2019

MODELING, SIMULATION AND INVESTIGATION OF INFLATABLE DROP-STITCH PANELS WITH FINITE ELEMENT ANALYSIS

Alena M. Alich
University of Rhode Island, alena.alich@gmx.de

Follow this and additional works at: <https://digitalcommons.uri.edu/theses>

Recommended Citation

Alich, Alena M., "MODELING, SIMULATION AND INVESTIGATION OF INFLATABLE DROP-STITCH PANELS WITH FINITE ELEMENT ANALYSIS" (2019). *Open Access Master's Theses*. Paper 1504.
<https://digitalcommons.uri.edu/theses/1504>

This Thesis is brought to you for free and open access by DigitalCommons@URI. It has been accepted for inclusion in Open Access Master's Theses by an authorized administrator of DigitalCommons@URI. For more information, please contact digitalcommons-group@uri.edu.

MODELING, SIMULATION AND INVESTIGATION OF INFLATABLE
DROP-STITCH PANELS WITH FINITE ELEMENT ANALYSIS

BY

ALENA M. ALICH

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE OF
MASTER OF SCIENCE
IN
MECHANICAL ENGINEERING

UNIVERSITY OF RHODE ISLAND

2019

MASTER OF SCIENCE THESIS
OF
ALENA M. ALICH

APPROVED:

Thesis Committee:

Major Professor David G. Taggart

Hongyan Yuan

George Tsiatas

Nasser H. Zawia

DEAN OF THE GRADUATE SCHOOL

UNIVERSITY OF RHODE ISLAND

2019

ABSTRACT

Due to the rising demand in lightweight and rapid deployable structures, inflatable drop-stitch structures are gaining importance. Despite the increasing interest in these structures, no reliable models to predict the behavior of the panels are available and the effects that lead to a pressure dependent stiffness are not fully understood.

In this research, classical lamination theory is applied to obtain homogenized engineering constants for the composite skin for use in predicting the panel response. In the next step, a finite element model of an inflatable drop-stitch panel under a four-point bending load is developed and the simulation results are validated by the comparison to experimental data. In addition, the simulation is used to validate the analytical model derived in a recent study (Smith, Michael 2019). An analytical model for the stress distribution within the panel is presented and compared to the numerical simulation results.

It is determined that the application of the classical lamination theory to predict skin properties is very sensitive to the orthotropic properties of the constituent layers. Obtaining good correlation with experimental data requires some minor adjustment of layer properties. The finite element simulation of a four-point bending test of an inflated drop-stitch panel showed that although the model does not correlate exactly with the experimental results, it does provide a reasonably good estimate of the panel response. The numerical model also provides a tool to evaluate the effects of various deformation mechanisms on the overall response. It is determined that the pressure dependent stiffness is associated with nonlinearities of the model.

ACKNOWLEDGMENTS

Throughout this year, too many people to name have supported me in a variety of ways in my research and my studies at the University of Rhode Island. I am very grateful to each one of them. Nevertheless, there are some persons I would like to formally acknowledge.

I would like to express my sincere appreciation to my major professor and advisor, Dr. David Taggart, whose guidance and expertise in the field of computational mechanics supported my learning process significantly throughout my research. I am really thankful that Dr. Taggart was always available to answer all my questions and provided helpful advice when I encountered problems.

I also want to thank Michael Smith for all the support that he gave me, from explaining his research, offering organizational and formatting tips, to making readily available all experimental results. I always enjoyed our discussions concerning our theses, which provided new ideas that I could apply to my research.

In addition, I want to thank Milo Ferrazzoli for his contribution to this research and the Department of Civil and Environmental Engineering as well as Dr. George Tsiatas for the use of the structures laboratory for the four-point bending tests used in this research.

Lastly, I would like to thank my friends for always cheering me up during difficult times, my parents for always believing in me and helping me achieve everything that I set my mind on, and my sister for understanding me and always being there when I need her. I wouldn't have made it this far without you.

TABLE OF CONTENTS

ABSTRACT	ii
ACKNOWLEDGMENTS	iii
TABLE OF CONTENTS	iv
LIST OF FIGURES	vii
LIST OF TABLES	xi
CHAPTER	
1 Introduction	1
1.1 Motivation	1
1.2 Goals of the Research	2
1.3 Methodology	2
2 Literature review	4
2.1 Inflatable Structures	4
2.2 Inflatable Drop-Stitch Panels	5
2.3 Computational Analysis of Inflatable Structures	7
3 Theoretical Foundation	9
3.1 Four-point bending Test	9
3.2 Pressure Vessels	10
3.3 Elastic Material Properties	12
3.4 Classical Lamination Theory	16
3.5 Finite Element Analysis	19

	Page
3.5.1 Foundations of Finite Element Analysis	19
3.5.2 Element Types	23
3.5.3 Contact Modeling	28
3.6 Inflatable Drop-Stitch Structures	30
3.6.1 Specific Characteristics of the Investigated Panel	33
4 Homogenization of the Skin Material with Classical Lamination Theory	37
4.1 Symmetrical Skin Material Laminate	37
4.2 Unsymmetrical Skin Material Laminate	40
4.3 Optimization of the Homogenization Result	42
4.4 Summary and Conclusion	45
5 Modeling of an Inflatable Drop-Stitch Panel with Finite Element Analysis	47
5.1 Modeling Process	47
5.2 Convergence Analysis	52
5.3 Pressure dependant Force vs. Displacement Behavior	56
5.3.1 Behavior of Whole Panel	56
5.3.2 Behavior of Drop-Stitch Yarns	60
5.3.3 Nonlinear vs. Linear Behavior	67
5.4 Pressure Dependent Beam Bending Deflection	69
5.5 Analytical Model for the Stress Distribution	74
5.5.1 Pressurization Step	75
5.5.2 Loading Step	83
5.6 Summary and Conclusion	92

	Page
6 Conclusions and Future Research	95
6.1 Conclusions	95
6.2 Future Research	97
LIST OF REFERENCES	98
APPENDIX	
A Matlab script for classical lamination theory	101
B Additional Analysis	103
B.1 Convergence S11	103
C MATLAB and Python scripts for the Evaluation of Results .	104
C.1 Postprocessing	104
C.1.1 Main Script - PostprocessingMatlab.m	104
C.1.2 Called Scripts	106
C.2 Stress Distribution	110
C.2.1 Main Script - plot_skin_stresses.m	110
C.2.2 Called Script - Stressreport.py	115
C.3 Cross-sectional Area from Simulation	118
C.3.1 Main Script- areacalc.m	118
C.3.2 Called Script - saveareafromodb.py	119
BIBLIOGRAPHY	122

LIST OF FIGURES

Figure		Page
1	Free body diagram of four-point bending test [1]	9
2	Free body diagram of a spherical pressure vessel [2]	11
3	Representation of the x_3 coordinate for different layers in a laminate [3]	18
4	Steps for Finite Element Analysis [4]	20
5	Degree of freedom of four node shell element[5]	26
6	Characteristic composition of the drop-stitch panel	31
7	Superposed stresses for a bending load [6]	32
8	Upper skin stress states for a bending load [6]	33
9	Inflatable drop-stitch panels	34
10	Dimensions of the panel	34
11	Definition of the warp and weft direction	34
12	Sideview of inflatable drop-stitch panel skin	35
13	Close-up (1×1 inch) of the different layers of the skin	35
14	Symmetric skin composition	37
15	Results for the Young's modulus of a symmetric skin composition obtained by the classical lamination theory	38
16	Stress vs. Strain curve for the whole skin [7]	40
17	Original skin composition	41
18	Results for the Young's modulus of the original skin composition obtained by the classical lamination theory	41
19	Results for the Young's modulus obtained by the classical lamination theory and an optimization process	43

Figure		Page
20	Stress vs. Strain curve for the Chafer layer [7]	44
21	Stress vs. Strain curve for the drop-stitch layer [7]	45
22	Change in thickness due to inflation [8]	49
23	Symmetry boundary conditions four-point bending test	50
24	Contact formulation between upper pin and panel	52
25	Loading and boundary condition for the whole assembly	52
26	Exemplary node for convergence analysis	53
27	Convergence for stress S_{22} with inflation pressure of 5 psi	54
28	Convergence for stress S_{22} with inflation pressure of 10 psi	54
29	Convergence for stress S_{22} with inflation pressure of 15 psi	55
30	Convergence for stress S_{22} with inflation pressure of 20 psi	56
31	Force displacement behavior for experiment and simulation (5psi)	57
32	Force displacement behavior for experiment and simulation (10psi)	58
33	Force displacement behavior for experiment and simulation (15psi)	58
34	Force displacement behavior for experiment and simulation (20psi)	59
35	Force displacement behavior for different inflation pressures	60
36	Nodes under loading pin used to investigate the loss of tension	61
37	Loss of tension in the drop-stitch yarns in the simulation	62
38	Stress 5 psi inflation pressure step time 1.2	63
39	Stress 5 psi inflation pressure step time 1.5	64
40	Stress 5 psi inflation pressure step time 1.7	65
41	Loss of tension points in force displacement behavior	66
42	Force-Displacement behavior including compression of drop-stitch yarns	66

Figure		Page
43	Linear model of inflated drop-stitch panel	68
44	Experiment and simulation bending for 5 psi inflation pressure .	69
45	Experiment and simulation bending for 10 psi inflation pressure	70
46	Experiment and simulation bending for 15 psi inflation pressure	70
47	Experiment and simulation bending for 20 psi inflation pressure	71
48	Explanation of the obtained kinking angle	72
49	Polynomial fits for inflation pressures 5 to 30 psi	73
50	Comparison of kinking angle in simulation and experiment . . .	74
51	Variables needed to characterize panel	75
52	Transverse stresses due to inflation pressure of 5 psi	77
53	Inflation Stresses 5 psi	81
54	Inflation Stresses 10 psi	82
55	Inflation Stresses 15 psi	82
56	Inflation Stresses 20 psi	83
57	Superposed bending stresses at the top of the panel	84
58	Superposed bending stresses at the bottom of the panel	85
59	Axial and transverse stresses at panel top skin for 5 psi	87
60	Axial and transverse stresses at panel bottom skin for 5 psi . . .	87
61	Axial and transverse stresses at panel top skin for 10 psi	88
62	Axial and transverse stresses at panel bottom skin for 10 psi . .	88
63	Axial and transverse stresses at panel top skin for 15 psi	89
64	Axial and transverse stresses at panel bottom skin for 15 psi . .	89
65	Axial and transverse stresses at panel top skin for 20 psi	90

Figure		Page
66	Axial and transverse stresses at panel bottom skin for 20 psi . .	90
B.67	Convergence for stress S_{11} for different inflation pressures . . .	103

LIST OF TABLES

Table		Page
1	Material characterization [7]	36
2	Values used in the final optimization iteration	43
3	Values for points of interest	63
4	Cross-sectional area and perimeter transverse direction	80
5	Cross-sectional area and perimeter axial direction	80

CHAPTER 1

Introduction

1.1 Motivation

In recent years, numerous applications requiring rapidly deployable and lightweight structures have been identified. Especially in marine and space applications, where added weight and space requirements lead to extreme or prohibitive increases in cost. Since inflatable structures provide the desired characteristics of lightweight, portability and rapid deployment, interest in inflatable structures is receiving increased attention.

Inflatable structures consist of a thin airtight skin [9]. When inflated, these structures develop high stiffnesses with a low weight. Prior to inflation, they are portable and use very little space [9]. Inflatable structures are versatile and are used in various applications like boats [10, 11], stand up paddling boards [12], shelters [13] and space applications [14]. Due to the inflation process, inflatable structures are pretensioned. The stiffness of these structures increases with the inflation pressure [15].

Drop-stitch structures, which are the focus of this research, are inflatable structures whose upper and lower skin are connected by drop yarns. These yarns ensure that the structure maintains the desired geometry and specific height when inflated [6]. The first reported application of drop-stitch structures was in the 1950s for the use in inflatable airplane wings [16]. Since then, and especially in the last ten years, these structures have been investigated and used in various applications. However, the material and construction parameters that control the mechanical response of inflatable drop-stitch panels are still not completely understood.

Determination of the effects of different materials and geometries on the mechanical behavior of drop-stitch panels is important to effectively design these

structures, optimize the structural properties for specific applications and ensure safety for users. A suitable approach to characterize the stiffness of inflatable drop-stitch panels is a four-point bending test [6]. However, construction of prototypes to test changes in skin and drop-stitch materials and construction parameters is expensive and time-consuming. To be able to predict the mechanical response and reduce the amount of physical testing, it is desired to develop accurate numerical and analytical models for predicting the behavior of new panels. Accurate predictive models for different geometries and materials will allow designers to explore a variety of constructions prior to building a prototype [15].

1.2 Goals of the Research

The primary goal of this research is to develop a finite element model of a four-point bending test on an inflatable drop-stitch panel to predict its mechanical response. The model has a potential to be modified to consider different skin and drop-stitch yarn properties and construction parameters. After developing the model, the goal is to validate assumptions and analytical theories that were derived in previous research [7]. Another goal is to understand the effects that lead to an inflation pressure dependent stiffness. It is also desired to determine the limits of the model in representing the actual behavior of an inflatable panel.

1.3 Methodology

This thesis begins with a review of previous research related to drop-stitch inflatable panels (Chapter 2). Then, in Chapter 3, various theoretical aspects related to this research are reviewed, including four-point bending testing, pressure vessel analysis, orthotropic elastic material behavior, classical lamination theory, finite element method and previously observed behavior of drop-stitch inflatable panels and their constituents.

The following Chapter 4 contains the evaluation of classical lamination theory to predict skin properties based on constituent material properties and layer construction. The theory is applied to obtain the homogenized characteristics of the skin, which can be used in the model of a four-point bending test. Results by the classical lamination theory are discussed and recommendations are given for its use in predicting the skin properties in analyzing inflatable panels.

The next phase of this research describes a computationally efficient finite element model of a four-point bending test of an inflatable drop-stitch panel, which is developed and investigated in Chapter 5. A mesh convergence study, examining the effect of element size, is performed to ensure accurate results. Using an appropriate mesh density, the force displacement behavior for simulation are compared to recent experimental results [7]. First, they are compared for the entire model to validate the simulation and to discuss the limits of the model in representing the actual behavior of the panel. The following steps evaluate the effects that could lead to the pressure dependent behavior. These effects include the loss of tension of the drop-stitch yarns and the general nonlinearity of the panel. In the next section of Chapter 5, the deformed shape of the simulation is compared to experimental measurements. This investigation includes the comparison of the "kinking angle" (introduced by [7]) predicted by the simulation to that observed experimentally by [7]. In the final section of Chapter 5, the stress distribution of the panel is estimated by an analytical model. The results are compared to the simulation results to evaluate the accuracy of the analytical model. In addition, it is used to validate the assumptions and existing knowledge about inflatable drop-stitch panels.

CHAPTER 2

Literature review

The following literature review is divided in three parts. In the first part, general research for inflatable structures is reviewed. The following section summarizes specific research for inflatable drop-stitch panels. In the last section of the chapter, the existing research about computational modeling of inflatable structures is studied.

2.1 Inflatable Structures

In 2006, Cavallaro et al [9] investigated the influence of different weaving structures and yarn properties on the loading behavior of inflatable beams. Building upon this, an analytical model is derived to describe the deflection of an inflatable beam. The derived analytical model includes shear deformation of the beam. Two different approaches are pursued to obtain the shear modulus that is needed for the analytical model. In the first approach, the woven skin material is assumed to behave like a continuum. The shear modulus is obtained by creating a unit cell model of the woven fibers for finite element analysis and calculating the resulting shear modulus of the unit cell. In the second approach, the shear modulus is obtained based on the distinct yarns [9]. The analytical model derived in Cavallaro et al [9] takes into consideration the pressure dependence and shear deformability of inflated beams. Based on this derived analytical model it is determined that the behavior of inflatable structures differs significantly from conventional non-inflatable structures [9].

2.2 Inflatable Drop-Stitch Panels

Most of the existing research about inflatable structures discusses inflatable beam structures. However, inflatable drop-stitch panels are still a rather new topic. It is known that drop-stitch structures have been used in the 1950s from the U.S. government to investigate the use of inflatable airplanes [16]. Since then, the demand and the need to accurately predict the behavior of inflatable drop-stitch panels is increasing. Therefore, research was conducted from different institutions on the properties specific to these unique structures.

In 2011, Falls and Waters [17] studied the use of the classical beam theory for inflatable drop-stitch panels. In the scope of this research, experiments for panels with different thicknesses are carried out. The results are compared with the classical beam theory. Due to a low correlation between theory and experiment it was concluded that the classical beam theory is not enough for describing the behavior of inflatable drop-stitch panels.

Further research on drop-stitch panels was conducted in 2013 by Cavallaro et al [6]. He developed an extended beam theory including shear deformability as in the beam theory by Timoshenko [18]. To obtain material properties of the panel, biaxial testing of the skin material and tension tests of the drop yarns are performed. The results of the derived shear deformable beam theory are compared to experimental results of four-point bending tests, which were carried out with different inflation pressures on the panel. The derived analytical equations take account for changes in rigidity of the drop-stitch panel due to different inflation pressures. A good correlation between theory and experiment was obtained for pressures higher than 10 psi [6].

In 2013, DiGiovanni [12] investigated drop-stitch panels by manufacturing small inflatable drop-stitch panel cubes for testing. The scope of this research was

to obtain the effective Young’s modulus and shear modulus of the panel cubes for the use in classical beam theory. The effective moduli of the panel are assumed as pressure dependent. Therefore, the panel cubes are tested with different inflation pressures. The moduli are used in the classical beam theory and the results are compared to experiments. Due to a low correlation between theory and analysis it is concluded that the classical beam theory is not enough to fully describe the behavior of drop-stitch panels.

More recently, Felicissimos [19] examined the material properties of inflatable and rigid drop-stitch fabrics. The research included tensile and shear testing of inflatable and rigid drop-stitch structures. From the testing results, it is concluded that the skin material of inflatable drop-stitch panels can assumed to be orthotropic. Due to the small thickness of the skin, the author concludes that it is appropriate to simplify the material behavior with transverse isotropic behavior for most applications [19].

In 2017, Hulton et al [15] investigated natural frequencies and mode shapes of inflatable drop-stitch panels using both experiment characterizations and a computational model. The computational aspects of this study are discussed below in section 2.3.

In 2019, Smith [7] investigated the tensile behavior of both the total skin of an inflatable drop-stitch panel and the individual distinct different layers. The tensile tests characterized the skin material as orthotropic and determined the directionally dependent Young’s moduli and Poisson’s ratios. The results obtained for the material parameter are used in a derived analytical model. The analytical model predicts the bending behavior of an inflatable drop-stitch panel for different inflation pressures. To validate the analytical model, four-point bending experiments for 5, 10, 15 and 20 psi inflation pressure were performed. The experiments show

that the correlation between experimental and analytical results increases with the inflation pressure. To obtain a better correlation for small pressures, a kinking angle, which describes the slope discontinuity at the loading points, was introduced. The kinking angle is observed to decrease with increasing inflation pressures. This research [7], was done in parallel with the study reported in this thesis, where the numerical results are directly correlated with Smith's experimental results.

2.3 Computational Analysis of Inflatable Structures

In the following section literature for computational analysis of inflatable structures is reviewed.

Graczykowski and Heinonen [20] investigated adaptive inflatable structures for protecting wind turbines against ship collisions [20]. The research focuses on the comparison of Abaqus Standard and Abaqus Explicit solutions regarding the use for fluid filled cavities. In the models used for the scope of this research no drop-stitch structures are used. The result of the research is that the use of Abaqus Standard is better for modeling fluid filled cavities because of the opportunity to use FORTRAN user subroutines to control the gas flow.

In 2005, Van and Wielgosz [21] developed analytical and numerical models for bending and buckling of inflatable beams. The analytical solution for beam bending and buckling is derived from the principal of virtual work including the effects of shear deformation. The equations are linearized around the already inflated, prestressed state. The equations are then solved for bending and buckling. In the next step a numerical model is created for 36 different beam geometries of varying radius, length and inflation pressure. 3D membrane elements are used for representing the skin behavior. Comparing the analytical and numerical results, it is shown that the difference in bending is smaller than 2.2 % and in buckling less than 1%. It is concluded that very good agreement is observed between analytical

and numerical results.

As mentioned above in section 2.2, Hulton et al [15] investigated natural frequencies and mode shapes experimental as well as computational. Experiments were carried out on the panel for six different inflation pressures. The first three bending and torsion modes were characterized experimentally. The next part of the research was to create a finite element based computational model. In this model, membrane elements are used for the skin and truss elements for the drop-stitch yarns. The computational analysis includes two steps, inflation and modal analysis. For the inflation step, Abaqus Explicit is used and for the modal analysis Abaqus Standard is used. Comparing the numerical results to the experimental results showed a high dependence on the inflation pressure, with the numerically predicted natural frequencies about 2-3 Hz smaller than the experimental results.

CHAPTER 3

Theoretical Foundation

3.1 Four-point bending Test

A four-point bending or flexure test is often used to determine the flexural response of a beam [22]. The four-point flexure test provides a simple test to determine the range of bending response of a beam. The advantage of a four-point bending in contrast to a three-point bending test, the maximum stress is uniformly distributed between the two load points [1].

In a four-point bending test, the beam is supported by two support points separated by a span, L . A total force, P , is applied on two loading pins as shown in Figure 1. The reaction forces, R , in the support are equal to $P/2$ to remain equilibrium in the beam [1]. A free body diagram of the beam in a four-point bending test is shown in Figure 1. The stiffness of the beam is defined as the ratio of the force P and the deflection at the midspan of the beam.

Figure 1: Free body diagram of four-point bending test [1]

The general equation to calculate the stresses in the beam due to bending is given below in Equation 1 [23]. In the center span, the bending moment, M , is given by the applied force times the distance, a , between the end support and the adjacent load application point (Equation 2) [23]. The variable y denotes

the distance from the neutral axis to the top surface of the beam. I is the area moment of inertia which can be calculated by the area integral over y^2 where the cross-sectional area is the yz -plane as can be seen in Equation 3 [23].

$$\sigma_b = \frac{My}{I} \quad (1)$$

$$M = Fa \quad (2)$$

$$I_z = \int_A y^2 dA \quad (3)$$

3.2 Pressure Vessels

Pressure vessels are used to store pressurized fluids or gases. It is important to know the stresses in pressure vessels for failure analysis and the construction of pressure vessels. The stresses in a thin-walled pressure vessel can be obtained by using a free body diagram and assuming force equilibrium. The pressure vessel is thin walled when the radius r is much smaller than the thickness t of the vessel $r \gg t$ [2]. In [24], a guiding value for using thin walled theory is defined. The thickness of thin walled pressure vessel should be smaller than 10% than the radius $\frac{r}{t} > 10\%$ [24]. For thicker pressure vessels a more complicated approach has to be chosen because the stress distribution within the walls needs to be included in the analysis [25].

To demonstrate the procedure to obtain the stresses in a pressure vessel, a spherical pressure vessel, as can be seen in the left part of Figure 2 is considered. To calculate the stresses, the spherical vessel is theoretically cut in half and the acting forces on the pressure vessel are displayed. For a better understanding of the force equilibrium, the free body diagram is divided in two parts, as can be seen in the center and right diagrams shown in 2.

