CAIO CESAR PEREIRA SANTOS

**Structural Analysis of Flexible Pipes and Umbilical Cables: a
bimaterial Finite Element modeling technique and a novel
experimental approach using a Digital Image Correlation system**

São Paulo

2019

CAIO CESAR PEREIRA SANTOS

**Structural Analysis of Flexible Pipes and Umbilical Cables: a
bimaterial Finite Element modeling technique and a novel
experimental approach using a Digital Image Correlation system**

Thesis presented to the Graduate Program in Mechanical Engineering of the Escola Politécnica da Universidade de São Paulo to obtain the degree of Doctor of Sciences.

São Paulo

2019

CAIO CESAR PEREIRA SANTOS

**Structural Analysis of Flexible Pipes and Umbilical Cables: a
bimaterial Finite Element modeling technique and a novel
experimental approach using a Digital Image Correlation system**

Revised version

Thesis presented to the Graduate Program in Mechanical Engineering of the Escola Politécnica da Universidade de São Paulo to obtain the degree of Doctor of Sciences.

Concentration area:
Mechanical Engineering

Advisor:
Celso Pupo Pesce

São Paulo
2019

Autorizo a reprodução e divulgação total ou parcial deste trabalho, por qualquer meio convencional ou eletrônico, para fins de estudo e pesquisa, desde que citada a fonte.

Este exemplar foi revisado e corrigido em relação à versão original, sob responsabilidade única do autor e com a anuência de seu orientador.

São Paulo, _____ de _____ de _____

Assinatura do autor: _____

Assinatura do orientador: _____

Catálogo-na-publicação

Santos, Caio Cesar Pereira

Structural Analysis of Flexible Pipes and Umbilical Cables: a bimaterial Finite Element modeling technique and a novel experimental approach using a Digital Image Correlation system / C. C. P. Santos -- versão corr. -- São Paulo, 219.

243 p.

Tese (Doutorado) - Escola Politécnica da Universidade de São Paulo. Departamento de Engenharia Mecânica.

1.Tubos flexíveis 2.Cabos umbilicais 3.Método dos Elementos Finitos
4.Correlação de imagem digital I.Universidade de São Paulo. Escola Politécnica. Departamento de Engenharia Mecânica II.t.

DEDICATORY

À minha futura esposa, à minha mãe, ao meu
irmão, a Airon.

ACKNOWLEDGMENTS

Um primeiro agradecimento à minha família, que me suportou e assistiu durante estes cinco longos anos. Em especial, à minha noiva e futura esposa Luana, pela paciência de cada dia e suporte emocional que tornaram reais cada página desta tese. À minha mãe e meu irmão, que sofreram comigo, mas sempre estiveram presentes no que foi preciso. Aos demais membros da minha família e amigos, que acompanharam essa longa jornada. Agradeço do fundo do meu coração.

Agradeço aos colegas de trabalho do Laboratório de Mecânica Offshore (LMO). Especialmente a dois grandes amigos que ganhei, com os quais tive a honra de dividir inúmeros almoços e pausas para o café: Rafael e Heloísa, sou muito grato. Também não posso deixar de citar alguns amigos do laboratório que foram de grande ajuda: Cristiano, sempre me auxiliando na realização dos mais diversos experimentos (úteis e inúteis); Larissa, ajudando com toda burocracia; Marcos Rabelo, presente no início desta caminhada.

Um agradecimento mais que especial ao meu orientador, amigo, conselheiro, Prof. Celso P. Pesce. Não somente pelos ensinamentos acadêmicos, mas por toda convivência e sabedoria difundidas ao longo dos anos. Também por sua paciência, que não foi pouca.

Para concluir, agradeço às intuições que tornaram este projeto de pesquisa possível. Ao grupo Prysmian, pela colaboração científica, fornecendo as amostras dos cabos umbilicais e permitindo a utilização de resultados experimentais. Ao CNPq, pela ajuda financeira no primeiro ano de pesquisa, por meio do processo 133317/2014-5. E à FAPESP, pela ajuda financeira durante três anos subsequentes, por meio do processo 2014/22528-0.

Obrigado!

"All models are wrong, but some are useful."

(George E. P. Box, 1976)

ABSTRACT

SANTOS, C. C. P. **Structural Analysis of Flexible Pipes and Umbilical Cables: a bimaterial Finite Element modeling technique and a novel experimental approach using a Digital Image Correlation system.** 2019. 243 p. Thesis (Doctoral) -- Escola Politécnica da Universidade de São Paulo, São Paulo.

The Finite Element Method is a powerful and widespread tool for the structural analysis of flexible pipes and umbilicals. However, it is unfeasible to represent in detail all layers and components of a flexible pipe or umbilical cable in a Finite Element (FE) model, since the calculation time would be unrealistic. Moreover, consistent numerical analysis requires support from experimental results. In this context, this thesis presents numerical and experimental research options, as well as the development of new strategies for the design of FE models of flexible pipes and umbilicals. Using the commercial FE software ABAQUS, the text highlights the development of innovative techniques to represent helical layers, as well as the concept of a two-dimensional FE analysis, supported by analytical formulation. Complementing the numerical approaches, pioneer experimental techniques herein developed are presented, based on optical instrumentation through a Digital Image Correlation (DIC) system. An unconventional use of the DIC system enables the development of an experimental methodology to study umbilicals under crushing loads.

Keywords: Structural analysis. Flexible pipes. Umbilical cables. The Finite Element Method (FEM). Digital Image Correlation (DIC).

RESUMO

SANTOS, C. C. P. **Structural Analysis of Flexible Pipes and Umbilical Cables: a bimaterial Finite Element modeling technique and a novel experimental approach using a Digital Image Correlation system**. 2019. 243 p. Tese (Doutorado) -- Escola Politécnica da Universidade de São Paulo, São Paulo.

Na análise estrutural de tubos flexíveis e cabos umbilicais, o Método de Elementos Finitos se destaca como uma ferramenta poderosa e bastante difundida. Contudo, é inviável representar em modelos baseados em Elementos Finitos (EF) um tubo flexível ou um cabo umbilical com toda sua riqueza de detalhes, pois os tempos de cálculo seriam irrealizáveis. Além disso, análises numéricas consistentes precisam de respaldo de resultados experimentais. Neste contexto, esta tese apresenta linhas de pesquisa numérica e experimental. O desenvolvimento de novas estratégias para a concepção de modelos em EF de tubos flexíveis e cabos umbilicais é apresentado. Utilizando o software comercial ABAQUS, destacam-se técnicas inovadoras para representação das camadas helicoidais, bem como a obtenção de modelos EF bidimensionais, amparados por formulação analítica. Complementando as abordagens numéricas, técnicas pioneiras de análise experimental são apresentadas, baseando-se em instrumentação óptica com sistema de correlação digital de imagens. Utilizado de forma não convencional, o monitoramento óptico permite o desenvolvimento de metodologia experimental para estudo de cabos umbilicais sob carregamentos de crushing.

Palavras-chave: Análise estrutural. Tubos flexíveis. Cabos umbilicais. Método dos elementos finitos. Correlação digital de imagem..

LIST OF FIGURES

Figure 1.1	Typical deep water field layout using flexible lines.	35
Figure 1.2	Evolution in the Brazilian oil production since 1977.	36
Figure 1.3	Typical layers and internal components of flexible pipes (left) and umbilicals (right).	37
Figure 1.4	Flexible lines with thousands of meters long.	38
Figure 2.1	At left, cable with two helical layers modeled by Knapp (1979). At right, cable with n helical layers modeled by Lanteigne (1985). . .	45
Figure 2.2	Definition of stresses in the analytical model proposed by F��ret and Bournazel (1987).	46
Figure 2.3	Diagram of the rotary bending test rig used by Witz and Tan (1995). .	46
Figure 2.4	Loading conditions taken into account by Ramos Jr. (2001).	47
Figure 2.5	Modeled umbilical cross-section and respective experimental test from the case study presented by Pesce et al. (2010b).	48
Figure 2.6	A typical model generated with the approach proposed by S��vik and Bruaseth (2005).	49
Figure 2.7	Details from the pressure armor modeling and symmetric consideration from Sousa et al. (2002).	50
Figure 2.8	Left: cross-section with initial ovalization (dotted line) considered by the analytical model; right: example of experimental crushing result compared with bounds obtained with the analytical approaches proposed by Pesce et al. (2012).	51
Figure 2.9	FE model and a typical bending result presented by Dobson et al. (2006).	52
Figure 2.10	FE model of a STU, developed by Probyn, Dobson and Martinez (2007); with details for the location of internal components.	53
Figure 2.11	“Quasi-3D” model presented by Dixon and Zhao (2008). Forces in the helically wound components to represent 3D behavior.	54
Figure 2.12	Examples of models with multi-helical components, presented by Dixon and Zhao (2008).	54
Figure 2.13	Umbilical model by Le Corre and Probyn (2009), using beam and shell elements.	55
Figure 2.14	At left, detail from the “3D/Periodic” model. At right, mesh of the “3D/Explicit” (two wires were removed for visualization).	55
Figure 2.15	Eight layers FE model developed in Sousa (2005).	56

Figure 2.16	Example of stress results from the umbilical model proposed by Bai, Lu and Cheng (2014), with beam and shell elements rendered. . . .	57
Figure 2.17	At left, an overview of the crushing model presented in Malta et al. (2013); at right, an output from the numerical analysis: radial displacements.	58
Figure 2.18	FE model of a STU developed by Guttner (2015).	58
Figure 2.19	First model proposed by Gay Neto and Martins (2009), using axisymmetries to simplify the geometry.	59
Figure 2.20	Second model proposed by Gay Neto and Martins (2009), respecting the helical geometries.	60
Figure 2.21	One of the numerical case studies presented by Malta (2016). Overview of the FE model and example of result.	61
Figure 3.1	The 'thin layer' and 'zero-thickness' approaches.	66
Figure 3.2	Section of the polymeric layer as a function of radius and thickness only.	68
Figure 3.3	Partial representation of a modeled plastic layer.	68
Figure 3.4	Representation of the tensile armor from a bimaterial pipe.	69
Figure 3.5	Partitions size on the tensile armor.	70
Figure 3.6	Equivalent rectangular section of the helical layers.	71
Figure 3.7	Partial representation of the helical layer with square section wire and equivalent bimaterial pipe.	72
Figure 3.8	Size of elements in the pipe length according to the pitch of the helical layer.	74
Figure 3.9	Element size in the circumferential direction due to the number of tensile armor wires. Different colors for different layers.	74
Figure 3.10	FE mesh of the tensile armor.	75
Figure 3.11	Example of contact interference elimination at the beginning of the calculation.	76
Figure 3.12	Representation of boundary conditions: master node in the center to control the rigid body displacements at the extremities.	76
Figure 3.13	Overview of the model developed for the flexible pipe from Witz (1996). Model length was shortened, and some components were removed for visualization.	80
Figure 3.14	Details of the mesh from the model developed for the flexible pipe from Witz (1996).	80
Figure 3.15	Models based on Witz (1996); number of DOF; different model lengths.	81

Figure 3.16	Models based on Witz (1996); number of elements; different model lengths; darker colors to user-defined elements; lighter colors to elements generated for contact.	82
Figure 3.17	Models based on Witz (1996); pre-allocated RAM memory; different model lengths.	84
Figure 3.18	Models based on Witz (1996); load case (A); different model lengths; axial stiffness with ends free to rotate.	85
Figure 3.19	Models based on Witz (1996); load case (B); different model lengths; axial stiffness with ends prevented from rotating.	86
Figure 3.20	Models based on Witz (1996); load case (C); different model lengths; torsional stiffness with ends free to elongate.	87
Figure 3.21	Models based on Witz (1996); load case (D); different model lengths; torsional stiffness with ends prevented from moving axially.	88
Figure 3.22	Model based on Witz (1996); model length: 1000mm; different load cases; required number of iterations to complete calculations. . . .	89
Figure 3.23	Model based on Witz (1996); model length: 1000mm; different load cases; total wall-clock time to complete the calculations.	90
Figure 3.24	Overview of the model developed for the flexible pipe from Ramos Jr. et al. (2008, 2014). Model length shortened, and some components were removed for visualization.	91
Figure 3.25	Details of the mesh from the model developed for the flexible pipe from Ramos Jr. et al. (2008, 2014).	93
Figure 3.26	Models based on Ramos Jr. et al. (2008, 2014); number of DOF; different model lengths.	94
Figure 3.27	Models based on Ramos Jr. et al. (2008, 2014); number of elements; different model lengths; darker colors to user-defined elements; lighter colors to elements generated for contact.	95
Figure 3.28	Models based on Ramos Jr. et al. (2008, 2014); pre-allocated RAM memory; different model lengths.	96
Figure 3.29	Models based on Ramos Jr. et al. (2008, 2014); axial stiffness from load cases (A) to (D); different model lengths.	99
Figure 3.30	Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; required number of iterations to complete calculations.	100
Figure 3.31	Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; total wall-clock time to complete the calculations.	101

Figure 3.32	Overview of the model developed for the flexible pipe from Sousa et al. (2013). Model length was shortened, and some components were removed for visualization.	103
Figure 3.33	Details of the mesh from the model developed for the flexible pipe from Sousa et al. (2013).	103
Figure 3.34	Models based on Sousa et al. (2013); number of DOF; different model lengths.	104
Figure 3.35	Models based on Sousa et al. (2013); number of elements and pre-allocated RAM memory; different model lengths; at left, darker colors to user-defined elements and lighter colors to elements generated for contact.	105
Figure 3.36	Models based on Sousa et al. (2013); load case (A); different model lengths; axial stiffness with ends prevented from rotating.	107
Figure 3.37	Models based on Sousa et al. (2013); load case (B), at left, without internal pressure; load case (D), at right, with internal pressure of $1000psi$; torsional stiffness with ends prevented from moving axially.	108
Figure 3.38	Models based on Sousa et al. (2013); load case (C); different model lengths; traction – internal pressure ratio with ends prevented from rotating.	109
Figure 3.39	Model based on Sousa et al. (2013); model length: $1000mm$; different load cases; required number of iterations and total wall-clock time to complete calculations.	110
Figure 3.40	Model based on Witz (1996); with bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$	112
Figure 3.41	Model based on Witz (1996); without bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$	112
Figure 3.42	Model based on Sousa et al. (2013); with bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$	114

Figure 3.43	Model based on Sousa et al. (2013); without bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$	114
Figure 3.44	Model based on Witz (1996); with and without bimaterial; bending moment – curvature diagram; model lengths of $500mm$; internal pressure of $30MPa$	115
Figure 3.45	Experimental results from Witz (1996) compared to bending response of the model with and without bimaterial; model lengths of $500mm$; internal pressure of $30MPa$	116
Figure 3.46	Models based on Sousa et al. (2013); with and without bimaterial; bending moment – curvature diagram; model lengths of $500mm$; internal pressure of $30MPa$	117
Figure 3.47	Model based on Sousa et al. (2013); without bimaterial; inner tensile armor layers before applying bending load; color indicate minor principal strain in elements with artificial material.	119
Figure 3.48	Model based on Sousa et al. (2013); without bimaterial; inner tensile armor layers after applying bending load and unload; color indicate minor principal strain in elements with artificial material.	119
Figure 3.49	Model based on Sousa et al. (2013); without bimaterial; outer tensile armor layer after bending; geometric interference between elements from adjacent metallic wires.	120
Figure 4.1	Schematic representation of the hysteretic moment-curvature diagram.	125
Figure 4.2	(a) Specimens manufactured from a HDPE tube; (b) Specimens dimensions according to ISO6259-3; (c) Example of a specimen. . . .	127
Figure 4.3	Example of the experimental HDPE relaxation tests.	128
Figure 4.4	Example of the experimental HDPE creep tests.	128
Figure 4.5	Normalized Prony series calculated with Equation (4.1) using the coefficients presented in Table 4.1.	129
Figure 4.6	3D view from the ABAQUS FE model of the 4-inches flexible pipe; model length of $500mm$	129
Figure 4.7	Cross section of 4-inches flexible pipe; 2D view obtained with <i>PipeDesign</i> .	131
Figure 4.8	HBS diagram obtained using a Coulomb's friction model and HDPE outer sheath with elastic behavior.	132

Figure 4.9	HBS diagram with frictionless interlayer contact and HDPE with viscoelastic behavior. Sinusoidal loading with period T	133
Figure 4.10	HBS diagram combining frictional ($\mu = 0.15$) interlayer contact with HDPE viscoelastic behavior.	134
Figure 4.11	HBS diagrams considering sinusoidal load periods of $1s$, $10s$, $100s$ and $1000s$	135
Figure 5.1	Tensioner used during a riser laying operation and detail of a caterpillar cross section.	141
Figure 5.2	Cable 1 cross section: central electrical core (power + signal); 12 steel tubes; double armored (flat wires).	142
Figure 5.3	Cable 2 cross section: central electrical cable; 9 thermoplastic hoses + thermoplastic HCR hoses; quadruple armored (flat wires).	142
Figure 5.4	Cable 3 cross section: central electrical core (power + signal); 12 thermoplastic hoses + 3 thermoplastic HCR hoses; double armored (flat wires).	143
Figure 5.5	Modeling and analysis flowchart.	144
Figure 5.6	Detail of the in-plane FE mesh. Steel tube (gray) and HDPE external coating (yellow) with two elements across the thickness.	147
Figure 5.7	At top, mesh detail of the bimaterial layer; at bottom, continuous contact surfaces (red) obtained by using bimaterial layers.	147
Figure 5.8	Radial body forces applied over the helical components.	149
Figure 5.9	Layers of a typical High Collapse Resistance (HCR) hose.	150
Figure 5.10	FE in-plane model of Cable 1 cross section.	151
Figure 5.11	Ramberg-Osgood fitting used to model the super-duplex steel rheology.	152
Figure 5.12	Crushing load calculated for the critical section of Cable 1. Axial tension: $540kN$. Internal pressure in tubular components: $3000psi$	153
Figure 5.13	von Mises stress field along the steel tube section during crushing. V-shoes displacement: $3.4mm$. Axial tension: $540kN$. Internal pressure in tubular components: $3000psi$	154
Figure 5.14	von Mises stress field over Cable 1 cross section. V-shoes displacement: $3.4mm$. Axial tension: $540kN$. Internal pressure in tubular components: $3000psi$	154
Figure 5.15	FE in-plane model of Cable 2 cross section.	155

Figure 5.16	Crushing load calculated for Cable 2 critical section; red dashed line indicates the first yielding of the equivalent tube that represents the reinforced hose. Axial tension: $742.8kN$. Internal pressure in tubular components: $1015psi$	156
Figure 5.17	von Mises stress field over Cable 2 cross section. V-shoes displacement: $4.9mm$. Axial tension: $742.8kN$. Internal pressure in tubular components: $1015psi$	157
Figure 5.18	von Mises stress field on the equivalent tube section that represents the HCR carcass. V-shoes displacement: $4.9mm$. Axial tension: $742.8kN$. Internal pressure in tubular components: $1015psi$	158
Figure 5.19	von Mises stress field on the thermoplastic hoses sections during crushing. V-shoes displacement: $4.9mm$. Axial tension: $742.8kN$. Internal pressure in tubular components: $1015psi$	158
Figure 5.20	FE in-plane model of Cable 3 cross section.	159
Figure 5.21	Crushing load calculated for Cable 3 critical section; red dashed line indicates the first yielding of the equivalent tube that represents the reinforced hose. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$	160
Figure 5.22	von Mises stress field over Cable 3 cross section. V-shoes displacement: $5.3mm$. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$	161
Figure 5.23	von Mises stress field on the equivalent tube section that represents the HCR carcass. V-shoes displacement: $5.3mm$. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$	162
Figure 5.24	von Mises stress field on the thermoplastic hoses sections during crushing. V-shoes displacement: $5.3mm$. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$	162
Figure 5.25	Comparison between 2D and 3D crushing modeling approaches presented by Guttner, Santos and Pesce (2017).	164
Figure 5.26	Crushing forces obtained from the 2D and 3D modeling approaches for Cable 1.	164
Figure 6.1	Overview of the three specimens from Cable 1 used in the experimental crushing tests.	166
Figure 6.2	Overview of the six specimens from Cable 2 used in the experimental crushing tests.	166

Figure 6.3	Overview of the six specimens from Cable 3 used in the experimental crushing tests.	166
Figure 6.4	Speckled pattern applied on samples of all three umbilicals and the planar shoes; the metal block at left is used as a reference for null strain measurement.	167
Figure 6.5	Experimental crushing tests assembly.	168
Figure 6.6	Side view of a specimen set on the shoes. Notice the irregular contact between the upper shoe and the umbilical sample.	169
Figure 6.7	Digital Image Correlation: from speckled subsets, to displacement fields, to strain fields.	170
Figure 6.8	Selected zone to be analyzed (red) and respective subset size (yellow).	170
Figure 6.9	Time-history diagram of a 'low-loading' test.	172
Figure 6.10	Time-history diagram of a 'high-loading-directly' test.	172
Figure 6.11	Time-history diagram of a 'high-loading-cyclic' test.	172
Figure 6.12	Crushing load alignment positions of Cable 1.	173
Figure 6.13	Crushing load alignment positions of Cable 2.	174
Figure 6.14	Crushing load alignment positions of Cable 3.	175
Figure 6.15	Crushing force vs. bottom shoe displacement diagrams for specimen C1-2. Results from Tests T1-02 to T1-07.	177
Figure 6.16	Crushing force vs. bottom shoe displacement diagrams for specimen C1-3. Results from Tests T1-09 to T1-14.	177
Figure 6.17	Crushing force vs. bottom shoe displacement diagrams for specimen C2-1. Results from Tests T2-01 to T2-06.	178
Figure 6.18	Crushing force vs. bottom shoe displacement diagrams for specimen C2-3. Results from Tests T2-09 to T2-14.	178
Figure 6.19	Crushing force vs. bottom shoe displacement diagrams for specimen C3-4. Results from Tests T3-04 to T3-09.	179
Figure 6.20	Crushing force vs. bottom shoe displacement diagrams for specimen C3-5. Results from Tests T3-11 to T3-16.	179
Figure 6.21	Crushing force vs. bottom shoe displacement diagrams for all 'high-loading' tests with Cable 1. Results from Tests T1-01, T1-08 and T1-15.	181
Figure 6.22	Crushing force vs. bottom shoe displacement diagrams for all 'high-loading' tests with Cable 2. Results from Tests T2-07, T2-08 and T2-15 to T2-18.	181

Figure 6.23	Crushing force vs. bottom shoe displacement diagrams for all 'high-loading' tests with Cable 3. Results from Tests T3-01 to T3-03, T3-10, T1-17 and T1-18.	181
Figure 6.24	Hysteresis loops for all 'low-loading' tests with specimens C1-2 and C1-3. Results from Tests T1-02 to T1-07 and T1-09 to T1-14. . .	182
Figure 6.25	Radial stiffness of specimens C1-2 and C1-3, for each tested position (Tests T1-02 to T1-07 and T1-09 to T1-14).	182
Figure 6.26	Hysteresis loops for 'high-loading' tests with all three specimens from Cable 1. Results from Tests T1-01, T1-08 and T1-15.	183
Figure 6.27	Cross sections of Cable 1 at maximum crushing load; at left, experimental test; at right, 2D FE model.	183
Figure 6.28	Hysteresis loops for all 'low-loading' tests with specimens C2-1 and C2-3. Results from Tests T2-01 to T2-06 and T2-09 to T2-14. . .	184
Figure 6.29	Radial stiffness of specimens C2-1 and C2-3, for each tested position (Tests T2-01 to T2-06 and T2-09 to T2-14).	185
Figure 6.30	Hysteresis loops for 'high-loading' tests with all three specimens from Cable 2. Results from Tests T2-07, T2-08 and T2-15 to T2-18. . .	186
Figure 6.31	Cross sections of Cable 2 at maximum crushing load; at left, experimental test; at right, 2D FE model.	186
Figure 6.32	Hysteresis loops for all 'low-loading' tests with specimens C3-4 and C3-5. Results from Tests T3-04 to T3-09 and T3-11 to T3-16. . .	187
Figure 6.33	Radial stiffness of specimens C3-4 and C3-5, for each tested position (Tests T3-04 to T3-09 and T3-11 to T3-16).	187
Figure 6.34	Hysteresis loops for 'high-loading' tests with all three specimens from Cable 3. Results from Tests T3-01 to T3-03, T3-10, T3-17 and T3-18.	187
Figure 6.35	Cross sections of Cable 3 at maximum crushing load; at left, experimental test; at right, 2D FE model.	188
Figure 6.36	Cable 2. Horizontal displacement field (<i>mm</i>) obtained with the 2D FE methodology; crushing load at $50N/mm$	189
Figure 6.37	Cable 2. Vertical displacement field (<i>mm</i>) obtained with the 2D FE methodology; crushing load at $50N/mm$	189
Figure 6.38	Cable 3. Horizontal displacement field (<i>mm</i>) obtained with the 2D FE methodology; crushing load at $33N/mm$	190
Figure 6.39	Cable 3. Vertical displacement field (<i>mm</i>) obtained with the 2D FE methodology; crushing load at $33N/mm$	190

Figure 6.40	Cable 2. Horizontal displacement fields (mm) obtained with the DIC system. Crushing load of $50N/mm$ after 5 crushing load cycle. Orientations A to F.	192
Figure 6.41	Cable 2. Vertical displacement fields (mm) obtained with the DIC system. Crushing load of $50N/mm$ after 5 crushing load cycle. Orientations A to F.	193
Figure 6.42	Cable 3. Horizontal displacement fields (mm) obtained with the DIC system. Crushing load of $33N/mm$ after 5 crushing load cycle. Orientations A to F.	194
Figure 6.43	Cable 3. Vertical displacement fields (mm) obtained with the DIC system. Crushing load of $33N/mm$ after 5 crushing load cycle. Orientations A to F.	195
Figure 6.44	Curvature tracking with the DIC system. Test T1-08 with specimen C1-2. Tracking of the external perimeter.	197
Figure 6.45	Enumerated tubes in the cross section of Cable 1 for identification during the DIC analysis.	199
Figure 6.46	Curvature tracking with the DIC system. Test T1-08 with specimen C1-2. Tracking of Tube 12.	200
Figure 6.47	Evolution in six steps of the crushing load, curvature variation and plastic strain field; test T1-08 with specimen C1-2; results for Tube 12.	202
Figure 6.48	Enumerated hoses in the cross section of Cable 2 for identification during the DIC analysis.	204
Figure 6.49	Curvature tracking with the DIC system. Test T2-17 with specimen C2-5. Tracking of Hose 01.	205
Figure 6.50	Evolution in six steps of the crushing load, curvature variation and plastic strain field; test T2-17 with specimen C2-5; results for Hose 01.	206
Figure 6.51	Curvature tracking with the DIC system. Test T2-17 with specimen C2-5. Tracking of Hose 07.	208
Figure 6.52	Enumerated hoses for identification in the Cable 3 during the DIC analysis.	210
Figure 6.53	Curvature tracking with the DIC system. Test T3-01 with specimen C3-1. Tracking of Hose 01.	211
Figure 6.54	Evolution in six steps of the crushing load, curvature variation and plastic strain field; test T3-01 with specimen C3-1; results for Hose 01.	212

Figure 6.55	Curvature tracking with the DIC system. Test T3-01 with specimen C3-1. Tracking of Hose 08.	213
Figure 6.56	Flowchart of the proposed experimental crushing analysis of umbilical cables.	214
Figure B.1	Wires helically wound in opposite directions around a core.	233
Figure B.2	Positive laying angle α induces positive torque in the counterclockwise sense.	233
Figure C.1	Deformation of material subjected to bending moment M	235
Figure C.2	Deformation of a fully integrated, linear element subjected to bending moment M	235
Figure C.3	Deformation of a fully integrated, quadratic element subjected to bending moment M	236
Figure C.4	Deformation of a linear element with reduced integration subjected to bending moment M	236
Figure C.5	Variation of deformation gradient in (a) an incompatible mode (enhanced deformation gradient) element and (b) a first-order element using a standard formulation.	236
Figure C.6	Typical mesh and loading.	237
Figure C.7	Ring buckling problem.	238
Figure D.1	Reinforced hose finite element mesh.	242
Figure D.2	Ramberg-Osgood fitting function used to represent the 304L stainless steel behavior.	243
Figure D.3	Result for the crushing of a single reinforced hose.	243

