EXPERIMENTATION OF COMPOSITE REPAIR TECHNIQUES FOR PIPELINES USING FINITE ELEMENT ANALYSIS WITH RESPECT TO ASME PCC-2, ISO 24817, AND PARAMETRIC MODELS

A Thesis

by

TAYLOR AUSTIN BARRON

Submitted to the Office of Graduate and Professional Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Chair of Committee, Chii-Der S. Suh
Committee Members, Thomas R. Lalk
Luciana R. Barroso
Head of Department, Andreas A. Polycarpou

August 2016

Major Subject: Mechanical Engineering

ABSTRACT

A relatively new method for corrosion repair in steel pipelines has been developed by externally wrapping damaged pipes with composites. Both ASME PCC-2 and ISO 24817, engineering standards concerning pipelines and pipe repairs, have analytical solutions to dictate the minimum composite repair thickness required to safely rehabilitate a corroded steel pipe. When the pipe is assumed to yield into the composite wrap, certain design allowances reduce the necessary composite thickness based upon the live pressure of the composite wrap.

Using finite element analysis, an investigation was carried out to determine whether ASME PCC-2 and ISO 24817 analytical solutions create code compliant composite wraps for various steel wall thinning percentages and live pressures. Results indicate that when considering live pressure, neither ASME nor ISO standards create max hoop strain compliant composite wraps across all wall thinning percentages and live pressures. A parametrically modified version of the ASME PCC-2 standard results in code compliant wraps that more efficiently satisfy the max hoop strain requirement for all tested wall thinning percentages and live pressures.

It is recommended that ASME PCC-2 equation governing composite repairs on pipes with live pressure considerations be updated to include parametric modifiers. This will lead to the creation of more cost efficient, code compliant composite wraps.

ACKNOWLEDGEMENTS

I would like to thank my committee chair, Dr. Chii-Der Suh, and my committee members, Dr. Thomas Lalk and Dr. Luciana Barroso for their guidance and support throughout the course of this research. I would specifically like to thank Dr. Suh for encouraging me to pursue graduate school. Through this experience, I have gained a greater appreciation for engineering as a whole and a greater understanding of the research process.

Secondly, I would like to thank Dr. Christopher Alexander for his guidance in the state of the art of composite wraps; whose contributions kept this research grounded and established the initial focus from day one.

I would also like to thank the Civil Engineering Department, Dr. Ray James, Dr. Glen Miller, and Dr. Martin Peterson for providing an opportunity to continue and enrich my educational career as a teaching assistant.

Finally, I would like to thank my mother and father for their encouragement and to my wife for her patience, love, and support.

CONTRIBUTORS AND FUNDING SOURCES

This work was supported by a thesis committee consisting of Dr. Chii-der Suh of the Mechanical Engineering Department (advisor), Dr. Thomas Lalk of the Mechanical Engineering Department, and Dr. Luciana Barroso of the Civil Engineering Department.

This work was funded independently of any primary research resource. Indirect support of this research was achieved through a teaching assistantship within the Civil Engineering Department, as well as through a paid summer internship at Stress Engineering Services.

This thesis is an independent extension of work completed under Dr. Christopher Alexander while at Stress Engineering Services. Working with Dr. Alexander established the initial problem statement. All other works were independently decided upon and completed by the student, including all finite element modelling, analysis, and coding.

NOMENCLATURE

D	Component outside diameter, mm (in.)
ε_c	Hoop strain in the composite
ε_{live}	Hoop strain at live pressure
ε_{clive}	Hoop strain at live pressure in the composite
ε_{slive}	Hoop strain at live pressure in the steel
E_c	Tensile modulus for the composite laminate in the circumferential direction determined by test, N/m2 (psi)
E_s	Tensile modulus for substrate material, N/m2 (psi)
MOWP	Maximum Operating Working Pressure SMYS derated as required by the appropriate construction code) of component, N/m2 (psi)
P	Internal design pressure, N/m2 (psi)
P_{live}	Internal pressure within the component during application of the repair, N/m2 (psi)
SMYS	Specified Minimum Yield Strength, N/m2 (psi)
s	SMYS of the steel component, N/m2 (psi)
t_s	Minimum remaining wall thickness of the component, mm (in.)
$t_{\min}$	Minimum repair thickness, mm (in.)

TABLE OF CONTENTS

Page

LIST OF FIGURES

LIST OF TABLES

Page

CHAPTER I

INTRODUCTION AND LITERATURE REVIEW

The following chapter has three sections. Section 1.1 establishes a general overview of composite wraps and the variables the testing is interested in. Section 1.2 reviews and critiques recent literature focusing on pipe repair stresses. Section 1.3 derives and critiques the ASME PCC-2 equation most central to this research endeavor.

1.1 Overview of Composite Wraps and ASME PCC-2

A relatively new method for corrosion repair in pipelines has been developed by externally wrapping damaged pipes with composites. Pipes that have external or internal corrosion where the corroded depth is less than 80% of the steel wall are eligible for composite over wrap rehabilitation. Both ASME PCC-2 and ISO 24817, engineering standards concerning pipelines and pipe repairs, have analytical solutions to dictate the minimum composite repair thickness required to safely rehabilitate a corroded pipe. For the ASME standard the specific equation in question is PCC-2 Underlying Substrate Yields Equation 5 (Section 3.4.3.2, 2015) [1]. When the pipe is assumed to yield into the composite wrap, certain design allowances reduce the necessary composite thickness based upon the installation pressure of the composite wrap. The following ASME equation dictates how thick the composite wrap needs to be. The equation is iteratively solved for the composite thickness, tc, as the hoop strain within the composite, ε_c, is mandated to be a maximum of 0.25% hoop strain.

$$\varepsilon_c = \frac{PD}{2E_c t_c} - s_y \frac{t_s}{E_c t_c} - \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{1}$$

In Equation 1 the remaining symbols are final design pressure P, outer steel diameter Ds, composite Young's modulus Ec, composite repair thickness tc, yield stress of the steel Sy, steel thickness ts, live wrapping pressure P_{live}, and Young's modulus of the steel Es. The first of the three terms represents the strain in the composite at the design pressure. The second term represents the strain held by the thinned (corroded) steel. The third term represents the strain at the live pressure (composite wrap installation pressure). Two main variables are examined, live pressure and remaining steel thickness. Live pressure refers to the installation pressure of the composite upon the corroded steel pipe. Steel thickness refers to the remaining wall thickness of the steel after the corrosion loss is measured. Each live pressure/steel thickness scenario creates an input into a finite element model of the pipe to be examined. More testing detail and the various governing standards are explained in the methodology section.

1.2 Literature Review

The following section outlines the most recent applicable research in composite repair techniques. Each article is presented, and then critically related to the research methods used to analyze how live pressure and steel thickness effect the viability of composite repairs.

1.1.1 Introduction to Composites in Pipeline Repair

Currently, a significant issue facing the long term life of oil and gas transmission pipelines is managing erosion, corrosion, and unintentional mechanical damage. The Interstate Natural Gas Association of America [2] estimated that 12% of pipeline infrastructure was installed prior to 1950, 37% prior to 1960, and 60% before 1970. A large majority of these pipelines were installed without modern anti corrosion methods such as interior coatings or cathodic protection. In the United States alone, between 2 and 3.3 billion dollars a year are lost to replacing or repairing corroded pipelines.

Composites offer benefits over traditional welded sleeves because composites do not require a pipeline to be taken offline. Composite repairs allow for pipeline owners to keep product moving and reduce opportunity costs losses associated with repairs. Also, there is no risk of explosion as composite repairs do not create high temperatures like welding does. Composite repair wraps can be used to rehab steel pipes that suffer from external or internal corrosion or mechanical damage that does not exceed 80% of the steel's thickness.

Figure 1 is a cutaway of a composite wrap applied to an in service pipeline. Notice the pit of corrosion is filled with an epoxy filler before the composite wrap is applied. The remaining steel thickness is measured from the bottom of the corroded pit.

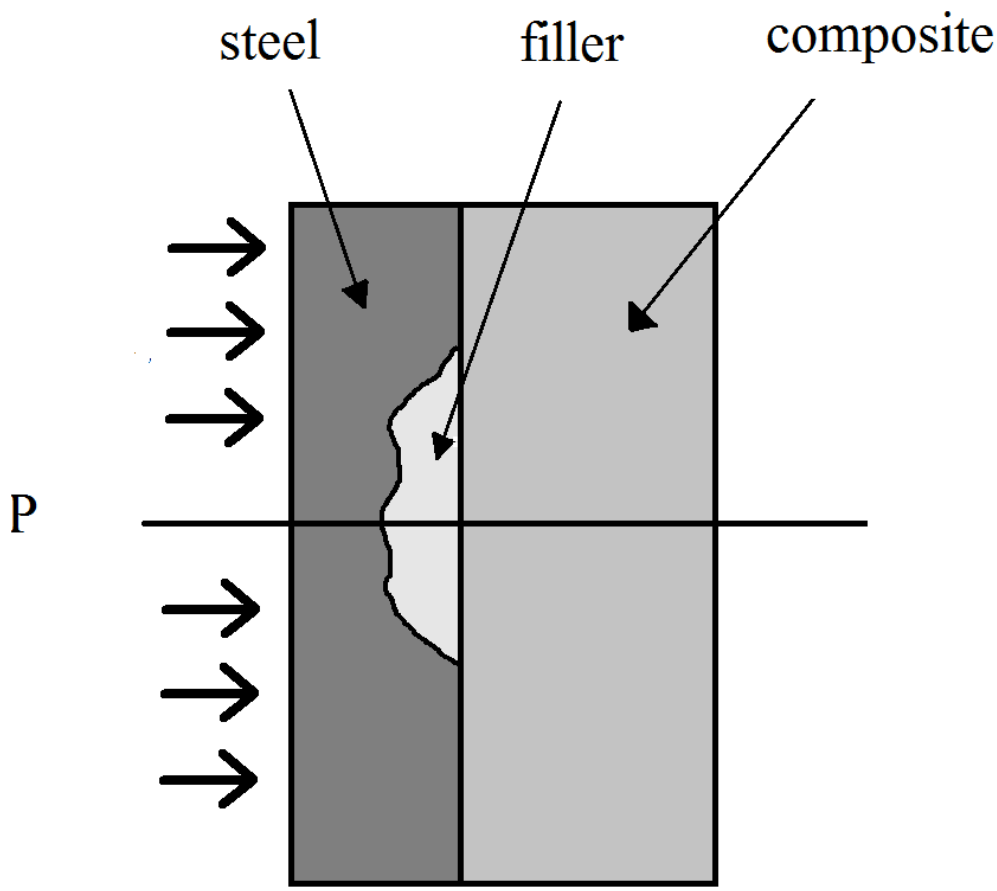


Figure 1. Zoomed in section of pipe with filler and composite wrap.

Before the installation of a composite wrap, the site is prepped by cleaning the surface of the pipe to a near white finish and then filling the corroded area with an epoxy filler. The composites are usually installed in one of two ways. Most repair systems use either carbon fiber or E-glass as the fibers in a resin matrix. There is strong cost savings pressure to use cheaper E glass over carbon fiber. The fibers are strong (1700 Mpa or 246 ksi) but brittle, they benefit from being surrounded in a protective matrix that toughens the resulting lamina. Each fiber is 10 microns in diameter and spun together on the order of the thousands and woven together into mats. For pipe repair the composite needs to be primarily strong in the hoop direction (along the circumference). Pipe repair composites are primarily unidirectional laminates for this reason.

After aligning the fibers, the mat is introduced into a liquid resin that will cure into a hard matrix around the fibers. When and how the resin is applied usually varies based on manufacturer. The resin can be partially cured in house and require a final cure in the field by being exposed to air or water; this is called prepreg. Another way to cure the composite is to mix the resin on site and apply the resin as the mats are being wrapped around the pipeline; this is called wet lay up. These methods represent a majority of methods used to install pipeline composite repairs.

1.2.2 Installation Pressure Research in Steel Sleeve Repairs

Chapetti, Otegui, Mandfredi, and Martin [3] published a full scale analysis of the stress state in metal sleeve repairs on gas pipelines. While the paper only addresses metal welding and not composite wraps, it addresses the resulting stresses from applying the repair to a partially pressurized pipeline. The authors state,

> It has been shown that keeping the gas pressure in the pipe near the normal operating values leads to lower stresses in the sleeves after they are welded…As soon as the pipe is pressurized again, very high hoop and longitudinal stresses are added to the sleeves, while the pipe material inside the repair remains almost unstressed…Although the choice of pressure during repair could be used as a tool to control stresses in pipe and sleeve and to optimize the integrity of the repair, pipe pressure during repair welding must be reduced at present due to safety reasons.

This reasoning is central to the ASME PCC-2 and ISO 24817 standards reduced composite thickness requirements at higher installation pressures. These findings validate that live pressure does play a role in the wrap's stress state. As a whole, Chapetti et al [3] was more focused on establishing safe steel sleeve procedures like buying high quality materials and proper welding procedures as this research was commissioned after a catastrophic steel sleeve failure. Installation pressure findings were just a byproduct of testing, not a main goal and as such did not present any more findings relating to it.

1.2.3 Analysis of Defect Width in Composite Repairs

Duell, Wilson, and Kessler [4] sought to compare how carbon composite wraps react to various widths of circumferential defects. They compared finite element models to real world testing. Their results showed that the width of the defect did not affect the composite repair's ultimate burst pressure as the filler putty served as a good strain interface between steel and composite wrap. They also showed that using ANSYS modelling predicted the burst pressure within 4% of the actual experimental pressure.

While the Duell et al., [4] publication addressed a significant part of pipeline research that hadn't been closely examined before, their research methods left something to be desired. Firstly, their theory section details ASME PCC-2 Substrate Yields equation as central to dictating the recommended composite thickness for their FEA model and experimental tests, but then does not use these numbers in either their FEA model or experimental tests. They use a value that is 25% thinner and likely not to pass code inspection at design pressure. For the composite thickness they do use (and don't explain how they got that number) they laud their FEA model for being within 4% accuracy of their tests, but do not highlight that they only compared two samples [4]. They give four sample burst pressures based on the defect width, but only report two experimental results without mentioning why there are missing data points. Also in their material properties they explain that their composite laminate material properties test produced +/- 12% variation with a 95% confidence interval. The small sample size, missing data points, and large variance in their material properties all detract from their claim that ANSYS came within 4% of the burst pressure. Furthermore, while their theory section derives ASME PCC-2 Equation, they do not produce any ANSYS or experimental data to test the wrapping at live pressure. This paper's greatest contribution lie within their solid theory section and ANSYS model. Their application of both leaves more to be desired.

1.2.4 Burst Pressure Modelling in Composite Repair

Freire, Veiera, Diniz, and Meniconi [5] published on the effectiveness of composite repairs with many types of defects as tested with various composite wraps with the goal of comparing models to real world pressure tests. Freire et al., [5] goals were to understand how composite layers withstand pressure, compare various reinforcement systems using experimental testing, and test if modifying the Remaining Strength Factor can be used to quantify composite wrap quality. Each pressure vessel was subjected to a rigorous pressurization test simulating what each composite wrap would be expected to perform across its life. The test cycle is quoted below. Each test vessel was wrapped at its maximum allowable operating pressure for a defect that was 70% corroded (30% remaining steel thickness).

> The design pressure considers the specified minimum yield strength of the pipe material decreased to 72%. The second test cycle reached 100% of the pipeline design pressure and was continually increased to 138.9% thereafter. If the specimen were able to withstand this pressure for 4 hours, the pressure would be released to 0 so that external inspection of the specimen could be performed and that the plastically deformed gages could be reset to zero. The third test cycle comprised 10 pressure cycles of 100% design pressure. The fourth pressure cycle consisted of a destructive test for the specimens that survived the first three test cycles.

The composite wraps of choice ranged from wet hand lay up, to precured, to prepreg partially precured. No composite wrap passed all the tests for both external and internal testing. Nonrepaired specimens saw greater burst pressures than would be calculated using Barlow's equation and some repairs just marginally beat the non-repaired specimens by 10-20%. The following long quote highlights their results.

> Test results showed that three repair systems allowed the pipes to reach the original design pressure. These repairs withstood pressures above those that ruptured similar specimens that were not repaired. However, only three of the eleven tested specimens passed the proposed protocol of pressure tests. Besides, none of the repair systems were approved in all strength verification tests for both internal and external defects.

While this conclusion infers that composite wraps can easily withstand burst pressure tests but not elevated cycling tests, there are some drawbacks to this method. One of the failure criterion is to pass a four hour 138% maximum operating working pressure (MOWP) test. Theoretically, all designs should fail a 138% MOWP pressure test as that is the pressure at which the pipe is designed to burst at. The cycle testing failure analysis is a valid test though. The 138% MOWP pressure testing showed the substrate steel yielding and then only elastically deformed when subjected to 100% cyclical tests. This highlights the elastic theory PCC-2 assumes to be governing the yielded substrate. One of the biggest concerns about this experiment is the lack of using codes to initially define the composite thickness. For the sake of consistency 25 millimeters was used across all composites, but the composites had varying material properties, some three times stiffer in the hoop direction than others. This confounds the failure criterion tests as samples that were less stiff failed in the most rigorous test the 138% MOWP four hour hold.

Despite the detracting factors, this experiment shines some light on the issue of ASME's level of conservative design when considering lifelong failure causes like fatigue and internal erosion.

1.2.5 Failure Pressure Estimations for Corroded Pipelines

Matherson L. da Silva and Heraldo da Costa Mattos [6] publication they look at the localized effects of defects when modelling the burst pressure of the pipe. Various defect sizes and locations were modelled with ASME B31G, RSTRENG .85, and the BG/DNV failure criterion [6]. These three failure criterion use different assumptions and correction factors to estimate burst pressure. All of them rely on imprecise, localized measurements around the defect site. These equations are designed to estimate the remaining strength of pipes that are not being rehabilitated. The equations were then modified and used to estimate the burst pressure of pipes that have composite wraps attached to them. Experiments for both damaged and rehabbed pipes were run.