For a force equilibrium in the pressure vessel, the net force due to the pressure p that is applied on the inside of the vessel must be equal the net force associated

Figure 2: Free body diagram of a spherical pressure vessel [2]

with the stress σ in the material. In the cross section of the spherical pressure vessel, the pressure p is applied on the inner area A of the vessel. Therefore, the resulting force is given as pA with an area of $A = \pi r^2$. The stress σ only appears in the walls of the spherical vessel. The resulting force of the wall stress is given by multiplying the stress with the area of the walls which is calculated by multiplying the perimeter L_p with the thickness of the material t . The resulting force is therefore $\sigma L_p t$ with the perimeter of the spherical vessel $L_p = 2\pi r$ [25]. Because of the equilibrium of these two forces this leads to Equation 4.

$$pA = \sigma L_p t \quad (4)$$

Rearranging the equation and solving for stress σ gives equation 5.

$$\sigma = \frac{pA}{L_p t} \quad (5)$$

Inserting the specific perimeter L_p and area A of the spherical vessel leads to Equation 6.

$$\sigma = \frac{p\pi r^2}{2\pi r t} \quad (6)$$

Simplifying Equation 6 gives Equation 7 for the stress in a spherical thin walled pressure vessel.

$$\sigma = \frac{pr}{2t} \quad (7)$$

Following this principle the stresses can be obtained for different geometries of pressure vessels [2].

3.3 Elastic Material Properties

In continuum mechanics, the properties of materials are described by constitutive equations. In the following, the constitutive equation for a linear elastic material is derived. Linear elasticity is first considered for the uni-axial stress state and then extended to a multi-axial stress state. With a purely elastic material, the deformation of a body is independent of the loading rate. The velocity at which a force is applied to the body is irrelevant for the deformation of the body. Only the amount of force has an influence on the degree of deformation of the body. Another important point of the elasticity model is that the deformation only exists as long as a force is applied to the body. If the force acting on the body is removed, the unloading of the body takes place in exactly the opposite direction to the initial loading. The load and unloading on the body are reversed. Thus, the deformations and distortions on a body are reversible [26].

In actual experiments, the elasticity assumptions are typically valid only for certain materials subjected to small deformations. If the model is limited to small deformations and stresses, the relationship between the strain ϵ and the stress σ can be represented by a linear relationship. The linear relationship is described by the proportionality constant E the modulus of elasticity [27]. The modulus of elasticity describes the resistance of a material to deformation [26]. For a uni-axial stress state, the relation between stress and strain is given in Equation 8 [27].

$$\sigma = E\epsilon \tag{8}$$

Extending Equation 8 to a three-dimensional stress state, the stresses and strains in all three direction x, y, z and the shear stresses and strains acting on the respective planes have to be considered. Still assuming a linear behavior of the material each

stress is related to the strains by constants. The relation for three-dimensional stress state, can be denoted with the fourth order elasticity tensor of constants C_{ijkl} as can be seen in the following Equation 9.

$$\sigma_{ij} = C_{ijkl}\epsilon_{kl} \quad (9)$$

In this form, the elasticity tensor C_{ijkl} has 81 components. Taking advantage of the symmetric property of the stress and strain tensors, the independent components of the tensor can be decreased to 36 [27]. Furthermore, including the strain energy function reduces the independent elastic constants to 21 [28]. A material that can be described by 21 independent coefficients is called fully anisotropic where the material behavior is dependent on the loading direction. The constitutive equation for a fully anisotropic material can be found in the following Equation 10 [27].

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ & & C_{33} & C_{34} & C_{35} & C_{36} \\ & & & C_{44} & C_{45} & C_{46} \\ & \text{Sym.} & & & C_{55} & C_{56} \\ & & & & & C_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ 2\epsilon_{xy} \\ 2\epsilon_{yz} \\ 2\epsilon_{zx} \end{Bmatrix} \quad (10)$$

For constitutive equations describing materials with symmetry axes, engineering constants are commonly used because they are directly related to a physical meaning. The engineering constants can be descriptively regarded as a measure of resistance to deformation [28].

The modulus of elasticity or Young's modulus is a value for the resistance of a material to compression or tension in the direction indicated by the index. E_1 , for example, indicates the resistance of a material to normal stress in the x-direction [29]. The shear modulus, G , is the resistance to shear forces. This is indicated by the ratio of the shear stress τ and the tangent of the shear angle γ . The shear angle is in the plane described by the indices. The relationship between shear

angle, shear stress and shear modulus is described in the following Equation 11 [29].

$$\tau_{ij} = G_{ij}\gamma_{ij} \quad (11)$$

A third elastic constant, the Poisson's ratio ν , indicates how large the deformation of a material is orthogonal to the direction of elongation. ν_{12} describes the ratio of elongation in the y-direction to elongation in the x-direction for a normal stress in the x-direction. The Poisson's ratio can be determined by the following Equation 12 [30].

$$\nu_{ij} = -\frac{\epsilon_{jj}}{\epsilon_{ii}} \quad (12)$$

A material with two symmetry planes can be described by nine independent coefficients. These coefficients are, for example: E_1 , E_2 , E_3 , G_{12} , G_{13} , G_{23} , ν_{12} , ν_{13} and ν_{23} . The constitutive equation dependent on the nine engineering constants can be found in Equation 13 [26].

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1-\nu_{23}\nu_{32}}{E_2 E_3 D} & \frac{\nu_{21}+\nu_{31}\nu_{23}}{E_2 E_3 D} & \frac{\nu_{31}+\nu_{21}\nu_{32}}{E_2 E_3 D} & 0 & 0 & 0 \\ & \frac{1-\nu_{13}\nu_{31}}{E_1 E_3 D} & \frac{\nu_{32}+\nu_{12}\nu_{31}}{E_1 E_3 D} & 0 & 0 & 0 \\ & & \frac{1-\nu_{12}\nu_{21}}{E_1 E_2 D} & 0 & 0 & 0 \\ & & & G_{23} & 0 & 0 \\ & \text{Sym.} & & & G_{31} & 0 \\ & & & & & G_{12} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ 2\epsilon_{xy} \\ 2\epsilon_{yz} \\ 2\epsilon_{zx} \end{Bmatrix} \quad (13)$$

with

$$D = \frac{1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{13}\nu_{31} - 2\nu_{21}\nu_{32}\nu_{13}}{E_1 E_2 E_3}$$

The three additional Poisson's ratio that appear in the equation are dependent on the other constants and can be calculated by Equation 14 [26].

$$\frac{E_1}{\nu_{12}} = \frac{E_2}{\nu_{21}}, \quad \frac{E_1}{\nu_{13}} = \frac{E_3}{\nu_{31}}, \quad \frac{E_3}{\nu_{23}} = \frac{E_2}{\nu_{23}} \quad (14)$$

To ensure stability of orthotropic materials, the conditions in the following equations 15 and 16 have to be fulfilled [31].

$$E_1 > 0, \quad E_2 > 0, \quad E_3 > 0 \quad (15)$$

$$\nu_{12}\nu_{21} < 1, \quad \nu_{23}\nu_{32} < 1, \quad \nu_{31}\nu_{13} < 1 \quad (16)$$

For materials that do not have a behavior that is dependent on the direction only two independent coefficients are needed to describe the stress-strain behavior. These materials are called isotropic materials. Starting from the constitutive equation for an orthotropic solid, all Young's moduli, shear moduli and Poisson's ratios are equal due to the directional independence. Thus, we are left with the three parameter E , G and ν . The shear modulus G is dependent on the Poisson's ratio and the Young's modulus. The shear modulus can be calculated by Equation 17.

$$G = \frac{E}{2(1 + \nu)} \quad (17)$$

The constitutive equation for a linear elastic isotropic solid is given in Equation 18.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{E(1-\nu)}{(1-2\nu)(1+\nu)} & \frac{E\nu}{(1-2\nu)(1+\nu)} & \frac{E\nu}{(1-2\nu)(1+\nu)} & 0 & 0 & 0 \\ & \frac{E(1-\nu)}{(1-2\nu)(1+\nu)} & \frac{E\nu}{(1-2\nu)(1+\nu)} & 0 & 0 & 0 \\ & & \frac{E(1-\nu)}{(1-2\nu)(1+\nu)} & 0 & 0 & 0 \\ & & & G & 0 & 0 \\ & \text{Sym.} & & & G & 0 \\ & & & & & G \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ 2\epsilon_{xy} \\ 2\epsilon_{yz} \\ 2\epsilon_{zx} \end{Bmatrix} \quad (18)$$

For thin plates, another simplification for the constitutive equation can be made. Assuming that the length and width direction are in the xy -plane and the thickness in z -direction, it can be approximated that stresses in z -direction, σ_{33} , τ_{13} and τ_{23} , are equal to zero. Therefore, the constitutive equation for the so-called plane stress state for an isotropic material reduce to Equation 19 [3].

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{E}{1-\nu^2} & \frac{\nu E}{1-\nu^2} & 0 \\ \frac{E\nu}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & G \end{bmatrix} \begin{Bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ 2\epsilon_{12} \end{Bmatrix} \quad (19)$$

Following the same procedure for an orthotropic material, the constitutive equation for an orthotropic material can be derived as seen in Equation 20 [3].

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{E_1}{1-\nu_{12}\nu_{21}} & \frac{\nu_{21}E_1}{1-\nu_{12}\nu_{21}} & 0 \\ \frac{E_1\nu_{12}}{1-\nu_{12}\nu_{21}} & \frac{E_2}{1-\nu_{12}\nu_{21}} & 0 \\ 0 & 0 & G_{12} \end{bmatrix} \begin{Bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ 2\epsilon_{12} \end{Bmatrix} \quad (20)$$

3.4 Classical Lamination Theory

Laminates are composite materials consisting of multiple thin layers typically with a dimension of order 1 mm (0.04 inch) [3]. The layers can be made of different materials and are bonded together. To predict the macroscopic behavior of the laminate, classical lamination theory can be applied. Classical lamination theory is used to calculate the homogenized properties of laminates. Important for the calculation is the use of the assumptions of the plate theory [3]. It is assumed that the thickness of the whole laminate is much thinner than the dimensions in length and width direction. As a result, each layer of the laminate and the whole laminate itself is subjected to plane stress. Therefore, for every layer and for the whole laminate the stresses σ_{13} , σ_{23} , and σ_{33} are zero and the strains ϵ_{13} , ϵ_{23} and ϵ_{33} are non-zero [3]. The constitutive equation for each layer is therefore given by Equations 19 and 20, for the cases of isotropic or orthotropic materials, respectively.

The constitutive equation for the whole laminate is then assembled by using the equations for the single layers. The resulting equation for the homogenized laminate can be found in Equation 21 [32]. The vector on the left side of the equation includes $\{N\}$, the in-plane resultant force vector, and $\{M\}$, the resultant moment vector [3]. The vector on the right side of the equation includes $\{\epsilon\}$, the vector of in-plane membrane strains, and $\{\kappa\}$, the vector of plate curvatures associated with bending [3]. Due to this, the stiffness matrix can be partitioned into three different block matrices: the A , B and D matrices. A is the extensional or membrane stiffness matrix, D is the bending stiffness or plate bending matrix and

B is the coupling matrix that couples the membrane and plate bending behavior [3].

$$\begin{Bmatrix} N_1 \\ N_2 \\ N_6 \\ M_1 \\ M_2 \\ M_6 \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} \\ A_{16} & A_{26} & A_{66} & B_{16} & B_{26} & B_{66} \\ B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{12} & B_{22} & B_{16} & D_{12} & D_{22} & D_{26} \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \epsilon_6 \\ \kappa_1 \\ \kappa_2 \\ \kappa_6 \end{Bmatrix} \quad (21)$$

The extensional stiffness matrix A describes the relation between in-plane forces and in-plane strains. It can be calculated by summing the stiffness matrices in-plane stress for each layer multiplied by the thickness of the layer as given in Equation 22. The total number of layers is denoted with n and the current layer is the k^{th} -layer. $Q_{ij}(k)$ is the stiffness matrix of the k^{th} -layer for plane stress and is given for isotropic behavior in Equation 19 and for orthotropic behavior in Equation 20. $x_3(k)$ is the distance from the laminate mid-plane to the k^{th} -layer as can be seen in Figure 3 [3].

$$A_{ij} = \sum_{k=1}^n Q_{ij}(k)[x_3(k) - x_3(k-1)] \quad i, j = 1, 2, 6 \quad (22)$$

The bending stiffness matrix D describes the relation of moments to flexural strains. It can be calculated by Equation 23. The same parameter Q_{ij} and x_3 are used, as described above for the membrane stiffness matrix in Equation 22 [3].

$$D_{ij} = \frac{1}{3} \sum_{k=1}^n Q_{ij}(k)[x_3^3(k) - x_3^3(k-1)] \quad i, j = 1, 2, 6 \quad (23)$$

The coupling matrix B couples membrane forces to flexural strains and moments to in-plane strains. It can be calculated using Equation 24. The peculiarity of the coupling matrix is that the terms of Equation 24 cancel each other out for a symmetric laminate with respect to the mid-plane. If all terms B_{ij} are equal to zero, no coupling between bending and stretching of the laminate exists [3].

$$B_{ij} = \frac{1}{2} \sum_{k=1}^n Q_{ij}(k)[x_3^2(k) - x_3^2(k-1)] \quad i, j = 1, 2, 6 \quad (24)$$

Figure 3: Representation of the x_3 coordinate for different layers in a laminate [3]

After assembling the ABD stiffness matrix, the representative engineering constants for the homogenized laminate structure can be obtained. The first step for doing this, is taking the inverse of the ABD matrix. In the following, the inverse of the ABD matrix is again displayed with sub-matrices as shown in Equation 25. For symmetric laminates, it can be shown that $A^{-1} = \bar{A}^{-1}$, $D^{-1} = \bar{D}^{-1}$ and $B^{-1} = \bar{B}^{-1} = 0$. However, for laminates that are not symmetric, $A^{-1} \neq \bar{A}^{-1}$, $D^{-1} \neq \bar{D}^{-1}$ and $B^{-1} \neq \bar{B}^{-1} \neq 0$ applies [33].

$$\begin{bmatrix} A & B \\ B & D \end{bmatrix}^{-1} = \begin{bmatrix} \bar{A}^{-1} & \bar{B}^{-1} \\ \bar{B}^{-1} & \bar{D}^{-1} \end{bmatrix} \quad (25)$$

In the calculation of the engineering constants, a distinction for different loads is made. If a laminate is only under in-plane load, the membrane stiffness matrix is used for the calculation [32]. The equations to obtain the Young's modulus, E , and shear modulus, G , are given in Equation 26.

$$E_x = \frac{1}{(\bar{A}^{-1})_{11}h}, \quad E_y = \frac{1}{(\bar{A}^{-1})_{22}h}, \quad G_{xy} = \frac{1}{(\bar{A}^{-1})_{66}h} \quad (26)$$

The equations to obtain the Poisson's ratio ν are given in Equations 27.

$$\nu_{xy} = -\frac{(A^{-1})_{12}}{(A^{-1})_{11}}, \quad \nu_{yx} = -\frac{(A^{-1})_{12}}{(A^{-1})_{22}} \quad (27)$$

If an out of plane load is applied to the laminate, the bending stiffness matrix D is used to obtain the effective engineering constants for the laminate. The equations to obtain the Young's modulus, E , and the shear modulus, G , are given in Equation 28.

$$E_{x,b} = \frac{12}{(\bar{D}^{-1})_{11}h^3}, \quad E_{y,b} = \frac{12}{(\bar{D}^{-1})_{22}h^3}, \quad G_{xy,b} = \frac{12}{(\bar{D}^{-1})_{66}h^3} \quad (28)$$

The equations to obtain the Poisson's ratio for bending are given in Equation 29.

[32]

$$\nu_{xy} = -\frac{(D^{-1})_{12}}{(D^{-1})_{11}}, \quad \nu_{yx} = -\frac{(D^{-1})_{12}}{(D^{-1})_{22}} \quad (29)$$

3.5 Finite Element Analysis

In this section, first the basics and procedure of finite element analysis is described. Then, the different element types used in the scope of this research are explained in more detail. Finally, modeling of contact interactions using Abaqus Standard is discussed.

3.5.1 Foundations of Finite Element Analysis

Finite element analysis is a tool to obtain the results of mechanical problems. Analytical solutions give a continuous result for the whole structure. On the contrary, numerical solutions such as finite element analysis give discrete results for specific points of the structure. In the finite element method, the whole structure/problem is divided into smaller parts, so-called finite elements, to obtain smaller problems that can be solved more easily. This makes the finite element analysis suitable for solving mechanical problems that include complex geometries, loading conditions and materials where analytical solutions cannot be obtained. By

dividing the larger problem in many smaller problems, the number of equations that have to be solved increases significantly. The advantage of using finite element analysis is strongly linked with the great computational power of modern computers [4]. Finite element analysis can also be used to solve other field problems such as heat transfer and fluid flow problems. In the scope of this research, only structural problems are considered. Therefore, the following description of the finite element analysis method focuses on structural problems [4]. The finite element analysis procedure can be divided into eight steps [4] that are shown in Figure 4 and are explained below.

Figure 4: Steps for Finite Element Analysis [4]

In the first step, the body is divided in finite elements. This process is called discretization or application of a mesh. The element size and type must be chosen in a way that the behavior of the structure is accurately modeled but at the same time computation time should remain low. Some element types that can be used for an analysis are explained in section 3.5.2. The necessary element size can be determined by a convergence analysis. With an infinite number of elements, the result of a finite element analysis is the same as the analytical result. The desired

accuracy of the solution must be considered for each problem. Sometimes it is useful to vary the element size over the structure. Small elements are necessary where geometrical changes or nonlinear behavior appears in the analysis. Large elements can be used when the results only undergo small changes during the analysis [4].

Step two includes the choice of displacement functions. The functions used are piecewise-continuous which means that they are continuous in the range that they are used. The choice of displacement functions differs with the number of nodes in each element. For a two-node truss element, for example, linear displacement functions are used. If the truss element has three nodes, quadratic displacement functions are used, etc. The displacement functions for equal elements in an analysis are always the same [4].

In step three, strain-displacement and stress-strain relations are established. A strain- displacement relation for a one-dimensional (1D) case is for example given in Equation 30 [4].

$$\epsilon_x = \frac{du}{dx} \quad (30)$$

The relationship between stress and strain is given by the constitutive equations which are described in section 3.3.

In the next step, the stiffness matrices for all elements in their local coordinate systems and their force displacement behaviors are derived. For this procedure, different approaches can be used. For 1D-elements and line-like structures like beams and trusses, the stiffness method is a useful approach. The stiffness matrix and force displacement equations are derived using force equilibrium equations [4]. For two-dimensional (2D) and three-dimensional (3D) structures, the work or energy method is better suited. For this method, the principle of virtual work, the principle of minimum potential energy or Castigliano's Theorem can be used for the

derivation of the stiffness matrix and equations. Using linear elastic materials, all three approaches lead to the same element equations. For nonlinear materials, only the principle of virtual work can be used [4]. The third method is known as methods of weighted residuals where, for example, Galerkin's method is used to obtain the stiffness matrix and force-displacement equations [4]. All three approaches give the same results but can be useful depending on the problem. The stiffness matrix and force-displacement equations are given in Equation 31 in the general form [4]. The force vector describing the forces applied on the nodes is denoted as f . The element stiffness matrix has the dimension of the number of degrees of freedom that the element has and is given by $[k]$. The displacement vector d describes the displacement in degrees of freedom of the element.

$$\{f\} = [k]\{d\} \quad (31)$$

Following the derivation of the local stiffness matrix and force-displacement equations, the system of equations for the whole structure in the global coordinate system is assembled. The global system of equations is given in matrix form in Equation 32 [4].

$$\{F\} = [K]\{D\} \quad (32)$$

In this state, the stiffness matrix is singular meaning that its determinant is zero. With a singular matrix, the system of equations cannot be solved. To obtain a nonsingular matrix, boundary conditions and constraints of the structure are applied to modify the system of equations [4].

In step six, the derived system of equations with a nonsingular stiffness matrix is solved for the displacements of each node. The dimension of the stiffness matrix is now equal to the number of degrees of freedom of the whole structure. To solve the equations, different solving techniques can be used. For example, an iterative approach is the Gauss Seidel method [4]. In Abaqus/Standard, the

Newton, modified Newton or quasi Newton methods are used to solve nonlinear problems [34].

After solving for the nodal displacements of the structure, these results can be used to obtain stresses and strains of the structure, which is the subject of step seven. To solve for the stresses and strains, the strain-displacement relationship and constitutive equations, as mentioned in step three, are applied.

The last step of a finite element analysis is the interpretation and evaluation of the results. At first, the obtained results should be verified. For verification purposes, there are different possibilities. For instance, the results can be compared to analytical solutions or to experimental results. In addition, the convergence of the solution should be observed. Verification of a model is very important to ensure that material behavior, element types and size are chosen properly. If the choice is not reasonable, there may be strong differences between the observed mechanical problem and the solution obtained by finite element analysis [34]. After the finite element results are verified, the obtained solutions can be used for different investigations. For example, it is important to look at high stresses and large displacement of a structure. These values are important for choosing the appropriate characteristics and materials for structures [4].

3.5.2 Element Types

For finite element analysis, different elements can be used to simplify the calculations. In Abaqus, the general notation that describe the elements contains 4 digits in 3 parts. The first digit denotes the kind of element. 'T' stands for truss, 'R' for rigid, 'S' for shell etc.. The next two digits describe the dimension of the element: '2D' for two-dimensional and '3D' for three-dimensional. For shell elements these two digits are not considered. The last digit gives the number of nodes that the element has. For some special elements, there are more digits

to describe their behavior. For instance, a four node shell element with reduced integration is called a 'S4R' element with 'R' for reduced integration.

In the following sections, the element types used in the scope of this research are presented briefly.

Truss elements

One of the simplest elements that can be used for finite element analysis is the truss element. A truss element can only take up forces in the axial direction. Transmitting moments through truss elements is not possible. A typical truss element has two nodes, each with two degrees of freedom in 2D or three degrees of freedom in 3D that allow the nodes to move in x- and y-direction and for 3D also in z-direction [4]. Truss elements can be used for line like structures, for example, wires or bridge structures. In finite element programs the geometry of the truss element is defined by its length, L , and cross-sectional area, A [34].

To calculate the relation between force, f , and displacement, d , Equation 31 is used. It is the same equation that relates force and displacement for a spring. However, the stiffness constant differs between spring and truss [4]. The stiffness for a truss is given in Equation 33, with E the Young's modulus. [4]. Details of the derivation of the stiffness are given in [4].

$$k = \frac{AE}{L} \quad (33)$$

Substituting the stiffness of a truss in Equation 31, the equation for a two node truss element in its local coordinate system is given in the following Equation 34 [4]. The derivation of the local stiffness matrix would be obtained in step four as explained in section 3.5.1.

$$\begin{Bmatrix} f_{1x} \\ f_{2x} \end{Bmatrix} = \frac{AE}{L} \begin{bmatrix} 1 & 1 \\ 1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \quad (34)$$

Step five would be the assembling of the global system of equations. If we assume for this case that there is only one element than the local coordinate system is equal to the global one and therefore step five can be skipped. And Equation 34 can be solved with respect to boundary conditions and loads in step six (see section 3.5.1). When the displacements are known it can be proceeded with step seven and the strains and stresses of the truss can be obtained by inserting the displacement in strain-displacement and stress-strain relations specific to the element type. The next step would then include the investigation of the results.

Shell elements

Shell elements are used for structures where the length and width dimensions are much greater than the thickness of the structure [3]. In Abaqus, the width and length direction of a shell is in the 1-2 surface [34]. Due to this geometry, general (thin) shell elements do not have a stress component in direction of the outward normal; $\sigma_{33} = 0$. Therefore, plane stress conditions can be used to simplify the material behavior of shell elements. The constitutive equation for orthotropic material in plane stress are given in Equation 20. The thickness of a shell can be defined in the property module of Abaqus/CAE. Depending on a linear or nonlinear analysis, the thickness of the shell can change. In the default settings of Abaqus/CAE, the analysis is linear and the thickness does not change. For large deformations, the setting of the analysis should be changed to nonlinear to include all nonlinear effects, including the change of thickness of the shells [34].

