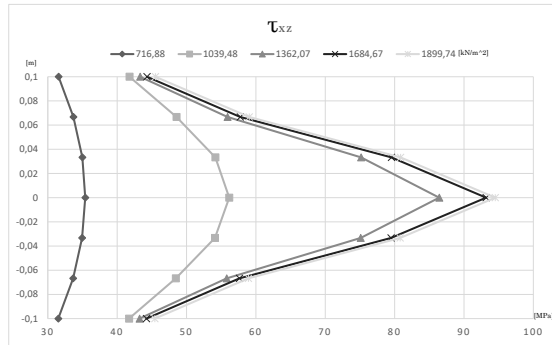
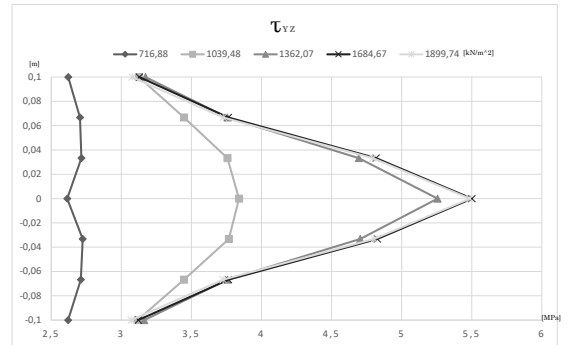


(e)  $\sigma_x$  stress for the NNRPIM



(f)  $\sigma_y$  stress for the NNRPIM

**Figure 7.23:**  $\sigma_x$  and  $\sigma_y$  stress diagrams along section D-D'

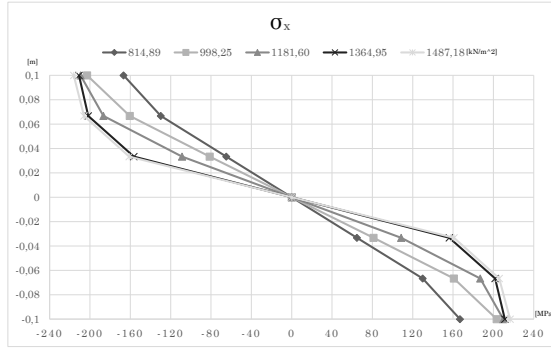
(a)  $\tau_{xz}$  stress for the FEM(b)  $\tau_{yz}$  stress for the FEM(c)  $\tau_{xz}$  stress for the RPIM(d)  $\tau_{yz}$  stress for the RPIM(e)  $\tau_{xz}$  stress for the NNRPIM(f)  $\tau_{yz}$  stress for the NNRPIM**Figure 7.24:**  $\tau_{xz}$  and  $\tau_{yz}$  stress diagrams along section D-D'

For both Figure 7.23 and 7.24, good agreement between numerical values is achieved. Again,  $\tau_{yz}$  close to zero, as expected, and some distortion. Parabolic shape obtained for the  $\tau_{xz}$ ,  $\sigma_x$  and  $\sigma_y$  stress distribution very similar between the three numerical values.

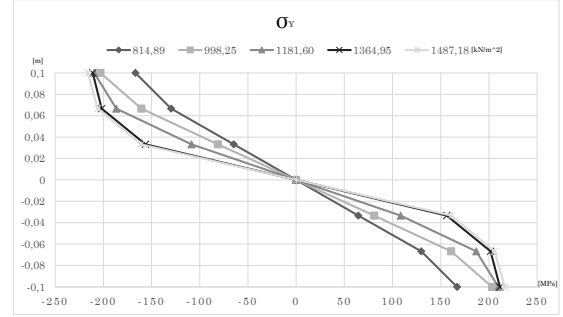
### 7.2.6 Stress Diagrams Along The Simply Supported Plate Thickness

In this section, the stress diagrams for the simply supported plate along its thickness will be presented. The units used are the same as the clamped plate.

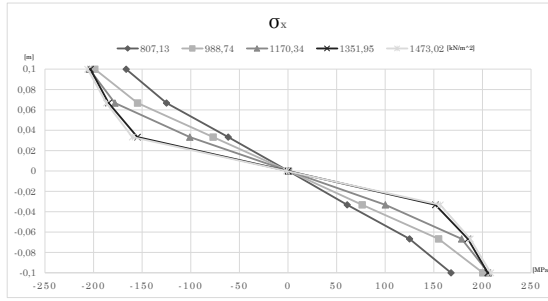
### 7.2.6.1 $\sigma_x$ , $\sigma_y$ , $\tau_{xz}$ , $\tau_{yz}$ stress diagrams along section A-A'



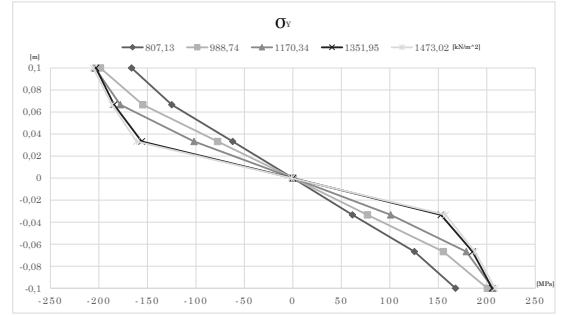
(a)  $\sigma_x$  stress for the FEM



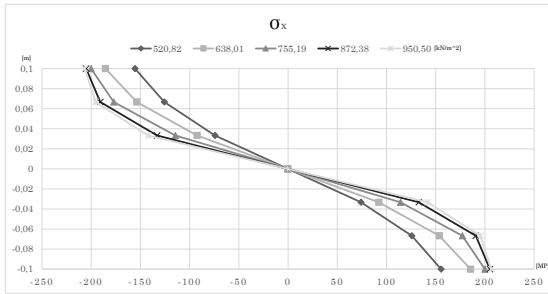
(b)  $\sigma_y$  stress for the FEM



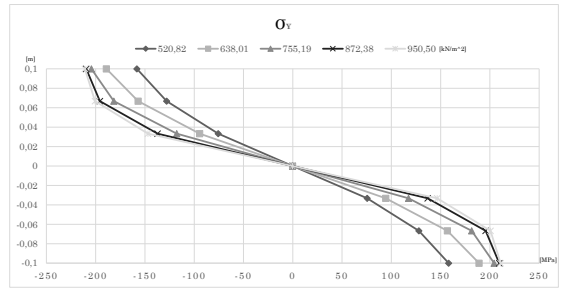
(c)  $\sigma_x$  stress for the RPIM



(d)  $\sigma_y$  stress for the RPIM

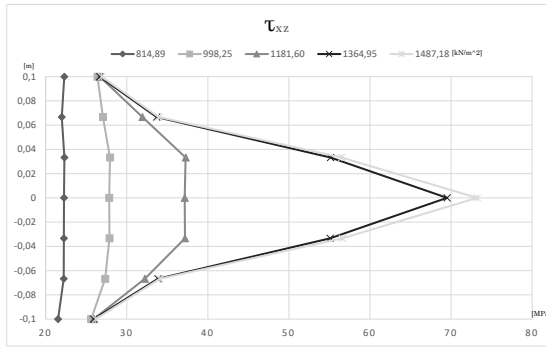
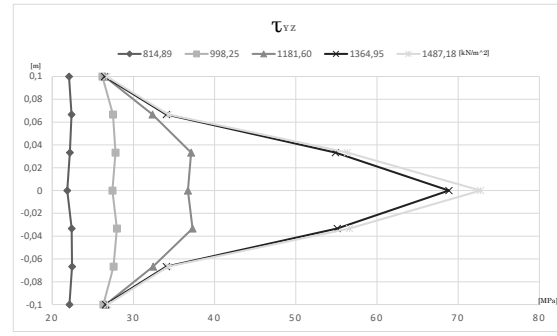
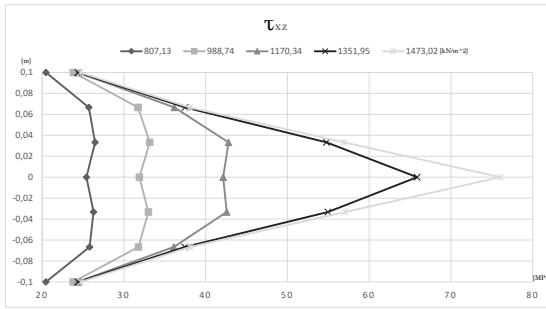
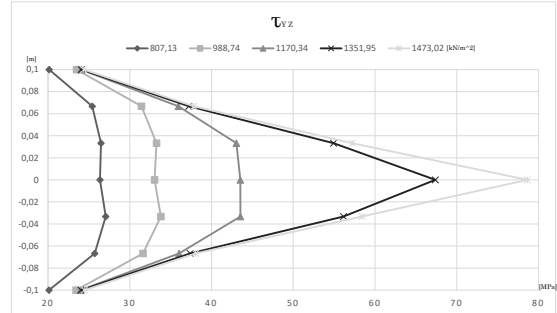
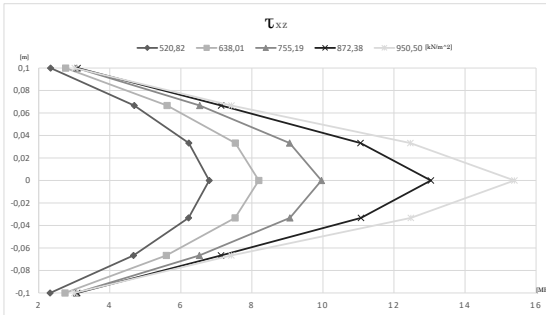
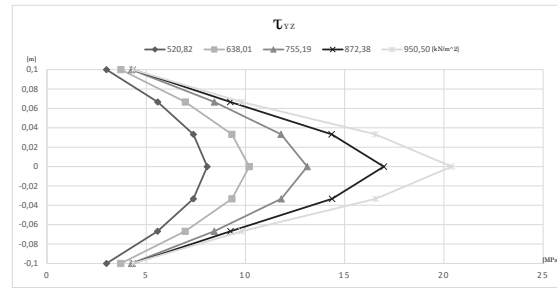


(e)  $\sigma_x$  stress for the NNRPIM



(f)  $\sigma_y$  stress for the NNRPIM

**Figure 7.25:**  $\sigma_x$  and  $\sigma_y$  stress diagrams along section A-A'

(a)  $\tau_{xz}$  stress for the FEM(b)  $\tau_{yz}$  stress for the FEM(c)  $\tau_{xz}$  stress for the RPIM(d)  $\tau_{yz}$  stress for the RPIM(e)  $\tau_{xz}$  stress for the NNRPIM(f)  $\tau_{yz}$  stress for the NNRPIM**Figure 7.26:**  $\tau_{xz}$  and  $\tau_{yz}$  stress diagrams along section A-A'

As for Figure 7.25 and 7.26, the same tendency and behaviour of the former diagrams is presented. The progressive evolution of the stresses,  $\sigma_x$  and  $\sigma_y$ , is clearly visible. Also, the evolution into a parabola for the  $\tau_{xz}$  and  $\tau_{yz}$  stresses is also clearly denoted.

### 7.2.7 Effective Stress Distribution Maps

In this section, the effective stress distribution maps for both plate configurations will be presented. All maps are obtained for the three numerical methods and using the KT-ALL algorithm.

As for units, the stress is measured in  $Pa$  while the load is in  $\frac{kN}{m^2}$ .

### 7.2.7.1 Effective Stress Distribution Maps For The Clamped Plate

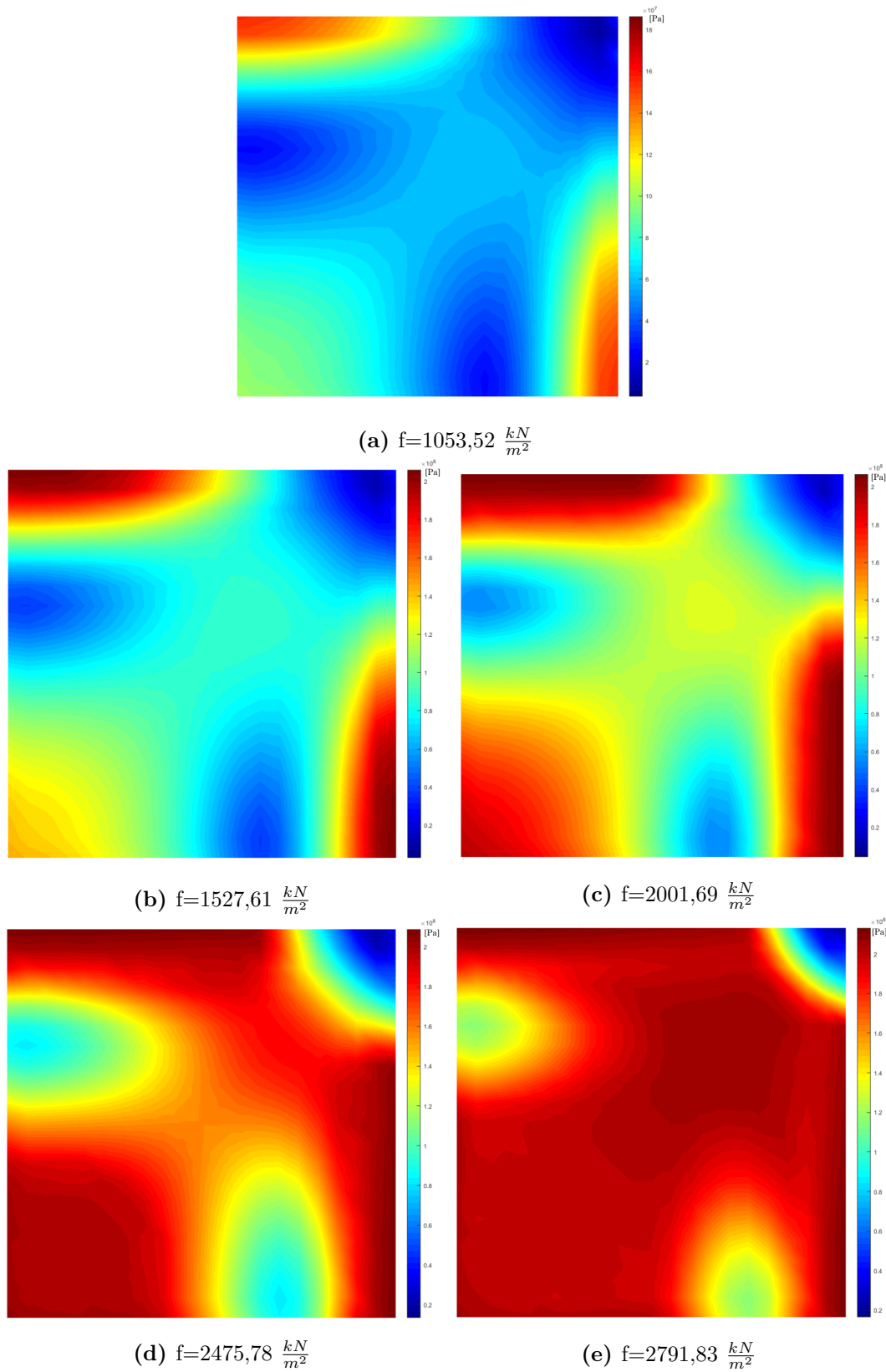
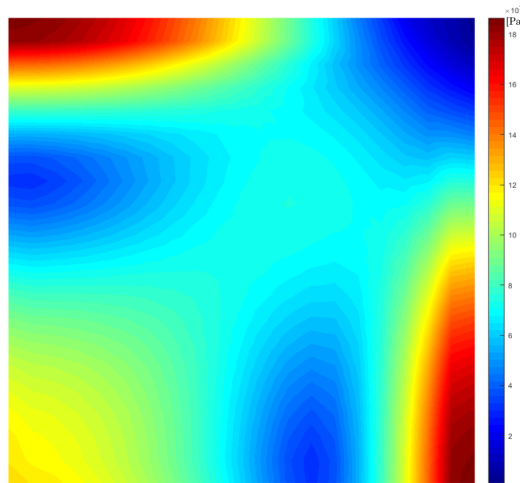
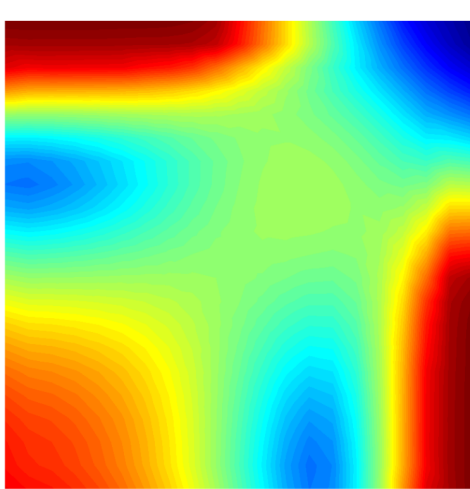


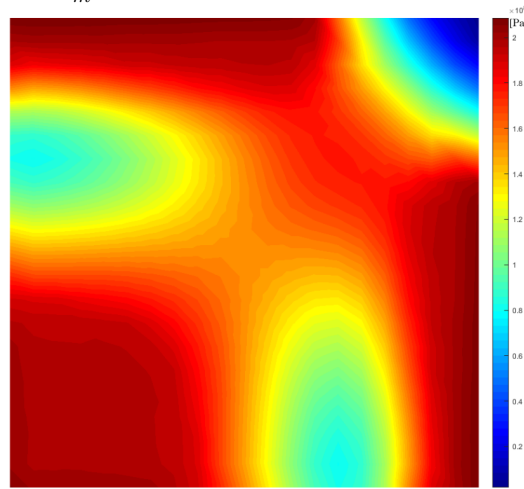
**Figure 7.27:** Effective stress distribution maps obtained by the FEM.