Their results show that modifying ASME B31G and RSTRENG equations by including the ultimate tensile stress instead of the flow stress resulted in higher expected burst pressures than the unmodified equation. ASME B31G tended to most accurately model the experimental burst pressure for damaged pipes. The modified RSTRENG and the BG/DNV models tended to overestimate the burst pressure, leading to less conservative designs. For composite wrapped pipes the modified ASME B31G averaged underestimating the burst pressure by almost a factor of two. The modified RSTRENG criteria underestimated the burst strength by only 10% [6]. Silva and Mattos [6] published an overall produced a sound report that did an in depth analysis of localized factors effecting the burst pressure. My only critique of their paper is not following any code procedure in deciding a composite thickness. As a focused academic endeavor, the paper did not need to address this issue but its concerns still remain.

1.2.6 Live Pressure Analysis of Composite Repairs

The final paper to be reviewed is Saeed, Ronagh, & Virk, [7] publication focused on using finite elements to model how the installation pressure effects the composite thickness. ASME PCC-2 and ISO 24817 allow for a reduction in thickness of the required composite as installation pressure increases. The theory is that the composite bears less load and the underlying steel bears more if the wrap is installed at higher installation pressures[7]. This is supported directly in the findings of Chapetti et al., [3], and present in the data in Freire et al., [5]. Saeed et al., [7] was the first publication to examine if the code's thickness allowances create composite repairs that are non-compliant with the codes own guidelines for max hoop strain. Both ASME and ISO dictate that any composite wrap cannot exceed an averaged hoop strain of 0.25% at design pressure. Using ABAQUS, Saeed et al., [7] tested a range of live pressures (0%-100%) at incremental, reduced steel wall thicknesses (30-80% missing wall thickness) [7]. Unlike previous research articles that only looked at individual sections with localized damage and epoxy fillers, the whole pipe was reduced in steel thickness. This is allowable because the purpose of this research wasn't to validate burst pressure calculations, but see if live pressure had any effects on max strain limitations. Also, previous research by Freire et al., [5] support that the stress state of the composite isn't influenced by the circumferential width of the defect.

Within Saeed et al., [7] the following equations were used to generate the composite thicknesses necessary within their finite element model. Each equation is iteratively solved for the composite thickness, tc, as the hoop strain within the composite, ε_c, is mandated to be a maximum of 0.25% hoop strain. As previously explained, reestablished below as Equation 1 is the ASME PCC-2 Underlying Substrate Yields Equation 5 (Section 3.4.3.2, 2015). Presented as Equation 2 is the corresponding ISO 24817 Underlying Substrate Yields Equation. Presented as Equation 3 is a derived equation that expands the circumstances in which ASME PCC-2 Underlying Substrate Yields Equation 6 should be used to all live pressures.

$$\varepsilon_c = \frac{PD}{2E_c t_c} - s_y \frac{t_s}{E_c t_c} - \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{1}$$

$$\varepsilon_c = \frac{PD}{2E_c t_c} - s_{y(72\%)} \frac{t_s}{E_c t_c} - \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{2}$$

$$\varepsilon_c = \frac{PD}{2E_c t_c} + s_y \frac{t_s}{E_c t_c} \tag{3}$$

In all equations the remaining symbols represent: the final design pressure P, outer steel diameter Ds, composite Young's modulus Ec, composite repair thickness tc, yield stress of the steel Sy, steel thickness ts, live wrapping pressure P_{live}, and Young's modulus of the steel Es. For Equation 2, $Sy_{(72\%)}$ refers to 72% of the steel's yield strength. Two main variables examined are live pressure and steel thickness. Live pressure refers to the installation pressure of the composite upon the corroded steel. Steel thickness refers to the remaining wall thickness of the steel after the corrosion lost is measured.

The ISO equivalent, Equation 2, has the same form as Equation 1 but only allows 72% of the specified minimum yield stress to be modelled within the steel component. Equation 3 is also mentioned within the ASME and ISO texts as a viable solution for solving for the necessary minimum composite thickness when the installation pressure is zero. Saeed et al., (2015) presented a derivation that concluded live pressure plays no part in the composite strain values, thus Equation 3 was a viable solution for solving for the minimum composite thickness for any live pressure.

Figure 2 illustrates the material property assumptions for both the steel and composite on a stress strain diagram. Using elastic perfectly plastic steel and brittle composites, Barlow's formula can be used to describe the model. Saeed et al., [7] derivation for Equation 3 is detailed below.

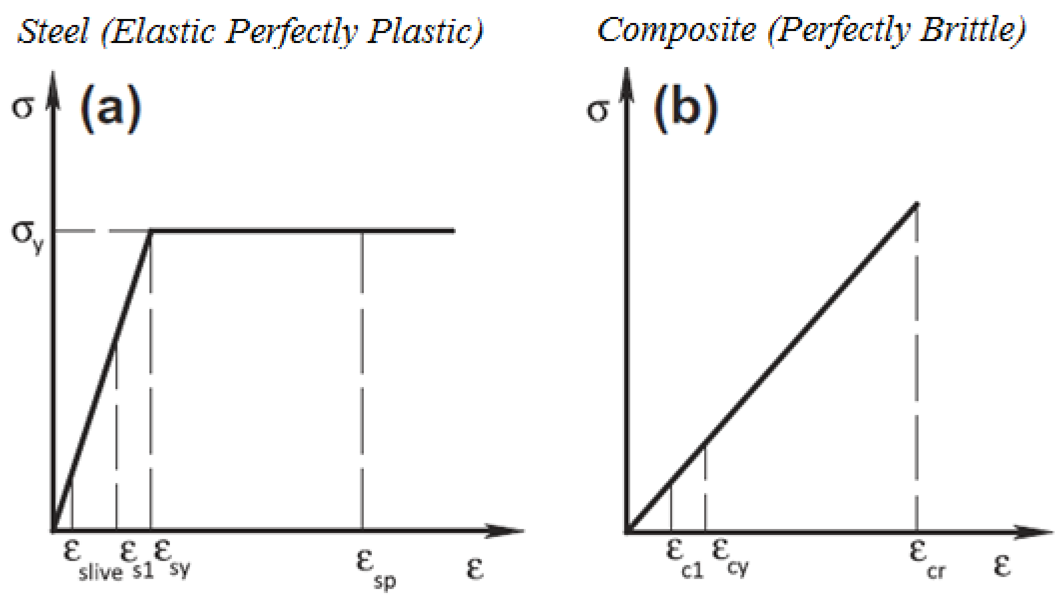


Figure 2. Saeed et al material assumptions.

The strain in the steel pipe wall at the installation pressure

$$\varepsilon_{slive} = \frac{P_{live} D}{2 t_s E_s} \tag{4}$$

The pressure is then increased to an arbitrary value (s1) greater than the live pressure but less than the steel yielding. The strain in the composite, ε_{c1}, is equal to the strain in the steel at an arbitrary point before yielding, ε_{s1}, minus the strain in the steel at installation pressure of the composite wrap, ε_{slive}.

$$\varepsilon_{c1} = \varepsilon_{s1} - \varepsilon_{slive} \tag{5}$$

Examining stress equilibrium at point 1, subscript s refers to steel and subscript c refers to composite:

$$2(\sigma_s t_s + \sigma_c t_c) = P_{\text{int}} D \tag{6]}$$

)Substituting Young's modulus and strain for stress in both the composite and steel:

$$2(E_s \varepsilon_{s1} t_s + E_c \varepsilon_{c1} t_c) = P_{\text{int}} D \tag{7}$$

Substituting Equation 5 into Equation 7:

$$E_s(\varepsilon_{c1} + \frac{P_{live} D}{2 t_s E_s}) t_s + E_c \varepsilon_{c1} t_c = \frac{P_{\text{int}} D}{2} \tag{8}$$

Rearranging Equation 8 for strain in the composite:

$$E_s(\varepsilon_{c1} + \frac{P_{live} D}{2 t_s E_s}) t_s + E_c \varepsilon_{c1} t_c = \frac{P_{\text{int}} D}{2} \tag{9}$$

$$\varepsilon_{c1}(E_s t_s + E_c t_c) = \frac{P_{\text{int}} D}{2} - \frac{P_{live} D}{2} \tag{10}$$

$$\varepsilon_{c1} = \frac{(P_{\text{int}} - P_{live}) D}{2(E_s t_s + E_c t_c)} \tag{11}$$

$$\varepsilon_{s1} = \frac{(P_{\text{int}} - P_{live}) D}{2(E_s t_s + E_c t_c)} + \frac{P_{live} D}{2 t_s E_s} \tag{12}$$

Pressurizing the pipe further to the steel yield point leads to the following equations as P1 and Py are linearly related (before steel yielding).

$$\varepsilon_{cy} = \frac{(P_y - P_{live}) D}{2(E_s t_s + E_c t_c)} \tag{13}$$

$$\varepsilon_{sy} = \frac{(P_y - P_{live}) D}{2(E_s t_s + E_c t_c)} + \frac{P_{live} D}{2 t_s E_s} \tag{14}$$

With the steel and composite relationship defined up to the steel yielding, Saeed et al., (2015) focuses on the design pressure. Saeed et al., (2015) defines that after reaching the steel yield point, any extra pressure is only resisted by the composite. This is due to the elastic perfectly plastic steel assumption. Furthermore, the steel strains and composite strains are locked together due to their adhesion. Saeed et al., 2015 continues the derivation with the following strain assumption. Where ε_{sp} equals the strain in the steel at the design pressure, ε_{sy} equals the strain in the steel at steel yielding, ε_{cr} equals the strain in the composite at the design pressure, and ε_{cy} equals the strain in the composite at steel yielding.

$$\varepsilon_{sp} - \varepsilon_{sy} = \varepsilon_{cr} - \varepsilon_{cy} \tag{15}$$

From Barlow's formula, the design pressure strain can be modeled as such.

$$2(\sigma_y t_s + \sigma_c t_c) = PD \tag{16}$$

$$2(\sigma_y t_s + \varepsilon_c E_c t_c) = PD \tag{17}$$

Ultimately, the strain in the composite can be rearranged as such.

$$\varepsilon_c = \frac{PD}{2E_c t_c} - \frac{\sigma_y t_s}{E_c t_c} \tag{18}$$

Which is equivalent to the ASME equation listed previously as Equation 3.

$$\varepsilon_c = \frac{PD}{2E_c t_c} - s_y \frac{t_s}{E_c t_c} \tag{19}$$

Saeed et al., [7] model consisted of an axisymmetric 2D model of 3,200 CAX4R elements (axisymmetric quadrilateral, reduced integration with hourglass controls). The live pressure was modelled by hand calculating the steel's circumferential expansion due to the hoop strain at live pressure and then offset the composite wrap by the calculated amount. The pipe is pressurized from zero psi to the design pressure in one step. Within the step the steel increases in diameter (due to hoop strain) at the live pressure (via the offset), makes contact with the composite, and then continues pressurizing onto the design pressure.

The results from Saeed et al., [7] highlight a few trends. The most noticeable trend is that using the ASME PCC-2 equation creates non code compliant wraps for all live pressures greater than 0%. ISO-24817 creates code compliant wraps for most scenarios but not all while at the same time producing overly thick wraps for increased live wrapping pressures. The recommended equation (ASME PCC-2 with no live pressure considerations) creates code compliant wraps, but doesn't create equally as conservative wraps across all wrapping pressures or reduced wall thicknesses.

Saeed et al., [7] concludes that his data supports using Equation 3 for all live pressure scenarios. Furthermore, he concludes that the ASME and ISO live pressure equations are not conservative enough and need to be revaluated.

After reviewing Saeed et al., [7] paper, two issues arise with their findings. Firstly, their derivation supported the removal of live pressure considerations by only focusing on the post steel yielding part of the loading dynamics. Saeed et al., [7] in great detail derives hoop composite strain up to the yielding point of steel, but does not carry any of these terms when deriving the post steel yielding behavior. By only examining the post yield steel behavior it is an obvious solution that live pressure doesn't play a role, because that section hasn't been included in the final analysis. In the next section, it is hypothesized why Saeed et al., [7] analysis only focused on the post steel yielding/design pressure regime. Secondly, Saeed et al., [7] finite element model (ABAQUS) is vulnerable to truncation errors. To simulate the composite wrap being joined to the steel surface after the steel already being pressurized, hand calculations were used to offset the steel from the composite. Doing any step outside of ABAQUS introduces the possibility of truncation errors, especially when dealing with small numbers such as .0025 strain rate. Furthermore, Saeed et al., [7] does not discuss which hand calculations they used. It is assumed they used Barlow's thin wall pressure vessel equations instead of Lame's formula. Lame's formula could account for the both the inner and outer wall of the steel and lead to higher data resolution. If Saeed et al., [7] did use Lame's formula, it should have been mentioned in their report. Internally, ABAQUS's finite element method would produce higher resolution data more in line with Lame's formula than Barlow's formula with a thin wall assumption.

Ultimately, Saeed et al., [7] presents a truncation vulnerable model and composite load derivation issues that leave gaps worth investigating. As this is the only published paper directly analyzing live pressure, it is worthwhile to test Saeed et al., [7] claims independently.

1.2.7 Note on Literature Review Papers Not Presented

There are a few papers important to the school of knowledge on composite wraps not detailed in this literature review. Their contributions, while important, were not directly related to research about live pressure installation for composite wraps. Important papers which discussed the thermal stresses with respect to burst pressure modelling are Esmaeel et al. [8], Goertzen and Kessler [9], and Mattos, Reis, Paim, da Silva, and Amorim [10]. Creep behavior in carbon fiber/epoxy matrices was examined in detail by Goertzen and Kessler (2006) [11]. Moisture concerns and effectiveness in water submersion of composite repairs were addressed by Keller et al., [12], Shamsuddoha, Islam, & Arayinthan [13], and Alexander and Ochoa [14]. These papers along with the previously discussed papers represent a well-rounded body of work in composite repairs.

1.2.8 Summation of Literature Review

The papers reviewed outline the current scope of published work focused on installation pressure of composite wraps for pipelines. Chapetti et al., [3] remarked that welding repair sleeves at higher internal pressures created less pressure on the repair sleeve itself. This was highlighted as a positive aspect that could create longer lasting pipe repairs. Chapetti et al., [3] noted that the high temperatures of welding did not allow for wrapping at high pressures as that increased the risk of explosion. As composite wraps became more popular, lots of research publications were produced analyzing the repaired pipe's burst pressure. Duell et al., [4] analyzed if circumferential defect width effected burst pressure, while Freire et al., [5], and Silva and Mattos, [6] independently analyzed various repair techniques, defect sizes, and field applicable predictions of burst pressure. While Freire et al., [5] mentioned live pressure considerations, it did not play a significant role in their results and was not the central part of their research. Up until this point, research tended to focus on better analysis of bursting pressure with no regard to code conformity. It is evident that researchers time and time again picked a 'conservative' or convenient composite thickness and then back solved for the pressure instead of using the codes to dictate the composite thickness like field service engineers need to do. Saeed et al., [7] initially set out to fill this gap in research regarding live wrapping pressure and hoop strain code compliance, but their research methodology is suspect and could benefit from more rigorous finite element modelling. This is especially true since Saeed et al., [7] calls for a complete overhaul of ASME PCC-2 and ISO 24817 based on their high strain value calculations.

Given this literature review, there is merit in researching how installation pressure effect code regulations with more detailed and rigorous modelling methods.

1.3 Review of ASME PCC-2

The following section theorizes how the ASME PCC-2 Equation 1 was derived and then discusses the limitations of an analytical model.

1.3.1 Derivation of ASME PCC-2 Live Pressure Equation

Equation 1's approach is to model the composite strain, ε_c, as a difference in hoop strain between the final design pressure, ε_{cr}, and the live wrapping pressure, ε_{clive}. Each of the strain values are taken from hoop stress equilibrium statements. The notation introduced by Saeed et al., (2015) is used throughout the paper for continuity.

$$\varepsilon_c = \varepsilon_{cr} - \varepsilon_{clive} \tag{20}$$

Using a modified version of Barlow's formula to accommodate the composite and steel walls, the equilibrium stress equation at the design pressure is:

$$2(\sigma_y t_s + \sigma_{cr} t_c) = PD \tag{21}$$

At the design pressure (ultimate pressure) the stress in the steel has yielded into the composite and is assumed to be elastic perfectly plastic. The stress in the composite can be substituted for strain values.

$$2(\sigma_y t_s + E_c \varepsilon_{cr} t_c) = PD \tag{22}$$

Rearranging for the composite strain at the design pressure reveals:

$$\varepsilon_{cr} = \frac{PD}{2E_c t_c} + \frac{\sigma_y t_s}{E_c t_c} \tag{23}$$

Evaluating the equilibrium equation at live pressure:

$$2(\sigma_{slive} t_s + \sigma_{clive} t_c) = P_{live} D \tag{24}$$

Substituting strain values in:

$$2(E_s \varepsilon_{slive} t_s + E_c \varepsilon_{clive} t_c) = P_{live} D \tag{25}$$

For the live wrapping pressure and higher, the steel strain, ε_{slive} , and the composite strains, ε_{clive} , are considered equal.

$$\varepsilon_{slive} = \varepsilon_{clive} \tag{26}$$

Substituting the composite strain back into the equilibrium equation and rearranging:

$$2\varepsilon_{clive}(E_s t_s + E_c t_c) = P_{live} D \tag{27}$$

$$\varepsilon_{clive} = \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{28}$$

Plugging Equations 28 and 23 back into Equation 20 results in ASME PCC2 Equation 1:

$$\varepsilon_c = \frac{P_D D}{2E_c t_c} - \frac{\sigma_y t_s}{E_c t_c} - \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{29}$$

It should be noted that the derivation of this equation has not been formally released by ASME. The previous derivation is a belief held by the author about how the committee arrived at their conclusion. At the very least, there are some reservations about how this equation was formulated. It does not account for any strain rate variations between the final design pressure and the live wrapping pressure. Most notably missing is the transition in strain dynamics as the steel yields.