LIST OF TABLES

Table 3.1	Workstation specifications.	78
Table 3.2	Flexible pipe properties from the case study of Witz (1996).	79
Table 3.3	Models based on Witz (1996); number of DOF; different model lengths.	81
Table 3.4	Models based on Witz (1996); number of elements; different model lengths.	82
Table 3.5	Models based on Witz (1996); pre-allocated RAM memory; different model lengths.	83
Table 3.6	Models based on Witz (1996); load case (A); different model lengths; axial stiffness with ends free to rotate.	85
Table 3.7	Models based on Witz (1996); load case (B); different model lengths; axial stiffness with ends prevented from rotating.	86
Table 3.8	Models based on Witz (1996); load case (C); different model lengths; torsional stiffness with ends free to elongate.	87
Table 3.9	Models based on Witz (1996); load case (D); different model lengths; torsional stiffness with ends prevented from moving axially.	88
Table 3.10	Model based on Witz (1996); model length: 1000mm; different load cases; required number of iterations to complete calculations. . . .	89
Table 3.11	Model based on Witz (1996); model length: 1000mm; different load cases; total wall-clock time to complete calculations.	90
Table 3.12	Flexible pipe properties from the case study of Ramos Jr. et al. (2008, 2014).	92
Table 3.13	Models based on Ramos Jr. et al. (2008, 2014); number of DOF; different model lengths.	93
Table 3.14	Models based on Ramos Jr. et al. (2008, 2014); number of elements; different model lengths.	94
Table 3.15	Models based on Ramos Jr. et al. (2008, 2014); pre-allocated RAM memory; different model lengths.	95
Table 3.16	Models based on Ramos Jr. et al. (2008, 2014); axial stiffness from load cases (A) to (D); different model lengths.	98
Table 3.17	Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; required number of iterations to complete calculations.	100
Table 3.18	Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; total wall-clock time to complete the calculations.	101

Table 3.19	Flexible pipe properties from the case study of Sousa et al. (2013). . .	102
Table 3.20	Models based on Sousa et al. (2013); number of DOF; different model lengths.	104
Table 3.21	Models based on Sousa et al. (2013); number of elements and pre-allocated RAM memory; different model lengths.	105
Table 3.22	Models based on Sousa et al. (2013); load case (A); different model lengths; axial stiffness with ends prevented from rotating.	106
Table 3.23	Models based on Sousa et al. (2013); load case (B), at left, without internal pressure; load case (D), at right, with internal pressure of $1000psi$; torsional stiffness with ends prevented from moving axially.	107
Table 3.24	Models based on Sousa et al. (2013); load case (C); different model lengths; traction – internal pressure ratio with ends prevented from rotating.	109
Table 3.25	Model based on Sousa et al. (2013); model length: $1000mm$; different load cases; required number of iterations and total wall-clock time to complete calculations.	110
Table 3.26	Main advantages and drawbacks of the bimaterial modeling technique.	121
Table 4.1	Prony series coefficients	127
Table 4.2	Layers of the Prysmian 4-inches flexible pipe.	130
Table 5.1	Admissible loads obtained for the numerical crushing of Cable 1, considering first yielding in the steel tubes, occurred at $0.8mm$ shoes displacement.	153
Table 5.2	Admissible loads obtained for the numerical crushing of Cable 2, considering first yielding in the equivalent rings, occurred at $1.8mm$ shoes displacement.	156
Table 5.3	Admissible loads obtained for the numerical crushing of Cable 3, considering first yielding in the equivalent rings, occurred at $1.0mm$ shoes displacement. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$	160
Table 6.1	Load cases parameters for Cable 1.	173
Table 6.2	Load cases parameters for Cable 2.	174
Table 6.3	Load cases parameters for Cable 3.	175

Table 6.4	Maximum curvature for each hose obtained with the DIC system during the 'high-loading' tests. Tests T1-01, T1-08 and T1-15 were performed with all three specimens from Cable 1.	199
Table 6.5	Maximum curvature for each hose obtained with the DIC system during the 'high-loading' tests. Tests T2-07, T2-08 and T2-15 to T2-18 performed with all six specimens of Cable 2.	204
Table 6.6	Maximum curvature for each hose obtained with the DIC system during the 'high-loading' tests. Tests T3-01 to T3-03, T3-10, T3-17 and T3-18 performed with all six specimens from Cable 3.	210
Table C.1	Normalized tip deflection of a cantilever beam (w_{fem}/w_{exact}).	238
Table C.2	Normalized eigenvalue pressure estimates (p_{fem}/p_{exact}). Bernoulli-Euler theory prediction: $p_{exact} = 51.71 KPa$ ($7.5 lb/in^2$).	239

LIST OF ABBREVIATIONS

ACSR	Aluminum Conductor Steel-Reinforced
DIC	Digital Image Correlation
DOF	Degrees of Freedom
EPUSP	<i>Escola Politécnica da Universidade de São Paulo</i>
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Method
FINEP	<i>Financiadora de Estudos e Projetos</i>
HBS	Hysteretic Bending Stiffness
HCR	High Collapse Resistant
HDPE	High-Density Polyethylene
LMO	Offshore Mechanics Laboratory
MBR	Minimum Bending Radius
MPC	Multi-Point Constraints
MPE	Minimum Potential Energy
RAM	Random Access Memory
SSD	Solid State Drive
STU	Steel Tube Umbilical
TDZ	Touch Down Zone

CONTENTS

1 INTRODUCTION	35
1.1 MOTIVATIONS AND GOALS	38
1.2 THESIS ORGANIZATION	39
2 LITERATURE REVIEW	43
2.1 ANALYTICAL APPROACHES	43
2.1.1 Distributed loads	44
2.1.2 Radial concentrated loads	49
2.2 NUMERICAL APPROACHES	51
2.2.1 Complete models	52
2.2.2 Crushing analyses	56
2.2.3 Core instabilities and bursting of flexible pipes	58
 PART I – FLEXIBLE PIPES	 63
3 A FE ALTERNATIVE MODELING STRATEGY	65
3.1 PREVIOUS ASSUMPTIONS	66
3.2 EQUIVALENT LAYERS	67
3.2.1 Polymeric layers	67
3.2.2 Tensile armor layers	68
3.2.3 Pressure armor layers	70
3.2.4 Interlocked carcass	71
3.2.5 Anti-wear and high-strength tapes	72
3.3 MODELING PARAMETERS	72
3.3.1 Type of Finite Element	73
3.3.2 Mesh size	73
3.3.3 Contacts	74
3.3.4 Boundary conditions	76
3.3.5 Model length	77
3.4 CASE STUDIES: COMPUTATIONAL PERFORMANCE WITH AXISYMMETRIC LOADS	77
3.4.1 Witz (1996)	78
3.4.2 Ramos Jr. et al. (2008, 2014)	91
3.4.3 Sousa et al. (2013)	100

3.5	INTERNAL PRESSURE AND BENDING	110
3.6	FINAL REMARKS	120
3.6.1	Further work	121
4	VISCOELASTIC	123
4.1	VISCOELASTICITY IN THE GLOBAL ANALYSES	123
4.2	HYSTERETIC BENDING BEHAVIOR	124
4.3	HDPE VISCOELASTICITY	126
4.4	CASE STUDY	127
4.4.1	HBS due to friction only	131
4.4.2	HBS due to viscoelasticity only	133
4.4.3	HBS due to friction and viscoelasticity	134
4.5	FINAL REMARKS	136
4.5.1	Further work	136
	 PART II – UMBILICAL CABLES	137
5	CRUSHING NUMERICAL IN-PLANE ANALYSIS	139
5.1	LAYING OPERATION AND LOADS	140
5.2	UMBILICAL CABLES UNDER ANALYSIS	141
5.3	NUMERICAL IN-PLANE MODELING METHODOLOGY	141
5.3.1	The two-dimensional approach	143
5.3.2	Hypotheses	144
5.3.3	Three-dimensional effects of the laying angle	145
5.3.4	Boundary conditions	145
5.3.5	Elements type and mesh	146
5.3.6	Contact formulation	146
5.3.7	Squeezing effect	148
5.3.8	Tensioner shoes	148
5.3.9	Reinforced hoses as equivalent pipes	149
5.4	CASE STUDIES: LAYING OPERATION CONDITIONS	150
5.4.1	Cable 1	151
5.4.2	Cable 2	155
5.4.3	Cable 3	159
5.5	FINAL REMARKS	163
6	CRUSHING EXPERIMENTAL ASSESSMENT USING DIC	165

6.1	EXPERIMENTAL ARRANGEMENT AND INSTRUMENTATION	165
6.1.1	Digital Image Correlation (DIC) processing	169
6.2	LOAD CASES	171
6.3	EXPERIMENTAL RESULTS	176
6.3.1	Radial stiffness	180
6.4	DISPLACEMENT FIELD ASSESSMENTS	188
6.5	CURVATURE TRACKING	196
6.5.1	Cable 1	198
6.5.2	Cable 2	203
6.5.3	Cable 3	209
6.6	FINAL REMARKS	214
6.6.1	Further work	215
7	CONCLUSIONS	217
	REFERENCES	219
	APPENDIX A – Viscoelasticity	231
	APPENDIX B – Adopted conventions	233
	APPENDIX C – Incompatible mode elements	235
	CANTILEVER BEAM WITH END SHEAR LOAD	237
	BUCKLING ANALYSIS OF RING UNDER EXTERNAL PRESSURE	238
	APPENDIX D – Reinforced hoses as equivalent pipes	241

1 INTRODUCTION

Flexible pipes and umbilical cables are complex structures composed by several polymeric and metallic layers, mounted in a concentric arrangement. As a well-known and proven concept in the offshore oil and gas industry, these flexible structures have been used to convey fluid and allow communication between the wellhead, on the sea floor, and the floating production units. Created circa forty years ago to operate in relatively shallow waters, such concepts have evolved substantially in order to be used for oil and gas extraction in deep, very deep and ultra-deep waters, such as the Brazilian pre-salt area or the Gulf of Mexico, which posed the challenge of adapting flexible pipes and umbilical cables to meet the mechanical requirements to deal with increasingly severe dynamic loads.

These structures have proved their efficiency over the last decades, especially in ultra-deep water fields, as pointed out by Novitsky and Sertã (2002). Figure 1.1 shows an example of an oil extraction system using several types of flexible lines, like flexible pipes and umbilical cables.

Figure 1.1: Typical deep water field layout using flexible lines.

Source: Extracted from Fergestad and Løtveit (2017).

In such cases, these flexible structures are subjected to several kinds of loads, such as internal pressure of the transported fluid, high external hydrostatic pressure applied by the water column, tensile or compression axial loads, torsion and bending generated by motions of the floating platform, the immersed weight, and the action of ocean currents. In addition to these loads, there are also effects that may damage the lines (e.g., corrosion, crushing, storage

and installation overbending, impact, abrasion and others).

Figure 1.2, from Petrobras, shows that the offshore oil extraction has been increasingly directed to deeper waters, which amplifies significantly not only the external pressure acting on the lines but also all other loads imposed on the structure.

Figure 1.2: Evolution in the Brazilian oil production since 1977.

Source: Petrobras (2016).

In order to resist this wide variety of structural loads, these lines are composed of several reinforcement and protection layers arranged concentrically, each with specific functions. Figure 1.3 shows typical configurations for the layers and internal components that may be present in the composition of a flexible pipe or an umbilical cable. Many studies devote entire chapters to some very detailed presentations of both structures and their different layers and components; Ramos Jr. (2001), Provasi (2013) and Fergestad and Løtveit (2017) offer exceptionally good descriptions of these two categories of flexible lines. Those studies present the typical layers and components that can normally be found in flexible pipes and umbilical cables, with illustrations and descriptions of their respective functionalities.

Figure 1.3 illustrates the geometric and functional arrangements of a typical flexible pipe and some umbilical cables. This arrangements result in a set that combines high flexibility with high resistance to radial and axial tensile loads. In this context, flexible pipes and umbilical cables are considered good engineering solutions, being widely used in offshore oil and gas exploitation fields.

The modular composition with individual layers and components allows each structure to be

Figure 1.3: Typical layers and internal components of flexible pipes (left) and umbilicals (right).

Source: Left image – Extracted from Fergestad and Løtveit (2017). Right Image – Adapted from Prysmian (2016).

adapted to specific environments and particular tasks. The helical layer sets combine a large resistance to pressure with a low flexural one (a typical pipe with 8 inches of internal diameter can bend to a radius of less than $2m$).

Each line is designed according to the needs of the production field. Figure 1.3 (above, right) depicts some umbilical cables with different types of internal components. Depending on the environmental conditions the umbilical will operate under, it can be designed with one or two pairs of tensile armors (the same goes for flexible pipes). Its internal components are planned according to the equipment to be operated and powered on the seabed. These conditions define the number of hoses, power cables, optical fiber cables and steel tubes, for example. Additionally, the voids in the umbilical's cross-section are occupied by polymeric components called “fillers”.

Concerning the flexible pipes, the number of layers depends on the imposed loads, with different geometries and number of wires. In addition to these conditions, the properties of the oil (e.g. pressure, temperature, viscosity and composition) must also be taken into account for a proper and safe design.

Furthermore, these lines are fabricated in lengths of thousands of meters and, due to their flexibility, they can be easily transported and installed (an entire line a few kilometers long can be stored in a single spool – an example can be seen in Figure 1.4).

However, it is important to be specially careful when designing these structures. Failures generated by fatigue, corrosion under stress or intense wear of the layers and internal components are more likely to happen when these structures operate in deep and ultra-deep waters. An accident can give rise to equipment damages, production interruptions that may cost millions of dollars, massive environmental damage and even loss of human lives. Therefore, the need to understand the structural behavior of the pipes is evident.

Figure 1.4: Flexible lines with thousands of meters long.

Source: Technip (2016).

1.1 MOTIVATIONS AND GOALS

As flexible pipes and umbilical cables are used in progressively greater water depths and, consequently, more severe environmental conditions, they are subjected to more intense loads. For this reason, it is crucial to employ an accurate structural analysis to design these lines.

This analysis consists in defining the distribution of internal efforts each layer will be able to resist, depending on the applied loads. Once the distribution is known, the structure's critical points can be addressed, allowing, for example, the determination of fatigue life of a pipe or a cable. This kind of analysis also enables the evaluation of the structure's stiffness (axial, radial, in torsion and bending), which are important features for detailed global dynamics and local structural analyses, as well as for a stability analysis.

The complexity of these lines and the involved loads led to the development of many models of structural analysis, based on analytical and semi-analytical approaches (some examples will be discussed in the next chapter). Although the analytical methods are usually more convenient in terms of the efficiency of design procedures, they require more restrictive assumptions. On the other hand, the Finite Element (FE) models are powerful tools to complement such analytical models.

The Finite Element Method (FEM) has a broader application, but it involves some other difficulties, such as the correct representation of geometries, applied loads and boundary conditions; the appropriate representation of contacts and even the mechanical behavior of materials that compose the structure. Analogously to the results obtained from analytical models, the quality of a FEM analysis also depends directly on the hypotheses and assumptions made. Therefore, it is necessary to find an appropriate methodology to simplify the numerical model, while preserving the quality of results as well as possible.

Once the methodology for a structural analysis (analytical or numerical) has been well established, it is desirable to validate and verify results. For this purpose, it is essential to work with experimental data from real cases. In the literature we are often presented structural analysis results without any correlation with experimental data, giving rise to doubts about the validity of such works. A structural analysis can become more reliable if validated by comparisons with experimental data, to ensure the representativeness of results.

In this context, this thesis proposes the development of innovative structural analyses for flexible pipes and umbilical cables, proposing innovative numerical modeling through the FEM, always corroborated by experimental results. The numerical results obtained during the thesis are firstly presented and confronted with analytical, numerical or experimental results from the literature. Moreover, in a second part of the text concerning umbilical cables, optical instrumentation and the FEM are employed to complement experimental campaigns herein developed. This thesis is composed by numerical and experimental structural analyses, using new modeling techniques and novel experimental approaches at once.

In fact, new experimental approaches are performed and developed with innovative methodologies to assess the crushing of umbilical cables. These techniques were developed with the purpose of capturing some phenomena that, to the extent of the author's knowledge, were never before monitored on umbilical cables. With the aid of optical instrumentation, unprecedented results were obtained and compared with numerical analyses.

Summarizing, the main goals of this thesis involve establishing new, reliable and robust numerical modeling methodologies and experimental approaches for flexible pipes and umbilical cables. The subjects covered herein focus on certain topics, which are listed in the following section. For the development of the numerical models, the FE commercial software ABAQUS was employed. It is a well-known computational tool within this research field.

1.2 THESIS ORGANIZATION

For the sake of organization, the content of this thesis was structured in seven chapters. After this opening chapter, Chapter 2 presents a literature review of some existing models developed for the structural analysis of flexible pipes and umbilical cables. The reviewed works are presented according to their main approach: analytical or numerical. For each section, the studies were reviewed in chronological order. The review continues throughout the remainder of the thesis, with specific studies mentioned at the beginning of each chapter.

After the literature review, the text is divided into two major parts, one focusing on flexible pipes and the other on umbilical cables. Part I is composed of two chapters. The first chapter of this part, Chapter 3, analyzes three different FE models of flexible pipes. The

author developed these models using modeling techniques that are unconventional for this kind of problem. During the three case-studies presented, axial, torsion and bending loads are simulated and studied. The computational performance of the FE models are evaluated first discarding then taking into account some modeling simplifications. The structural response of the resultant models are correlated with some analytical, experimental and/or numerical results from the literature. The goal of this chapter is to present some modeling techniques developed by the author aiming to simplify the design and construction of a FE model of flexible pipes, as well as its numerical complexity, without compromising their structural analysis capacity.

After setting this particular modeling methodology for flexible pipes, the study presented in Chapter 4 provides an assessment of the viscoelastic effect of the outer polymeric layer. This chapter demonstrates (1) the use of the stated modeling methodology and (2) how the time-dependent rheology impacts the structural bending behavior of the pipe. Although it is well-known that the bending stiffness of flexible pipes is strongly dominated by the external polymeric layer, existing numerical and analytical models do not take into account the viscoelastic mechanical behavior, which characterizes most of the polymeric external layers. This chapter addresses this particular point numerically. The first part of the thesis is concluded by emphasizing the response of flexible pipes to bending loads and reflecting on the reliability of the new numerical modeling techniques.

The second part of the thesis presents the studies with umbilical cables. Chapter 5 presents an innovative numerical methodology to study the response of umbilicals to crushing loads, using a two-dimensional modeling approach. The first purpose of the developed in-plane FE models is to predict stress and strain fields on the umbilical cable components under crushing loads. Such loads are outcomes from the laying operation and comprise the caterpillar shoes load and the squeezing effects (associated not only with the tensile armors but also with other helical structural components subjected to tension). The referred model comprises a joint analysis using a two-dimensional Finite Element Analysis (FEA) fed by analytical equations, to represent three-dimensional effects. The combined analytical-numerical approach is meant to obtain results efficiently, without requiring extraordinary computational capacity. The chapter presents and discusses modeling hypotheses and methodology, describing how three-dimensional effects and interactions among cable components are treated. It also makes use of three case studies with umbilical cables.

Proceeding with the crushing analysis on umbilicals, Chapter 6 presents an innovative experimental analysis assisted by optical tracking techniques. Focusing on monitoring the integrity of some tubular components during the laying operation, the special crushing test setup assesses displacement fields on cross-sections of the umbilicals. Using a high resolution Digital Image Correlation (DIC) system, experiments with the same three umbilical cables presented

in Chapter 5 are carried out in a $250kN$ mechanical tests rig. Besides measuring the components' displacements with the DIC system, the optical measurements are also used to track the profile of each tubular component during the experimental tests, so that FE models and experimental tests can be compared. Numerical-experimental correlations are presented, for all three umbilical cables, comparing the DIC-measured data with the FE models detailed in the previous chapter.

Finally, a closing chapter on page 217 discusses results obtained and the general conclusions supported by this study, besides presenting some suggestions for future work. Additionally, appendices are provided to supplement certain text points.

2 LITERATURE REVIEW

Flexible pipes and umbilicals have been studied for at least four decades. Over those years, many different approaches for structural analysis have been developed based on analytical and numerical models. Model-based analysis should be supported by experimental results so that experimental methods may complement analytical and numerical calculations, serving to test hypotheses as well as paradigms. If a structural analysis obtains consistent results from both approaches, it can be considered effective and representative of reality.

Nowadays, FE models prevail in the structural analysis of flexible pipes and umbilical cables. The need to keep up with technological challenges and the ever-increasing computational processing power pushed structural modeling to evolve from consistent analytical-numerical approaches to full nonlinear Finite Element Analysis (FEA), where material properties and geometric nonlinearities may be properly taken into account. Numerical models are currently the norm due to the combination of computational processing power available and the high complexity of flexible pipes and umbilicals.

This literature review starts by listing some pioneering studies in structural mechanics in the offshore industry, conducted when FEM were not yet widespread. The text presents some approaches based on analytical and analytical-numerical models, where the analytical formulations are used in combination with numerical methods. Subsequently, this chapter presents works based on numerical models, mainly FEA developed with commercial software, as well as some studies based on their own FE formulations.

2.1 ANALYTICAL APPROACHES

The early days of structural analysis of flexible pipes and umbilical cables consist mostly of analytical and experimental studies, borne from the need to understand the structural behavior of these flexible structures, combined with the limited computational processing power available at the time. As a result, analytical approaches were the first ones to be explored further, providing a broader understanding of what was being studied and serving as good benchmarks for results obtained by other analyses (experimental or numerical).

As in all theoretical approaches, analytical formulations are always based on assumptions, hypotheses and simplifications. These factors restrict the validity of the models and cannot be forgotten in order to prevent their models from being used in situations where the formulations are not valid. Therefore, the basis of this traditional approach ends up being one of its major drawbacks, restricting its use. It is worth mentioning that numerical and experimental approaches have the same weakness, but their simplifications are usually less restrictive.

Analytical models for flexible pipes and umbilical cables are normally very complex, reflecting the complexity of these structures. Even with the adopted simplifications, the models that estimate the structural behavior of a flexible pipe or an umbilical end up with a system of nonlinear algebraic equations. Generally, such systems are solved numerically with a computer. For this reason, these approaches are typically considered analytical-numerical.

The constitutive equations for these analytical models come from the classical Theory of Elasticity. With a simpler geometry, the plastic sheaths are normally modeled by thin or thick-walled cylinder formulations. The real challenge in the analytical development concerns formulations for the helical components. The proposed approaches are always based on equilibrium equations, geometric compatibilities and constitutive relations. For example, many models build from the beam equations found in Timoshenko and Gere (2009), and thus work with this theory's assumptions and hypotheses. On the other hand, the helically wound components found in flexible lines are often modeled by the geometrically nonlinear equations for curved beams, by Clebsch-Kirchhoff, as presented in Love (1944). Such equations allow the analysis of a beam, whose axis can be described by any curve in space, subjected to a generic load.

With this literature review, it was verified that the analytical models for flexible pipes and umbilicals are developed according to their application. The models that enable analyses with tension, pressure, torsion and bending loads cannot consider radial concentrated loads, such as crushing conditions. As this thesis deals with these two categories of loading, the review presented herein for analytical models is divided into distributed and radial concentrated load studies.

2.1.1 Distributed loads

The first analytical models with helically wound components date back to the 1970's, in Costello's work. Three examples containing helical springs and twisted wire cables can be found in the following studies: Phillips and Costello (1973), Blanco and Costello (1974) and Costello (1977). Over the years, a less restrictive and more complete analytical model was developed by Knapp (1979). Using the principle of Minimum Potential Energy (MPE), the author analytically calculates the coefficients of the stiffness matrix of cables with helical wires of circular section. The analytical model developed considers tension and torsion loads on a cable with two helical layers and a compressible core (Figure 2.1, left). The strains are calculated using geometric compatibilities, considering the hypothesis of small displacements and that the planar cross-sections of the undeformed cable remain plane after deformations.

After Knapp's seminal study, other similar cable models considering wires with helically

wound geometry were developed. Lantaigne (1985) presents a model that studies the mechanical behavior of an Aluminum Conductor Steel-Reinforced (ACSR) cable. The proposed model can consider n helical layers, as shown in Figure 2.1 (right). It considers combined loads of tension, torsion and bending, estimating the structure's stiffness matrix with its nine coefficients (axial, torsional, flexural and coupling parameters).

Figure 2.1: At left, cable with two helical layers modeled by Knapp (1979). At right, cable with n helical layers modeled by Lantaigne (1985).

Source: Knapp (1979) and Lantaigne (1985).

Moving on to analytical models of flexible lines specifically designed for offshore applications, Fret and Bournazel (1987) presents an analytical model to obtain the distribution of efforts on a flexible pipe's layers. The theoretical approach estimates stresses and contact pressures between layers due to axisymmetric loads. The definition of stresses for the helical wires and for the plastic sheaths can be seen in Figure 2.2. In addition, Fret's model also estimates contact efforts and slipping between layers when the pipe is subjected to a bending load.

Witz and Tan (1992a) develop a considerably more complex nonlinear analytical model to predict the structural behavior of three different types of flexible lines for offshore application, subjected to tension-torsion combined loads. According to the authors, the proposed methodology can be applied to flexible pipes, umbilicals and marine cables. The model is based on the theory of thin-walled tubes to model the polymeric sheaths and on Love's equations to model the helical layers. The authors present case studies for all three aforementioned structure types comparing analytical and experimental results for tension and torsion loads. Witz and Tan (1992b) follow the study with an analytical model capable of predicting the flexural behavior of the same three structures. As in the first paper, the responses of the analytical

Figure 2.2: Definition of stresses in the analytical model proposed by Féret and Bournazel (1987).

Source: Féret and Bournazel (1987).

model are compared with experimental data. They present diagrams illustrating the bending moment-curvature relationship of each flexible structure, and there is great compatibility between analytical and experimental data. More results using this same approach are presented by the same authors in Witz and Tan (1995). In this paper, an umbilical's structural bending behavior is assessed by means of an analytical model and experimental methods, with good correlations. For the experiments, cyclic loads are applied using the rotary bending rig illustrated in Figure 2.3.

Figure 2.3: Diagram of the rotary bending test rig used by Witz and Tan (1995).

Source: Witz and Tan (1995).

An interesting case study with several analytical results can be found in Witz (1996). In this work, the author organizes calculations from ten different institutions for the analysis of the same flexible pipe. The theoretical results from each source are presented and compared with experimental data. All results from different sources broadly agree with the experimental data for the axial-torsional structural response. However, very different results were obtained for the flexural structural response.

An even more complete analytical model is developed by Ramos Jr. (2001), in his doctoral thesis. The presented methodology proposes analytical models for both flexible pipes and umbilical cables with a generic configuration. Tension, torsion, internal and external pressure and bending loads are taken into account, and may be considered individually or combined (Figure 2.4). Additionally, this method is also based on Love's equations for modeling the helically wound layers and the author evaluates the thin-walled tube hypothesis for the plastic sheaths as well. Throughout the study, Ramos explains in detail all adopted assumptions and simplifications, leaving no doubts to the reader about the comprehensiveness of the model. The final modeling comprises consistent analytical relations for each layer of the flexible line, resulting in a system with several algebraic equations. The system's solution allows the calculation of stresses and strains in all layers, interlayer pressures and gaps, as well as the estimation of the structure's stiffness matrix. Finally, the author compares the results obtained by his methodology with experimental results from different cables and pipes.

Ramos' thesis originated an article (Ramos Jr. and Pesce (2004)) that describes a study with a flexible pipe model subjected to combined loads (axisymmetric and bending loads). This article emphasizes that the bending stiffness results in a "full-slip" condition when the slippage between the layers is considered frictionless. Ramos' analytical model is also the basis of a software called *PipeDesign*¹, which was used in the calculations of some analytical results of the present thesis. The software was developed to assist the design of flexible pipes, working as a local analysis tool and providing initial results for structural verification due to several loads. The program calculates analytically the distribution of mechanical efforts, pressures and displacements in all layers of a given section.

Figure 2.4: Loading conditions taken into account by Ramos Jr. (2001).

Source: Ramos Jr. (2001).