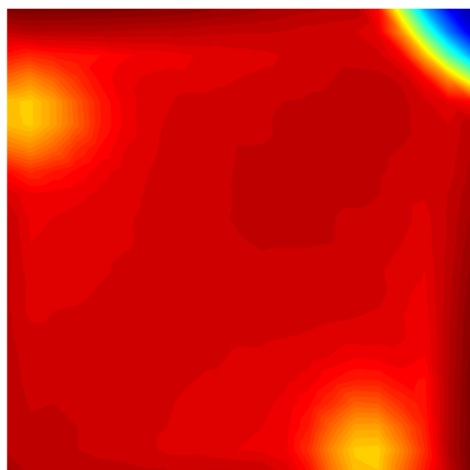
(a)  $f=1251,68 \frac{kN}{m^2}$



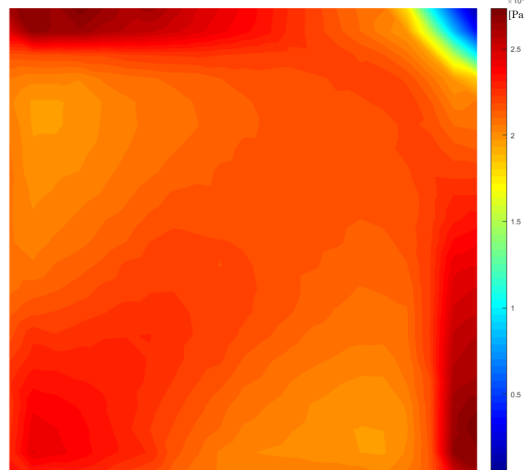
(b)  $f=1814,94 \frac{kN}{m^2}$



(c)  $f=2378,19 \frac{kN}{m^2}$



(d)  $f=2941,45 \frac{kN}{m^2}$



(e)  $f=3316,95 \frac{kN}{m^2}$

**Figure 7.28:** Effective stress distribution maps obtained by the RPIM.

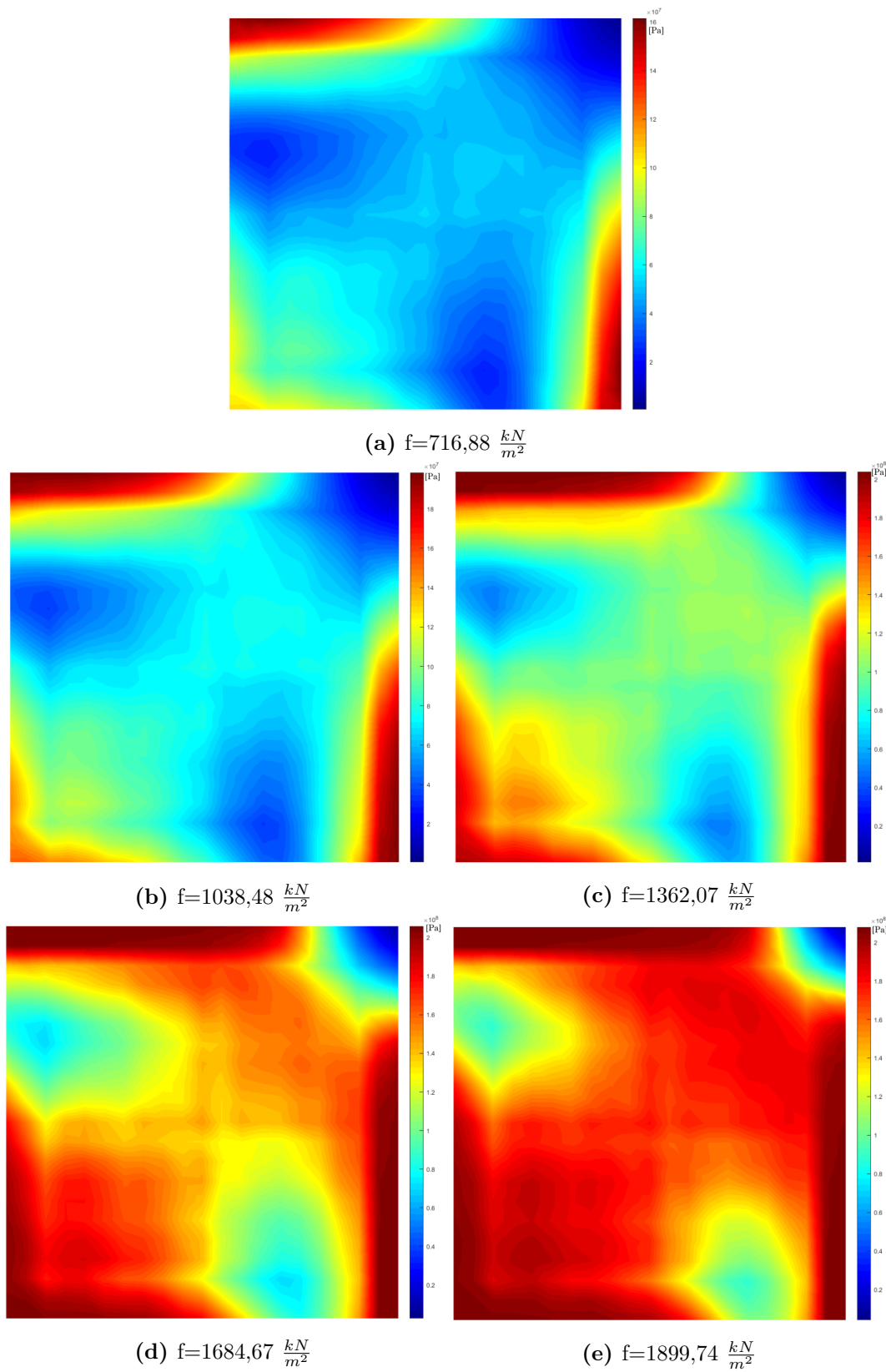


**Figure 7.29:** Effective stress distribution maps obtained by the NNRPIM.

As for the effective stress maps present in Figure 7.27, 7.28 and 7.29, for all three methods, they are quite similar. As expected, for the clamped plate, plasticity first occurs on the upper left and lower right areas of the plate. As the load increases, material plasticity evolves towards the center of the plate.

Both FEM and RPIM stress distribution are quite smooth while the NNRPIM solution is more coarse. This is due to the plot function of the program being used, which uses triangular shapes to represent the stress distribution along the plate surface, that, as a consequence, results in a lower smoothness solution.

Nevertheless, it is important to note the high proximity between solutions.

### 7.2.7.2 Effective Stress Distribution Maps For The Simply Supported Plate

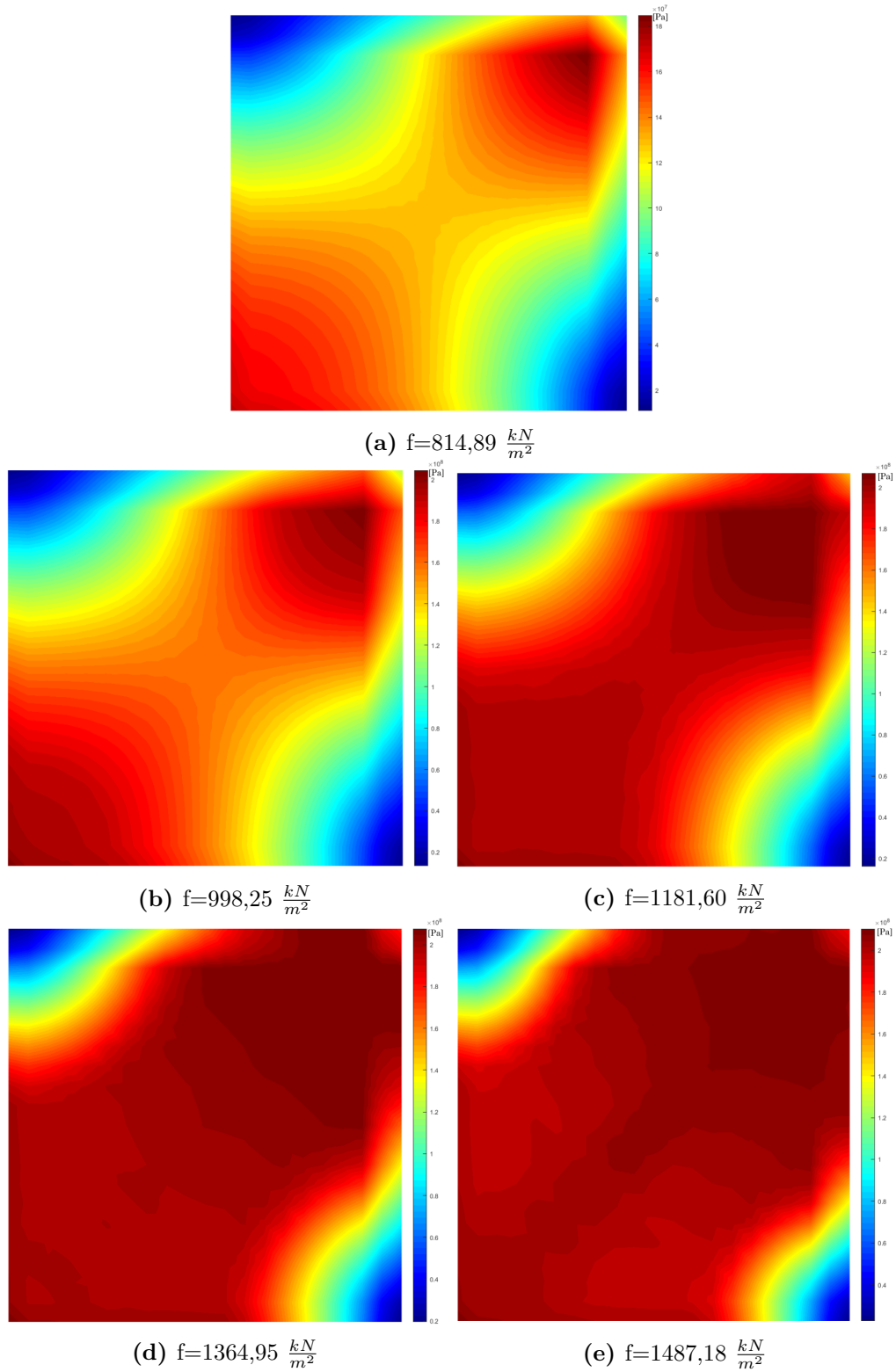


**Figure 7.30:** Effective stress distribution maps obtained by the FEM.

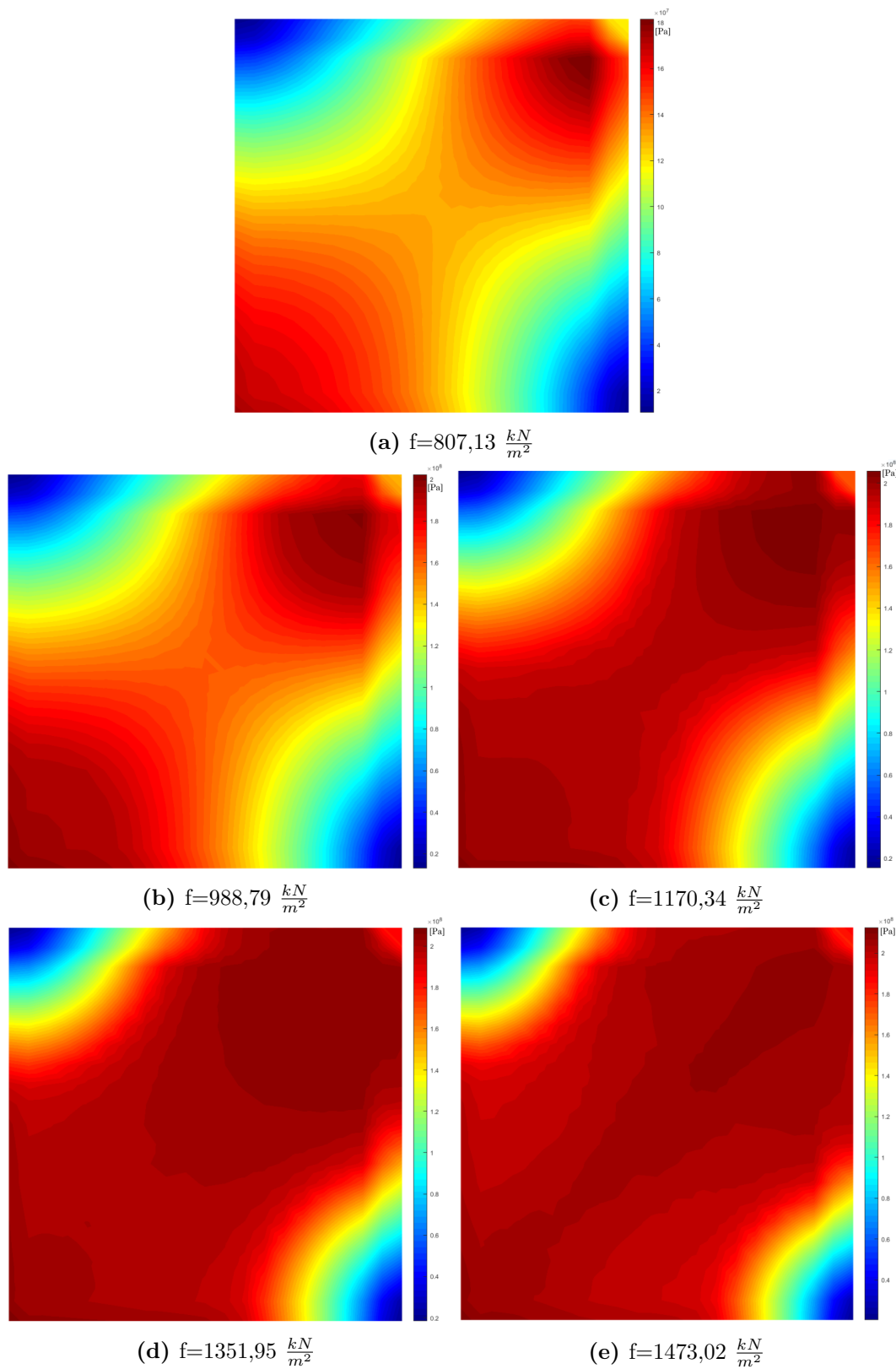


**Figure 7.31:** Effective stress distribution maps obtained by the RPIM.

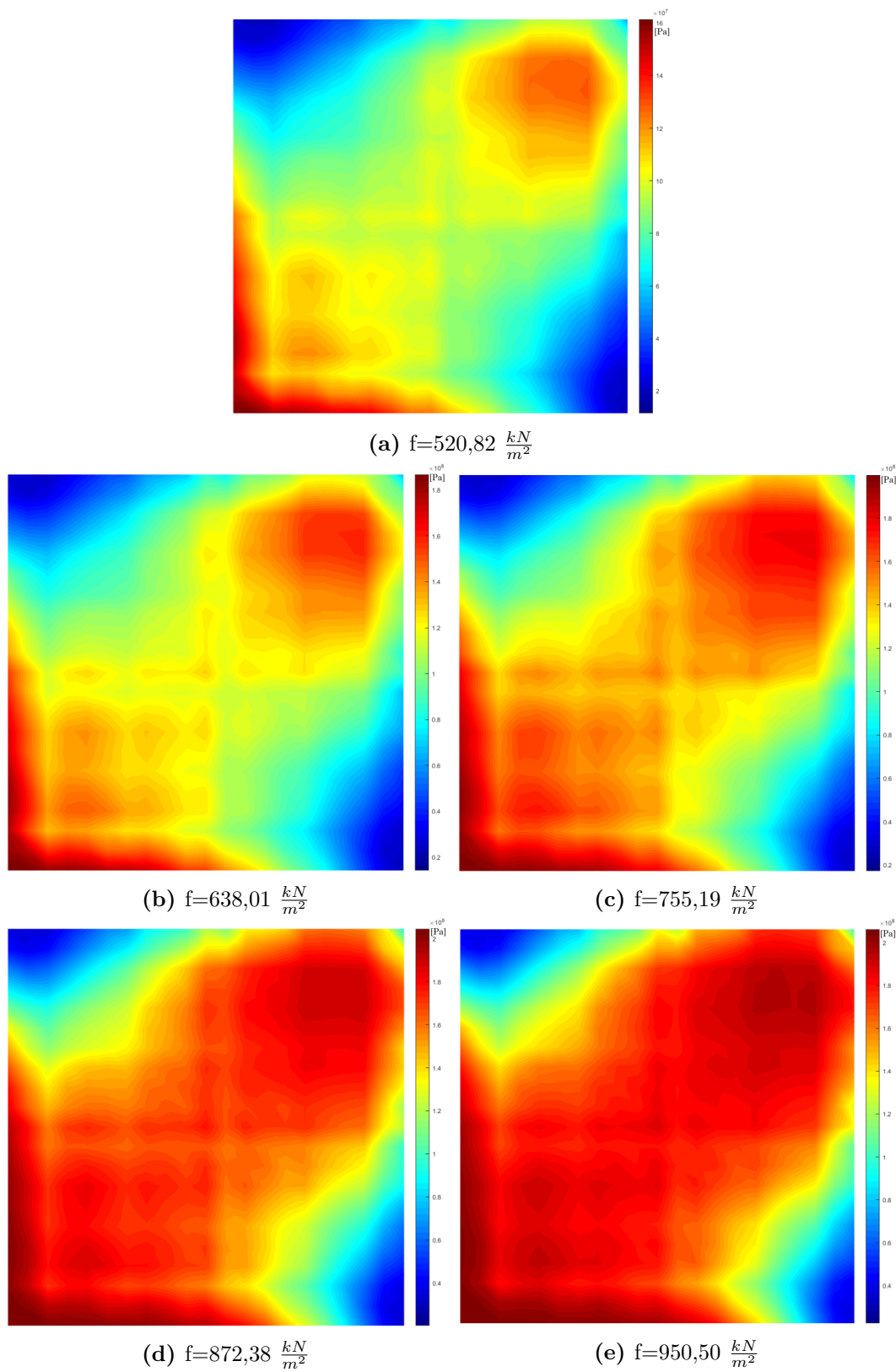


**Figure 7.32:** Effective stress distribution maps obtained by the NNRPIM.

For the simply supported plate, once again the stress distribution for the three numerical methods, Figure 7.30, 7.31 and 7.32, are quite close to each other. As expected, plasticity occurs first at the center and upper right areas of the plate, and as the load increases, plasticity spreads towards the center of the plate.

The NNRPIM solution shows the same issue as the clamped plate solution did. Nevertheless, the stress distribution map is clearly similar to the ones obtained by the FEM and RPIM.

## 7.3 Elastoplastic Analysis Of 2D Frames

In this section, the elastoplastic analysis of selected 2D frames will be presented.

The FEM, RPIM and NNRPIM were used, combined with the KT-ALL non-linear solution algorithm.

### 7.3.1 Two Bay Asymmetric Frame

The selected frame analyzed is based on the research paper by Argyris et al [23].

For this example, a nodal discretization of 6063 nodes was used for the FEM, RPIM and NNRPIM.

#### 7.3.1.1 Geometry, Boundary and Load Conditions

The geometry and dimensions of the two bay asymmetric frame considered are represented in Figure 7.33 and 7.34.

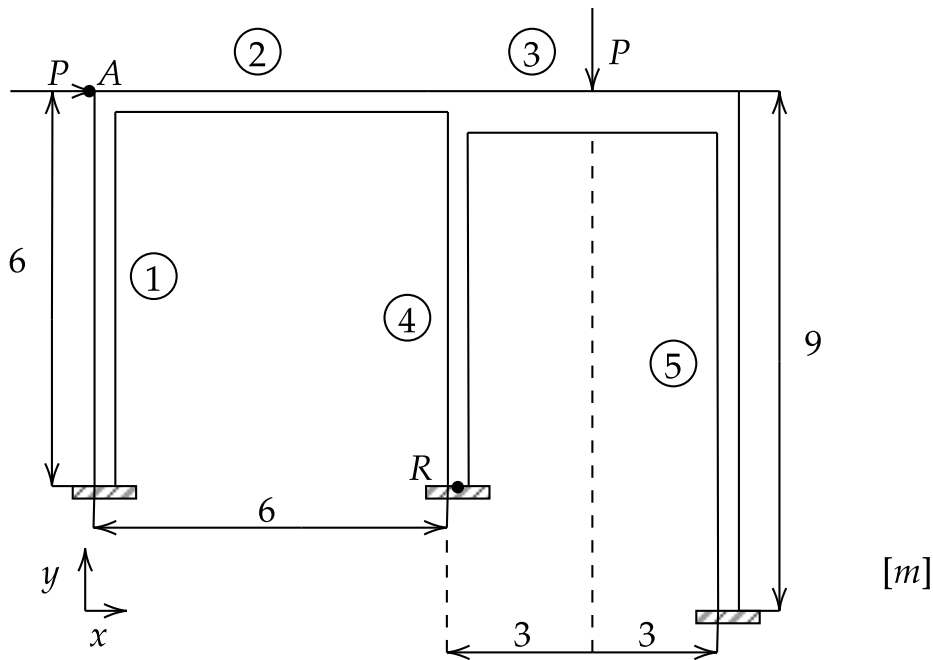


**Figure 7.33:** Geometry, boundary and load conditions of the two bay asymmetric frame.

The base of element 1 and 5 are completely blocked in all degrees of freedom ( $u$  and  $v$ ), while a connection at the base of element 4 is considered ( $R$ ), allowing rotation of the element, with the displacements being blocked as well.

The length of the elements considered are shown in Figure 7.33, while the cross-section dimensions are shown in Figure 7.34.

#### 7.3.1.2 Mechanical Properties

The mechanical properties of the material are as follows,

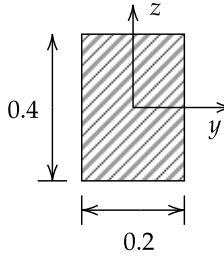
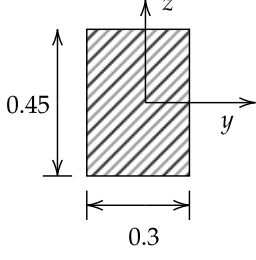
| Element No. | Cross Section Dimensions [m]                                                       |
|-------------|------------------------------------------------------------------------------------|
| 1,2,4,5     |  |
| 3           |  |

**Figure 7.34:** Cross-section dimensions.

| Mechanical Properties | Value       |
|-----------------------|-------------|
| $E$ [Pa]              | 19613,3E+06 |
| $E_T$ [Pa]            | 19613,3E+02 |
| $\nu$                 | 0,3         |
| $\sigma_Y$ [Pa]       | 98,0665E+06 |

**Table 7.10:** Mechanical properties of the two bay asymmetric frame.

### 7.3.1.3 Horizontal Displacement of Point A

The horizontal displacement graphs of point **A**, presented in Figure 7.33 are now presented. As for units, the  $x$  axis is normalized by the relationship  $\frac{u_A}{6}$ , and the  $y$  axis is correspondent to the load in  $\frac{\text{kilopound}}{10}$ .