By default, the displacement, stresses and strains are calculated for the mid plane of the shell element. This setting can be changed if necessary by applying an offset. A typical 3D shell element used in Abaqus has four nodes each with six degrees of freedom. The nodes can move in x, y and z direction and also have rotational degrees of freedom in each of the three axis. This four node shell

element is shown in Figure 5. The force displacement equation for each shell node

Figure 5: Degree of freedom of four node shell element[5]

is given in Equation 35. The whole stiffness matrix of a four node shell would have dimensions of 24×24 . On the left side of the equation, the force vector is given. Forces can be applied in all directions of the node. f_u is the force in 1 direction, f_v in 2 direction and f_w is the force in 3 direction. The remaining three forces are moments about the axis denoted in the index. On the right side of the equation, the stiffness matrix and displacement vector as defined in Equation 31 can be found. The displacement vector includes the displacements in all degrees of freedom. In the local coordinate system, the nodal displacement θ_z is zero, as can be seen in the stiffness matrix. The stiffness matrix of a shell is assembled by the stiffness matrix of a plane stress element and the stiffness for a plate bending element [35].

$$\begin{Bmatrix} f_u \\ f_v \\ f_w \\ f_{\theta_x} \\ f_{\theta_y} \\ f_{\theta_z} \end{Bmatrix} = \begin{bmatrix} \mathbf{K}_{\text{PS}}^{(e)} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & \mathbf{K}_{\text{PB}}^{(e)} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u \\ v \\ w \\ \theta_x \\ \theta_y \\ \theta_z \end{Bmatrix} \quad (35)$$

A plane stress element has two degrees of freedom in the 1, or u , and 2, or v , directions. In the stiffness matrix in Equation 35 the 2×2 stiffness matrix

for a plane stress element is denoted with $\mathbf{K}_{\text{PS}}^{(\text{e})}$ and can be found in the upper left. A plate bending element has three degrees of freedom, w , θ_y and θ_z . The plate bending element stiffness matrix is denoted with $\mathbf{K}_{\text{PB}}^{(\text{e})}$. The derivation of the element stiffness matrices for plane stress and plate bending can be found in [4]. For the derivation of the shell element stiffness matrix, the reader is referred to [35].

Rigid elements

Rigid body elements are used for very stiff structures where the stress distribution in the rigid body is not needed. The advantage of rigid elements is the decrease of computational time that is needed for the analysis. A case where the use of rigid bodies is useful, for example, is the simulation of tooling. There are different types of rigid elements including 3D, 2D and plane stress elements that can be used. The choice of element type depends on the geometry and behavior of the body [34]. All elements of a rigid body are linked to a reference node. The distance and displacement of all nodes of the rigid body stay constant in relation to each other and the reference node. Therefore, boundary conditions, constraints and loads are applied only at the reference node of the rigid body. A rigid body does not deform but can move by large distances in every direction that is given by the degree of freedoms of the specific element. There are some restrictions in the use of rigid body elements. For example, the contact between two rigid bodies is not possible in Abaqus/Standard. In general, a contact between an elastic material and a rigid body is possible as long as the master surface is a surface of the rigid body and the slave surface of the elastic body. The explanation of contact modeling in Abaqus can be found in the following section 3.5.3.

3.5.3 Contact Modeling

Contact modeling in Abaqus is used to describe the interaction between structures. In Abaqus/Standard, there are two different algorithms to implement contacts, contact pairs and general contact [34]. General contact is a more universal formulation of contact than contact pairs. For general contact, whole areas of the model that could be in contact to each other or to itself are defined. It is also possible to automatically include every part of the model for possible contacts. This allows an easy definition for contacts. For general contact, the same approach as for contact pairs is used but the surfaces are selected automatically [34]. In the scope of this research, contact pairs are used and explained in more detailed below.

For a desired contact between two bodies, the contact pair approach can be used. There are two different contact discretizations that are used for contact pairs. The node-to-surface approach and the surface-to-surface approach. For contact pairs, the surfaces interacting at the beginning or at any time in the simulation with each other are selected. One surface is assigned as master surface and one is specified as slave surface [34]. The master surface should be the one that is stiffer than the slave surface. When a contact including a rigid body is defined the surface belonging to the rigid body must therefore always be the master surface. In addition, the slave surface of the contact pair should have a finer mesh than the master surface. The right choice of slave and master surface is important to ensure an accurate solution and low computational effort [34, 36].

In a contact pair that uses a node-to-surface approach, every node of the slave surface interacts with a projection point of the master surface which is obtained by interpolation between the master nodes next to the slave node. In a classical node-to-surface approach the slave nodes can not penetrate the master surface but the master surface can penetrate the slave nodes [34].

The surface-to-surface approach is especially well suited for surface pairs with opposing normals. In contrary to the node-to-surface approach, in the surface-to-surface approach an integral over the adjacent nodes of the observed slave node is applied to consider the shape of the surfaces. With this approach much smaller and fewer penetrations of the nodes of the contact surfaces are attained. In addition, the surface-to-surface contact leads to a better convergence and more accurate results in contact stresses.

After defining the surfaces for the contact pair, a sliding option has to be chosen. One of the two options finite sliding and small sliding can be selected. For small sliding the master surface is approximated with a planar gliding surface for each slave node. This approximation is only considered for the undeformed master surface. Therefore, small sliding can only give accurate results for relatively small movements of the slave surface to the master surface. In the finite sliding approach, the deformation of the surfaces during the simulation is included in the calculation. Therefore, finite sliding is the more general approach but also leads to a higher computational effort.

If shell elements are used for contact pairs, the user needs to specify if the upper surface, lower surface or both sides are in contact. In addition, it can be specified if the thickness of the shell elements should be included in the contact or not.

After specifying the contact interaction, the contact properties must be defined. The contact properties can, for example, be defined for the behavior in tangential and normal direction of the contact. It is also possible to define damping or damage in the contact definition[34]. In the tangential behavior, it is specified if the contact includes friction. If friction is desired, different algorithms that enforce the friction behavior are available. In the normal contact behavior, it can be

decided if a hard contact or a soft contact should be used.

In general, the contact algorithm introduces a nonlinear behavior into the system. Therefore, steps including contact interactions have to be computed increment-wise and nonlinearity must be activated in the step settings.

3.6 Inflatable Drop-Stitch Structures

Inflatable drop-stitch structures consist of an airtight shell. Depending on the application, the sides of the skin are connected with drop-stitch yarns of different lengths. The described structures are versatile and can be used in a broad range of applications, including marine, space and recreational applications [14, 15]. The usage of inflatable structures ranges from stand up paddle boards and kayaks to space shelters or loading ramps.

Inflatable structures offer different advantages in comparison to conventional structures. They are lightweight, easy to transport and offer high stiffnesses when inflated. In addition, the stiffness of the structure can be varied by changing the air pressure in the panel [6]. Inflatable drop-stitch structures can be built in different shapes and sizes depending on the application.

In Figure 6, the different layers of the skin of an inflatable drop-stitch panel are shown. The inner layer is the woven drop-stitch fabric. The drop-stitch fabric for the upper and lower skin is woven concurrently [6]. In regular intervals a yarn is dropped from one skin to the other. Thus, the connection of the upper and lower skin by the drop-stitch yarns is produced. The length of the dropped yarns determines the overall thickness of the panel. Different weaving techniques can be used to obtain the desired properties of the material. A plain weave, with an orthogonal arrangement of the yarns leads to a high extensional stiffness but low shear stiffness. A braided weaving technique, on the contrary, effects a higher shear stiffness and a low extensional stiffness [9].

Figure 6: Characteristic composition of the drop-stitch panel

The woven drop-stitch fabric is coated with a layer of a polymer. On top of this layer a second woven layer, which is often referred to as the chafer layer, is added. The outermost layer of the skin is a second layer of a polymer. Due to the polymer coating the air-tightness of the skin is achieved and the panel can be inflated with air. A common air pressure used for panels is 15 psi or higher [37].

The load carrying behavior of inflatable panels is comparable to sandwich structures [6]. Sandwich structures consist of two thin layers surrounding a relatively thick core. The core is a lightweight material and the facings consist of a high strength material to ensure stiffness of the structure. The core of a sandwich structure should support shear loads [3]. For an inflatable drop-stitch panel the core is the pressurized air in the panel. The shear stiffness is given by the drop-stitches that constrain the relative movement of the upper and lower skin. The facings of the sandwich structure are analogous to the skin of the drop-stitch panel [6].

By the inflation of the panel, the skin is bi-axially pretensioned. Due to the inflation pressure the panel gets stiff and can resist a mechanical loading. In Figure 7, the stress state of an inflatable panel under bending load is illustrated. The figure shows only half of the panel because it behaves symmetrically. To explain

the behavior of the panel, it can be divided into two regions. In region one, the panel only bends. In region two, the panel has to resist bending as well as shear forces. In general, the stresses imposed due to bending are superposed with the stresses due to the inflation. The bending of the panel leads to a compression in the upper skin and tension in the lower skin [6]. Before failure due to an overload

Figure 7: Superposed stresses for a bending load [6]

condition appears, the so-called wrinkling can be detected for a panel. The onset of wrinkling occurs because of a local loss in tension. Wrinkling is explained for an upper skin element in Figure 8. In Figure 8a, the stress state after the inflation can be seen. Due to the inflation pressure the skin is bi-axially pretensioned with the hoop stress σ_{Hoop} in the width direction and the longitudinal stress σ_{Long} in the length direction, the stress due to bending is superposed with the inflation pressure [6]. The bending stress acts in the opposite direction as the inflation pressure as can be seen in Figure 7. On this upper surface, a bending stress equal to the stress due to inflation the stresses cancel each other and longitudinal stresses are zero for the local skin element. This state is called wrinkling onset state and

can be seen in Figure 8b. After the wrinkling onset state is reached the load capability of the panel decreases and buckling and collapse are much more likely to occur. In the next stress state (Figure 8c), the stresses due to bending exceed the stresses due to inflation and the longitudinal stresses are negative, resulting in a compression of the skin element. This state is called post wrinkled state. An advantage of the wrinkling that occurs before the failure of a panel is that wrinkling is visually detectable. Therefore, the onset of failure can be predicted and necessary countermeasures can be performed [6]. If an overload condition

Figure 8: Upper skin stress states for a bending load [6]

appears despite the countermeasures, a fail-safe collapse is possible for inflatable panels. This means that the panel can go back to the original condition when the overload is removed [6].

3.6.1 Specific Characteristics of the Investigated Panel

The scope of this research is panel-shaped structures. An inflatable drop-stitch panel which is representative for this research can be seen in Figure 9. On the left side of the Figure, the panel can be seen uninflated, and on the right side in its inflated state. The dimensions of the panel are given in Figure 10. The length of the panel is 100 inches and the width 24 inches. The thickness of the Panel is 4 inches and is given through the length of the drop-stitch yarns. The radius at the edges of the inflated panel measures half the stated thickness of the panel.

Figure 9: Inflatable drop-stitch panels

Figure 10: Dimensions of the panel

As explained in section 3.6, the skin includes woven fabric. The weaving technique used for the panel in this research is plain weave. Thus, due to the process of weaving, the yarn in the skin of the panel is arranged orthogonally in warp and weft directions. The yarn that is dropped from the upper to the lower skin is included in the yarn for the warp direction. The warp direction of the drop-stitch fabric is in the length direction of the panel and the weft direction is in the orthogonal width direction as shown in Figure 11 [6].

Width: weft direction

Length: warp direction

Figure 11: Definition of the warp and weft direction

The skin material includes four layers as can be seen in the cross-section of the skin in Figure 12. At the edges of the panel the upper and lower skin overlap, resulting in a double layer of the skin material at these locations [6].

Figure 12: Sideview of inflatable drop-stitch panel skin

The outermost layer of the skin material is made of neoprene to ensure the airtightness of the panel (see Figure 13a). The following chafer layer is a woven fabric as can be seen in Figure 13b. The chafer layer highly contributes to the overall stiffness of the skin. The chafer layer is followed by another layer of neoprene (see Figure 13a). The inner layer is another woven layer, the drop-stitch fabric, which is shown in Figure 13c. In addition, the drop yarns that connect the upper and lower skin can be partially seen in Figure 13c. It is observed that there are approximately 24 drop-stitch yarns per square inch.

Figure 13: Close-up (1×1 inch) of the different layers of the skin

The characteristic linearized elastic properties for the distinct layers and the overall skin have been determined Smith [7] in prior research. Tension tests have been carried out to determine the engineering constants. The Young's modulus and Poisson's ratio as well as the thickness of each layer are summarized in Table 1. For the neoprene layer a range of the Young's modulus is given because the specific durameter of the neoprene used for the panel skin is unknown [7]. For the methodology and discussion of the obtained engineering constants, the reader is referred to Smith [7].

Characteristic	Skin	Neoprene	Chafer	Drop-stitch	Yarns
Thickness [in]	0.055	0.01	0.02	0.015	unknown
Dependent on Dir.	yes	no	yes	yes	no
E_{1-weft} [psi]	18662	376 - 1189	35133	42986	45000
E_{2-warp} [psi]	68100	-	96310	15810	-
ν_{12}	0.297	0.49	0.715	0.37	0.3
ν_{21}	0.727	-	0.123	0.136	-

Table 1: Material characterization [7]

CHAPTER 4

Homogenization of the Skin Material with Classical Lamination Theory

In the following sections, the classical lamination theory, as explained in Section 3.4, is used to obtain the homogenized properties of the skin material of an inflatable drop-stitch panel. The overall engineering constants for the skin material are determined for different compositions of the skin.

4.1 Symmetrical Skin Material Laminate

In the first attempt, the classical lamination theory is applied to an idealized symmetric composition of the skin. The drop-stitch fabric layer is theoretically divided into two parts to obtain the symmetry as can be seen in Figure 14 (see original composition in Figure 12). The skin is assumed to be symmetric since the calculation for the classical lamination theory is simplified for this case. It is assumed that rearranging the composition of the skin has only a negligible effect on the overall in-plane stiffness. For a symmetric laminate no coupling between membrane extension and plate bending exists. Therefore, the terms of the coupling matrix B are zero [3]. Due to the symmetry assumed for the skin it is not necessary

Figure 14: Symmetric skin composition

to calculate the full ABD -matrix. The skin material of the inflatable panel is under an extensional loading condition. For extensional loading conditions the material can be simplified by membrane behavior. Therefore, it is enough to only compute

the membrane matrix A . The membrane matrix can be calculated by assembling the distinct matrices for each layer with respect to their thickness as given in Equation 22. The engineering constants given in Table 1 are used for the different layers of the skin. For neoprene a range of values for the Young's modulus is given in the table. For the computation in this section a Young's modulus of 376 psi is used. The engineering constants for the homogenized skin are calculated with the inverse of the membrane matrix A as given in Equation 26. The Young's modulus for the distinct layers and the resulting Young's modulus for the full skin are obtained using the classical lamination theory with the assumption of symmetry of the skin are displayed in Figure 15. The Young's modulus is shown individually for the warp and weft direction of the panel (see warp and weft direction in Figure 11). As

Figure 15: Results for the Young's modulus of a symmetric skin composition obtained by the classical lamination theory

shown in the Figure 15 the material contributing the most to the stiffness in weft direction is the drop-stitch fabric, followed by the chafer layer. Both the chafer layer and the drop-stitch fabric are stiffer than the homogenized skin material. The neoprene layer acts to reduce the overall skin stiffness and since the stiffness of neoprene is negligible in comparison to the other layers, the value used for the

neoprene Young's modulus has little effect on the overall skin stiffness.

The Young's modulus computed with the classical lamination theory is slightly higher than the Young's modulus obtained experimentally by Smith [7]. The reason for this result is that, as mentioned above, the Young's modulus for the chafer and drop-stitch layer are much higher than the resulting Young's modulus of the skin.

In the warp direction of the material, the stiffness of the chafer layer is about six times higher than the stiffness of the drop-stitch fabric. The Young's modulus of the chafer layer is higher than the Young's modulus of the overall skin. The drop-stitch fabric on the other side, has a much lower stiffness than the overall skin.

The result for the resulting Young's modulus obtained by the classical lamination theory is much smaller than the experimentally obtained value [7] for the skin. The reason for this result is the much smaller Young's modulus of the neoprene and drop-stitch layers in comparison to the overall skin stiffness in warp direction.

In Figure 16 the resulting Young's modulus obtained by the classical lamination theory is compared more precisely with the experimental stress-strain curves. The lamination theory result for the weft direction, shown in blue, is within the error bounds or only slightly above. The lamination theory result for the warp direction, shown in red, is below the experimental result even when considering the error bounds. The slope for the warp direction obtained by the classical lamination theory is about midway between the experimentally obtained curves for the warp and weft direction.

Figure 16: Stress vs. Strain curve for the whole skin [7]

4.2 Unsymmetrical Skin Material Laminate

Due to the deviations between the computed and experimentally obtained Young's modulus of the skin in the previous section, a different approach is investigated in this section. The classical lamination theory is applied on the original unsymmetrical skin composition as can be seen in Figure 17. For an unsymmetrical composite a coupling between membrane extension and plate bending exists. Due to this, a calculation of the full ABD -matrix is necessary. Therefore, it is essential to know the shear modulus of the distinct layers. The shear modulus was not obtained experimentally by [7]. Due to the isotropic behavior of neoprene, the shear modulus is dependent on the Young's modulus and Poisson's ratio and can be calculated with Equation 17. The chafer layer and drop-stitch fabric behave differently dependent on the direction and are characterized as orthotropic [7]. Therefore, a calculation of the shear modulus with the known constants is not possible. To obtain an estimate of the shear modulus, it is first calculated for the warp and weft directions using the specific Young's modulus and Poisson's ratio and Equation 17. In the next step the average value for the shear modulus calculated in warp and weft directions is taken as an estimate for the orthotropic

shear modulus G_{12} . This procedure to obtain an estimate for the shear modulus of an orthotropic material is proposed in [38]. After calculating the shear modulus for each layer, the A , B and D matrices are obtained by assembling the distinct elasticity matrices of the distinct layers as given in Equation 22, 23 and 24. In the

Figure 17: Original skin composition

next step, the engineering constants are again calculated for membrane extension. The first step for this is calculating the inverse of the full ABD - Matrix. In the next step the upper left quadrant of the inverse is used for the A^{-1} matrix in Equation 22, 23 and 24 to calculate the engineering constants. The results for the calculation of the engineering constants can be seen in the following Figure 18.

Figure 18: Results for the Young's modulus of the original skin composition obtained by the classical lamination theory

As can be seen the Young's modulus obtained by the classical lamination theory is smaller in both the weft and warp directions compared to the symmetric

skin composition. This leads to a smaller deviation to the experimental result in weft direction but a larger deviation in warp direction.

4.3 Optimization of the Homogenization Result

In the last two sections, a high deviation of the results obtained by the classical lamination theory for the symmetric as well as the unsymmetrical composition of the skin was obtained, especially for the warp direction of the skin. In this section it is investigated if the high deviation for the calculation of the Young's modulus with the classical lamination theory could be explained by inaccuracies in the experimentally determined Young's moduli. This investigation seeks to determine if classical lamination theory can be applied in future research to estimate the skin stiffness from the constituent layer properties. The influence of the accuracy of the experimentally obtained engineering constants is investigated by varying the Young's moduli of the different layers by $\pm 10\%$. Similarly, the Poissons ratio ν_{12} is varied in a reasonable range complying with the stability conditions of orthotropic material.

The optimization tool Isight is used in connection with MATLAB to obtain the combination of layer properties which provide the best correlation with experimentally measured skin properties. The MATLAB code implements the lamination theory calculation for a given combination of layer properties. The optimization algorithm in Isight selects the combination of properties, within the user-specified range, for each iteration.

Using a range of $\pm 10\%$ for the Young's moduli of the different layers, it is possible to obtain the same value with the classical lamination theory as experimentally obtained. The results for the optimal combination of Young's moduli and Poisson's ratios for the different layers are shown in Figure 19. The values used for the engineering constants of the different layers that provide the best correlation

Figure 19: Results for the Young's modulus obtained by the classical lamination theory and an optimization process

with experimental results are summarized in Table 2. In the right column the deviation from the original value for the engineering constant is given. As can be seen, the deviation for the Young's modulus of the chafer layer in weft direction and for both directions of the drop-stitch layer are approximately 8 – 9%, just slightly within the specified range of values. The value for the chafer layer Young's modulus in warp direction, on the other hand, deviates by less than 2%.

Engineering constant	Value in optimization	Deviation
Chafer E_1 [psi]	31886.9	9.2%
Chafer E_2 [psi]	97809.8	1.5%
Chafer ν_{12}	0.579	19%
Drop-stitch E_1 [psi]	39135.65	8.9%
Drop-stitch E_2 [psi]	17267.3	8.4%
Drop-stitch ν_{12}	0.593	60%
Neoprene E [psi]	544.2	—

Table 2: Values used in the final optimization iteration

In Figure 20 the optimized chafer layer Young's moduli are compared to the experimentally obtained stress vs. strain curves [7]. As can be seen in the Figure, the value used for the optimized result in warp direction is within the error bounds

Figure 20: Stress vs. Strain curve for the Chafer layer [7]

for strains up to 0.5%. For higher strains, the experimental curves become nonlinear and diverges from the idealized linear stress-strain behavior. Comparing the stress strain behavior in weft direction, the optimization value is within the error bounds for strains up to 2.5%.

The stress strain curve of the drop-stitch fabric was obtained by [8]. No error bounds were determined in this procedure. Nevertheless, for strains up to 5%, the Young's moduli used for the warp and weft direction in the optimization are very close to the experimental stress-strain response of the material. As can be seen in Table 2 the values for the Poisson's ratios are above 0.5. Although this is not possible for isotropic materials, it can be used in orthotropic materials if the stability conditions given in equation 16 are fulfilled which is the case for these values.

In summary, it can be concluded that the optimization process is able to identify a combination of reasonable layer engineering constants that result in homogenized skin properties that correlate well with experimental results.

Figure 21: Stress vs. Strain curve for the drop-stitch layer [7]

4.4 Summary and Conclusion

In the previous sections the classical lamination theory is applied for different skin compositions. In Section 4.1, the classical lamination theory is used on a symmetric skin composition. The deviation between the experimentally obtained Young's modulus and the calculated Young's modulus is significant. Therefore, it is investigated, if the application of the classical lamination theory on the original unsymmetrical skin composition would give an improved result. However, the use of the classical lamination theory on the unsymmetrical skin composition also leads to significant deviations.

There are two possible explanations for the deviating results obtained using the classical lamination theory. The first reason is that the dry fabric could behave differently when tested on its own than when tested in a composite. When the dry fabric is built into the composite the fibers cannot move freely and the material can be simplified as a continuum. The dry fabric separated from the composite however does not behave like a continuum.

The second possible source for the deviating results are inaccuracies in the experimentally obtained engineering constants for the individual layers. In the

Section 4.3 an optimization approach is chosen to evaluate if the deviation of the results could be explained by the experimental inaccuracies. It is shown that it is possible to get a good result by using the classical lamination theory by varying the engineering constants of the different layers in a reasonable range.

Therefore, it is possible that inaccuracies are the reason for the deviation of the results. In future studies, it is recommended to reapply the classical lamination theory with more accurate experimental results for the engineering constants of the individual layers of the skin.

CHAPTER 5

Modeling of an Inflatable Drop-Stitch Panel with Finite Element Analysis

In the following section, the modeling process of an inflatable drop-stitch panel in a four-point bending test is discussed. Hereafter, the obtained model is validated analytically and with experimental results. After the validity of the finite element model has been verified, different effects leading to the specific behavior of inflatable drop-stitch panels are investigated.