¹The *PipeDesign* software was jointly developed and it is the co-property of Prysmian Cables and Systems and the Offshore Mechanics Laboratory (LMO) of *Escola Politécnica da Universidade de São Paulo* (EPUSP).

A few years later, the methodology proposed by Ramos Jr. (2001) was extended with an analytical-experimental study by Pesce et al. (2010c), focused on umbilical cables. This study was presented in two papers, one focused on the development of the analytical model and the other on the comparisons with experimental results. In the first article, identified as Part I, the analytical model is introduced along with considerations about the algorithm for the numerical solution of the analytical system. The generic model, implemented in UTILFLEX® (a software developed by the authors), allows the consideration of layers with different components, diameters and thicknesses. The resultant model calculates the stresses at critical points of these internal components. This first part of the study summarizes a technical report written by Pesce et al. (2008c), in the context of a Research & Development (R&D) project, focused on umbilical cable design tools and structural analysis methodologies, funded by a partnership between *Financiadora de Estudos e Projetos* (FINEP) and the Prysmian Group.

The other part of the study was published in a second paper (Pesce et al. (2010a)) which shall be referred to as Part II. It reports experimental tests performed with an umbilical cable in order to compare their results with those obtained with the analytical approach. Figure 2.5 illustrates the analyzed umbilical's cross-section and the experimental setup. The case study presents a good correlation between analytical and experimental results, indicating that the proposed model is consistent. This second paper is based on another technical report from Pesce et al. (2008b), in the context of the same R&D project.

Figure 2.5: Modeled umbilical cross-section and respective experimental test from the case study presented by Pesce et al. (2010b).

Source: Pesce et al. (2010b).

Another similar study developing a nonlinear formulation for umbilical cables and flexible pipes was conducted by Custódio and Vaz (2002). The authors present a model to predict the response of the aforementioned flexible lines to the application of axisymmetric loads

or displacements. Material nonlinearities, gap formation, lateral contact between wires and changes in the curvature of the wires are taken into account through an iterative solution method. The effectiveness of the nonlinear approach is illustrated by two case studies, one for each type of structure.

Sævik and Bruaseth (2005) continue the study proposed by Custódio and Vaz (2002) with an improved model using FE techniques. They develop FE formulations to model the components of an umbilical cable. Figure 2.6 presents a typical model generated with their methodology. The authors successfully correlate the outcomes from the analytical-numerical method with the experimental tests and conclusions from Custódio and Vaz (2002).

Figure 2.6: A typical model generated with the approach proposed by Sævik and Bruaseth (2005).

Source: Sævik and Bruaseth (2005).

Recently, the structural behavior of flexible pipes subjected to axisymmetric loads (tension, torsion, internal and external pressure) has been readdressed analytically by Ramos Jr. and Kawano (2015), under a rigorous first-order variational approach.

2.1.2 Radial concentrated loads

There are analytical approaches that take radial loads into account, using models that represent the cross-sections of flexible pipes. These models consider the cross-sections subjected to crushing loads that originate during the installation procedures. To the best of the author's knowledge, analytical models studying crushing loads address only flexible pipes. Models representing umbilical cables were not found.

The first reviewed study that discusses concentrated loads with an analytical approach is the paper by Sousa et al. (2002). In this study, the authors propose an analytical model to assess the stresses in the layers of a flexible pipe subjected to crushing loads. The study uses

a FE model to validate the analytical approach. However, no correlation with experimental results is performed. Figure 2.7 illustrates the cross-section taken into account, as well as the symmetric considerations used to simplify the model.

Figure 2.7: Details from the pressure armor modeling and symmetric consideration from Sousa et al. (2002).

Source: Sousa et al. (2002).

One year later, Martins, Pesce and Aranha (2003) presented a study focused on the interlocked carcass of flexible pipes. The analytical model proposed therein was created in the context of a research project developed between the *Escola Politécnica da Universidade de São Paulo* (EPUSP) and *Pirelli Cabos S.A.* This study was first reported in Martins, Pesce and Aranha (1996). The analytical model consists of an equivalent pipe that reproduces the structural behavior of the interlocked carcass under radial loads. This analysis achieves considerable gains: the simplicity of the model and a good correlation with experimental data. As a result, many subsequent numerical models have adopted this equivalent pipe model to simplify geometries. This model has been improved by the authors over the years, as shown in Pesce et al. (2010b) and Franzini et al. (2011). The following studies improve the analytical model considering the pressure armor layer and effects from initial ovalizations.

In addition, Pesce et al. (2012) increases the complexity of the analytical approach even further by adopting an elastic-plastic behavior to consider nonlinearities from the metallic carcass and pressure armor rheologies. Presenting a case study, this analysis uses experimental results to prove the efficiency of the proposed analytical approach. Figure 2.8 illustrates the cross section considered by the author in the analytical formulation, which takes into account initial ovalizations (left), and an example of case study results (right). The proposed approach establishes an upper and a lower bound, presenting good correlation with experimental data.

Figure 2.8: Left: cross-section with initial ovalization (dotted line) considered by the analytical model; right: example of experimental crushing result compared with bounds obtained with the analytical approaches proposed by Pesce et al. (2012).

Source: Pesce et al. (2012)

2.2 NUMERICAL APPROACHES

With the increase of computational power and the possibility of processing large FE calculations, the numerical approaches based on FEA became an alternative to the analytical models. Basic references presenting the fundamentals of the FEM can be found, for example, in Bathe (1996) or Cook et al. (2001). The concepts presented in these works provide the base of all commercial software currently available.

Regarding the different numerical approaches used to model flexible pipes and umbilicals, some studies develop their own tools and even specific FE formulations to address the subject (e.g., Sævik (1993) and Provasi (2013)). On the other hand, most researchers and engineers use commercial FE software with a wide application, such as the ABAQUS software, which was used to develop all numerical models used in this thesis.

The next subsections present reviews of some existing works on structural analysis of flexible lines using FE modeling, sorted into three categories. The structural modeling approaches mentioned in this literature review were categorized by studied phenomenon. We begin with “Complete models”, which are designed to study the response of a structure under axisymmetric loads, torsion and bending. Normally, they represent all layers and components using different types of FE, such as beams (1D), shells (2D) or solid elements (3D). A second subsection is dedicated to the crushing analysis. The FE models developed to study this installation load usually work with simplifying assumptions, like simpler geometries and symmetries. These considerations are used to produce equivalent conditions, in order to investigate local behaviors that present greater complexity. Closing this FE modeling review, the last subsection presents

some models developed to study instability problems. All the scientific productions herein listed are the basis of this research project.

2.2.1 Complete models

Complete models represent a segment of the given pipe or cable and model all layers and internal components. A case study involving this approach and using a commercial FE software can be found in Dobson et al. (2006). The authors develop a three-dimensional FE model of a Steel Tube Umbilical (STU) with ABAQUS. Figure 2.9 illustrates it, as well as some bending results. The steel tubes are modeled using shell elements, with an isotropic elastoplastic material behavior; electrical cables are represented by solid elements with composite properties; tapes, by shell elements; fillers and outer-sheaths are modeled using solid elements, with isotropic elastic material behavior. Unfortunately, the rheological models considered are not mentioned. The material coefficients are defined via experimental tests, which are also not provided. The results from the FE model are compared to full-scale test data, with the umbilical subjected to tension, bending and crushing loads. There is a good correlation between the numerical results and the experimental tests.

Figure 2.9: FE model and a typical bending result presented by Dobson et al. (2006).

Source: Dobson et al. (2006).

In a very similar study, Probyn, Dobson and Martinez (2007) present a second case study with another umbilical cable. The position of the electrical cables and the steel tubes can be seen in Figure 2.10. The modeling techniques seem to be the same proposed in Dobson et al. (2006). Comparison with full-scale tests presents a good correlation in tension, bending and crushing.

Dixon and Zhao (2008) present a discussion on the 3D modeling process of umbilical cables,

Figure 2.10: FE model of a STU, developed by Probyn, Dobson and Martinez (2007); with details for the location of internal components.

Source: Probyn, Dobson and Martinez (2007).

demonstrating its importance to calculate the fatigue life of these structures. At first, the authors present a two-dimensional modeling approach, identified as “Quasi-3D”. This approach allows the creation of in-plane models of the umbilical cross-section and the representation of the 3D helical behavior with analytical equations. Unfortunately, the paper does not give details about this methodology, briefly illustrated in Figure 2.11. Following, the authors also mention a sophisticated in-house package called 3DUST (3D Umbilical Simulation Toolkit). Coupled with the ABAQUS software, the mentioned toolkit is capable of generating three-dimensional models of umbilicals with multi-helical components. Figure 2.12 illustrates two examples. The discussion ends with comparisons of some fatigue life calculations using both “Quasi-3D” and “Full-3D” numerical approaches.

Another FE modeling approach is presented by Le Corre and Probyn (2009). The authors use ABAQUS to evaluate a segment of an umbilical cable, representing the layers with different types of FE (shell, beam and membrane elements). Figure 2.13 illustrates the model. Axial traction and cyclic bending loads are applied to the structure. The kinematics of sliding obtained numerically are compared to analytical model results, showing a very good correlation. Closing the study, the FE model is validated against full-scale experimental data.

In the same year, Bahtui, Alfano and Bahai (2009) present a complete model of a flexible pipe with six layers. The interlocked carcass is modeled as an equivalent orthotropic pipe, with first-order solid elements. In addition, the tensile armor layers and the anti-wear tapes are represented by first-order shell elements. The evaluated loads are combinations of cyclic bending with differential internal-external pressures. Although the hypotheses and assumptions adopted are not clearly explained, the numerical results show a good correlation with an

Figure 2.11: “Quasi-3D” model presented by Dixon and Zhao (2008). Forces in the helically wound components to represent 3D behavior.

Source: Dixon and Zhao (2008).

Figure 2.12: Examples of models with multi-helical components, presented by Dixon and Zhao (2008).

Source: Dixon and Zhao (2008).

analytical model.

Leroy et al. (2010) compare three different models of the same flexible pipe. The goal is to evaluate the structure under axial traction. The first model, named “Life6”, is analytical. The second one, numerical and developed in ABAQUS, is called “3D/Periodic”. This model focuses on the tensile armors, whose wires are represented by first-order hexahedral solid elements. Periodic conditions are used to represent the flexible pipe with a model a few centimeters long (Figure 2.14, left). Finally, the third model does not consider the same restrictive assumptions adopted in “3D/Periodic”. The new flexible pipe model, called “3D/Explicit”, is more realistic, allowing the representation of a greater length of the pipe and eliminating the need for the periodic hypothesis used in “3D/Periodic” (Figure 2.14, right). Nevertheless, the disadvantage of this last model lays on the heavier numerical computing performance required.

Another well-known complete model of flexible pipe can be found in three papers: Merino

Figure 2.13: Umbilical model by Le Corre and Probyn (2009), using beam and shell elements.

Source: Le Corre and Probyn (2009).

Figure 2.14: At left, detail from the “3D/Periodic” model. At right, mesh of the “3D/Explicit” (two wires were removed for visualization).

Source: Leroy et al. (2010).

et al. (2009), Merino et al. (2010), and Sousa et al. (2009). The model is based on previous works by Cruz (1996) and Sousa (2005). The modeled flexible pipe, illustrated in Figure 2.15), is composed of eight layers of equivalent cylinders, connected by spring elements. This model, developed with ANSYS, was used with multiple purposes by its authors. In Merino et al. (2009), the main goal is to evaluate the tension-torsion coupling effect on the pipe. In Sousa et al. (2009), the influence of a wire rupture on the distribution of tension in the tensile armors is analyzed. Finally, in Merino et al. (2010) the same model was used to analyze the deformations of the flexible pipe under pure torsion. In all three studies, the model was compared to experimental results, mostly with good correlations, barring a few exceptions.

Another three-dimensional FE modeling for umbilical cables can be found in Sævik and Gjøsteen (2012). In this study, the authors chose to model the helical components (tensile armors and tubes) with a thin and slender beam formulation. The formulation developed takes into account the contact between components using contact elements with 2 or 3 nodes. The

Figure 2.15: Eight layers FE model developed in Sousa (2005).

Source: Translated from Sousa (2005).

model also considers material nonlinearities, besides gaps and frictions between individual bodies. The paper indicates that the model can also take into account contacts with external structures, but the case study presented considers only the response to axisymmetric loads. The numerical results agree well with experimental data.

Bai, Lu and Cheng (2014) also use beam and shell elements to develop another FE model of an umbilical cable, using the commercial software ABAQUS. Figure 2.16 illustrates the umbilical model with rendered elements. The simulations consider tension-torsion loads. In the same study, the authors extend the analytical approach from Knapp (1979) to correlate numerical and analytical results. The theoretical approach is improved in order to consider different materials and geometries of helical components in a same layer, as well as Poisson effects in the helical and central components. The results from both approaches present good correlation.

2.2.2 Crushing analyses

A second category of studies focuses on crushing modeling. The FE models addressing this subject normally consider a partial representation of the structural layers and internal components. The models are smaller and do not reproduce a substantial length of the line, as the phenomenon studied is locally concentrated. These models are created based on strategies to represent the different parts of the line with equivalent structural elements, and usually take advantage of possible existing symmetries in the problem.

A study by Gay Neto et al. (2009) presents an axisymmetric version of an interlocked carcass model, also considering a polymeric outer sheath. This study replicates the crushing

Figure 2.16: Example of stress results from the umbilical model proposed by Bai, Lu and Cheng (2014), with beam and shell elements rendered.

Source: Bai, Lu and Cheng (2014).

loading conditions during the laying operation, applied by the caterpillar shoes of a tensioner. The results of this model are compared with experimental data, being fully satisfactory for the carcass deformation in linear regime. The same interlocked carcass model was explored in more than one publication to investigate buckling instabilities. The other study will be addressed in the next subsection.

Malta et al. (2013) continue this FE crushing study by exploring the response of a three-dimensional model that considers the pressure armor layer and the interlocked carcass. Based on a crushing load with three tensioners, the study takes advantage of the symmetry, considering a model of a 120° sector. The main FE model considers the exact profiles of both metallic layers. The authors also develop some simplified three-dimensional models, replacing the metallic layers with equivalent layers (using the simplifying approach developed by Martins, Pesce and Aranha (2003)). Figure 2.17 shows an overview of the main model. The responses of all numerical simulations are compared, concluding that the equivalent layer assumptions change the structural behavior of the metallic layers. Higher contrasts can be noticed for the interlocked carcass simplifications.

A three-dimensional FE crushing study of a STU is presented by Guttner (2015). The proposed modeling methodology represents all components using solid elements, as can be seen in Figure 2.18. The numerical case study is developed using ABAQUS, without considering symmetries or periodicities to reduce complexity, working with a model that is 3 meters long. Unlike the studies considering periodicity and symmetry, this study allows the assessing of stresses in the entry/exit transition region of the tensioner shoes, verifying an increase of the

Figure 2.17: At left, an overview of the crushing model presented in Malta et al. (2013); at right, an output from the numerical analysis: radial displacements.

Source: Malta et al. (2013).

stress levels in the steel tubes in these regions.

Figure 2.18: FE model of a STU developed by Guttner (2015).

Source: Guttner (2015).

Concluding this section, Mendonça (2016) implement FE simulations to perform a parametric investigation of crushing in flexible pipes. The study is based on previous models herein presented (Gay Neto et al. (2009) and Malta et al. (2013)), among others. This crushing analysis proposes several case studies to explore different parameters that affect the crushing behavior of a flexible pipe.

2.2.3 Core instabilities and bursting of flexible pipes

Recently, there has been a surge of research efforts focused on developing analyses using FE models to simulate certain instability problems (e.g., birdcaging and lateral buckling of the

tensile armors, wet collapse, burst, etc.). Although this thesis does not address topics related to this kind of phenomenon, some of those studies are still worth mentioning.

Gay Neto and Martins (2009) present a comparison between three models of interlocked carcass, one analytical and two numerical. The purpose of this analysis was to study the buckling instability of this structural layer under radial loads. The analytical model is based on an equivalent pipe approach. The first numerical model recreates the carcass profile and supposes axisymmetry, neglecting the laying angle of the wires. Figure 2.19 presents an overview of this configuration. In turn, the second model respects the helically wound geometry and the layer's small pitch, as illustrated in Figure 2.20. The boundaries of the axisymmetric model are fixed and only a quarter of the structure circumference is represented, taking advantage of the symmetries. Finally, the models present a very similar response in the subcritical region (pre-buckling), which means that all simplifications and assumptions considered for the axisymmetric model are valid for the tested conditions.

Figure 2.19: First model proposed by Gay Neto and Martins (2009), using axisymmetries to simplify the geometry.

Source: Gay Neto and Martins (2009).

Subsequently, focusing on burst analysis, Gay Neto et al. (2013b) compare four different models. Based on the models of interlocked carcass developed for the previous study, the authors now simulate the pressure armor layer, with the purpose of resisting to internal pressurization. The first of the new models is analytical and based on an equivalent pipe concept. This model is compared with three numerical models. The first numerical analysis uses a two-dimensional model composed of one half of an annulus. The planar model, which is axisymmetric and formed by two layers, represents the pressure armor and the inner polymeric layer. Both layers are constrained at the ends and subjected to a pressure load per unit length. The other two models are three-dimensional. One of them considers the proximity of the helical geometry under axisymmetric conditions and the other one preserves the helically

Figure 2.20: Second model proposed by Gay Neto and Martins (2009), respecting the helical geometries.

Source: Gay Neto and Martins (2009).

wound configuration of the pressure armor layer. In both cases, the geometry of the pressure armor profile is taken into account and the models include a polymeric layer to distribute the pressure load. The comparisons between models showed that the two-dimensional model yields satisfactory results. Additionally, there are differences owed to the neglected geometric characteristics.

Gay Neto and Martins (2014) study the wet collapse of flexible pipes in two different models, considering the interlocked carcass, the inner polymeric layer and the pressure armor. The FE model is based on a previous study by the same authors (Gay Neto and Martins (2012)), focused on wet collapse buckling of the interlocked carcass. Improving upon this study with the addition of the pressure armor layer, the second model aims to assess the same phenomenon in this outer layer. Axisymmetric hypotheses were considered, allowing the representation of a quarter of the cross-section. The first model represents the geometry of all layers accurately, while the second one is simplified. The pressure armor is modeled using an equivalent pipe, a technique already used in the studies mentioned in the previous paragraph. This approach simplifies the mesh and reduces the computational cost of the model.

Sousa et al. (2012) perform a study of the instability phenomenon called birdcaging — when the tensile armor tends to expand in response to compressive axial loads and/or external hydrostatic pressure. This work uses FE models previously developed by the authors and compares simulation results with experimental data obtaining sufficient representativeness and accuracy. Rabelo (2014) builds on Sousa et al. (2012) to develop a study of the instability mechanism. Rabelo et al. (2015) proposes a deflagration criterion for the birdcaging phenomenon, in which the structural instability of the outer polymeric layer plays a key role.

Malta and Martins (2014) also provides valuable information on the birdcaging phenomenon.

The paper presents an ABAQUS FEA of a flexible pipe with external sheath, a pair of tensile armors, a high-strength tape and a rigid core. This model considers different sorts of nonlinearities (e.g. contacts, gaps, friction, plasticity and large displacements). As the study does not employ simplifying assumptions using symmetries or periodicity, each individual wire of the tensile armor is free to move in any direction. Numerical case studies are presented and compared. Malta (2016) continue the assessment of the birdcaging phenomenon in his doctoral research. His thesis is based on FE models of flexible pipes developed with ABAQUS, presenting some numerical case studies. The models are used to investigate the mechanism that governs birdcaging instability through parametric analyses. Figure 2.21 illustrates one of his models. The results assessing the influence of the sample length on the birdcaging phenomenon were also published in Malta and Martins (2016).

Figure 2.21: One of the numerical case studies presented by Malta (2016). Overview of the FE model and example of result.

Source: Translated from Malta (2016).

This brief review shows that the foundations of the structural analysis of flexible pipes and umbilicals cables have constantly evolved over the years. While the first studies were based on analytical approaches, the rapid development of computational processing power soon allowed numerical methods to become widespread. Although all studies with FE models are quite recent, great progress has been achieved in that short period.

Other important works concerning specific topics addressed in each chapter are cited throughout the text.

PART I – FLEXIBLE PIPES

3 A FE ALTERNATIVE MODELING STRATEGY

This chapter describes the FE modeling technique developed during this study with the goal of simplifying the structural analysis of a flexible pipe by using “equivalent layers”. It provides details on the modeling methodology for each layer, aside from presenting considerations on boundary conditions, meshing and nonlinearities caused by interactions between layers. The early stages of this development were briefly presented by Santos (2013) and Santos, Pesce and Ramos Jr. (2015b).

One of the most challenging aspects when dealing with a FE model of a flexible pipe is to find an efficient way to model contacts, particularly those related to the considerable number of helical wires and the layers that surround them. Faced with that challenge, the author came up with a strategy: resorting to an alternative method to simulate the mechanical contact between all metallic and polymeric components.

Wriggers (2006) lists a great many studies concerning computational contact mechanics that were developed over the last decades. Special attention is given to the method of Lagrange multipliers and the penalty method. These two methods are widely recognized and can be found in most FE software that deal with contact problems.

In search of alternative ways to manage the contact between two bodies in a FE model, some authors propose different approaches to model the interface zone. As illustrated by Figure 3.1, the contact area can also be represented by a very thin layer of elements. This method replaces the commonly implemented zero-thickness boundary elements, resulting in a single mesh to represent both bodies and their interface. While the ‘zero-thickness’ approach is widely known when treating contact problems (it is the basis of the Lagrange and penalty methods), several works (e.g. Desai et al. (1984), Sharma and Desai (1992), Wang and Wang (2006), Wriggers, Schröder and Schwarz (2013), Bog, Zander and Rank (2015)) develop models using the ‘thin layer’ concept, thus avoiding the complexity of the classical schemes, which is greatly owed to the exact formulation and enforcement of contact constraints.

The ‘thin layer’ approach can be easily implemented to calculate the contact between two identical bodies with equal FE meshes. The alternative strategy proposed in this thesis builds upon these ideas, and its originality lies in the synergistic use of both techniques to model the mechanical contacts in a flexible pipe: the more common ‘zero-thickness’ approach is used to model the contact between layers, and the ‘thin layer’ approach is implemented between adjacent helical wires. This alternative method avoids the typical calculation of contact between metallic wires pertaining to the same armor layer. The layer of elements placed between wires is assigned with an artificial material, originating a concept of an equivalent pipe to represent helical layers. That equivalent pipe is modeled with two distinct materials, disposed

Figure 3.1: The 'thin layer' and 'zero-thickness' approaches.

Source: Cerfontaine et al. (2015)

in helical arrangements, representing wires and voids. The strategy reduces the amount of contact interaction and is meant to speed up numerical convergence, as well as to increase the computational performance. Section 3.2 exemplifies the technique.

3.1 PREVIOUS ASSUMPTIONS

In order to simplify the analysis, some premises were considered for the modeling methodology. These premises, listed below, are based mainly on Ramos Jr. (2001) and on Ramos Jr. and Pesce (2003), where the authors strongly emphasize the discussion and the establishment of well-grounded assumptions.

1. All materials are continuous, homogeneous, isotropic and they have a linear elastic behavior;
2. The central axis of the pipe remains straight during the application of axisymmetric loads;
3. The loads applied on the extremities are considered constant and evenly distributed;
4. The loads are static; dynamic effects are not considered.

Concerning the first assumption, this simple rheological behavior is adopted to simplify analysis and comparisons. Therefore, when dealing with a FE model, other rheological conditions can easily be implemented if necessary. Assumptions 2 and 3 are consequences of the boundary conditions employed, which are described in Section 3.3.4. Furthermore, as only a segment of the flexible pipe is considered to obtain numerical responses (elongations and rotations) from

the numerical models, elongation and rotation angle per unit length are treated as constant for this post-processing.

Regarding the last assumption, all cases simulated for this study consider static loads, so that all inertia effects are negligible. However, nonlinearities are not ignored; large-displacement effects, contact interactions and friction are accounted for. Newton's method is used to solve the nonlinear equilibrium equations. The solutions are calculated with a series of increments, with iterations to obtain equilibrium within each increment. In order to compare the computational efficiency of each simulation, there are no changes to the size of each increment, so that the number of required iterations can be used as a parameter to measure calculation performance. The increments are fixed at 5% of the total load and this configuration is adopted for all cases (i.e., all loading and unloading were applied using 20 increments, which were enough to ensure convergence of calculations).

3.2 EQUIVALENT LAYERS

Modeling simplifications herein developed intend to estimate the mechanical behavior of any flexible pipe. The alternative modeling strategies avoid the need for contact calculations between helical components pertaining to the same layer, but without interfering with contact interactions between layers. Since these contacts are of fundamental importance, the analysis considers a nonlinear FEM.

The methodology herein presented was constructed in ABAQUS, but it can be replicated using any FE software. All flexible pipe components are modeled with solid elements (3D), therefore excluding beam (1D) or shell elements (2D), which are commonly used to simplify FE models. Regarding layer geometry, the proposed methodology establishes two priority dimensions: thicknesses and diameters, both of which are always observed to generate the modeled layers and to ensure geometric compatibilities. The following sections describe the concepts used to represent each layer.

3.2.1 Polymeric layers

The plastic layers have a rather simple geometry when compared to other layers and there are no major geometric simplifications to be made. In the proposed modeling methodology, they are represented by a pipe with the corresponding radius and thickness (Figure 3.2).

The material is represented with linear elastic isotropic behavior. Both the Young's modulus and the Poisson's ratio (always about 0.5 to represent a typical almost incompressible material) correspond to the respective polymer. Figure 3.3 illustrates a sample of modeled layer with a

Figure 3.2: Section of the polymeric layer as a function of radius and thickness only.

reduced length.

Figure 3.3: Partial representation of a modeled plastic layer.

3.2.2 Tensile armor layers

In this new modeling methodology, tensile armor layers are characterized by a single pipe with the diameter and thickness of the actual layer. The wires are represented as helical partitions of this base-pipe, and the same goes for the gaps between them, which are represented by a hypothetical material with a very low modulus of elasticity. Figure 3.4 illustrates this representation.

The helically wound wires are subjected mainly to axial loads. Torsion and bending of

Figure 3.4: Representation of the tensile armor from a bimaterial pipe.

metallic wires play a secondary role. For this reason, the model prioritizes the cross-sectional area of the wires. This simplification supposes that changing the area moments of inertia has no considerable effect on the structural behavior of the assembly. Load cases influenced by this parameter have to be investigated in the future. Figure 3.5 illustrates how the partition's width L is calculated according to the laying angle α , the thickness t and the area of the wire cross-section A . All these data must be based on the actual analyzed structure.

Once the L dimension of the given layer's cross-section has been calculated, n representative wires are disposed along the circumference of the pipe, generating $2n$ different partitions. The 'equivalent layer' is created based on the 'thin layer' approach presented in Figure 3.1. An artificial material is assigned for each partition that characterizes the voids between steel wires. Figure 3.4 shows the basis-pipe with $2n$ regions: yellow cells represent the gaps (artificial material with a linear elastic behavior, considering a negligible Young's modulus of $1Pa$ and a null Poisson's ratio) and gray cells represent the steel wires (Young's modulus around $200GPa$, depending on the metallic material, and Poisson's ratio of 0.3).²

Since the layer is modeled as a bimaterial pipe, there is no need to calculate interactions between wires. These interactions are represented simply by the union of meshes from different partitions; in other words, the interface nodes belong to both regions, and therefore the creation of contact zones becomes unnecessary. This approach is advantageous because the coupling of wires is made directly on the model's stiffness matrix, thus circumventing the use of boundary

²Previous studies on the Young's modulus of the artificial material indicated that this value is low enough to influence neither the structural behavior of the model nor its computational performance.

Figure 3.5: Partitions size on the tensile armor.

conditions. Another significant advantage of the present approach is that it spares software users a great deal of time and effort, since they do not need to set several contact surfaces.

This methodology is similar to a spring approach, with the behavior of artificial elements representing the interaction between adjacent metallic wires. The advantage is that it can be easily implemented in any FE software, since the user just needs to assign the material properties in helical partitions of a base-pipe. There are gains in computational performance without affecting the quality of results, as the loads do not induce considerable deformations in the wire or on the gap region. However, if the elements of these gap regions become too distorted, the calculation will certainly face convergence difficulties. This issue is addressed in Section 3.5.