As it can be seen in Figure 7.35, 7.36 and 7.37, the results obtained coincide well with the reference solution for all three numerical methods. As for the marks labeled with respective numbers, these concern the plastic hinges developed as the load increases. These plastic hinges are represented in Figure 7.38.

### 7.3.1.4 Effective Plastic Strain Distribution Maps - Development Of The Plastic Hinges

The effective plastic strain distribution maps for the FEM, RPIM and NNRPIM are presented in Figure 7.39, 7.40 and 7.41.

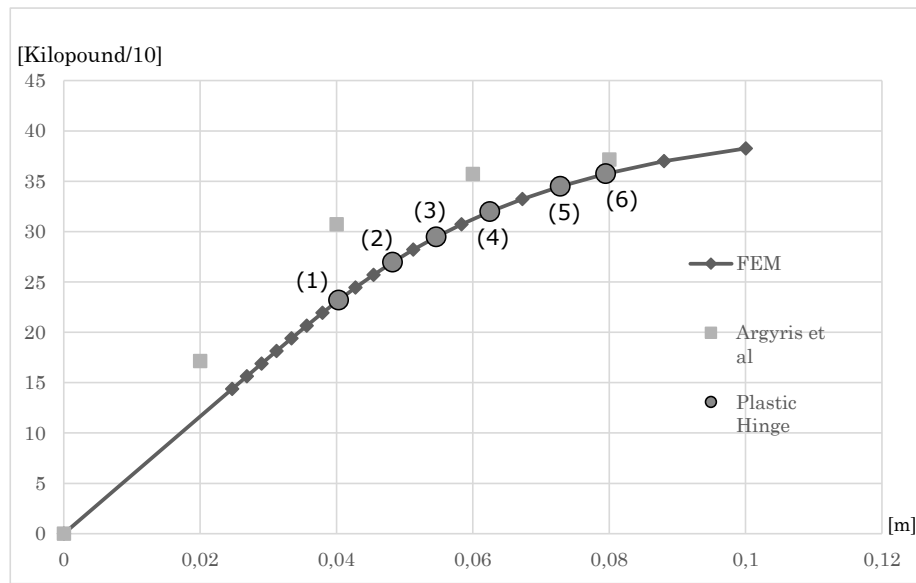


**Figure 7.35:** Horizontal displacement of point **A** given by the FEM

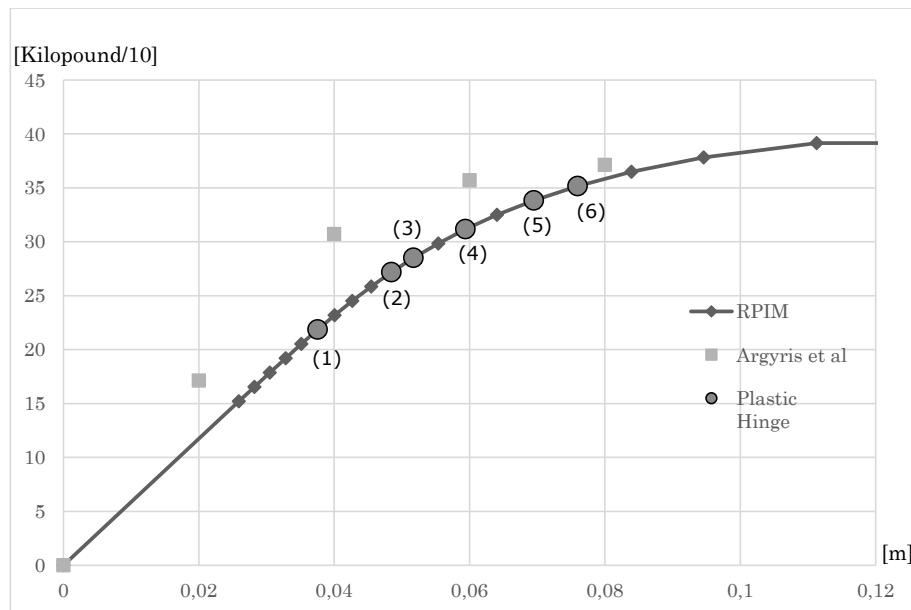


**Figure 7.36:** Horizontal displacement of point **A** given by the RPIM

As it can be seen, as the load increases, the amount of effective plastic strain at a certain point of the structure increases, creating the plastic connections or plastic hinges. These hinges are created due to the plastification of certain points in the structure that are subjected to high stress considering the load input. This way, it is possible to identify the critical points of the structure.

Comparing Figure 7.39, 7.40 and 7.41, the three methods return the same critical points, with the same development of the plastic hinges between them.

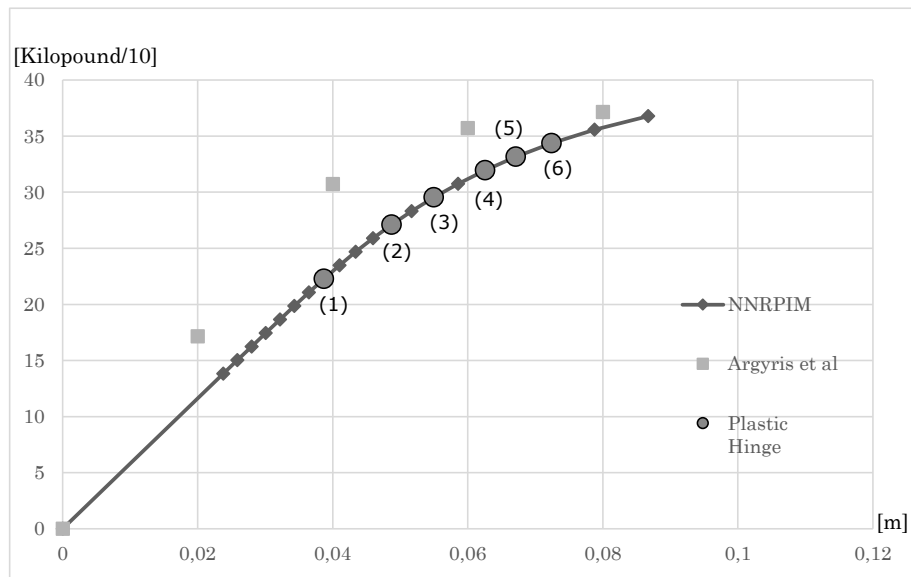


**Figure 7.37:** Horizontal displacement of point A given by the NNRPIM

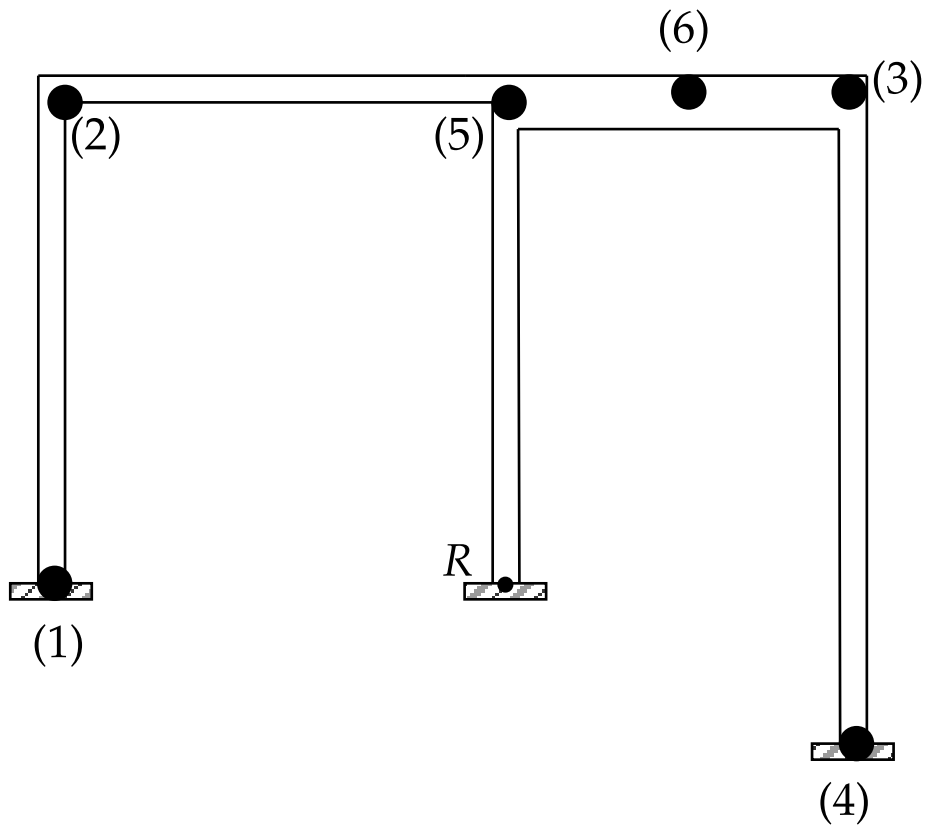


**Figure 7.38:** Plastic hinges in chronological order.

### 7.3.1.5 Evolution Of The Effective Stress On The Plastic Hinges

The diagrams showing the evolution of the effective stress on the identified plastic hinges are presented in Figure 7.42, 7.43 and 7.44.

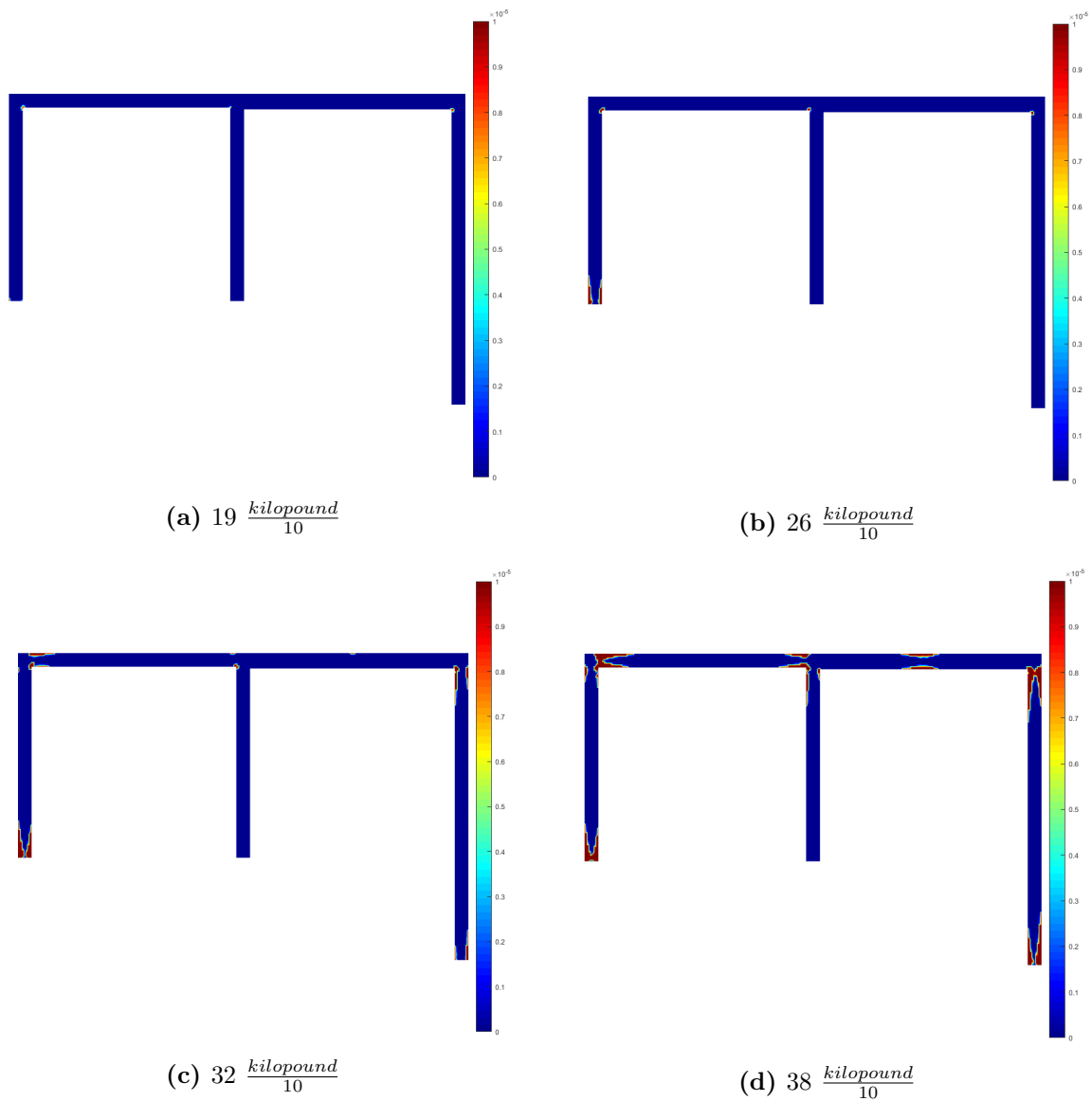


**Figure 7.39:** Development of the plastic hinges for the FEM

As it can be seen in Figure 7.42, 7.43 and 7.44, for all three numerical methods used, the evolution of the effective stress is quite similar.

An interesting detail to be analyzed in the evolution of the 1<sup>st</sup> and 2<sup>nd</sup> plastic hinge. In fact, the 2<sup>nd</sup> plastic hinge enters in plastic regime first, when compared to the 1<sup>st</sup> plastic hinge. When compared with Figure 7.39, 7.40 and 7.41, in terms of effective plastic strain, both data contradict.

However, in truth, the 1<sup>st</sup> plastic hinge has a very rapid transition between elastic and plastic behaviour, while the 2<sup>nd</sup> plastic hinge is slower in the same transition, as it can be assessed by the curvature of both plastic hinges in the diagrams exposed.

Another important aspect about the diagrams presented is the stabilization of the stress level at a certain load applied. After a certain load is applied, the effective stress maintains its value, as it should in complete plastic regime. We can clearly see the

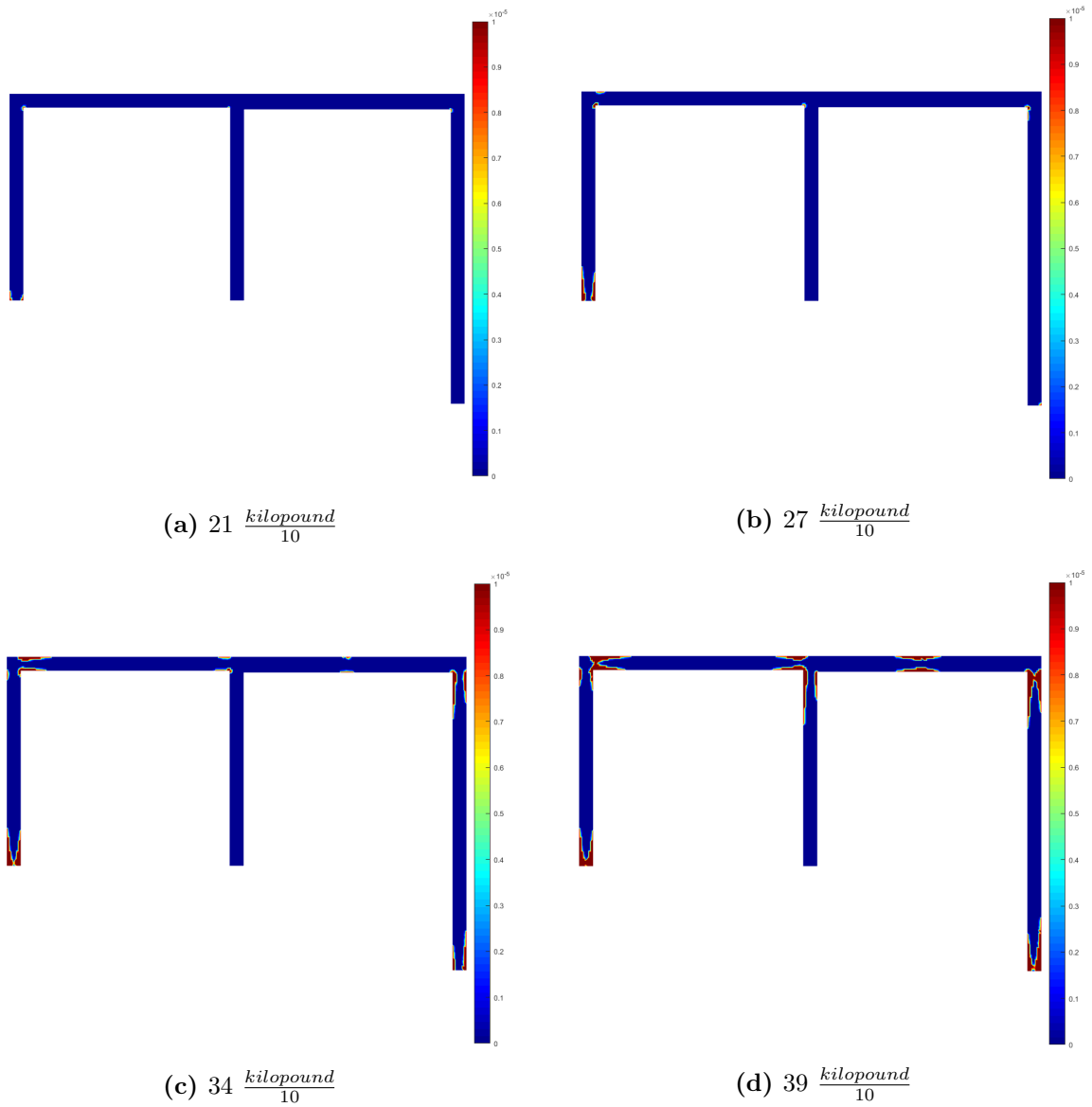


**Figure 7.40:** Development of the plastic hinges for the RPIM

evolution and transition from linear elastic regime to plastic regime, ending with the stabilization of the effective stress, even when the load is increased.

