Towards the reliability prediction of mechanical components for nuclear safety cases

A THESIS SUBMITTED TO THE

UNIVERSITY OF CENTRAL LANCASHIRE

FOR THE DEGREE OF MASTER OF SCIENCE

IN THE SCHOOL OF ENGINEERING

2017

By

Stephen D Moore, BSc (Hons)

Abstract

The thesis presents a modelling method to increase accuracy in reliability predictions of mechanical components. Such predictions are used innuclear safety case documentation that is required for a nuclear site license to be granted. The methodology proposed is the use of front-end Finite Element fatigue analysis using the ANSYS software to effectively evaluate the mechanical reliability of nuclear safety systems/mechanisms, to evaluate the Mean Cycles To Failure of any steel three-dimensional component. No inference is attempted with reference to the validity of the reliability values calculated, though they are shown to be reasonable when a simplistic constant hazard rate reliability model is employed. Some rudimentary evidence is provided showing that reductions of the fatigue safety factor by just less than a quarter may increase the design life of the component four-fold, ergo significantly reducing the on-site expected failure rates. The methodologies explored herein therefore effectively show that the ANSYS-Workbench software can be used to predict the life, ergo the reliability of mechanical components *in-situ*.

Declaration

No portion of the work referred to in this thesis has been submitted in support of an application for another degree or qualification of this or any other university or other institute of learning.

COPYRIGHT

Acknowledgements

The author would like to thank the supervisory team Dr J. Whitty and Dr J. Francis for their guidance and suggestions throughout the composition of this Thesis. Especially Dr J. Whitty for his patience, direction and wisdom during the more challenging aspects of this work.

A special thank you goes out to the author's wife Kanya for all her support, encouragement, understanding and for putting up with all the late nights and absences during the past years.

Contents

List of Figures

List of Tables

Chapter 1

Introduction

The safety of nuclear installations in the UK are managed and assured by a system of regulatory control that is based on a licensing process. The license is granted to a corporate body to use a site for specified activities for a specified time frame under specific conditions. The nuclear safety case forms an integral part of the process for gaining this license. The Office for Nuclear Regulation (ONR) guidance, notes that a licensee is expected to: *make and implement adequate arrangements for the production and assessment of safety cases consisting of documentation to justify safety during the design, construction, manufacture, commissioning, operation and decommissioning phases of the lifecycle* [1]. The research described in this thesis is a continuation of work conducted at undergraduate level achieving (1st class Honours) [2]. It is envisaged that the living safety case can be improved by use of state-of-the-art computational modelling methods, in order to predict the lifespan of bespoke mechanical components at the design stage as well as during service. What follows is a new modelling application which can be employed to generate predictions which will provide data for use in the aforementioned safety case and therefore, help to optimize maintenance schedules generated; thus reducing overall lifetime operational costs.

1.1 Safety case

Safety cases have been used in the United Kingdom (UK) for over 40 years [3], as a technique to help manage the major risks in high hazard industries, such as nuclear, chemical and oil and gas. A safety case is described as: a structured argument, supported by a body of evidence that provides a compelling, comprehensive and valid case that a system is safe for a given application in a given operating environment [1][3]. The development of safety cases in essence was to provide a system for managing risk to a tolerable standard. The aim being to reduce the probability of an accident occurring.

1.1.1 Nuclear safety case

This may refer to a nuclear site, a nuclear power plant, part of a plant or a plant modification. With the purpose of establishing and presenting a case demonstrating that all applicable legislation requirements are met and that the system(s) will continue to be acceptably safe to use within its environment throughout its lifetime. The purpose of the safety case is to demonstrate and establish in written form that the plant process, activities and modifications being proposed are:

- Soundly assessed and meet the required safety principles.
- Safe during both normal and in fault condition and will remain fit for purpose.
- Conform to the appropriate standards, codes of practice and good nuclear engineering.
- As Low As Reasonably Practicable (ALARP) principles are applied to protect both the public and worker(s).

- Have a defined and acceptable operating envelope, with defined conditions and limits; with a means to keep within these limits.

defined by the HSE as: *the totality of documented information and arguments which substantiates the safety of the plant, activity, operation or modification in question. It provides a written demonstration that relevant standards have been met and that risks have been reduced ALARP* [4]. Safety cases are important not only to minimize safety risks but also to reduce commercial and project risks, not their intended purpose but a consequential benefit. High risk industries such as nuclear are heavily influenced by public opinion and confidence in how safe they are to operate. This became evident in the decline of the nuclear sector internationally after the Three Mile Island accident 1959, the Chernobyl disaster 1986 and more recently, the Fukushima Daiichi incident 2011, (which resulted in a slow down of nuclear programmes around the world).