3.2.3 Pressure armor layers

A typical pressure armor may present wires with considerably complex sections. In order to simplify the modeling, complex profiles are replaced by rectangular sections. Since the proposed methodology prioritizes radius and thicknesses, the only dimension that remains to be estimated is the width of the equivalent rectangular profile to get an accurate superposition of layers.

Besides diameter and thickness, the model's design of helical layers also respects the laying angle. In addition, to reproduce an analogous behavior, the rectangular area has to be the same as the original profile area; consequently, the width L of the section can be calculated

as a function of the area and thickness, as illustrated in Figure 3.6. Again, it is important to highlight that the rectangular section was chosen for the sake of simplicity. According to Martins, Pesce and Aranha (2003) and Pesce et al. (2012), the flexural-torsional stiffness of the profile cross-section would have to be preserved if the study focused on squeezing or crushing loads. Further analysis is required to quantify the differences borne from considering other sections. Lastly, this geometrical simplification does not recover concentrated stresses that could appear in the pressure armor wires.

Figure 3.6: Equivalent rectangular section of the helical layer

Eliminating the complexity of the wire section, the pressure armor layer is simplified by rectangular profile springs. Similarly to what was described in the previous section on the tensile armor layer, the gaps between wires are also modeled using the ‘thin layer’ approach, as illustrated in Figure 3.7. The linear elastic behavior is also considered for this layer (Young’s modulus around $200GPa$ and Poisson’s ratio of 0.3) and for the gaps (Young’s modulus close to zero and a null Poisson’s ratio).

3.2.4 Interlocked carcass

The interlocked carcass is a formidable challenge in the FE modeling of flexible pipes. Currently, the CAD software allow a faithful representation of this layer with its complex geometry; nevertheless, a model with such level of detail would require a very high and unfeasible calculation time (not to mention convergence problems due to self-contacts and interlayer contacts). Therefore, we must resort to the same procedure adopted with the other helically wound layers: the interlocked profile is also replaced by rectangular sections with the same area. The width of the rectangular section is imposed to preserve this area, as well as the layer thickness. The obtained “equivalent pipe” will have a similar shape to the one presented in Figure 3.7 for the

Figure 3.7: Partial representation of the helical layer with square section wire and equivalent bimaterial pipe.

pressure armor.

3.2.5 Anti-wear and high-strength tapes

The high-strength and anti-wear tapes are ignored in this modeling approach. A future study may be done in order to represent these layers, which might be expressly required in certain situations, particularly those prone to *birdcaging* or involving compressive loads; see, e.g., Malta and Martins (2016). Hypothetically, the aramid tapes could be modeled as helical wires with a high modulus of elasticity in tension, a very low one in compression and a null Poisson's ratio. In addition, the anti-wear tapes could be represented by variations in the contact properties, such as a reduced friction coefficient.

3.3 MODELING PARAMETERS

Along with the geometrically simplified layers, this section explains the choices made regarding the type of Finite Element, the mesh configuration of the different layers and the length of the model. The mesh setup is chosen to simplify the model as much as possible, in order to reduce number of elements. On the other hand, it also must address the correct calculation of contact pressures and model stiffnesses.

3.3.1 Type of Finite Element

A common feature of all layers is the use of a single Finite Element in their thickness (no internal nodes in the layers, only at the interfaces). For this reason, Finite Elements with full-integration and incompatible modes are used for the entire model.

The C3D8I (ABAQUS index) are 8-nodes solid elements, reinforced with incompatible modes to improve bending behavior. In addition to the conventional Degrees of Freedom (DOF), incompatible deformation modes are added inside the element. The main effect of the additional DOF inside the element is to eliminate the false shear stresses that appear when the fully integrated element is loaded with bending (*shearlocking*). These additional DOF eliminate the artificial stiffness due to the Poisson effect in bending.

In a standard linear Finite Element with full-integration, the expected linear stress variation caused by bending is followed by a linear stress variation in the perpendicular direction, leading to incorrect stresses evaluation and overestimation of stiffness. As the layers have a single element in thickness, the C3D8I elements are used to avoid this numerical phenomenon. A more detailed explanation concerning the bending behavior of solid Finite Elements and numerical problems associated with it is provided in Appendix C.

3.3.2 Mesh size

As described in the previous section, the type of element is selected regarding the absence of internal nodes inside the layers; therefore, a single element is considered in each layer thickness, and as a result the only nodes are at the interfaces. In order to work with a regular mesh, this section presents the choices for mesh sizing in axial and circumferential directions.

Axially, the size is based on the geometry of each layer. For helical layers, for example, it is not necessary to establish the element size in this direction, since there is only one element in the wire section. Consequently, the choice of element size in the axial direction applies only to polymeric layers, being designated as equal to half of the pitch size of the pressure armor. This mesh size in the axial direction is chosen to obtain the maximum correlation between adjacent meshes, which are in contact. This correlation is important for the repeatability of contact conditions over the length of the model, thus improving numerical convergence. Therefore, element size is imposed axially by the pitch of the pressure armor, as illustrated in Figure 3.8. Some consequences of this choice are presented and discussed in Section 3.5.

Seeking a mesh correlation also in the circumferential direction, the mesh size along the circumference becomes a function of the number of wires in the tensile armor layers. As for the pressure armor, the sections of the tensile armor wires are represented by a single element.

Figure 3.8: Size of elements in the pipe length according to the pitch of the helical layer.

Thus, the number of elements along the circumference of these layers is already defined and it is proportional to the number of wires. The same rule applies to neighboring layers. Figure 3.9 shows an example: for a model with 35 wires in the inner tensile armor layer and 37 wires in the outer tensile armor layer, there are 36 elements in the circumference of the other layers.

Figure 3.9: Element size in the circumferential direction due to the number of tensile armor wires. Different colors for different layers.

These choices tend to be favorable for computational convergence. The correlation of adjacent meshes benefits the contact calculations between layers and reduces numerical issues. To conclude this section, Figure 3.10 illustrates the mesh of a tensile armor layer. In order to avoid distortion of elements, their length is defined by the wire pitch itself. For this configuration, the extremities end up presenting 6-nodes elements.

3.3.3 Contacts

As previously mentioned, contacts and friction effects generate nonlinearities that change the boundary conditions for each layer. Contact between layers indicates that the contact surfaces will not be free to move in any direction. Once limited, the displacement attempts

Figure 3.10: FE mesh of the tensile armor.

are converted into reaction forces. In other words, interactions between the layers modify the local boundary conditions of the model with the introduction of nonlinearities. Wriggers (2008) presents an entire chapter to treat contact problems and Wriggers (2006) discusses different formulations, algorithms and techniques to deal with numerical contact problems.

In a flexible pipe model with n layers, there will be $n - 1$ pairs of layer contacts. For the sake of simplicity, all contacts are set up with the same parameters. The helical geometry of the metallic layers introduces slips in the interfaces, and the radial contractions and expansions generate contact pressure. Therefore, the normal behavior is implemented with “hard” contact and the tangential behavior is modeled by a Coulomb model with a friction coefficient $\mu = 0.10$. In order to obtain an accurate result in the calculation of contact pressures, the “surface-to-surface” method was chosen.

The proposed methodology works with layers attached to each other. The external diameter of an inner layer is the internal diameter of the one that envelops it. Consequently, when the cylindrical surfaces are meshed, interferences or slight clearances appear in the interfaces of meshes. Even though these variations are small (order of magnitude of hundredths of a millimeter), they are enough to induce convergence problems in the contact iterations. Therefore, all interferences are eliminated at the beginning of the calculation, as illustrated in Figure 3.11, without introducing contact pressures (the software algorithm detects nodes penetrating surrounding layers and moves them without generating stresses). After removing node penetrations, a minor external/internal pressure 0.1MPa (or 1atm) is applied to the model so that the layers come fully into contact.

Figure 3.11: Example of contact interference elimination at the beginning of the calculation.

After a few pairs of interlayer contacts have been defined, a single simulation requires time to locate all interactions between elements, which results in a considerable increase of the total calculation time. Post-processing data show that interactions represent, on average, 80% of wall-clock time necessary to complete calculations. Therefore, mastering this factor is essential for the computational performance of the analysis. All parameters above-presented were chosen for the sake of simplicity and based on the computational contact recommendations from the ABAQUS software. Other contact parameters were not tested and are to be further explored and evaluated.

3.3.4 Boundary conditions

While there is not a perfect way to represent boundary conditions, it is possible to establish conditions that match the behavior required by the study. However, in order to distribute the load evenly, one must simultaneously enforce the sections at the ends to remain plane and impose rigid body constraints to their nodes. These nodes are associated with a single (master) node, created in the center of the section, as shown in Figure 3.12. The loads are then imposed at this central (master) node of the section.

Figure 3.12: Representation of boundary conditions: master node in the center to control the rigid body displacements at the extremities.

Other small influence strategies could have been chosen. The rigid body kinematic constraint

was selected for the sake of simplicity. As a consequence, a large model is required to mitigate the undesirable effects of boundary conditions on the structural behavior at mid sections.

3.3.5 Model length

Another important aspect of the modeling methodology is to represent a certain length of the flexible pipe, enabling a feasible calculation time. The model must be long enough to suppress the extremities effects yet of a length that does not compromise calculation time. Considering a FE model of a flexible pipe with all layers modeled, this dimension will certainly be but a small fraction of the total length of a real flexible pipe. However, it should be large enough in relation to the pipe's diameter that variations in tensile, torsional, flexural and pressure loads are null in the center of the model. For this reason, model length is treated as a parameter in all case studies presented in the next chapter, so that we may analyze its influence on all tested conditions.

3.4 CASE STUDIES: COMPUTATIONAL PERFORMANCE WITH AXISYMMETRIC LOADS

Following the presentation of the methodology employed to obtain the numerical models, this section aims to measure the computational performance of three models developed using all aforementioned considerations (including the bimaterial approach). Three case studies are herein carried out, considering experimental data available in the literature. The studies chosen as paradigms for the developed numerical models contain not only experimental results, but also the main characteristics of all layers of the respective flexible pipes. Complementing the analyses, analytical models are also considered for comparison. Some of the chosen works have their own analytical approaches. For those that do not, the aforementioned *PipeDesign* software (based on the analytical model developed by Ramos Jr. (2001)) was used to complement the analysis. Among other results, *PipeDesign* allows users to calculate the axial, torsional and flexural stiffnesses of a flexible pipe in a chosen equilibrium position.

The first flexible pipe used to test the new numerical approaches is described in Witz (1996). As pointed out in the literature review, this study presents experimental results, dealing with axial, torsional and bending loads separately. Witz (1996) also presents an average of analytical results from ten different sources. The second flexible pipe chosen to test the new modeling strategy is the one found in Ramos Jr. et al. (2008) and Ramos Jr. et al. (2014). That study presents experimental and analytical results for a flexible pipe experimentally tested under axial loading and subjected to different boundary conditions. The last case study uses a flexible pipe found in Sousa et al. (2013). This last work analyses four different load cases, including axial

tension, torsion and internal pressure. For all load cases, experimental results are presented and compared with numerical and analytical approaches.

Before we proceed to the case study reports, a few words about the computational equipment: all calculations were processed in a workstation with the specifications presented in Table 3.1. Parallel execution was used to reduce processing times. The ABAQUS solver uses thread-based parallelization and, in order to guarantee the same conditions for all calculations performed, all 32 available threads were always employed. Another aspect worthy of notice is the use of a Solid State Drive (SSD) to decrease storage time and the amount of Random Access Memory (RAM), which is enough to run all models herein described.

Table 3.1 : Workstation specifications.

	2 processors
CPU	Intel® Xeon® CPU E5-2620 v4 (with 8 cores/each and 16 threads/each)
Graphics	NVIDIA Quadro K1200
RAM	256 GB (DDR4)
Storage	SSD (1 TB)

3.4.1 Witz (1996)

The first case study refers to a 2.5-inches (internal diameter) flexible pipe composed by eight layers. Geometrical and material properties of each layer are provided in Table 3.2, reproduced from Witz (1996). The laying angles are specified based on the conventions presented in Appendix B. We would like to draw attention to the fact that only the second layer is polymeric; the other layers are composed by tapes and metallic helical components, and the external polymeric sheath is absent. As mentioned in the previous sections, the structural tapes are not modeled by the proposed FE approach.

The modeling methodology is applied to create an FE model from the data in Table 3.2. Figure 3.13 provides an overview of the resulting model, with all simplifications described in the previous chapter. Some components are suppressed so that we may have a good visualization of all layers. The absence of the external sheath is a notable feature of this case study.

Figure 3.14 shows details of the FE mesh generated for the model based on the flexible pipe from Witz (1996). Materials are identified by color. The white elements are assigned with the

Table 3.2 : Flexible pipe properties from the case study of Witz (1996).

Layer	Properties
1: Carcass	ID = 63.5 mm
	Layer thickness = 3.5 mm
	Number of wires = 1
	Lay angle = -87.00°
	Cross-sectional area = 18 mm^2
	Young's modulus = 199 GPa
	Poisson's ratio = 0.3
2: Inner sheath	Layer thickness = 4.9 mm
	Young's modulus = 284 MPa
	Poisson's ratio = 0.47
3: Pressure armor	Layer thickness = 6.2 mm
	Number of wires = 2
	Lay angle = $+85.5^\circ$
	Cross-sectional area = 51.5 mm^2
	Young's modulus = 200 GPa
	Poisson's ratio = 0.3
4: Anti-friction tape *	Layer thickness = 1.5 mm
5: Inner tensile armor	Layer thickness = 3.0 mm
	Number of wires = 40
	Lay angle = $+35.0^\circ$
	Cross-sectional width = 6.0 mm
	Young's modulus = 200 GPa
	Poisson's ratio = 0.3
6: Anti-friction tape *	Layer thickness = 1.5 mm
7: Outer tensile armor	Layer thickness = 3.0 mm
	Number of wires = 44
	Lay angle = -35.0°
	Cross-sectional width = 6.0 mm
	Young's modulus = 200 GPa
	Poisson's ratio = 0.3
8: Fabric tape *	Layer thickness = 0.5 mm

**structural tapes are not modeled by the proposed FE approach*

Source: Adapted from Witz (1996).

Figure 3.13: Overview of the model developed for the flexible pipe from Witz (1996). Model length was shortened, and some components were removed for visualization.

artificial material to represent voids. Metallic materials are gray, while the polymeric sheath is blue.

The cross-section image points out the 1-thickness-element layers, as well as the agreement between adjacent meshes. It is worth noting that the elements in the tensile armor layers are very regular and structured, also aiming at a good computational performance of the model.

Figure 3.14: Details of the mesh from the model developed for the flexible pipe from Witz (1996).

The proposed modeling methodology was applied to generate models with different lengths, varying between $50mm$ and $5000mm$, to enable the study of the influence of this parameter in the structural behavior. The models were designed considering two approaches, with and without the bimaterial concept. For the models without bimaterial layers, the white elements illustrated in the images above were simply suppressed. The models without the artificial material were created with $50mm$, $100mm$, $200mm$, $500mm$, $1000mm$, and $2000mm$. For

the bimaterial models, an additional length of 5000mm was also considered, resulting in a total of 13 different cases.

All other geometric simplifications described in Section 3.2 were implemented in all cases. Table 3.3 and Figure 3.15 present the number of degrees of freedom (DOF) for all generated FE models, with and without the bimaterial concept, for all considered model lengths. Notice that, for a same length, the models developed with the bimaterial concept present less DOF. The decrease in DOF when using the bimaterial layers is close to 20% for all model lengths, regardless of whether the models present more user-defined elements (i.e., elements from solid instances that represent a geometric part of the model, usually defined by the software user).

Table 3.3 : Models based on Witz (1996); number of DOF; different model lengths.

Model length [mm]	Degrees of freedom	
	without bimaterial	with bimaterial
50	38 970	31 908
100	74 076	59 985
200	143 784	116 451
500	356 220	285 180
1000	707 130	568 983
2000	1 412 022	1 132 209
5000		2 809 884

Figure 3.15: Models based on Witz (1996); number of DOF; different model lengths.

Concerning the number of elements, Table 3.4 presents the total number of elements for each model. The models with the bimaterial concept obviously contain more elements defined by the user (the difference lays in the white elements depicted in Figure 3.14). However, FE

software generate less internal elements for contact interactions. If the bimaterial concept is used, these contact elements are only needed in the interfaces between layers. Without the bimaterial approach, more internal contact elements are required, so that the contact between helical wires in the same layer can also be evaluated. For this reason, the total number of elements is reduced when considering the helical bimaterial approach. The graphics presented in Figure 3.16 enhance the data presented in the table above. Darker colors represent the user-defined elements, while the lighter colors denote the elements generated for contact.

Table 3.4 : Models based on Witz (1996); number of elements; different model lengths.

Model length [mm]	Elements	
	without bimaterial	with bimaterial
50	7 270	6 417
100	13 908	12 273
200	26 989	23 989
500	67 015	59 085
1000	133 054	118 133
2000	265 748	235 236
5000		583 946

Figure 3.16: Models based on Witz (1996); number of elements; different model lengths; darker colors to user-defined elements; lighter colors to elements generated for contact.

When the bimaterial approach is used, the number of user-defined elements increases in almost 80%, as expected. On the other hand, the elements generated for contact drop by half, reducing the total number by nearly 11%. While the models without bimaterial have 30% of geometric elements and 70% of contact elements, the models with bimaterial present around

60% of geometric elements and 40% of elements generated for contact. Consequently, the solvers for models with the artificial material in the interfaces between the wires will certainly require less from the contact algorithm.

A brief comparison with the size of a FE model presented in Zhang et al. (2015) can be informative. Using exclusively solid elements, Zhang et al. (2015) have also developed a model based on the flexible pipe from Witz (1996). Their numerical model was designed preserving the cross-section of metallic wires in the interlocked carcass and the pressure armor. The considered length was $1288mm$, and the resulting model had 5×10^5 elements. The modeling techniques herein employed reduce this number considerably, as the cross sections of the wires were greatly simplified.

The pre-allocated RAM was also quantified for each of the 13 models. Table 3.5 and Figure 3.17 present the memory required to run each calculation. Notice that the solver requires more memory when the bimaterial approach is used. For the longer lengths ($500mm$, $1000mm$ and $2000mm$), this increase is even more pronounced (over twofold), even with the decrease of DOF observed in Figure 3.15 on page 81. The geometric elements require more memory than the elements generated for contact.

Table 3.5 : Models based on Witz (1996); pre-allocated RAM memory; different model lengths.

Model length [mm]	RAM memory [MB]	
	without bimaterial	with bimaterial
50	110	145
100	469	771
200	811	2 540
500	4 546	9 263
1000	9 220	19 252
2000	19 833	42 120
5000		110 844

The same 4 load cases presented in Witz (1996) were re-simulated using the methodology proposed in this thesis, taking precautions to ensure that the numerical results from both works could be compared³. The load cases are named from Case (A) to Case (D), adopting the same terminology used in Witz (1996):

- **(A)** Axial traction with both ends of the flexible pipe free to rotate; the flexible pipe is loaded and unloaded four times with a maximum axial traction of $500kN$;

³Witz (1996) also presents a fifth load case with bending, which will be discussed in the next section.

Figure 3.17: Models based on Witz (1996); pre-allocated RAM memory; different model lengths.

- **(B)** Axial traction with both ends of the flexible pipe prevented from rotating; the flexible pipe is loaded and unloaded four times with a maximum axial traction of $500kN$ •;
- **(C)** Clockwise and anti-clockwise twisting with ends free to elongate; the flexible pipe is twisted in the anti-clockwise sense until a twisting angle of $+2.3^\circ/m$. Next, the twisting load is reversed until the model reaches a twisting angle of $-2.3^\circ/m$. To conclude, the flexible pipe is brought back to a zero-load condition;
- **(D)** Clockwise and anti-clockwise twisting with ends prevented from moving axially; the flexible pipe is twisted and untwisted identically to Case (C).

It is worth mentioning that all loads are applied in the reference points from both extremities of the model, as seen in Figure 3.12 on page 76. One extremity has its reference point fixed (all displacements and rotations are restrained), while the second reference point has its displacements imposed according to the respective load case to be simulated.

The first load case to be analyzed is the axial traction with both ends of the flexible pipe free to rotate. For the sake of simplicity, the axial stiffness from each model was chosen as the parameter to be compared. All stiffnesses evaluated in this Chapter were estimated through a linear regression of the force-displacement curve, since the structural responses obtained with the numerical models were approximately linear. Table 3.6 and Figure 3.18 summarize the 13 results comprising both modeling approaches and all tested lengths. As expected, the axial stiffness tends to decrease for longer models. The influence of boundary conditions on the short models makes them stiffer. By increasing the length, the boundary effects are reduced and the axial stiffness converges asymptotically.

The comparison allows us to verify that the axial stiffness of the model without the artificial material is always slightly lower than that of the model with bimaterial. However, this difference

Table 3.6 : Models based on Witz (1996); load case (A); different model lengths; axial stiffness with ends free to rotate.

Model length [mm]	Axial stiffness [MN]	
	without bimaterial	with bimaterial
50	249.9	287.5
100	183.3	192.8
200	161.8	166.3
500	152.0	154.3
1000	148.9	150.4
2000	147.3	148.6
5000		147.7

Figure 3.18: Models based on Witz (1996); load case (A); different model lengths; axial stiffness with ends free to rotate.

tends to decrease in the longer models, indicating that axial stiffness is not significantly affected by the artificial elements in the interface of the helical components.

Like many numerical models found in the literature, the axial stiffness values obtained with our FE models are above the analytical values predicted by the analytical models in Witz (1996). Considering the average of the 10 analytical models from Witz (1996), the axial stiffness of the models with their ends free to rotate was 128 MN , with a standard deviation of 21 MN . The asymptotic values of the FEA axial stiffness were 15% greater.

For Case (B), Table 3.7 and Figure 3.19 show the results obtained with the numerical models. Like the previous case, the axial stiffness converges to a plateau as model length is increased.

With both extremities prevented from rotating, the axial stiffness tends to be slightly higher.

Table 3.7 : Models based on Witz (1996); load case (B); different model lengths; axial stiffness with ends prevented from rotating.

Model length [mm]	Axial stiffness [MN]	
	without bimaterial	with bimaterial
50	251.6	289.4
100	185.3	194.9
200	163.7	168.3
500	153.9	156.1
1000	150.8	152.2
2000	149.2	150.5
5000		149.5

Figure 3.19: Models based on Witz (1996); load case (B); different model lengths; axial stiffness with ends prevented from rotating.

This is confirmed both by results of the numerical models and by the analytical values presented in Witz (1996). The mean axial stiffness of the analytical models with the restricted twisting was $129MN$ (also with standard deviation of $21MN$). In the best case scenario, this value is 16% lower than those calculated by the FE models. Notice that the presence of bimaterial layers had no significant impact on the models' axial behavior. It can be stated that the higher axial stiffness obtained with both numerical models is due to reasons other than the bimaterial approach.

For load case (C), with the flexible pipe's ends free to move axially, the direction of rotation influences the torsional stiffness. For the clockwise sense, the outer tensile armor layer tends to expand its diameter, detaching from the rest of the pipe. In this case, a lower stiffness is expected. In the opposite sense, this outer layer tends to squeeze the inner layers. Thus, a

higher torsional stiffness is expected. Table 3.8 and Figure 3.20 show the torsional stiffness values calculated for Case (C) considering clockwise and counterclockwise senses, with model lengths up to $1000mm$. The models with and without the bimaterial approach present similar behavior, showing that the 1-meter length seems enough to mitigate the effects of the boundary conditions.

Table 3.8 : Models based on Witz (1996); load case (C); different model lengths; torsional stiffness with ends free to elongate.

Model length [mm]	Torsional stiffness GI [kN.m ² /deg]			
	Clockwise sense		Counterclockwise sense	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
50	5.52	6.06	6.28	6.52
100	1.23	1.54	4.77	4.86
200	0.13	0.20	4.26	4.30
500	0.05	0.11	4.02	4.03
1000	0.05	0.09	3.94	3.95

Figure 3.20: Models based on Witz (1996); load case (C); different model lengths; torsional stiffness with ends free to elongate.

Considering the counterclockwise direction, the analytical results presented by Witz (1996) for torsional stiffness with the ends of the flexible pipe free to move axially were inconclusive due to excessive deviations. In the clockwise direction, the torsional stiffness obtained by Witz (1996) was $3.03kN.m^2/deg$, with a standard deviation of $0.54kN.m^2/deg$. As already verified regarding axial stiffness, this value is below the one predicted by numerical models, presenting a deviation of 30% for the longer models.

The results for Case (D) are presented in Table 3.9 and in Figure 3.21, verifying the comments above. These results also confirm that the $1000mm$ length model is representative.

Table 3.9 : Models based on Witz (1996); load case (D); different model lengths; torsional stiffness with ends prevented from moving axially.

Model length [mm]	Torsional stiffness GI [$kN.m^2/deg$]			
	Clockwise sense		Counterclockwise sense	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
50	5.54	6.09	6.31	6.55
100	2.25	2.42	4.81	4.88
200	1.67	1.76	4.32	4.34
500	1.58	1.63	4.06	4.07
1000	1.55	1.59	3.99	4.00

Figure 3.21: Models based on Witz (1996); load case (D); different model lengths; torsional stiffness with ends prevented from moving axially.

Correlating these results with the analytical data from Witz (1996), the stiffness in the clockwise direction presented a deviation of less than 4% for the longer models. The torsional stiffness presented by Witz (1996) was $1.48 \pm 0.16 kN.m^2/deg$. In the counterclockwise sense, the difference between the analytical and numerical approaches was less than 13%. The mean of the analytical values was $3.54 \pm 0.91 kN.m^2/deg$ for this case.

With the adequate length of 1000mm and the models with and without the bimaterial approach presenting similar results, the computational performances of the models were analyzed for the four load cases with this length. Table 3.10 and Figure 3.22 show the number of iterations required to complete the simulations for all load cases.

Exclusively axial traction loads were completed with about 500 iterations. Case (B), with the ends of the flexible pipe prevented from rotating, required a slightly smaller number of iterations to complete calculations and did not exceed 500 iterations. There is no relevant difference in the number of iterations required by each model for Cases (A) and (B).

The calculations for torsional load cases required a greater number of iterations to be com-

Table 3.10 : Model based on Witz (1996); model length: 1000mm; different load cases; required number of iterations to complete calculations.

Load case	Iterations	
	without bimaterial	with bimaterial
A	550	532
B	491	451
C	859	1 906
D	629	1 533

Figure 3.22: Model based on Witz (1996); model length: 1000mm; different load cases; required number of iterations to complete calculations.

pleted. Even though there was only one loading and unloading cycle to be simulated in each torsional sense, the contacts between the layers undergo severe discontinuities during this change of direction in the twisting angle, which certainly increases the number of iterations required for convergence. A comparison between the results of both models reveals a substantial difference. The models with artificial material require a much higher number of iterations to complete the analysis.

However, regarding the computational performance, it is more significant to observe the total simulation times. Table 3.11 and Figure 3.23 show the total simulation times, in minutes, obtained with the 1000mm long model. The simulations involving axial traction were noticeably much faster when the bimaterial layers were used, even though the number of iterations is similar. For Cases (C) and (D) concerning twisting loads, the models without bimaterial were somewhat more advantageous.

These results show that the iterations of the models with bimaterial are faster than the iterations of the models without bimaterial. For Cases (C) and (D), although the simulations required over twice the number of iterations, total processing times were slightly longer. The

Table 3.11 : Model based on Witz (1996); model length: 1000mm; different load cases; total wall-clock time to complete calculations.

Load case	Total wall-clock time [minutes]	
	without bimaterial	with bimaterial
A	418	185
B	366	158
C	474	488
D	343	410

Figure 3.23: Model based on Witz (1996); model length: 1000mm; different load cases; total wall-clock time to complete the calculations.

reduced number of contact elements presented in Figure 3.16 on page 82 seems to positively influence iteration processing time, making them faster. However, the bimaterial approach becomes advantageous only if the number of iterations for convergence does not increase considerably. Depending on the type of load, the calculation times can be favored or compromised.