5.1 Modeling Process

For the modeling process of an inflatable panel, it is important to have a precise knowledge of the geometry and characteristics of the panel and the four-point bending test in general. The characterization of the panel has been conducted in prior research and is summarized in section 3.6.1. The assembly and implementation of a four-point bending test is analyzed in section 3.1. The length between the support denoted as L in Figure 1 is 76 inches. The distance between the support and the loading pins labeled as a in Figure 1 is 26 inches. Due to the symmetry of a four-point bending test in axial as well as in transverse directions, it is possible to reduce the model for finite element analysis to one quarter of the whole structure. The computation time can be decreased significantly by modeling only a quarter of the structure because the number of elements and the associated degrees of freedoms are reduced.

The next step of the modeling process is defining the number of parts and the necessary characteristics that are needed for modeling the inflatable panel under four-point bending. To apply the four-point bending load, four support bars are needed: two bars that act as support under the panel and two bars that apply the

load on top of the panel. The bars need to have a high rigidity to ensure that only the panel is deforming and not the bars. Therefore, the bars are modeled as three-dimensional rigid body elements with four reference nodes (R3D4) in Abaqus. Due to the symmetry conditions, the number of bars can be reduced to two.

For the panel, two different parts are required. The first part is the skin of the panel. The skin makes up a large part of the overall model. Therefore, computationally inexpensive elements should be used to keep the computation time as low as possible. The skin is very thin in comparison to the dimension in length and width directions. For this reason, we can assume a plane stress state and it is possible to use shell elements without a loss of accuracy in the computation. The elements used are four node elements with reduced integration (S4R). A plane stress state is assumed for the skin material. Therefore, the material behavior "elastic lamina" is chosen. This material requires the Young's moduli and the Poisson's ratios for only the in-plane directions and the shear moduli for all three planes. The Young's moduli and Poisson's ratios for the skin are taken from Table 1. The shear moduli are not experimentally determined. To obtain an estimate, the isotropic shear moduli are obtained by taking the Young's modulus and Poisson's ratio for each direction and using Equation 17. The mean value of the two obtained shear moduli of 16,724 psi is used as an estimate for all shear moduli. Two sections are defined for the whole skin to account for the different thicknesses as explained in Section 3.6.1: one section for the sidewalls has a thickness of 0.11 inches and one section for the remaining skin has a thickness of 0.055 inches.

The second part required for the panel is the drop-stitch yarns. Since there are a high number of yarns, computationally inexpensive elements are preferred. The drop-stitch yarns can only absorb forces in axial tension. Therefore, truss elements are suitable for representing the behavior of the yarns. Three-dimensional truss

elements with 2 nodes (T3D2) and the restriction of tension only are used in Abaqus. The part representing the drop-stitch yarns is instantiated multiple times to obtain 16 drop-stitch yarns per inch. As mentioned in Section 3.6.1, the actual number of drop-stitch yarns is 24 yarns per inch. However, this difference does not influence the behavior of the panel, if the stiffness of the yarns is modified to account for the number of yarns. The stiffness of the drop-stitch yarns is given by the area, Young's modulus and length as shown in Equation 33. The stiffness of the drop-stitch yarns is determined experimentally by the thickness to pressurization behavior of the panel, which can be seen in Figure 22.

Figure 22: Change in thickness due to inflation [8]

To obtain the stiffness from the experimental data, only one square inch of the material is considered. The applied pressure can be transformed to a force by multiplying it with the area of the regarded skin. Due to this transformation, the units of the given slope in Figure 22 change to $\frac{in}{lb}$. By taking the reciprocal of the slope, we obtain the sum of the stiffnesses of all drop-stitch yarns in one square inch. By assuming 16 drop-stitch yarns per square inch, we can divide the obtained stiffness by this number to obtain the stiffness of a single drop-stitch yarn. In the next step, the stiffness, the Young's modulus, given in Table 1, and the length of

four inches can be inserted in Equation 33 to obtain the cross-sectional area of one drop-stitch yarn. The area is $8.4175e - 5 \text{ in}^2$. The material chosen for the drop-stitch yarns is elastic isotropic. The option "tension only" is applied to achieve zero stiffness for a compression of the drop-stitch yarns.

In Figure 23, the model with the assembled parts and the applied symmetry conditions are shown. Due to the usage of symmetry to reduce the model complexity, it is necessary to apply symmetry boundary conditions. The symmetry boundary conditions are applied in y- and z-directions as indicated in Figure 23. Due to the symmetry boundary conditions and the arrangement of the drop-stitch yarns, the cross-sectional area of the drop-stitch yarns located on the symmetry planes must be modified. The cross-sectional area of the drop-stitch yarns on the symmetry plane is changed to half of the initial cross-sectional area. The single drop-stitch yarn at the location where the two symmetry planes intersect has an effective area of one quarter of the originally defined area.

Figure 23: Symmetry boundary conditions four-point bending test

The process of the four-point bending test can be divided into two steps. The first step includes the inflation of the drop-stitch panel. The second step represents the loading. For both steps the interaction attribute "tie" is used to tie the drop-stitch yarns to the skin. Furthermore, all degrees of freedom for the supporting

bars under the panel are inhibited so that no movement of the supporting bars is possible. The inflation process is modeled by applying a pressure on the internal skin surface. For this model, no fluid elements are used to model the air inside of the panel. This simplification is possible because the volume change measured in the conducted four-point bending experiment was negligible[8]. In other cases, for example in a compression test of a drop-stitch panel, the volume change inside the panel would be too big to neglect the air inside the panel and the related pressure increase.

The loading condition of the second step is realized by applying a displacement boundary condition on the upper bars. A displacement in y-direction is carried out on the bar and induces bending of the panel. Because of the distance of the upper bar to the panel, the approximate deflection of the panel is smaller than the displacement of the upper bar. In the second step, two contact regions must be considered. The contacts occur between the supporting bar and the lower skin and between the moving upper bar and the upper skin of the drop-stitch panel. For both contacts, a hard contact formulation without friction is chosen. The rigid body bar is assigned as "master surface" and the associated skin part of the panel as "slave surface" as explained in Section 3.5.3. Due to the deformation of the panel due to the applied loading, the bars are in contact with part of the sidewall of the panel. For this reason, half of the sidewalls are added to the respective slave surfaces for the upper and lower skin parts to ensure that no penetration of the surfaces occurs. The contact area for the contact between the upper pin and the upper part of the panel can be seen in the following Figure 24.

Because of the contact formulation and the large displacements of the panel, nonlinearity is considered in both steps of the simulation. The applied loading and boundary conditions for both steps are illustrated in the following Figure 25.

Figure 24: Contact formulation between upper pin and panel

Figure 25: Loading and boundary condition for the whole assembly

5.2 Convergence Analysis

As explained in Section 3.5.1, it is important to analyze the convergence of the solution. Without a converged solution, the results are not meaningful.

For the investigation of the convergence, a representative node marked in red shown in Figure 26 is examined. The node is chosen because of its location between the upper bar and the support bar. In addition, the node is on the edge of the quarter model and therefore in the middle of the inflatable panel. Most of the investigations that are carried out in the following sections refer to data that is connected to the middle of the panel. The convergence of the solution is examined for pressures between 5 and 20 psi. To evaluate the convergence, the stresses in x- direction, denoted as S_{11} , and the stresses in z-direction, denoted as S_{22} , are

Figure 26: Exemplary node for convergence analysis

observed for five different mesh sizes. The largest mesh size considered is two inches and the smallest is 0.125 inches. In Figures 27, 28, 29 and 30, the stresses for the z-direction are considered for the different inflation pressures. The time step on the y-axis of the graph denotes the time of the simulation. For time steps between zero and one, the inflation of the panel takes place. For time steps between one and two, the bending load is applied.

As can be seen for all different pressures in the Figures 27, 28, 29 and 30, there is no visible difference in the stress for the inflation step. The subsequent bending step is more critical for the convergence of the solution. Bending exhibits a much more complex behavior of the skin material than the inflation.

The stresses for an inflation with 5 psi can be seen in Figure 27. Investigating the second step, it can be seen that there are high variations between the results for the stress in z-direction for the different mesh sizes. It can be seen that the difference between the solution for the stress from one mesh size to the larger mesh size gets smaller, the smaller the mesh size. The difference between the mesh size of 0.125 and 0.25 inches, leading to four times more elements, is 2.3 psi. The stress value 2.3 psi is only 1% compared to the range of stresses between -43 psi and 158 psi. Therefore, it can be concluded that the solution for an inflation pressure of 5 psi is converged within the accuracy needed for the scope of this research. Another mesh refinement would highly increase the computation time and would only lead

Figure 27: Convergence for stress S_{22} with inflation pressure of 5 psi

to an insignificantly more accurate result.

In Figure 28, the results for the stresses in z-direction are displayed for 10 psi. As can be seen, the simulation for a mesh size of two inches aborts after the

Figure 28: Convergence for stress S_{22} with inflation pressure of 10 psi

inflation step. All other simulations completed. Similar to the simulation for an inflation pressure of 5 psi, the differences for the resulting stress decrease with the mesh size. The difference between the stress for a mesh size of 0.125 and 0.25 inches is biggest at time step 1.8. The difference amounts to 6.3 psi, which is

about 2% compared to the range of stresses in this simulation between 0 and 318 psi. Therefore, it can be confirmed for the inflation pressure of 10 psi that a mesh of 0.125 inches leads to a sufficient accuracy.

The stress results for an inflation pressure of 15 psi can be seen in Figure 29. As can be seen, the simulation for a 2 and 1 inch mesh size aborts before the

Figure 29: Convergence for stress S_{22} with inflation pressure of 15 psi

completion of the simulation. Comparing the stress results for a 0.25 and 0.125 inches mesh size, it can be seen that the highest difference of 5.5 psi occurs at time step 2. Therefore, the difference is about 1% in relation to the range from 0 to 484 psi. A sufficient accuracy can therefore be confirmed also for an inflation pressure of 15 psi.

The last Figure 30 shows the stresses for an inflation pressure of 20 psi. As can be seen, only the simulations for the mesh sizes 2, 0.25 and 0.125 inches completed. The largest difference between the smallest mesh sizes can again be seen for the time step 2. The difference is 6.2 psi. Comparing this to the range of 0 to 656 psi, the difference only amounts to 1%. Therefore, we can confirm that a mesh size of 0.125 inches is also sufficient for 20 psi inflation pressure.

In summary, we can conclude that the mesh size of 0.125 inches is sufficient

Figure 30: Convergence for stress S_{22} with inflation pressure of 20 psi

to display the stresses in z-direction at the investigated node. The stresses in the transverse x-direction have also been investigated. The results show the same tendency and even better convergence. To avoid repetition, the results are not discussed in this section. If required, the results can be viewed in the appendix B.

Due to the small differences in the stresses for changing the mesh size from 0.25 to 0.125 inches for the investigated node, it is assumed that this mesh size is also sufficient for the whole structure.

5.3 Pressure dependant Force vs. Displacement Behavior

5.3.1 Behavior of Whole Panel

In this section, the force displacement behavior of the panel under a four-point bending load is analyzed. The results obtained by the model created in the prior section 5.1 are compared with existing experimental results. The behavior is investigated for inflation pressures of 5 psi, 10 psi, 15 psi and 20 psi.

To analyze the force displacement behavior, the displacement in the finite element model is computed at the middle of the panel. Both the top and bottom of the panel are considered. The force needed to deflect the panel is equal to the reaction force in the bars that are pushing the panel down. Therefore, the force

needed to investigate the force displacement behavior is obtained by outputting the reaction force on the reference node of the upper pin. Due to the fact that only one quarter of the model is simulated, the reaction force has to be multiplied by four to obtain the overall force applied on the panel. The code that is used to extract the force and displacement and the code evaluating the output can be found in appendix C.

In Figure 31, the force displacement behavior for an inflation pressure of 5 psi is displayed.

Figure 31: Force displacement behavior for experiment and simulation (5psi)

As can be seen, the displacement of the top and bottom in the middle of the panel does not differ. The initial slope for the simulation of 71 lbf/in is slightly higher than for the experimental result of 77 lbf/in. The experimental and simulation slope differ by about 8%. Furthermore, the slope of the experimental result decreases after a deflection of about 1 inch. The slope for the simulation starts to decrease later than in the experiment. It starts at about 2.8 inches.

In Figure 32, the force displacement curve for an inflation pressure of 10 psi is presented. The experimental and simulation slope are similar for displacements smaller than 1 inch. The difference in initial slope amounts to 8 lbf/in which is

Figure 32: Force displacement behavior for experiment and simulation (10psi)

about 9%. For displacements larger than 1 inch, the simulation result is stiffer than the experiment. For displacements larger than 4 inches, the difference between experiment and simulation exceeds 100 lbf. A stagnating slope is observed for displacements higher than 4 inches for the experiment and higher than 4.5 inches for the simulation.

In Figure 33, the force displacement behavior for an inflation pressure of 15 psi is presented. It can be observed that initially the force needed for a deflection

Figure 33: Force displacement behavior for experiment and simulation (15psi)

is smaller for the simulation than for the experiment. The initial slope in the simulation is smaller compared to the experiment by 12.5%. After a displacement of about 1.8 inches, this characterization changes. The slope of the curve obtained in the simulation is higher than in the experiment. A decrease of slope can be detected for the experimental results after a deflection of about 3.5 inches. For the simulation results, a decrease cannot be detected in the depicted range of deflections.

In Figure 34, the force displacement behavior for the last inflation pressure of 20 psi is investigated. As can be seen, the initial slope of experiment and

Figure 34: Force displacement behavior for experiment and simulation (20psi)

simulation already differs by about 20%. The stiffness of the panel detected in the experimental four-point bending test is higher than the stiffness obtained for the simulation. For the simulation, the curve in the investigated range has a linear behavior. In the experiment, however, there is a nonlinear decrease of the slope at a displacement of about 4 inches.

In Figure 35, all obtained force displacement curves for experiment and simulation are displayed for an easier comparison.

As explained above and as can be seen here, the simulation cannot fully rep-

Figure 35: Force displacement behavior for different inflation pressures

resent the actual behavior of the panel. While the initial slope for the experiment varies between 71 and 120 lbf/in, which is a change of about 40%, the slopes in the simulation vary from 77 to 96 lbf/in which is about 20%. Thus, the slope changes about twice as much in the experiment as in the simulation. Nevertheless, the simulation curves are, despite their different slopes, in the same range as the experimentally obtained curves and the pressure dependence can be represented up to a certain point. Therefore, an approximate representation of the actual behavior by the simulation can be confirmed. The largest difference between the experiment and simulation are the more significant pressure dependence and the earlier occurring decreases in slope for the experiment.

5.3.2 Behavior of Drop-Stitch Yarns

In the following section, the simulation results are examined more closely to investigate the loss of tension in the drop-stitch yarns and the resulting effect on the decrease in slope of the force displacement curves. To analyze this effect, the difference in the distance between the upper skin and bottom skin under the loading pin is observed. The distance is examined by detecting the distance between the

nodes in the middle under the pin. The nodes can be seen in Figure 36 marked in red. To investigate the loss of tension of the drop-stitch yarns, only the second step

Figure 36: Nodes under loading pin used to investigate the loss of tension

is investigated in detail. In the first step, the pressurization takes place. After this step, the drop-stitch yarns for the four inflation pressures 5 psi, 10 psi, 15 psi, 20 psi have different lengths due to the applied pressure. For a better comparison between the different inflation pressures, the difference in length due to the applied force is observed instead of the specific length. The simulation results for the change in length due to applied force can be seen in Figure 37. There are two points of interest in these curves. The first point is where locally the drop yarns lose tension and less force is needed for the loading pin to push the panel further down. The point is called "Loss of tension" in Figure 37. At this point, the loading pin is still supported by the adjacent drop-stitch yarns, which have not lost tension yet. Therefore, force still must be added to increase the displacement. The second important point is the point where most drop-stitch yarns under the loading pin lost tension and no additional force is needed to push the panel further down. This point is called "Negative slope" in Figure 37 because it leads to a negative slope

due to the decrease in needed force and the increase in the difference between the top and bottom skins.

Figure 37: Loss of tension in the drop-stitch yarns in the simulation

In general, up to a force of about 180 lbf, the drop-stitch yarn length decreases about the same amount for all different inflation pressures. The loss of tension occurs for higher forces when the inflation pressure increases. In addition, the difference between top and bottom is higher before a loss of tension occurs when the inflation pressure is higher. The point of negative slope, on the contrary, happens for lower displacement differences of the top and bottom skins as the pressure increases. It can be seen that the point of loss of tension and the point of negative slope move closer in terms of the displacement difference of the top and bottom skins for higher pressures. The exact differences between the top and bottom of the panel and the force for the point of loss of tension and negative slope can be found in Table 3.

Representative for all pressure cases, the stresses in the drop-stitch yarns are examined more closely in terms of the two points of interest for an inflation pressure of 5 psi. In Figure 38, the stresses in the drop-stitch yarns at a time step of 1.2 can be seen in the region of the upper loading pin. At the time step 1.2, the loading

Point of interest	5 psi	10 psi	15 psi	20 psi
Loss of Tension Force [lbf]	208.9	316.6	434.1	548.8
Loss of Tension Difference [in]	0.0345	0.063	0.0997	0.131
Loss of Tension time step [-]	1.51	1.63	1.78	1.92
Negative slope Force [lbf]	232.6	352.6	467.6	-
Negative slope Difference [in]	0.367	0.21	0.194	-
Negative slope time step [-]	1.7	1.76	1.91	-

Table 3: Values for points of interest

pins are not yet in contact with the panel. Therefore, there is zero force. The stresses in the panel are induced by the inflation pressure only. As can be seen, the drop-stitch yarns all have the same stresses except the yarns that can be seen in the back. These drop-stitch yarns have different stresses because they are at the sidewalls of the panel. This effects the drop-stitch yarns because they do not have adjacent yarns on one side. Therefore, the tension is distributed differently from the panel skin onto the drop-stitch yarns. In Figure 39, the stresses in the panel

Figure 38: Stress 5 psi inflation pressure step time 1.2

at time step 1.5 are displayed. This time step is shortly before the loss of tension point (see Table 3). As can be seen, the loading pin is now in contact with the panel. Due to the changed color scale, it appears as the stresses have changed all

over the panel. But the main part of the panel is only exposed to small changes in stress. Significant changes only take place in the area of contact between the panel and the pin. Especially at the back of the picture, near the sidewall of the panel, a large region of drop-stitch yarns already lost tension. But only two rows of drop-stitch yarns have zero stresses all the way through the panel. Due to the distance of the drop-stitch yarns of 0.25 inches, there is a cross-section region of close to 1 inch in which the drop-stitch yarns have lost their tension and no support for the loading pin is given by them. The loading pin has a diameter of 3 inches. Therefore, the loading pin is still supported by the adjacent drop-stitch yarns.

Figure 39: Stress 5 psi inflation pressure step time 1.5

In Figure 40, the stresses in the drop-stitch yarns are shown for the time step 1.7, which is the point of negative slope for an inflation pressure of 5 psi (see Table 3). Again, the color scale is deceptive. The change of stress mainly occurs in the region of contact between the loading pin and the panel. Compared to the prior Figure 39 at time step 1.5, the difference in the stresses of the drop-stitch yarns between the front of the panel and the back (at the sidewall) is smaller. There are now nine rows of drop-stitch yarns which have zero stress through the panel.

This is equivalent to a cross-section of close to 2.75 inches. Due to the fact that the loading pin does not completely penetrate the panel, it is assumed that the cross-sectional length of the loading pin that is contact with the panel does not exceed 2.75 inches. Therefore, it can be observed that from this time step on, there are no drop-stitch yarns beneath the loading pin that could add support. Thus, no additional force is needed after this point to push the panel further down.

Figure 40: Stress 5 psi inflation pressure step time 1.7

In Figure 41, the points of interest are related to the force displacement curves of the whole panel discussed above (see Figure 35). As can be seen in this figure, the characteristic changes in the curves are related to the loss of tension and negative slope point obtained in Figure 37. Before the loss of tension in the drop-stitch yarns, the force displacement behavior of the panel is almost linear. Between the loss of tension point and the negative slope point, a substantial decrease of slope can be observed. After the negative slope point is reached for the nodes directly under the loading pin, a negative slope is also detected for the force displacement curve of the whole panel.

Due to the relation of the observed points of interest "loss of tension" and

Figure 41: Loss of tension points in force displacement behavior

"negative slope" for the drop-stitch yarns to the overall force displacement curve of the panel, it is clear that the loss of tension of the drop-stitch yarns has a significant effect on the overall panel response.

To further investigate this effect, the nonlinear effects of the drop-stitch yarns are neglected by allowing the drop-stitch yarns to support compressive loads. The force displacement behavior of the panel including the compression of the drop-stitch yarns is displayed in Figure 42. The initial slope for the panel differs only

Figure 42: Force-Displacement behavior including compression of drop-stitch yarns

slightly from the slope that was obtained for no compression of the drop-stitch yarns. The slope including compression of the drop-stitch yarns is higher by 1 lbf/in for all inflation pressures. Significant differences in the behavior of these curves can be seen after the loss of tension points shown in Figure 41. After this point, the differences between the slopes for the distinct inflation pressures are much smaller than observed when compression is excluded.

The shape of the force displacement curves is characterized first by a linear curve followed by a rapid decrease in slope, which is observed for both the simulation and experiment in Figure 35. This curve shape, however, cannot be observed for the curves in Figure 42. It is therefore confirmed that the loss of tension of the drop-stitch yarns is significant for the characteristic behavior of inflatable drop-stitch panels.

5.3.3 Nonlinear vs. Linear Behavior

It is apparent that the pressure dependent stiffness of the inflatable drop-stitch panel is associated with nonlinearities in the finite element model. To investigate this hypothesis, a new modified model that excludes nonlinear effects is developed. A new model has to be created because it is not possible to use a linear analysis on the model described in section 5.1, due to the contact that occurs between the panel and the loading pin (see contact properties 3.5.3). Therefore, the new model cannot include contacts. Instead, the load is applied through displacement and pressure boundary conditions. The panel itself is not changed from the previously obtained model. Instead of the rigid supporting pin contacting the panel, a boundary condition that allows no displacement in y-direction and no rotation about the z axis is applied at the same location as the supporting pin. Instead of the loading pin, a pressure is applied on an area of one inch by ten inches at the point where the loading pin would contact the panel. In addition, compression is

allowed for the drop-stitch yarns. The resulting force displacement behavior for inflation pressures of 0 to 20 psi can be seen in Figure 43. As can be seen by

Figure 43: Linear model of inflated drop-stitch panel

the equal slope of the curves for all pressures, no pressure dependence is observed for a linear model. All pressures lead to the same slope of the force displacement curve as the initial slope for 5 psi with nonlinear effects, obtained in Section 5.3. Therefore, the hypothesis that nonlinearities lead to the pressure dependent behavior is confirmed. Nonlinear effects in the model include, for example, the loss of tension in the drop-stitch yarns. The loss of tension in the drop-stitch yarns has been investigated in Section 5.3.2. It could be determined that the tension only property does not affect the initial slope. However, the loss of tension in the drop-stitch yarns increases the pressure dependent behavior in the later slope of the force displacement curve. Not understood for now is, which nonlinearities lead to the change in initial slope. Due to no appearance of kinking at low loads it is assumed that the contact interaction leads to the pressure dependence in the initial force displacement slope. This hypothesis must be investigated in further research.

5.4 Pressure Dependent Beam Bending Deflection

In this section, the bending shape of the inflated panel is analyzed. The investigation is carried out by observing the shape of about a 7 inch deflected panel. The shapes are compared for experimental and simulation data for pressures between 5 and 20 psi. The experimental data was obtained in a four-point bending test. Images of the test have been evaluated in prior research [7] to extract the bending shape.