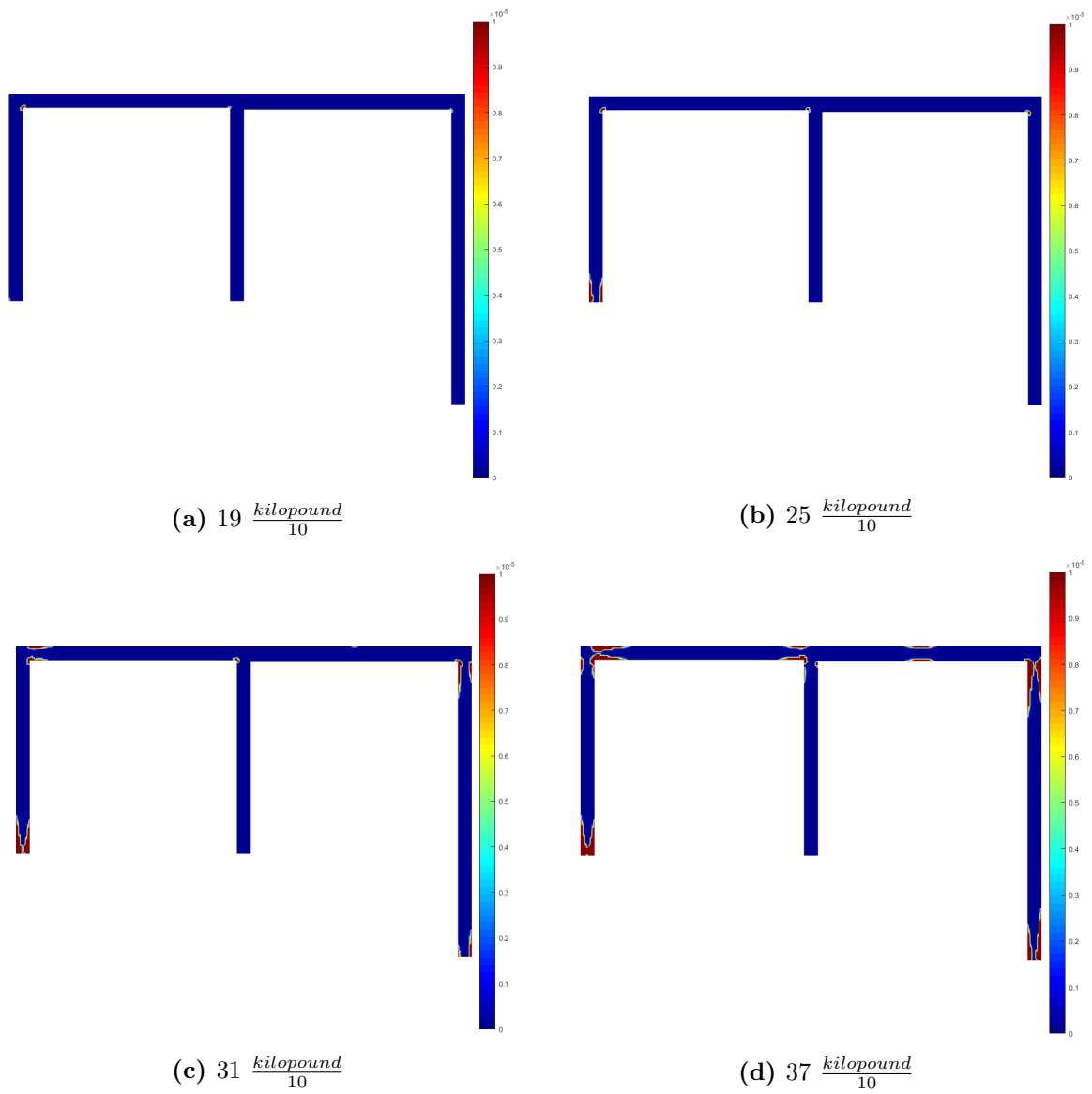


**Figure 7.41:** Development of the plastic hinges for the NNRPIM

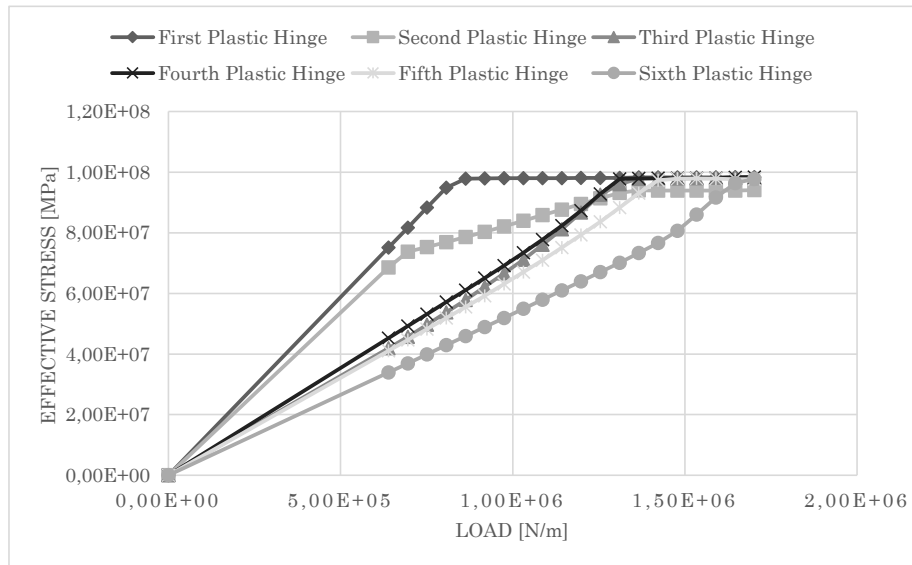


**Figure 7.42:** Evolution of the effective stress on the plastic hinges given by the FEM

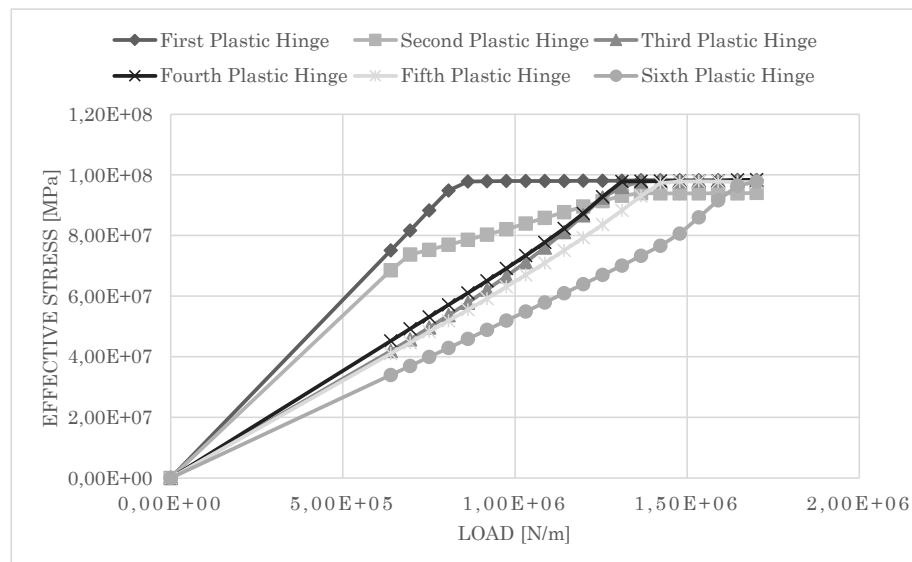


**Figure 7.43:** Evolution of the effective stress on the plastic hinges given by the RPIM

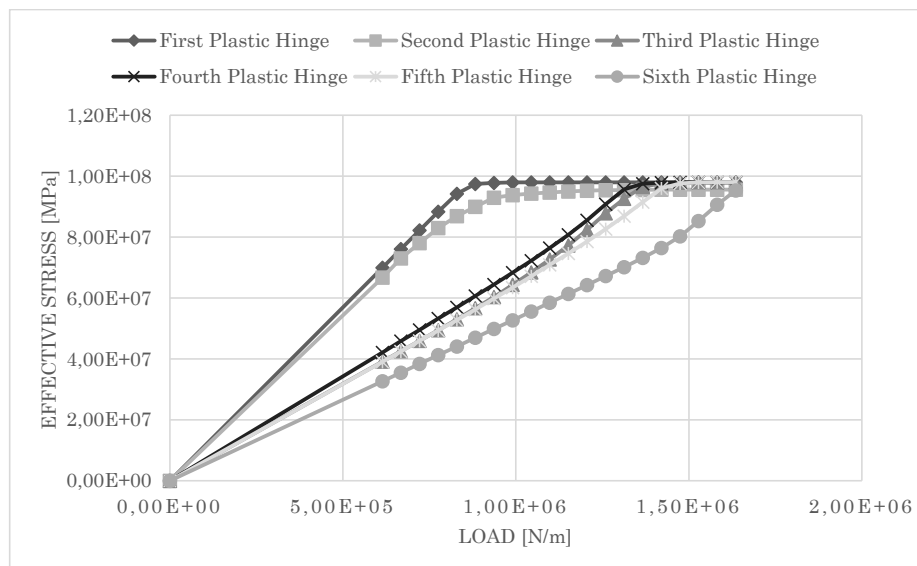


**Figure 7.44:** Evolution of the effective stress on the plastic hinges given by the NNR-PIM

### 7.3.2 Bowstring Bridge

This example was developed by the author. The purpose of this specific problem is the elastoplastic analysis of a commonly used structure in the industry.

For the 2D model, the structure was analyzed with a nodal density of 6024 nodes, using the FEM, RPIM and NNRPIM,, combined with the KT-ALL non-linear solution algorithm.

#### 7.3.2.1 Geometry, Boundary and Load Conditions

Due to symmetry of the solid considered, only half of the total bridge will be analyzed.

The geometry and dimensions of the bowstring bridge considered are represented in Figure 7.45.

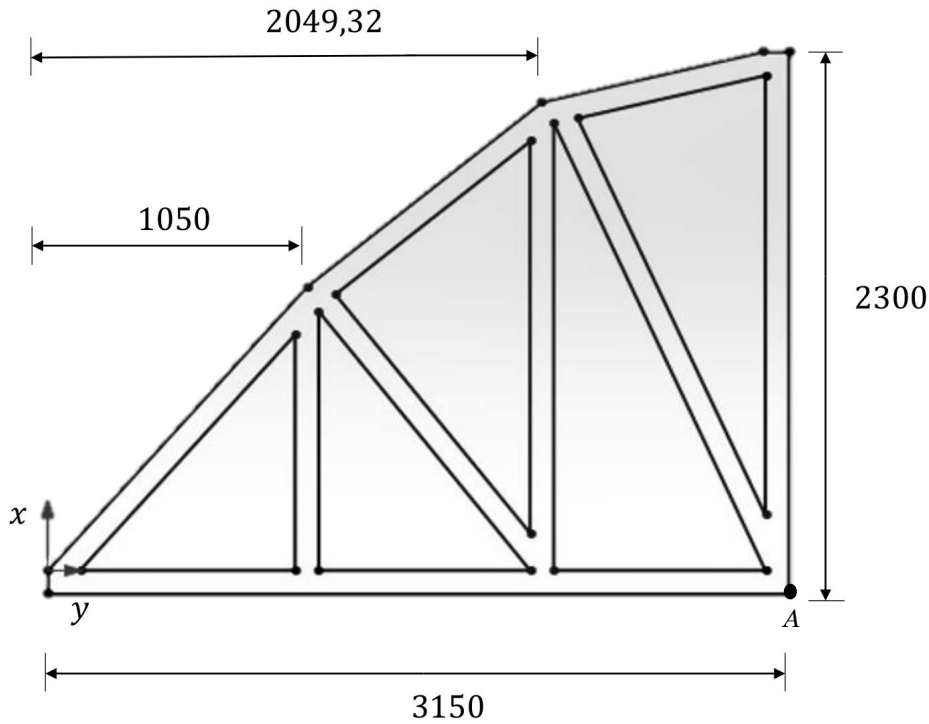


**Figure 7.45:** Dimensions of the bowstring bridge in [mm]

Every element present in the structure has a thickness of 100 [mm]

The boundary and load conditions are represented in Figure 7.46

#### 7.3.2.2 Mechanical Properties

The mechanical properties of the material, a common structural steel, are presented in Table 7.11.

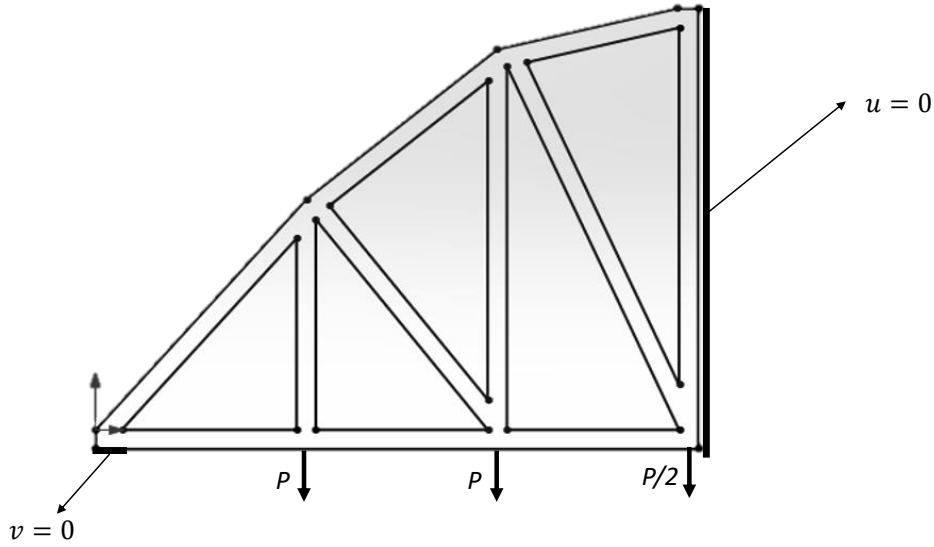


**Figure 7.46:** Boundary and load conditions of the bowstring bridge

| Mechanical Properties | Value  |
|-----------------------|--------|
| $E$ [MPa]             | 205000 |
| $E_T$ [MPa]           | 6700   |
| $\nu$                 | 0,3    |
| $\sigma_Y$ [MPa]      | 230    |

**Table 7.11:** Mechanical properties of the bowstring bridge

### 7.3.2.3 Vertical Displacement Of Point A

The vertical displacement of point **A**, represented in Figure 7.45, for all three numerical methods and for the KT-ALL non-linear algorithm are presented in Figure 7.47.

As it can be seen in Figure 7.47, the FEM and NNRPIM obtain almost equal results, being that the load/displacement curves for the two numerical methods are almost equal to each other. As for the RPIM curve, it follows the trend of the FEM and NNRPIM up to a certain point, it then deviates to a lower level when compared with the other two curves. Nevertheless, all three results are very similar to each other and a good agreement between them is verified.

### 7.3.2.4 Effective Plastic Strain Maps - Development

The effective plastic strain distribution maps for the FEM, RPIM and NNRPIM, are presented in Figure 7.48, 7.49 and 7.50.

Analyzing Figure 7.48, 7.49 and 7.50, we can clearly see the critical areas of the structure as the load increases. For all three methods, plastic strain begins in the same point, and propagates extensively along the upper arch of the structure. Such arch is under compressive load, while the lower beam is under traction load. As plastification ensues, the load applied to the bridge is mainly supported by this lower beam. The distribution of compressive and traction stress is due to the trusses included in the

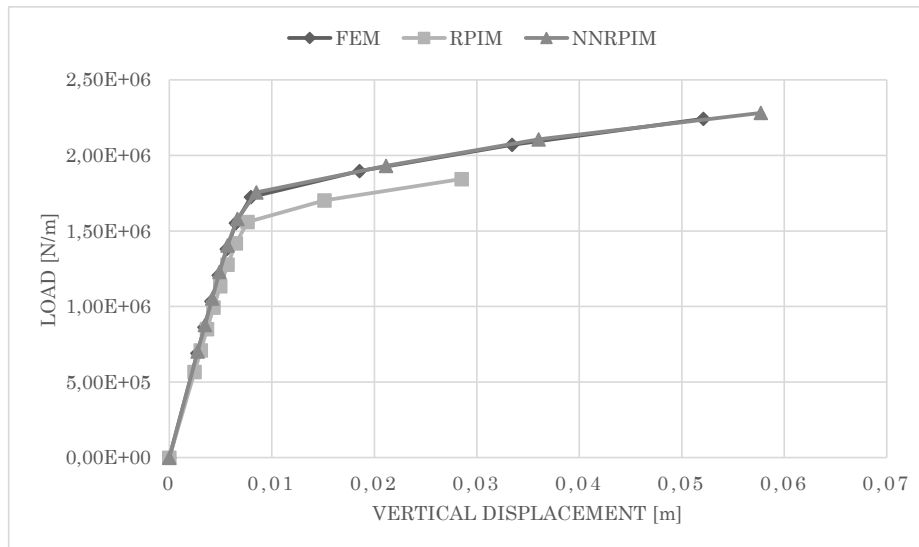


**Figure 7.47:** Vertical displacement of point **A** of the 2D bowstring bridge

design of the structure.

As for the comparison between the three numerical methods, the FEM and NNRPIM return almost equal results, which is to be expected given the almost equal load/displacement diagram shown in Figure 7.47. As for the RPIM, the plastification of the structure is not as broad and in the other two numerical methods, due to the fact that the load applied is not as great as in the FEM and NNRPIM, which limits the plastic strain.