1.1.2 Nuclear safety cases in the UK

After the Windscale reactor fire in Cumbria in 1957 the UK nuclear industry realized the requirement for a safety case. The introduction of the Nuclear Installations Act 1965 (NIA 65) introduced a licensing regime [5], with the production and maintenance of a safety case justifying safety during all phases of operation being one of the site license conditions to be met.

However, as plants and system become more complex so do the safety cases supporting these facilities. Safety cases have developed over the years, varying from sector to sector. The nuclear safety case is (i) a deterministic or stochastic analysis of the hazards and faults which could arise and cause injury, disability or loss-of-life from the plant either on or off site, (ii) a demonstration of the sufficiency and adequacy of the arrangements, (engineering and procedures) ensuring that the combined frequencies of such events will be acceptably low (i.e. ALARP). In order to demonstrate safety, a safety case will conform to Safety Assessment Principles (SAPs) detailing the information that the safety case should contain [1, 3]. The ONR document, ONR Nuclear Safety Technical Assessment Guide states among other things that: *A safety case may comprise a hierarchy of documents. The top tier will contain the core of the safety arguments and increasingly detailed technical documents and supporting analysis will be presented in lower tiers. At the lowest level there are likely to be the engineering calculations, experimental results and data on reliability and operational experience.* The research is aimed at studying these so called lower tiers in more depth, particularly the engineering principles and design for reliability and reliability claims.

1.2 Ageing/Degradation

With engineering Structures System and Components (SSCs) steadily growing in complexity an increasing awareness of the risks, hazards, and liabilities related to the operation of these engineering systems is required [6, 7]. However, the costs associated with renewing and/or replacing these SSC's is increasing and it is often the least complex SSC's which are life limiting (e.g. civil structures that are driving organizations to extend the life of their SSC's); typically nuclear reactors are designed to operate over a 40 year period which may be extended for several decades [7].

Nuclear facilities are complex, being composed of many interdependent systems which must operate over long periods of time. These long periods of operation mean that the plant will undergo changes throughout this period; these changes can be attributed to a number of things (e.g. ageing and/or obsolescence of the facilities SSC's or a change in regulations [7]).

The prediction of damage caused by environmental and or mechanical forces remains a serious challenge during the lifetime of a nuclear facility; and so, understanding these mechanisms, how SSC's may age or degrade over time and the stressors that may accelerate the ageing process is increasingly important [1]. Roberge discusses in his handbook of corrosion engineering that modelling is an essential benchmarking process for engineers in the selection and life-prediction of new materials and process [6].

1.3 Reliability & Risk

Estimating the design life of mechanical equipment is a difficult task, as many life limiting failure modes such as corrosion, erosion, creep and fatigue operate on the component at the same time and have a synergistic effect on reliability. Additionally the loading on the component may be static, cyclic or dynamic at different points during the component's life-cycle. Other variables affecting prediction and reliability include the severity of the loading; and material variability. In the work described in this thesis, Failure Mode Effect Analysis (FMEA) was initially carried out in order to generate data.

The diagram shown in Figure 1.1 is a pictorial representation of the application described in this thesis; and intends to progress to the next stage. It should be noted that this project mainly concentrates on determination of life of safety critical components using the iterative *Design-Model-Analysis-Results* loop indicated in Figure 1.1. By combining knowledge and skills of multiple engineering disciplines it is intended to derive more realistic reliability figure(s) for be-spoke items. The idea is to be able to determine on the bathtub curve when an item is about to fail from wear, fatigue or stress or a combination of factors. Ergo, enabling nuclear plants to reduce maintenance costs, increase operating time and reduce catastrophic failures of components in service. Additionally, other powerful analysis and statistical tools (like Design For Six Sigma: DFSS) would then be utilized to reduce the failure region of the SSC.

The nuclear industry can be thought of as space exploration on earth as they both suffer from similar difficulties i.e. both require SSC's to operate in harsh environments, both sets of SSC's are subjected to high levels of radiation and they both have to be fixed remotely if they break down, providing they can be repaired. Safety cases are important not only to minimize safety risks but also to reduce commercial and project risks. High risk industries such as nuclear are heavily influenced by public opinion, increasing confidence in how safe they are to operate through meeting the objectives of this study is the intention of the author (§§1.5.2). Recent dialogue with Sellafield Ltd [8] identified a possible area of investigation, which is to look at the public domain probabilistic design data provided by NASA in the design and construction of the Space Shuttle[1]. It has become apparent that this early example of so called probabilistic design [9] was modestly successful in the prediction of the reliability values of a number of mechanical components, hence would be an important starting point (salient lemma) for the work presented here.