In the following Figure 44, the bending shape for a panel with 5 psi inflation pressure is shown. As can be seen, there is a significant difference between the slope for the part between the loading points from -12 to 12 inches. While the sides of the panel are almost a straight diagonal line, the middle part is nearly a horizontal line. In Figure 45, the bending shape of a panel with 10 psi inflation pressure is

Figure 44: Experiment and simulation bending for 5 psi inflation pressure

shown. As can be seen, the simulation on average fits the points obtained in the experiment. The kink between the side of the panel and the middle of the panel is smaller for 10 psi than for the prior figure with 5 psi inflation pressure. The smaller kink is connected to the higher curvature of the part in the middle of the panel.

Figure 45: Experiment and simulation bending for 10 psi inflation pressure

In Figures 46 and 47, where the bending shape for 15 psi and 20 psi is investigated, a continuation of the tendency described above can be observed. The higher the pressure, the more curvature the middle part of the panel exhibits. Due to the higher curvature, a decrease in the kink can be detected for increasing inflation pressures.

Figure 46: Experiment and simulation bending for 15 psi inflation pressure

As discussed above, for all bending shapes for the different pressures, a change in slope or kink can be observed at the point where the loading pins contact the

Figure 47: Experiment and simulation bending for 20 psi inflation pressure

panel. A kink does not appear for the bending of a conventional beam and is therefore not included in classical Timoshenko or Bernoulli beam theories [18]. In [7], an analytical model is derived to predict the bending shape of the panel. To fit the analytical model to the experimental data including the kinking effect, the analytical model is optimized by a so-called "kinking angle". The kinking angle is the difference between the angle from the middle part of the panel to the x-axis and the angle of the side part to the x-axis, determined by the slope at the observed point. It is explained in following Figure 48. In [7], the kinking angle is obtained by optimizing the fit between analytical model and experiment.

Due to the good fit of the experimental data to the simulation for a deflection of about 7 inches, it is apparent that the bending shape of the finite element model can represent the bending shape of the original model in general. In order to have more confidence in this conclusion, future research should include the comparison of experiment and simulation at different deflection levels.

To validate the usage of the kinking angle in [7] and to compare the behavior for different pressures, the kinking angle is computed for the simulation results at different pressures. To obtain the kinking angle, polynomial fits in MATLAB are

Figure 48: Explanation of the obtained kinking angle

used to map the bending shape of the panel. One polynomial is used for the middle part of the panel from -11.5 inches to 11.5 inches and one polynomial is used for each side of the panel. The polynomial fits are applied for inflation pressures of 5, 10, 15, 20, 25 and 30 psi and can be seen in Figure 49. Only half of the shape of the panel is displayed because of the symmetric bending of the panel.

In the next step, the inverse of the tangent of the slopes for both polynomials, in the middle and at the side of the panel at 11.5 inches, is computed. The kinking angle is then obtained by taking the difference of these two angles as shown in Figure 48. The angles obtained from the simulation results are displayed together with the optimization values for the kinking angle obtained by [7] in Figure 50. In [7], the kinking angles are obtained for inflation pressures of 5, 10, 15 and 20 psi. As can be seen in Figure 50, the kinking angle for the simulation and the experiment are very close. For pressures between 5 and 15 psi, the kinking angle obtained from the simulation is slightly smaller than the value in [7]. For 20 psi, on the contrary, the value from the simulation is slightly above the value in [7]. For both the kinking angle obtained in the simulation and obtained by [7], it is clear that the value of the kinking angle decreases with increasing pressure. In [7],

Figure 49: Polynomial fits for inflation pressures 5 to 30 psi

the tendency of the decreasing kinking angle is described as linear. For inflation pressures of 5 to 20 psi, the decrease could also be described as linear for the simulation results. But due to the fact that inflation pressures up to 30 psi are investigated in the simulation, it can be seen that the decrease is more exponential than linear. The value of the kinking angle was predicted to be 0° for 25 psi in [7]. In the simulation, the kinking angle approaches zero for high pressures. To obtain a value of zero for the kinking angle is theoretical only possible for pressures

approaching infinity. Practically the kinking angle is probably not detectable for pressures much higher than investigated in Figure 50.

Figure 50: Comparison of kinking angle in simulation and experiment

In summary, it is clear that a kink in the deformed panel shape occurs at the loading point. The kinking angle decreases with increasing pressure. In general, the predicted kinking [7] correlates well with the simulation results. While [7] predicts zero kinking for inflation pressures higher than 25 psi, the simulation shows a small amount of kinking observed at higher pressures.

5.5 Analytical Model for the Stress Distribution

In the following two sections, an analytical model for the axial and transverse stress distribution in the inflatable drop-stitch panel is derived. The derivation of the stress distribution contributes to a better understanding of the panel behavior and can be used as a tool for building optimized panels in the future.

In the first section, the stresses due to inflation are investigated by using the thin pressure vessel equations introduced in section 3.2 and including the cross-sectional increase.

In the second section, the beam bending stresses introduced in section 3.1 are

superposed on the inflation stresses as described by [6] and depicted in Figure 7.

MATLAB is used to compute and display the stresses. The obtained stresses are compared to the stresses obtained in the simulation for the top and bottom skin in transverse and axial directions in the middle of the panel. The MATLAB script used to calculate the stresses and the Python script which extracts the stresses from the simulation, can be found in appendix C.2.

5.5.1 Pressurization Step

To calculate the stresses due to inflation, thin pressure vessel theory is used. The theory states that there is an equilibrium between the inflation pressure and the stresses in the skin. To calculate the stresses by the equilibrium equation, the radius r , the thickness t , the height h and the length l or width w shown in Figure 51a are used. Due to the equal cross-sectional shape of the panel in transverse and in axial directions, both directions are summarized in the figure. The only variable that changes in the cross section is the length or respectively the width denoted as l/w in the figure. For the stress in the axial direction, the width dimension is needed and for the transverse stress, the length dimension. The first step for the equilibrium equation is calculating the cross-sectional area of the inner panel ($A1$) and the skin ($A2$) as displayed in Figure 51b.

Figure 51: Variables needed to characterize panel

The inner area of the skin can be simply calculated by adding the area of a rectangle and a circle. The inner area $A1$ is given in equations 36 and 37 for the transverse and axial directions.

$$A1_{trans} = h l + \pi r^2 \quad (36)$$

$$A1_{axial} = w l + \pi r^2 \quad (37)$$

The area of the skin $A2$ can be again calculated in two parts. The first part is taking the length or respectively width of the panel twice and multiplying it with the thickness. The second part is the sidewall of the panel. To calculate the area of the sidewall, the area of a circle with the outer radius is calculated and then the area of a circle with the inner radius is subtracted. The equations for the cross-sectional area of the skin are given in equations 38 and 39.

$$A2_{trans} = 2 l t + \pi ((r + 2 t)^2 - r^2) \quad (38)$$

$$A2_{axial} = 2 w t + \pi ((r + 2 t)^2 - r^2) \quad (39)$$

Using the force equilibrium principle of equations 4 and 5, we can derive the stress in the skin due to inflation using equation 40.

$$p A1 = \sigma A2 \quad (40)$$

The axial and transverse stresses due to inflation can be found respectively in equations 41 and 42.

$$\sigma_{trans} = \frac{p A1_{trans}}{A2_{trans}} = \frac{p (h l + \pi r^2)}{2 l t + \pi ((r + 2 t)^2 - r^2)} \quad (41)$$

$$\sigma_{axial} = \frac{p A1_{axial}}{A2_{axial}} = \frac{p (w l + \pi r^2)}{2 w t + \pi ((r + 2 t)^2 - r^2)} \quad (42)$$

Equations 41 and 42 give us the average axial or transverse stresses in the panel. Due to the double thickness of the sidewalls, the stresses are half of the stresses in

the top and bottom skins of the panel. This effect can also be seen in the simulation. In Figure 52, the transverse stress distribution due to an inflation pressure of 5 psi is displayed. While the stress in the middle of the panel is about 185 psi, the stress at the sidewall is approximately 91 psi and therefore approximately half of the stress in the middle of the panel. Due to the implementation of the double sidewall thickness, a more complex stress distribution is reached.

Figure 52: Transverse stresses due to inflation pressure of 5 psi

In the following investigation of the inflation pressures, the scope is therefore restricted to the examination of the upper and lower skin excluding the sidewalls. Because the equations only give the average value of the inflation stress, the sidewalls falsify the result if only the top and bottom of the panel are investigated. To take account for this, equations 41 and 42 are modified. The area of the sidewalls is divided by two as can be seen in the following equations 43 and 44.

$$\sigma_{trans} = \frac{p(hl + \pi r^2)}{2lt + \pi/2((r + 2t)^2 - r^2)} \quad (43)$$

$$\sigma_{axial} = \frac{p(hw + \pi r^2)}{2wt + \pi/2((r + 2t)^2 - r^2)} \quad (44)$$

Furthermore, it is desired to include the expansion of the panel. During the inflation process, the panel expands and the inner area, A_1 , and skin area, A_2 , change. The area in equations 43 and 44 must be modified to maintain equilibrium. The expansion in the panel is included in the analytical model, by the change of the variables r , w , l , and h .

The change in height is calculated using the experimentally measured slope in 22 that relates the pressure to the change in height of the panel in equation 45.

$$\Delta h = p \cdot 6.6 \cdot 10^{-3} \frac{in}{psi} \quad (45)$$

To calculate the change in radius, width and length, the compliance matrix of the orthotropic skin material is used. The compliance matrix is the inverse of the stiffness matrix given in equation 20. It is assumed that the effects due to the shearing can be neglected for this case. Therefore, the 4x4 compliance matrix of the orthotropic skin is given in the following equation 46. The values of the engineering constants can be found in Table 1.

$$[S] = \begin{bmatrix} \frac{1}{E_{2-warp}} & \frac{-\nu_{21}}{E_{2-warp}} \\ \frac{-\nu_{21}}{E_{2-warp}} & \frac{1}{E_{1-wefl}} \end{bmatrix} \quad (46)$$

The strain in the material can be calculated by equation 47 with σ_{trans} and σ_{axial} from equations 43 and 44.

$$\begin{Bmatrix} \epsilon_{axial} \\ \epsilon_{trans} \end{Bmatrix} = [S] \begin{Bmatrix} \sigma_{axial} \\ \sigma_{trans} \end{Bmatrix} \quad (47)$$

Strain is the percentage change in length. Therefore, the total change in length and width directions can be directly obtained by multiplying the strain with the initial value as shown in equations 48 and 49.

$$\Delta w = \epsilon_{trans} w \quad (48)$$

$$\Delta l = \epsilon_{axial} l \quad (49)$$

For the sidewalls, a similar approach is pursued. However, due to the curvature of the sidewalls, the transverse stress or hoop stress in the sidewalls is assumed to be different than in the upper and lower parts of the skin. The same stress as for the hoop stress of a cylindrical pressure vessel is used instead. The hoop stress

for a cylindrical pressure vessel is the same as the stress for a spherical pressure vessel and is given in equation 7 for a single wall thickness. The axial stress in the sidewalls is assumed to be half of the stress at the top or bottom of the panel due to the double thickness. The stress vector for the sidewall thus results in equation 50.

$$\begin{Bmatrix} \sigma_{axial} \\ \sigma_{trans} \end{Bmatrix} = \begin{Bmatrix} \sigma_{axial}/2 \\ \frac{pr}{4t} \end{Bmatrix} \quad (50)$$

Inserting the stress vector of the sidewall in equation 47, the strains for the sidewall can be obtained. The change in radius can be calculated with ϵ_{trans} as can be seen in the following equation 51.

$$\Delta r = \epsilon_{trans(Sidewall)} r \quad (51)$$

After calculating the changes in the dimensions, they are added to the original values and then inserted in equations 43 and 44 again. This process is repeated until the error between the new calculated transverse and axial stress and the values for the stresses from the prior iteration is smaller than 10^{-6} .

To validate the calculated change in length, width, height and radius, the cross-sectional perimeter and inner area $A1$ is compared to the simulation. The calculated values and obtained values from the simulation can be found in Tables 4 and 5. As can be seen for the transverse cross-section in Table 4, the area and the perimeter in the simulation and the calculated area are very close. The difference between the simulation values and the analytically obtained values does not exceed 0.3%. The analytically obtained values for higher pressures are slightly smaller than the values in the simulation. For the area and perimeter in the axial direction, the error between the simulation and analytically calculated values does not exceed 0.4%. The analytically obtained values exceed the change obtained by the simulation by a small amount for higher pressures. The difference between the simulation and analytical model increases with pressure for both the axial and

Pressure	Simulation		Analytical Calculation	
	Area	Perimeter	Area	Perimeter
0 psi	92.56	52.56	92.57	52.57
5 psi	93.91	52.9	93.9	52.9
10 psi	95.33	53.25	95.26	53.24
15 psi	96.81	53.62	96.66	53.6
20 psi	98.37	54	98.09	53.96

Table 4: Cross-sectional area and perimeter transverse direction

Pressure	Simulation		Analytical Calculation	
	Area	Perimeter	Area	Perimeter
0 psi	396.56	204.56	396.57	204.57
5 psi	399.64	204.48	399.95	204.67
10 psi	402.84	204.39	403.35	204.78
15 psi	406.15	204.29	406.74	204.9
20 psi	409.6	204.2	410.15	205.01

Table 5: Cross-sectional area and perimeter axial direction

transverse directions. However, the differences are very small. Therefore, good agreement between the simulation and analytical changes of the panel shape due to inflation is observed.

Using the values for the width, length, height and radius after expansion of the panel, the stresses due to inflation are obtained by equations 43 and 44. The resulting stresses for inflation pressures of 5 psi, 10 psi, 15 psi and 20 psi in transverse and axial directions are displayed in Figures 53, 54, 55 and 56. Due to the symmetry of the panel, the stresses are the same for the top and bottom of the panel. Therefore, only the stresses at the top are shown. The stresses obtained with the analytical model are compared with the stresses in the middle of the panel in the simulation. As mentioned at the beginning of the section, the analytically calculated inflation stress is the average stress. Therefore, the mean value for the simulation is displayed in addition to the actual stress for a better comparison.

In Figure 53, the inflation stresses for 5 psi are displayed. Based on this

figure, the general stress distribution in the panel can be interpreted as follows. Observing the stresses at the axial positions 0 and 96 inches, extreme changes

Figure 53: Inflation Stresses 5 psi

in the stress occur for both the transverse and axial directions. These extreme changes are associated with the transition from a single to a double skin thickness. The same effect can also be observed for the other investigated inflation pressures. Neglecting the effects at the transition to the sidewalls, the transverse stress is increasing from 0 to about 20 inches. From about 20 inches to the middle of the panel at 48 inches axial position, only small changes in stress can be observed. Due to the symmetry of the panel, the same behavior is mirrored for the other half from 48 to 96 inches. The axial stress in the panel decreases up to the approximate point of the supporting pin and then increases to reach the highest stress point in the middle of the panel. The stress distribution is mirrored at the middle of the panel due to the symmetry. The same tendency for the axial and transverse behavior with a different amplitude of the values can be seen for the other investigated inflation pressures. Comparing the mean value of the axial and transverse stresses to the analytically obtained values, only minor differences smaller than 1% are observed for an inflation pressure of 5 psi.

Figure 54 shows the stresses due to an inflation pressure of 10 psi. As already mentioned, the same tendencies for the axial and transverse stresses can be detected for 10 psi as for 5 psi. However, the differences between the average value of the simulation and the theoretical value are slightly higher than for an inflation pressure of 5 psi.

Figure 54: Inflation Stresses 10 psi

This increase is also observed for inflation pressures of 15 psi in Figure 55 and 20 psi in Figure 56.

Figure 55: Inflation Stresses 15 psi

While the difference amounts to about 1% for 5 psi, it increases to about 1.3% for 10 psi, 2% for 15 psi and about 3% for 20 psi. These slight increases in error can partially be explained by differences in the skin expansion of the panel.

Figure 56: Inflation Stresses 20 psi

In summary, it can be concluded that the theoretical calculation provides reliable estimates for the transverse and axial stresses due to inflation.

5.5.2 Loading Step

In the second step of the finite element analysis, the four-point bending load is applied. The axial stresses due to the four-point bending load are superposed to the inflation stresses as shown in Figure 7.

To calculate the bending stress, equation 1 is used. The maximum bending stress occurring in the panel can be obtained by equation 52, using the total distance between supporting pin and loading pin, denoted as a in Figure 1. This distance is 26 inches for the bending test investigated in this research. The force F in equation 52 is the force applied by one loading pin and therefore half the total force applied on the panel.

$$\sigma_{bmax} = \frac{F a \frac{h}{2}}{I} \quad (52)$$

The moment of inertia I for the specific geometry of this panel is derived in [7]. The equation derived by [7] is modified to the notation used in this research, shown in Figure 51a, and given in equation 53. The length, radius, width and height after the inflation, obtained in Section 5.5.1, are used for the calculation of the moment of inertia.

$$I = 2 \left[\frac{w t^3}{12} + w t \left(\frac{h}{2} + \frac{t}{2} \right)^2 \right] + \frac{\pi}{4} \left[(r + 2t)^4 - r^4 \right] \quad (53)$$

The superposing of the bending load differs for the top and bottom and for the different regions of the panel. Therefore, the superposition of the bending load is illustrated in Figures 57 and 58. As can be seen in these figures, there are three different regions in the panel.

Figure 57: Superposed bending stresses at the top of the panel

The first region, to the left of the first supporting pin and to the right of the second supporting pin, is not exposed to the bending load. Only the inflation pressure contributes to stresses in this region.

In the second region, in the middle of the panel, the maximum bending stress given in equation 52 is superposed on the inflation stress. For the top of the panel, the maximum bending stress is a compressive stress and is subtracted from the

Figure 58: Superposed bending stresses at the bottom of the panel

inflation stress. On the bottom of the panel, on the other hand, the bending stress is tensile and is added to the inflation stress.

The remainder of the panel makes up the third region. In the third region, bending stresses are superposed, but the bending stresses are dependent on the distance from the supporting pin to the loading pin. The bending stress at the specific location is calculated by using the same equation as for the maximum stress, but using the distance from the supporting pin to the observed point instead of a . Therefore, the bending stress directly at the supporting pin is zero and the bending stress at the loading pin is equal to the maximum bending stress. The bending stress obtained for the specific location of the panel is then superposed again to the inflation stresses. At the top of the panel, the bending stress is subtracted, and at the bottom it is added.

Thus, for the top of the panel, the middle part is exposed to the smallest axial stresses and for the bottom, it is exposed to the highest axial stresses.

To obtain the transverse stresses, equation 47 is used. Due to the larger dimensions in axial direction, it is assumed that the transverse strain is restricted and therefore remains constant during the four-point loading. Thus, the transverse strain effected by the inflation pressure is used to obtain the transverse stress. For

the axial stress, the respective axial stress at the specific location is imposed.

Rearranging of equation 47 and inserting the compliance matrix [S] gives the result for the transverse stress σ_{trans} for the top and bottom of the panel. These stresses are given by equations 54 and 55.

$$\sigma_{trans}(top) = \epsilon_{trans} E_{1-waft} + \frac{\nu_{21} E_{1-waft}}{E_{2-warp}} \sigma_{axial}(top) \quad (54)$$

$$\sigma_{trans}(bottom) = \epsilon_{trans} E_{1-waft} + \frac{\nu_{21} E_{1-waft}}{E_{2-warp}} \sigma_{axial}(bottom) \quad (55)$$

To compare the theoretical values for the transverse and axial stresses to the simulation, the reaction force from the simulation at time step 1.4 is used to calculate the theoretical bending stress. The time step 1.4 is chosen because at this time the loading pin has reached the panel, but no kinking occurs yet. Kinking would complicate the stress distribution of the panel.

In the following eight figures, the transverse and axial stresses for the top and bottom skins of the panel for the inflation pressures 5, 10, 15 and 20 psi are displayed for both the finite element simulation and the analytical theory. For the simulation, the middle of the panel is chosen to extract the stress values. The four peaks that can be seen in each simulation stress distribution of the panel are associated with the contact stresses at the support and loading pins.

To investigate the accuracy of the theory, the middle region of the panel, with the maximum (bottom) or respective minimum (top) bending stress, is further observed for each of the eight cases. To compare the simulation with the theory, the local stress concentrations at the loading pins are neglected.

In Figures 59 and 60, the transverse and axial stresses for 5 psi inflation pressure are shown for the total force of 144.3 lbf.

In the middle region, at the top of the panel (Figure 59), the predicted transverse stress by the theory is 115.2 psi and the axial stress is -151.2 psi. Neglecting

Figure 59: Axial and transverse stresses at panel top skin for 5 psi

Figure 60: Axial and transverse stresses at panel bottom skin for 5 psi

the stress concentrations at the loading points, an average difference of 19.2% can be observed for the axial stress and -14.2% for the transverse stress . For the middle region of the bottom skin (Figure 60), the theory gives a transverse stress of 239.5 psi and axial stress of 472.2 psi. This leads to a difference from the simulation of -2% for the axial stress and 0.5% for the transverse stress. In Figures 61 and 62, the transverse and axial stresses for 10 psi inflation pressure for the total force of 155.3 lbf are shown.

Figure 61: Axial and transverse stresses at panel top skin for 10 psi

Figure 62: Axial and transverse stresses at panel bottom skin for 10 psi

For the top skin (Figure 61), a transverse stress of 291.9 psi and axial stress of -5.9 psi is predicted by the theory. Compared to the average of the simulation, an error of -9.8% is reached for the transverse direction. The axial direction is not a good approximation in this case. The average of the simulation deviates by -151%. For the bottom skin (Figure 62) a better approximation is reached. The predicted value for the transverse stress is 423.1 psi and for the axial stress 652.3 psi. Both values lead to an error of 2 to 3% compared to the mean value of the

middle region in the simulation. For the inflation pressure of 15 psi, the transverse and axial stresses are displayed in Figures 63 and 64 for the total force of 159.7 lbf.

Figure 63: Axial and transverse stresses at panel top skin for 15 psi

Figure 64: Axial and transverse stresses at panel bottom skin for 15 psi

For the middle of the top of the panel with an inflation pressure of 15 psi (Figure 63), a transverse stress of 474.4 psi and an axial stress of 156.8 psi are predicted. Comparing these values to the average of the simulation in the middle, an error of -9% is obtained for the transverse direction and -22.5% for the axial

direction. For the bottom of the panel (Figure 64), an error of -4% is obtained by comparing the simulation to the theoretical transverse stress of 606.6 psi and axial stress 820.5 psi. Finally, the transverse and axial stresses for a total applied load of 172.6 lbf are shown for 20 psi inflation pressure in Figure 65 and 66.

Figure 65: Axial and transverse stresses at panel top skin for 20 psi

Figure 66: Axial and transverse stresses at panel bottom skin for 20 psi

For the middle region of the top skin for an inflation pressure of 20 psi (65), a transverse stress of 652.2 psi and an axial stress of 305 psi are predicted. The percentage error to the average of the simulation therefore results in -9.1% in

the transverse direction and -16.3% in the axial direction. The detected error for the bottom skin (Figure 66) is smaller and amounts to -5.3% in axial as well as transverse directions. The predicted value by the theory in the transverse direction is 796.4 psi and in the axial direction 1009 psi.

In general, it can be said that for all investigated inflation pressures, the theoretical axial stress for the bottom skin is a valid estimate of the stresses obtained in the simulation. For the axial stress at the top of the panel, the stresses in the middle of the panel are slightly overestimated by the theory. In addition, the region at the ends of the panel where the theory predicts inflation stress only, the simulation shows stress changes during bending. The estimate of the axial stress in the middle region is more accurate at the bottom of the panel. The lower accuracy at the top of the panel could be explained by an additional superposing of stresses due to wrinkling of the top skin.