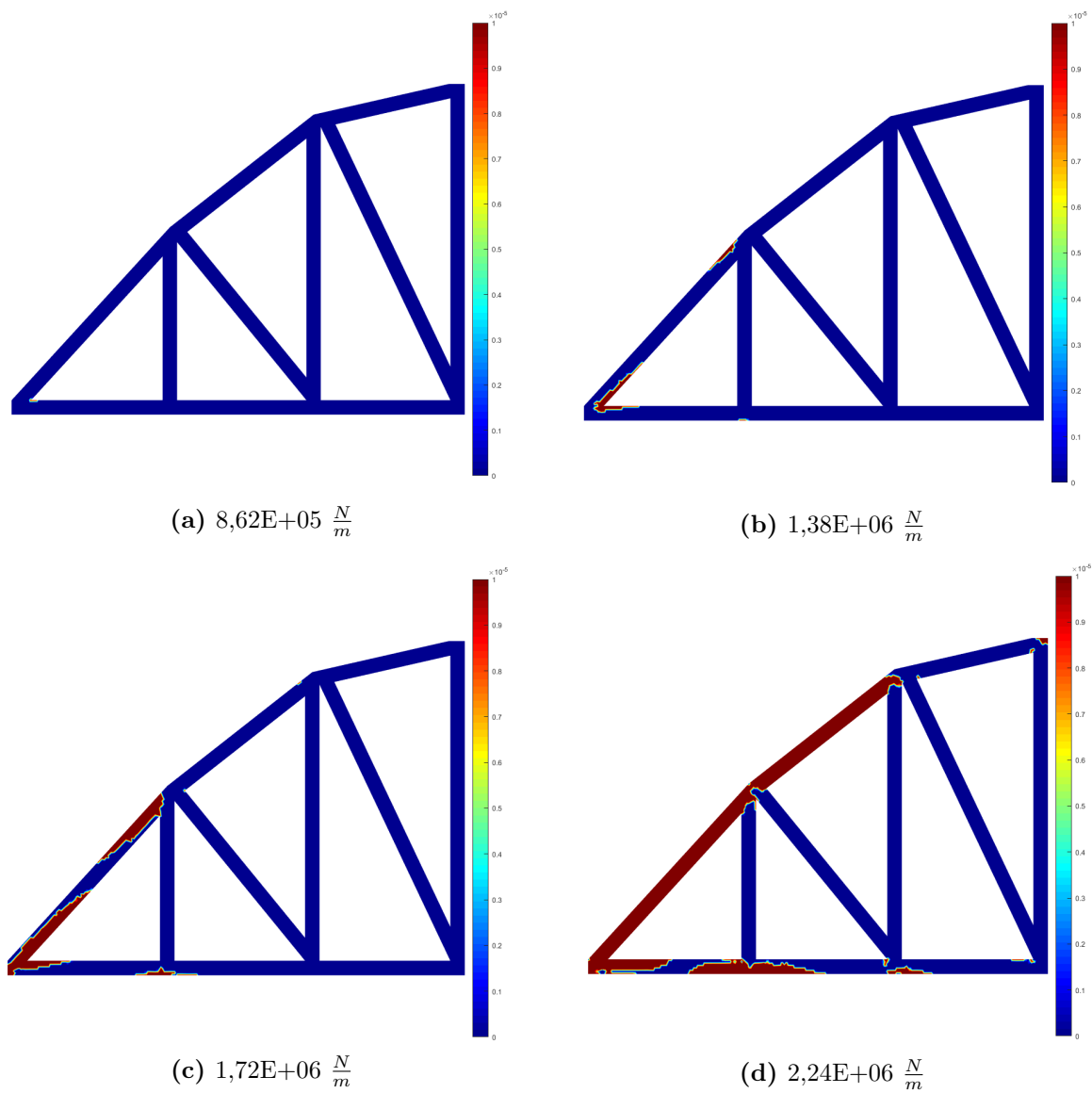


**Figure 7.48:** Development of the effective plastic strain for the FEM

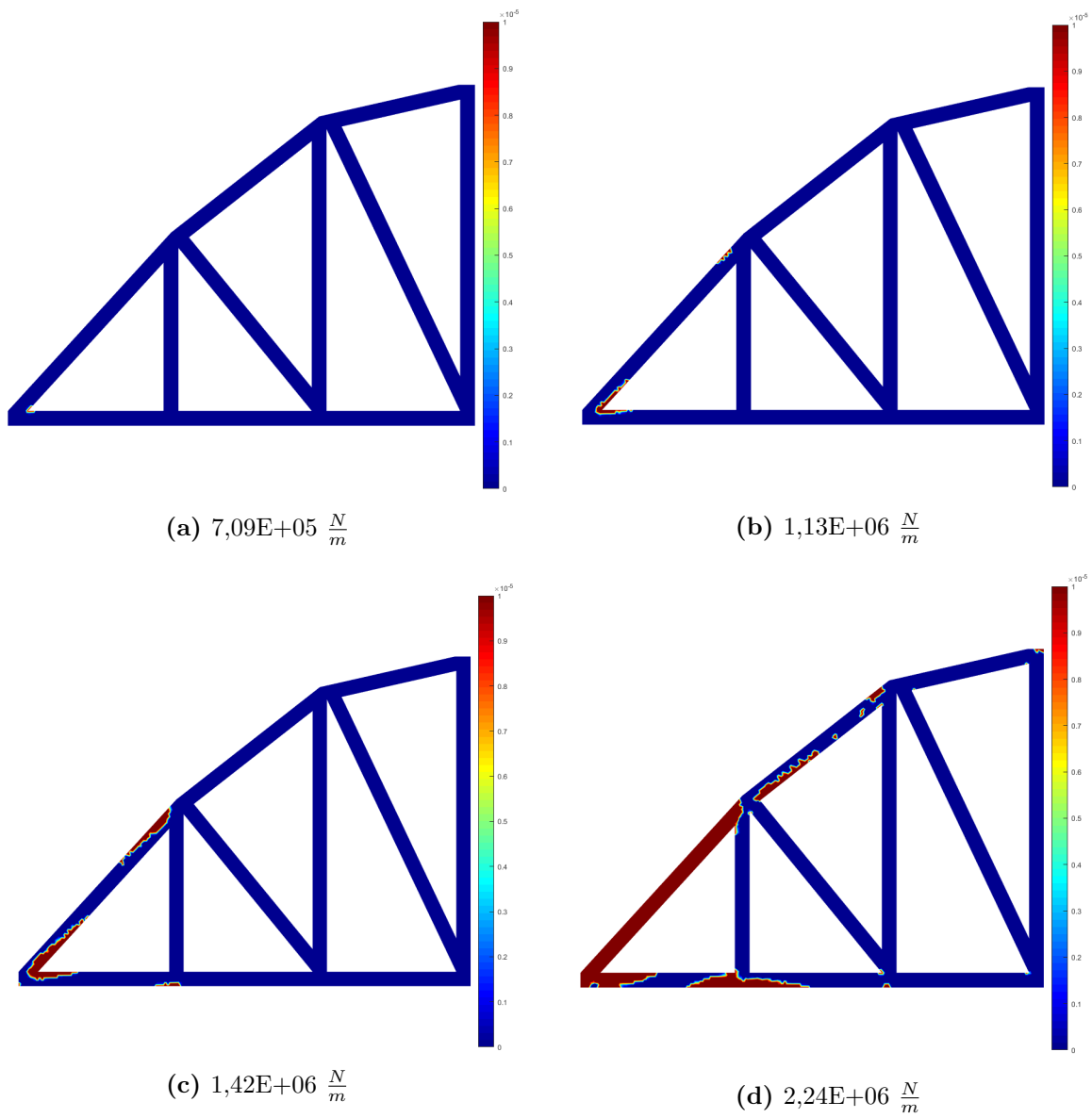


**Figure 7.49:** Development of the effective plastic strain for the RPIM

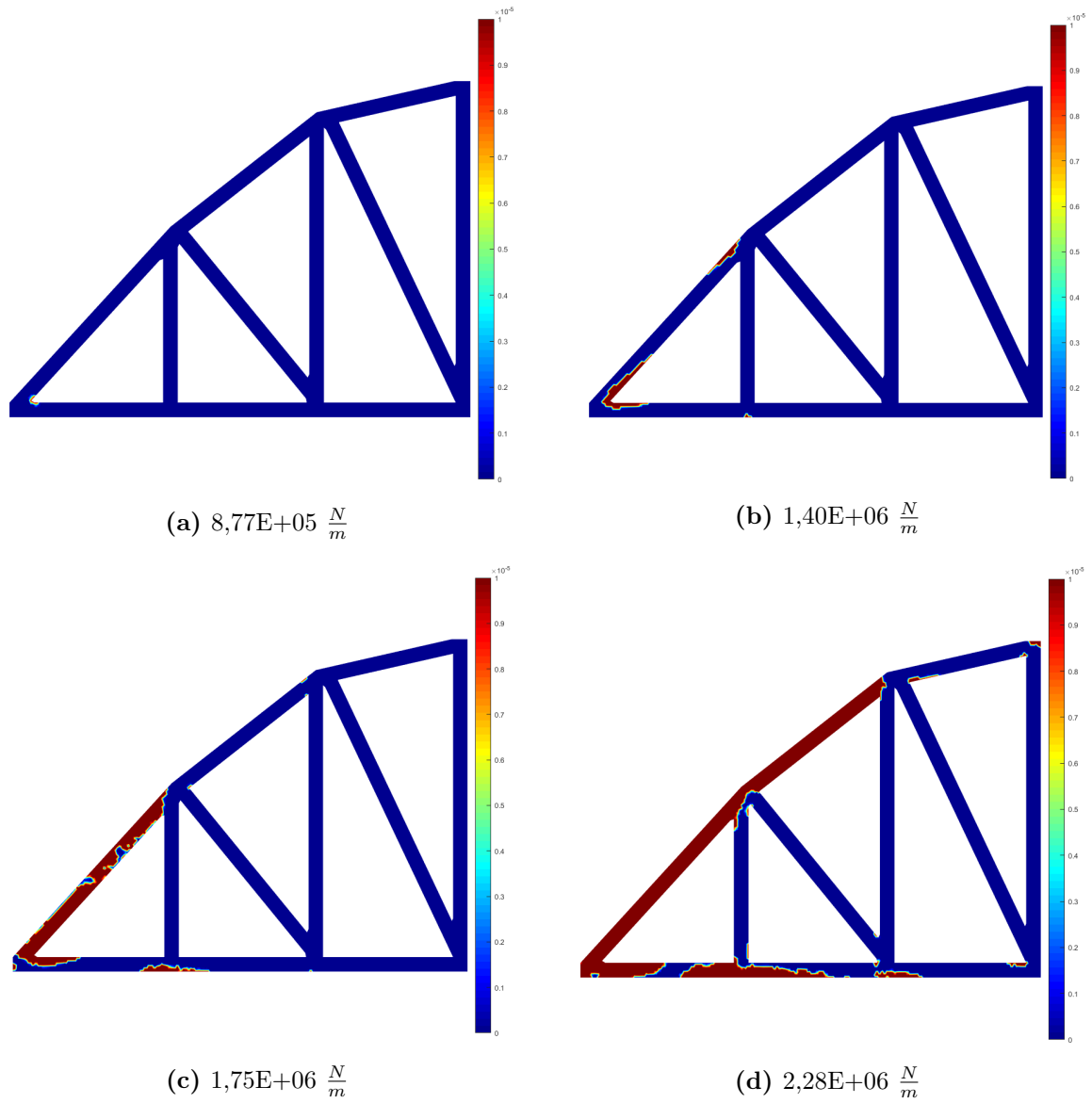


**Figure 7.50:** Development of the effective plastic strain for the NNRPIM

## 7.4 Elastoplastic Analysis Of 3D Structures

In this section, the analysis of the 2D structures presented earlier will be made, but considering 3D geometry.

### 7.4.1 Two Bay Asymmetric Frame

As for the modifications in terms of geometry for the 3D model of the two bay asymmetric frame, the same dimensions as in the 2D case are applied. In terms of thickness, only half of the structure is analyzed due to symmetry.

As for nodal density, a discretization of 2481 nodes was used.

The thickness variation for the 3D model is represented in Figure 7.51, as an upper view of the model.

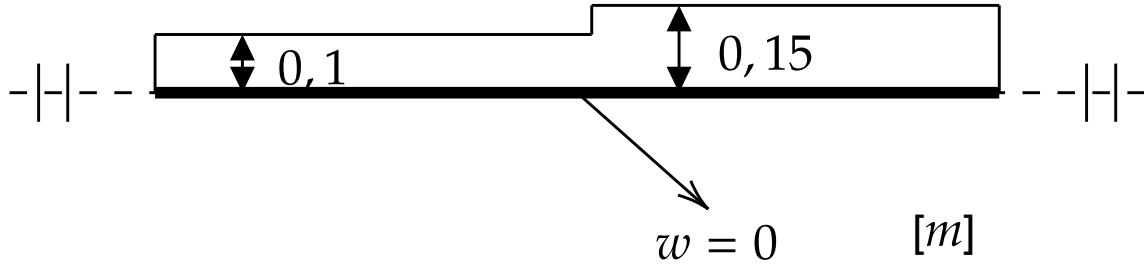


**Figure 7.51:** Thickness variation and additional boundary condition for the 3D model  
- Upper view.

Additionally, due to symmetry, the  $w$  displacement is locked, which is also represented in Figure 7.51.

#### 7.4.1.1 Horizontal Displacement Of Point A

Similarly with the 2D case, for the 3D case, the horizontal displacement of point A are presented in Figure 7.52, 7.53, 7.54, and will be compared with the reference solution. The units used are the same as in the 2D case.

As it can be seen in Figure 7.52, 7.53 and 7.54, the obtained results, although not as coincident with the reference solution given by Argyris et al when compared with the 2D solutions, nonetheless they coincide with the reference values. This deviation is mainly because of the nodal density used. For 3D geometries, the computational cost of a high nodal density, considering the computational power that is available, was very high. Therefore, considering the time that would take for the calculations to be complete, the nodal density used was considered satisfactory due to the results given.

#### 7.4.1.2 Effective Plastic Strain Distribution Maps - Development Of The Plastic Hinges

The effective plastic strain distribution maps for the FEM, RPIM and NNRPIM, considering the 3D case, are presented in Figure 7.55, 7.56 and 7.57.

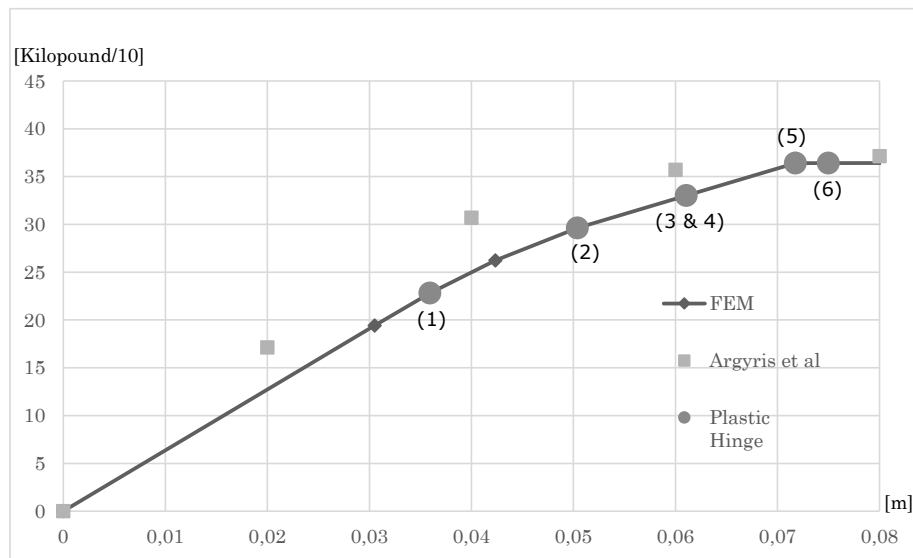


**Figure 7.52:** Horizontal displacement of point A given by the FEM

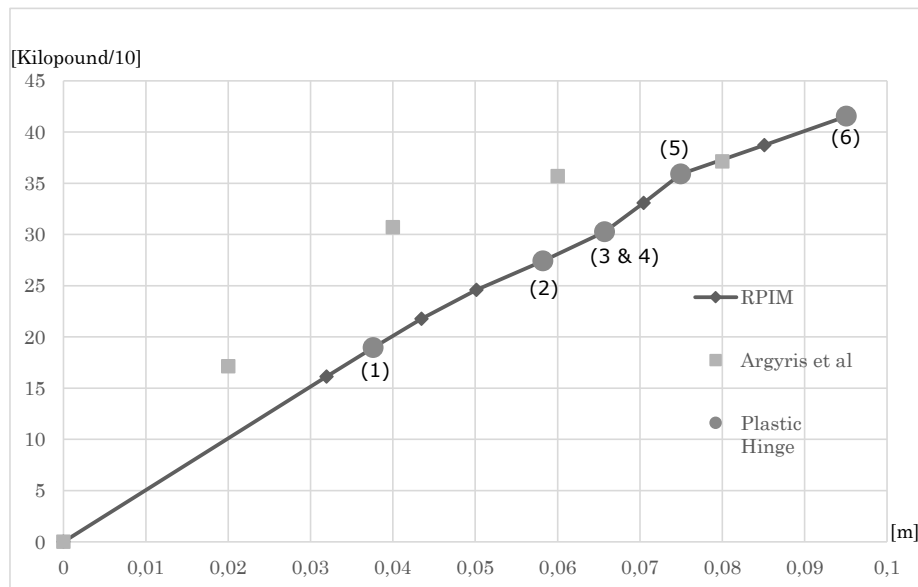


**Figure 7.53:** Horizontal displacement of point A given by the RPIM

Analyzing Figure 7.55, 7.56 and 7.57 and comparing them to the 2D case, the same plastic hinges are developed. In the 3D case, these plastic hinges are more broad, meaning that the plastic strain on these critical points is more severe.

In terms of progression of the plastic hinges, as the load increases, for all three numerical methods, we can see the same evolution. An interesting detail to point out is the absence of the 6<sup>th</sup> plastic hinge in Figure 7.55d. The plastic hinge is not shown due to the fact that the load increment after  $36 \frac{\text{kilopound}}{10}$  lead to a complete hardening of the material, and due to that fact the plastic hinge is not evident. However, as it is shown in Figure 7.52, this plastic hinge is developed but at a lower load.

Overall, the results are in agreement, as they should, with the 2D case.

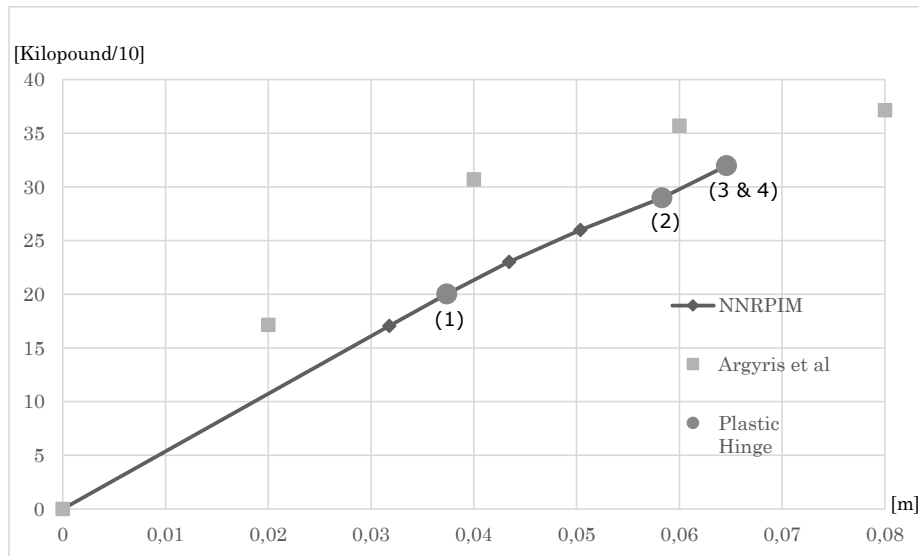


**Figure 7.54:** Horizontal displacement of point A given by the NNRPIM

#### 7.4.1.3 Evolution Of The Effective Stress On The Plastic Hinges

The diagrams showing the evolution of the effective stress on the identified plastic hinges are presented in Figure 7.58, 7.59, 7.60

The results obtained by the three methods are similar between them. The same detail presented for the 2D case, concerning the entrance of the 2<sup>nd</sup> plastic hinge in elastoplastic regime firstly, with the 1<sup>st</sup> plastic hinge having a rapid transition between elastic and plastic behaviour, is also visible.

The simultaneous entrance into plasticity by the 3<sup>rd</sup> and 4<sup>th</sup> plastic hinge is also visible, with the curve shift for both plastic hinges occurring at the same time.

#### 7.4.1.4 Comparison Between The 2D And 3D Results

The results presented for the horizontal displacement of point A for the 2D and 3D case will now be presented in a combined diagram, in Figure 7.61, as for a more complete comparison.

The combined results shown in Figure 7.61 show good agreement between the 2D and 3D results, for all three numerical methods, and a combined agreement with the reference solution by Argyris et al.