The safety of a nuclear facility depends on controlling the risk of exposure to radiation from both routine operational activities and from potential accidents. The legal framework for the nuclear industry is based around the Health and Safety at Work Act 1974 (HASWA) the Energy Act 2013 and NIA 65, regulated by the Office for Nuclear Regulation (ONR) in the UK. One of a number of factors to ensure safety is the assessment of a robust design with limits and conditions for operation. The ONR uses Safety Assessment Principles (SAPs) to help inspectors judge the adequacy of a licensees' safety case [10, 11].

There are 36 conditions to each nuclear site licence [10]. The Nuclear Site Licence Conditions (NSLC) form part of a legal framework which must be drawn on in an assessment, and require the licensee to make adequate arrangements, in the interests

[1]e.g. http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/20100023397.pdf (2010)

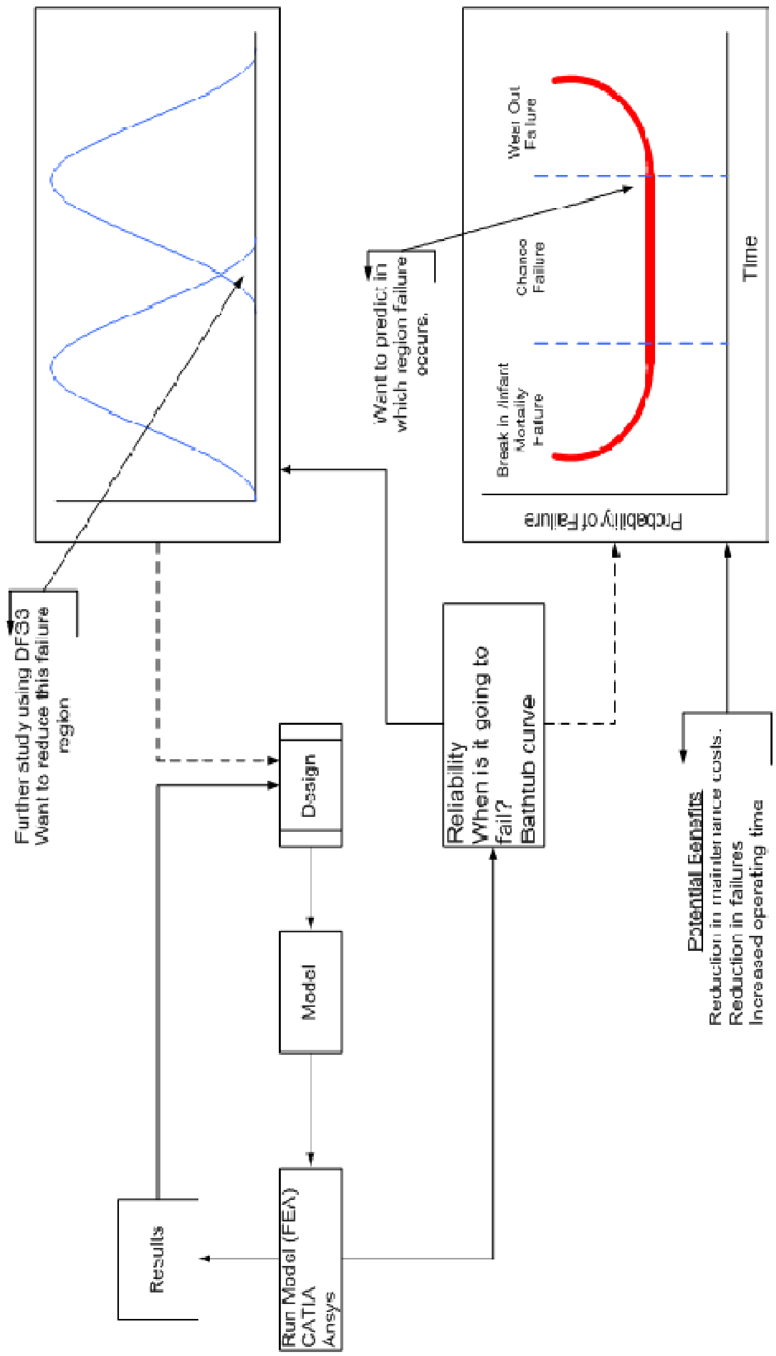


Figure 1.1: Diagram showing interaction between the processes.

of safety, to secure certain objectives in general [12]. The principal NSLCs relevant to the work described throughout this thesis are NSLC 14, NSLC 23, NSLC 27 and NSLC 28.