For the transverse stresses, a good representation by the theory is achieved for the top and bottom skin in the middle of the panel and especially for the bottom skin. At the sides of the panel, the theory underestimates the stresses at the top and overestimates them at the bottom. The deviation of the theoretical stress to the simulation increases the closer to the ends of the panel it is. This inaccuracy can be explained by the transverse strains which are less restricted by the larger axial dimensions the closer they are to the ends of the panel. At the ends of the panel the restriction only appears to one side. Therefore, the assumption of constant transverse strain loses its validity close to the ends of the panel.

In summary, it can be said that it is possible to obtain a reasonable estimate for the stress distribution from about 20 to 76 inches by the theory. Thus, the assumptions made to obtain the theory could be confirmed excluding the regions close to the end of the panel.

5.6 Summary and Conclusion

In Section 5.1 of this chapter, the creation of the finite element model of a four-point bending test on an inflatable drop-stitch panel is described. In Section 5.2 a convergence analysis is performed. It is determined that the solution is converged within the needed accuracy of this research using a mesh size of 0.125 inches.

After the convergence of the solution could be confirmed, the force displacement behavior of the inflated panel was investigated in section 5.3. The investigation of the whole panel in Section 5.3.1 shows that the simulation does not exactly match the behavior of the actual panel. While the simulation results do exhibit pressure dependent panel stiffness, the predicted stiffnesses are not as sensitive to inflation pressure as that observed experimentally. The simulation results, however, are in the same range of stiffness as the experimental results. Therefore, it can be concluded that the simulation delivers a good estimate of the behavior of the actual panel.

To further investigate the pressure dependence, the influence of the "tension only" option for the drop-stitch yarns on the pressure dependence is investigated in Section 5.3.2. To investigate the tension loss of the drop-stitch yarns, two points of interests have been examined: the point where the first drop-stitch yarns under the loading pin lose tension and the point where all drop-stitch yarns under the loading pin lose their tension. These points are mapped to the simulation's force displacement curves in Section 5.3.1. In addition, the behavior when both yarn tension and compression are allowed is analyzed. From these investigations, it is determined that the loss of tension of the drop-stitch yarns is a key mechanism that contributes to pressure dependent loss of stiffness for large deflections of the panel.

Due to the observed result that the nonlinear drop-stitch behavior has a signif-

ificant influence on the pressure dependent behavior, the difference between a non-linear and linear analysis is investigated in 5.3.3. A linear model without contact interactions was designed for this purpose. It is determined that the nonlinearity of the model is the decisive factor for the pressure dependence of the model. The hypothesis is made that the contact interaction is an important factor contributing to pressure dependent stiffness of the panel.

In Section 5.4, the deformed shape for about 7 inches deflection is compared for the simulation and the experiment. In addition, the experimentally measured kinking angle is examined and compared to the theoretical results in [7]. A good agreement between the deflection shape of the experiment and simulation is observed, which further validates the simulation results. Furthermore, the observed kinking angles for the simulation are close to the theoretically obtained values. However, the tendency slightly differs. While [7] predicts a linear tendency and no kinking angle for pressures over 25 psi, the simulation shows an exponential behavior with nonzero angles even at higher pressures. Therefore, the theoretically obtained kinking angles could be validated even though the assumption for zero kinking angle for inflation pressures larger than 25 psi would lead to small inaccuracies.

In Section 5.5, an analytical model for the stress distribution is derived. Thin pressure vessel theory, introduced in section 3.2, is used for the pressurization step and the beam bending stress introduced in Section 3.1 for the loading step.

Comparing the simulation results to the theoretical results for the pressurization step in Section 5.5.1, large deviations occur locally but comparing the average value a good agreement of the stresses due to inflation is reached.

For the loading step in Section 5.5.2, the bending stress is superposed on the inflation stress. Different composition of the stresses in different regions of the

panel are assumed for the axial stress. The transverse stress is calculated by the assumption of a constant transverse strain during bending. It could be determined that the theory can give a good estimate for the transverse and axial stress in the middle region within about 20 inches distance to the end of the panel. The assumption of constant transverse strain due to a much larger dimension in axial direction could therefore only be confirmed for the middle region of the panel.

CHAPTER 6

Conclusions and Future Research

6.1 Conclusions

In Chapter 4, classical lamination theory is applied to analyze the skin material of the panel. Using experimentally measured orthotropic layer properties does not give good agreement between the lamination theory results and experimentally obtained data for the whole skin. Reasons for this could be that the fabric used for the investigated panel does not behave like a continuum when tested separately or that inaccuracies in the testing data exist. To investigate the possibility of inaccuracies in the experimental data an optimization approach is pursued. By adjusting the values of the layer engineering constants, good agreement between experimental skin properties and lamination theory predictions can be obtained. It is determined that the Young's modulus for the optimized combination of layer stiffness constants are found to be in the error bounds of the experimental results. The optimized layer Poisson's ratios, however, differ significantly from the experimental values. However, the optimized Poisson's ratios appear to be of reasonable magnitude. Thus, the experimentally obtained Poisson's ratios could be inaccurate and should be further investigated. It can be concluded from the optimization that the deviating results for the original values could be caused by experimental inaccuracies. Therefore, continued investigation of the application of classical lamination theory in further research is recommended. If successful, this tool would allow designers to effectively consider a variety of skin materials and layups before constructing prototype panels.

In Chapter 5, a finite element model is designed for an inflatable drop-stitch panel under a four-point bending load. The model includes the possibility to change the material properties and inflation pressure. The simulation results are

first compared to the experimental force displacement results. It could be detected the model cannot fully represent the actual behavior. However, the pressure dependence of the force displacement results can be depicted by the model and the simulation and experimental results are in the same range of stiffnesses. Therefore, it can be concluded that the model can give a good approximation of the actual behavior of the panel. In addition, it could be determined that the reason for the pressure dependence of the inflatable drop-stitch panels are nonlinear effects. The difference for larger deflections is caused by the loss of tension in the drop-stitch yarns. It is assumed that especially the contact contributes to the initial difference in stiffness of the panels.

Furthermore, the bending shape of the deflected panels for a displacement of about 7 inches is compared for the simulation and the experimental four-point bending test. A good agreement between the experimental and simulation could be established. Hence, the kinking angle between the middle and side region of the panels is obtained by using a polynomial fit and the tangent slope. The obtained values for the angle are then compared to the kinking angles obtained by an optimization approach in [7]. It could be determined that the kinking angle obtained by the simulation and in [7] are very close. The obtained values in [7] could thus be validated by the simulation. But while [7] predicts a zero kinking angle for pressures higher than 25 psi, the simulation in this research shows nonzero but small angles for higher pressures.

Subsequently, an analytical model is derived to predict the axial and transverse stresses in the finite element model. The stresses due to the inflation pressure are obtained by using force equilibrium as in the thin pressure vessel theory. The theoretical value for the inflation pressure shows a good agreement compared to the average value of the stresses in the model. For the loading, the inflation

stresses are superposed with the theoretical bending stress obtained from beam bending theory to obtain an estimate in the axial direction. The transverse stress is calculated by assuming a constant transverse strain during the loading. It could be determined that the derived analytical model gives a good estimate for the middle region of the panel between about 20 and 76 inches. The validity of the model decreases closer to the ends of the panel because the transverse stress cannot be assumed as constant for the loading step anymore.

6.2 Future Research

In the future, it is recommended to repeat the research of [7] and this research with a different panel to analyze the outcome, and to evaluate the methods derived in [7] and this research. Repeating of the classical lamination theory in future research should include the shear modulus. In addition, it is recommended to find a more accurate method to obtain the Poisson's ratio. A possible method would be 3D-Digital Image Correlation (3D-DIC). It should also be further investigated whether the dry fabric behavior deviates significantly from a continuum behavior. If this is the case, it is recommended to use the layer wise composition of shells in Abaqus where the user subroutine VFABRIC could be used for all fabric layers.

Furthermore, a 3D-DIC could be used on an inflatable drop-stitch panel during a four-point bending test to obtain the axial and transverse stresses of the panel. Stress results obtained by this analysis could then be compared to the simulation results and the analytical model for the stress distribution. Thus, the assumptions made for the theoretically obtained stresses could be further investigated. In addition, the influence of the contact interaction on the pressure dependent stiffness should be investigated in future research. Moreover, the finite element model designed in this research should be extended to include fluid interactions, to make the investigation of compression tests or water filled panels possible.

LIST OF REFERENCES

- [1] *D6272-17 Standard Practice for Determining Sandwich Beam Flexural and Shear Stiffness*, ASTM International Std., 2016.
- [2] D. Roylance, “Pressure vessels,” Department of Materials Science and Engineering, Massachusetts Institute of Technology, Cambridge, Tech. Rep., 2001.
- [3] A. J. K. W. Altenbach, Holm, *Mechanics of Composite Structural Elements*, 2nd ed. Singapore: Springer Singapore, 2018.
- [4] D. L. Logan, *A first course in the finite element method*. Cengage Learning, 2011.
- [5] what-when-how.com. “Fem for plates and shells (finite element method) part 2.” June 2019. [Online]. Available: <http://what-when-how.com/the-finite-element-method/fem-for-plates-and-shells-finite-element-method-part-2/y>
- [6] P. V. Cavallaro, C. J. Hart, and A. M. Sadegh, “Mechanics of air-inflated drop-stitch fabric panels subject to bending loads,” in *ASME 2013 International Mechanical Engineering Congress and Exposition*. American Society of Mechanical Engineers, 2013, pp. V009T10A055–V009T10A055.
- [7] M. Smith, “Mechanical response of polymer-fabric skin materials used in inflatable drop-stitch structures,” Master’s thesis, University of Rhode Island (URI), 2019.
- [8] M. Ferrazzoli, Navatek, 2018-2019, data set, personal correspondence.
- [9] P. V. Cavallaro and A. M. Sadegh, “Air-inflated fabric structures,” NAVAL UNDERSEA WARFARE CENTER DIV NEWPORT RI, Tech. Rep., 2006.
- [10] P. Halswell, P. Wilson, and D. Taunton, “Hydroelastic inflatable boats: Relevant literature and new design considerations,” *International Journal of Small Craft Technology*, vol. 154 (B1), 2012.
- [11] J. Hodge, “Sea eagle 472rl razorlite inflatable drop stitch kayak,” Sea Eagle, 2018, accessed on 4-05-2019. [Online]. Available: <https://www.seaeagle.com/RazorLite/473rl>
- [12] L. L. R. DiGiovanna, “Characterizing the mechanical properties of drop stitch inflatable structures,” Ph.D. dissertation, Massachusetts Institute of Technology, 2013.

- [13] Inflatable Images. “Inflatable tents.” July 2019. [Online]. Available: <https://www.inflatableimages.com/custom-shelters-tents/#>
- [14] P. V. Cavallaro and R. W. Smith, “Conceptual inflatable fabric structures for protective crew quarters systems in space vehicles and space habitat structures,” Naval Undersea Warfare Center Division Newport, United States, Tech. Rep., 2015.
- [15] A. W. Hulton, P. V. Cavallaro, and C. J. Hart, “Modal analysis and experimental testing of air-inflated drop-stitch fabric structures used in marine applications,” in *ASME 2017 International Mechanical Engineering Congress and Exposition*. American Society of Mechanical Engineers, 2017, pp. V009T12A030–V009T12A030.
- [16] D. G. Bagnell, “Recent advancements in the development of inflatable multi-hull boats utilizing drop-stitch fabric,” in *11th International Conference on Fast Sea Transportation*, 2011.
- [17] J. Falls and J. Waters, “Bending tests of inflatable dropstitch panels,” in *11th International Conference on Fast Sea Transportation, FAST*, 2011.
- [18] C. M. Wang, “Timoshenko beam-bending solutions in terms of euler-bernoulli solutions,” *Journal of engineering mechanics*, vol. 121, no. 6, pp. 763–765, 1995.
- [19] R. Felicissimo, “Exploration of the mechanical properties of both ridged and inflated drop stitch fabric material,” Master’s thesis, City University of New York (CUNY), 2015.
- [20] C. Graczykowski and J. Heinonen, “Adaptive inflatable structures for protecting wind turbines against ship collisions,” IPPT - Institute of Fundamental Technology Research (Poland), Tech. Rep. VVT Working Papers 59, September 2006.
- [21] A. L. Van and C. Wielgosz, “Bending and buckling of inflatable beams: Some theoretical results,” *Thin-Walled Structures*, vol. 43(8), pp. 1166–1187, 2005.
- [22] *D7250/D7250M-16 Standard Test Method for Flexural Strength of Advanced Ceramics at Ambient Temperature*, ASTM International Std., 2018.
- [23] J. Lubliner and P. Papadopoulos, *Introduction to solid mechanics*. Springer, 2016.
- [24] K. Gramoll. eCourses. “Mechanics - theory- thin-walled pressure vessels.” May 2019. [Online]. Available: https://www.ecourses.ou.edu/cgi-bin/ebook.cgi?doc=&topic=me&chap_sec=07.6&page=theory

- [25] V. Vullo, *Circular Cylinders and Pressure Vessels : Stress Analysis and Design (3)*, ser. Springer Series in Solid and Structural Mechanics. Cham: Springer International Publishing, 2014. [Online]. Available: <http://dx.doi.org/10.1007/978-3-319-00690-1>
- [26] H. Altenbach, *Kontinuumsmechanik : Einführung in die materialunabhängigen und materialabhängigen Gleichungen*, 3rd ed. Berlin: Springer Vieweg, 2015.
- [27] M. H. Sadd, *Elasticity: theory, applications, and numerics*. Academic Press, 2009.
- [28] W. M. Lai, D. H. Rubin, E. Krempf, and D. Rubin, *Introduction to continuum mechanics*. Butterworth-Heinemann, 2009.
- [29] C. Mittelstedt and W. Becker, *Strukturmechanik ebener Laminate*. Technische Universität Darmstadt FB Mechanik, Bibliothek, 2017.
- [30] H. Altenbach, *Holzmann/Meyer/Schumpich Technische Mechanik Festigkeitslehre*, 12nd ed. Springer, 2016.
- [31] Y. Li and J. Barbič, “Stable orthotropic materials,” in *Proceedings of the ACM SIGGRAPH/Eurographics Symposium on Computer Animation*, ser. SCA '14. Aire-la-Ville, Switzerland, Switzerland: Eurographics Association, 2014, pp. 41–46. [Online]. Available: <http://dl.acm.org/citation.cfm?id=2849517.2849524>
- [32] H. Schürmann, *Konstruieren mit Faser-Kunststoff-Verbunden*. Springer, 2005.
- [33] J. N. Reddy, *Mechanics of laminated composite plates and shells: theory and analysis*. CRC press, 2004.
- [34] D. S. S. Corp., *ABAQUS/Standard Documentation*. Dassault Systmes Simulia Corp., 2018.
- [35] E. Oñate, *Structural analysis with the finite element method. Linear statics: volume 2: beams, plates and shells*. Springer Science & Business Media, 2013.
- [36] S. King and T. Richards, “Solving contact problems with abaqus,” Dassault Systemes, Coventry, UK, 2013.
- [37] Navatek. “Drop stitch.” Feb. 2019. [Online]. Available: <https://www.navatekltd.com/drop-stitch.html>
- [38] Y. Li and J. Barbič, “Stable orthotropic materials,” in *Proceedings of the ACM SIGGRAPH/Eurographics Symposium on Computer Animation*. Eurographics Association, 2014, pp. 41–46.

APPENDIX A

Matlab script for classical lamination theory

```
clear all
close all
format long
%% Define Properties of Layers
Layern=4;
%Layers can be changed to symmetric skin composition
Layers=["Dso"; "Neoprene"; "Chafer"; "Neoprene"];

%Chafer Layer
Chafert=0.02;
ChaferE1=35133;
ChaferE2=96310;
Chaferv12=0;
Chaferv21=0;
Chaferv12=Chaferv21*ChaferE1/ChaferE2;
ChaferG12=((ChaferE1+ChaferE2)/2)/(2*(1+Chaferv12));

%Dropstitch Layer
Dsot=0.015;
DsoE1=42986;
DsoE2=15810;
Dsov12=0;
Dsov21=0;
Dsov21=Dsov12*DsoE2/DsoE1;
DsoG12=((DsoE1+DsoE2)/2)/(2*(1+Dsov12));

%Neoprene
Neoprenet=0.01;
NeopreneE1=375;
NeopreneE2=375;
Neoprenev12=0.49;
Neoprenev21=0.49;
NeopreneG12=NeopreneE1/(2*(1+Neoprenev12));

%Whole Skin
Skint=0.055;
SkinE1=19662;
SkinE2=68100;
SkinG12=0;
Skinv12=0.297;
Skinv21=0.727;

ChaferE=makeE(ChaferE1,ChaferE2,Chaferv12,Chaferv21,ChaferG12);
NeopreneE=makeE(NeopreneE1,NeopreneE2,Neoprenev12,...
    Neoprenev21,NeopreneG12);
DsoE=makeE(DsoE1,DsoE2,Dsov12,Dsov21,DsoG12);
%% Classical Lamination Theory

A=zeros(3);
D=zeros(3);
B=zeros(3);
%Summation over Layers
for n=1:Layern
    A=A+eval(strcat(Layers(n),'E'))*eval(strcat(Layers(n),'t'));
end
x(1)=-Skint/2;
for n=1:Layern
    x(n+1)=x(n)+eval(strcat(Layers(n),'t'));
end
```

```

for n=1:Layern
    D=D+eval(strcat(Layers(n),'E'))*(1/3)*(x(n+1)^3-x(n)^3);
end

for n=1:Layern
    B=B+eval(strcat(Layers(n),'E'))*(1/2)*(x(n+1)^2-x(n)^2);
end

h=figure;
ABD=[A,B;B,D];
Ainv=inv(ABD);
%Determination of the resulting engineering constants
E1res=1/(Ainv(1,1)*Skint);
E2res=1/((Ainv(2,2)*Skint));
%Engineering constants for bending
%E1res=12/(Ainv(4,4)*Skint^3)
%E2res=12/(Ainv(5,5)*Skint^3)
G12=1/((Ainv(3,3)*Skint));
v21res=-Ainv(1,2)/Ainv(1,1);
%Visualization of the results
Elbar=[ChaferE1,DsoE1,NeopreneE1,SkinE1,E1res];
E2bar=[ChaferE2,DsoE2,NeopreneE2,SkinE2,E2res];
c = categorical({'E1-weft','E2-warp'});
bar(c,[Elbar;E2bar]);
ylabel('Youngs Modulus [psi]')
legend('Chafer','Dropstitch','Neoprene','Whole Skin',...
    'Resulting YM CLT','location','northwest')
grid on

function [E] = makeE(E1,E2,v12,v21,G12)
%UNTITLED3 Summary of this function goes here
% Detailed explanation goes here
E=zeros(3);
E(1,1)=E1/(1-v12*v21);
E(1,2)=E1*v21/(1-v12*v21);
E(2,2)=E2/(1-v12*v21);
E(2,1)=E2*v12/(1-v12*v21);
E(3,3)=G12;
end

```

APPENDIX B

Additional Analysis

B.1 Convergence S11

Figure B.67: Convergence for stress S_{11} for different inflation pressures

APPENDIX C

MATLAB and Python scripts for the Evaluation of Results

C.1 Postprocessing

C.1.1 Main Script - PostprocessingMatlab.m

```
system(['abaqus cae script=Postprocessing.py'])
clear all;
close all;
%% Force Displacement for different pressures
Pressure=[5,10,15,20];
for c=1:4
    h=figure;
    A=importdata(join([int2str(Pressure(c)), 'psiforcedis.txt'...
        ]), ' ', 5);
    Sim=A.data(:,2:4);
    hold on;
    plot(Sim(:,2), 4*Sim(:,1), 'b');
    plot(Sim(:,3)-Sim(1,3)+Sim(1,2), 4*Sim(:,1), 'r--');
    x=Sim(22-abs(4-c):30,2);
    y=4*Sim(22-abs(4-c):30,1);
    p=polyfit(x,y,1);
    y1=polyval(p,x);
    data=csvread(join(['Panel52617_', int2str(Pressure(c)), ...
        'psi.csv'], ''));
    plot([0;data(1:810,2)], [0;data(1:810,1)], 'g')
    xe=[0;data(1:80,2)];
    ye=[0;data(1:80,1)];
    pe=polyfit(xe,ye,1);
    yle=polyval(pe,xe);
    hold on;
    grid on;
    title(join([int2str(Pressure(c)), 'psi']))
    xlim([0,5]);
    xlabel('Displacement [in]')
    ylabel('Force [lbf]')
    legend('Simulation Bottom', 'Simulation Top', 'Experiment', ...
        'Location', 'southeast')
    annotation(gcf, 'textbox', ...
        [0.150008752567469, 0.682860682506624, 0.326657344386969, ...
        0.196011558481519], 'String', {'Initial slope: '}, ...
        ['Experiment = ' num2str(round(pe(1))), ' lbf/in'], ...
        ['Simulation = ' num2str(round(p(1))), ' lbf/in']})...
        , 'Interpreter', 'latex', 'EdgeColor', 'none', ...
        'BackgroundColor', [1 1 1], 'FitBoxToText', 'on');
end
%%
load('BeamDat.mat');
start=[96,100,101,100];
for c=1:4
    [xdis,y]=readpaneldis(Pressure(c));
    x=[0:0.125:48];
    h=figure;
    grid on
    hold on
    y=-y(start(c),1:385);
    x=x-xdis(start(c),:);
    plot([-x,x], [y,y], '.', 'MarkerSize', 8)
    xlim([-48 48]);
    ylim([-7.5 0]);
```

```

        plot(S{c,1}.x_exp-38,S{c,1}.v_exp, '.', 'MarkerSize',8)
        title(join([int2str(Pressure(c)), ' psi']))
        xlabel('Position z [in]')
        ylabel('Position y [in]')
        legend({'Simulation', 'Experiment'}, 'location', 'southeast')
    end
    getkinkingangle()
    %% Force Displacement all Pressures
    p=figure;
    for c=1:4
        A=importdata(join([int2str(Pressure(c)), ...
            'psiforcedis.txt']), ' ',5);
        Sim=A.data(:,2:4);
        hold on;
        plot(Sim(:,2),4*Sim(:,1), 'b');
        data=csvread(join(['Panel52617-',int2str(Pressure(c)), ...
            'psi.csv'], ''));
        plot([0;data(1:810,2)], [0;data(1:810,1)], 'g')
        hold on;
        grid on;
        xlim([0,5]);
        xlabel('Displacement [in]')
        ylabel('Force [lbf]')
        legend('Simulation ', 'Experiment', 'Location', 'southeast')
    end
    title('Force vs. Displacement 5, 10, 15 and 20 psi')
    %% Loss of tension under loading pin
    h=figure;
    for c=1:4
        kinky=0;
        kinkx=0;
        LOTy=0;
        LOTx=0 ;
        hold on
        grid on
        x=[0:0.1:1];
        A=importdata(join(['LOT',int2str(Pressure(c)), 'psi.txt']), ...
            ' ',3);
        Forcetxt=importdata(join([int2str(Pressure(c)), ...
            'psiforcedis.txt']), ' ',5);
        Force=Forcetxt.data(:,2)*4;
        LOT=A.data;
        bottompin=(LOT(:,2)-LOT(:,3))-(LOT(1,2)-LOT(1,3));
        toppin=(LOT(:,4)-LOT(:,5))-(LOT(1,4)-LOT(1,5));
        plot(Force,toppin);
        m(1)=0;
        for i=2:length(toppin)
            m(i)=(toppin(i)-toppin(i-1))/(Force(i)-Force(i-1));
            if ((Force(i)-Force(i-1))== 0)
                m(i)=0;
            end
            if (m(i)<-0.001 && kinky==0)
                kinky=toppin(i-1);
                kinkx=Force(i-1);
            elseif m(i)>0.001 && LOTy==0
                LOTy=toppin(i-1);
                LOTx=Force(i-1);
            end
        end
        if(LOTy > 0.02)
            plot(LOTx,LOTy, 'kx', 'MarkerSize',10, ...
                'HandleVisibility', 'off');
            LOTall(c,1)=LOTx;
            LOTall(c,2)=LOTy;
        else