1.3.2 Indeterminacy of Analytical Solutions

In the literature review section, it was highlighted that Saeed et al., [7] derivation of composite thickness equation without live pressure considerations only focused on the design pressure end condition (Equation 19). This was after deriving the composite strain up to the underlying steel substrate yielding (Equation 13). While Saeed et al., [7] did not comment on why the composite strain defined up to the steel yield point is not apparent in the composite strain at design pressure, there is good reason to avoid inserting the pressure at steel yielding, Py, into any final analytical equation.

$$\varepsilon_{cy} = \frac{(P_y - P_{live})D}{2(E_s t_s + E_c t_c)} \tag{30}$$

The pressure at the steel yield point is unknowable as an input. While the stress and strain states between the live wrapping pressure and the steel yielding are simple Barlow's formulas. The exact pressure at which the steel yields after the composite is applied is unknowable. It is not an input like P_{live}, nor is it an output like P_{design}. It exists indeterminately after the steel and composite have both been stressed at differing levels depending upon the steel thickness, composite thickness, and live wrapping pressure. The pressure at which the steel yields is a necessary term to fully model how the composite and steel dynamically share the load as the steel's Young's modulus is no longer a simple linear model. The change in steel's ability to withstand loading conditions plays an undeniable factor in the necessary composite thickness. Given the piecewise nature of the dynamic conditions, the composite strain at the design pressure could theoretically be modelled as such.

$$\varepsilon_c = \frac{(P_y - P_{live})D}{2(E_s t_s + E_c t_c)} + \frac{(P_{design} - P_y)D}{2(E_{s,y} t_s + E_c t_c)} \tag{31}$$

The Young's modulus of the steel, post yielding is represented by $E_{s,y}$. Equation 31 only serves to prove a point that it is necessary to include the pressure at which steel yields, P_y, as a reference point to account for the change in the composite strain after the steel yields. This equation still wouldn't be of practical use since the pressure at steel yield cannot be independently solved for. There would be two unknowns (composite repair thickness, and pressure at steel yielding) and one equation.

CHAPTER II

RESEARCH OBJECTIVES

The following chapter consists of three sections. Section 2.1 states the research objectives, Section 2.2 formulates the hypothesis, and Section 2.3 explains the creation of a parametric model.

2.1 Research Objectives

Saeed et al., [7] validation effort suffered from two major shortcomings. There are concerns with relatively large truncation errors because the steel strain at live pressure was hand calculated and then entered directly into finite element analysis. Furthermore, their theory stating there is no need for live pressure considerations goes against the body of literature and brings into question their suspect derivation. Given these issues it is worthwhile to establish finite element results using a model that simulates all steps inside the model. Using finite element analysis through ABAQUS, the objective is determine whether ASME PCC-2 and ISO 24817 result in code compliant composite wraps for corrosion levels between 30%-80% of the steel wall and live wrapping pressures between 0%-100% of the pipe's max operating working pressure.

Furthermore, there has been no published effort to establish what composite thicknesses perfectly relate to 0.25% hoop strain for each live pressure and corrosion thickness. The results of this research can be used to provide a means to solve for the composite thickness at each corrosion percentage/live pressure scenario that results in

0.25% hoop strain. An additional objective is to create a parametric model that results in more efficient modelling of composite thicknesses that includes live pressure considerations.

2.2 Formulation of Hypothesis

Any effort to analytically model the strain in the composite needs to directly address the model's dynamic change at the steel yielding point. Since the steel yielding point is not knowable as an input within an analytical solution, a parametric method is introduced. Constants were added to each term of Equation 1 (ASME PCC-2 with live pressure considerations) to best fit the true composite thicknesses that results in 0.25% composite hoop strain. The true composite thicknesses for each testing scenario that result in perfect 0.25% hoop strain are discerned through running an iterative loop of finite element model tests with changing composite thicknesses until each steel thickness and live pressure scenario outputs 0.25% strain.

It is the author's hypothesis that an parametric adaptation of ASME PCC-2 Live Pressure (presented as Equation 32) will create composite thicknesses closer to 0.25% hoop strain at design pressure than ASME PCC-2 or ISO 24817 equations alone.

2.3 Parametric Formulation

The parametric version of Equation 1, listed at Equation 32, introduces constants in front of every term within the composite thickness equation. These terms allow for each section's influence on the governing equation to be visualized. The constants were chosen to be simple values and include higher order terms. Including higher order terms would introduce inconsistencies and obscure meaning from any findings of better fitting results. When comparing to the true optimized results, the parametric data was visually inspected to be better fitting than previous iterations. It was not deemed necessary to run a method of least squares regression to find the absolute closest match with large amounts of significant digits. The purpose of this parametric model is to gain general insight into how the model reacts to various constants. The method of least squares method was also ruled out because the model should err on the side of overly conservative composite thicknesses. A simple least squares method could not guarantee that all data points would be slightly greater than the optimized results, as some points would be lower. Recommending composite thicknesses thinner than the optimized values would generate non code compliant wraps that exceeded the 0.25% hoop strain limit. The parametric equation is presented below.

$$\varepsilon_c = \mathrm{A} * \frac{PD}{2E_c t_c} - B * s_y \frac{t_s}{E_c t_c} - C * \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{32}$$

Equation 32 dictates how thick the composite wrap needs to be. The equation is iteratively solved for the composite thickness, tc, as the hoop strain within the composite, ε_c, is mandated to be a maximum of 0.25% hoop strain. In Equation 32 the remaining symbols represent: the final design pressure P, outer steel diameter Ds, composite Young's modulus Ec, composite repair thickness tc, yield stress of the steel Sy, steel thickness ts, live wrapping pressure P_{live}, and Young's modulus of the steel Es. The first of three terms represents the strain in the composite at the design pressure. The second term represents the strain held by the thinned (corroded) steel. The third term represents the strain at the live pressure (composite wrap installation pressure). Two main variables are examined, live pressure and steel thickness due to corrosion wall thinning. Live pressure refers to the installation pressure of the composite upon the corroded steel. Steel thickness refers to the remaining wall thickness of the steel after the corrosion lost is measured. Each live pressure/steel thickness scenario creates an input into a finite element model of the pipe to be examined.

CHAPTER III

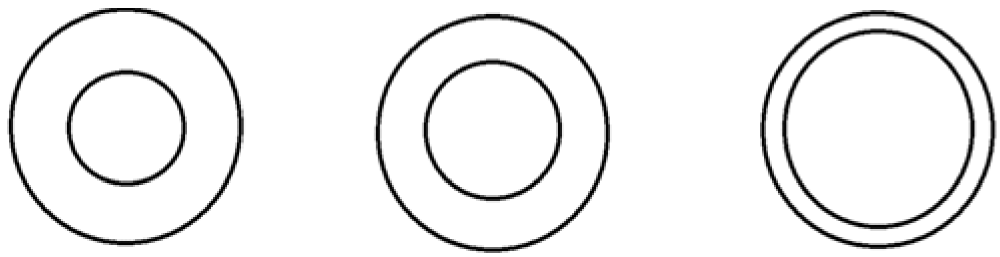
METHODOLOGY

The following section is broken up into four sections. Section 3.1 explains the variables examined within the research. Section 3.2 discusses the differences between the ASME and ISO standards. Section 3.3 explains the finite element model optimization process. Section 3.4 illustrates how the parametric model was best fit to the optimized finite element model.

3.1 Explanation of Variables

The two main variables studied were thickness of steel and live wrapping pressure. The live wrapping pressure (also referred to as installation pressure) as this represents the pressure when the composite wrap is installed. The variable representing steel thickness was declared to be corrosion depth as measured by the % nominal wall thinning (also referred to as % corrosion). It is assumed within this model that the corrosion could be modelled as an even removal of steel from the inner layer of steel. Thinning along the inner wall was selected as to not affect the OD and subsequently Barlow's stress calculations. In the real world corrosion can form on the outside of the pipe as well as the inside, but this model is indifferent to which wall is corroded due to the thin wall pressure vessel assumption. The finite element model examined various levels of corrosion ranging from 30% wall thinning to 80% wall thinning. Figure 3 shows how the corrosion was modelled.

** Not to scale, actual model is a thin wall pressure vessel with a OD/t ratio of 23.*



0% Wall Thinning 30% Wall Thinning 80% Wall Thinning

Figure 3. Visualization of reduced nominal wall thickness

The percent wall thinning was adjusted in increments of 10% nominal wall thickness. For the live wrapping pressure, increments of 25% between 0 MPa and 100% max operating working pressure (MOWP) were used for each thickness. The Maximum Operating Working Pressure is 72% of the Specified Minimum Yield Stress of the material (72% SMYS) For example, X42 steel has a MOWP (72% SMYS) of 1,778 psi at its nominal thickness of 0.375" and 12.75" outer diameter. When designating wrapping pressures for each nominal wall thickness, the MOWP is recalculated for each wall thickness. At 90% nominal wall thickness, the MOWP is 1,600 psi. For 50% nominal wall thickness the MOWP is 890 psi. For 20% nominal wall thickness the MOWP is 356 psi. For each nominal wall cases (30%-80% wall thinning), the live wrapping pressure is designated as 0, 25%, 50%, 75%, and 100% of the steel's specified MOWP.

3.2 ASME and ISO Standards Finite Element Model Testing

Using finite element software package ABAQUS, the ASME PCC-2 and ISO 24817 equations relating to composite thickness with respect to live pressure were examined to see if their resulting finite element model hoop strains matched the initial inputs of 0.25% hoop strain. The following four analytical solutions listed below were tested. They represent the ASME PCC-2 and ISO 24817 live pressure and no live pressure equations. The only difference between the ISO and ASME standards is that the yield stress in the steel in ASME standards is the true specified minimum yield stress (SMYS) for the material while the ISO standard is 72% of the specified minimum yield stress (72% SMYS). Equations that have been presented before are listed with their original equation number.

ASME PCC-2 Underlying Substrate Yields, Live Pressure:

$$\varepsilon_c = \frac{PD}{2E_c t_c} - s_y \frac{t_s}{E_c t_c} - \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{1}$$

ISO 24817 Underlying Substrate Yields, Live Pressure:

$$\varepsilon_c = \frac{PD}{2E_c t_c} - s_{y(72\%)} \frac{t_s}{E_c t_c} - \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{2}$$

ASME PCC-2 Underlying Substrate Yields, No Live Pressure Considerations:

$$\varepsilon_c = \frac{PD}{2E_c t_c} + s_y \frac{t_s}{E_c t_c} \tag{3}$$

ISO 24817 Underlying Substrate Yields, No Live Pressure Considerations:

$$\varepsilon_c = \frac{PD}{2E_c t_c} + s_{y(72\%)} \frac{t_s}{E_c t_c} \tag{33}$$

These four standards equations were each examined over a range of corrosion thicknesses and live wrapping pressures. For all standards, the steel wall thinning percentages were [30%, 40%, 50%, 60%, 70%, 80%]. 30% was the minimum as each steel pipe had 1.38 safety factor built in due to the MOWP requirements. Testing below 30% would not induce steel yielding. 80% wall thinning was the upper limit as that is the maximum corrosion that can be wrapped using this method, as mandated by ASME and ISO standards. For each corrosion percentage, five live wrapping pressure percentages were tested, [0%, 25%, 50%, 75%, 100%]. APPENDIX II displays all the tables illustrating the inputs for the four tested standards.

3.3 Finite Element Model Optimization Loop

The finite element model in conjunction with python scripting was used to reveal which composite thickness has exactly 0.25% hoop strain. Input values of steel thickness, composite thickness, live pressure, and design pressure were inputted into a parameterized model. The python script outputted the max stress and strain values for the steel and composite into an output file. The original ASME PCC-2 Equation 1 was used as a baseline to establish whether the composites needed to be incrementally increased or decreased in thickness. The optimization loop exited once a live pressure and steel thickness data point had a composite hoop strain between 0.25% and 0.2505%. It is worth mentioning that the 30% corrosion thickness data points were not optimized as the necessary composite thickness to establish 0.25% hoop strain ranged between 0.1mm and 0.0001mm. These values were seen as impractical and also unfeasible for pipe repairs.

3.4 Parametric Model Matching

Once the optimized finite element model established the true composite thicknesses, the parametric version of Equation 1 (presented as Equation 32) was evaluated for best fit. The following figures illustrate the curve fitting process. Equation 32, the parametrically modified version of Equation 1, is restated below for convenience.

$$\varepsilon_c = \mathrm{A} * \frac{PD}{2E_c t_c} - B * s_y \frac{t_s}{E_c t_c} - C * \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \tag{32}$$

The C values were adjusted first, with a tested range between 1.50 and 0.05. As the C value varied, the strain due to live pressure for each given corrosion level varied as well. Upon inspection, for each corrosion percentage the values only ‘pivoted’ around each 0% live pressure data point. Figure 4 depicts the C value variations. It was not observed that all data points translated up or down based on C value modification. At C=1.5, the subtracted strain due to live pressure tended to overshoot the optimized finite element model composite thickness, creating unsafe composites. With C=0.05, the subtracted strain due to live pressure was not enough to match the live pressure reduction rate produced by the optimized finite element model results. This leads to overly conservative composite thicknesses. With C=0.20, the subtracted strain due to live pressure tended to best match the optimized data. It is worth noting for lower corrosion levels (40% and 50%)

that no C value properly matched the discounting curve without overestimating the discount rate at higher wall thinning percentages. This issue is addressed in detail in the discussion section.

The B values were adjusted between 0.95 and 1.15. Modifying the B values resulted in a relatively consistent increase or decrease of all strain values. Figure 5 depicts the B value variations. With B=0.95, all composite thicknesses were overly conservative as compared to the optimized finite element model results. With B=1.15, all composite thicknesses were under conservative and produced unsafe composite wraps. By inspection, B=1.06 resulted in near perfect matching between the parametric model and the optimized finite element model results for the 0% live pressure data points of each corrosion level.

The A values were not illustrated, as changing the A value resulted in the same up and down translations of all data points that modifying the B value created. In an effort to discern the most knowledge from these parametric evaluations, it was easiest to only modify A or B and keep the other equal to one.

The best fit parametric model had values of A=1.0, B=1.06, and C=0.20.

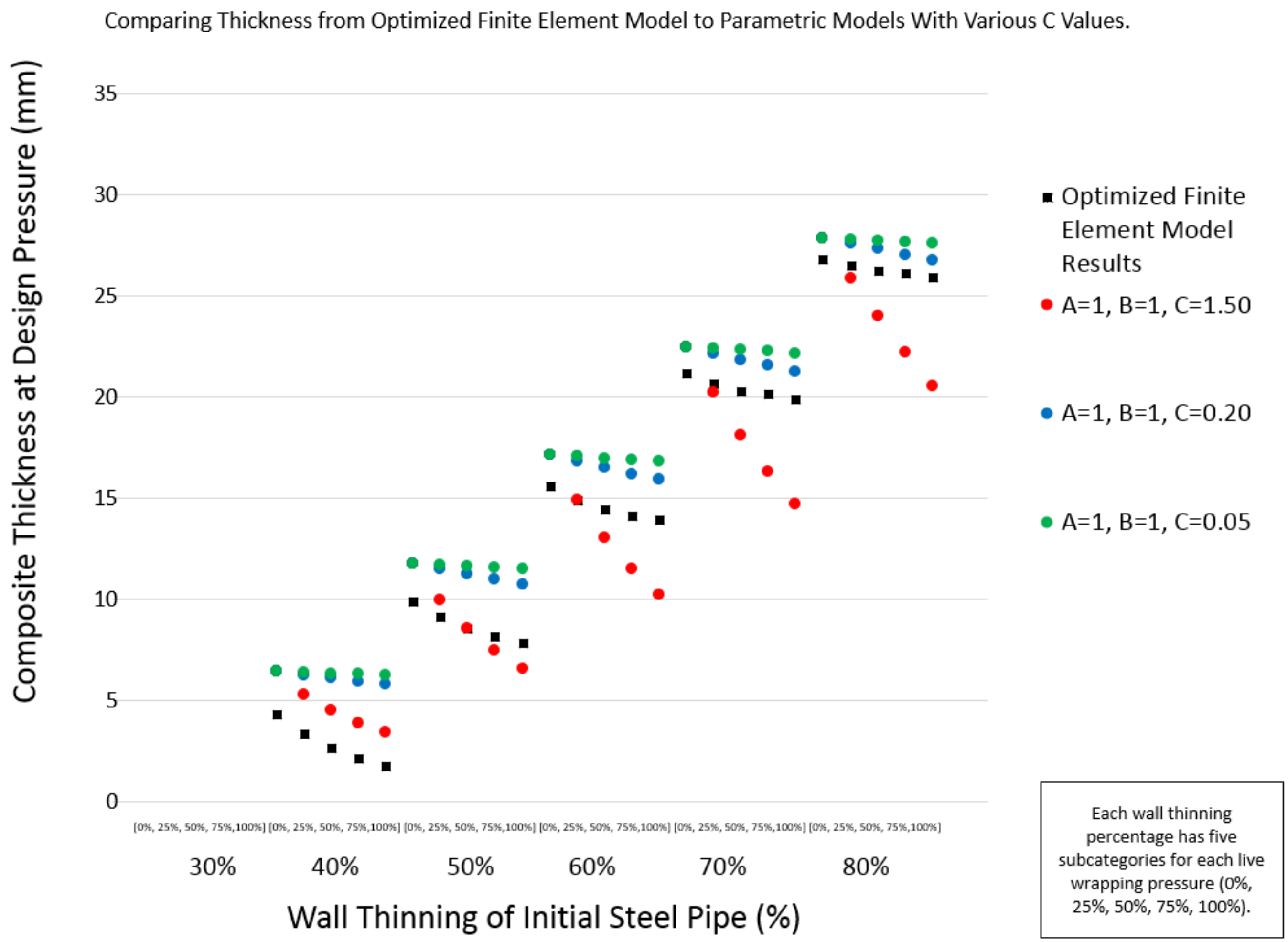


Figure 4. Comparing thicknesses from optimized finite element model results to parametric models, various C values.