Looking more closely at what happens during twisting loads (C) and (D), two main factors have likely hindered computational performance. Firstly, when the direction of the twisting angle is reversed, there is a change in the contact conditions between the tensile armor layers. The counterclockwise rotation increases the contact pressure between layers. Next, the clockwise rotation causes the appearance of a gap between these two helical layers. These discontinuities in contact conditions during the calculation increase the number of iterations required for convergence. However, this phenomenon affects both the models with bimaterial and those without.

The second factor that may explain the greater number of iterations of the model with bimaterial is the high distortion of the elements assigned with the artificial material when the

model is twisted in the clockwise sense. With the radius of the outer layer is increased, the helical wires tend to move apart from each other, producing an excessive distortion in the elements that represent voids. These deformations can also delay convergence and require a greater number of iterations, being a disadvantage of the bimaterial concept model.

Next, two case studies will allow a deeper understanding of computational performance gains and losses of using layers with or without bimaterial.

3.4.2 Ramos Jr. et al. (2008, 2014)

The 2.5-inches flexible pipe used for this case study also presents five structural layers: an interlocked carcass, an inner polymeric sheath, an internal tensile armor layer, an external tensile armor layer and an outer polymeric sheath. The main material properties and geometric data of this pipe can be found in Table 3.12. More details are presented in both articles from Ramos Jr. et al. (2008, 2014).

These articles do not identify the materials that compose the pipe. However, both articles provide the respective Young's moduli and Poisson's ratios, thereby making this analysis possible. Figure 3.24 presents an overview of the FE model based on this 2.5-inches flexible pipe. The authors do not mention structural tapes in the pipe composition, so the numerical model proposed represents accurately all layers of the described structure.

Figure 3.24: Overview of the model developed for the flexible pipe from Ramos Jr. et al. (2008, 2014). Model length shortened, and some components were removed for visualization.

The smaller number of tensile armor wires is essential for the reduced size of the model, when compared to the case study based on Witz (1996). Another important factor that greatly simplifies this second case study is the absence of a pressure armor layer. This implies larger laying angles for the tensile armors, since these layers have to resist both axially and radially. Details of the FE mesh obtained are shown in Figure 3.25.

Compared with the models from the previous case study, the ones now obtained contain less degrees of freedom (DOF) for the same lengths. In some of them, the reduction is up to 50%.

Table 3.12 : Flexible pipe properties from the case study of Ramos Jr. et al. (2008, 2014).

Layer	Properties
1: Carcass	ID = 66.74 mm Layer thickness = 1.63 mm Number of wires = 1 Lay angle = +85.8° Cross-sectional area = 19.56 mm ² Young's modulus = 190 GPa Poisson's ratio = 0.3
2: Inner sheath	Layer thickness = 6.0 mm Young's modulus = 280 MPa Poisson's ratio = 0.4
3: Inner tensile armor	Layer thickness = 2.0 mm Number of wires = 29 Lay angle = +55.5° Cross-sectional width = 5.0 mm Young's modulus = 207 GPa Poisson's ratio = 0.3
4: Outer tensile armor	Layer thickness = 2.0 mm Number of wires = 29 Lay angle = -55.5° Cross-sectional area = 5.0 mm Young's modulus = 207 GPa Poisson's ratio = 0.3
5: Outer sheath	Layer thickness = 5.0 mm Young's modulus = 320 MPa Poisson's ratio = 0.4

Source: Based on Ramos Jr. et al. (2014).

Figure 3.25: Details of the mesh from the model developed for the flexible pipe from Ramos Jr. et al. (2008, 2014).

Table 3.13 and Figure 3.26 present the number of DOF for the developed models, considering the same lengths from the previous case study. Notice that unlike the differences previously observed, in this case the models with and without the bimaterial approach have very similar numbers of DOF. It is worth remembering that the two structures have five layers. However, the flexible pipe from Witz (1996) has 4 helically wound layers and only one polymeric sheath, while this flexible pipe from Ramos Jr. et al. (2008, 2014) has 3 helical layers and two polymeric ones.

Table 3.13 : Models based on Ramos Jr. et al. (2008, 2014); number of DOF; different model lengths.

Model length [mm]	Degrees of freedom	
	without bimaterial	with bimaterial
50	19 683	19 860
100	37 881	37 827
200	73 788	73 878
500	181 224	181 314
1000	359 832	359 922
2000	718 698	718 764
5000		1 792 974

Regarding the number of elements in each model, Table 3.14 and Figure 3.27 present the values for all 13 considered lengths. While the use of the bimaterial approach reduced the

Figure 3.26: Models based on Ramos Jr. et al. (2008, 2014); number of DOF; different model lengths.

number of elements in the FE models based on Witz (1996), in this case, the models with bimaterial contain more elements.

Table 3.14 : Models based on Ramos Jr. et al. (2008, 2014); number of elements; different model lengths.

Model length [mm]	Elements	
	without bimaterial	with bimaterial
50	3 353	4 173
100	6 524	8 102
200	12 799	15 991
500	31 568	39 505
1000	62 751	78 586
2000	125 426	157 114
5000		392 178

Again, darker colors indicate user-defined elements. The expected increase in the number of this kind of elements in the models is confirmed for the bimaterial approach. Lighter colors identify the number of elements generated for contact calculations. Since this second case study deals with a lower number of wires in the helical layers, the reduction in the number of contact elements is not as effective. The models without bimaterial present 40% of geometric elements. Using the artificial material at the wires' interfaces, the ratio of contact elements falls from 60% to 48%. In comparison with the previous case study, the differences between models with and without bimaterial are slighter, as this second pipe presents less helical layers and less wires in the tensile armors.

Observing the pre-allocated RAM for each model, again the modeling technique with bima-

Figure 3.27: Models based on Ramos Jr. et al. (2008, 2014); number of elements; different model lengths; darker colors to user-defined elements; lighter colors to elements generated for contact.

material requires a larger amount of memory. Table 3.15 and Figure 3.28 compute the memory required for the models with different lengths. The increase between approaches with and without bimaterial is not as expressive as in the first case study, but it is still a factor worth considering.

Table 3.15 : Models based on Ramos Jr. et al. (2008, 2014); pre-allocated RAM memory; different model lengths.

Model length [mm]	RAM memory [MB]	
	without bimaterial	with bimaterial
50	42	44
100	366	400
200	1 298	1 461
500	4 280	5 259
1000	9 186	11 866
2000	19 200	24 516
5000		62 322

The load cases presented by Ramos Jr. et al. (2008, 2014) consider only axial tensile loads under four different conditions. The experimental approach contemplates tests with and without internal pressure and axial loads applied with ends fixed and free to rotate. The combination of these conditions provides four different cases. As each case is tested twice, the experimental study consists of eight assessments.

To compare the results from Ramos Jr. et al. (2014) with the results from the numerical models herein developed, similar load cases were simulated. In the FE simulations, the loading

Figure 3.28: Models based on Ramos Jr. et al. (2008, 2014); pre-allocated RAM memory; different model lengths.

conditions were replicated, and the traction was also applied cyclically from zero to $50kN$, loading and unloading the model 3 times. With the same nomenclature from Ramos Jr. et al. (2008, 2014), the load cases simulated with and without bimaterial are summarized as follows:

- **(A)** Three cycles of axial traction varying from $0kN$ to $50kN$; both ends of the flexible pipe prevented from rotating; no internal pressure;
- **(B)** Three cycles of axial traction varying from $0kN$ to $50kN$; both ends of the flexible pipe free to rotate; no internal pressure;
- **(C)** Three cycles of axial traction varying from $0kN$ to $50kN$; both ends of the flexible pipe prevented from rotating; constant internal pressure of $1000psi$;
- **(D)** Three cycles of axial traction varying from $0kN$ to $50kN$; both ends of the flexible pipe free to rotate; constant internal pressure of $1000psi$.

In the FEA, axial loads are always applied at the reference points at both extremities of the models. Displacements and rotations are imposed to these points, assisting convergence. Only the internal pressure load is applied through the nodes of the internal wall of the inner polymeric sheath. Similarly to the experimental procedures described in Ramos Jr. et al. (2014), the internal pressure is applied before the cyclic axial traction.

As in the first case study, the axial stiffness was chosen as a parameter to be compared, using as paradigm the corresponding results presented in Ramos Jr. et al. (2014). With no internal pressure applied to the models, cases (A) and (B) have their axial stiffness results presented below. Next, cases (C) and (D) replicate the same loading conditions from cases (A) and (B) respectively, but with the models pressurized at $1000psi$. The axial stiffness of

the flexible pipe with internal pressure is also given below. All results are concisely presented in Table 3.16 and in Figure 3.29.

The results for all 4 load cases are extremely similar. The downward trend of values as the length of the model increases is also quite similar. The results tend to plateaus, so that for lengths greater than $1000mm$ the variation in axial stiffness is no longer substantial. For all tested loading conditions, the differences between models with and without bimaterial also tends to decrease when the length of the models increases.

Compared to the results from Ramos Jr. et al. (2014), the numerical models herein presented are stiffer. For cases (A) and (B), Ramos Jr. et al. (2014) analytically estimated axial stiffnesses of $4.421MN$ and $4.394MN$, respectively. Experimental results from Ramos et al. (2014) broadly agree with the analytical approach. The numerical models show a deviation of 8% compared to these analytical values. Both analyses indicate a very slight stiffness reduction when the ends of the flexible pipes are free to rotate.

Cases (C) and (D) are distinguished from (A) and (B) by the application of the $1000psi$ internal pressure, as already described. However, the analytical model used in Ramos Jr. et al. (2014) indicates that this internal pressure has nearly no influence on the axial stiffness of the flexible pipe. The *force x elongation* curves simply shift when the internal pressure is considered. However, the numerical models were about 5% stiffer when pressurized. Compared with analytical results, differences increased to 13%.

Regarding the number of iterations required to complete each load case, the $2000mm$ long model was chosen for comparison. Table 3.17 and Figure 3.30 show the number of iterations to complete the load cases (A) through (D). Notice that the models with and without bimaterial presented very similar values. Cases (A) and (C) with non-rotating ends presented better convergence, as expected. By restricting the rotation at the ends of the model, the numerical system becomes more stable. Comparing the models with and without internal pressure, the variation in the number of iterations is not considerable, and it is not possible to determine whether the internal pressure influences the computational performance in this case study.

As for the previous case study, the iterations were faster when the bimaterial approach was used. The advantage of the use of the artificial material is evident when the total times of calculation are observed. Table 3.18 and Figure 3.31 present the simulation times for models with $2000mm$ in length.

Some conclusions can be drawn from the correlation between numbers of iterations and processing times of each load case. As concluded for the analysis with the flexible pipe from Witz (1996), simulations involving axial traction without considerable twisting angles benefit from the bimaterial approach. All four load cases considered for the flexible pipe from Ramos Jr. et al. (2008, 2014) have achieved reduced processing times with the bimaterial layers.

Table 3.16 : Models based on Ramos Jr. et al. (2008, 2014); axial stiffness from load cases (A) to (D); different model lengths.

Model length [mm]	Axial stiffness [MN]			
	(A)		(B)	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
50	61.16	148.83	61.11	148.93
100	10.82	12.16	10.79	12.12
200	6.33	6.68	6.31	6.66
500	5.08	5.27	5.07	5.25
1000	4.77	4.92	4.75	4.90
2000	4.63	4.76	4.61	4.74
5000		4.67		4.66

	(C)		(D)	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
50	61.45	150.69	61.39	149.56
100	11.53	12.85	11.51	12.82
200	6.69	7.04	6.67	7.01
500	5.35	5.55	5.33	5.53
1000	5.02	5.18	5.00	5.16
2000	4.87	5.01	4.85	4.99
5000		4.92		4.90

Figure 3.29: Models based on Ramos Jr. et al. (2008, 2014); axial stiffness from load cases (A) to (D); different model lengths.

Table 3.17 : Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; required number of iterations to complete calculations.

Load case	Iterations	
	without bimaterial	with bimaterial
A	402	409
B	454	458
C	371	384
D	494	495

Figure 3.30: Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; required number of iterations to complete calculations.

The computational performance gains were not as great as those obtained for the flexible pipe from Witz (1996) due to the above-mentioned causes: smaller number of helical layers and less wires in the tensile armor layers.

Comparing simulations of models restrained from twisting with those with their ends free to rotate, the first ones are slightly faster. Correlating with the number of iterations already presented, it is noticeable that the time per iteration is slightly higher for the models with internal pressure. Using the bimaterial approach, the gains in processing time were greater than 20% for all simulated cases.

3.4.3 Sousa et al. (2013)

The last case study was performed with a flexible pipe from a study by Sousa et al. (2013). As in the case studies presented in the previous sections, four load cases are considered, with numerical and analytical results. These results will be compared with the numerical results

Table 3.18 : Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; total wall-clock time to complete the calculations.

Load case	Total wall-clock time [minutes]	
	without bimaterial	with bimaterial
A	354	280
B	412	316
C	413	314
D	528	412

Figure 3.31: Model based on Ramos Jr. et al. (2008, 2014); model length: 2000mm; different load cases; total wall-clock time to complete the calculations.

herein obtained. The flexible pipe from Sousa et al. (2013) is a 2.5-inches 9-layers pipe composed by: an interlocked carcass, an inner polymeric sheath, a pressure armor, a pair of tensile armor layers, an outer sheath, and structural tapes. Geometric and material data from these layers can be found in Table 3.19.

The modeling methodology proposed in this thesis does not require any more data than presented above; for further details regarding this flexible pipe, please refer to Sousa et al. (2013). It is worth remembering that, just as in the first case, the structural tapes were not modeled. The following images illustrate the FE model obtained. Figure 3.32 illustrates a portion of the model, exhibiting its layers with simplified geometry. Next, Figure 3.33 illustrates the mesh considering the prescribed methodology.

This flexible pipe does not have as many wires in its tensile armors as the pipe from Witz (1996). However, it has one more layer: the outer sheath. To generate the FE mesh, the number of elements in the perimeter is similar to the amount of wire in the pair of tensile armors. The cross-sectional image of the mesh illustrates this congruence between meshes of

Table 3.19 : Flexible pipe properties from the case study of Sousa et al. (2013).

Layer	Properties	
1: Carcass	ID = 63.5 mm	Young's modulus = 205 GPa
	Layer thickness = 3.5 mm	Poisson's ratio = 0.3
	Number of wires = 1	
	Lay angle = +87.6°	
	Cross-sectional area = 19.6 mm ²	
2: Inner sheath	Layer thickness = 5.0 mm	Young's modulus = 345 MPa Poisson's ratio = 0.3
3: Pressure armor	Layer thickness = 6.2 mm	Young's modulus = 205 GPa
	Number of wires = 2	Poisson's ratio = 0.3
	Lay angle = +85.6°	
	Cross-sectional area = 54.1 mm ²	
4: Anti-wear tape *	Layer thickness = 2.0 mm	
5: Inner tensile armor	Layer thickness = 2.5 mm	Young's modulus = 205 GPa
	Number of wires = 32	Poisson's ratio = 0.3
	Lay angle = +30.0°	
	Cross-sectional width = 8.0 mm	
6: Anti-wear tape *	Layer thickness = 1.5 mm	
7: Outer tensile armor	Layer thickness = 2.5 mm	Young's modulus = 205 GPa
	Number of wires = 34	Poisson's ratio = 0.3
	Lay angle = -30.0°	
	Cross-sectional width = 8.0 mm	
8: Fabric tape *	Layer thickness = 0.5 mm	
9: Outer sheath	Layer thickness = 5.0 mm	Young's modulus = 215 MPa
		Poisson's ratio = 0.3

**structural tapes are not modeled by the proposed FE approach*

Source: Adapted from Sousa et al. (2013).

Figure 3.32: Overview of the model developed for the flexible pipe from Sousa et al. (2013). Model length was shortened, and some components were removed for visualization.

Figure 3.33: Details of the mesh from the model developed for the flexible pipe from Sousa et al. (2013).

neighboring layers.

Regarding model sizes, we worked with the same lengths from the previous case studies, resulting in 13 models with and without the bimaterial approach. Table 3.20 and Figure 3.34 illustrate the number of degrees of freedom (DOF) for each model.

The bimaterial approach reduces the amount of DOF for each given length, as already verified in the FE models from the first case study. This indicates that a smaller number of equations must be solved for each iteration. The models with bimaterial presented circa 17% less DOF than the models without bimaterial.

Analogously to both previous studies, Table 3.21 and Figure 3.35 show the number of elements and the amount of pre-allocated memory for each model.

The results obtained are similar to those of the previous case studies, and the same considerations apply. The use of a modeling methodology with bimaterial increases the number of user-defined elements and decreases the number of elements generated to calculate contacts.

Table 3.20 : Models based on Sousa et al. (2013); number of DOF; different model lengths.

Model length [mm]	Degrees of freedom	
	without bimaterial	with bimaterial
50	29 940	25 080
100	57 120	47 544
200	111 672	92 562
500	273 684	226 248
1000	544 992	450 030
2000	1 087 038	897 306
5000		2 239 608

Figure 3.34: Models based on Sousa et al. (2013); number of DOF; different model lengths.

In this case, the total number of elements was slightly reduced in the models with bimaterial. Again, an increase of the amount of memory required to run calculations is confirmed. For this flexible pipe, bimaterial models require around 40% more memory to be processed.

To compare the computational performance of the models with and without bimaterial, the same load cases found in Sousa et al. (2013) have been precisely reproduced. It is worth remembering that loads are always applied by imposing displacements and rotations.

- **(A)** The model is axially tensioned with its ends prevented from rotating; firstly, axial traction is increased until $700kN$; next, 3 cycles of amplitude $250kN$ and mean value of $575kN$ are applied;
- **(B)** The model is axially tensioned and then twisted; with an initial pre-tension of $575kN$, the model is firstly twisted in the counter-clockwise sense; 3 cycles of amplitude $0.581^\circ/m$ and mean value of $0.516^\circ/m$ are imposed; next, the torsional load is reversed and a

Table 3.21 : Models based on Sousa et al. (2013); number of elements and pre-allocated RAM memory; different model lengths.

Model length [mm]	Elements		RAM memory [MB]	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
50	5 496	5 004	100	100
100	10 580	9 694	456	613
200	20 741	19 063	1 663	2 462
500	50 891	46 870	5 709	8 662
1000	101 394	93 417	12 223	18 064
2000	202 289	186 460	26 099	37 047
5000		465 770		98 517

Figure 3.35: Models based on Sousa et al. (2013); number of elements and pre-allocated RAM memory; different model lengths; at left, darker colors to user-defined elements and lighter colors to elements generated for contact.

clockwise twist is applied; 3 cycles of amplitude $0.387^\circ/m$ and mean value of $0.688^\circ/m$ are considered; during the twisting loads, the model's ends are prevented from moving axially;

- **(C)** The model is axially tensioned with its ends prevented from rotating, and then pressurized; firstly, axial traction is increased until $575kN$; next, the model is pressurized until $15MPa$ and depressurized; this zero to maximum pressure load is simulated considering 3 load cycles;
- **(D)** The model is axially tensioned, pressurized and then twisted; first, axial traction is increased until $575kN$ and the model is pressurized until $15MPa$; the following loading conditions are the identical to case (B).

Notice that during all 4 load cases the model is pre-tensioned and it is never completely

unloaded. The pre-tension ensures that all layers are always in contact and prevents severe discontinuities, like those from the first case study, for load cases (C) and (D). This type of loading favors numerical convergences and comparisons with other models. The same stiffness parameters addressed by Sousa et al. (2013) were calculated for the numerical models herein developed (case (A): axial stiffnesses; cases (B) and (D): torsional stiffnesses; case (C): the ratio between axial traction and internal pressure).

Concerning the axial stiffness calculated within Case (A), Sousa et al. (2013) presented experimental and numerical values of $153MN$ and $151MN$, respectively. Both approaches considered a $4650mm$ long flexible pipe. The axial stiffness results from Table 3.22 and Figure 3.36 show that the behavior of axial stiffness for different model lengths is the same observed in previous case studies. It has been verified that the pipe from Sousa et al. (2013) is long enough and apparently a model with 1-meter length would already present converged axial stiffness values. The introduction of the artificial material practically does not change the structural behavior of longer models. However, the axial stiffness values herein obtained are around $163MN$, about 7% higher than the experimental value from Sousa et al. (2013).

Table 3.22 : Models based on Sousa et al. (2013); load case (A); different model lengths; axial stiffness with ends prevented from rotating.

Model length [mm]	Axial stiffness [MN]	
	without bimaterial	with bimaterial
50	296.6	351.1
100	207.4	216.6
200	181.6	185.0
500	168.9	170.1
1000	165.0	165.7
2000	163.2	163.5
5000		162.3

The models herein developed are axially stiffer, as expected for a numerical approach. However, the torsional stiffness results are below the values found by Sousa et al. (2013). For case (B), without internal pressure, the experimental and numerical values obtained by Sousa et al. (2013) were around $3.51kN.m^2/deg$ and $3.25kN.m^2/deg$, respectively. With internal pressure, the experimental value rose to $3.60kN.m^2/deg$ while the numerical values were reduced to around $3.18kN.m^2/deg$. The changes were not significant, showing that this internal pressure of $1000psi$ practically does not modify the torsional stiffness of the flexible pipe. It is worth mentioning that as the pipe is always pre-tensioned during load cases (B) and

Figure 3.36: Models based on Sousa et al. (2013); load case (A); different model lengths; axial stiffness with ends prevented from rotating.

(D), the same stiffness is obtained for both the clockwise and the counterclockwise loading senses.

Table 3.23 and Figure 3.37 present torsional stiffness results obtained for the models with and without bimaterial. The values without internal pressure are those on the left, while the values obtained with the pressurized models are listed at the right-hand side.

Table 3.23 : Models based on Sousa et al. (2013); load case (B), at left, without internal pressure; load case (D), at right, with internal pressure of 1000psi; torsional stiffness with ends prevented from moving axially.

Model length [mm]	Torsional stiffness GI [kN.m ² /deg]			
	(B)		(D)	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
50	4.91	5.11	4.93	5.20
100	3.33	3.38	3.34	3.40
200	2.91	2.93	2.91	2.94
500	2.70	2.71	2.70	2.72
1000	2.64	2.65	2.64	2.65
2000	2.61	2.61	2.61	2.62
5000		2.60		2.60

The differences in stiffness between cases (B) and (D) are not expressive enough to show up in the graphics. It reproduces the structural behavior seen in Sousa et al. (2013). However, the torsional stiffnesses of the models herein developed are below the earlier experimental and numerical values. The 5000mm long model presented torsional rigidity about 19% lower.

Figure 3.37: Models based on Sousa et al. (2013); load case (B), at left, without internal pressure; load case (D), at right, with internal pressure of 1000psi ; torsional stiffness with ends prevented from moving axially.

Once again, the differences in stiffness between models with and without bimaterial were very small. This indicates that the reduced torsional stiffness is an outcome of other simplifications implemented. Two major factors can be pointed out for these deviations, both involving the contact interactions between layers, which seem to have been underestimated.

First, as already described in the previous chapter, structural tapes were disregarded and the diameters of the layers were adapted on account of these absences. These small changes in geometry may have greatly altered the torsional stiffness. The second factor that may have generated these differences is the simplicity of the contact model chosen. In Sousa et al. (2013), the contact between layers is modeled with an initial adhesion, and the slip between layers only appears after this adhesion is overcome. The models herein developed do not consider this initial adhesion and the tangential behavior of the contacts is only managed by a Coulomb model with a penalty coefficient of $\mu = 0.1$. In this way, the neighboring layers present sliding from the beginning of the twisting load and the models do not reproduce the same hysteresis of the experimental and numerical tests developed in Sousa et al. (2013). A higher friction could increase the torsional stiffness of the models, but this parameter was fixed for all the analyses, to guarantee the comparison between models and between case studies.

Load case (C) corresponds to a cyclic internal pressure varying from zero to 15MPa , with the pipe's ends restrained from moving axially. The relationship between axial traction and internal pressure was monitored in each model, following the same methodology adopted by Sousa et al. (2013). Experimentally, Sousa et al. (2013) obtained a ratio of 2.506kN/MPa , and their numerical model presented 2.753kN/MPa . Table 3.24 and Figure 3.38 present the results obtained with the models herein developed. The values are slightly higher than the results from both approaches presented by Sousa et al. (2013). Once again, the tendency to reach a plateau and similar results for models with and without bimaterial are observed.

Regarding the computational performance of the numerical approaches, Table 3.25 and Figure 3.39 present the monitored values for the 1000mm long model. With the exception of

Table 3.24 : Models based on Sousa et al. (2013); load case (C); different model lengths; traction – internal pressure ratio with ends prevented from rotating.

Model length [mm]	Traction – Internal pressure ratio [kN/MPa]	
	without bimaterial	with bimaterial
50	3.04	3.36
100	2.85	2.99
200	2.83	2.88
500	2.80	2.82
1000	2.80	2.81
2000	2.80	2.80
5000		2.80

Figure 3.38: Models based on Sousa et al. (2013); load case (C); different model lengths; traction – internal pressure ratio with ends prevented from rotating.

case (D), the models with bimaterial required a greater number of iterations. However, the bimaterial approach has faster calculation times for the simulated loading conditions; for the cases involving twisting, the gains in simulation time surpass 25%. Again, the calculations of the iterations using the bimaterial are faster.

Table 3.25 : Model based on Sousa et al. (2013); model length: 1000mm; different load cases; required number of iterations and total wall-clock time to complete calculations.

Load case	Iterations		Total wall-clock time [minutes]	
	without bimaterial	with bimaterial	without bimaterial	with bimaterial
A	355	451	283	212
B	495	586	403	282
C	357	429	284	205
D	555	533	480	295

Figure 3.39: Model based on Sousa et al. (2013); model length: 1000mm; different load cases; required number of iterations and total wall-clock time to complete calculations.

3.5 INTERNAL PRESSURE AND BENDING

To emphasize the differences between both modeling approaches, the following analyses made in this section investigate only the models' mechanical responses and no longer examine their computational performances. The 500mm-long models developed for the Witz (1996) and Sousa et al. (2013) cases with were tested subjected to bending and pressure loads. This way, the contact between layers and the consequences of using the bimaterial approach can be better explored. The bending load induces greater variations in the contact conditions between layers. When the model is bent, artificial elements are more requested. This load case allows us to gain a deeper understanding of this modeling technique and its consequences.

Only the flexible pipes from Witz (1996) and Sousa et al. (2013) were chosen for this final analysis, since both have pressure armor layers and laying angles of tensile armors around 30°. Besides, those works provide bending results that can be correlated with the ones herein obtained.

Before applying the bending load, an internal pressure was introduced to make sure that

all contacts between layers were well-established. This pressure also ensures an initial friction force between the layers and is meant to obtain the typical bending hysteresis cycles. With stable contacts between layers, the convergence of calculations becomes less computationally expensive. The solver ends up needing fewer iterations to converge. The imposed internal pressure also minimizes variations between open-closed contact conditions, which would interfere with convergence.

With the same boundary conditions mentioned in Section 3.3.4, the models were first pressurized with an internal pressure of $10MPa$. The pressure was applied directly to the inner plastic sheath, in such a way as to ensure that the contact loads between this layer and the pressure armor were monitored. Figure 3.40 illustrates the contact pressures between the plastic layer and the pressure armor of the model with bimaterial based on Witz (1996), after applying the internal pressure of $10MPa$. Below, Figure 3.41 illustrates the same results for the model without bimaterial. The unit of calculated contact pressures is MPa .

There is a substantial difference between the results from both modeling approaches. The geometric continuity of the layers with bimaterial distributes the contact pressure uniformly along the model's length, while the model without bimaterial presents variation of the contact pressure on the surface of the inner sheath. The nodal contact forces are influenced by the nodal area in contact. The model with bimaterial has the same nodal contact area for all nodes because its surfaces are cylindrical and continuous. Therefore, all nodes over the surface distribute the contact pressure regularly. With the exception of both extremities, which have zero pressure values as a function of the rigid body boundary conditions, the contact pressure for the model with bimaterial is around $10MPa$. The plastic layer transfers practically all of the applied internal pressure, as expected.

Figure 3.41 illustrates the results of the model without bimaterial. The model with helical components to represent the metallic wires has nodal contact areas varying along the mesh interfaces. These areas change according to the combination of adjacent meshes. Depending on the mesh size of the inner plastic sheath, there will be nodes in contact with the wires of the pressure armor and there will be nodes that do not establish any contact. Consequently, null values of contact pressure are expected, which explains the presence of blue regions in results. Concentrations (in red) appear near these null values, to counterbalance efforts.