```

```

        LOTall(c,1)=-1;
        LOTall(c,2)=-1;
    end
    if(kinky > 0.02)
        plot(kinkx,kinky,'gx','MarkerSize',10,...
            'HandleVisibility','off');
        Kink(c,1)=kinkx;
        Kink(c,2)=kinky;
    else
        Kink(c,1)=0;
        Kink(c,2)=0;
    end
end
plot(-1,-1,'kx','MarkerSize',10);
plot(-1,-1,'gx','MarkerSize',10);
ylim([0 0.6]);
xlabel('Force [lbf]')
ylabel('Difference top and bottom [in]')
title('Loss of tension in drop stitch yarns')
legend('5 psi','10 psi','15 psi','20 psi','Loss of tension',...
    'Negative slope','location','northeast')
%% loss of tension in Force Displacement graph
p=figure;
for c=1:4
    A=importdata(join([int2str(Pressure(c)),...
        'psiforcedis.txt']),',',5);
    Sim=A.data(:,2:4);
    hold on;
    plot(Sim(:,2),4*Sim(:,1));
    hold on;
    grid on;
    i=1;
    while Sim(i,1)< Kink(c,1)/4
        i=i+1;
    end
    n=1;
    while Sim(n,1)< LOTall(c,1)/4
        n=n+1;
    end
    plot(Sim(i,2),Kink(c,1),'gx','MarkerSize',10,...
        'HandleVisibility','off');
    plot(Sim(n,2),LOTall(c,1),'kx','MarkerSize',10,...
        'HandleVisibility','off');
    xlim([0,6.7]);
    xlabel('Displacement [in]');
    ylabel('Force [lbf]');
end
plot(-1,-1,'kx','MarkerSize',10);
plot(-1,-1,'gx','MarkerSize',10);
legend('5 psi','10 psi','15 psi','20 psi','Kink',...
    'Negative slope','Location','northwest')
title('Loss of tension in relation to whole model')

```

C.1.2 Called Scripts

Postprocessing.py

```

# -*- coding: mbcs -*-
#
# Abaqus/CAE Release 2018 replay file
# Internal Version: 2017_11_07-12.21.41 127140
# Run by alena on Thu May 16 13:13:41 2019
#
# from driverUtils import executeOnCaeGraphicsStartup
# executeOnCaeGraphicsStartup()

```

```

#: Executing "onCaeGraphicsStartup()" in the site directory ...
from abaqus import *
from abaqusConstants import *
session.Viewport(name='Viewport: 1', origin=(0.0, 0.0),
    width=98.2601165771484, height=190.285705566406)
session.viewports['Viewport: 1'].makeCurrent()
session.viewports['Viewport: 1'].maximize()
from caeModules import *
from driverUtils import executeOnCaeStartup
executeOnCaeStartup()
Pressures=['5psi', '10psi', '15psi', '20psi', '25psi', '30psi']
for i in range(0,6):
    o1 = session.openOdb(name='Job-'+Pressures[i]+'.odb')
    session.viewports['Viewport: 1'].setValues(displayedObject=o1)
    odb = session.odbs['Job-'+Pressures[i]+'.odb']
    session.XYDataFromHistory(name=
        'RF2 PI: BOTTOM_PIN N: 7201 NSET TOP_PIN-1', odb=odb,
        outputVariableName=
            'Reaction force: RF2 PI: BOTTOM_PIN Node 7201 in NSET TOP_PIN',
        steps=('mechanical_loading', ), __linkedVpName__=
            'Viewport: 1')
    session.XYDataFromHistory(name=
        'U2 PI: SKIN-1 N: 8 NSET MID_SPAN-1', odb=odb,
        outputVariableName=
            'Spatial displacement: U2 PI: SKIN-1 Node 8 in NSET MID_SPAN',
        steps=('mechanical_loading', ), __linkedVpName__=
            'Viewport: 1')
    session.XYDataFromHistory(name=
        'U2 PI: SKIN-1 N: 10 NSET TOP_MID_SPAN-1', odb=odb,
        outputVariableName=
            'Spatial displacement: U2 PI: SKIN-1 Node 10 in NSET TOP_MID_SPAN',
        steps=('mechanical_loading', ), __linkedVpName__=
            'Viewport: 1')
    x0 = session.xyDataObjects[
        'RF2 PI: BOTTOM_PIN N: 7201 NSET TOP_PIN-1']
    x1 = session.xyDataObjects[
        'U2 PI: SKIN-1 N: 8 NSET MID_SPAN-1']
    x2 = session.xyDataObjects[
        'U2 PI: SKIN-1 N: 10 NSET TOP_MID_SPAN-1']
    session.writeXYReport(fileName=Pressures[i]+'forcedis.txt',
        appendMode=OFF, xyData=(x0, x1, x2))
    del session.xyDataObjects[
        'RF2 PI: BOTTOM_PIN N: 7201 NSET TOP_PIN-1']
    del session.xyDataObjects[
        'U2 PI: SKIN-1 N: 8 NSET MID_SPAN-1']
    del session.xyDataObjects[
        'U2 PI: SKIN-1 N: 10 NSET TOP_MID_SPAN-1']
    odb = session.odbs['Job-'+Pressures[i]+'.odb']
    for n1 in range(1,3):
        session.XYDataFromHistory(name='U2 PI: SKIN-1 N: '
            +str(n1)+' NSET MIDDLELINE-1', odb=odb,
            outputVariableName=
                'Spatial displacement: U2 PI: SKIN-1 Node '
            +str(n1)+' in NSET MIDDLELINE',
            steps=('mechanical_loading', ),
            __linkedVpName__='Viewport: 1')
        session.XYDataFromHistory(name='U3 PI: SKIN-1 N: '
            +str(n1)+' NSET MIDDLELINE-1', odb=odb,
            outputVariableName=
                'Spatial displacement: U3 PI: SKIN-1 Node '
            +str(n1)+' in NSET MIDDLELINE',
            steps=('mechanical_loading', ),
            __linkedVpName__='Viewport: 1')
    for n2 in range(11,394):
        session.XYDataFromHistory(name='U2 PI: SKIN-1 N: '

```



```

+str(n2)+' NSET MIDDLELINE-1', odb=odb,
outputVariableName=
'Spatial displacement: U2 PI: SKIN-1 Node '
+str(n2)+' in NSET MIDDLELINE',
steps=('mechanical_loading', ),
__linkedVpName__='Viewport: 1')
session.XYDataFromHistory(name='U3 PI: SKIN-1 N: '
+str(n2)+' NSET MIDDLELINE-1', odb=odb,
outputVariableName=
'Spatial displacement: U3 PI: SKIN-1 Node '
+str(n2)+' in NSET MIDDLELINE',
steps=('mechanical_loading', ),
__linkedVpName__='Viewport: 1')
x=[]
x.append(session.xyDataObjects[
'U2 PI: SKIN-1 N: 1 NSET MIDDLELINE-1'])
x.append(session.xyDataObjects[
'U2 PI: SKIN-1 N: 2 NSET MIDDLELINE-1'])
for n in range(11,394):
    x.append(session.xyDataObjects[
'U2 PI: SKIN-1 N: '+str(n)+' NSET MIDDLELINE-1'])
x.append(session.xyDataObjects[
'U3 PI: SKIN-1 N: 1 NSET MIDDLELINE-1'])
x.append(session.xyDataObjects[
'U3 PI: SKIN-1 N: 2 NSET MIDDLELINE-1'])
for n in range(11,394):
    x.append(session.xyDataObjects[
'U3 PI: SKIN-1 N: '+str(n)+' NSET MIDDLELINE-1'])
session.writeXYReport(fileName=Pressures[i]+
'paneldis.txt', xyData=(x))
del session.xyDataObjects[
'U2 PI: SKIN-1 N: 1 NSET MIDDLELINE-1']
del session.xyDataObjects[
'U2 PI: SKIN-1 N: 2 NSET MIDDLELINE-1']
del session.xyDataObjects[
'U3 PI: SKIN-1 N: 1 NSET MIDDLELINE-1']
del session.xyDataObjects[
'U3 PI: SKIN-1 N: 2 NSET MIDDLELINE-1']
for c in range(11,394):
    del session.xyDataObjects['U2 PI: SKIN-1 N: '
+str(c)+' NSET MIDDLELINE-1']
    del session.xyDataObjects['U3 PI: SKIN-1 N: '
+str(c)+' NSET MIDDLELINE-1']
###
xy0 = session.XYDataFromHistory(name=
'U2 PI: PART-42-1-LIN-41-1 N: 1-1', odb=odb,
outputVariableName=
'Spatial displacement: U2 PI: PART-42-1-LIN-41-1 Node 1',
steps=('mechanical_loading', ),
__linkedVpName__='Viewport: 1')
xy1 = session.XYDataFromHistory(name=
'U2 PI: PART-42-1-LIN-41-1 N: 2-1', odb=odb,
outputVariableName=
'Spatial displacement: U2 PI: PART-42-1-LIN-41-1 Node 2',
steps=('mechanical_loading', ),
__linkedVpName__='Viewport: 1')
xy2 = session.XYDataFromHistory(name=
'U2 PI: PART-42-1-LIN-145-1 N: 1-1', odb=odb,
outputVariableName=
'Spatial displacement: U2 PI: PART-42-1-LIN-145-1 Node 1',
steps=('mechanical_loading', ),
__linkedVpName__='Viewport: 1')
xy3 = session.XYDataFromHistory(name=
'U2 PI: PART-42-1-LIN-145-1 N: 2-1', odb=odb,
outputVariableName=

```

```

        'Spatial displacement: U2 PI: PART-42-1-LIN-145-1 Node 2',
        steps=('mechanical_loading', ),
        __linkedVpName__='Viewport: 1')
x0 = session.xyDataObjects['U2 PI: PART-42-1-LIN-41-1 N: 1-1']
x1 = session.xyDataObjects['U2 PI: PART-42-1-LIN-41-1 N: 2-1']
x2 = session.xyDataObjects['U2 PI: PART-42-1-LIN-145-1 N: 1-1']
x3 = session.xyDataObjects['U2 PI: PART-42-1-LIN-145-1 N: 2-1']
session.writeXYReport(fileName='LOT'+Pressures[i]+'.txt',
    appendMode=OFF, xyData=(x0, x1, x2, x3))
del session.xyDataObjects['U2 PI: PART-42-1-LIN-41-1 N: 1-1']
del session.xyDataObjects['U2 PI: PART-42-1-LIN-41-1 N: 2-1']
del session.xyDataObjects['U2 PI: PART-42-1-LIN-145-1 N: 1-1']
del session.xyDataObjects['U2 PI: PART-42-1-LIN-145-1 N: 2-1']
###
session.odbs['Job-'+Pressures[i]+'.odb'].close()
sys.exit()

```

readpaneldis.m

```

function [disx,y] = readpaneldis(pressure)
%reads in the Displacement shape of the Panel and
%modifies it for later use
A=importdata(join([int2str(pressure),...
    'psipaneldis.txt' ]),',',5);
datau3=A.data(:,387:end);
datau2=A.data(:,2:386);
datau2=[datau2(:,1),datau2(:,3:end),datau2(:,2)];
datau3=[datau3(:,1),datau3(:,3:end),datau3(:,2)];
y=datau2;
disx=datau3;
end

```

getkinkingangle.m

```

function [] = getkinkingangle()
Pressure=[5,10,15,20,25,30];
Expangle=[10.51,7.31,4.26,1.53];
start=[96,100,101,100,99,98];
angleloc=94
for c=1:6
    [xdis,y]=readpaneldis(Pressure(c));
    x=[0:0.125:48] ;
    a=figure;
    hold on
    y=-y(start(c),1:385);
    x=x-xdis(start(c),:);
    xlim([0 40]);
    ylim([-7.5 0]);
    title(join([int2str(Pressure(c)), ' psi']));
    xlabel('Position z [in]');
    ylabel('Position y [in]');
    x1=x(1:angleloc);
    y1=y(1:angleloc);
    x2=x(angleloc:end);
    location(c)=x(angleloc)
    y2=y(angleloc:end);
    p1=polyfit(x1,y1,2);
    py1=polyval(p1,x1);
    p2=polyfit(x2,y2,3);
    py2=polyval(p2,x2);
    plot(x1,py1,'b');
    plot(x2,py2,'c');
    k = polyder(p1);
    d1=polyval(k,x1);

```

```

k2 = polyder(p2);
d2=polyval(k2,x2);
plot(x1(angleloc),py1(angleloc),'kx','MarkerSize',10);
plot(x2(1),py2(1),'kx','MarkerSize',10);
grid on
legend('Polynomial fit for x < 11.5',...
       'Polynomial fit for x > 11.5',...
       'Location of kinking angle','location','northwest')
phi1= rad2deg(atan(d1(angleloc)))
phi2= rad2deg(atan(d2(1)))
theta(c)=phi2-phi1
end
h=figure
plot(Pressure,theta)
hold on
grid on
plot(Pressure(1:4),Expangle,'x','MarkerSize',10)
xlabel('Pressure [psi]')
ylabel('Kinking Angle [ $\circ$ ]')
title('Pressure dependant kinking angle')
legend('Kinking Angle: Simulation',...
       'Kinking Angle: Beam Theory with Shear Deformation')
end

```

C.2 Stress Distribution

C.2.1 Main Script - plot_skin_stresses.m

```

function plot_skin_stresses
clc; clear all; close all
system(['abaqus cae script=Stressreport.py'])
Ea=68100;
Et=18662;
nu_at=0.727;
S11=1/Ea;
S22=1/Et;
S12=-nu_at/Ea;
S=[S11 S12; S12 S22];
for pressure_case=1:4
    w=20; l=96; h=4; r=2; t=0.055;
    if pressure_case==1
        p=5;
        [sig_a,sig_t,I,h]=get_inflation_stress(p,w,l,h,r,t)
        RF=get_force(pressure_case);
        pressure='5 psi';
        fea_result=get_5psi_pressure_only;
    elseif pressure_case==2
        p=10;
        [sig_a,sig_t,I,h]=get_inflation_stress(p,w,l,h,r,t)
        RF=get_force(pressure_case);
        pressure='10 psi';
        fea_result=get_10psi_pressure_only;
    elseif pressure_case==3
        p=15;
        [sig_a,sig_t,I,h]=get_inflation_stress(p,w,l,h,r,t)
        RF=get_force(pressure_case);
        pressure='15 psi';
        fea_result=get_15psi_pressure_only;
    else
        p=20;
        [sig_a,sig_t,I,h]=get_inflation_stress(p,w,l,h,r,t)
        RF=get_force(pressure_case);
        pressure='20 psi';
        fea_result=get_20psi_pressure_only;
    end
end

```

```

%
% Inflation induced stresses
%
fea_size=size(fea_result);
npts=(fea_size(1)-1);
x=[fea_result(:,1); 96-fea_result(npts:-1:1,1)];
%
sig_trans_top=[fea_result(:,2); fea_result(npts:-1:1,2)];
sig_axial_top=[fea_result(:,3); fea_result(npts:-1:1,3)];
sig_trans_bottom=[fea_result(:,4); fea_result(npts:-1:1,4)];
sig_axial_bottom=[fea_result(:,5); fea_result(npts:-1:1,5)];

axial_th=sig_a*ones(769,1);
trans_th=sig_t*ones(769,1);

figure
plot(x,sig_axial_top,'r',x,axial_th,'r--',x,...
      sig_trans_top,'b',x,trans_th,'b--')
set(gca,'fontsize',12);
title([pressure ' Inflation Stresses'])
xlabel('Axial position (in)')
ylabel('Stress (psi)')
legend('axial stress','axial-theory','transverse stress',...
       'transverse-theory','Location','northwest')
if pressure_case==1
    axis([0,100,100,250])
elseif pressure_case==2
    axis([0,100,250,475])
elseif pressure_case==3
    axis([0,100,400,700])
else
    axis([0,100,500,950])
end
annotation(gcf,'textbox',...
[0.620285714285713,0.684208198200663,0.262738664198503,...
0.20579180179934],'String',...
{'mean axial'=' num2str(round(mean(sig_axial_top),...
5,'significant'))},['axial theory'='...
num2str(round(mean(axial_th),5,'significant'))},...
['mean trans'=' num2str(round(mean(sig_trans_top),...
5,'significant'))},['trans theory'='...
num2str(round(mean(trans_th),5,'significant'))}],...
'Interpreter','latex','FitBoxToText','on');
%
% Inflation + bending induced stresses
%
if pressure_case==1
    fea_result=get_5psi_bending;
elseif pressure_case==2
    fea_result=get_10psi_bending;
elseif pressure_case==3
    fea_result=get_15psi_bending;
else
    fea_result=get_20psi_bending;
end
%
fea_size=size(fea_result);
npts=(fea_size(1)-1);
x=[fea_result(:,1); 96-fea_result(npts:-1:1,1)];
%
sig_trans_top=[fea_result(:,2); fea_result(npts:-1:1,2)];
sig_trans_top_mean=mean(sig_trans_top(301:469))
sig_axial_top=[fea_result(:,3); fea_result(npts:-1:1,3)];
sig_axial_top_mean=mean(sig_axial_top(301:469))
sig_trans_bottom=[fea_result(:,4); fea_result(npts:-1:1,4)];

```

```

    sig_trans_bottom_mean=mean(sig_trans_bottom(301:469))
    sig_axial_bottom=[fea_result(:,5); fea_result(npts:-1:1,5)];
    sig_axial_bottom_mean=mean(sig_axial_bottom(301:469))

    axial_th=sig_a*ones(385*2,1);
    trans_th=sig_t*ones(385*2,1);

figure
%
sig_bending=(2*RF*26*(h/2))/I
%
% stresses & strains
sig=[sig_a;sig_t];
eps=S*sig;
eps_t=eps(2);
for i=1:769
    if x(i)<=10
        axial_top_th(i)=axial_th(i);
        axial_bottom_th(i)=axial_th(i);
    elseif x(i)<=36
        axial_top_th(i)=axial_th(i)-sig_bending*(x(i)-10)/26;
        axial_bottom_th(i)=axial_th(i)+sig_bending*(x(i)-10)/26;
    elseif x(i)<=60
        axial_top_th(i)=axial_th(i)-sig_bending;
        axial_bottom_th(i)=axial_th(i)+sig_bending;
    elseif x(i)<=86
        axial_top_th(i)=axial_th(i)-sig_bending*(86-x(i))/26;
        axial_bottom_th(i)=axial_th(i)+sig_bending*(86-x(i))/26;
    elseif x(i)>86
        axial_top_th(i)=axial_th(i);
        axial_bottom_th(i)=axial_th(i);
    end
    trans_top_th(i)=eps_t/S22-(S12/S22)*axial_top_th(i);
    trans_bottom_th(i)=eps_t/S22-(S12/S22)*axial_bottom_th(i);
end
%
%bottom
%transverse deviation
trans_devi_bottom=(trans_bottom_th(400)-sig_trans_bottom_mean)/sig_trans_bottom_mean
%axial deviation
axial_devi_bottom=(axial_bottom_th(400)-sig_axial_bottom_mean)/sig_axial_bottom_mean
%top
%transverse deviation
trans_devi_top=(trans_top_th(400)-sig_trans_top_mean)/sig_trans_top_mean*100
%axial deviation
axial_devi_top=(axial_top_th(400)-sig_axial_top_mean)/sig_axial_top_mean*100

plot(x,sig_axial_top,'r',x,axial_top_th,'r--',x,...
    sig_trans_top,'b',x,trans_top_th,'b--')
set(gca,'fontsize',12);
title([pressure ' RF= ' num2str(round(4*RF,4,'significant'))...
    ' - Top skin (compression)'])
xlabel('Axial position (in)')
ylabel('Stress (psi)')
legend('axial stress','axial-theory','transverse stress',...
    'transverse-theory','Location','southeast')
if pressure_case==1
    axis([0,100,-400,300])
elseif pressure_case==2
    axis([0,100,-250,450])
elseif pressure_case==3
    axis([0,100,-100,650])
else

```

```

        axis([0,100,0,850])
    end
    % annotation(gcf,'textbox',...
    % [0.620285714285713,0.684208198200663,0.262738664198503,...
    % 0.20579180179934],'String',...
    % {'deviation middle axial' = ' num2str(round(axial_devi_top,...
    % 3,'significant'))},...
    % ['deviation middle trans' = ' num2str(round(trans_devi_top,...
    % 3,'significant'))}],...
    % 'Interpreter','latex','FitBoxToText','on');

    figure
    plot(x,sig_axial_bottom,'r',x,axial_bottom_th,'r--',x,...
        sig_trans_bottom,'b',x,trans_bottom_th,'b--')
    set(gca,'fontsize',12);
    title([pressure ' RF= ' num2str(round(4*RF,4,'significant'))...
        ' - Bottom skin (tension)'])
    xlabel('Axial position (in)')
    ylabel('Stress (psi)')
    legend('axial stress','axial-theory','transverse stress',...
        'transverse-theory','Location','south')
    if pressure_case==1
        axis([0,100,0,525])
    elseif pressure_case==2
        axis([0,100,200,750])
    elseif pressure_case==3
        axis([0,100,300,1000])
    else
        axis([0,100,500,1100])
    end
    % annotation(gcf,'textbox',...
    % [0.620285714285713,0.684208198200663,0.262738664198503,...
    % 0.20579180179934],'String',...
    % {'deviation middle axial' = ' num2str(round(axial_devi_bottom,...
    % 3,'significant'))},...
    % ['deviation middle transv' = ' num2str(round(trans_devi_bottom,...
    % 3,'significant'))}],...
    % 'Interpreter','latex','FitBoxToText','on');
end
for i=1:12
    figure(i)
    grid on
    figname=['Fig-' num2str(i) '.png'];
    saveas(gcf,figname)
end

function [sig_a,sig_t,I,h]=get_inflation_stress(p,w,l,h,r,t)
%
% Moment of inertia
%

$$I = 2 * (w * t^3 / 12 + w * t * (h / 2 + t / 2)^2) + (\pi / 4) * ((r + 2 * t)^4 - r^4)$$

% nominal stresses
%
sig_a = p * (w * h +  $\pi$  * r^2) / (2 * w * t + ( $\pi$  / 2) * ((r + 2 * t)^2 - r^2));
sig_t = p * (l * h +  $\pi$  * r^2) / (2 * l * t + ( $\pi$  / 2) * ((r + 2 * t)^2 - r^2));
sig=[sig_a; sig_t]
Ea=68100;
Et=18662;
nu_at=0.727;
S11=1/Ea;
S22=1/Et;
S12=-nu_at/Ea;
S=[S11 S12; S12 S22];