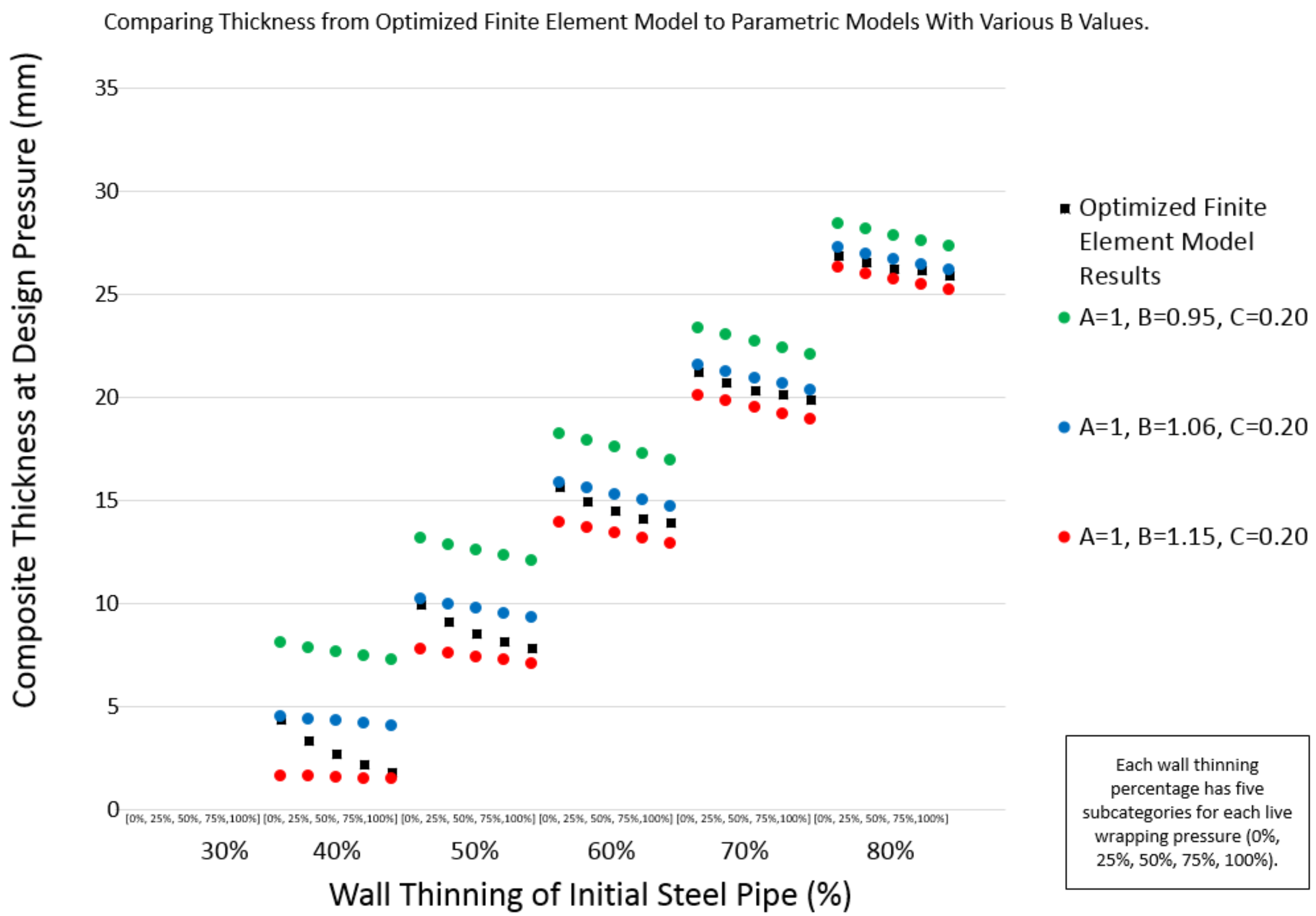


Figure 5. Comparing composite thicknesses from optimized finite element model results to parametric models, various B values.

CHAPTER IV

MODEL DEVELOPMENT

The following chapter discusses the creation of the finite element model using ABAQUS and details all the steps that went into the model creation. Section 4.1 introduces the finite element method. Section 4.2 establishes the finite element analysis. Section 4.3 walks through the creation of the quarter pipe with all the necessary assumptions.

4.1 Introduction to the Finite Element Method

The finite element method is used for finding approximate solutions to partial differential equations through steady state analysis or converting PDE's into ordinary differential equations which are solved [15]. The strength of the finite element method lies in the ability to discretize a complex region (or domain) into a collection of geometrically simple shapes (elements) [15]. On each element, the governing equation is formulated using a variational method. The finite element method first systematically divides the domain into subdomains where each subdomain represents a set of elemental equations relating to the problem. Once the subdomains are established, an assembly of elements is created based upon each elements continuity. The global system of equations is solved using the known general solutions for the problem. Solving each subdomain results in a matrix with the form $[K^e]\{c^e\} = [F^e]$ where each element will have more unknowns than elemental equations.

4.2 Introduction to Finite Element Analysis

Finite element analysis is the consolidation of elements using the finite element method into solvable matrices with boundary conditions to balance unknown variables. To evaluate a finite element model, the weak form needs to be established so reduce the derivative rigor required when solving. To solve for the weak form the approximate solution is sought over each element. Within a typical finite element Ω_e is assumed to be

$$u_h^e = \sum_{j=1}^{n} u_j^e \psi_j^e(x) \tag{34}$$

Where u_h^e are individual solutions of $u(x)$ at the nodes of element Ω_e and ψ_j^e represents the approximation function over the element.

The weak form for a linear analysis can be described as:

$$B^e(w,u) = \int_{x_a}^{x_b} \left(a \frac{dw}{dx} \frac{du}{dx} + cwu \right) dx \tag{35}$$

$$l^e(w) = \int_{x_a}^{x_b} \left(wfdx + w(x_a)Q_1 + w(x_b)Q_2 \right) dx \tag{36}$$

Where Q_n and u represent the boundary conditions at end points a and b.

$$u_h^e(x_a) = u_1^e \tag{37}$$

$$\left(-a \frac{du}{dx} \right)_{x=x_a} = Q_1^e \tag{38}$$

$$u_h^e(x_b) = u_2^e \tag{39}$$

$$\left(-a\frac{du}{dx}\right)_{x=x_b}=Q_2^e \tag{40}$$

In more universal terms, the weak form can be described simply as:

$$B^e(w,u)=l^e(w) \tag{41}$$

Solving for the coefficient (or stiffness) matrix for either linear or quadratic elements:

$$K_{ij}^e=B^e(\psi_i^e,\psi_j^e)=\int_{x_a}^{x_b}\left(a\frac{d\psi_i^e}{dx}\frac{d\psi_j^e}{dx}+c\psi_i^e\psi_j^e\right)dx \tag{42}$$

$$f_i^e=\int_{x_a}^{x_b} f\psi_i^e dx \tag{43}$$

After each element is approximated, a global nodal mesh is created to coordinate local element nodes to the global nodes. Forces and boundary conditions are introduced and the K matrix representing the whole model is inverted to solve for the unknowns at each node.

4.3 Quarter Pipe Analysis

The following section outlines the methodology of creating the finite element model as well as assumptions within the finite element model. The model was required to replicate the whole simulation of partially pressurizing the pipe, applying the composite wrap, and then continue internal pressurization to the design pressure. It was not considered acceptable to have any step done outside of ABAQUS or rely on hand calculations to simulate a step.

4.3.1 Model Simulation Methodology

Simulating the entire loading process entirely within the finite element software package of ABAQUS was accomplished by using the modelchange() command where the composite elements were deactivated during the initial loading and then reactivated for pressurization to the design pressure. The first step takes a bare steel pipe up to the designated live wrapping pressure. This is performed using the modelchange(), REMOVE command to remove the composite shell elements. After the pipe is pressurized in the first step, the composite shell elements are added back as a skin on the steel substrate by using the modelchange(), ADD STRAIN FREE command. These two steps accurately replicate the real life procedure of pressurizing the pipe and then adding the composite laminate. Using this method preserves the resulting stress and strain calculations within ABAQUS. One run produces all the necessary results while reducing the modeler's chance of error by having to enter or subtract strains manually.

The model is created in a python code. This allows access to the modelchange() command as it is not available in the ABAQUS CAE GUI. The python code is also parametrized to allow for each iteration to be run based off an input file. The necessary input variables (steel shell thickness, composite shell thickness, live wrapping pressure, and ultimate design pressure) are called from a tab delineated line in a text file. After each job is created and ran, the output database is analyzed for the stresses and strains of both the composite and steel during the live pressure step and the design pressure step. The python script searches for the element with the max stress for both the composite and the steel sections. The max steel stress element for the first step is outputted and both the max stress composite element and max stress steel element for the second step is outputted. These values are outputted to results text file. The iteration loop continues onto the next line of input text and repeats the process for all testing needs.

4.3.2 Quarter Pipe Model

The finite element model consisted of a 3D, deformable shell element, 90^{o} revolution of the steel pipe skinned with a composite laminate. Figure 6 depicts the pipe shell before loading and Figure 7 illustrates the pipe model after pressurization. Since the diameter to radius ratio is greater than 10, the thin wall pressure vessel assumption holds and shell elements were used. A 90 degree shell model was used because axisymmetric models could not easily accommodate the modelchange() command to simulate the initial steel only pressurization as well as shared nodes for skins.

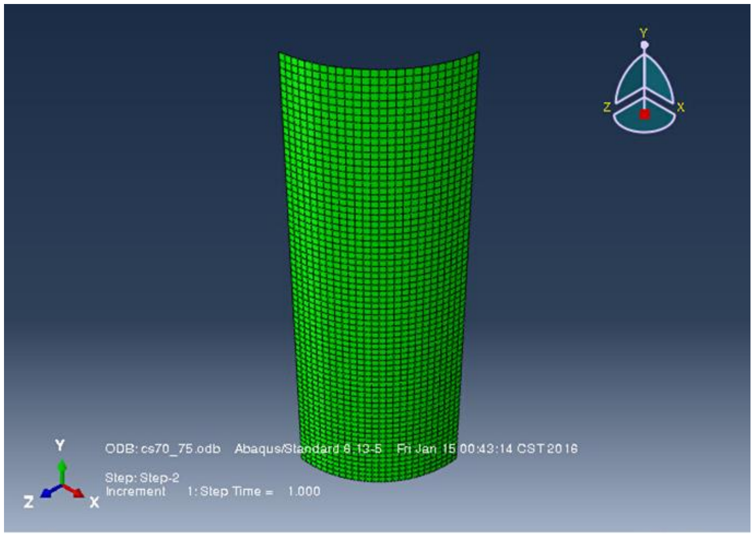


Figure 6. ABAQUS quarter pipe shell model, before loading.

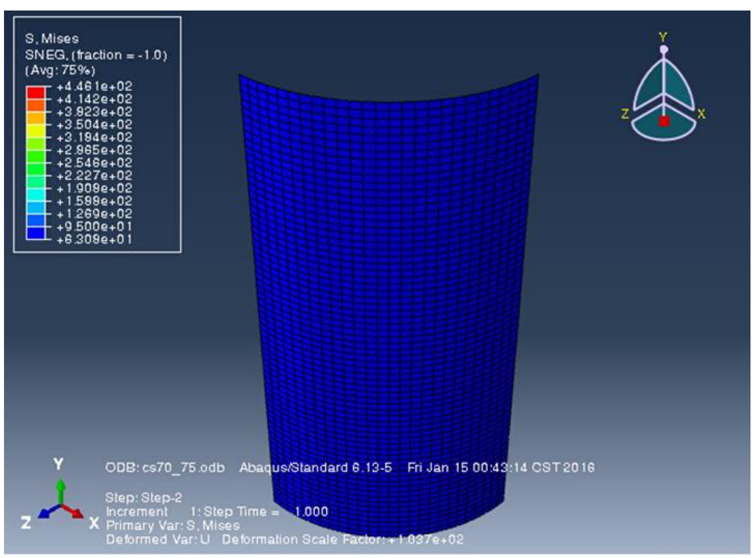


Figure 7. ABAQUS quarter pipe shell model, fully pressurized.

4.3.3 Material Properties and Assumptions

Table 1 illustrates the material properties for the steel pipe and composite considered for finite element testing.

Table 1.

Material Properties of Composite Laminate and Steel Pipe		
Composite Laminate Properties		
Modulus in hoop direction (E11)	23,800	Mpa
Modulus in axial direction (E22)	24,500	Mpa
Modulus in thickness direction (E33)	11,600	Mpa
Poisson's ratio (v31)	0.100	
Poisson's ratio (v32)	0.071	
Poisson's ratio (v12)	0.107	
Shear modulus (G31)	3,600	Mpa
Shear modulus (G32)	2,600	Mpa
Shear modulus (G12)	4,700	Mpa
Steel Pipe Properties (API 5L X65)		
Yield Strength	448	Mpa
Ultimate Strength	530	Mpa
Young's modulus	200,000	Mpa
Outside Diameter	168.3	mm
Wall Thickness	7.11	mm

The X65 steel is modelled with strain hardening in Figure 8 instead of the standard elastic perfectly plastic assumption. X65 steel was used as large majority of pipelines use this steel [16]. This is to gain better insight as to how the yielded steel helps share the load of the composite repair. The stress strain curve for X65 steel at ambient temperature was created using ASME Section VIII Div. 2 references.

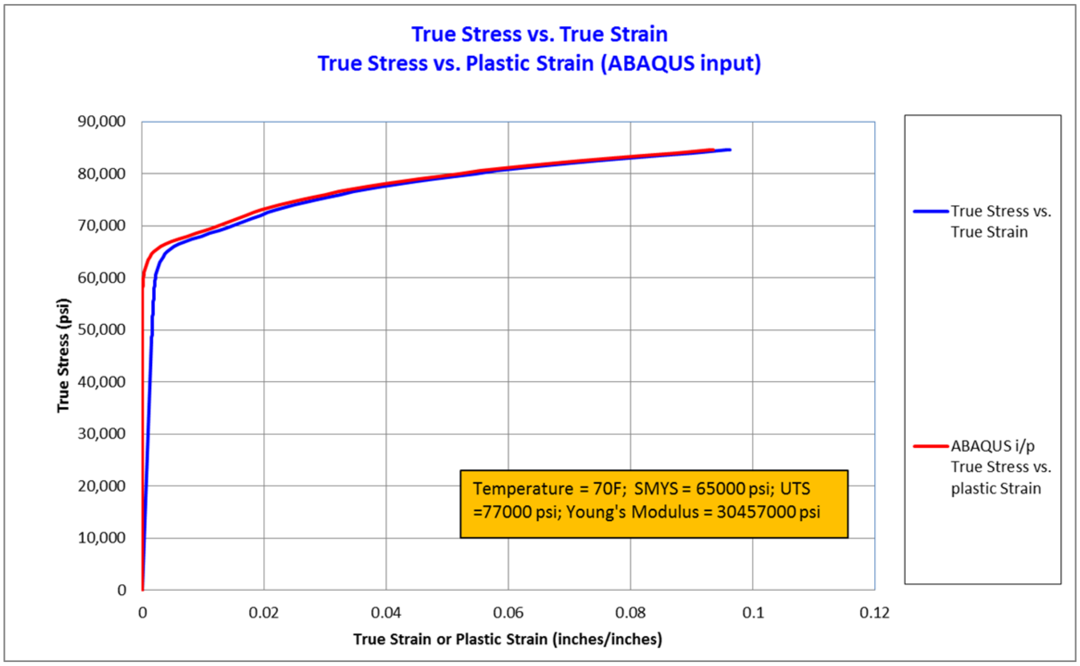


Figure 8. True stress vs. true strain of X65 steel.

For the composite's stress strain properties, the material is modelled as perfectly brittle with catastrophic failure. This is acceptable for uniaxial direction composites used in hoop stress based repairs. Within the finite element model, the composite itself is modelled as a homogenous solid with transversely isotropic material properties. Using lamina properties were not necessary as the composite is not loaded to failure. Both ASME and ISO standards require composites to have at least 1.0% strain to failure and loading conditions have a max hoop strain of 0.25%, a fourth of the required minimum strain.

4.3.4 Forcing Conditions on Quarter Pipe Model

The pressure was modelled as an internal load applied to interior surface of the quarter pipe model. In the first loading step, the pipe was pressurized to the live pressure dictated by the corrosion level and live wrapping percentage. In the second loading step, after the composite elements have been reactivated, the pipe is pressurized fully to the design pressure (27.25 MPa).

4.3.5 Boundary Conditions on Quarter Pipe Model

In an effort to make efficient use of the computational resources, only a quarter pipe was modelled instead of the full circumference of the pipe. Using symmetry boundary conditions, the finite element model can produce accurate results without the full pipe. The top and bottom of the pipe are both bound with Y-symmetrical boundary conditions (U2=0, UR1=0,UR3=0). These two boundary condition makes the effective length of the pipe three times the original pipe. It is good practice for the length of the pipe modeled to be at least twice as long as the diameter. These boundary conditions bring the effective axial length/diameter ratio up to six. The right edge of the pipe (viewer's orientation on Figure 6 and 7) is bound by a Z symmetrical boundary condition (U3=0, UR1=0, UR2=0) this effectively reflects the model another 90 degrees to the right. The left edge of the pipe (viewers orientation on Figures 6 and 7) is bound by an X symmetry boundary condition (U1=0, UR2=0, UR3=0). Similarly to the right edge boundary condition, the left edge boundary condition effectively simulate the whole pipe another 90 degrees to the left.

4.3.6 Elements

The quarter section pipe is modeled using a total 4,836 S8R shell elements (standard, reduced integration, quadratic) for the steel and composite wrap. In order to simulate the adhesion between the steel and composite, both sets of elements shared a common plane of nodes. The steel section was assigned to the ‘top plane’ of the shell while the composite section was assigned to the ‘bottom plane’ of the shell. Shell elements were used over 3D stress elements as the transverse stress was negligible due to the radius/thickness ratio being 23.6. Radius to thickness ratios greater than ten are commonly held to satisfy the thin wall pressure vessel assumption. Plate elements were not used due to the lack of external bending moments. Standard (implicit) element type was chosen over explicit as this model is not concerned with the time dependent dynamic steps. The stiffness based matrix controlling method of standard is better suited for this quasi-static model. The explicit element solver includes inertial forces and controls the mass matrix with time. This is not necessary for this model and could lead to unnecessary divergences within the model. Eight node quadratic elements were selected over four node linear shell elements to decrease the necessary elements needed to simulate a curved surface. Linear elements can only draw straight lines between node points and would require and extremely fine mesh to fully minimize mesh based approximation errors. Quadratic eight node elements have a node in between each element corner along element edges. This allows for quadratic approximation methods to fully capture the curved surface of the pipe and quickly reduce mesh based approximation errors.