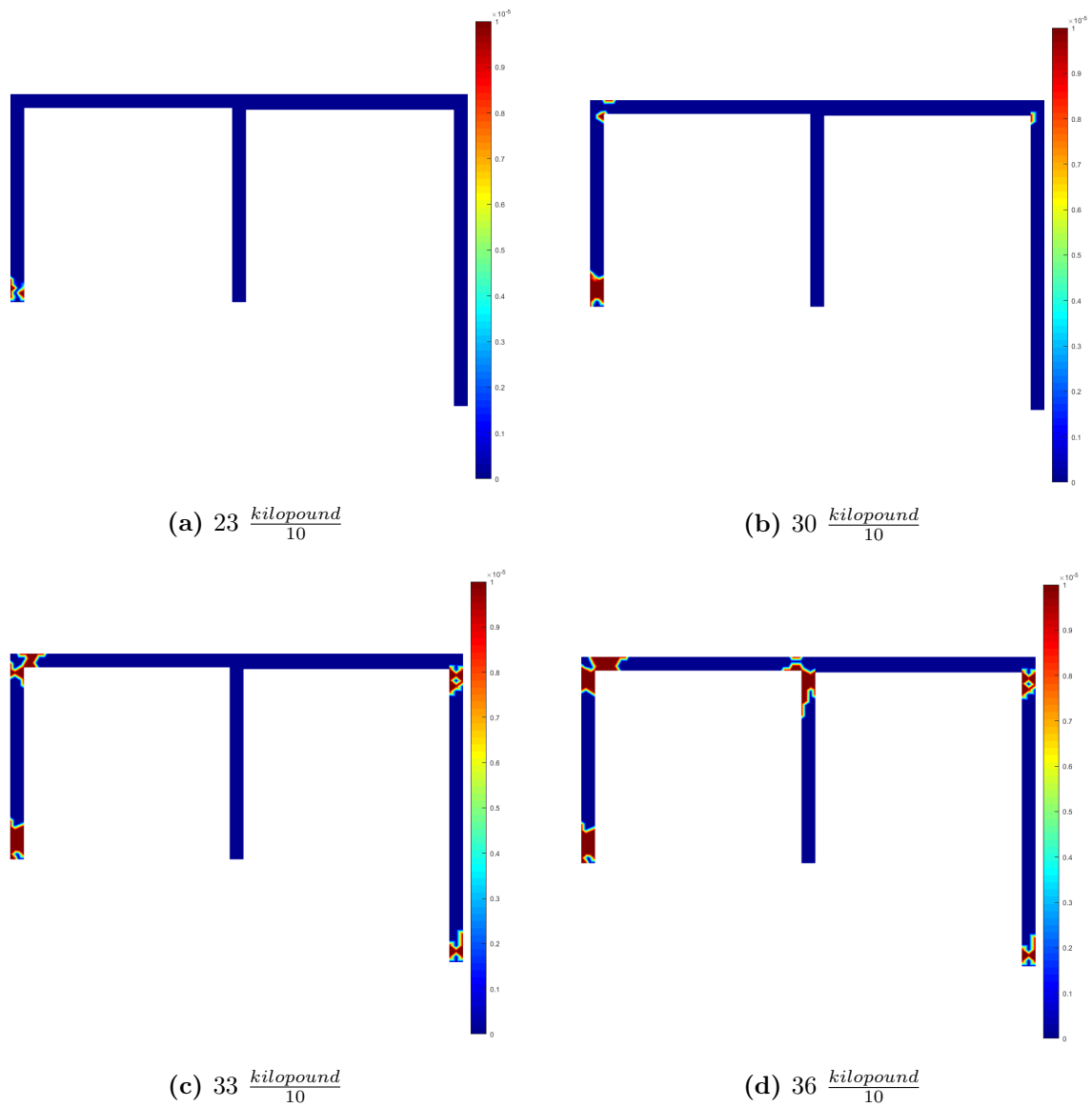


**Figure 7.55:** Development of the plastic hinges for the FEM - 3D case

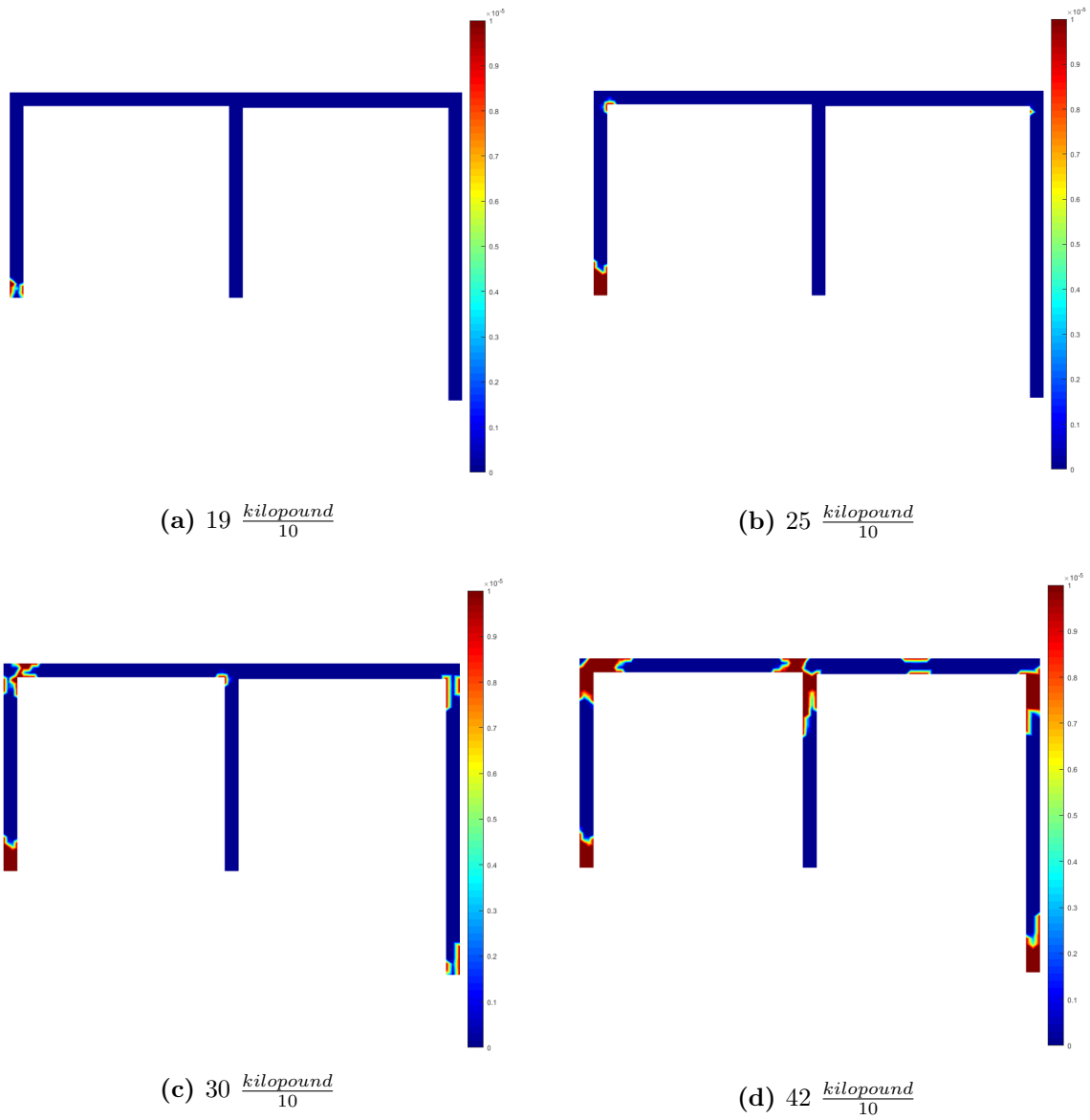


**Figure 7.56:** Development of the plastic hinges for the RPIM - 3D case

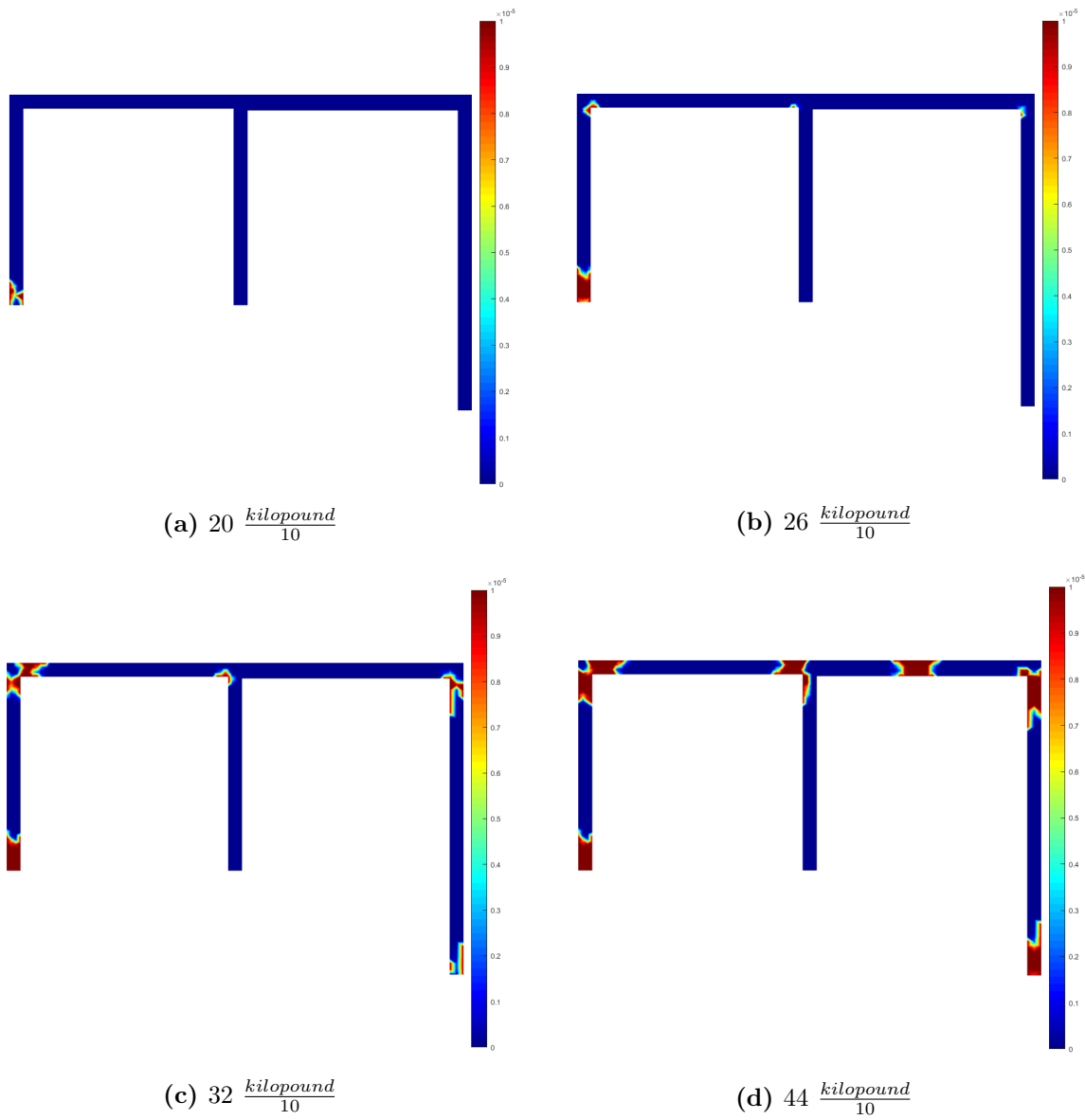


**Figure 7.57:** Development of the plastic hinges for the NRPIM - 3D case

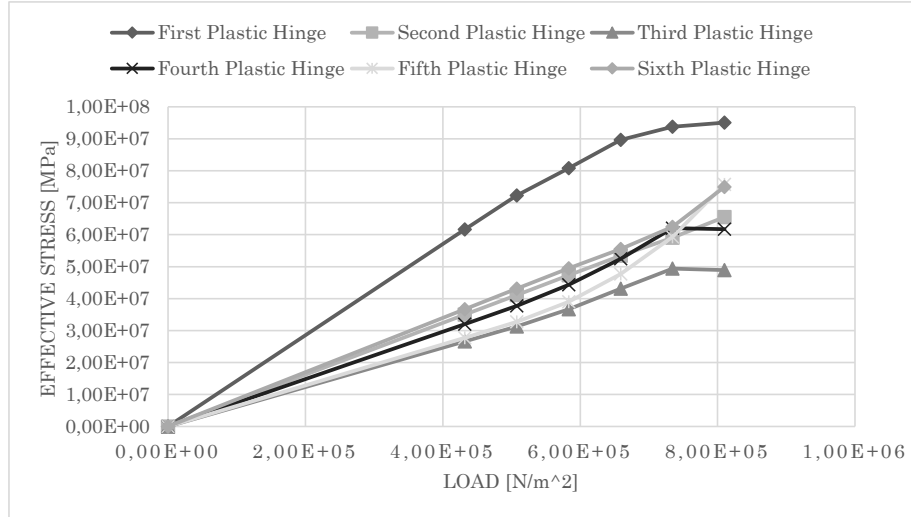


**Figure 7.58:** Evolution of the effective stress on the plastic hinges given by the FEM - 3D case

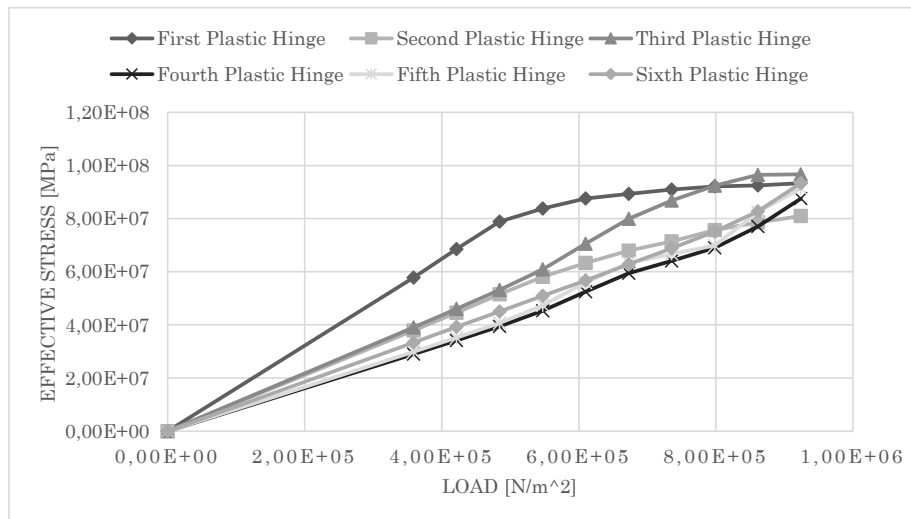


**Figure 7.59:** Evolution of the effective stress on the plastic hinges given by the RPIM - 3D case

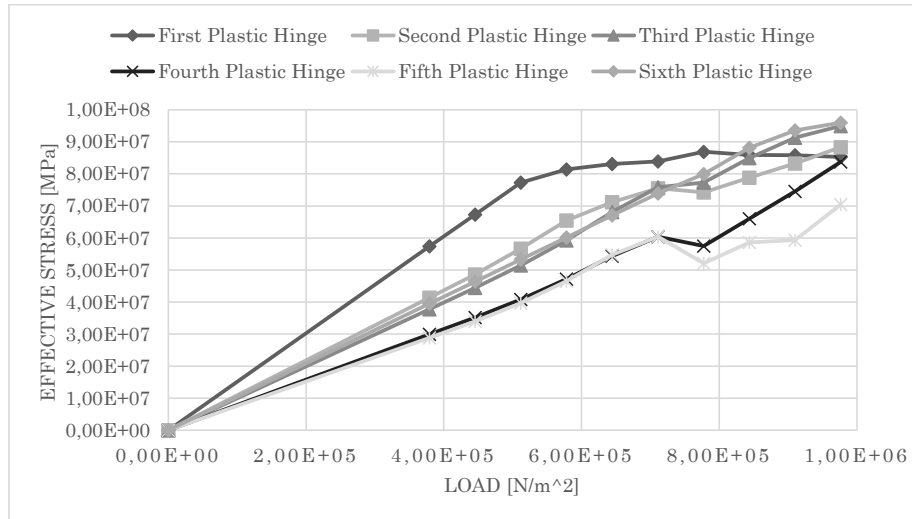


**Figure 7.60:** Evolution of the effective stress on the plastic hinges given by the NNR-PIM - 3D case

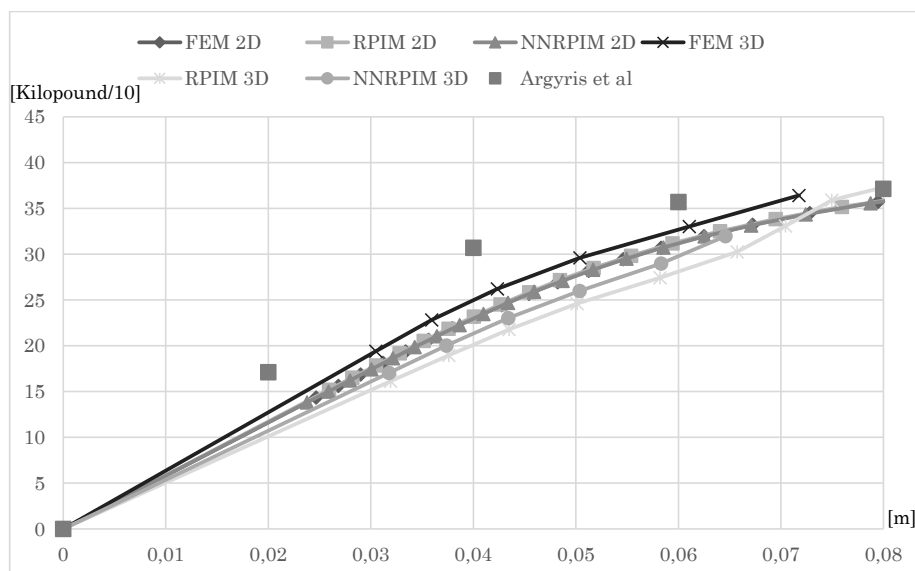


**Figure 7.61:** 2D and 3D results for the horizontal displacement of point A

## 7.4.2 Bowstring Bridge

As it was done with the two bay asymmetric frame, the 3D case of the bowstring bridge will be analyzed. The 3D geometry consists of a complete extrusion of the 2D section with a thickness of 0,1 [m]. A nodal density of 3032 nodes was used. The mechanical properties and dimensions are the same.

### 7.4.2.1 Vertical Displacement Of Point A

The vertical displacement of point **A**, for all three numerical methods, is presented in Figure 7.62.

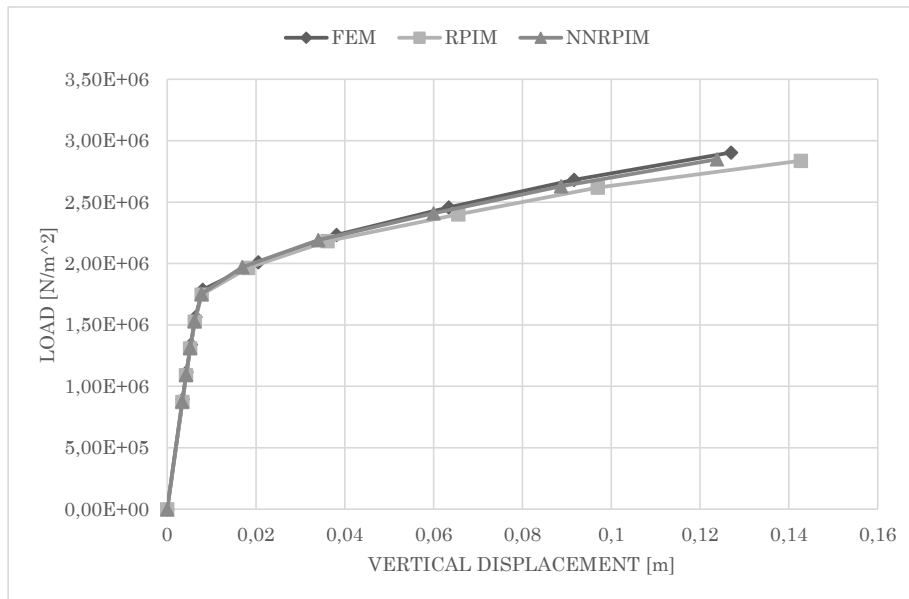


**Figure 7.62:** Vertical displacement of point **A** of the 3D bowstring bridge

The results presented in Figure 7.62 show excellent agreement between the 3 numerical methods used, and are also in agreement with the values obtained for the 2D case presented earlier.

### 7.4.2.2 Effective Plastic Strain Maps - Development

The development of the effective plastic strain along the structure as the load increases is presented in Figure 7.63, 7.64 and 7.65.

The results obtained are in good agreement with the 2D results. The plastic strain initiates on the same point and propagates through the same areas. In the 3D case, the plastic strain is higher, and affects the support trusses of the bridge, while in the 2D case, the plastic strain barely reaches these elements.

Analyzing the three numerical methods, the results are very similar between them.

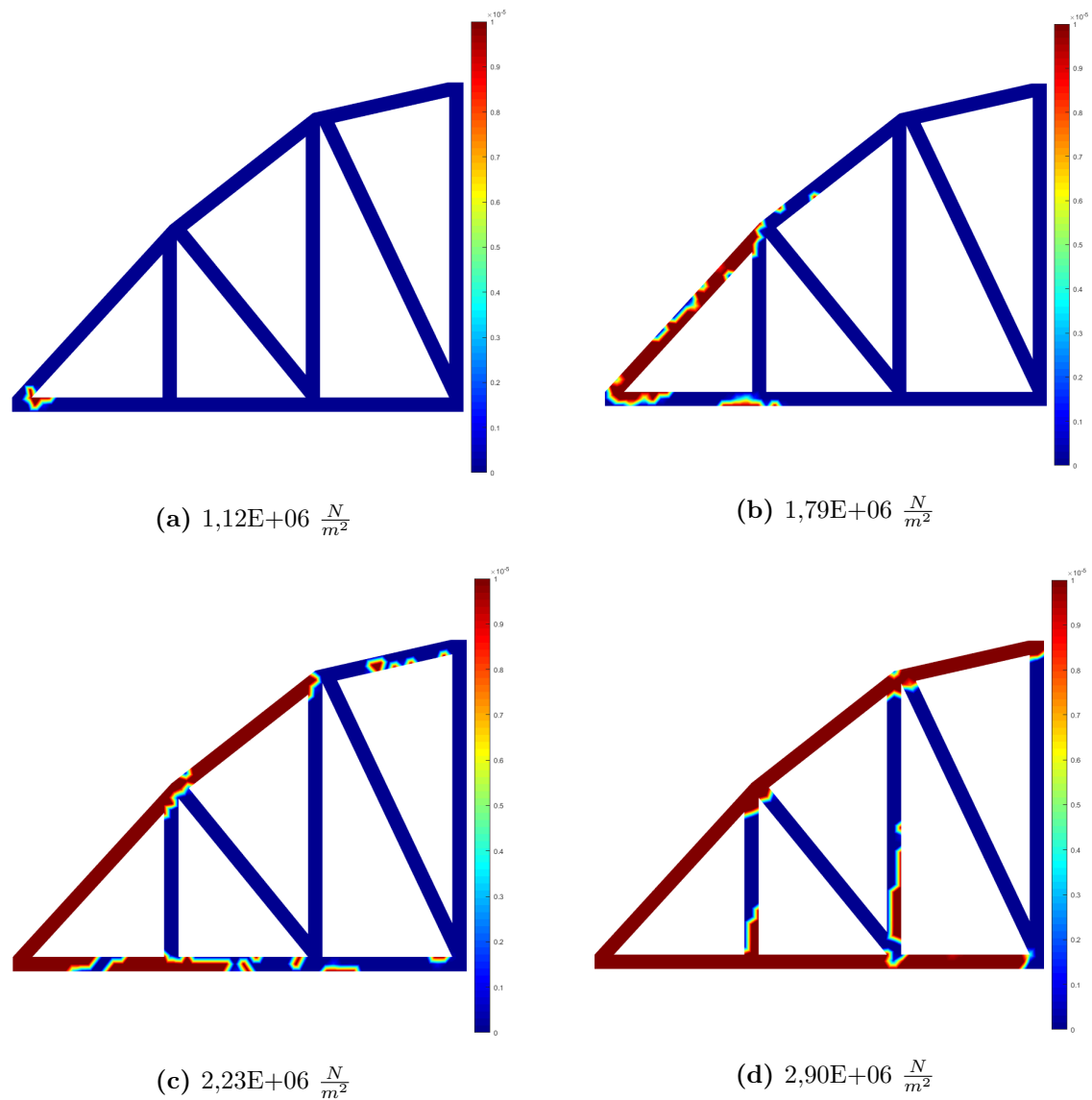


**Figure 7.63:** Development of the effective plastic strain for the FEM - 3D case

#### 7.4.2.3 Comparison Between The 2D and 3D Results

The combined results for the vertical displacement of point **A** are presented in Figure 7.66.