To correctly represent this distribution of contact pressures in the model without bimaterial, the mesh of the plastic sheath needs to be refined. The chosen mesh size is insufficient and a numerically induced spatial aliasing phenomenon appears in the contact between layers.

Figure 3.42 presents the same results for the model with bimaterial based on Sousa et al. (2013), with an internal pressure of $10MPa$. With bimaterial, this second model also presents a uniform result for the contact pressure between the inner plastic layer and the

Figure 3.40: Model based on Witz (1996); with bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$.

Figure 3.41: Model based on Witz (1996); without bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$.

pressure armor. The model without bimaterial presents variations (Figure 3.43), due to the same reasons already detailed regarding the previous model.

When compared to what happens in reality, both approaches present unrealistic results. The model with bimaterial presents an average of contact pressures between layers. It is an underestimated value, as the contact area between layers is larger than the real one.

On the other hand, models without bimaterial would need more refined meshes to achieve more realistic results. With a sufficiently refined mesh, nodes from the plastic sheath in contact with helical wires would have contact pressure greater than $10MPa$, whereas nodes between wires would present null pressure values. It would be possible to identify the position of the helical wires just by looking at the distribution of the contact pressure in the plastic layer. However, a more refined mesh is not within the scope of this work; our aim is to simplify the FE model as much as possible.

To analyze how this difference in the distribution of contact pressures affects the structural behavior of the models, bending loads were simulated. The bending was introduced simply by imposing rotation at both ends of the model, which behave like rigid bodies (as described in Section 3.3.4 on page 76). To guarantee an approximately uniform curvature at mid span, only the model length of $500mm$ was considered. Tests with longer models resulted in concentrated bending near to the extremities. For this reason, the models with $500mm$ length were chosen to be analyzed.

After applying an internal pressure of $30MPa$, rotations were applied at both ends, in order to bend the model. To prevent results from being influenced by the boundary conditions, curvature and bending moment were taken from the central section of the model. Thus, the models were loaded considering one bending cycle, varying the curvature of the central section up to $1m^{-1}$. Figure 3.44 illustrates the curvature-moment diagram for both models based on Witz (1996). Following the same color code used in the previous section, red indicates the response of the model with bimaterial and blue the response of the model without bimaterial.

The hysteretic cycles obtained are typical of bending behavior of flexible pipes. For small curvatures, the initial adhesion between layers precludes slipping and a “no-slip” condition prevails. Before this friction limit, the quasi-linear moment-curvature response is characterized by a high bending stiffness. By increasing the curvature, the interlayer friction limit is eventually overcome, allowing slippage between components. This second phase is known as “full-slip”. When the bent flexible pipe begins to be unloaded and the curvature reversed, the metallic layers are prevented from sliding once again until curvature reaches a new critical value. Therefore, load and unload paths in the moment-curvature diagram are not the same. This slip-stick mechanism leads to a typical hysteretic loop.

Figure 3.44 illustrates the hysteresis cycles obtained for the model with and without bima-

Figure 3.42: Model based on Sousa et al. (2013); with bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$.

Figure 3.43: Model based on Sousa et al. (2013); without bimaterial; contact pressure in MPa between the pressure armor and the inner sheath; visualization with and without the armor pressure layer; internal pressure of $10MPa$.

Figure 3.44: Model based on Witz (1996); with and without bimaterial; bending moment – curvature diagram; model lengths of 500mm; internal pressure of 30MPa.

terial. As expected, the differences in the calculation of contact pressures change the bending behavior. In the “no-slip” condition, both bending stiffnesses are similar. For the FE models, this initial stiffness depends greatly on the contact parameters, in particular on the stick stiffness, as observed by Dai, Ye and Sævik (2017). Also, the correlation of this stiffness with analytical models is somewhat compromised, since the plane-sections-remain-plane hypothesis from the analytical approaches ignores the shear strain of the plastic layers. This hypothesis makes analytical models stiffer in the “no-slip” region. Thus, the following analyses evaluate exclusively the “full-slip” region.

Comparing models with and without bimaterial, the bending stiffnesses in the “full-slip” region present considerable differences. The model with bimaterial presents a stiffness of 0.895 kN.m^2 in the “full-slip” region, against 1.886 kN.m^2 of the model without bimaterial. Both stiffnesses were calculated through a linear fit considering only the first part of the bending cycle. A linear regression was conducted in the range of curvatures between 0.3 m^{-1} and 1.0 m^{-1} , with the slope of each fitting taken as the tangent bending stiffness for the respective cases. These linear fittings are depicted in Figure 3.44 as dashed lines.

With the same internal pressure of 30MPa, Witz (1996) obtained experimental values for the “full-slip” bending stiffness of 0.725 kN.m^2 and 0.896 kN.m^2 . The second value is very close to the one calculated for the model with bimaterial. Nevertheless, it is noteworthy that this “full-slip” stiffness is highly dependent on the calculation method. A slightly different curvature range for the linear regression changes the estimated stiffnesses considerably. For this reason, correlation between results from different approaches and different sources must

be done carefully and with a deep understanding of all factors involved. The comparisons cannot be summarized by a simple assessment of stiffness values.

Figure 3.45 illustrates the hysteresis cycles again, adding the experimental data from Witz (1996). The correlation between numerical and experimental results is satisfactory.

Figure 3.45: Experimental results from Witz (1996) compared to bending response of the model with and without bimaterial; model lengths of 500mm ; internal pressure of 30MPa .

In the “full-slip” condition, none of the three results exhibits a strict linear behavior. As the curvature increases, there is a slight tendency for the bending stiffness to increase. It is worth remembering that usually this stiffness is mainly ruled by the outer polymeric layer, as demonstrated by several analytical models (e.g. the model briefly presented in Sousa et al. (2015)). As this flexible pipe does not have an outer sheath, the contribution of the inner sheath is considerable and the other layers also play an important role in the bending mechanism. Since no single layer stands out over the others, contact interactions become more relevant.

The variances observed in contact pressure results explain the different bending stiffnesses between models with and without bimaterial. The model without bimaterial presents a higher stiffness, indicating that the helical tendons have greater difficulty sliding. The non-uniformity of contact pressures certainly induces regions where the bonded contact prevails, increasing stiffness. As seen above, for the model with bimaterial, the contact pressures are smaller and have no peaks. This uniform contact pressure benefits sliding between layers and a condition closer to the theoretical “full-slip” condition. For this reason, the bending stiffness of the model with bimaterial is smaller.

Notice that the red hysteresis cycle is closed, while the blue curve presents higher moment

after completing the cycle. This indicates, for the model without bimaterial, that contact conditions and position of the wires have changed throughout the cycle and have not returned to their initial position. These changes are responsible for the vertical drift of the blue curve at the end of the cycle.

Figure 3.46 shows the same bending cycles for the flexible pipe based on Sousa et al. (2013). Unlike the above case, this flexible pipe has an outer polymeric sheath.

Figure 3.46: Models based on Sousa et al. (2013); with and without bimaterial; bending moment – curvature diagram; model lengths of 500mm ; internal pressure of 30MPa .

The diagram shows the hysteresis cycles obtained for the models with and without bimaterial, following the same color identification used throughout this study. The load case is similar to the one shown above, also with an internal pressure of 30MPa . The bending stiffnesses in the diagram (dashed lines) are similarly calculated. With the presence of an outer sheath, the differences between results from models with and without bimaterial are less pronounced during the beginning of the first loading cycle. The bending stiffness is ruled by the outer layer and the contact interactions between layers do not have the same relative importance as in the previous case.

Sousa et al. (2015) presents values for the bending stiffness of this flexible pipe, obtained analytically and with a FE model. The stiffnesses in the “no-slip” condition are over 10 times higher than those herein found, due to the reasons already mentioned. Both analytical and FE models from Sousa et al. (2015) ignore the shear of the plastic layers (the FE model uses shell elements for the polymeric sheaths). The “full-slip” analytical bending stiffness presented by Sousa et al. (2015) was 1.09kN.m^2 . The value obtained with the FE model was 1.67kN.m^2 . These values have the same order of magnitude of those herein obtained. As for the flexible

pipe from Witz (1996), the model with bimaterial is slightly less rigid in the “full-slip” region. The reasons are the same explained above.

In this case study, when the curvature starts to be reversed, the model with bimaterial presented a higher stiffness, until the calculations were stopped due to convergence problems. Notice that before returning to the straight configuration (null curvature), the calculation for the model with bimaterial was interrupted. The model without bimaterial presented no convergence problems and the loading and unloading cycle was completed. As in the previous model, at the end of the bending cycle the moment is slightly higher and the cycle is not closed.

The reason why the bimaterial model presented convergence problems is illustrated in the following images. Figure 3.47 shows the internal tensile armor layer at the beginning of the simulation. The colors indicate the minor strain on the elements assigned with artificial material. Results consider the main directions of strain. At the beginning of the simulation, there is no loading and Figure 3.47 presents a zero-strain condition. The image above shows a side view of the layer, while the image below illustrates a closer look at some elements.

Following, Figure 3.48 shows the same result for the last calculated iteration, just before the convergence problem appears. After being loaded and unloaded, the metallic wires do not return to the original position. This variation in their geometric configuration explains the increase in bending stiffness of the model with bimaterial. During unloading, some wires did not change completely from a stick to a slip condition.

Even though the model is almost straight, Figure 3.48 shows the artificial elements compressed, with strains greater than 400%. As the artificial elements are not able to simulate a close contact condition between wires, convergence problems appear. Excessive deformation of the artificial elements gives rise to numerical problems. This could be solved by increasing the modulus of elasticity of these elements, but this parameter remains to be investigated in future works. What can be stated is that, although the simulation is interrupted, the lack of numerical convergence indicates the imminence of contact between adjacent wires. The downside is that the simulation is interrupted.

The same does not happen for the model without bimaterial. If the contact between adjacent helical wires is not represented, the simulation is not interrupted by numerical issues. However, this apparently good numerical behavior should be regarded with due care. As an example, Figure 3.49 illustrates the outer tensile armor layer for a given moment of the simulation. As the lateral surfaces of the helical tendons are not assigned for contact, calculations do not detect geometrical interferences between adjacent wires during bending. The simulation does not face convergence problems. Nevertheless, it is not able to represent the bending behavior beyond values of curvature at which contact between wires of the same layer is achieved.

Figure 3.47: Model based on Sousa et al. (2013); without bimaterial; inner tensile armor layers before applying bending load; color indicate minor principal strain in elements with artificial material.

Figure 3.48: Model based on Sousa et al. (2013); without bimaterial; inner tensile armor layers after applying bending load and unload; color indicate minor principal strain in elements with artificial material.

Figure 3.49: Model based on Sousa et al. (2013); without bimaterial; outer tensile armor layer after bending; geometric interference between elements from adjacent metallic wires.

The zoomed image provides a clear view of the geometric interference between elements. To avoid these interferences, the contacts must be reconfigured by increasing modeling efforts. A layer with n metallic wires needs to be modeled assigning all n existing contact interfaces. The contact algorithms would certainly be much more computationally expensive. These tests are also to be carried out in further work.

3.6 FINAL REMARKS

The case studies achieved satisfactory results for the proposed load cases. In general, the response of the numerical models was satisfactorily coherent with results from other works. The case studies demonstrate the effectiveness and potential of the bimaterial modeling approach proposed. However, the results have also indicated that the methodology needs to be further explored and evaluated.

The axisymmetric loads did not induce large deformations in the elements assigned with artificial material. For this reason, the bimaterial approach has proved advantageous in these cases. For the conditions tested, the bimaterial provides a slight decrease in processing times. However, for other situations, this approach may not be advantageous.

As the bending results have confirmed, the bimaterial layers lead to numerical problems if the elements assigned with the artificial material present excessive distortion. On the other

hand, the bimaterial approach is beneficial to calculate contact loads between layers. The continuity of the cylindrical contact surfaces aids calculation of interactions between layers without requiring a very refined mesh for the polymeric sheaths.

Results for other load cases must also be investigated to validate, or not, the modeling approach under different conditions. Regarding the computational requirements to run the models, the bimaterial approach involved more memory during calculations. However, the longer models with and without bimaterial always presented similar stiffness. This indicates that the bimaterial approach does not effect significant changes on the structural behavior of the model. The following table summarizes the main advantages and drawbacks of the proposed modeling technique:

Table 3.26 : Main advantages and drawbacks of the bimaterial modeling technique.

Advantages	<ul style="list-style-type: none"> - Modeling convenience for software users, without the need of declaring several contact surfaces between metallic wires; - Processing times are slightly lower when artificial elements are not too much distorted; - Structural behavior of layers not affected by the bimaterial approach; - Less refined mesh for plastic sheaths without compromising contact calculations.
Drawbacks	<ul style="list-style-type: none"> - Calculations require more pre-allocated RAM memory; - Convergence difficulties if artificial elements are highly distorted.

It is worth remembering that this analysis pertains only to the adopted calculation parameters. Nothing can be said about the computational performance of the bimaterial approach under different calculation conditions. The use of another FE software or another solver may lead to differences in computational performance.

3.6.1 Further work

Some points remain to be studied and further explored. The quality of results herein presented tends to validate the simplification methodology, but the points listed below are yet to be analyzed:

- The representation of structural tapes: as pointed out by the partial results presented, the absence of these tapes in the interface of metallic layers may have caused important

differences in the structural behavior of the models. These tapes could be modeled through geometric elements or considering more complex contact formulations between layers;

- The study of friction coefficients: the Coulomb model to represent the tangential contact behavior between layers is to be investigated. This parameter could change not only the structural behavior of the model, but is also guaranteed to influence the computational performance;
- More complex load cases: for example, instability loads could be studied with this innovative modeling approach. Loading conditions inducing higher distortion in the artificial elements must also be better analyzed. Questions regarding whether the proposed methodology is able to reproduce the *birdcaging* or the wire lateral buckling instabilities remain to be answered.

4 VISCOELASTIC

As already mentioned, flexible pipes are complex structures composed of several metallic helical layers and polymeric extruded sheaths, disposed in concentric arrangements. The importance of structural analysis based on numerical FE models is described in the first chapters of this thesis. These powerful models, when used wisely, enable analysts to assess stress and strain fields in all structural pipe components.

Internal friction is a commonly studied feature, since its effect is associated to the hysteretic bending behavior. On the other hand, it is well-known that the bending stiffness of flexible pipes is strongly dominated by the extruded external polymeric layer. The analytical models reviewed in Chapter 2 allow one to evaluate the individual contributions of each layer to the overall bending stiffness. Ramos Jr. and Pesce (2004) focuses solely on this assessment.

In spite of the great number of studies on structural pipe components, it is not easy to find numerical and analytical models that take into account the viscoelastic mechanical behavior, which characterizes most of the polymeric external layers. This chapter addresses this particular point numerically, investigating the effect of the external sheath's viscoelastic behavior on the local bending response of flexible pipes.

The author developed a FE model that takes into account internal friction between layers, as well as the viscoelastic behavior of the external polymeric sheath. Experimental data from Rabelo (2014) is used to assess the viscoelastic response of High-Density Polyethylene (HDPE). To represent the time-dependent response of the polymeric material, a linear viscoelastic rheological model is calibrated using these experimental results.

Subsequently, ABAQUS software is used to generate a representative model of a 4-inches flexible pipe with a viscoelastic outer sheath. This model is then used in a case study to assess and discuss the viscoelastic effect on the bending of a typical flexible pipe. The study compares internal friction with viscoelastic effects, first by considering them separately and then by combining both models. The results and development herein presented are partially based on Santos et al. (2015a).

4.1 VISCOELASTICITY IN THE GLOBAL ANALYSES

During the design stage of a flexible pipe, also known as riser in this context, its structural behavior is of major importance, both in a global and in a local scale. An optimized riser project involves many interactions between global dynamic analysis and local structural calculations; Pesce (1997). The global dynamic analysis generates loads used to design the flexible pipe's cross section. Then, considering the flexible pipe structure, the design engineer estimates the

main mechanical characteristics to be used as inputs into a second global dynamic analysis, thus closing a step of the design optimization loop.

It is well-known (see Aranha, Martins and Pesce (1997), Pesce and Martins (2005)) that riser global dynamic response is dominated by geometric rigidity, and therefore controlled by tension. However, locally, at the top and the Touch Down Zone (TDZ) or inside regions of sudden variation of curvature (e.g., lazy-wave hump), bending stiffness effects are highly significant.

Linear bending behavior is usually considered as a first step of the design loop. However, when the riser is subjected to high mechanical loads this results in considerable contact pressures between layers. Consequently, internal friction between components may play a substantial role. This effect dissipates energy by adding hysteresis to the bending stiffness behavior. A number of authors describe this phenomenon in detail (e.g., Boom and de Boer (1992), Sævik (2011) and Sousa et al. (2012)), presenting analytical and numerical tools capable of predicting the bending stiffness in different situations. In addition, some papers (e.g., Tan, Quiggin and Sheldrake (2007) and Leroy et al. (2010)) estimate the hysteretic cycle due to a cyclic bending load.

The bending stiffness of the flexible pipe is a main input in the global modeling process, because it has a major influence on the resulting curvatures along the line and thereby influences directly the loads applied to the structure. Furthermore, it also affects the dynamic analysis, since the hysteretic bending behavior can be regarded as an energy dissipation factor. Péronne et al. (2015) describes a refinement of the dynamic global analysis and the fatigue analysis with the Hysteretic Bending Stiffness (HBS) of flexible pipes, showing the importance of taking energy dissipation into account during the design project.

As frequently pointed out, the external polymeric sheath dominates the bending stiffness behavior during the second phase of bending, once internal slippage has initiated. On the other hand, it is well-known that polymers are commonly viscoelastic materials. Despite this fact, at least to the authors' knowledge, no attempt has been made to assess the role a viscoelastic behavior could play in the bending of flexible pipes. In the existent models, polymers are usually characterized by a simple linear elastic constitutive law.

4.2 HYSTERETIC BENDING BEHAVIOR

This section describes according to literature the typical hysteretic bending behavior of flexible pipes due only to the contact interactions between layers. As previously mentioned, many authors describe the bending stiffness of risers as depending only on the curvature of the central axis. The Hysteretic Bending Stiffness (HBS) response is ruled by a slip-stick

mechanism, activated by the contact pressures between layers. These contact pressures happen when the pipe is subjected to axisymmetrical loads (tension, torsion, pressure), increasing as these loads become significant. Figure 4.1 shows a sketch of a typical HBS response, depending only on the curvature of the pipe. The hysteresis cycle is very similar to the numerical results obtained in the previous chapter, concerning bending loads (Figure 3.44 on page 115 and Figure 3.46 on page 117).

Figure 4.1: Schematic representation of the hysteretic moment-curvature diagram.

Source: Péronne et al. (2015).

When the applied curvature is small, the initial friction between layers precludes slipping. Until this friction limit is overcome, the bending load is resisted conjunctly by all metallic and polymeric components. It leads to a quasi-linear moment-curvature response, with a high flexural stiffness. The tangent to the curve for this small curvature state is usually called “no-slip” bending stiffness, illustrated as EI_{Rigid} in Figure 4.1.

By increasing the curvature, the limit of interlayer friction is eventually overcome, allowing the slippage between components. Due to the relative movement between the helically wound wires and their adjacent layers, the metallic ones cease to resist significantly to the bending loads. In this second phase, analogously to a bent helical spring, the metallic layers of the flexible pipe offer a very low bending resistance, which causes the contribution of the polymeric layers to prevail. This is the “full-slip” bending stiffness phase, when the polymeric layers regulate the bending stiffness of the riser, designated as EI_{linear} in Figure 4.1. The curvature value corresponding to the internal friction limit (C_c in Figure 4.1) is usually called “critical curvature”. It is worth mentioning that when polymers are represented by a linear elastic material law, the resultant moment-curvature function must be nearly linear.

If the critical curvature is exceeded as the flexible pipe is loaded and unloaded, there is

hysteretic behavior. It is characterized by the (quasi-) bi-linear behavior shown in Figure 4.1. When the bent flexible pipe is unloaded, reversing the curvature, the metallic layers are prevented from sliding once again by interlayer friction, until curvature reaches a new critical value. Therefore, load and unload paths in the moment-curvature diagram are not the same. This mechanism leads to a hysteretic loop, with the area inside the loop being proportional to the energy dissipation (per unit of pipe length) in a given bending cycle.

Grealish, Smith and Zimmerman (2006), Smith et al. (2007) and Péronne et al. (2015) emphasize that this hysteretic response has a substantial impact on the fatigue life of flexible pipes. They also show that curvature amplitudes resultant from the global dynamic of the riser depend on its HBS. It is pointed out that a correct calculation of the HBS provides more accurate fatigue predictions. Meanwhile, the authors present different theoretical approaches leading to quite different curves. Substantial differences can be found between the coefficient of friction used by the authors, with values varying between 0.07 and 0.20.

4.3 HDPE VISCOELASTICITY

To consider the viscoelastic structural effects of the external polymeric sheath in the bending stiffness of flexible pipes, a linear viscoelastic constitutive behavior was implemented in a FE model with bimaterial. The proposed approach is described in the next section, through a case study with a 4-inches flexible pipe.

Experimental results of relaxation and creep tests on HDPE carried out by Rabelo (2014) were used to calculate the Prony coefficients for the linear viscoelastic constitutive model. The process is described in greater detail in Appendix A. Figure 4.2 illustrates the specimens produced from a HDPE pipe. Rabelo (2014) conducted relaxation tests at three different strain levels (0.45%, 1.00% and 2.50%) and creep tests at three different stress levels (3MPa, 8MPa and 18MPa). Figure 4.3 illustrates some relaxation test results and Figure 4.4 illustrates creep test results.

The averaged experimental curves from Rabelo (2014) were used to fit a Prony series with four coefficients, presented in Table 4.1.

Equation (4.1) recalls the normalized Prony series given in Appendix A by Equation (A.5), but now considering four coefficients. The graphical representation is illustrated in Figure 4.5. The E_0 for the HDPE (instantaneous elastic modulus used to normalize the Prony series) is presented in the next section, Table 4.2.

$$\frac{Y(t)}{E_0} = 1 - [p_1 (1 - e^{-t/\tau_1}) + p_2 (1 - e^{-t/\tau_2}) + p_3 (1 - e^{-t/\tau_3}) + p_4 (1 - e^{-t/\tau_4})] \quad (4.1)$$

Figure 4.2: (a) Specimens manufactured from a HDPE tube; (b) Specimens dimensions according to ISO6259-3; (c) Example of a specimen.

(c)
Source: Rabelo (2014).

Table 4.1 : Prony series coefficients

i	p_i	$\tau_i[s]$
1	0.132	0.512
2	0.211	13.4
3	0.185	156
4	0.171	2294

4.4 CASE STUDY

As previously mentioned, the HBS is affected by the external polymeric sheath. The material of this outer layer is commonly HDPE, a viscoelastic material whose time-dependent response can be reproduced by a Prony series. The present case study runs a FE model with the bimaterial concept. It is similar to the models presented in the previous chapter, using the same considerations and the same model length of the bending cases ($500mm$). The proposed numerical approach is capable of reproducing the HBS of the modeled flexible pipe. Nevertheless, the improved model herein presented takes into account both the interlayer friction (already considered in the previous model) and the viscoelastic behavior of the external sheath.

The FE model reproduces a 4-inches flexible pipe. Table 4.2 recalls its main properties,

Figure 4.3: Example of the experimental HDPE relaxation tests.

Source: Rabelo (2014)

Figure 4.4: Example of the experimental HDPE creep tests.

Source: Rabelo (2014).

Figure 4.5: Normalized Prony series calculated with Equation (4.1) using the coefficients presented in Table 4.1.

while Figure 4.6 presents the overview of the numerical model. Figure 4.7 illustrates its cross section generated with *PipeDesign* , a software briefly mentioned in the literature review on page 47.

Figure 4.6: 3D view from the ABAQUS FE model of the 4-inches flexible pipe; model length of 500mm.

As seen in the previous chapter, the proposed FE model is capable of reproducing the HBS for this flexible pipe under specific conditions set by the user. An important detail to represent the flexible pipe's HBS is to keep all layers in contact. A well-established contact between layers also helps reduce numerical issues and improve convergence. In order to ensure this interlayer contact during calculations, the modeled pipe is subjected to a small external pressure of $1MPa$.

Using the same procedures adopted in the previous analysis, a sinusoidal bending load is

Table 4.2 : Layers of the Prysmian 4-inches flexible pipe.

Layer	Properties
1: Carcass	ID = 101.60 mm Layer thickness = 5.00 mm Number of wires = 1 Lay angle = -87.97° Cross-sectional area = 35.71 mm^2 Young's modulus = 191 GPa Poisson's ratio = 0.3
2: Inner sheath	Layer thickness = 5.00 mm Young's modulus = 284 MPa Poisson's ratio = 0.47
3: Pressure armor	Layer thickness = 6.45 mm Number of wires = 1 Lay angle = -88.58° Cross-sectional area = 53.73 mm^2 Young's modulus = 200 GPa Poisson's ratio = 0.3
4: Inner tensile armor	Layer thickness = 2.00 mm Number of wires = 46 Lay angle = 36.00° Cross-sectional area = 14 mm^2 Young's modulus = 200 GPa Poisson's ratio = 0.3
5: Outer tensile armor	Layer thickness = 2.00 mm Number of wires = 49 Lay angle = -34.00° Cross-sectional area = 14 mm^2 Young's modulus = 200 GPa Poisson's ratio = 0.3
6: Outer sheath	Layer thickness = 6.00 mm Young's modulus $E_0 = 442 \text{ MPa}$ Poisson's ratio = 0.47

Figure 4.7: Cross section of 4-inches flexible pipe; 2D view obtained with *PipeDesign*.

applied around the non-deformed configuration to obtain the hysteretic response. This load is inputted using Multi-Point Constraints (MPC) and imposing a rotation at both extremities until the desired curvature is reached. The maximum curvature was chosen as 50% of the value corresponding to the Minimum Bending Radius (MBR) of this particular pipe, which is equal to $1.36m$. Therefore, curvature in the HBS diagrams is between $\pm 0.37m^{-1}$.

The Coulomb friction coefficient μ and the Prony coefficients are parameters of this analysis. Thus, the influence of these two inputs in the hysteretic bending stiffness (HBS) of the flexible pipe is analyzed separately. Considering these two parameters at a time, the model reveals the influence of these nonlinear effects on the bending load. We are now able to more accurately reproduce a realistic case by modeling both effects together.

4.4.1 HBS due to friction only

The first analyzed case is typically found in the literature: friction is the sole factor considered, all materials assigned with elastic constitutive laws, ruled by a Young's modulus E and a Poisson's ratio ν (using parameters presented in Table 4.2). Figure 4.8 presents the HBS for different coefficients of friction μ . The black curve indicates the null friction case, while the other three curves refer to $\mu = 0.10$; 0.15 ; 0.20 . It is worth mentioning that this example works with a Young's modulus of the HDPE corresponding to a case where there is no viscous behavior. To not consider viscoelasticity is similar to having $t \rightarrow \infty$ in Equation (4.1). In other words, the outer sheath in this first analysis is modeled with the long-term

Young's modulus E_∞ given by Equation (4.2).

$$Y(\infty) = E_\infty = [1 - p_1 - p_2 - p_3 - p_4] E_0 = 0.301E_0 \quad (4.2)$$

Figure 4.8: HBS diagram obtained using a Coulomb's friction model and HDPE outer sheath with elastic behavior.

As expected, the simulations with a frictionless contact do not present a hysteretic cycle (black curve). With all materials assigned with linear-elastic constitutive laws, dissipation effects are not present and the result is characterized by a linear bending stiffness behavior in the whole range of imposed curvatures of imposed curvature.

By introducing a Coulomb's frictional contact, the model now displays a flexible pipe's typical HBS. As expected, the area of the hysteretic cycles increases with the friction coefficient, thus signaling the unequivocal relation between this parameter and the energy dissipation per bending cycle.

This first HBS diagram, only considering Coulomb's frictional contact mechanisms, is not time-dependent. As previously noted, the bending load was applied sinusoidally, with a period of oscillation of 1s. The same results are obtained even if a different period of oscillation or wave format are used, as the contact model does not depend on the load history and no structural dynamics are implied.