4.3.7 Convergence Testing

Figure 9 and Table 2 illustrate the mesh convergence test on the shell model. The figure represents the von Mises stress on the composite shell at the design pressure. The data point 40% corrosion thickness, 25% live pressure was used. Any data point would have been suitable as the von Mises values were equivalent across all elements on the shell's surface. Both linear shell elements (S4R) and quadratic shell elements (S8R) were examined to test for convergence due to increasing nodes per element, as well as increasing elements per model. As expected, S8R elements produced more accurate results at lower mesh resolutions. This is due to the mid line nodes on the each element allowing for curved surfaces through quadratic values. Global mesh size of 30, S8R produced very close results to the fully converged mesh of global mesh size of 5, S8R. Though, (5,S8R) was selected to perform the testing analysis. (2.5, S8R) was extremely resource taxing and did not improve the outputting data over the less taxing (5, S8R) mesh and element combination.

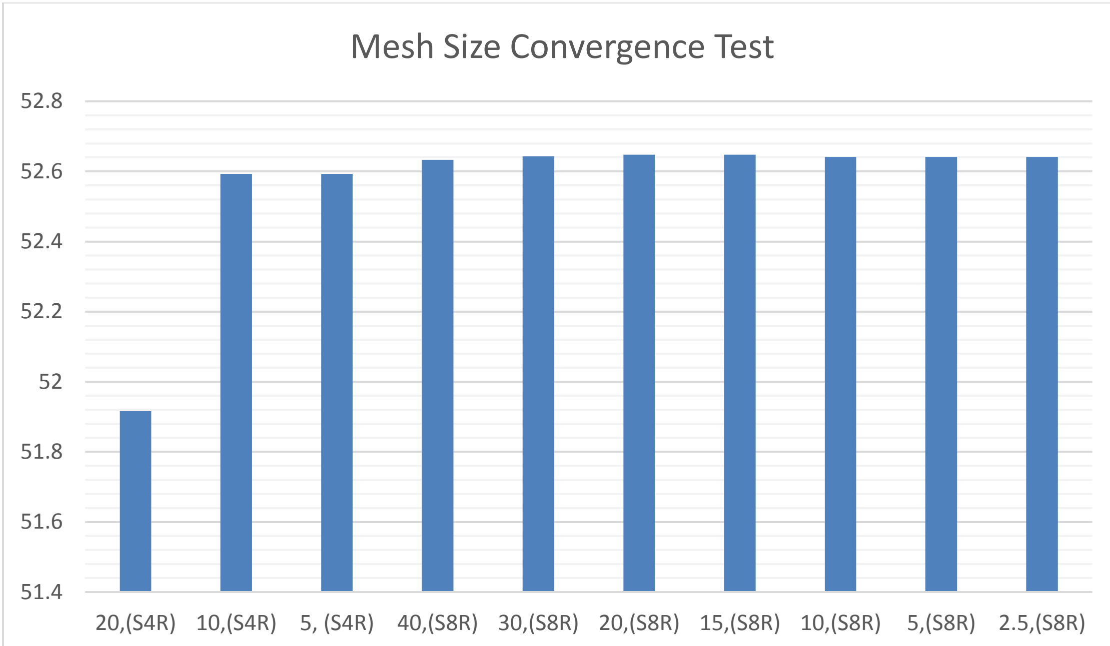


Figure 9. Mesh size convergence test.

Table 2.

Convergence Results

Mesh size, (element)	Von Mises Stress (Mpa)
20,(S4R)	51.9162
10,(S4R)	52.593
5, (S4R)	52.593
40,(S8R)	52.633
30,(S8R)	52.6434
20,(S8R)	52.6478
15,(S8R)	52.6481
10,(S8R)	52.6415
5,(S8R)	52.6416
2.5,(S8R)	52.6416

4.3.8 Model Benchmarking

Currently, only one published paper directly examines this research problem, Saeed et al., [7]. The published paper was not used as a definitive benchmark test due to research group's suspect finite element model. Furthermore, real world data that can be used to benchmark this model poses multiple issues. Firstly, this specific area of research is relatively new and live testing of composite wraps is reviewing papers from a small sample size. Of those composite wrap real world experiments, most research groups tended to focus on testing burst pressure models instead of live wrapping pressures. The remaining groups focusing on live pressure scenarios exclusively tested by machining localized defects instead of testing consistently thinner pipe like this model assumes.

This leaves benchmarking a computer generated model by analytical solutions. As already established in Chapter I, this specific model cannot be accurately analyzed using analytical solutions due to the steel yielding term being a fundamental lynchpin for describing strains.

The resolve the issue of error analysis, the hoop stress in the steel-only first step of the model was compared to the exact analytical solution (Barlow's formula) to see the error between the analytical stress and the elemental stress. As illustrated in Table 3, the finite element model averaged 0.00057% difference across all sample inputs tested when compared to the max stress elements to the Barlow's formula solutions. Barlow's formula is listed as Appendix I for reference.

Table 3.

Error analysis between analytical solution and ABAQUS mode for first steel step

Wall Thinning (%)	Live Pressure (%)	Analytical Hoop Stress (Mpa)	ABAQUS Model Hoop Stress (Mpa)	Percent Error (%)
	0%	0	0	N/A
	25%	80.62980	80.62956	0.000305%
30%	50%	161.25791	161.25742	0.000305%
	75%	241.88771	241.88696	0.000311%
	100%	322.51582	322.51483	0.000308%
	0%	0	0	N/A
	25%	80.62896	80.62869	0.000336%
40%	50%	161.25791	161.25737	0.000336%
	75%	241.88687	241.88606	0.000334%
	100%	322.51582	322.51474	0.000336%
	0%	0	0	N/A
	25%	80.64000	80.63955	0.000559%
50%	50%	161.28000	161.27933	0.000415%
	75%	241.92000	241.91792	0.000860%
	100%	322.56000	322.55725	0.000853%
	0%	0	0	N/A
	25%	81.36867	81.36832	0.000426%
60%	50%	161.25791	161.25723	0.000423%
	75%	241.88687	241.88585	0.000420%
	100%	322.51582	322.51447	0.000419%
	0%	0	0	N/A
	25%	80.64000	80.63920	0.000995%
70%	50%	161.28000	161.27879	0.000750%
	75%	241.92000	241.91838	0.000670%
	100%	322.56000	322.55795	0.000636%
	0%	0.0	0.0	N/A
	25%	80.64000	80.63905	0.001174%
80%	50%	161.28000	161.27870	0.000806%
	75%	241.92000	241.91776	0.000926%
	100%	322.56000	322.55740	0.000806%
				0.000571% Average

CHAPTER V

RESULTS

The following section presents and discusses the results from the all finite element models tested. There are three sections, Section 5.1 discusses the ASME PCC-2 and ISO 24817 testing. Section 5.2 discusses the optimized finite element model results and the best fit parametric model results. Section 5.3 compares the best fit parametric model results directly to the ASME PCC-2 models and the ISO 24817 models.

It is worth noting that 30% wall thinning results, while considerable in the real world, did not create good data for comparing models. Pipes are generally designed to operate at a max 72% specified minimum yield stress (SMYS) for safety reasons. Simulating 30% steel thinning produces uncharacteristically low results as compared to the rest of the data points as the steel has barely yielded if at all. For this research endeavor 30% corrosion results are shown for presentation’s sake when applicable but not included in the ABAQUS optimization loop efforts.

5.1 ASME and ISO Model Results

Both the ASME and ISO standards provide equations for solving for the necessary composite thickness with live pressure considerations as well as without. These four solutions all suggest different composite thicknesses for the same loading conditions. Figure 10 shows how varied each of the four solutions are across various corrosion thicknesses and installation pressures. ISO 24817, No Live Pressure Considered produces the most conservative composite out of the four solutions for all testing scenarios. The ASME PCC-2 with Live Pressure solution produces the least conservative composite wrap out of the four solutions. At relatively little corrosion (30%), the ISO equations are extremely conservative compared to the ASME PCC-2 equations. This is a result of the ISO equations artificially reducing the max stress carried by the steel from the actual yield stress to 72% yield stress. Modifying the max stress the steel can handle only serves to make the ISO formula a safety factor based design instead of attempting to match the originally inputted hoop strain. For the ASME standard, as the steel becomes thinner, the level of live pressure discounting of the composite thickness increases. At 40% corrosion, the subtracted strain due to the live pressure is considerably less than at 80%. Table 4 at the end of this chapter documents the exact strain values of all the standards and the parametric model.

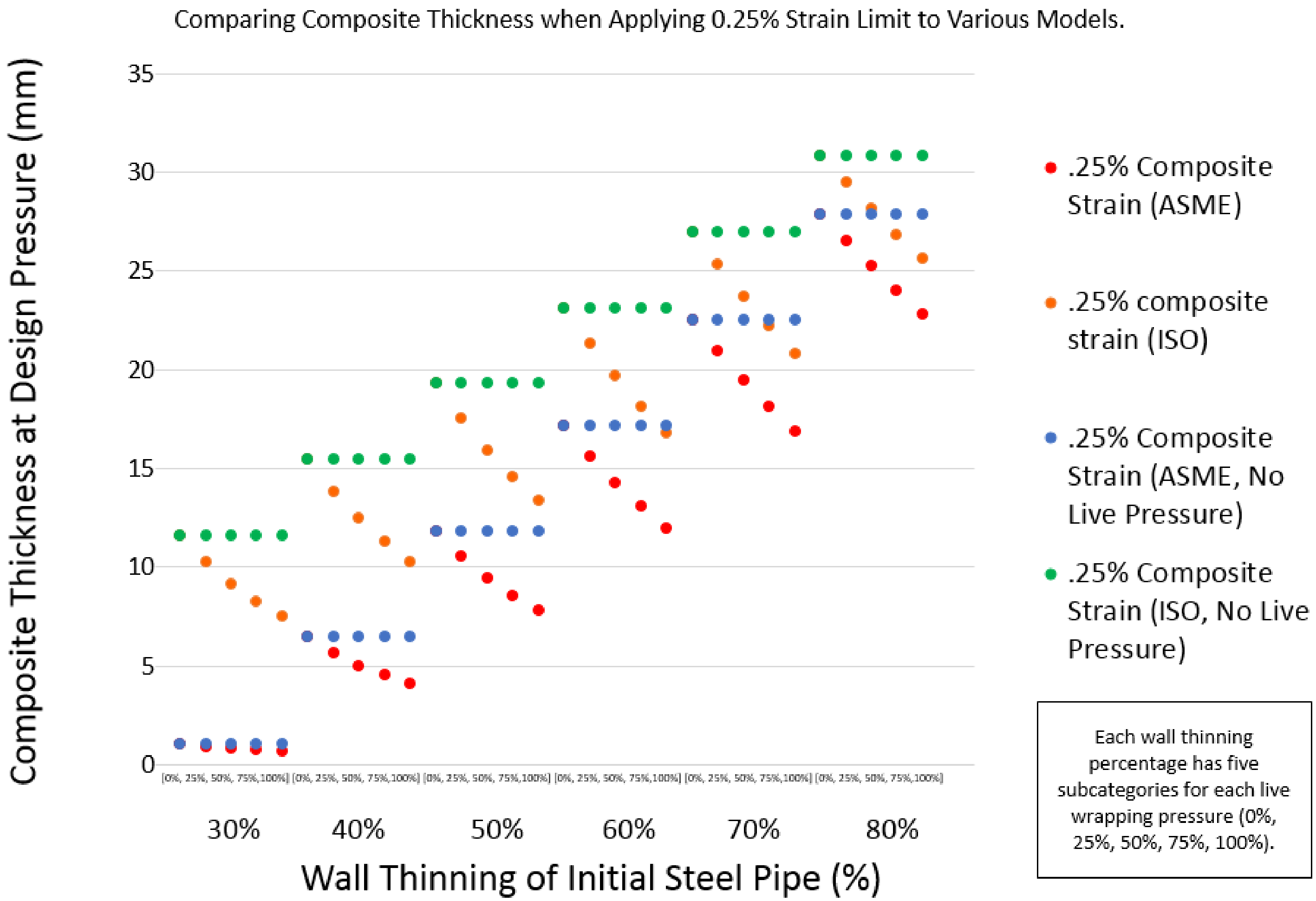


Figure 10. Comparison of resulting composite thicknesses modelled by ASME PCC-2 and ISO 24817 equations with and without live pressure.

5.1.1 Resulting Composite Strain Using ASME PCC-2.

In Figures 11 through 14, the resulting strains from the ASME and ISO composite thickness minimums illustrated in Figure 16 are presented. Each hoop strain in Figures 1 through 14 are outputs of finite element models for each corrosion and live pressure scenario as the pipe is pressurized fully to the original design pressure. If any of the ASME and ISO solutions were perfect, the finite element results would show consistent 0.25% hoop strain for every corrosion and live pressure scenario. This is not the case for any of the four models (ASME and ISO with and without live pressure). Only a few data points across the four models actually match the 0.25% hoop strain input for the governing equation. The following sections highlight how each model over estimates or underestimates 0.25% hoop strain. Models that show strains below 0.25% indicate that the composite for that specific corrosion level and wrapping pressure was overly thick. Strains above 0.25% hoop strain indicate that the composite thickness was too thin. Too thin composite thicknesses produces non code compliant composite wraps and increases the risk of pipe rupture. Too thick composite thicknesses cut into the economic advantages of composite wraps.

For Figure 11, the ASME PCC-2 Substrate Yielding equation (Equation 1) has less than 0.25% hoop strain at zero live pressure for each steel corrosion level. This means, for those data points, the composite thicknesses were overly conservative and too thick. Regarding the live pressure considerations, for 30% and 40% wall thinning wrapping at any live pressure creates increasingly over conservative thicknesses. This means that the

ASME PCC Substrate Yielding Equation does not discount the composite thickness nearly enough as the wrapping pressure increases. It is worth noting that for 30% wall thinning the recommended composite thickness is between 2.0 and 0.4 millimeters. The 30% wall thinning results can be disregarded as explained in the beginning of this chapter. For 50%-80% steel wall thinning there is a reversal on the effects of live pressure. The transition from creating more conservative wraps to creating less conservative wraps happens between 40% and 50% steel wall thinning due to corrosion. For 50%-80% wall thinning, an increase in wrapping pressure corresponds with a decrease in the conservativeness of the pipe. Some specific live wrapping pressures result in near perfect 0.25% strain matching (60% wall thinning, 50% live wrapping pressure), but this is quickly overshadowed as a significant amount of test cases exceed the 0.25% hoop strain maximum. The ASME PCC-2 Equation with live pressure considerations should not be recommend for pipe repairs as it is not conservative enough during high corrosion wraps and is also too conservative for lesser corroded pipes.

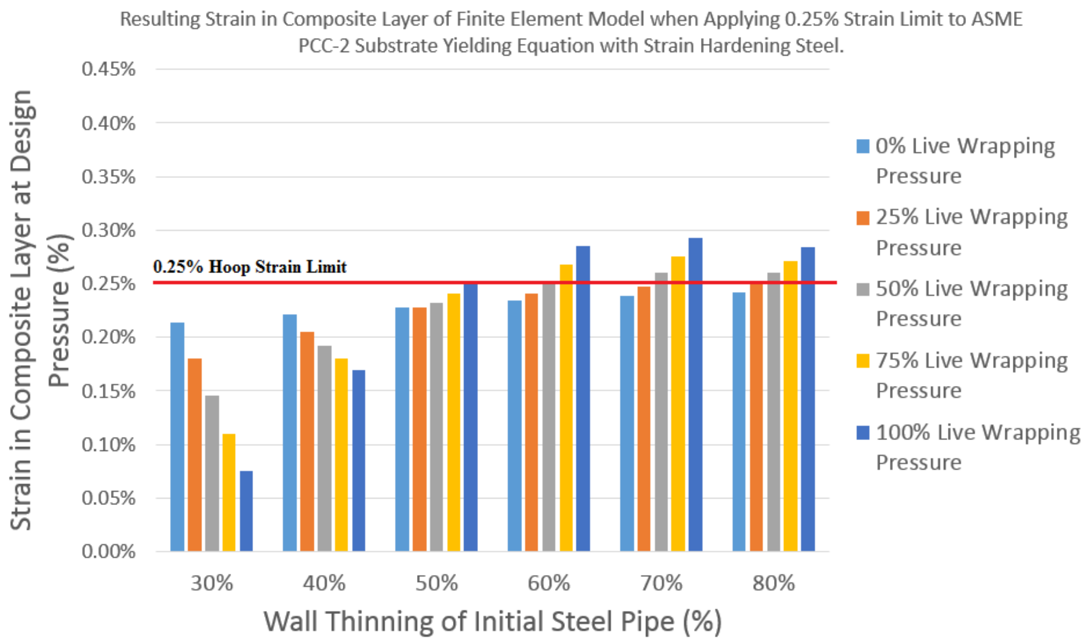


Figure 11. Strain in composite layer modelled with ASME PCC-2 substrate yielding equation with strain hardening steel.

5.1.2 Resulting Composite Strain Using ASME PCC-2 with No Live Pressure

Figure 12 illustrates the resulting hoop strains in the finite element models created with the ASME PCC-2 solution with no live pressure considerations. The results include a shadow of the Figure 11 results that include the live pressure considerations. This is so comparisons between the PCC-2 models with and without live pressure can be made. Without discounting for live pressure, all wall thinning percentages see a drop in hoop strain as the wrapping pressure increases. All tested scenarios are less than 0.25% hoop strain, but there is room for improvement. Testing without live pressure considerations highlights the fact that installation pressure plays a role in developing the necessary composite thickness. If installation pressure didn’t, the composite hoop strain would be equal across all live pressures for a given corrosion level. The results show that wrapping at higher wrapping pressure results in a need for a thinner composite as the hoop strain decreases. Between no live pressure consideration and PCC-2’s recommendation lies a composite thickness that can closely match 0.25% hoop strain.