The results shown in Figure 7.66 show excellent agreement between the three numerical methods for both cases. The 3D case returns higher vertical displacements due to a higher load input. However, for the same loads, the vertical displacement is very similar.

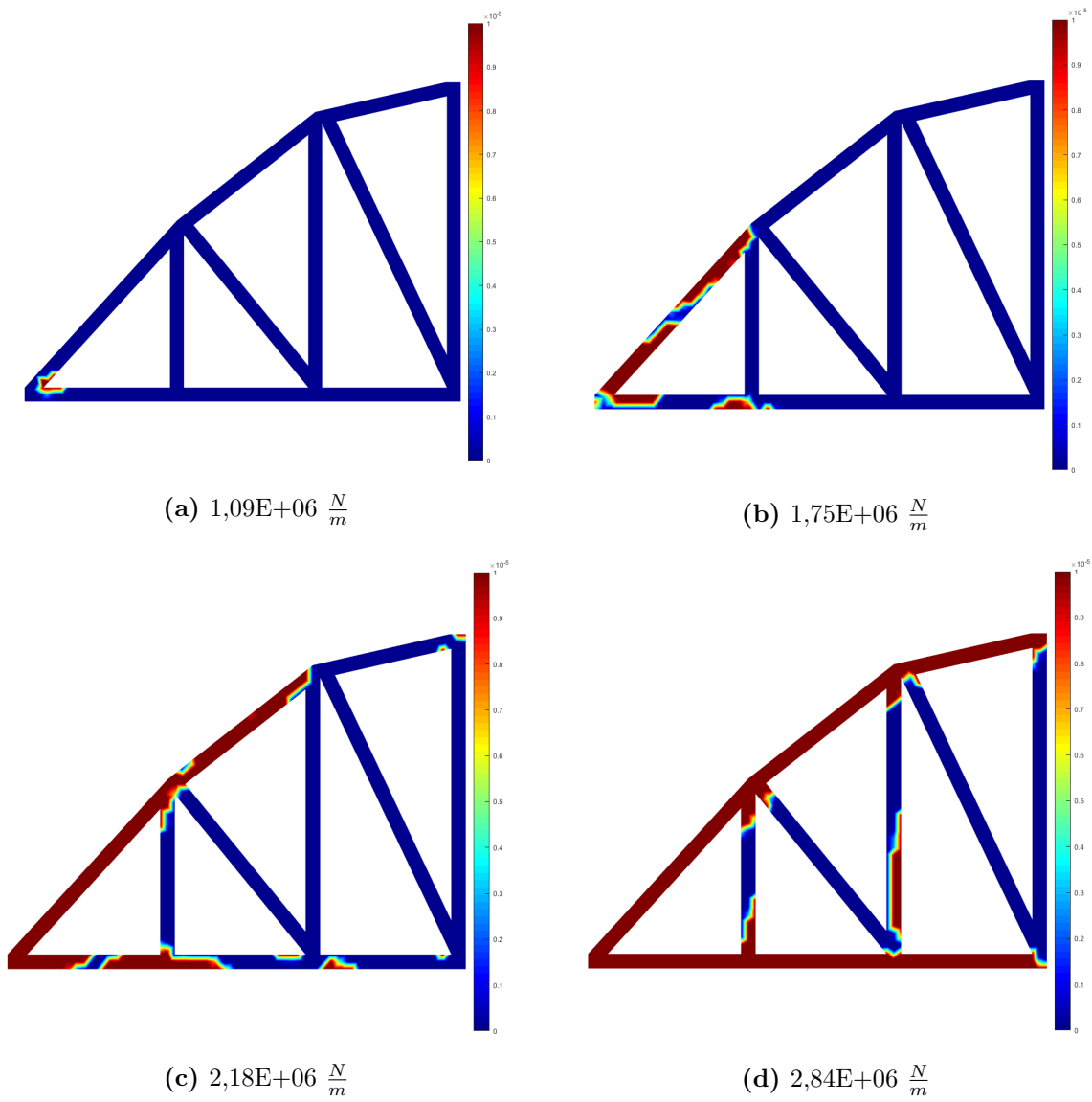


**Figure 7.64:** Development of the effective plastic strain for the RPIM - 3D case

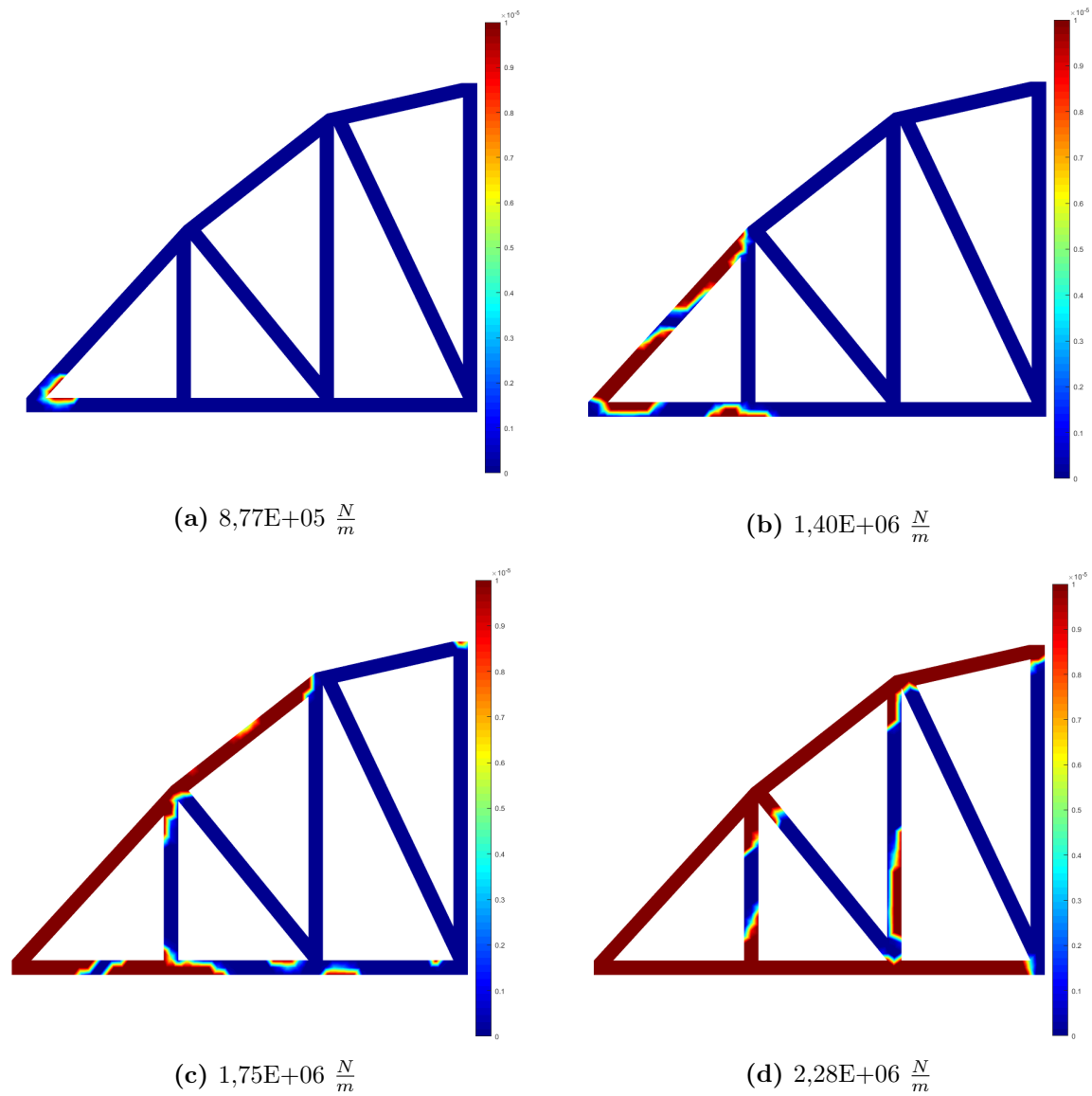


**Figure 7.65:** Development of the effective plastic strain for the NNRPIM

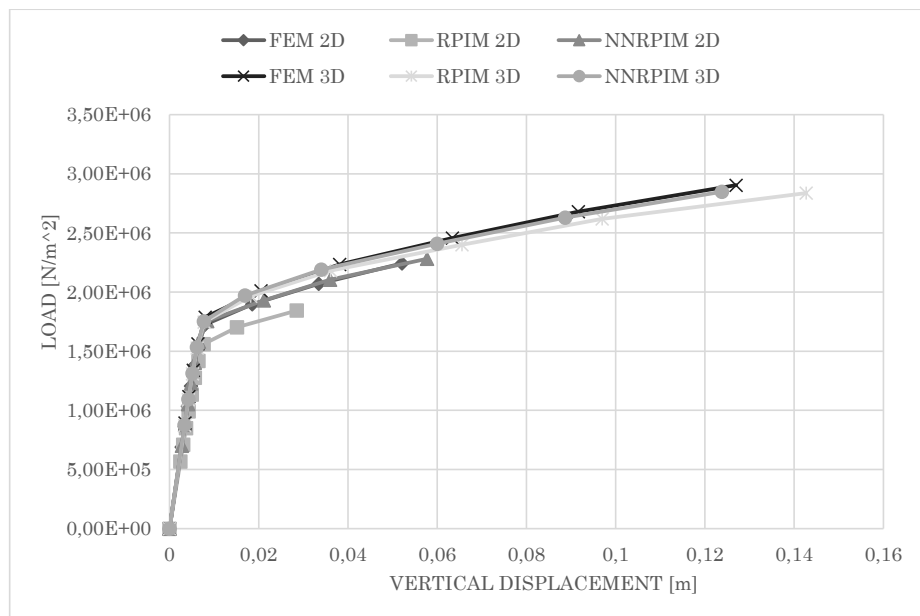


**Figure 7.66:** 2D and 3D results for the vertical displacement of point A

## 7.5 Numerical Analysis of Three-Point Bending Test On 3D Printed PLA Specimens

In this section, the experimental work developed will be presented. With the advancement of additive manufacturing in the engineering world, and the endless possibilities that the technology allows, experimental work considering 3D printing manufacturing of structural elements and the use of meshless methods as a analysis tool was considered interesting and of use to the scientific community.

For this purpose, specimens were manufactured using additive manufacturing. The material considered for this purpose was polylactide or PLA, which is a thermoplastic aliphatic polyester derived from renewable biomass, such as corn starch or sugarcane. It is one of the most popular bioplastics, used in applications from plastic cups to medical implants.

### 7.5.1 Three-Point Bending Test

A three-point bending test consist in the application of a central load on the specimen, which is supported at each end by supporting pins. This type of solicitation inflict flexural stresses and strains and it is used to measure the flexural strength of the specimen.

Figure 7.67 is a representation of the test in itself.

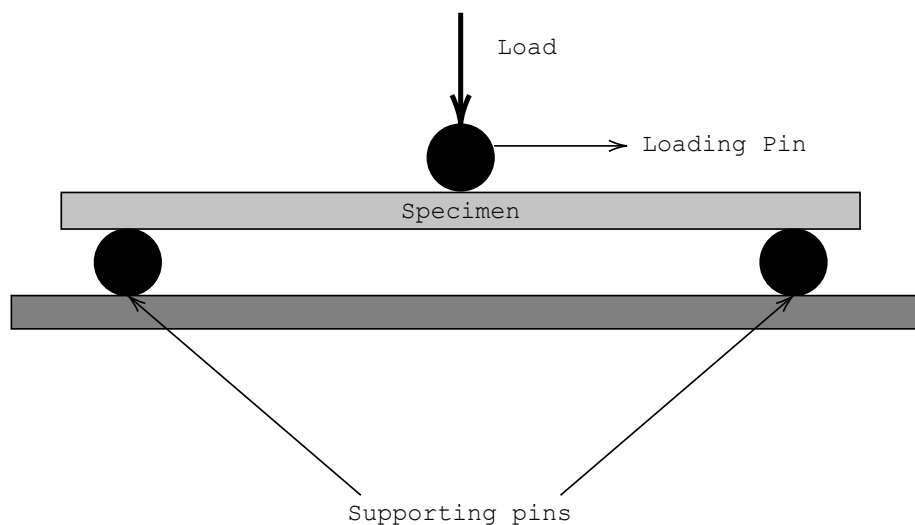


**Figure 7.67:** Three-point flexural test representation

As it can be seen in Figure 7.67, the specimen is supported at each end by the supporting pins and loaded, centrally, by a loading pin.

As for the norm used in the mechanical test, the ISO 178 [63] norm was considered.

### 7.5.2 Manufacturing and Geometry

A 3D printer can print in various directions. This direction has a influence on the strength of the element being printed, considering the type of load that is going to be

applied. Normally, the printing direction should be perpendicular to the load applied.

In this case, considering the load applied, a printing direction of  $0^\circ$  was considered, represented in Figure 7.68.

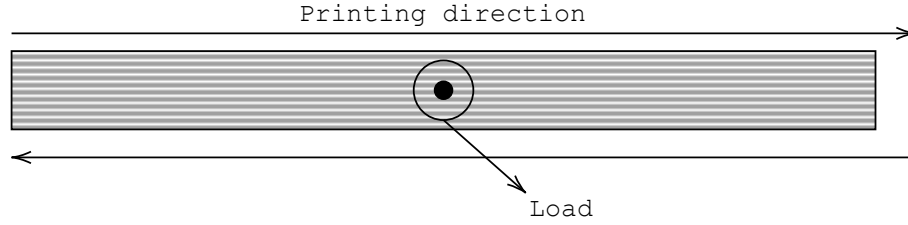


**Figure 7.68:** Printing direction of the specimen

As for the geometry and dimensions, these are represented in Figure 7.69

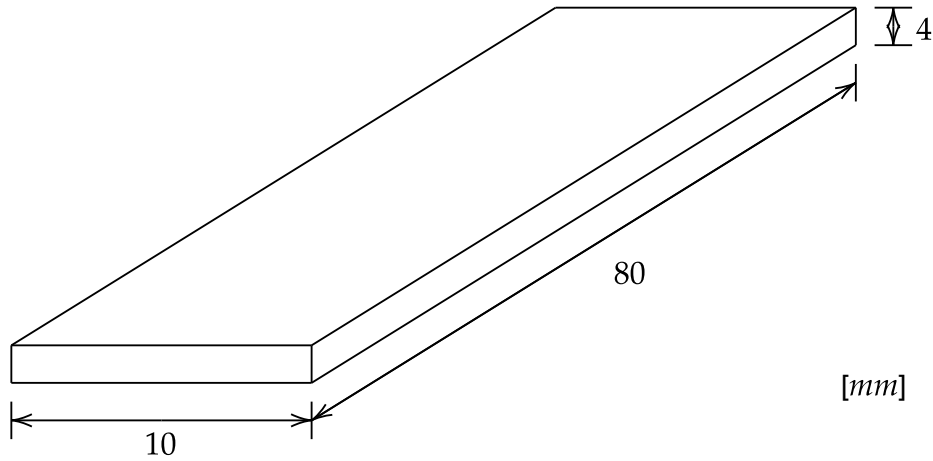


**Figure 7.69:** Geometry and dimensions of the specimen

### 7.5.3 Load and Boundary Conditions

Due to symmetry, only a quarter of the beam will be analyzed.

The load and boundary conditions for the beam are represented in Figure 7.70.

As it can be seen in Figure 7.70, the boundary conditions result from the symmetry conditions of analyzing a quarter of the beam. The vertical displacement block,  $v = 0$ , results from the supporting pins being placed under the beam, not allowing the mentioned displacement.

### 7.5.4 Mechanical properties of PLA

In order to determine the mechanical properties of the material, compressive tests applied to specimens, with the dimensions explicit in Figure 7.71, were made.

The compressive test only applies compressive forces to the specimen, allowing to estimate the material mechanical properties.

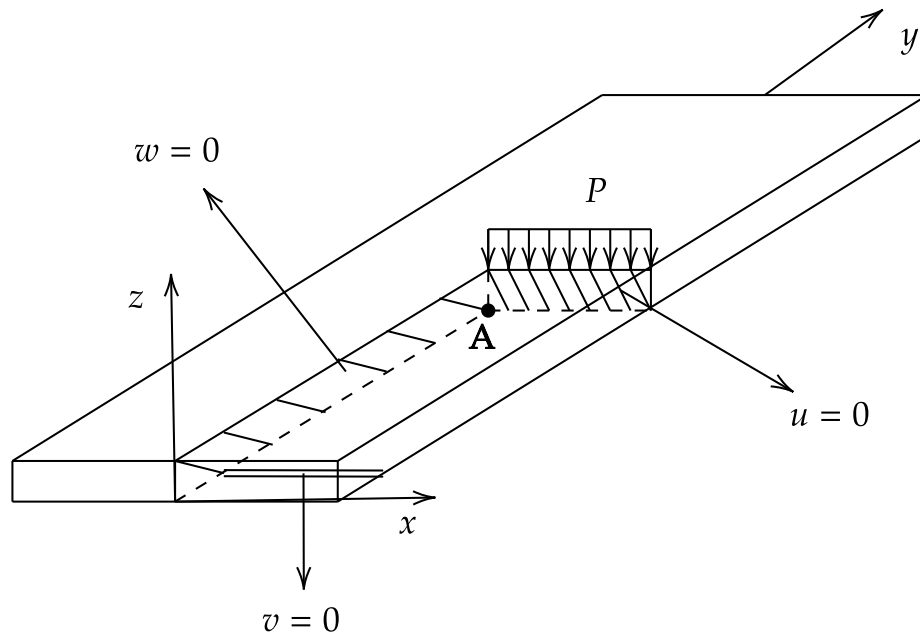


**Figure 7.70:** Boundary and load conditions for the beam

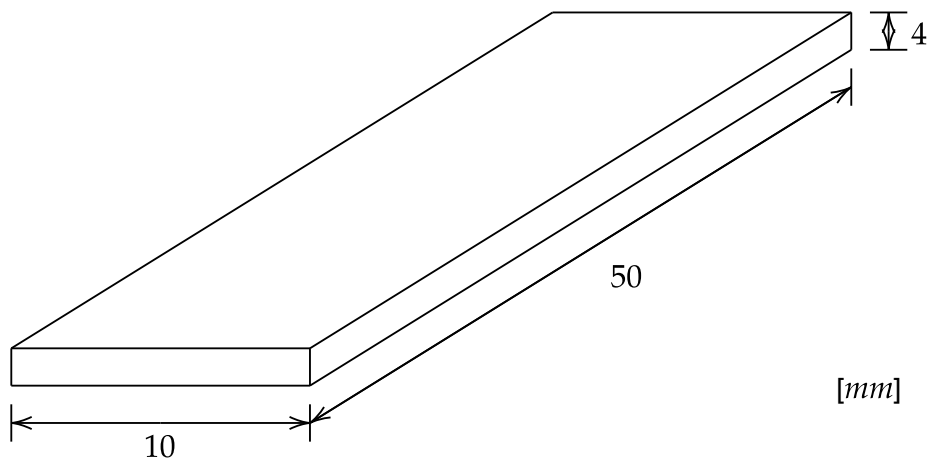


**Figure 7.71:** Dimensions of the specimens used in compressive tests

Five specimens, printed at  $0^\circ$ , were tested, and the resultant effective stress/effective deformation curve was obtained, presented in Figure 7.72

After obtaining the results presented in Figure 7.72, the elastic modulus,  $E$ , was calculated for each specimen tested, being that the average elastic modulus was the value established for the elastic modulus of the PLA. The values obtained are presented in Table 7.12

With the compressive tests and the elastic modulus determined, the mechanical properties of the PLA can be established.

The mechanical properties of the PLA material used are explicit in Table 7.13.