4.4.2 HBS due to viscoelasticity only

Momentarily leaving aside the interlayer friction, the next HBS diagram was generated taking into account exclusively the effect of the external layer's viscoelasticity. For this second case, the response of the FE model depends on load dynamics.

The bending load was applied with four different oscillation periods: $1s$, $10s$, $100s$ and $1000s$. The results are presented in Figure 4.9. This wide range spans all the typical load periods a real flexible pipe might be subjected to, from vortex-induced vibrations ($1s - 10s$), floating platform induced motions ($10s - 500s$) to quasi-static loading; Pesce and Martins (2005). Notice that these oscillation periods are also compatible with and convenient for the use of four coefficients in the Prony Series (one decade for each coefficient). The black curve representing linear elastic case without friction in Figure 4.8 is recalled in Figure 4.9 as a reference.

Figure 4.9: HBS diagram with frictionless interlayer contact and HDPE with viscoelastic behavior. Sinusoidal loading with period T

Two main conclusions can be drawn from the results presented in Figure 4.9. Firstly, as expected, when considering the viscoelasticity, the bending stiffness of the flexible pipe appears to be proportional to load speed. The yellow hysteresis cycle, obtained with an oscillation period of $1s$, has the highest bending stiffness. Decreasing the load speed for bigger oscillation periods, the bending stiffness clearly decreases, until the period of $1000s$. The speed load depicted by the red curve is quasi-static and the viscoelasticity has no significant effect

on the average bending stiffness of the flexible pipe. That makes the inclination of the red ellipse comparable to the inclination of the black curve (no friction, linear-elastic materials).

A second remark about Figure 4.9 is that slow bending cycles reveal larger hysteretic loop areas, indicating that more energy is dissipated per cycle. The area of the red ellipse obtained by cyclically loading the flexible pipe with a period of 1000s is larger than the other ones. On the other hand, the dissipated energy per time (power) is lower.

4.4.3 HBS due to friction and viscoelasticity

A third analysis was developed considering simultaneously the effects of frictional interlayer contact and the viscoelasticity of the external HDPE layer. The results, presented in Figure 4.10, were obtained setting the friction coefficient $\mu = 0.15$ and varying the loading oscillation period. The black curve (linear-elastic material, considering only friction) is presented again as reference.

Figure 4.10: HBS diagram combining frictional ($\mu = 0.15$) interlayer contact with HDPE viscoelastic behavior.

Figure 4.10 shows the same effect noticed in Figure 4.9: “full-slip” bending stiffness is proportional to load speed. On the other hand, the “no-slip” bending stiffness is not significantly affected by the constitutive law of the external layer, as expected.

Another visible relation is that the energy dissipated per cycle is slightly affected by load speed. However, viscoelasticity clearly produces a marked increase in the energy dissipated per

cycle, if compared to the pure friction contact case. This result shows that the viscoelasticity of the external HDPE layer has a clear and substantial contribution to the dissipation of energy during the bending cycle of flexible pipes.

Finally, Figure 4.11 makes a comparison for each simulated loading period ($1s$, $10s$, $100s$ and $1000s$). Each graphic compares the following curves: only friction considered, no viscoelastic effect (black); only viscoelasticity considered, no frictional contact (red); the combination of both effects considered (blue).

Figure 4.11: HBS diagrams considering sinusoidal load periods of $1s$, $10s$, $100s$ and $1000s$.

The loop areas are proportional to the energy dissipation per cycle, and thus the results in Figure 4.11 reveal an interesting conclusion: the sum of the areas corresponding to the isolated effects (black and red curve areas) is smaller than the area of the combined case (blue curve area). In other words, combining these two effects is not the same as adding their distinct values.

Together, the two effects – dry friction and viscous ones – act in synergy, dissipating more

energy. As already noticed, the “no-slip” bending stiffness is not considerably affected by the behavior of the riser’s external layer. The black curves and the blue curves are almost coincident in the “no-slip” zone. In contrast, in the “full-slip” regime, the external sheath behavior rules the hysteretic mechanism. The slope of the blue loop in the “full-slip” zone visibly follows the inclination of the red ellipses. Therefore, when considering both effects (interlayer friction and viscoelasticity), the mechanisms are synergistically combined. The viscoelasticity affects directly both the hysteretic loops and the energy dissipation during bending. It is worth remembering that structural damping is dominant in structural dynamics, substantially affecting curvature amplitudes and, in consequence, the fatigue life of flexible pipes.

4.5 FINAL REMARKS

This chapter has briefly assessed the influence of viscoelasticity in the hysteretic behavior of a flexible pipe subjected to bending. In a first step of the presented case study, the well-known typical loop of moment-curvature diagram was reproduced, considering only interlayer friction. In a second step, a new (and almost elliptical) hysteretic behavior was obtained in a frictionless simulation, considering a linear viscoelastic constitutive law for the external polymeric layer material. When both effects were considered together, the influence of the viscoelasticity is still significant: the hysteretic loops change with the load history and their areas are relatively larger than the typical ones corresponding to pure frictional interlayer contact.

This study shows that, at least for the conditions simulated with this particular flexible pipe, the effect of the viscoelasticity of the polymeric sheaths is important. It is worth mentioning that the viscous behavior was considered only for the outer layer, and since typical flexible pipes are composed of two (even three) polymeric sheaths, the effects herein observed might be even more expressive than expected.

4.5.1 Further work

Different combinations of coefficients of friction and speed loads are to be tested. The graphics herein presented lend themselves to further in-depth discussion, if we are to quantify the curve areas and the dissipated power per cycle. Other flexible pipes and even risers can be tested with this methodology.

In conclusion, this assessment can be complemented by a global dynamic analysis using as inputs the obtained hysteretic loops to quantify the influence of viscoelasticity on the fatigue life of a flexible pipe.

PART II – UMBILICAL CABLES

5 CRUSHING NUMERICAL IN-PLANE ANALYSIS

The main purpose of this chapter is to present the numerical methodology developed to study umbilical cable components under crushing loads. The loads from the laying operation comprise the radial load from the tensioner shoes and the squeezing effects, associated not only with the tensile armors, but also with all helically wound components under tension.

The results and development herein presented are partially based on Santos et al. (2015c), as well as on results from a research project on umbilical cables developed as a partnership between *Prysmian Group* and the Offshore Mechanics Laboratory (LMO) at the *Escola Politécnica da Universidade de São Paulo* (EPUSP). The respective technical reports are listed in the final references.

As umbilical cables combine elements with distinct mechanical behavior and geometries, predicting their structural response turns out to be a very complex task. The literature review carried out in Chapter 2 demonstrated that most authors have been dealing analytically with such a complex problem for flexible pipes. On the other hand, the sophistication of these structures and the existence of several nonlinearities pose restrictions for the analytical approaches, leading the studies with umbilicals to resort to techniques involving numerical simulations. Chapter 2 has exemplified the current state of the art concerning umbilical FE modeling.

All umbilical studies previously reviewed are based on case studies with existing flexible structures. The enormous variability of cross section arrangements makes it very difficult to study these structures in general terms (e.g. addressing a general case and implementing a generic approach). Each oil field application demands unique designs. In addition, studies on umbilical cables under crushing loads are not commonly found in the technical literature.

The strategy adopted by this research is meant to establish a methodology applicable to any chosen cross section. It aims to provide a deeper understanding of umbilical performance and technical risks during laying operation. The whole context motivates the pursuit of an alternative way to perform this local structural analysis with umbilicals. There is need of a tool general enough to allow the modeling of any cross section, yet simple enough to overcome modeling difficulties, reducing computer efforts and allowing fast analyses. In addition, the developed tool needs to be proven reliable and representative of the crushing phenomenon. This last assumption requires experimental data to correlate numerical and experimental results.

The first part of this study with umbilical cables proposes a two-dimensional FE model capable of taking into account some important three-dimensional effects, such as the squeezing caused by the tensioning of the helical armor layers. A similar idea was reviewed in Chapter 2, in a paper from Dixon and Zhao (2008). Their two-dimensional model is illustrated in Figure 2.11 on page 54. Unfortunately, the authors do not give any details about the numerical analysis.

The numerical approach herein developed is asserted by three numerical case studies: one Steel Tube Umbilical (STU) and two umbilicals composed by High Collapse Resistant (HCR) hoses.

In Chapter 6, the second part of this crushing study complements the numerical analysis. A special crushing test setup is developed to assess displacements on umbilical cable cross sections. Using a high-resolution Digital Image Correlation (DIC) system, experiments with the same umbilical cables are carried out in a $250kN$ mechanical test rig. Besides measuring the movements of internal components, applied crushing forces and shoe displacements are acquired.

Determining the umbilical cable crushing limits is not an easy task, even experimentally. Monitoring deformations without interfering in the components' behavior is not trivial. The focus of the structural analysis is the integrity of the tubular components, particularly the steel tubes and HCR hoses, addressed by both the numerical and the experimental approaches. The collapse of a steel tube or a HCR hose can compromise the cable.

5.1 LAYING OPERATION AND LOADS

The laying operation of umbilical cables is an important stage to be analyzed during the structural design. Installation loading may generate stresses that can damage some important components, thoroughly compromising an umbilical's functionality. The magnitude of these efforts grows with the water depth.

Umbilicals are installed by launching vessels equipped with tensioners to control the release of the line. These tensioners synchronize their launch speed with the displacement of the ship. To hold and control the launch operation, radial loads are applied on the umbilical through V-shaped shoes installed in tensioners (caterpillars). These radial loads, typically applied by three or four shoes with an opening angle between 120° and 180° , must support the line's weight and the dynamic tension due to the vessel's response to waves. Figure 5.1 shows a typical tensioner and a detail of the caterpillar geometry.

During installation, radial and axial efforts intensify as the water depth increases. The radial compressive loads applied to the external polymeric sheath are transmitted and distributed throughout the umbilical's internal components. Additionally, the tensioned helical components squeeze the underlying layers. Therefore, installation loads are separated into two groups:

- Axial loads, due to static and dynamic tensions acting on the line, mostly resisted by the tensile armors, sometimes along with steel tubes;

Figure 5.1: Tensioner used during a riser laying operation and detail of a caterpillar cross section.

Source: Costa (2003).

- Transversal loads (both proportional to the axial loading):
 - Crushing: related to the compressive loads from the caterpillar shoes;
 - Squeezing: due to the helically wound tensioned components.

5.2 UMBILICAL CABLES UNDER ANALYSIS

As already mentioned, this crushing analysis focuses on the integrity of the tubular components; more specifically, the steel tubes in the case of a steel tube umbilical (STU) and the HCR hoses for umbilicals with these components.

The next pages illustrate three umbilical cables from *Prysmian Group* used in the numerical and experimental case studies⁴. For the sake of organization, the three umbilicals are hereafter named from 1 to 3. Figure 5.2 illustrates the cross section of Cable 1, a STU; Figure 5.3 shows the cross section of Cable 2, an umbilical with thermoplastic and HCR hoses; and Figure 5.4 presents the cross section of Cable 3, which also has HCR and thermoplastic hoses.

5.3 NUMERICAL IN-PLANE MODELING METHODOLOGY

The developed modeling methodology comprises a joint analysis using a two-dimensional FE model fed by an analytical model, which introduces some three-dimensional effects. A combined analytical-numerical approach is much simpler to implement than a full three-dimensional

⁴For the sake of confidentiality, we cannot provide more detailed descriptions.

Figure 5.2: Cable 1 cross section: central electrical core (power + signal); 12 steel tubes; double armored (flat wires).

Source: Prysmian Group.

Figure 5.3: Cable 2 cross section: central electrical cable; 9 thermoplastic hoses + thermoplastic HCR hoses; quadruple armored (flat wires).

Source: Prysmian Group.

Figure 5.4: Cable 3 cross section: central electrical core (power + signal); 12 thermoplastic hoses + 3 thermoplastic HCR hoses; double armored (flat wires).

Source: Prysmian Group.

one and it is meant to obtain results efficiently, without requiring considerable computational capacity.

The next sections present and discuss modeling hypotheses and methodology, describing how three-dimensional effects and interactions among all cable components are treated.

5.3.1 The two-dimensional approach

The two-dimensional FEA was developed using ABAQUS. The numerical model is meant to obtain computational results efficiently. Even though the working model is two-dimensional, the proposed methodology takes into account some three-dimensional effects associated to the helical geometry of structural components. The umbilical cross section is modeled in order to assess load distribution and strain fields in internal components.

The analysis flowchart below (Figure 5.5 on the next page) summarizes the procedures described in the next subsections. The block on the left-hand side represents the analytical equations, which feed the two-dimensional model.

The next sections introduce the hypotheses this methodology is based on, the analytical equations whose results are inputs for the FEA and details concerning modeling, meshing and

Figure 5.5: Modeling and analysis flowchart.

contact formulation.

5.3.2 Hypotheses

As mentioned, the proposed FE approach uses analytical relations whose development requires some simplifying assumptions, such as perfect circularity, leading to a uniform distribution of the squeezing load. A full 3D model would preclude such loading hypotheses. Being simpler, the 2D approach depends on the loading hypotheses from the three-dimensional equilibrium equations, which govern the analytical model. Such hypotheses, considered exclusively to calculate the squeezing loads, are consistent with the analytical studies reviewed in Chapter 2 on page 43 (e.g., Witz and Tan (1992a), Ramos Jr. and Pesce (2004), Pesce et al. (2010c)):

- Plane sections remain plane heuristically (extending Navier hypotheses);
- Isotropic linear elastic materials;
- Friction between layers is negligible;
- All layers have the same elongation and twisting angle per unit length.

5.3.3 Three-dimensional effects of the laying angle

As the laying angles of the helical layers may not be directly taken into account in the two-dimensional model, in order to preserve the proper axial stiffness of the umbilical an equivalent elastic modulus is defined and used for materials of each helical component. According to Pesce et al. (2008c), the axial load supported by each helical layer j with laying angle α_j is given by:

$$F_j = n_j A_j E \varepsilon_j \cos \alpha_j \quad (5.1)$$

Accordingly, the axial strain ε_j for a helical component is given by:

$$\varepsilon_j = \left(\cos^2 \alpha_j \frac{\Delta L}{L} + \sin \alpha_j \cos \alpha_j R_j \frac{\Delta \varphi}{L} + \sin^2 \alpha_j \frac{\Delta R_j}{R_j} \right) \quad (5.2)$$

The first two terms of Equation (5.2) are associated respectively to cable elongation and rotation; the third one is due to the corresponding layer radius variation. The order of relative importance of the last two terms is about 1% of the first one, if the helical laying angle is sufficiently small ($\sin^2 \alpha_j \approx 0$) and if the cable is well-designed considering torque balancing ($\Delta \varphi / L \approx 0$). In fact, if the rotation is small, the second term in Equation (5.2) is almost null. Neglecting these two terms, the axial force from Equation (5.1) can be rewritten as in Equation (5.3), defining an equivalent Young's modulus E_{eq} for the materials of each helical component in Equation (5.4).

$$F_j = n_j A_j (E \cos^3 \alpha_j) \left(\frac{\Delta L}{L} \right) = n_j A_j E_{eq} \left(\frac{\Delta L}{L} \right) \quad (5.3)$$

$$E_{eq} = E \cos^3 \alpha_j \quad (5.4)$$

5.3.4 Boundary conditions

When dealing with a two-dimensional model, some constraints must be used to avoid rigid body motions. The degrees of freedom of the central node are fixed, avoiding translation of the core. In addition, one single node from the external layer is constrained to move only into the radial direction, preventing rigid body rotation of the section.

5.3.5 Elements type and mesh

To introduce the axial loading in the two-dimensional model, the methodology considers generalized plane strain Finite Elements. This type of plane strain element allows inputting a load or a strain in the out-of-plane direction, i.e., $\varepsilon_z \neq 0$.

The two-dimensional model is discretized allowing at least two elements across each thick component. Figure 5.6 exemplifies the mesh of a steel tube (in gray) with a HDPE external coating (in yellow), both assigned with two elements across the thickness.

However, as these components may be subjected to radial loads that might induce flexural deformations and possibly lead to collapse, elements with incompatible modes are considered. This type of Finite Element is used to improve the flexural behavior, as shown in Appendix C. In short, the model considers two-dimensional elements within a generalized plane strain scheme and incompatible modes.

5.3.6 Contact formulation

Contacts are modeled with a surface-to-surface formulation and a linear penalty method. Geometrical nonlinearities from large displacements are also taken into account during calculations. The load is applied in increments and the stiffness matrix is constantly reevaluated, considering the updated geometry of the components. The iterative method continues until the whole load is applied.

The same modeling techniques deployed in Chapters 3 and 4 to model helical components are adapted herein for the two-dimensional armor layers. Considering armor wires one by one results in a significant increase of the complexity of the model, especially the contact interactions. Therefore, the same modeling strategy is herein adapted: bimaterial rings are used to characterize tensile armor layers, as shown in the upper image in Figure 5.7.

The gray colored regions that represent the tensile armor wires are assigned with the equivalent elastic modulus E_{eq} calculated with Equation (5.4), together with the actual Poisson's ratio of steel. The gaps (voids) between wires are filled with a hypothetical component, perfectly bonded to the wires. This component, represented by the white-colored elements, is assigned with the same artificial material from the previous chapters: negligible Young's modulus ($1MPa$) and null Poisson's ratio.

This approach is the same introduced in Chapter 3 to simplify FE modeling. Instead of having one interaction for each wire, the solver works only with one additional contact surface per layer, as also illustrated by the bottom image in Figure 5.7, where the red lines indicate contact surfaces.

Figure 5.6: Detail of the in-plane FE mesh. Steel tube (gray) and HDPE external coating (yellow) with two elements across the thickness.

Figure 5.7: At top, mesh detail of the bimaterial layer; at bottom, continuous contact surfaces (red) obtained by using bimaterial layers.

5.3.7 Squeezing effect

When tensioned, helical layers squeeze inner components. Besides the squeezing effect, these components are also subjected to axial elongation. In the 2D approach, to each structurally relevant helical component a corresponding radial load is considered and represented by an equivalent body force. This effect is taken into account through the equilibrium equation (5.5) below, proposed by Pesce et al. (2008c), based on Ramos Jr. and Pesce (2004). This equation evaluates the differential contact pressure on a particular helical component, from the previous knowledge of the following data: the elongation, $\Delta L/L$; the mean radius, R_j ; the torsion angle, $\Delta\varphi$; the radius variation, ΔR ; the elastic modulus of the layer, E ; the thickness, t_j ; and the laying angle, α_j .

$$\frac{(p_{c,j} - p_{c,j-1})}{E_j} = - \left(\frac{t_j}{R_j} \right) \sin^2 \alpha_j \left\{ \cos^2 \alpha_j \frac{\Delta L}{L} + \sin \alpha_j \cos \alpha_j R_j \frac{\Delta \varphi}{L} + \sin^2 \alpha_j \frac{\Delta R_j}{R_j} \right\} \quad (5.5)$$

By rearranging terms in Equation (5.5) and neglecting the last two terms, as done before for Equation (5.2), the equivalent radial body force q'' is defined as presented in Equation (5.6).

$$q''_j = \frac{(p_{c,j} - p_{c,j-1})}{t_j} = - \left(\frac{E_j}{R_j} \right) \sin^2 \alpha_j \cos^2 \alpha_j \left(\frac{\Delta L}{L} \right) \quad (5.6)$$

The squeezing effects are represented in the two-dimensional model by radial body forces q''_j combined with the axial traction. The axial load corresponding to each element is evaluated by imposing an elongation (uniformly distributed in the generalized plane strain section). The umbilical axial stiffness EA can be calculated as the ratio between the imposed elongation and the total axial load.

As Equation (5.6) depends on the Young's modulus of the component, the equivalent radial body load applied to the non-steel elements has almost no effect on the von Mises stress field obtained along the cross section. If compared to the squeezing loads caused by the tensile armors, it can be neglected (in an analytical model). However, for the sake of completeness, this load is considered in the two-dimensional model. In a full three-dimensional model, this load would be intrinsically represented. Figure 5.8 illustrates an example of the radial body forces squeezing the internal components.

5.3.8 Tensioner shoes

The radial loads due to the shoes are applied over the cross section using contact with rigid surfaces. These loads are not considered as concentrated, since they refer to the geometry

Figure 5.8: Radial body forces applied over the helical components.

of the shoes. In analytical approaches, concentrated loads are usually considered in order to simplify calculations, leading to conservative results (e.g., Martins, Pesce and Aranha (2003)).

On the other hand, the FE modeling allows a more realistic consideration of the configurations and geometry of the shoes, so that distinct and more representative laying conditions can be simulated. For the numerical case studies in the first part of this crushing assessment, four V-shaped shoes have been considered, with the same configuration presented in Figure 5.1 on page 141. They are diametrically displaced, compressing the cable section. Controlling displacements, the model can evaluate the shoes' reactive force as the crushing follows. Combined with tension and squeezing-due-to-tension loads previously described, the model characterizes the crushing condition.

5.3.9 Reinforced hoses as equivalent pipes

The reinforced hoses are one of the main targets of the present FE modeling strategy and must be well represented. Figure 5.9 illustrates a typical HCR hose, with an internal interlocked

carcass layer to increase the collapse resistance of the assembly.

Figure 5.9: Layers of a typical High Collapse Resistance (HCR) hose.

Source: ScoreSubsea (2016).

For a representative and reliable model of this tubular component, the crushing behavior of one HCR hose was assessed separately before being integrated in the umbilical section model. A main geometrical simplification related to the metallic carcass was conducted. It is impossible to geometrically represent the helical interlocked carcass in a two-dimensional model. Consequently, the original geometry was replaced by an equivalent tube (seen as a ring in the two-dimensional model), with an equivalent thickness and an identical stiffness to radial loads. The equivalent thickness calculation is based on the studies mentioned in Chapter 2: Pesce et al. (2010a), after Martins, Pesce and Aranha (2003). Details about this simplifying methodology and the crushing analysis of one single hose are provided in Appendix D.

5.4 CASE STUDIES: LAYING OPERATION CONDITIONS

Following the proposed modeling methodology, this section presents three numerical case studies with the umbilicals previously described, analyzing the combined effects of crushing loads due to rigid shoes and squeezing from the helically wound components. The FE models herein developed aim to calculate the maximum admissible crushing load applied by the shoes during a laying operation, monitoring damage (yielding) in steel tubes and HCR hoses. For the sake of simplicity, the first yielding of metallic components was considered as the crushing failure criteria. Nevertheless, it is important to highlight that the appearance of plastic strains does not necessarily mean that steel tubes and HCR hoses would have been compromised.

Each umbilical cross section is modeled using the methodology proposed and then axially loaded. As described in Section 5.3.5, generalized plane strain elements allow loads in the out-of-plane direction. For the load cases herein considered, the axial tension applied corresponds to half of the maximum working tension indicated in each cable's datasheet.

Once the axial load has been applied, the analytical equations presented in Section 5.3.7 are used to calculate the radial squeezing body forces. These body forces are applied to all helical components present in the cross section, as illustrated in Figure 5.8. Once these three-

dimensional effects are considered, radial displacements are imposed to four V-shaped shoes, crushing the umbilical cross section.

5.4.1 Cable 1

As previously stated, Cable 1 is a STU (Figure 5.2 on page 142) and the target of this first analysis is the integrity of the steel tubes. Figure 5.10 illustrates the cross section of the two-dimensional FE model obtained with the proposed methodology. Each color represents a different material. The stress-strain behavior of the super-duplex steel tubes material is modeled through a classic Ramberg-Osgood fitting function, whose parameters are specified in Figure 5.11.

Figure 5.10: FE in-plane model of Cable 1 cross section.

The applied axial traction corresponds to $540kN$ (50% of the maximum working tension of Cable 1). The pressurization in the steel tubes is also considered, according to the API Specification 17E and the umbilical cable datasheet, which indicates that all steel tubes are pressurized up to $3000psi$ during the laying operation.

Assessing the stress field in the steel tubes during the simulation, an admissible shoe dis-

Figure 5.11: Ramberg-Osgood fitting used to model the super-duplex steel rheology.

Source: Material parameters from the Offshore Mechanics Laboratory (LMO) database.

placement is estimated monitoring the plastic strain and associating it to steel tube damage. Figure 5.12 illustrates the shoe's reactive loads as function of its displacement (total intensity, not aligned with the applied displacements). Since it is a two-dimensional model, the result is a load per unit length.

The linearity of the results at the beginning of the loading is due to the material's constitutive properties and to minor displacements in the cross section. As the shoes' displacement increases, leading to higher loads and stresses, geometrical nonlinearities and plastic behavior become important, inducing a nonlinear response.

The red dashed line in Figure 5.12 indicates the moment when plastic strain first appears in the steel tubes. It is important to highlight that the onset of plastic strains does not necessarily mean that the steel tube is already compromised. Since this result corresponds to the most critical situation (the steel tube is aligned with the center of an electrical power cable and the center of the umbilical cross section), it brings a lower bound estimate for the critical crushing load. The admissible load values for the four shoes are presented in Table 5.1.

Apparently, it seems that the reactive forces are unbalanced. However, the four shoes are not planar and the 160° V-shape is enough to hold the umbilical and generate resultant forces in directions slightly misaligned with displacements.

Shoe displacements are imposed up to 3.4mm in this first case, leading to the von Mises stress field for the most critical steel tube depicted in Figure 5.13 and to the one for the entire cross section shown in Figure 5.14. It is worth mentioning that the obtained results are valid for this given cross section configuration, which presents a steel tube aligned with an electrical cable and one caterpillar shoe (Figure 5.10).

Figure 5.12: Crushing load calculated for the critical section of Cable 1. Axial tension: $540kN$. Internal pressure in tubular components: $3000psi$.

Table 5.1 : Admissible loads obtained for the numerical crushing of Cable 1, considering first yielding in the steel tubes, occurred at $0.8mm$ shoes displacement.

Shoe	Load magnitude [tf/m]
Upper	12.8
Lower	11.3
Left	8.0
Right	8.1

Figure 5.13: von Mises stress field along the steel tube section during crushing. V-shoes displacement: $3.4mm$. Axial tension: $540kN$. Internal pressure in tubular components: $3000psi$.

Figure 5.14: von Mises stress field over Cable 1 cross section. V-shoes displacement: $3.4mm$. Axial tension: $540kN$. Internal pressure in tubular components: $3000psi$.

5.4.2 Cable 2

A similar loading condition was simulated for Cable 2. This second umbilical has three HCR hoses, modeled using the equivalent thickness described previously in Section 5.3.9 and Appendix D. It also presents four armor layers that distribute the crushing load from the caterpillar shoes on the inner components. As in the case study with Cable 1, the critical section was considered with a HCR hose aligned with the power core and the upper caterpillar shoe. Figure 5.15 illustrates the second cross section configuration.

Figure 5.15: FE in-plane model of Cable 2 cross section.

After considering the same procedures adopted for the first analysis, displacements of the shoes are imposed, in order to reproduce the laying operation conditions. The applied axial tension ($742.8kN$) corresponds to half of the maximum working tension allowed for this umbilical cable, as indicated on its datasheet. All hoses are pressurized with $1015psi$, according to API Specification 16E. Figure 5.16 illustrates the shoe's reactive load.

The red dashed line indicates the onset of plastic deformation, this time in the equivalent tubes that represent HCR hoses carcasses. Again, it is worth mentioning that this result is only valid for the critical crushing configuration seen in Figure 5.15, and provides a lower bound estimation. The real stress field on the carcass should definitely be accessed further, through

a dedicated full three-dimension FE model. Furthermore, even under plastic deformation, the hose may continue to work properly, as it is highly conservative to associate the equivalent tube's first yielding with the admissible crushing load. The admissible load values for the four shoes are presented in Table 5.2.

Figure 5.16: Crushing load calculated for Cable 2 critical section; red dashed line indicates the first yielding of the equivalent tube that represents the reinforced hose. Axial tension: $742.8kN$. Internal pressure in tubular components: $1015psi$.

Table 5.2 : Admissible loads obtained for the numerical crushing of Cable 2, considering first yielding in the equivalent rings, occurred at 1.8mm shoes displacement.

Shoe	Load magnitude [tf/m]
Upper	28.4
Lower	25.1
Left	28.2
Right	28.2

The shoes' displacements were simulated up to $4.9mm$, as can be seen in the graphic above, leading to the von Mises stress field presented in Figure 5.17. This field is detailed in Figure 5.18 for the 0.5-inch reinforced hose equivalent model and in Figure 5.19 for the 3/8-inch thermoplastic hose.