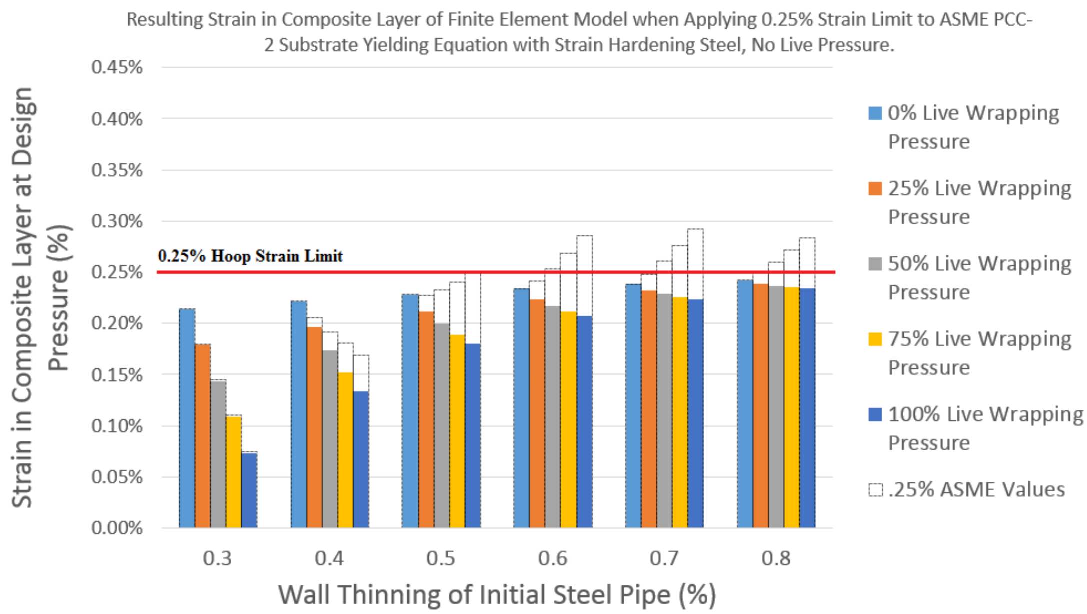


Figure 12. Strain in composite layer modelled with ASME PCC-2 substrate yielding equation with strain hardening and no live pressure.

5.1.3 Resulting Composite Strain Using ISO 24817

Figure 13 illustrates the resulting hoop strains in the finite element model created with the ISO 24817 solution with live pressure considerations. The results include a shadow of the Figure 11 results that include the live pressure considerations. This is so comparisons between the PCC-2 and ISO 24817 models can be made. The ISO solution follows the same trends as the PCC-2 live pressure equation. Both show transitions of conservativeness between 40% and 50% wall thinning. The ISO standard produces results under 0.25% hoop strain for every testing scenario except the most extreme case (80% wall thinning, 100% live wrapping pressure). While not all tested scenarios produce less than 0.25% hoop strain, all scenarios are less than the ASME PCC-2 counterpart. This is due to the only difference between the ISO and ASME standards; the ISO standard limits the max stress allowed to be carried by the steel. This difference creates thicker composites and thus less strain within each composite compared to each scenario's PCC-2 counterpart.

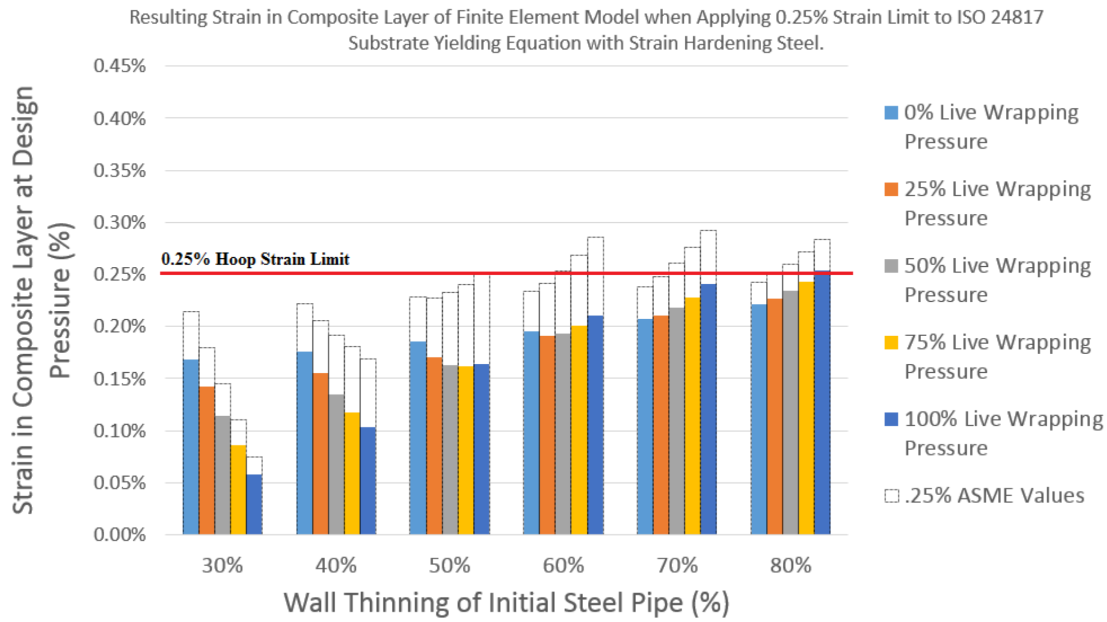


Figure 13. Strain in composite layer modelled with ISO 24817 substrate yielding equation with strain hardening.

5.1.4 Resulting Composite Strain Using ISO 24817 with No Live Pressure

Figure 14 illustrates the resulting hoop strains in the finite element model created with the ISO 24817 solution without live pressure considerations. The results include a shadow of the Figure 11 results that include the live pressure considerations. This is so comparisons between the PCC-2 live pressure and ISO 24817 no live pressure models can be made.

For every wall thinning and live pressure scenario tested, the ISO no live pressure model resulted in the most conservative composite wraps with lesser composite strain than the ASME live pressure model. While this is the only model meeting the minimum requirements of creating code compliant composite wraps, the ISO no live pressure model is overly conservative for all scenarios. Furthermore, as the wrapping pressure increases, each resulting composite thickness becomes increasingly overly conservative.

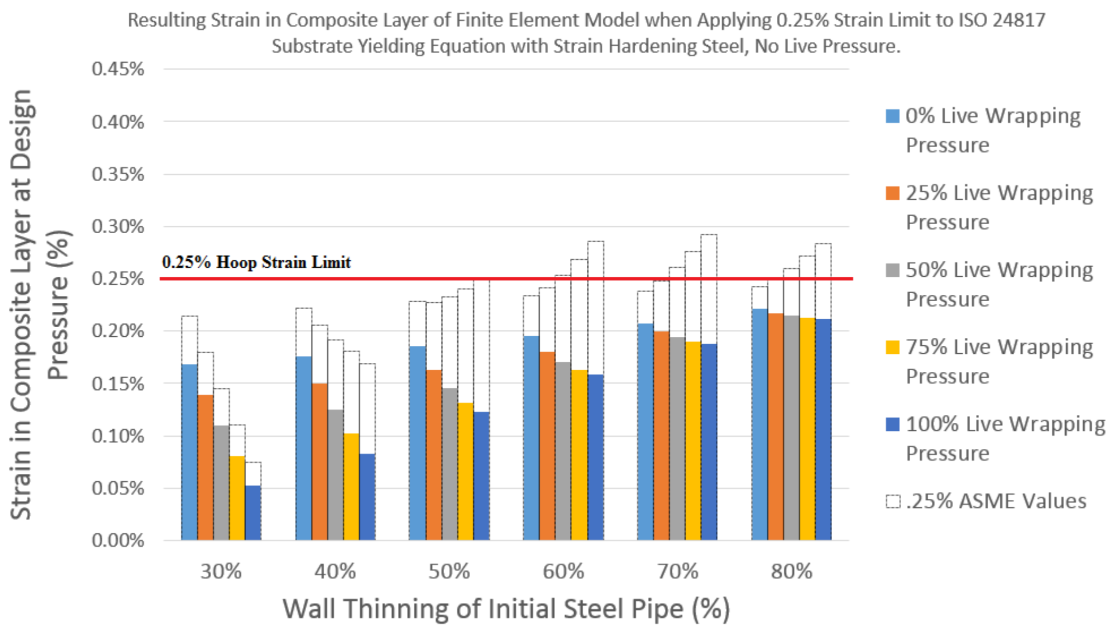


Figure 14. Strain in composite layer modelled with ISO 24817 substrate yielding equation with strain hardening and no live pressure.

5.2 Results from Optimized Finite Element Model and Best Fit Parametric Model

All four of the standards solutions did not produce 0.25% hoop strain values for all wall thinning levels and live wrapping pressures. The finite element software package ABAQUS and python scripting were used to search for the necessary composite thickness to equal 0.25% hoop strain. The resulting composite thicknesses were then used to fit a parametric model with constants in front each part of the ASME PCC-2 Live Pressure equation (Equation 32). Each of the constant's values were changed to output thickness values that best fit the optimized finite element model results that had perfect 0.25% hoop strain.

It was found that the parametric version of ASME PCC-2 Live Pressure created the best fit with values of A=1, B=1.06, C=0.20. The following section presents the results from the optimized finite element model and the best fit parametric model. The parametric equation (Equation 32) is reestablished below for easier referencing.

$$\varepsilon_c = \mathrm{A} * \frac{PD}{2E_c t_c} - B * s_y \frac{t_s}{E_c t_c} - C * \frac{P_{live} D}{2(E_s t_s + E_c t_c)} \quad (32)$$

5.2.1 Optimized Finite Element Model Results Equaling 0.25% Composite Strain

Figure 15 illustrates the resulting hoop strains in the finite element model created with optimized composite thickness loop. The results include a shadow of the Figure 11 results. Optimized results for 30% wall thinning were not presented as the finite element model optimization loop was recommending composite thicknesses less than 0.1 millimeters. 30% corrosion does not represent enough steel thinning to warrant a composite wrap solution using these equations. Table 4 showcases the finite element optimization loop results that generated perfect 0.25% hoop strains across every live pressure and wall thinning scenario.

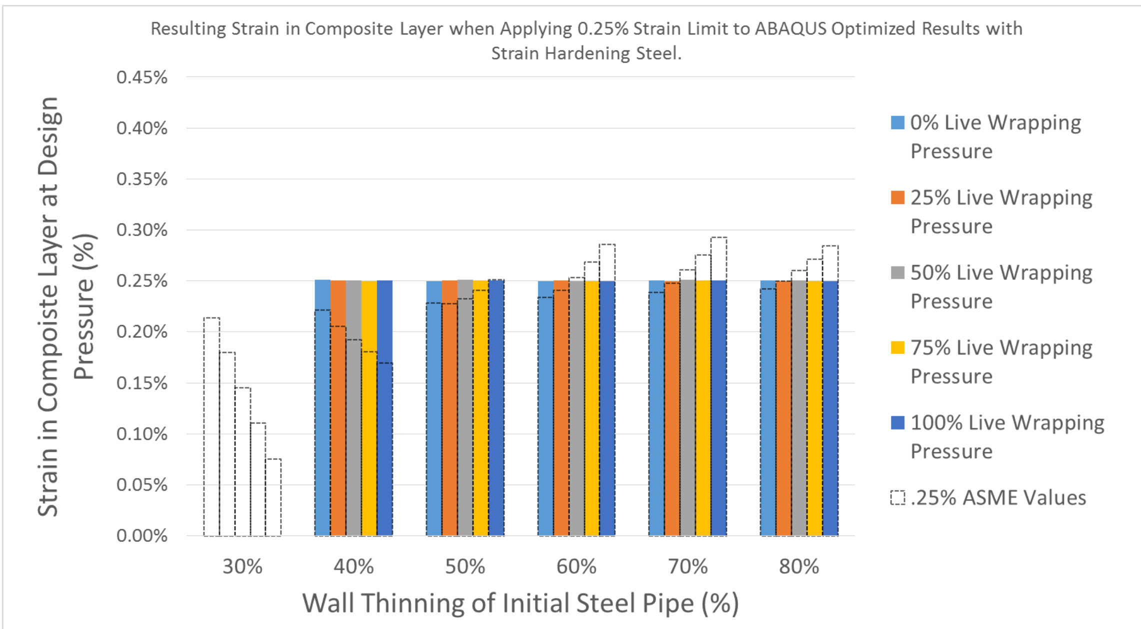


Figure 15. Strain in composite layer modelled by finite element model optimization loop.

Table 4.

Resulting ABAQUS inputs from optimization loop that equal 0.25% hoop strain

Corrosion Percentage (%)	Live Pressure Percentage (%)	Steel Thickness (mm)	Composite Thickness (mm)	Live Pressure (MPa)	Design Pressure (MPa)
	0%	4.977	-	-	-
	25%	4.977	-	-	-
30%	50%	4.977	-	-	-
	75%	4.977	-	-	-
	100%	4.977	-	-	-
	0%	4.266	4.3188	0.0	27.25
	25%	4.266	3.3337	4.0875	27.25
40%	50%	4.266	2.6519	8.1750	27.25
	75%	4.266	2.1589	12.2625	27.25
	100%	4.266	1.7642	16.3500	27.25
	0%	3.555	9.9200	0.0	27.25
	25%	3.555	9.1000	3.4067	27.25
50%	50%	3.555	8.5158	6.8134	27.25
	75%	3.555	8.1400	10.2201	27.25
	100%	3.555	7.8200	13.6269	27.25
	0%	2.844	15.5900	0.0	27.25
	25%	2.844	14.9034	2.7500	27.25
60%	50%	2.844	14.4469	5.4500	27.25
	75%	2.844	14.1000	8.1750	27.25
	100%	2.844	13.9000	10.9000	27.25
	0%	2.133	21.2000	0.0	27.25
	25%	2.133	20.6500	2.0440	27.25
70%	50%	2.133	20.2692	4.0881	27.25
	75%	2.133	20.1199	6.1321	27.25
	100%	2.133	19.8728	8.1761	27.25
	0%	1.422	26.8325	0.0	27.25
	25%	1.422	26.4900	1.3627	27.25
80%	50%	1.422	26.2176	2.7254	27.25
	75%	1.422	26.1000	4.0881	27.25
	100%	1.422	25.9000	5.4507	27.25

5.2.2 Parametric Model Based Off Optimized Finite Element Model Results

Figures 16 and 17 illustrate the resulting hoop strains in the finite element model created from the best fit parametrically modified ASME PCC-2 solution with live pressure considerations (Equation 32, where A=1, B=1.06, C=0.2) . The results include a shadow of the Figure 11 results that include the live pressure considerations. Figure 16 is presented against the ASME PCC-2 Live Pressure values, while Figure 17 is presented against the ASME PCC-2 No Live Pressure values. In Figure 16, the best fit parametric model does a better job of decreasing composite thicknesses that are under 0.25% hoop strain, and increasing the composite thicknesses for values over 0.25% hoop strain than any of the original standards analytical solutions. It is worth noting that the parametric formula does not perfectly match the optimized finite element results and thus does not result in perfect 0.25% hoop strains across all corrosion and live pressure levels. The 40% wall thinning scenario has the furthest departure from 0.25% hoop stress. This is expected as the 30% and 40% wall thinning sections are the steel thicknesses most strongly influenced by the steel yielding. The 30% and 40% wall thinning sections have the largest percentage of composite straining between the live wrapping pressure and the steel yielding. This is due to the excess of remaining steel material that prevents steel yielding at lower pressures when compared to thinner wall thicknesses. As stated previously, the inability to know the pressure at steel yielding directly will hamper analytical models from fully describing the composite's hoop strain at all wall thinning percentages.

Figure 17 shows a direct comparison of the best fit parametric model's improvements over the ASME PCC-2 No Live Pressure solution. For all comparisons within Figure 23, the parametric model offers a composite wrap more closely aligned with 0.25% hoop strain.

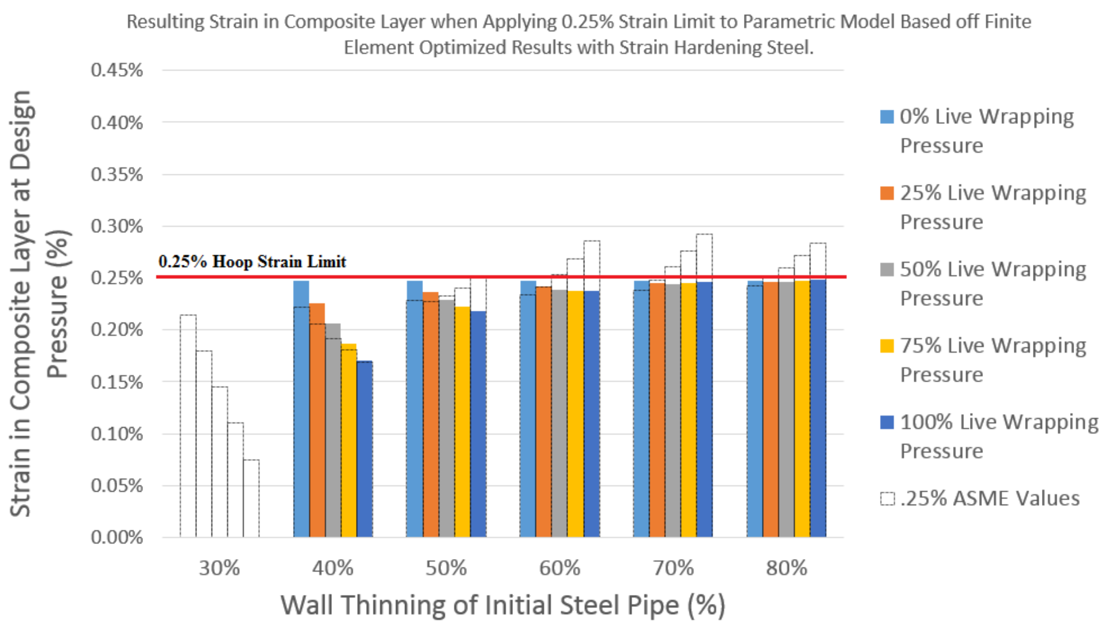


Figure 16. Strain in composite layer of the finite element model using the best fit parametric model, compared to ASME PCC-2 values.