NSLC 14 requires the licensee to make and implement adequate arrangements for the production and assessment of safety cases. The safety case will normally contain Probabilistic Safety Analysis (PSA) as well as deterministic analysis. While NSLC 23 stipulates that the safety case identifies the conditions and limits necessary in the interests of safety i.e. keep within the safety envelope, it is the expectation of the ONR that both PSA and deterministic aspects contribute to this process. Additionally, the ONR expects that the PSA will furnish suitable and sufficient safety mechanisms, devices and circuits, as required by NSLC 27. Moreover engineering substantiation will enable identification of safety functions and safety measures (SSC/plant) that may affect safety for which regular Examination, Inspection, Maintenance and Testing (EIM&T) will be needed, as per NSLC 28.

The depth of the PSA for a given facility will vary depending on the magnitude of the radiological hazard and the risks posed by that hazard. Using techniques such as the FMEA to categorise the SSC, the correct level of rigour can be applied to the PSA. The PSA should account for all contributors to risk (i.e. during the Reliability Centered Maintenance (RCM) activity, specifying the correct maintenance periodicity's, as human errors can lead to maintenance induced faults, e.g. shaft misalignment's) [12].

The design of engineered SSC's need to meet the required safety function with the appropriate reliability, according to the scale and frequency of the radiological hazard, so that confidence in the robustness of the design can be gained. The functional requirements and classification are defined outputs of the deterministic PSA. Each project life-cycle is influenced by the classification of the SSC (i.e. the correct level of rigour is applied), some examples are listed:

- Design approach
- Level of design substantiation
- Applied codes and standards
- Material selection
- Operational phase, what level of EIM&T required.

However, the varied range of mechanical engineering SSC's makes it complex to specify generic codes, standards and procedures to an assigned nuclear safety classification. Currently there are no generic UK nuclear specific codes and standards that define the requirements for the categorisation and classification of mechanical engineering aspects [13]. To satisfy that the reliability of the SSC is robust and that the risk is As Low As Reasonably Practical (ALARP) a robust design process that is integrated with the safety case process is of utmost importance [10, 13].

1.3.1 Safety function classification

All SSC's are specified and designed to provide a required engineering functionality. This functionality will have an influence on safety and requires an appropriate safety classification to be assigned [14]. This Safety Function Classification (SFC) is used to identify the level of confidence that substantiation should provide; and thus, will affect the design methods and standards, material selection, fabrication and inspections as well as the maintenance requirements for example. SFC's range from 0 - 3 with SFC1 requiring the highest level of confidence, Table 1.1 provides the SFC description.

What is presented throughout this thesis is a first undertaking in marrying the deterministic analysis of Design Based Analysis (DBA) with the stochastic approaches of PSA, via the assessment of the fatigue life of the SSC; and via the evaluation

Safety Function Classification number	Description
SFC 1	Safety Function Class 1: Failure of which alone would lead directly to an off site release which would result in a dose >10mSv to the public or >250mSv to a worker.
SFC 2	Safety Function Class 2: Failure of which, in the absence of correct functioning safety measures, could lead directly to an off site release which resulted in a dose >0.1mSv to the public or >20mSv to a worker.
SFC 3	Safety Function Class 3: Failure of which, in the absence of correct functioning safety measures, could lead directly to an off site release which resulted in a dose of between 0.001 to 0.1mSv to the public or a dose between 2 to 20mSv to a worker.
SFC 0	Safety Function Class 0: Failure of which, in the absence of correct functioning safety measures, could lead directly to an off site release which resulted in a dose of between <0.001mSv to the public or a dose between <2mSv to a worker.

Table 1.1: Safety Function Classification description.

of reliability figures of mechanical components resident on a nuclear site, using the aforementioned design fatigue life estimates and computational mechanics software. In simple terms one needs/wants to understand how the SSC is going to perform/age throughout its life-cycle; to achieve this it is important that the design intent is understood and managed appropriately so that the correct level of asset investment can be applied to the right SSCs.