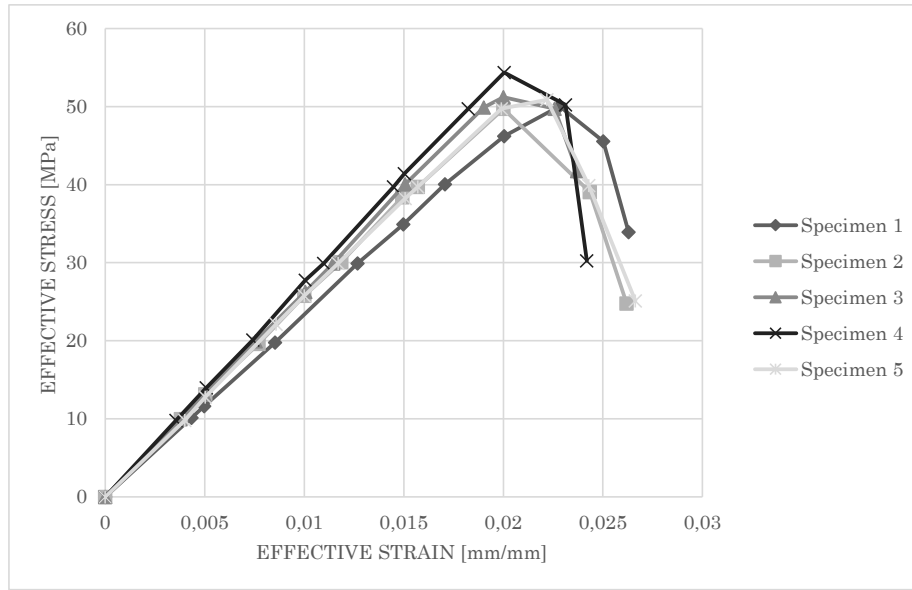


**Figure 7.72:** Results obtained from the compressive tests

| Specimen | Elastic Modulus [ $MPa$ ] |
|----------|---------------------------|
| 1        | 2333,88                   |
| 2        | 2568,65                   |
| 3        | 2684,57                   |
| 4        | 2726,92                   |
| 5        | 2552,36                   |
| Average  | 2573,276                  |

**Table 7.12:** Elastic modulus values obtained via the compressive tests

| Mechanical Properties | Value        |
|-----------------------|--------------|
| $E$ [ $Pa$ ]          | 2573,276E+06 |
| $E_T$ [ $Pa$ ]        | 857,759E+06  |
| $\nu$                 | 0,36         |
| $\sigma_Y$ [ $Pa$ ]   | 93,641E+06   |

**Table 7.13:** Mechanical properties of PLA

### 7.5.5 Numerical Results

The three-point bending test was numerically simulated for the 2D and 3D case. For the 2D case, a nodal density of 6611 nodes, while in the 3D case, a nodal density of 2499 nodes was used. For both cases, the FEM, RPIM and NNRPIM were used combined with the KT-ALL non-linear solution algorithm.

### 7.5.5.1 Effective Stress/Effective Strain

The effective stress/effective strain diagram, for the 2D and 3D case, is presented in Figure 7.73. The experimental results are also presented as to compare with the numerical ones.

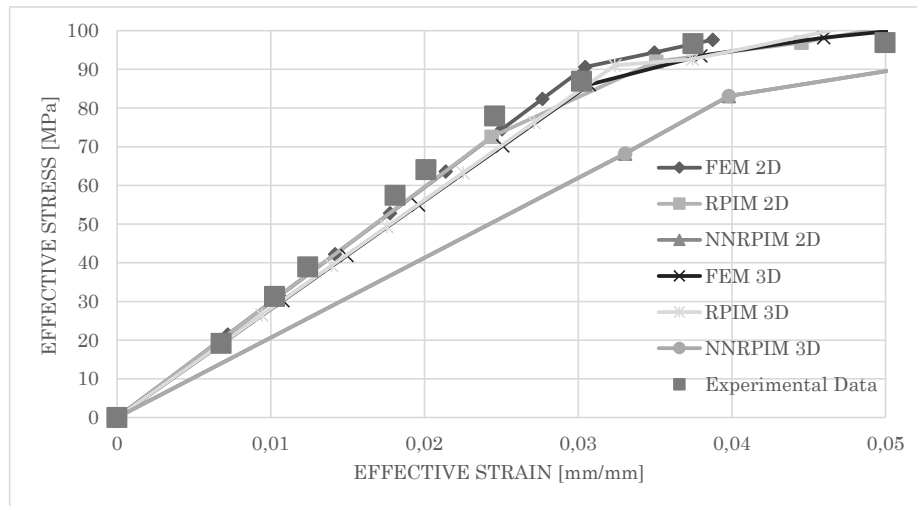


**Figure 7.73:** 2D and 3D results for the effective stress/effective strain

Analyzing the results presented in Figure 7.73, it is possible to observe an excellent agreement between the numerical methods and the experimental results. For both 2D and 3D cases, the FEM and RPIM achieve almost equal results compared with the experimental ones.

On the other hand, for both geometrical cases, the NNRPIM curve is below the expected experimental one. Nevertheless, the results still follow the expected trend.

# Chapter 8

## Conclusions

In this work, the Radial Point Interpolation Method (RPIM) and the Natural Neighbour Radial Point Interpolation Method (NNRPIM) were used in the elastoplastic analysis of 2D and 3D structural elements.

In order to verify the methods used, a convergence analysis was made using Cook's membrane. The three numerical methods were compared with a reference solution. From that study, it was concluded that:

- For a low nodal density, the meshless method used, the RPIM and NNRPIM, had a closer first solution value than the FEM, which leads to the conclusion that for low nodal densities meshless methods perform better than the FEM.
- All three methods, as expected, as the nodal density rises, the results obtained are increasingly more accurate. Higher nodal densities lead to better results for all three numerical methods.
- The comparison between the 2D and 3D leads to the conclusion that the results obtained are very similar, and all methods are very close to each other.

Afterwards, the elastoplastic analysis of uniformly loaded plates was presented. The main objective of this example was not only the non-linear analysis of a 3D structural element but also to analyze the behaviour of the non-linear solution algorithms.

From this analysis, we can conclude that:

- The results obtained for the three numerical methods are in good agreement with the ones reported in literature.
- The KT-1 algorithm, for the clamped plate, returned results that more accurately approached the ones given by the reference solution, although all three algorithms return good results. For the simply supported plate, the KT-0 and KT-ALL adjust quite well to the literature solution.
- In terms of computational power and time, the KT-ALL algorithm was by far the fastest one to run. For this specific case, the KT-0 algorithm took around 7 hours to compute the solutions presented.

- It was found that the NNRPIM needs a higher nodal density than the FEM and RPIM in order to apply the same load values.
- The stress diagrams along the plates thickness clearly present the effects of plasticity on the element. Regarding the comparison between the numerical methods, good agreement between them was found, with the meshless methods returning the smoothest solutions, specially the NNRPIM.

Next, the elastoplastic analysis of frames was executed. A selected example from Argyris et al and a novel example by the author are presented. It is concluded that:

- For the two bay asymmetric frame, considering both 2D and 3D cases, the FEM, RPIM and NNRPIM reach good agreement with the reference solution.
- The three methods reach the same results not only in terms of order but also in terms of location of the plastic connections.
- For the 3D case, a higher nodal density for a more accurate analysis is needed, with this fact being dependent on the computational power that is available.
- For the bowstring bridge, excellent agreement for all three methods is reached, with the RPIM for the 2D case not reaching the load input to equal the other methods.
- For both examples, the development of the plastic hinges and overall material plasticity is more broad in the 3D case, mainly due to the lower nodal density used.

Lastly, the numerical analysis of 3D printed specimens, considering a three-point bending test was made. Compression tests were executed in order to determine the mechanical properties of the material. After the study, the main conclusions are:

- Excellent agreement between the experimental data and the numerical data was reached.
- For the NNRPIM, for both 2D and 3D case, the result deviate somewhat from the experimental data, but following the expected tendency. Once more, the need for the NNRPIM to use a higher nodal density is evidenced.

The work developed throughout this work shows that the meshless methods used, the RPIM and NNRPIM, reach good results not only when compared to the FEM but with the reference solutions presented. They accurately analyze elastoplastic behaviour, not only of simple structural elements as plates but of complex structures as a bowstring bridge. It is shown that these methods behave better than the FEM for low nodal densities, with the NNRPIM having a disadvantage in terms of pre-processing time. The RPIM, depending on the Gauss quadrature chosen, has a low pre-processing time phase.

As future work, regarding the elastoplastic analysis of structures, the meshless methods used could be applied to:

- Vibrations analysis and a dynamic analysis of structural elements;
- Structural optimization, and how the non-linear behaviour of the structural element is affected;
- Anisotropic and orthotropic behaviour applied to structural elements;
- The consideration of other hardening rules, such as kinematic and independent hardening.



# Bibliography

- [1] Jorge Belinha. Meshless methods in biomechanics. *Bone Tissue Remodelling Analysis*, 2014.
- [2] Ted Belytschko, Yury Krongauz, Daniel Organ, Mark Fleming, and Petr Krysl. Meshless methods: an overview and recent developments. *Computer methods in applied mechanics and engineering*, 139(1-4):3–47, 1996.
- [3] YuanTong Gu. Meshfree methods and their comparisons. *International Journal of Computational Methods*, 2(04):477–515, 2005.
- [4] Vinh Phu Nguyen, Timon Rabczuk, Stéphane Bordas, and Marc Duflot. Meshless methods: a review and computer implementation aspects. *Mathematics and computers in simulation*, 79(3):763–813, 2008.
- [5] Cheng-An Wang, Hamou Sadat, Vital Ledez, and Denis Lemonnier. Meshless method for solving radiative transfer problems in complex two-dimensional and three-dimensional geometries. *International Journal of Thermal Sciences*, 49(12):2282–2288, 2010.
- [6] E. Oñate. *Structural Analysis with the Finite Element Method: Linear Statics*. Number v. 1 in Lecture notes on numerical methods in engineering and sciences. Springer., 2009.
- [7] Walter Ritz. Über eine neue methode zur losung gewisser variationsprobleme der mathematischen physik. *Journal fur Mathematik*, 135:s–1, 1909.
- [8] Boris Grigoryevich Galerkin. Series solution of some problems of elastic equilibrium of rods and plates. *Vestn. Inzh. Tekh*, 19:897–908, 1915.
- [9] William Weaver and James M Gere. *Matrix analysis framed structures*. Springer science & business media, 2012.
- [10] John H Argyris and Sydney Kelsey. *Energy theorems and structural analysis*, volume 60. Springer, 1960.
- [11] MJ Turner. Stiffness and deflection analysis of complex structures. *journal of the Aeronautical Sciences*, 23(9):805–823, 1956.
- [12] Ray W Clough. The finite element method in plane stress analysis. In *Proceedings of 2nd ASCE Conference on Electronic Computation, Pittsburgh Pa., Sept. 8 and 9, 1960*, 1960.

- [13] Olek C Zienkiewicz and Robert L Taylor. *The finite element method for solid and structural mechanics*. Elsevier, 2005.
- [14] Susanne Brenner and Ridgway Scott. *The mathematical theory of finite element methods*, volume 15. Springer Science & Business Media, 2007.
- [15] Augustus Edward Hough Love. Xvi. the small free vibrations and deformation of a thin elastic shell. *Philosophical Transactions of the Royal Society of London.(A.)*, (179):491–546, 1888.
- [16] Raymond D Mindlin. Influence of rotatory inertia and shear on flexural motions of isotropic, elastic plates. *J. appl. Mech.*, 18:31–38, 1951.
- [17] Junuthula Narasimha Reddy. An introduction to the finite element method. *New York*, 1993.
- [18] Klaus-Jürgen Bathe. *Finite element procedures*. Klaus-Jurgen Bathe, 2006.
- [19] Hyochoong Bang and Young W Kwon. *The finite element method using MATLAB*. CRC press, 2000.
- [20] M Nurhaniza, MKA Ariffin, Aidy Ali, F Mustapha, and AW Noraini. Finite element analysis of composites materials for aerospace applications. In *IOP Conference Series: Materials Science and Engineering*, volume 11, page 012010. IOP Publishing, 2010.
- [21] Horea Gozar, Alexandru Chira, Örs Nagy, and Zoltán Derzsi. Medical use of finite element modeling of the ankle and foot. *Journal of Interdisciplinary Medicine*, 3(1):34–38, 2018.
- [22] Ajaya K Gupta, Bijan Mohraz, and William C Schnobrich. Elasto-plastic analysis of three-dimensional structures using the isoparametric element. *Nuclear Engineering and Design*, 22(2):305–317, 1972.
- [23] JH Argyris, B Boni, U Hindenlang, and M Kleiber. Finite element analysis of two- and three-dimensional elasto-plastic frames—the natural approach. *Computer Methods in Applied Mechanics and Engineering*, 35(2):221–248, 1982.
- [24] Xucheng Wang and Xiaoning Wang. Simplified method for elasto-plastic finite element analysis of hardening materials. *Computers & structures*, 55(4):703–708, 1995.
- [25] Bahattin Kanber and O Yavuz Bozkurt. Finite element analysis of elasto-plastic plate bending problems using transition rectangular plate elements. *Acta Mechanica Sinica*, 22(4):355–365, 2006.
- [26] Ehsan Hashemi and Behrooz Farshi. Finite element based elasto-plastic analysis of classical and first order beams with armstrong–frederick kinematic hardening model. *Transactions of the Canadian Society for Mechanical Engineering*, 38(1):1–14, 2014.

- [27] Fabio De Angelis and Robert L Taylor. A nonlinear finite element plasticity formulation without matrix inversions. *Finite Elements in Analysis and Design*, 112:11–25, 2016.
- [28] Robert A Gingold and Joseph J Monaghan. Smoothed particle hydrodynamics: theory and application to non-spherical stars. *Monthly notices of the royal astronomical society*, 181(3):375–389, 1977.
- [29] Larry D Libersky and Albert G Petschek. Smooth particle hydrodynamics with strength of materials. In *Advances in the free-Lagrange method including contributions on adaptive gridding and the smooth particle hydrodynamics method*, pages 248–257. Springer, 1991.
- [30] Saba Gharehdash, Milad Barzegar, Igor B Palymskiy, and Pavel A Fomin. Blast induced fracture modelling using smoothed particle hydrodynamics. *International Journal of Impact Engineering*, 2019.
- [31] B Nayroles, G Touzot, and P Villon. Generalizing the finite element method: diffuse approximation and diffuse elements. *Computational mechanics*, 10(5):307–318, 1992.
- [32] Peter Lancaster and Kes Salkauskas. Surfaces generated by moving least squares methods. *Mathematics of computation*, 37(155):141–158, 1981.
- [33] Wing Kam Liu, Sukky Jun, and Yi Fei Zhang. Reproducing kernel particle methods. *International journal for numerical methods in fluids*, 20(8-9):1081–1106, 1995.
- [34] Satya N Atluri and Tulong Zhu. A new meshless local petrov-galerkin (mlpg) approach in computational mechanics. *Computational mechanics*, 22(2):117–127, 1998.
- [35] Gui-Rong Liu and YuanTong Gu. A point interpolation method for two-dimensional solids. *International Journal for Numerical Methods in Engineering*, 50(4):937–951, 2001.
- [36] JG Wang and GRs Liu. A point interpolation meshless method based on radial basis functions. *International Journal for Numerical Methods in Engineering*, 54(11):1623–1648, 2002.
- [37] LMJS Dinis, RM Natal Jorge, and J Belinha. Analysis of 3d solids using the natural neighbour radial point interpolation method. *Computer Methods in Applied Mechanics and Engineering*, 196(13-16):2009–2028, 2007.
- [38] Jorge Américo Oliveira Pinto Belinha et al. The natural neighbour radial point interpolation method: solid mechanics and mechanobiology applications. 2010.
- [39] Somnath Ghosh and Suresh Moorthy. Elastic-plastic analysis of arbitrary heterogeneous materials with the voronoi cell finite element method. *Computer Methods in Applied Mechanics and Engineering*, 121(1-4):373–409, 1995.

- [40] William Barry and Sunil Saigal. A three-dimensional element-free galerkin elastic and elastoplastic formulation. *International Journal for Numerical Methods in Engineering*, 46(5):671–693, 1999.
- [41] David Roger Jones Owen. Finite elements in plasticity, theory and practice. 1980.
- [42] Jorge Américo Belinha. Análise elasto-plástica considerando o método livre de elementos de galerkin: Problemas bidimensionais, placas e laminados. 2004.
- [43] LMJS Dinis, RM Natal Jorge, and J Belinha. The natural neighbour radial point interpolation meshless method applied to the non-linear analysis. In *AIP Conference Proceedings*, volume 1353, pages 1175–1178. AIP, 2011.
- [44] YM Cheng, FN Bai, and MJ Peng. A novel interpolating element-free galerkin (iefg) method for two-dimensional elastoplasticity. *Applied Mathematical Modelling*, 38(21-22):5187–5197, 2014.
- [45] Malgorzata A Jankowska. On elastoplastic analysis of some plane stress problems with meshless methods and successive approximations method. *Engineering Analysis with Boundary Elements*, 95:12–24, 2018.
- [46] Jagabanduhu Chakrabarty. *Theory of plasticity*. Elsevier, 2012.
- [47] Wen Chen, Zhuo-Jia Fu, and Ching-Shyang Chen. *Recent advances in radial basis function collocation methods*. Springer, 2014.
- [48] Behzad V Farahani, J Belinha, Rui Amaral, Paulo J Tavares, and Pedro MPG Moreira. Extending radial point interpolating meshless methods to the elastoplastic analysis of aluminium alloys. *Engineering Analysis with Boundary Elements*, 100:101–117, 2019.
- [49] LMJS Dinis, RM Natal Jorge, and Jorge Belinha. Analysis of plates and laminates using the natural neighbour radial point interpolation method. *Engineering Analysis with Boundary Elements*, 32(3):267–279, 2008.
- [50] Rolland L Hardy. Theory and applications of the multiquadric-biharmonic method 20 years of discovery 1968–1988. *Computers & Mathematics with Applications*, 19(8-9):163–208, 1990.
- [51] JG Wang and GR Liu. On the optimal shape parameters of radial basis functions used for 2-d meshless methods. *Computer methods in applied mechanics and engineering*, 191(23-24):2611–2630, 2002.
- [52] Robin Sibson. A vector identity for the dirichlet tessellation. In *Mathematical Proceedings of the Cambridge Philosophical Society*, volume 87, pages 151–155. Cambridge University Press, 1980.
- [53] Georges Voronoi. Nouvelles applications des paramètres continus à la théorie des formes quadratiques. deuxième mémoire. recherches sur les paralléloèdres primitifs. *Journal für die reine und angewandte Mathematik*, 134:198–287, 1908.

- [54] Singiresu S. Rao. Chapter 8 - basic equations and solution procedure. In Singiresu S. Rao, editor, *The Finite Element Method in Engineering (Fifth Edition)*, pages 277 – 309. Butterworth-Heinemann, Boston, fifth edition edition, 2011.
- [55] Ernest Hinton and David Roger Jones Owen. *Finite elements in plasticity: Theory and practice*. Pineridge Press Limited, 2017.
- [56] Andreas Ochsner. *Elasto-plasticity of frame structure elements*. Springer, 2016.
- [57] J Bauschinger. On the change of the elastic limit and strength of iron and steel by tension and compression, by heating and cooling and by often repeated loading. *Mitt. Mech*, 13(8):1, 1886.
- [58] René De Borst, Mike A Crisfield, Joris JC Remmers, and Clemens V Verhoosel. *Nonlinear finite element analysis of solids and structures*. John Wiley & Sons, 2012.
- [59] G Shi, Y Liu, and X Wang. Accurate, efficient, and robust q4-like membrane elements formulated in cartesian coordinates using the quasi-conforming element technique. *Mathematical Problems in Engineering*, pages 8–12, 2015.
- [60] J Belinha and LMJS Dinis. Elasto-plastic analysis of plates by the element free galerkin method. *Engineering computations*, 23(5):525–551, 2006.
- [61] Joaquim de Azevedo Figueiras. Ultimate load analysis of anisotropic and reinforced concrete plates and shells. 1983.
- [62] José Augusto Trigo Barbosa. Análise não linear por elementos finitos de placas e cascas reforçadas: aplicação a laminados anisotrópicos. 1992.
- [63] International Organization for Standardization. 176 - plastics – determination of flexural properties. 2019.