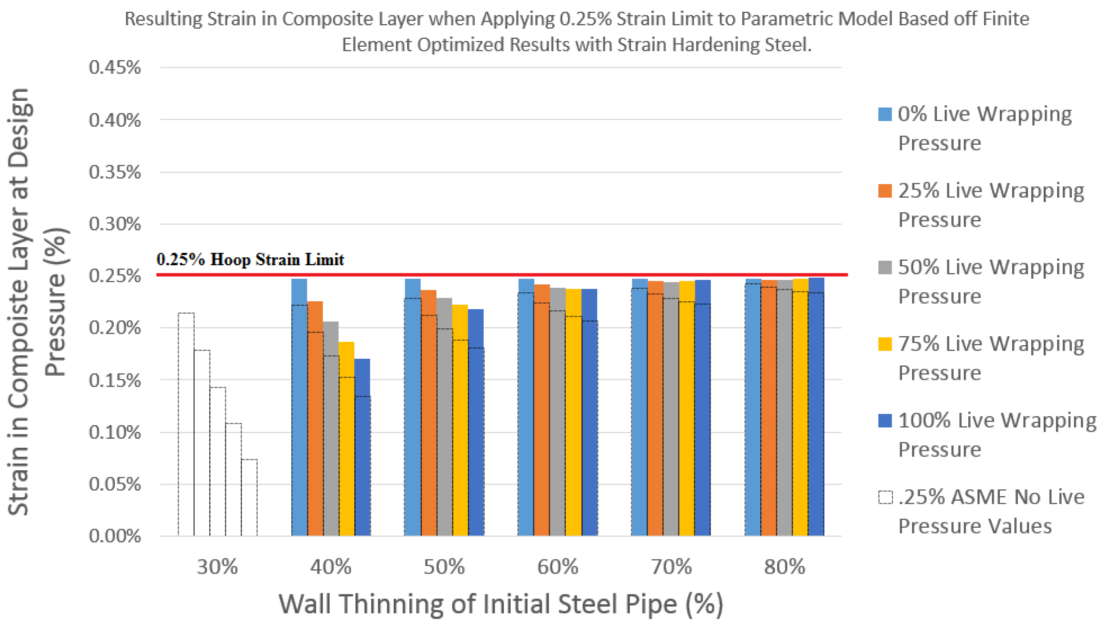


Figure 17. Strain in the composite layer of the finite element model using the best fit parametric model, compared to ASME PCC-2 no live pressure values.

5.3 Comparison of Best Fit Parametric Model to ASME PCC-2 and ISO 24817

As a visual, Figures 18 and 19 compare the output thickness of the best fit parametric model to the optimized finite element results and both ASME and ISO standards. The best fit parametric model tracks closer to .25% hoop strain more so than any of the ASME or ISO standards equation. Table's 5 and 6 showcase all the tested theories hoop strains and composite thicknesses side by side. Equation 32 resulted in all of the testing scenarios safely conforming to the max hoop strain limit. Furthermore the best fit parametric model produced the closest values to 0.25% hoop strain across all wall thinning percentages and live pressures. There is still not perfect agreeance between the best fit parametric model and consistent 0.25% hoop strains across every walling thinning level and wrapping pressure. The parametric model still produces considerably overly conservative composite wraps for greater than 50% live wrapping pressures at 40% wall thinning. Though, the parametric model was still the best performing method for 40% wall thinning, it still did not fully replicate the nature of the loading dynamics. The lack of hoop strain matching at 30% and 40% wall thinning rates points back to the model's inability to establish a pressure at steel yielding term. It is theorized that the 30% and 40% wall thinning percentages have considerable amounts of composite straining before the steel yields. All of the tested models do not establish clear methods of accounting for composite strain after live pressure but before steel yielding. The best fit parametric model had B values of 1.06 and C values of 0.20. These rates tend to suggest that the ASME codes need to reexamine the standards to account for these inconsistencies. It is theorized that the steel yielding term, Y_s, should be increased to a steel stress equivalent to no greater than 0.25% strain. This would better reflect the 6% increase in the steel stress term the parametric model recommends. As to why the C values need to be reduced to 20% of the current term, there is not as obvious of an answer at this time. Given no straightforward answer, the author chooses not to speculate but consider it a starting point for further research.

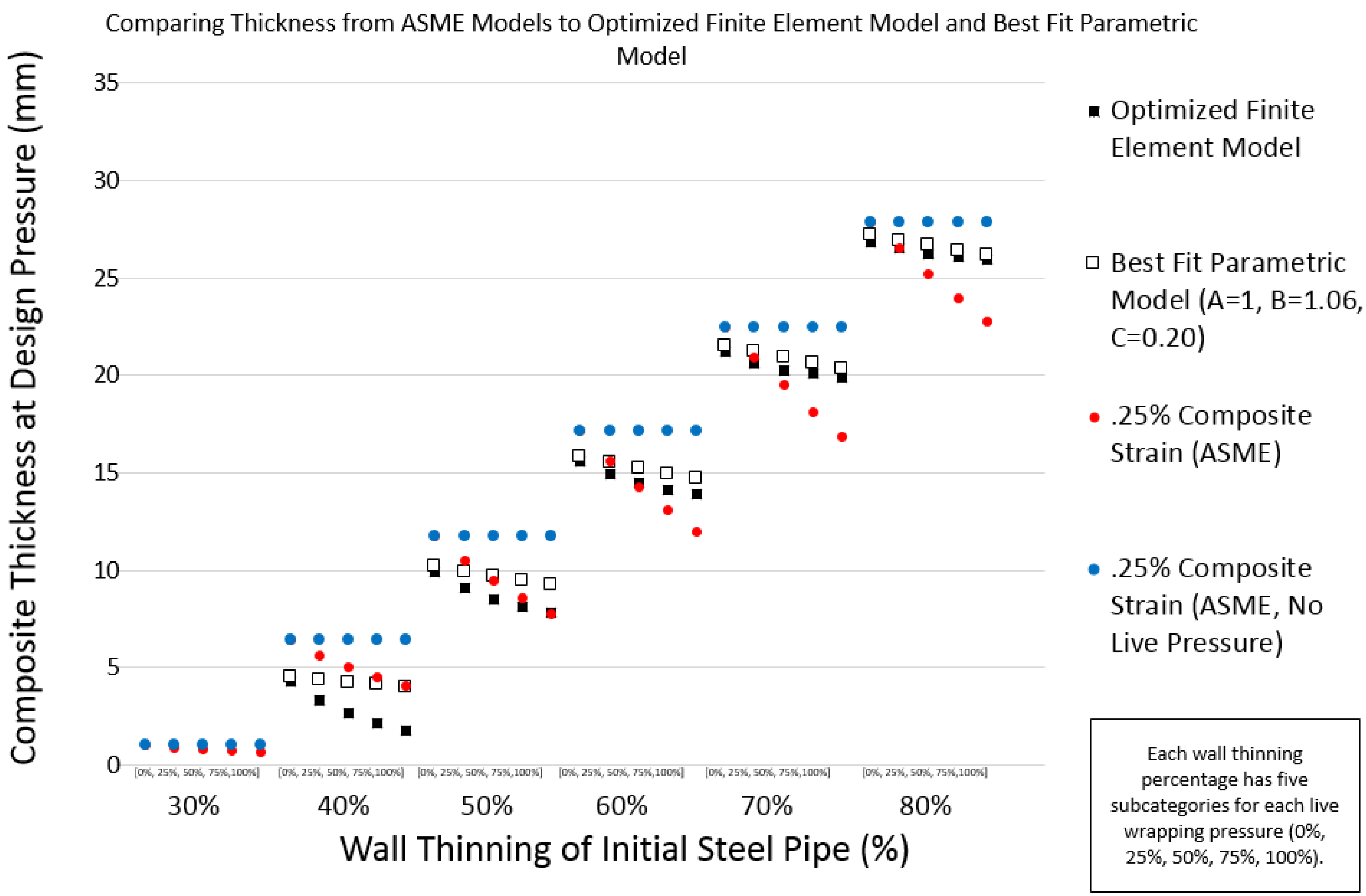


Figure 18. Comparison of composite thickness outputs from best fit parametric model to ASME PCC-2 equations as well as the optimized finite element model results.

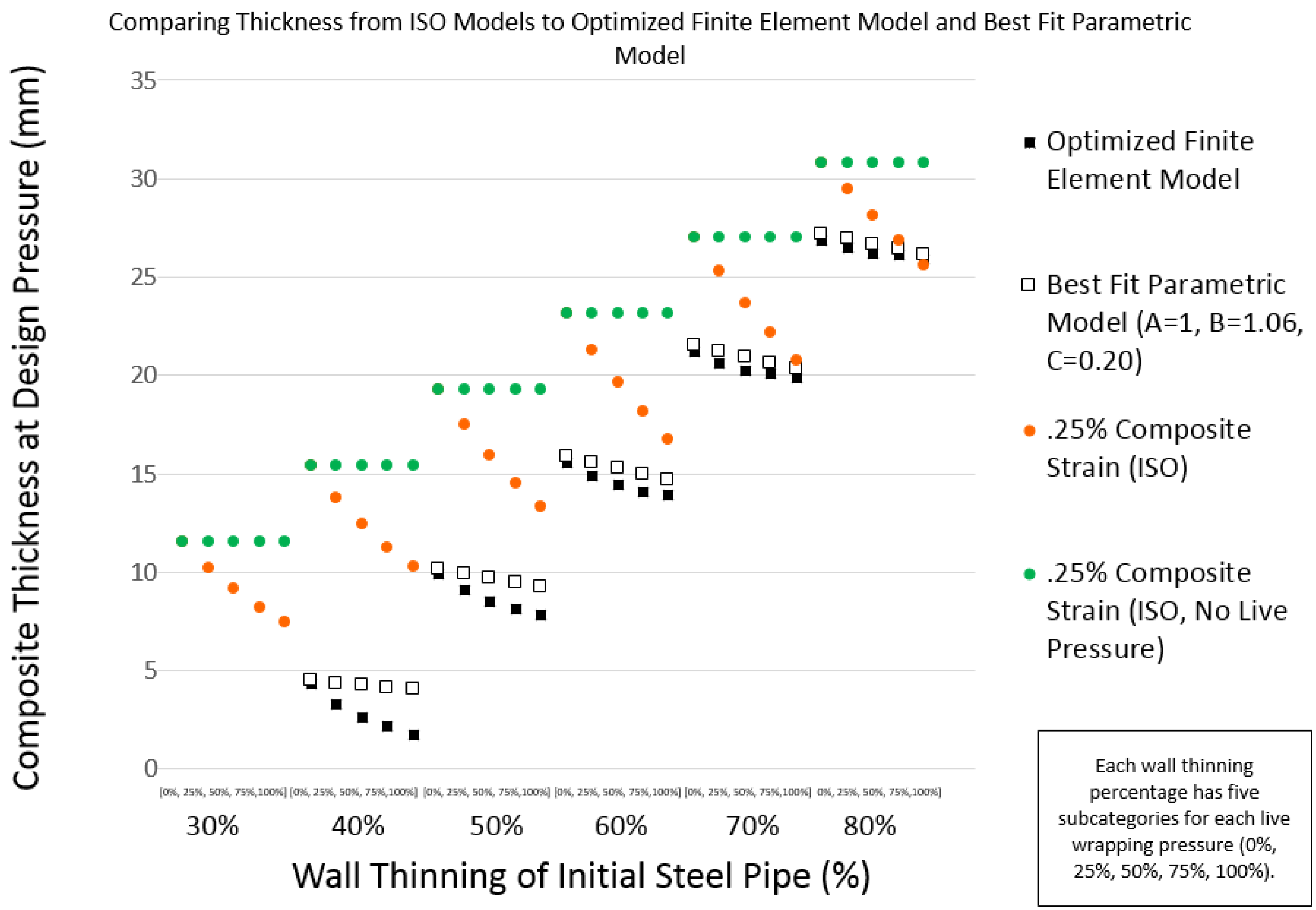


Figure 19. Comparison of composite thickness outputs from the best fit parametric model to ISO 24817 equations as well as the optimized finite element model results

Table 5.

Comparison of strain rates between ASME standard, ISO standard and parametric model

Corrosion Percentage (%)	Live Pressure Percentage (%)	ASME PCC-2, Live Pressure strain (mm/mm)	ASME PCC-2, No Live Pressure strain (mm/mm)	ISO 24817, Live Pressure strain (mm/mm)	ISO 24817, No Live Pressure strain (mm/mm)	Empirical PCC-2, Live Pressure (A=1, B=1.06, C=.20) strain (mm/mm)
	0%	0.2138%	0.2138%	0.1680%	0.1680%	-
	25%	0.1798%	0.1786%	0.1420%	0.1387%	-
30%	50%	0.1451%	0.1434%	0.1146%	0.1094%	-
	75%	0.1102%	0.1084%	0.0864%	0.0806%	-
	100%	0.0749%	0.0734%	0.0580%	0.0525%	-
	0%	0.2214%	0.2214%	0.1761%	0.1761%	0.2473%
	25%	0.2055%	0.1962%	0.1550%	0.1500%	0.2257%
40%	50%	0.1921%	0.1732%	0.1344%	0.1249%	0.2057%
	75%	0.1803%	0.1526%	0.1174%	0.1021%	0.1872%
	100%	0.1694%	0.1339%	0.1039%	0.0834%	0.1702%
	0%	0.2282%	0.2282%	0.1852%	0.1852%	0.2472%
	25%	0.2278%	0.2121%	0.1709%	0.1634%	0.2364%
50%	50%	0.2326%	0.1994%	0.1632%	0.1453%	0.2285%
	75%	0.2408%	0.1889%	0.1615%	0.1321%	0.2228%
	100%	0.2509%	0.1805%	0.1637%	0.1225%	0.2185%
	0%	0.2339%	0.2339%	0.1955%	0.1955%	0.2471%
	25%	0.2410%	0.2240%	0.1906%	0.1803%	0.2419%
60%	50%	0.2532%	0.2169%	0.1937%	0.1702%	0.2392%
	75%	0.2682%	0.2113%	0.2012%	0.1633%	0.2378%
	100%	0.2854%	0.2070%	0.2109%	0.1585%	0.2378%
	0%	0.2384%	0.2384%	0.2077%	0.2077%	0.2471%
	25%	0.2476%	0.2326%	0.2102%	0.1992%	0.2448%
70%	50%	0.2605%	0.2285%	0.2180%	0.1940%	0.2442%
	75%	0.2757%	0.2255%	0.2283%	0.1905%	0.2447%
	100%	0.2926%	0.2231%	0.2404%	0.1879%	0.2458%
	0%	0.2419%	0.2419%	0.2213%	0.2213%	0.2471%
	25%	0.2499%	0.2388%	0.2265%	0.2172%	0.2463%
80%	50%	0.2600%	0.2367%	0.2343%	0.2148%	0.2465%
	75%	0.2714%	0.2350%	0.2435%	0.2130%	0.2474%
	100%	0.2841%	0.2338%	0.2538%	0.2117%	0.2486%

Table 6.

Comparison of composite thicknesses between ASME standard, ISO standard, optimized results, and the parametric model

Corrosion Percentage (%)	Live Pressure Percentage (%)	ASME PCC-2, Live Pressure strain (mm/mm)	ASME PCC-2, No Live Pressure strain (mm/mm)	ISO 24817, Live Pressure strain (mm/mm)	ISO 24817, No Live Pressure strain (mm/mm)	ABAQUS Optimization Results	Empirical PCC-2, Live Pressure (A=1, B=1.06, C=.20) strain (mm/mm)
	0%	1.0654	1.0654	11.5581	11.5581	-	-
	25%	0.9202	1.0654	10.2324	11.5581	-	-
30%	50%	0.8093	1.0654	9.1391	11.5581	-	-
	75%	0.7220	1.0654	8.2310	11.5581	-	-
	100%	0.6516	1.0654	7.4699	11.5581	-	-
	0%	6.4188	6.4188	15.4125	15.4125	4.3188	4.4916
	25%	5.6337	6.4188	13.8053	15.4125	3.3337	4.3661
40%	50%	5.0029	6.4188	12.4349	15.4125	2.6519	4.2467
	75%	4.4889	6.4188	11.2657	15.4125	2.1589	4.1330
	100%	4.0642	6.4188	10.2651	15.4125	1.7642	4.0248
	0%	11.7722	11.7722	19.2670	19.2670	9.9200	10.1662
	25%	10.5178	11.7722	17.4882	19.2670	9.1000	9.9259
50%	50%	9.4558	11.7722	15.9178	19.2670	8.5158	9.6941
	75%	8.5549	11.7722	14.5365	19.2670	8.1400	9.4705
	100%	7.7877	11.7722	13.3236	19.2670	7.8200	9.2547
	0%	17.1256	17.1256	23.1215	23.1215	15.5900	15.8408
	25%	15.6034	17.1256	21.3051	23.1215	14.9034	15.5371
60%	50%	14.2469	17.1256	19.6441	23.1215	14.4469	15.2405
	75%	13.0435	17.1256	18.1335	23.1215	14.1000	14.9509
	100%	11.9787	17.1256	16.7657	23.1215	13.9000	14.6682
	0%	22.4791	22.4791	26.9759	26.9759	21.2000	21.5154
	25%	20.9223	22.4791	25.2845	26.9759	20.6500	21.2022
70%	50%	19.4692	22.4791	23.6853	26.9759	20.2692	20.8931
	75%	18.1199	22.4791	22.1802	26.9759	20.1199	20.5883
	100%	16.8728	22.4791	20.7699	26.9759	19.8728	20.2877
	0%	27.8325	27.8325	30.8304	30.8304	26.8325	27.1901
	25%	26.5043	27.8325	29.4595	30.8304	26.4900	26.9231
80%	50%	25.2176	27.8325	28.1256	30.8304	26.2176	26.6578
	75%	23.9745	27.8325	26.8308	30.8304	26.1000	26.3942
	100%	22.7769	27.8325	25.5769	30.8304	25.9000	26.1322

5.4 Summary

The four equations representing the ASME and ISO standards all resulted in differing composite thickness recommendations for the same testing scenarios. Equation 1, ASME PCC-2 Live Pressure Considered, violated the 0.25% max hoop strain requirement on pipes with for corrosion percentages greater than 60%. Furthermore, as the live wrapping pressure increased the strain rates increased as well. This means that the live pressure considerations discounted the composite thickness too much. A perfect live pressure thickness discount would result in equal hoop strains across all live wrapping pressures.