```

```

%
error=1;
count=0;
while error>1e-6
    count=count+1;
    % side wall (use thin walled cylinder for hoop stress)
    sig_side_wall=[sig_a/2; p*r/(4*t)];
    eps_side_wall=S*sig_side_wall;
    eps_hoop=eps_side_wall(2);
    % strains
    eps=S*sig;
    del_w=eps(2)*w;
    del_l=eps(1)*l;
    del_h=p*6.6e-3;
    del_r=r*eps_hoop;
    %
    w=20+del_w;
    l=96+del_l;
    h=4+del_h;
    r=2+del_r;

    sig_a_new=p*(w*h+pi*r^2)/(2*w*t+(pi/2)*((r+2*t)^2-r^2));
    sig_t_new=p*(l*h+pi*r^2)/(2*l*t+(pi/2)*((r+2*t)^2-r^2));

    error=sqrt((sig_a_new-sig_a)^2+(sig_t_new-sig_t)^2);
    %
    sig_a=sig_a_new;
    sig_t=sig_t_new;
    sig=[sig_a; sig_t];
end
disp(['Number of iterations: ' num2str(count)])
%     eps
%     sig
disp(['Pressure case: ' num2str(p)])
Area.and.Perimeter.width=[(w*h+pi*r^2),(2*w+2*pi*r)]
Area.and.Perimeter.length=[(l*h+pi*r^2),(2*l+2*pi*r)]
%
%
% Moment of inertia
%
I=2*(w*t^3/12+w*t*(h/2)^2)+(pi/4)*((r+2*t)^4-r^4);

function fea_result=get_5psi_pressure_only
%
% [x S11-Path1 S22-Path1 S11-Path2 S11-Path2]
%   where S11 - transverse stress
%   where S22 - axial stress
%   where Path1 - top (compression)
%   where Path2 - bottom (tension)
A=importdata('5psipressurization.txt',' ',3);
fea_result=A.data;

function fea_result=get_5psi_bending
%
% [x S11-Path1 S22-Path1 S11-Path2 S11-Path2]
%   where S11 - transverse stress
%   where S22 - axial stress
%   where Path1 - top (compression)
%   where Path2 - bottom (tension)
A=importdata('5psibending.txt',' ',3);
fea_result=A.data;

function RF=get_force(p)
%

```

```

if p==1
    A=importdata('5psiforcedis.txt',' ',4);
    A=A.data;
elseif p==2
    A=importdata('10psiforcedis.txt',' ',4);
    A=A.data;
elseif p==3
    A=importdata('15psiforcedis.txt',' ',4);
    A=A.data;
elseif p==4
    A=importdata('20psiforcedis.txt',' ',4);
    A=A.data;
end
for i=1:length(A)
    if A(i,1)==1.4
        RF=A(i,2)
    end
end

function fea_result=get_10psi_pressure_only
%
A=importdata('10psipressurization.txt',' ',3);
fea_result=A.data;

function fea_result=get_10psi_bending
%
A=importdata('10psibending.txt',' ',3);
fea_result=A.data;

function fea_result=get_15psi_pressure_only
%
A=importdata('15psipressurization.txt',' ',3);
fea_result=A.data;

function fea_result=get_15psi_bending
%
A=importdata('15psibending.txt',' ',3);
fea_result=A.data;

function fea_result=get_20psi_pressure_only
%
A=importdata('20psipressurization.txt',' ',3);
fea_result=A.data;

function fea_result=get_20psi_bending
%
A=importdata('20psibending.txt',' ',3);
fea_result=A.data;

```

C.2.2 Called Script - Stressreport.py

```

# -*- coding: mbcs -*-
#
# Abaqus/CAE Release 2018 replay file
# Internal Version: 2017-11-07-12.21.41 127140

# from driverUtils import executeOnCaeGraphicsStartup
# executeOnCaeGraphicsStartup()
from abaqus import *
from abaqusConstants import *
session.Viewport(name='Viewport: 1', origin=(0.0, 0.0),
    width=121.125, height=113.5546875)
session.viewports['Viewport: 1'].makeCurrent()
session.viewports['Viewport: 1'].maximize()
from caeModules import *

```



```

from driverUtils import executeOnCaeStartup
executeOnCaeStartup()
session.viewports['Viewport: 1']
    ].partDisplay.geometryOptions.setValues(
        referenceRepresentation=ON)
Pressures=['5psi', '10psi', '15psi', '20psi']
for i in range(0,4):
    ol = session.openOdb(name='Job-'+Pressures[i]+'.odb')
    session.viewports['Viewport: 1'].setValues(
        displayedObject=ol)
    if i==0:
        frames= [5 , 40]
    elif i==1:
        frames= [5,40]
    elif i==2:
        frames= [5,40]
    elif i==3:
        frames= [7,43]

leaf = dgo.LeafFromElementSets(elementSets=("SKIN-1.SET-1", ))
session.viewports['Viewport: 1']
    ].odbDisplay.displayGroup.replace(leaf=leaf)
session.viewports['Viewport: 1']
    ].view.setValues(session.views['Left'])
session.viewports['Viewport: 1']
    ].view.setProjection(projection=PARALLEL)
session.viewports['Viewport: 1'].view.setValues(nearPlane=
    96.3464, farPlane=109.939, width=4.57463, height=2.15468,
    cameraPosition=(-97.1427, 0.0437741, 0.774715),
    cameraTarget=(6, 0.0437741, 0.774715))
session.viewports['Viewport: 1'].view.fitView()
session.Path(name='Path-1', type=NODE_LIST, expression=((
    'SKIN-1', (7, '2435:2053:-1', 10, )), ))
session.Path(name='Path-2', type=NODE_LIST, expression=((
    'SKIN-1', (6, '1160:1542', 8, )), ))
session.viewports['Viewport: 1'].odbDisplay.setFrame(step=0,
    frame=frames[0])
session.viewports['Viewport: 1'].view.fitView()
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S11'))
pth = session.paths['Path-1']
session.XYDataFromPath(name='XY-Path-1-S11', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S22'))
pth = session.paths['Path-1']
session.XYDataFromPath(name='XY-Path-1-S22', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S11'))
pth = session.paths['Path-2']
session.XYDataFromPath(name='XY-Path-2-S11', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)

```

```

session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S22'))
pth = session.paths['Path-2']
session.XYDataFromPath(name='XY-Path-2-S22', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS, numIntervals=10,
    projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
x0 = session.xyDataObjects['XY-Path-1-S11']
x1 = session.xyDataObjects['XY-Path-1-S22']
x2 = session.xyDataObjects['XY-Path-2-S11']
x3 = session.xyDataObjects['XY-Path-2-S22']
session.writeXYReport(
    fileName=Pressures[i]+'pressurization.txt',
    appendMode=OFF, xyData=(x0, x1, x2, x3))
session.viewports['Viewport: 1'].odbDisplay.setFrame(step=1,
    frame=frames[1])
session.viewports['Viewport: 1'].view.fitView()
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S11'))
pth = session.paths['Path-1']
session.XYDataFromPath(name='XY-Path-1-S11', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S22'))
pth = session.paths['Path-1']
session.XYDataFromPath(name='XY-Path-1-S22', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S11'))
pth = session.paths['Path-2']
session.XYDataFromPath(name='XY-Path-2-S11', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='S', outputPosition=INTEGRATION_POINT,
    refinement=(COMPONENT, 'S22'))
pth = session.paths['Path-2']
session.XYDataFromPath(name='XY-Path-2-S22', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS,
    numIntervals=10, projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
x0 = session.xyDataObjects['XY-Path-1-S11']
x1 = session.xyDataObjects['XY-Path-1-S22']
x2 = session.xyDataObjects['XY-Path-2-S11']
x3 = session.xyDataObjects['XY-Path-2-S22']
session.writeXYReport(fileName=Pressures[i]+'bending.txt',
    appendMode=OFF, xyData=(x0, x1, x2, x3))
session.odbs['Job-'+Pressures[i]+'.odb'].close()
sys.exit()

```

C.3 Cross-sectional Area from Simulation

C.3.1 Main Script- areacalc.m

```
system(['abaqus cae script=saveareafromodbnew.py'])
%%
clear all
close all
set(groot, 'defaulttextinterpreter', 'latex');
set(groot, 'defaultAxesTickLabelInterpreter', 'latex');
set(groot, 'defaultLegendInterpreter', 'latex');
Pressure=[5,10,15,20];
tab=zeros(5,4);
for c=1:4
    h=figure
    x=importdata(join(['Path1x',int2str(Pressure(c)),...
        'psi.txt']), ' ', 2);
    y=importdata(join(['Path1y',int2str(Pressure(c)),...
        'psi.txt']), ' ', 2);
    x=x.data;
    y=y.data;
    xy=[x(:,1) y(:,1) x(:,2) y(:,2)];
    xyundef=xy(:,1:2);
    xydef=xyundef+xy(:,3:4);
    plot(xyundef(:,1),xyundef(:,2), 'x')
    hold on
    plot(xydef(:,1),xydef(:,2), '.');
    xlim([-0.5,14]);
    ylim([-1,4.5]);
    xyundef=[xy(:,1:2)];
    xydef=[xydef];
    [k,v]=boundary(xyundef(:,1),xyundef(:,2));
    area=v*2;
    grid on
    [k,v]=boundary(xydef(:,1),xydef(:,2));
    area2=v*2;
    legend('Undeformed area','Deformed area','location','best')
    title(strcat('Area $ \ $',num2str(Pressure(c)),...
        'psi transverse'))
    % Perimeter
    Perimundef=0
    for i=1:length(xyundef)-1
        Perimundef=Perimundef+...
            norm(xyundef(i,:)-xyundef(i+1,:));
    end
    Perimundef=Perimundef*2
    Perimdef=0
    for i=1:length(xydef)-1
        Perimdef=Perimdef+norm(xydef(i,:)-xydef(i+1,:));
    end
    Perimdef=Perimdef*2;
    dim = [0.3 0.3 0.3 0.3];
    str = {strcat('undeformed Area = ',num2str(area)),...
        strcat('undeformed Perimeter = ',num2str(Perimundef)),...
        strcat('deformed Area = ',num2str(area2)),...
        strcat('deformed Perimeter = ',num2str(Perimdef))};
    annotation('textbox',dim,'String',str,'FitBoxToText',...
        'on','Interpreter','latex');
    tab(c+1,1)=area2;
    tab(c+1,2)=Perimdef;
end
tab(1,1)=area;
tab(1,2)=Perimundef;
%%
clearvars -except tab
```

```

Pressure=[5,10,15,20]
for c=1:4
    h=figure;
    x=importdata(join(['Path2z',num2str(Pressure(c)),...
        'psi.txt']),',',2);
    y=importdata(join(['Path2y',num2str(Pressure(c)),...
        'psi.txt']),',',2);
    x=x.data;
    y=y.data;
    xy=[x(2:end,1) y(2:end,1) x(2:end,2) y(2:end,2)];
    xyundef=xy(:,1:2);
    xydef=xyundef+xy(:,3:4);
    plot(xyundef(:,1),xyundef(:,2),'x')
    hold on
    plot(xydef(:,1),xydef(:,2),'.')
    xlim([-5,50])
    ylim([-0.5,4.5])
    xyundef=[xy(:,1:2)];
    xydef=[xydef];
    [k,v]=boundary(xyundef(:,1),xyundef(:,2));
    area=v*2;
    grid on
    [k,v]=boundary(xydef(:,1),xydef(:,2));
    area2=v*2;
    legend('Undeformed area','Deformed area','location','best')
    title(strcat('Area $\ $ ',num2str(Pressure(c)),'psi axial'))
    % Perimeter
    Perimundef=0
    for i=1:length(xyundef)-1
        Perimundef=Perimundef+norm(xyundef(i,:)-...
            -xyundef(i+1,:));
    end
    Perimundef=Perimundef*2
    Perimdef=0
    for i=1:length(xydef)-1
        Perimdef=Perimdef+norm(xydef(i,:)-xydef(i+1,:))
    end
    Perimdef=Perimdef*2;
    dim = [0.3 0.3 0.3 0.3];
    str = {strcat('undeformed Area = ',num2str(area)),...
        strcat('undeformed Perimeter = ',num2str(Perimundef)),...
        strcat('deformed Area = ',num2str(area2)),...
        strcat('deformed Perimeter = ',num2str(Perimdef))};
    annotation('textbox',dim,'String',str,'FitBoxToText',...
        'on','Interpreter','latex');
    tab(c+1,3)=area2;
    tab(c+1,4)=Perimdef;
end
tab(1,3)=area;
tab(1,4)=Perimundef;
Area_width=tab(:,1);
Perimeter_width=tab(:,2);
Area_length=tab(:,3);
Perimeter_length=tab(:,4);
AreaT=table({'0 psi';'5 psi';'10 psi';'15 psi';'20 psi'},...
    Area_width,Perimeter_width,Area_length,Perimeter_length)
writetable(AreaT,'Areachange.csv','Delimiter','\n')

```

C.3.2 Called Script - saveareafromodb.py

```

# -*- coding: mbcs -*-
#
# Abaqus/CAE Release 2018 replay file
# Internal Version: 2017-11-07-12.21.41 127140

```

```

# from driverUtils import executeOnCaeGraphicsStartup
# executeOnCaeGraphicsStartup()
#: Executing "onCaeGraphicsStartup()" in the site directory ...
from abaqus import *
from abaqusConstants import *
session.Viewport(name='Viewport: 1', origin=(0.0, 0.0),
    width=98.2601165771484, height=190.285705566406)
session.viewports['Viewport: 1'].makeCurrent()
session.viewports['Viewport: 1'].maximize()
from caeModules import *
from driverUtils import executeOnCaeStartup
executeOnCaeStartup()
Pressures=['5psi', '10psi', '15psi', '20psi']
for i in range(0,4):
    if i ==3:
        fr=7
    else:
        fr=5
    o1 = session.openOdb(name='Job-'+Pressures[i]+'.odb')
    session.viewports['Viewport: 1'].setValues(displayedObject=o1)
    session.Path(name='Path-1', type=NODE_LIST, expression=(
        ('SKIN-1', (8, '1081:1159', 4, '801:824', 1, '1543:1566',
        9, '1974:2052', 10, )),))
    session.Path(name='Path-2', type=NODE_LIST, expression=((
        'SKIN-1', (8, 10, '2053:2435', 7, '1001:953:-1', 6,
        '1160:1542', 8, )), ))
    session.viewports['Viewport: 1'].odbDisplay.setFrame(step=0,
        frame=fr )
    session.viewports['Viewport: 1'].odbDisplay.display.setValues(
        plotState=(DEFORMED, ))
    session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
        variableLabel='U', outputPosition=NODAL, refinement=(
        INVARIANT, 'Magnitude'))
    session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
        variableLabel='U', outputPosition=NODAL, refinement=(
        COMPONENT, 'U1'))
    pth = session.paths['Path-1']
    session.XYDataFromPath(name='Path1x', path=pth,
        includeIntersections=False, projectOntoMesh=False,
        pathStyle=PATH.POINTS, numIntervals=10,
        projectionTolerance=0, shape=UNDEFORMED,
        labelType=X.COORDINATE)
    session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
        variableLabel='U', outputPosition=NODAL, refinement=(
        COMPONENT, 'U2'))
    pth = session.paths['Path-1']
    session.XYDataFromPath(name='Path1y', path=pth,
        includeIntersections=False, projectOntoMesh=False,
        pathStyle=PATH.POINTS, numIntervals=10,
        projectionTolerance=0, shape=UNDEFORMED,
        labelType=Y.COORDINATE)
    x0 = session.xyDataObjects['Path1x']
    x1 = session.xyDataObjects['Path1y']
    session.writeXYReport(fileName='Path1'+Pressures[i]+'.txt',
        appendMode=OFF, xyData=(x0, x1))
    x0 = session.xyDataObjects['Path1x']
    session.writeXYReport(fileName='Path1x'+Pressures[i]+'.txt',
        appendMode=OFF, xyData=(x0, ))
    x0 = session.xyDataObjects['Path1y']
    session.writeXYReport(fileName='Path1y'+Pressures[i]+'.txt',
        appendMode=OFF, xyData=(x0, ))
    pth = session.paths['Path-2']
    session.XYDataFromPath(name='Path2y', path=pth,
        includeIntersections=False,
        projectOntoMesh=False, pathStyle=PATH.POINTS,

```

```

        numIntervals=10, projectionTolerance=0,
        shape=UNDEFORMED, labelType=Y_COORDINATE)
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='U', outputPosition=NODAL,
    refinement=(COMPONENT, 'U1'))
pth = session.paths['Path-2']
session.XYDataFromPath(name='Path2x', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS, numIntervals=10,
    projectionTolerance=0, shape=UNDEFORMED,
    labelType=X_COORDINATE)
x0 = session.xyDataObjects['Path2x']
session.writeXYReport(fileName='Path2x'+Pressures[i]+'.txt',
    appendMode=OFF, xyData=(x0, ))
x0 = session.xyDataObjects['Path2y']
session.writeXYReport(fileName='Path2y'+Pressures[i]+'.txt',
    appendMode=OFF, xyData=(x0, ))
session.viewports['Viewport: 1'].odbDisplay.setPrimaryVariable(
    variableLabel='U', outputPosition=NODAL, refinement=(
    COMPONENT, 'U3'))
pth = session.paths['Path-2']
session.XYDataFromPath(name='Path2z', path=pth,
    includeIntersections=False, projectOntoMesh=False,
    pathStyle=PATH_POINTS, numIntervals=10,
    projectionTolerance=0, shape=UNDEFORMED,
    labelType=Z_COORDINATE)
x0 = session.xyDataObjects['Path2z']
session.writeXYReport(fileName='Path2z'+Pressures[i]+'.txt',
    appendMode=OFF, xyData=(x0, ))
session.odbs['Job-'+Pressures[i]+'.odb'].close()
sys.exit()

```

BIBLIOGRAPHY

- Altenbach, Holm, A. J. K. W., *Mechanics of Composite Structural Elements*, 2nd ed. Singapore: Springer Singapore, 2018.
- Altenbach, H., *Kontinuumsmechanik : Einführung in die materialunabhängigen und materialabhängigen Gleichungen*, 3rd ed. Berlin: Springer Vieweg, 2015.
- Altenbach, H., *Holzmann/Meyer/Schumpich Technische Mechanik Festigkeitslehre*, 12nd ed. Springer, 2016.
- D6272-17 Standard Practice for Determining Sandwich Beam Flexural and Shear Stiffness*, ASTM International Std., 2016.
- D7250/D7250M-16 Standard Test Method for Flexural Strength of Advanced Ceramics at Ambient Temperature*, ASTM International Std., 2018.
- Bagnell, D. G., “Recent advancements in the development of inflatable multi-hull boats utilizing drop-stitch fabric,” in *11th International Conference on Fast Sea Transportation*, 2011.
- Cavallaro, P. V., Hart, C. J., and Sadegh, A. M., “Mechanics of air-inflated drop-stitch fabric panels subject to bending loads,” in *ASME 2013 International Mechanical Engineering Congress and Exposition*. American Society of Mechanical Engineers, 2013, pp. V009T10A055–V009T10A055.
- Cavallaro, P. V. and Sadegh, A. M., “Air-inflated fabric structures,” NAVAL UNDERSEA WARFARE CENTER DIV NEWPORT RI, Tech. Rep., 2006.
- Cavallaro, P. V. and Smith, R. W., “Conceptual inflatable fabric structures for protective crew quarters systems in space vehicles and space habitat structures,” Naval Undersea Warfare Center Division Newport, United States, Tech. Rep., 2015.
- Corp., D. S. S., *ABAQUS/Standard Documentation*. Dassault Systmes Simulia Corp., 2018.
- DiGiovanna, L. L. R., “Characterizing the mechanical properties of drop stitch inflatable structures,” Ph.D. dissertation, Massachusetts Institute of Technology, 2013.
- Falls, J. and Waters, J., “Bending tests of inflatable dropstitch panels,” in *11th International Conference on Fast Sea Transportation, FAST*, 2011.

- Felicissimo, R., “Exploration of the mechanical properties of both ridged and inflated drop stitch fabric material,” Master’s thesis, City University of New York (CUNY), 2015.
- Ferrazzoli, M., Navatek, 2018-2019, data set, personal correspondence.
- Graczykowski, C. and Heinonen, J., “Adaptive inflatable structures for protecting wind turbines against ship collisions,” IPPT - Institute of Fundamental Technology Research (Poland), Tech. Rep. VVT Working Papers 59, September 2006.
- Gramoll, K. eCourses. “Mechanics - theory- thin-walled pressure vessels.” May 2019. [Online]. Available: https://www.ecourses.ou.edu/cgi-bin/ebook.cgi?doc=&topic=me&chap_sec=07.6&page=theory
- Halswell, P., Wilson, P., and Taunton, D., “Hydroelastic inflatable boats: Relevant literature and new design considerations,” *International Journal of Small Craft Technology*, vol. 154 (B1), 2012.
- Hodge, J., “Sea eagle 472rl razorlite inflatable drop stitch kayak,” Sea Eagle, 2018, accessed on 4-05-2019. [Online]. Available: <https://www.seaeagle.com/RazorLite/473rl>
- Hulton, A. W., Cavallaro, P. V., and Hart, C. J., “Modal analysis and experimental testing of air-inflated drop-stitch fabric structures used in marine applications,” in *ASME 2017 International Mechanical Engineering Congress and Exposition*. American Society of Mechanical Engineers, 2017, pp. V009T12A030–V009T12A030.
- Inflatable Images. “Inflatable tents.” July 2019. [Online]. Available: <https://www.inflatableimages.com/custom-shelters-tents/#>
- King, S. and Richards, T., “Solving contact problems with abaqus,” Dassault Systemes, Coventry, UK, 2013.
- Lai, W. M., Rubin, D. H., Krempl, E., and Rubin, D., *Introduction to continuum mechanics*. Butterworth-Heinemann, 2009.
- Li, Y. and Barbič, J., “Stable orthotropic materials,” in *Proceedings of the ACM SIGGRAPH/Eurographics Symposium on Computer Animation*. Eurographics Association, 2014, pp. 41–46.
- Li, Y. and Barbič, J., “Stable orthotropic materials,” in *Proceedings of the ACM SIGGRAPH/Eurographics Symposium on Computer Animation*, ser. SCA ’14. Aire-la-Ville, Switzerland, Switzerland: Eurographics Association, 2014, pp. 41–46. [Online]. Available: <http://dl.acm.org/citation.cfm?id=2849517.2849524>

- Logan, D. L., *A first course in the finite element method*. Cengage Learning, 2011.
- Lubliner, J. and Papadopoulos, P., *Introduction to solid mechanics*. Springer, 2016.
- Mittelstedt, C. and Becker, W., *Strukturmechanik ebener Laminate*. Technische Universität Darmstadt FB Mechanik, Bibliothek, 2017.
- Navatek. “Drop stitch.” Feb. 2019. [Online]. Available: <https://www.navatekltd.com/drop-stitch.html>
- Oñate, E., *Structural analysis with the finite element method. Linear statics: volume 2: beams, plates and shells*. Springer Science & Business Media, 2013.
- Reddy, J. N., *Mechanics of laminated composite plates and shells: theory and analysis*. CRC press, 2004.
- Roylance, D., “Pressure vessels,” Department of Materials Science and Engineering, Massachusetts Institute of Technology, Cambridge, Tech. Rep., 2001.
- Sadd, M. H., *Elasticity: theory, applications, and numerics*. Academic Press, 2009.
- Schürmann, H., *Konstruieren mit Faser-Kunststoff-Verbunden*. Springer, 2005.
- Smith, M., “Mechanical response of polymer-fabric skin materials used in inflatable drop-stitch structures,” Master’s thesis, University of Rhode Island (URI), 2019.
- Van, A. L. and Wielgosz, C., “Bending and buckling of inflatable beams: Some theoretical results,” *Thin-Walled Structures*, vol. 43(8), pp. 1166–1187, 2005.
- Vullo, V., *Circular Cylinders and Pressure Vessels : Stress Analysis and Design (3)*, ser. Springer Series in Solid and Structural Mechanics. Cham: Springer International Publishing, 2014. [Online]. Available: <http://dx.doi.org/10.1007/978-3-319-00690-1>
- Wang, C. M., “Timoshenko beam-bending solutions in terms of euler-bernoulli solutions,” *Journal of engineering mechanics*, vol. 121, no. 6, pp. 763–765, 1995.
- what-when-how.com. “Fem for plates and shells (finite element method) part 2.” June 2019. [Online]. Available: <http://what-when-how.com/the-finite-element-method/fem-for-plates-and-shells-finite-element-method-part-2/y>