Probably the most used distribution employed in the prediction of hazard rates in the aforementioned NSC is that of the Weibull distribution, which has the reliability function [15]:

$$R(t) = \exp\left\{-\left(\frac{t}{\eta}\right)^{\beta}\right\} \tag{1.1}$$

where β is the shape factor which corresponds to premature failure $\beta < 1$, constant hazard rate β=1 and wear out β>1, interestingly $\beta \geq 2$ is usually attributed to fatigue failure. Additionally, it can be shown [15] that the Mean Time To Failure (MTTF) is given by $\mu = \eta\Gamma\left(1 + \frac{1}{\beta}\right)$. It is worth noting that when a constant hazard rate is assumed then equation (1.1) reduces to an exponential distribution with a MTTF of $\mu = \eta\Gamma\left(2\right) = \eta$.

1.4 Overview of this thesis

The remainder of this chapter with be dedicated to the presentation of the aims and objectives (the proceeding section) followed by a discussion of the fundamentals of material behavior e.g. basic stress-strain response of materials, yield criteria, and the fundamentals of metal fatigue and its interactions with other failure mechanisms; relevant to the FE analyses presented later in the thesis. The next chapter will attempt to describe some of the pertinent literature dedicated to the historical perspective of fatigue, including the key causes and mechanisms of the phenomenon and the modelling approaches and design rules of avoidance.

In addition to identifying possible maintenance strategies, derived from Failure Mode and Effect Analysis (FMEA), in order to focus on the onset of fatigue of components *in-situ* on a nuclear site. In the interest of completeness a short review of the fundamentals of pressure vessel design is presented using the American Society of Mechanical Engineers (ASME) codes, with particular emphasis given to Pressure Vessel (PV) design and avoidance of excessive plastic deformation. Thereby justifying wall thickness values. The valve models detailed in the case studies later in the thesis are examples of this also; here some attention is given to the ASME code approaches used in the design against fatigue.

The third chapter is dedicated to the methods employed, viz. code verification, relief valve geometrical construction, Finite Element (FE) meshing and boundary conditions in order to simulate high pressure saturated steam at 360 °C. In order to verify methods used, a text book (benchmark) case study was performed in which the FE modelling stress and strain predictions were compared with values obtained from analytic model analogues evident in the literature. It should be noted that the work presented throughout this thesis is not intended to be a clear or concise treatise of finite element modelling of any of the SSC's selected. More a vehicle in which to explore life predictions of the software ergo provide estimates of failure and reliability values of mechanical components in order to inform the DBA and PSA elements of a nuclear safety case.

Chapter 4 presents the specific ANSYS FE modelling fatigue life hence reliability figures for the relief valves being considered in this thesis. The penultimate chapter discusses some of the main observations of the results presented, developing the main findings whilst attempting to explore the wider context of the work herein. As well as presenting a method in order to provide first estimates of reliability values of mechanical components using the fatigue predictions from the nuclear industry validated ANSYS workbench code. The final chapter begins with a reflection of the aims and objectives of the work before drawing out the salient technical conclusions; the thesis concludes with some brief recommendations for further work.

1.5 Aims and objectives

The primary goal of this research is to determine a means of predicting the life-time of mechanical components which could be employed in a Nuclear safety case. Use has been made of computational simulation methods resident in the nuclear industry standard code ANSYS-Workbench, in order to provide reasonable approximations to *in-situ* reliability values; and thus inform the maintenance protocol and/or the PSA data resident in the site nuclear safety case documentation. These goals are summarized by virtue of the following research hypothesis.

> *Nuclear Industry standard design simulation software codes (e.g. ANSYS) can be used to predict the life, ergo the reliability of mechanical components.*

In order to rigorously evaluate this hypothesis the aims and objectives of the thesis of the proceeding sub-section are suitably identified. It is noted here that all models presented throughout this thesis are for representative and demonstrative purposes only. Thereby, hypothesizing further that the aforementioned simulation code can be used to provide reliability values to *any* mechanical component. It is not the aim of this Masters thesis to test this further hypothesis rather, this work is intended to demonstrate or prove a concept.

1.5.1 Aims

- Evaluate the mechanical reliability of mechanical safety mechanisms using the ANSYS-Workbench simulation suite (see objectives 1-4, §§1.5.2).
- Inform the living nuclear safety case by increasing the accuracy of mechanical life-time prediction of specific be-spoke SSC's (see objectives 4-6, §§1.5.2).