Equation 2, ASME PCC-2 No Live Pressure Considered, did not violate the 0.25% max hoop strain requirement for any corrosion level or live wrapping pressure. While it satisfied the main constraint of the code, it still does not fully serve its purpose of matching 0.25% hoop strain across all corrosion rates and wrapping pressures. The majority of testing scenarios produce overly conservative thickness recommendations, especially at lower corrosion percentages. Furthermore, the results from Equation 2 all experienced reduction in strain rates as the live wrapping pressure increased. Since Equation 2 recommended the same composite thickness for each set of live pressures, and that the hoop strain decreased as the live pressure increased, it can be deduced that live pressure does play a role in determining the necessary composite thickness. ASME and PCC-2 need a live pressure component of their analytical solutions.

Equation 3, ISO 24817 Live Pressure Considered, violated the max hoop strain requirement for the most aggressive testing scenario (80% corroded wall, 100% live wrapping pressure). For all other testing scenarios the ISO standard produced overly conservative pipe wraps, considerably more so than Equation 2. Equation 3 suffers from issues on both ends of the spectrum, as it is not conservative enough for edge cases like 80% wall thinning and 100% live wrapping pressure as well as being too conservative for every other testing scenario.

Equation 33, ISO 24817 No Live Pressure Considered does not violate any max hoop strain conditions on any wrapping pressure or corrosion percentage, but Equation 33 is the most overly conservative composite thickness equation of the standards. Between 40% and 70% wall thinning, the resulting hoop strain percentages hover around 0.16-0.17% hoop strain. This is a considerable difference than the inputted 0.25% hoop strain these equations are trying to match. The reason ISO values are so much more conservative than the ASME standard is directly attributed to only allowing 72% yield stress to be modelled in the steel yielding equation. This artificially forces the composites to be conservative for the sake of being conservative. The ISO equations should not be viewed as attempts to directly discern the resulting hoop strain.

Equation 32, the best fit parametric version of Equation 1, resulted in all of the testing scenarios safely conforming to the max hoop strain limit. Furthermore the parametric model produced the closest values to 0.25% hoop strain across all corrosion percentages and live pressures. There is still not perfect agreeance between the parametric model a consistent 0.25% hoop strains across every corrosion level and wrapping pressure. The best fit parametric model had B values of 1.06 and C values of 0.20. These rates tend to suggest that the ASME codes need to reexamine the standards to account for these inconsistencies. It is theorized that the steel yielding term, Y_s, should be increased to a steel stress equivalent when strain hardened to no greater than 0.25% strain. This would better reflect the 6% increase in the steel stress term the parametric model recommends. As to why the C values need to be reduced to 20% of the current term, there is not as obvious of an answer at this time. Given no straightforward answer, the author chooses not to speculate but consider it a starting point for further research.

The overall best performing model was the optimized finite element model. Using python scripting, it iteratively solved for the perfect composite thickness that created a perfect 0.25% hoop strain. While this model performed in an outstanding fashion, it does not serve the professional standards well. The standards need an analytical solution that technicians and operators can easily use in the field that is clearly defined. Relying on finite element model does not offer as concrete of a final answer as an analytical solution.

CHAPTER VI

CONCLUSIONS AND FUTURE WORK

6.1 Conclusions

Based off the results and discussion the following conclusions are revealed. ASME PCC-2 Substrate Yields Equation with Live Pressure is not conservative enough for safe use, especially for significant wall thinning. It is not recommend this equation be used to develop composite pipe wrap. ASME PCC-2 Substrate Yields Equation with No Live Pressure Considerations allows for the creation of safe pipe wraps but is overly conservative. The data also supports a live pressure component be included. It is not recommend this equation be used to develop composite pipe wraps.

ISO 24817 Substrate Yields Equation with Live Pressure was also not conservative enough for safe use. It is not recommend this equation be used to develop composite pipe wraps. ISO 24817 Substrate Yields Equation with No Live Pressure Considerations created safe pipe wraps but were significantly over conservative to the point of majorly effecting the financial viability of a composite wrap. It is not recommend this equation be used to develop composite pipe wraps.

The best performing model was the optimized finite element model which used python scripts to solve for the perfect composite thickness. While this ensured no waste on code compliant wraps, it is not feasible for professional standards to refer to a finite element model over an analytical solution. It is not recommended that professional standards use the finite element model to recommend and enforce composite wraps with live pressure considerations.

The best fit parametric model performed better than all the standards solutions as it created consistently safe wraps while recommending the least amount of excessive wrapping. Furthermore, the best fit parametric model revealed that the steel yield strength term within all the analytical solutions would be better served as a strain hardening stress value greater than the yield stress. An in depth analysis of the theory governing load transfer in composite wraps validated that parametric modelling is a necessity as the exact composite thickness cannot be analytically solved for. Given these conclusions, it is the final recommendation for the ASME PCC-2 and ISO 24817 to adopt the best fit parametric model to create safe, resource efficient composite wraps.

6.2 Future Work

While this research has established new insights into how live pressure and wall thinning effect composite repair hoop strains, it unveiled more questions along with its answers. There is plenty of opportunity for follow up on this topic. Currently, it is not fully understood why the parametric values selected produce the best fit. The parametric formula as a whole should be vetted with real world pressure testing. Furthermore, these models represent a level of abstraction from real world corrosion scenarios as the corrosion is modelled as perfect thinning around the whole pipe. It is recommended to continue testing the ISO, ASME and the best fit parametric model with localized machined defects.

REFERENCES

[1] *Repair of Pressure Equipment and Piping ASME PCC-2-2015*. New York, NY: The American Society of Mechanical Engineers, 2015.

[2] “The role of pipeline age in pipeline safety - File.aspx.” [Online]. Available: http://www.ingaa.org/File.aspx?id=19307. [Accessed: 13-Jun-2016].

[3] M. D. Chapetti, J. L. Otegui, C. Manfredi, and C. F. Martins, “Full scale experimental analysis of stress states in sleeve repairs of gas pipelines,” *International Journal of Pressure Vessels and Piping*, vol. 78, no. 5, pp. 379–387, May 2001.

[4] J. M. Duell, J. M. Wilson, and M. R. Kessler, “Analysis of a carbon composite overwrap pipeline repair system,” *International Journal of Pressure Vessels and Piping*, vol. 85, no. 11, pp. 782–788, Nov. 2008.

[5] J. l. f. Freire, R. d. Vieira, J. l. c. Diniz, and L. c. Meniconi, “Part 7: Effectiveness of composite repairs applied to damaged pipeline,” *Experimental Techniques*, vol. 31, no. 5, pp. 59–66, Sep. 2007.

[6] M. L. da Silva and H. da Costa Mattos, “Failure pressure estimations for corroded pipelines,” *Materials Science Forum*, vol. 758, pp. 65–76, Jun. 2013.

[7] N. Saeed, H. Ronagh, and A. Virk, “Composite repair of pipelines, considering the effect of live pressure-analytical and numerical models with respect to ISO/TS 24817 and ASME PCC-2,” *Composites Part B*, vol. 58, pp. 605–610, Mar. 2014.

[8] R. A. Esmaeel, M. A. Khan, and F. Taheri, “Assessment of the environmental effects on the performance of FRP repaired steel pipes subjected to internal pressure,” *J. Pressure Vessel Technol*, vol. 134, no. 4, pp. 041702–041702, Jul. 2012.

[9] W. K. Goertzen and M. R. Kessler, “Dynamic mechanical analysis of carbon/epoxy composites for structural pipeline repair,” *Composites Part B: Engineering*, vol. 38, no. 1, pp. 1–9, Jan. 2007.

[10] H. S. da Costa Mattos, J. M. L. Reis, L. M. Paim, M. L. da Silva, F. C. Amorim, and V. A. Perrut, “Analysis of a glass fibre reinforced polyurethane composite repair system for corroded pipelines at elevated temperatures,” *Composite Structures*, vol. 114, pp. 117–123, Aug. 2014.

[11] W. K. Goertzen and M. R. Kessler, “Creep behavior of carbon fiber/epoxy matrix composites,” *Materials Science and Engineering: A*, vol. 421, no. 1–2, pp. 217–225, Apr. 2006.

[12] M. W. Keller, B. D. Jellison, and T. Ellison, “Moisture effects on the thermal and creep performance of carbon fiber/epoxy composites for structural pipeline repair,” *Composites Part B: Engineering*, vol. 45, no. 1, pp. 1173–1180, Feb. 2013.

[13] M. Shamsuddoha, M. M. Islam, T. Aravinthan, A. Manalo, and K. Lau, “Effectiveness of using fibre-reinforced polymer composites for underwater steel pipeline repairs,” *Composite Structures*, vol. 100, pp. 40–54, Jun. 2013.

[14] C. Alexander and O. O. Ochoa, “Extending onshore pipeline repair to offshore steel risers with carbon–fiber reinforced composites,” *Composite Structures*, vol. 92, no. 2, pp. 499–507, Jan. 2010.

[15] J. N. Reddy, *Introduction to the finite element method*, Third Edition. McGraw Hill, pp. 100-149.

APPENDIX I

EXPLANATION OF THIN WALL PRESSURE VESSELS

Given that a cylindrical pressure vessel has a radius to thickness ratio greater than 10, the given assumptions of a thin walled pressure vessel apply. The thin wall assumption states there is negligible transverse stress within the wall, thus the outside radius is equal to the interior radius. Only hoop and axial stresses are considered for internally pressurized vessels.

The axial stress is solved by analyzing the pressures exerted on the end cap of a pressure vessel. Internal pressure P results in a longitudinal stress in the cylinder. The force exerted on the pressure vessel endcap is simply the pressure multiplied by the area.

$$F = P(\frac{\pi * D^2}{4}) \tag{44}$$

The reactionary force experienced by the steel wall is equal to the average longitudinal stress inside the wall multiplied by the cross sectional area of the pipe wall.

$$F = \sigma_2 (2\pi r t) \tag{45}$$

Setting these forces equal to one another establishes an equation for the longitudinal stress.

$$\sigma_2 = \frac{PD}{4t} \tag{46}$$

Cutting the cylinder in the parallel to the longitudinal axis reveals the hoop stress interactions. The internal pressure P results in a hoop stress. The force exerted upon the half shell by the internal pressure is simply the pressure multiplied by the surface area.

$$F = P(\pi D * \Delta L) \qquad (47)$$

The reactionary force experienced by the steel is equal to the average hoop stress multiplied by the area.

$$F = \sigma_1 (2t * \Delta L) \qquad (48)$$

Setting these forces equal to one another establishes an equation for hoop stress.

$$\sigma_1 = \frac{PD}{2t} \qquad (49)$$

These derivations form the basis for Barlow's formula which relates the internal pressure of a pipe to the strength of the material. Barlow's formula is a reduced version of Lame's equations for pressure vessels.

APPENDIX II

FINITE ELEMENT INPUTS FOR EACH TESTED STANDARDS EQUATION

Table 7.

ABAQUS inputs for ASME PCC-2 live pressure considered (0.25% hoop strain)

Corrosion Percentage (%)	Live Pressure Percentage (%)	Steel Thickness (mm)	Composite Thickness (mm)	Live Pressure (MPa)	Design Pressure (MPa)
30%	0%	4.977	1.0654	0	27.25
	25%	4.977	0.9202	4.7688	27.25
	50%	4.977	0.8093	9.5375	27.25
	75%	4.977	0.722	14.3063	27.25
	100%	4.977	0.6516	19.075	27.25
40%	0%	4.266	6.4188	0	27.25
	25%	4.266	5.6337	4.0875	27.25
	50%	4.266	5.0029	8.175	27.25
	75%	4.266	4.4889	12.2625	27.25
	100%	4.266	4.0642	16.35	27.25
50%	0%	3.555	11.7722	0	27.25
	25%	3.555	10.5178	3.4067	27.25
	50%	3.555	9.4558	6.8134	27.25
	75%	3.555	8.5549	10.2201	27.25
	100%	3.555	7.7877	13.6269	27.25
60%	0%	2.844	17.1256	0	27.25
	25%	2.844	15.6034	2.75	27.25
	50%	2.844	14.2469	5.45	27.25
	75%	2.844	13.0435	8.175	27.25
	100%	2.844	11.9787	10.9	27.25
70%	0%	2.133	22.4791	0	27.25
	25%	2.133	20.9223	2.0440	27.25
	50%	2.133	19.4692	4.0881	27.25
	75%	2.133	18.1199	6.1321	27.25
	100%	2.133	16.8728	8.1761	27.25
80%	0%	1.422	27.8325	0	27.25
	25%	1.422	26.5043	1.3627	27.25
	50%	1.422	25.2176	2.7254	27.25
	75%	1.422	23.9745	4.0881	27.25
	100%	1.422	22.7769	5.4507	27.25

Table 8.

ABAQUS inputs for ASME PCC-2 no live pressure considered (0.25% hoop strain)

Corrosion Percentage (%)	Live Pressure Percentage (%)	Steel Thickness (mm)	Composite Thickness (mm)	Live Pressure (MPa)	Design Pressure (MPa)
	0%	4.977	1.0654	0	27.25
	25%	4.977	1.0654	4.7688	27.25
30%	50%	4.977	1.0654	9.5375	27.25
	75%	4.977	1.0654	14.3063	27.25
	100%	4.977	1.0654	19.0750	27.25
	0%	4.266	6.4188	0	27.25
	25%	4.266	6.4188	4.0875	27.25
40%	50%	4.266	6.4188	8.1750	27.25
	75%	4.266	6.4188	12.2625	27.25
	100%	4.266	6.4188	16.3500	27.25
	0%	3.555	11.7722	0	27.25
	25%	3.555	11.7722	3.4067	27.25
50%	50%	3.555	11.7722	6.8134	27.25
	75%	3.555	11.7722	10.2201	27.25
	100%	3.555	11.7722	13.6269	27.25
	0%	2.844	17.1256	0	27.25
	25%	2.844	17.1256	2.7500	27.25
60%	50%	2.844	17.1256	5.4500	27.25
	75%	2.844	17.1256	8.1750	27.25
	100%	2.844	17.1256	10.9000	27.25
	0%	2.133	22.4791	0	27.25
	25%	2.133	22.4791	2.0440	27.25
70%	50%	2.133	22.4791	4.0881	27.25
	75%	2.133	22.4791	6.1321	27.25
	100%	2.133	22.4791	8.1761	27.25
	0%	1.422	27.8325	0	27.25
	25%	1.422	27.8325	1.3627	27.25
80%	50%	1.422	27.8325	2.7254	27.25
	75%	1.422	27.8325	4.0881	27.25
	100%	1.422	27.8325	5.4507	27.25

Table 9.

ABAQUS inputs for ISO 24817 live pressure considered (0.25% hoop strain)

Corrosion Percentage (%)	Live Pressure Percentage (%)	Steel Thickness (mm)	Composite Thickness (mm)	Live Pressure (MPa)	Design Pressure (MPa)
	0%	4.977	11.5581	0.00001	27.25
	25%	4.977	10.2324	4.7688	27.25
30%	50%	4.977	9.1391	9.5375	27.25
	75%	4.977	8.231	14.3063	27.25
	100%	4.977	7.4699	19.075	27.25
	0%	4.266	15.4125	0.00001	27.25
	25%	4.266	13.8053	4.0875	27.25
40%	50%	4.266	12.4349	8.175	27.25
	75%	4.266	11.2657	12.2625	27.25
	100%	4.266	10.2651	16.35	27.25
	0%	3.555	19.267	0.00001	27.25
	25%	3.555	17.4882	3.4063	27.25
50%	50%	3.555	15.9178	6.8125	27.25
	75%	3.555	14.5365	10.2188	27.25
	100%	3.555	13.3236	13.625	27.25
	0%	2.844	23.1215	0.00001	27.25
	25%	2.844	21.3051	2.725	27.25
60%	50%	2.844	19.6441	5.45	27.25
	75%	2.844	18.1335	8.175	27.25
	100%	2.844	16.7657	10.9	27.25
	0%	2.133	26.9759	0.00001	27.25
	25%	2.133	25.2845	2.0437	27.25
70%	50%	2.133	23.6853	4.0875	27.25
	75%	2.133	22.1802	6.1312	27.25
	100%	2.133	20.7699	8.175	27.25
	0%	1.422	30.8304	0.00001	27.25
	25%	1.422	29.4595	1.3625	27.25
80%	50%	1.422	28.1256	2.725	27.25
	75%	1.422	26.8308	4.0875	27.25
	100%	1.422	25.5769	5.45	27.25

Table 10.

ABAQUS inputs for ISO 24817 no live pressure considered (0.25% hoop strain)

Corrosion Percentage (%)	Live Pressure Percentage (%)	Steel Thickness (mm)	Composite Thickness (mm)	Live Pressure (MPa)	Design Pressure (MPa)
	0%	4.977	1.0654	0	27.25
	25%	4.977	1.0654	4.7688	27.25
30%	50%	4.977	1.0654	9.5375	27.25
	75%	4.977	1.0654	14.3063	27.25
	100%	4.977	1.0654	19.0750	27.25
	0%	4.266	6.4188	0	27.25
	25%	4.266	6.4188	4.0875	27.25
40%	50%	4.266	6.4188	8.1750	27.25
	75%	4.266	6.4188	12.2625	27.25
	100%	4.266	6.4188	16.3500	27.25
	0%	3.555	11.7722	0	27.25
	25%	3.555	11.7722	3.4067	27.25
50%	50%	3.555	11.7722	6.8134	27.25
	75%	3.555	11.7722	10.2201	27.25
	100%	3.555	11.7722	13.6269	27.25
	0%	2.844	17.1256	0	27.25
	25%	2.844	17.1256	2.7500	27.25
60%	50%	2.844	17.1256	5.4500	27.25
	75%	2.844	17.1256	8.1750	27.25
	100%	2.844	17.1256	10.9000	27.25
	0%	2.133	22.4791	0	27.25
	25%	2.133	22.4791	2.0440	27.25
70%	50%	2.133	22.4791	4.0881	27.25
	75%	2.133	22.4791	6.1321	27.25
	100%	2.133	22.4791	8.1761	27.25
	0%	1.422	27.8325	0	27.25
	25%	1.422	27.8325	1.3627	27.25
80%	50%	1.422	27.8325	2.7254	27.25
	75%	1.422	27.8325	4.0881	27.25
	100%	1.422	27.8325	5.4507	27.25