Another point to be further discussed is the deformation of the thermoplastic hoses. As seen in Figure 5.19, the strains are not negligible. The important distortions revealed could compromise the cable's operation and the thermoplastic hoses could be the cable's weakness, instead of the reinforced hoses.

Figure 5.17: von Mises stress field over Cable 2 cross section. V-shoes displacement: $4.9mm$. Axial tension: $742.8kN$. Internal pressure in tubular components: $1015psi$.

Figure 5.18: von Mises stress field on the equivalent tube section that represents the HCR carcass. V-shoes displacement: 4.9mm . Axial tension: 742.8kN . Internal pressure in tubular components: 1015psi .

Figure 5.19: von Mises stress field on the thermoplastic hoses sections during crushing. V-shoes displacement: 4.9mm . Axial tension: 742.8kN . Internal pressure in tubular components: 1015psi .

5.4.3 Cable 3

Concluding these crushing numerical case studies, Cable 3 also presents three HCR hoses, this time with power and signal cables in the inner core. The critical crushing configuration considered consists of, regarding the orientation of the cross section relative to the shoes, the HCR hoses aligned with power cables and the upper HCR hose aligned with the upper shoe, as shown in Figure 5.20.

Figure 5.20: FE in-plane model of Cable 3 cross section.

Following the same methodology, Figure 5.21 presents the laying operation loads as a function of the displacements of caterpillar shoes.

The admissible loads considering first yield in the HCR hoses are listed in Table 5.3. These values were reached for a magnitude of displacements equal to $1mm$.

Simulating displacements up to $5.3mm$, the resultant von Mises stress field is presented in Figure 5.22. This resultant field is detailed in Figure 5.23 for the 0.5-inch reinforced hose equivalent model and in Figure 5.24 for the 3/8-inch thermoplastic hose.

Figure 5.21: Crushing load calculated for Cable 3 critical section; red dashed line indicates the first yielding of the equivalent tube that represents the reinforced hose. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$.

Table 5.3 : Admissible loads obtained for the numerical crushing of Cable 3, considering first yielding in the equivalent rings, occurred at $1.0mm$ shoes displacement. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$.

Shoe	Load magnitude [tf/m]
Upper	12.1
Lower	9.8
Left	14.2
Right	14.2

Figure 5.22: von Mises stress field over Cable 3 cross section. V-shoes displacement: $5.3mm$. Axial tension: $938kN$. Internal pressure in tubular components: $1015psi$.

Figure 5.23: von Mises stress field on the equivalent tube section that represents the HCR carcass. V-shoes displacement: 5.3mm . Axial tension: 938kN . Internal pressure in tubular components: 1015psi .

Figure 5.24: von Mises stress field on the thermoplastic hoses sections during crushing. V-shoes displacement: 5.3mm . Axial tension: 938kN . Internal pressure in tubular components: 1015psi .

5.5 FINAL REMARKS

A two-dimensional modeling methodology to analyze an umbilical cross section under crushing was successfully established. The in-plane modeling technique represents radial loads from the caterpillar shoes, as well as effects associated to the helical geometry of some internal components and armor wires. These inherent three-dimensional effects are included in the models by using analytical equations. The methodology was tested with three case studies.

The simple failure criterion adopted for the study can be reconsidered. The appearance of plasticity in metallic components may be rather conservative. Steel tubes and HCR hoses undergoing major deformations do not necessarily lose functionality or have their structural capacity compromised. The large deformations obtained for thermoplastic hoses also need to be better evaluated.

Concerning the validity of the 2D models, a study by Guttner, Santos and Pesce (2017) evaluated some results of 2D models of Cable 1's cross section. The study used a fully three-dimensional model as paradigm. Comparisons indicated that the two-dimensional approach is a good solution, given the quality of results and processing times. According to Guttner, Santos and Pesce (2017), the processing times of the 2D models were about 500 times faster than those of the 3D simulations.

Figure 5.25 illustrates a comparison of the stress fields obtained with both approaches. Below, Figure 5.26 illustrates the crushing force-displacement curves from 2D and 3D approaches. The four slender curves indicate results from the two-dimensional approach, considering different orientations of Cable 1's cross section. Results from the 3D models are noticeably quite consistent.

In the next chapter, this in-plane approach is used to reproduce some experimental crushing tests. Results from 2D models are compared to experimental measurements, to further explore the numerical approach herein presented.

Figure 5.25: Comparison between 2D and 3D crushing modeling approaches presented by Guttner, Santos and Pesce (2017).

Source: Guttner, Santos and Pesce (2017).

Figure 5.26: Crushing forces obtained from the 2D and 3D modeling approaches for Cable 1.

Source: Guttner, Santos and Pesce (2017).

6 CRUSHING EXPERIMENTAL ASSESSMENT USING DIC

This second part of the crushing analysis describes the development of a new experimental methodology. A pioneer and innovative assessment is presented, using a high-resolution Digital Image Correlation (DIC) system to track displacements of internal components in an umbilical cable cross section subjected to crushing in a $250kN$ mechanical test rig.

This experimental methodology is used to assess the same umbilical cables examined in the previous chapter. An experimental setup with two flat shoes (upper and lower) has been adopted in order to simplify the study. In a preliminary assessment with conventional instrumentation to measure the crushing force applied and the displacement of the bottom shoe, the radial stiffnesses of the cross sections are evaluated and compared with results from the two-dimensional approach previously proposed. Next, the experimental study continues with the analysis of the components in the umbilical cross sections. A DIC system is employed to monitor the integrity of steel tubes, HCR and thermoplastic hoses. Unlike the conventional use of a DIC system (usually employed to monitor strain fields), the goal here is to track the circumferential profiles of deformed hoses. This analysis is concluded using the measured optical data to feed a FE model of a single component, in order to evaluate stresses in the tubular components.

6.1 EXPERIMENTAL ARRANGEMENT AND INSTRUMENTATION

The crushing experimental campaign was carried out with samples from Cables 1, 2 and 3. Figure 6.1 illustrates the three available Cable 1 samples, while Figure 6.2 and Figure 6.3 show the six available samples from Cables 2 and 3, respectively. All samples were prepared to be $300mm$ long. This length was chosen by taking into account the maximum loading capacity of the hydraulic test rig and the size of the flat metallic shoes.

As can be seen in the images above, samples of a same cable are not identical. In the case of an umbilical with helically wound components arranged with different laying angles, it is improbable to obtain replicated samples. This fact was taken into account when analyzing results.

To track displacements with the DIC system during the experimental crushing tests, one extremity of each sample was faced, polished and painted with a speckled pattern. Figure 6.4 illustrates one specimen of each cable painted and prepared. Such pattern was meticulously handmade with small black specks covering the previously white painted surface. In addition to this treatment of the sample surface, the shoes' front surfaces were also painted and prepared in order to assess the rigidity of the supports.

Figure 6.1: Overview of the three specimens from Cable 1 used in the experimental crushing tests.

Figure 6.2: Overview of the six specimens from Cable 2 used in the experimental crushing tests.

Figure 6.3: Overview of the six specimens from Cable 3 used in the experimental crushing tests.

Figure 6.4: Speckled pattern applied on samples of all three umbilicals and the planar shoes; the metal block at left is used as a reference for null strain measurement.

Figure 6.4 also shows the flat shoes used to apply the experimental crushing load. The flat configuration was chosen to simplify contacts, even though it does not represent the actual case of a crushing load during a laying operation. This simplification facilitates experimental setup and reduces friction effects considerably. The DIC results confirmed that the metallic shoes can be considered rigid when in contact with the cables. This fact certifies that the 2D models previously developed can adopt rigid surfaces to represent the shoes.

Figure 6.4 also shows a small rectangular metallic block next to each sample. This piece was used as reference of null strain state and to verify the quality of the measured DIC data. Moreover, it allows the researcher to monitor the bottom shoe displacement and synchronize optical and conventional measurements.

Images in Figure 6.5 illustrate the experimental assembly with all of the equipment used. The images illustrate the LMO's universal MTS 250kN servo-hydraulic rig used to apply the crushing force. The umbilical cable sample is set on the rig, supported by both 300mm long flat shoes. The crushing load is applied by the lower shoe moving upwards; the upper shoe is fixed. The 8-megapixels monochromatic digital camera is shown at the front of the pictures, with one LED lamp on each side. The optical images acquired are transferred to the laptop, where the DIC system is installed. The images are registered and processed with the software *VIC-2D v6.0.0*, by *Correlated Solutions, Inc.* The desktop PC is used to control the rig.

Figure 6.5: Experimental crushing tests assembly.

Figure 6.6 shows a side view of one of the samples set on the shoes. As the lengths of the

sample and the shoes are noticeably similar, there is contact throughout the sample's entire length. However, after the samples are withdrawn from the umbilical, the diameter of their extremities increases slightly. This small variation in diameter can be noticed in Figure 6.6. The upper contact between the flat metallic surface and the 300mm long sample happens only at both extremities. Consequently, a preload was required to establish uniform crushing contact along the whole sample.

Figure 6.6: Side view of a specimen set on the shoes. Notice the irregular contact between the upper shoe and the umbilical sample.

6.1.1 Digital Image Correlation (DIC) processing

The Digital Image Correlation (DIC) system previously mentioned was used during the tests to track the profiles of tubular components. Sutton, Orteu and Schreier (2009) is a very good reference to study this optical system and the algorithms used for tracking and calculating displacements. Basically, the system uses subsets to track the movement of speckle patterns, as illustrated in Figure 6.7. Each subset, represented by the squares in Figure 6.7, is monitored.

After a sequence of images is acquired during the experimental test, the DIC system correlates the subsets from each image at subsequent time steps (t and $t + dt$), estimating the displacement vector field from one image to the other. Once the displacement vector field has been determined, local numerical derivatives provide the corresponding strain field, if desired.

Figure 6.8 presents a zoom in the steel block used as reference for displacement and null strain field, illustrating the subset size used during this study. For all analyses herein presented, the subset size was set with 31-pixels side length and 6-pixels pitch. This means that from the center of one particular subset to its neighbors there is a distance of 6 pixels. With a 31-pixels side the subsets overlap, differently from what is shown in Figure 6.8.

Figure 6.7: Digital Image Correlation: from speckled subsets, to displacement fields, to strain fields.

Source: LaVision (2017).

Figure 6.8: Selected zone to be analyzed (red) and respective subset size (yellow).

6.2 LOAD CASES

The experimental campaign consisted of several tests using all samples from the three umbilical cables. The position of the cross sections was changed for some tests while the load speed was fixed at around $10kN/min$. Figure 6.12 provides a clear view of the different positions tested for Cable 1, while Figure 6.13 and Figure 6.14 present these positions for Cables 2 and 3, respectively. Position A means that the radius A is aligned with the crushing load direction and so on, until Position F. This load speed was chosen based on a previous study made with one of the specimens. The study was presented in Santos et al. (2016).

Three different loading time-histories were developed for the crushing tests, so that each specimen could be used more than once. The types of loading histories developed are specified below:

- The 'low-loading' case was planned to not affect the integrity of internal components in the umbilical cross section. This load case consists of 5 cycles of crushing load and unload, with a small crushing force up to $50N/mm$ that surely does not affect the integrity of internal components (as per the numerical results previously obtained). As the internal components remain intact with this kind of test, it is repeated with the same specimen for all 6 positions, from A to F. The time-history diagram of the crushing load of a 'low-loading' test is presented in Figure 6.9.
- The 'high-loading-directly' case consists of only 1 cycle of crushing load and unload, with a high crushing force up to $670N/mm$ that certainly damages some internal components. The time-history diagram of the crushing load of a 'high-loading-directly' test is presented in Figure 6.10. This case has considered solely cross sections in Position A.
- The 'high-loading-cyclic' case consists of 4 cycles of crushing load and unload, with the crushing force amplitude growing between cycles up to $670N/mm$. The time-history diagram of the crushing load of a 'high-loading-cyclic' test is presented in Figure 6.11. Position A was always considered.

Three tables on the next pages specify, for each cable, all tests that were carried out. Each table indicates the respective sample, its cross section position and the adopted load case. The C1, C2 and C3 indexes help identify the specimens of Cables 1, 2 and 3, respectively. Likewise, indexes T1, T2 and T3 identify the tests carried out with Cables 1, 2 and 3, respectively.

Figure 6.9: Time-history diagram of a 'low-loading' test.

Figure 6.10: Time-history diagram of a 'high-loading-directly' test.

Figure 6.11: Time-history diagram of a 'high-loading-cyclic' test.

Figure 6.12: Crushing load alignment positions of Cable 1.

Table 6.1 : Load cases parameters for Cable 1.

Test ID	Sample	Position	Load case
Test T1-01	Specimen C1-1	A	High-loading-cyclic
Test T1-02		A	
Test T1-03		B	
Test T1-04		C	Low-loading
Test T1-05		D	
Test T1-06		E	
Test T1-07		F	
Test T1-08	Specimen C1-2	A	High-loading-cyclic
Test T1-09		A	
Test T1-10		B	
Test T1-11		C	Low-loading
Test T1-12		D	
Test T1-13		E	
Test T1-14		F	
Test T1-15	Specimen C1-3	A	High-loading-directly

Figure 6.13: Crushing load alignment positions of Cable 2.

Table 6.2 : Load cases parameters for Cable 2.

Test ID	Sample	Position	Load case
Test T2-01	Specimen C2-1	A	Low-loading
Test T2-02		B	
Test T2-03		C	
Test T2-04		D	
Test T2-05		E	
Test T2-06		F	
Test T2-07	Specimen C2-2	A	High-loading-directly
Test T2-08		A	High-loading-cyclic
Test T2-09	Specimen C2-3	A	Low-loading
Test T2-10		B	
Test T2-11		C	
Test T2-12		D	
Test T2-13		E	
Test T2-14		F	
Test T2-15	Specimen C2-4	A	High-loading-cyclic
Test T2-16		A	High-loading-directly
Test T2-17	Specimen C2-5	A	High-loading-directly
Test T2-18	Specimen C2-6	A	High-loading-cyclic

Figure 6.14: Crushing load alignment positions of Cable 3.

Table 6.3 : Load cases parameters for Cable 3.

Test ID	Sample	Position	Load case
Test T3-01	Specimen C3-1	A	High-loading-directly
Test T3-02	Specimen C3-2	A	High-loading-cyclic
Test T3-03	Specimen C3-3	A	High-loading-directly
Test T3-04	Specimen C3-4	A	Low-loading
Test T3-05		B	
Test T3-06		C	
Test T3-07		D	
Test T3-08		E	
Test T3-09		F	
Test T3-10	Specimen C3-5	A	High-loading-directly
Test T3-11		A	Low-loading
Test T3-12		B	
Test T3-13		C	
Test T3-14		D	
Test T3-15		E	
Test T3-16		F	
Test T3-17	Specimen C3-7	A	High-loading-cyclic
Test T3-18		A	High-loading-cyclic

6.3 EXPERIMENTAL RESULTS

This section presents the crushing force vs. shoe displacement diagrams for all 51 tests outlined in the tables in the previous section. Next, the methodology to achieve the radial stiffness of the three umbilical cables is presented, along with the stiffness values obtained. These calculations are compared with the numerical results from the 2D FE approach presented in the previous chapter.

The 'low-loading' test case was applied to two specimens of each umbilical: for Cable 1, C1-2 (Figure 6.15) and C1-3 (Figure 6.16) on the facing page; for Cable 2, C2-1 (Figure 6.17), C2-3 (Figure 6.18) on page 178; and for Cable 3, C3-4 (Figure 6.19) and C3-5 (Figure 6.20) on page 179. As described before, each specimen underwent 6 non-destructive tests, rotating the sample from position A to position F.

Despite the 'low-loading' rate of $0.5\text{N}/\text{mm}/\text{s}$ (adopted based on Santos et al. (2016)), the cycles of load and unload originated a hysteresis loop. It is remarkable that the loops from Cable 1 are much smaller than the ones found for Cables 2 and 3. However, these loops indicate the presence of a dissipation mechanism. As the load increases, the umbilical components certainly begin to move and suffer a slight change in position. During the unloading phase, the relative positions of the structural elements are not restored, giving rise to the hysteresis cycle. The viscoelastic behavior of some plastic and elastomeric components may contribute to the hysteresis as well, but it is important to remember that at these small loads plasticity effects are not mobilized, at least not significantly.

Starting always at the same point and controlling the load to the same maximum value and at the same speed, the subsequent hysteresis loops are formed between different levels of shoe displacements. Nevertheless, the cycles remain quasi-parallel to each other, revealing a sort of independence of the radial stiffness with respect to the cable position. Moreover, changes in the hysteresis loop shapes are imperceptible from one position to another. The shift of loops to the right seems to be solely due to the first (accommodating) loading cycle.

It is worth noting that results for position B and D of Cable 1 presented different displacement levels. Besides the cross section symmetries that can be seen in Figure 6.12 on page 173, these positions cannot be considered equivalents. The frontal cross sections of the specimens of all cables present symmetries, but positions apparently similar or symmetric present differences due to the components helically placed along the samples.

The other tests presented in the load case tables considered higher crushing forces, reaching a maximum load per unit length of $670\text{N}/\text{mm}$. The 'high-loading' test results are illustrated for Cable 1 (Figure 6.21), Cable 2 (Figure 6.22) and Cable 3 (Figure 6.23) on page 181. Some remarkable aspects are the formation of the hysteresis loops and the similarity of results for

Figure 6.15: Crushing force vs. bottom shoe displacement diagrams for specimen C1-2. Results from Tests T1-02 to T1-07.

Figure 6.16: Crushing force vs. bottom shoe displacement diagrams for specimen C1-3. Results from Tests T1-09 to T1-14.

Figure 6.17: Crushing force vs. bottom shoe displacement diagrams for specimen C2-1. Results from Tests T2-01 to T2-06.

Figure 6.18: Crushing force vs. bottom shoe displacement diagrams for specimen C2-3. Results from Tests T2-09 to T2-14.

Figure 6.19: Crushing force vs. bottom shoe displacement diagrams for specimen C3-4. Results from Tests T3-04 to T3-09.

Figure 6.20: Crushing force vs. bottom shoe displacement diagrams for specimen C3-5. Results from Tests T3-11 to T3-16.

different specimens of a same umbilical, whether the load is applied cyclically or directly.

6.3.1 Radial stiffness

Figure 6.24 illustrates in the same diagrams all hysteresis loops for the six ‘low-loading’ tests with specimens C1-2 and C1-3. The first offset in displacement seems to be heavily dependent on sample position. However, after internal components have accommodated inside the specimen, hysteresis loops are very similar for all orientations.

This repeatability means that the radial stiffness can be calculated based on the slope of the hysteresis loop. For all cycles, after a preload (around $10N/mm$) that equalizes the contact between shoes and sample has been reached, the inclination of the force-displacement curve is nearly constant with the increase of the crushing load. This indicates that the radial stiffnesses of the cables are constant as the load increases within this given range.

The umbilical radial stiffness can be evaluated by looking at this inclination of each cycle. The first load path, when the sample components have not yet re-accommodated, stands apart from the rest. Therefore, the radial stiffness calculations neglected this first cycle. The experimental values for instantaneous radial stiffness were then considered as an average taking into account cycles 2 to 5.

Figure 6.25 presents the final estimation of the radial stiffness for each position tested. The stiffnesses are almost constant for positions A through F after the preload mentioned above has been reached. To compute these stiffness values, the slope of a linear fit was calculated for several small intervals in the force-displacement diagrams. For this reason, the results in Figure 6.25 do not start at a null force level neither finish at the maximum force value. The crushing load was normalized by the maximum force $F_{max} = 33N/mm$ reached during the tests with Cable 1. Figure 6.25 illustrates how stiffnesses change with loading.

Figure 6.25 also shows the range where the results obtained with a 2D FE model were placed. Using the crushing modeling methodology presented in the previous chapter, 2D FE models were designed to reproduce the experimental tests conditions. The 2D models previously presented have been adjusted to simulate the experimental conditions as faithfully as possible: shoe geometries were changed to flat and the squeezing loads were set to null, as no axial tension is applied during the experiments.

As seen in the numerical case studies presented in the previous chapter, simulation results depend on the cross section orientation and on how the internal components are placed. In the numerical tests, the load-displacement curve changes significantly with the position of the umbilical components relatively to the load direction. Therefore, all possible cross sections were simulated in order to estimate upper and lower bounds for the crushing load. The FE

Figure 6.21: Crushing force vs. bottom shoe displacement diagrams for all 'high-loading' tests with Cable 1. Results from Tests T1-01, T1-08 and T1-15.

Figure 6.22: Crushing force vs. bottom shoe displacement diagrams for all 'high-loading' tests with Cable 2. Results from Tests T2-07, T2-08 and T2-15 to T2-18.

Figure 6.23: Crushing force vs. bottom shoe displacement diagrams for all 'high-loading' tests with Cable 3. Results from Tests T3-01 to T3-03, T3-10, T1-17 and T1-18.

Figure 6.24: Hysteresis loops for all 'low-loading' tests with specimens C1-2 and C1-3. Results from Tests T1-02 to T1-07 and T1-09 to T1-14.

Figure 6.25: Radial stiffness of specimens C1-2 and C1-3, for each tested position (Tests T1-02 to T1-07 and T1-09 to T1-14).

radial stiffnesses calculated from the lower and upper bounds of the numerical results are shown in Figure 6.25. For a coherent correlation, a linear numerical analysis was used to calculate these bounds from the numerical model. The nonlinear effects (e.g. from geometry changes, contact interactions or material properties) in the 'low-loading' test are hardly significant and were therefore disregarded in these FE calculations. For this reason, the FEA presented a linear result and independence from the load, with constant bounds in the radial stiffness diagrams. It is worth mentioning that for the evaluation of the numerical radial stiffness, the in-plane linear numerical analysis considered bonded contacts between components. These bonded contacts certainly disregard possible internal accommodations of umbilical components.

The results in Figure 6.25 clearly show that experimental radial stiffness does not change considerably with position. The differences are larger for loads smaller than 30% of F_{max} , before the preload necessary to equalize contacts. Once this preload has been overcome, the

samples presented stiffness values inside the boundaries provided by the 2D FE simulations, indicating a coherence between the numerical and experimental approaches.

Concerning the ‘high-loading’ tests for Cable 1, Figure 6.26 shows the experimental hysteresis loops obtained. Either directly with specimen C1-3 or in cycles with specimens C1-1 and C1-2, the final shapes are very similar. The diagram also presents the numerical force-displacement curve obtained by the in-plane FE approach. For this ‘high-loading’ case, the 2D FEA was set up to consider nonlinearities, reproducing the crushing load and unload with a maximum load per unit length of 670 N/mm . Figure 6.27 illustrates the cross sections of both experimental and numerical approaches at this maximum load.

Figure 6.26: Hysteresis loops for ‘high-loading’ tests with all three specimens from Cable 1. Results from Tests T1-01, T1-08 and T1-15.

Figure 6.27: Cross sections of Cable 1 at maximum crushing load; at left, experimental test; at right, 2D FE model.

The numerical hysteresis loop in Figure 6.26 is comparable with the experimental results. Since the FE approach does not take into account viscoelasticity effects from the polymeric

components and layers, this numerical loop is certainly due to the accommodation of internal components in the cross section. Figure 6.27 emphasizes the similarity of cross section configurations at the maximum crushing load, even if the deformation and relative position of internal components in the numerical and experimental approaches were not identical.

Analyzing the force-displacement curve obtained with the in-plane FE model, a lower radial stiffness can be noticed at the beginning of the crushing process. Because the model is two-dimensional, in its initial configuration the components may dislocate more freely than happens in reality. In the experimental tests, the helically wound geometries outside the monitored plane inhibit the movement of the components and, for this reason, the radial stiffness presents substantial values in the beginning of the loading process. In the FE model, as the components no longer have to be rearranged, the radial stiffness increases, so that at the maximum load compatible displacements take place in both approaches.

The same approach to calculate the radial stiffness was developed for the other two umbilical cables. For Cable 2, Figure 6.28 presents the hysteresis loops obtained for the ‘low-loading’ tests with specimens C2-1 and C2-3. From these loops, Figure 6.29 estimates the radial stiffnesses, following the procedures previously described. It shows how axial stiffness decreases with the crushing load for positions A to F. The crushing load is again normalized by the maximum force $F_{max} = 50\text{N/mm}$ reached during the tests shown in Figure 6.28.

Figure 6.28: Hysteresis loops for all ‘low-loading’ tests with specimens C2-1 and C2-3. Results from Tests T2-01 to T2-06 and T2-09 to T2-14.

Similar to what happened with Cable 1, only low crushing loads generate pronounced differences for the radial stiffness of Cable 2. The four tensile armor layers distribute the loading through the internal components and therefore the stiffness values are not dependent on specimen position. The helical wire pitches of this 4-armor-layer cable are between 874mm (inner layer) and 1253mm (outer layer). The adopted sample length (300mm) represents nearly 30% of the average wire pitch. It seems that this fraction is enough to provide an homoge-

Figure 6.29: Radial stiffness of specimens C2-1 and C2-3, for each tested position (Tests T2-01 to T2-06 and T2-09 to T2-14).

neous and single behavior for the radial stiffness of the cable, irrespective of load orientation. Nevertheless, the linear FE approach clearly overestimates the radial stiffness for Cable 2. The disregarding of nonlinear effects due to geometry changes does not reproduce the stiffness reduction observed in the experimental tests.

Concerning results with Cable 2 from the 'high-loading' tests, Figure 6.30 presents experimental and numerical hysteresis loops. It is noticeable that the numerical in-plane model is stiffer, even when considering geometrical nonlinearities. At the maximum crushing load, the shoe displacement reached 30mm in the 2D FE model, while the experimental displacements were around 25 higher.

Figure 6.31 illustrates the cross section configurations at this maximum crushing load, suggesting the causes of the higher radial stiffness in the numerical approach. Differently from Cable 1, the cross section of Cable 2 does not present a configuration with elements prone to re-accommodate themselves. Therefore, the numerical model presented a considerable stiffness since the loading began. Another contrast between the results from experimental and numerical approaches was the deformation of the electrical core. This core is clearly stiffer in the 2D FE model, resulting in a higher radial stiffness of the assembly. More differences between the experimental and numerical results are pointed out in the next section, when displacement fields are analyzed.

Closing this radial stiffness assessment, results from the 'low-loading' crushing tests with Cable 3 are presented in Figure 6.32 and Figure 6.33. Once again, there are no considerable changes in the radial stiffness of this last cable as a result of the cross section position, at least for the chosen specimens (C3-4 and C3-5). The differences are noticeable for loads smaller than 30% of F_{max} and, again, the bounds obtained by the linear FE calculations overestimate the stiffness.

Figure 6.30: Hysteresis loops for 'high-loading' tests with all three specimens from Cable 2. Results from Tests T2-07, T2-08 and T2-15 to T2-18.

Figure 6.31: Cross sections of Cable 2 at maximum crushing load; at left, experimental test; at right, 2D FE model.

For higher crushing loads, the 'high-loading' tests with Cable 3 are shown in Figure 6.34. The curve inclinations in the force-displacement diagram indicate that Cable 3's radial stiffness begins to increase as larger loads are applied. For this last case, the load-unload curve obtained with the 2D FE model is comparable to the experimental results. With a core similar to Cable 1, the radial stiffness of Cable 3 presented an analogous behavior. At the beginning of loading, the accommodation of internal components allows higher displacements without an increase of the crushing force. After a shoe displacement of around 22mm , the internal components aligned with the load are responsible for the increase of the radial stiffness. Figure 6.35 presents both experimental and numerical cross sections at the maximum crushing load, illustrating this alignment of internal components in the FE model.

Figure 6.32: Hysteresis loops for all 'low-loading' tests with specimens C3-4 and C3-5. Results from Tests T3-04 to T3-09 and T3-11 to T3-16.

Figure 6.33: Radial stiffness of specimens C3-4 and C3-5, for each tested position (Tests T3-04 to T3-09 and T3-11 to T3-16).

Figure 6.34: Hysteresis loops for 'high-loading' tests with all three specimens from Cable 3. Results from Tests T3-01 to T3-03, T3-10, T3-17 and T3-18.