• Produce this thesis in partial fulfillment of the degree of MSc by research in the John Tyndall Institute for Nuclear Research at the University of Central Lancashire (UCLan); this thesis having industrial context objectives 3-8, of the proceeding subsection.

1.5.2 Objectives

1. Provide suitable selection metrics for the evaluation of lifetime predictions of SSC's for use in nuclear safety cases.

2. Critically evaluate possible failure mechanisms of a particular SSC commonly employed on a nuclear site.

3. Formulate lifetime and hence reliability requirements of a particular SSC evident in substantiating a Nuclear safety case.

4. Present lifetime safety bench-marking case-studies and/or predictions of a variety of SSCs.

5. Underpin said case-studies using numerical design assessment and/or analytic methods evident in the literature.

6. Develop methodologies to accurately predict the lifespan of mechanical SSCs during the front-end design stages and *in-situ*.

7. Use fatigue analysis data to approximate reliability values (§5.5.1) of a variety of SSC's which may be present on a nuclear decommissioning site.

8. Use generated lifetime predictions in an industrial setting (e.g. the T-RAM system[2] at Sellafield Ltd).

[2] The *in-situ* Throughput-Reliability Availability and Maintainability SL system

1.6 Fundamentals of material behaviour

In order to achieve the thesis third objective (§§1.5.2) it will be necessary to classify typical materials and their respective failure mode, used in the construction of the SSC's. There are four distinct categories of materials: Metals, Polymers, Ceramics and Composites [16]. The particular failure mode of any mechanical component is dependent on the classification of a particular material. Such classifications are achieved from the bonding type present within them (either on the molecular, micro or on a macro level). Such data and hence failure modes have been entered into the modelling protocols in *demonstratum est* objectives 3 & 4 (§§1.5.2).

1.6.1 Stress strain

Stress strain curves are graphical representations of a material's mechanical properties. Probably the most important test of a materials mechanical response is the tensile test, in which one end of a rod or wire is clamped in a loading frame and the other end is subjected to a controlled displacement. Connected instrumentation provides a reading of the load corresponding to the displacement [17, 18].

The engineering measures of stress and strain are denoted by the following sigma (σ) stress and epsilon (ϵ) strain. Stress is used to express the loading in terms of force applied to a certain cross-sectional area of an object, stress is the applied force or system of forces that tend to deform a body. Strain is a measure of the response of a system to an applied stress i.e. how much it deforms. Engineering strain is defined as the amount of deformation in the direction of the applied force divided by the initial length of the material [18]. When the stress is plotted against the strain, an engineering stress - strain graph Figure 1.2 is produced.

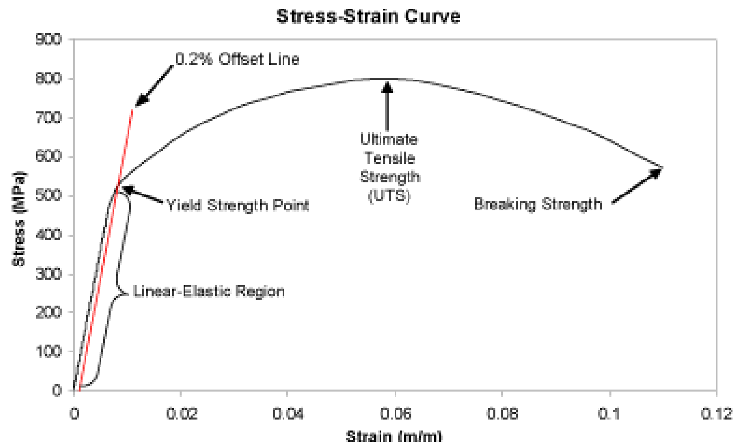


Figure 1.2: Stress-Strain curve.

1.6.2 Static failure

For a material to fail, a force needs to be applied to that material. This will have the effect of the material exerting a resultant force to balance the applied force; and this resultant force will induce a stress in the material. Stress is defined as the force or load being transmitted divided by the cross-sectional area transmitting the load. The most common types of stresses are tensile, compressive and shear stresses according to Black & Kohser [19] and Vernon [20]. Tensile stress tends to stretch or lengthen the material as the load is applied. Compressive stress is the opposite of tensile stress, where the material is shortened or squashed when the force is applied. Shear stress results from two offset forces acting on a body. The amount of elongation or distortion in the material (i.e. the change in length), whether it is positive or negative, is called the *strain*; for this thesis SI units will be used. Strain is expressed in terms of millimeters per metre, but can be expressed in other forms [19]. When the forces applied to a material are constant and time-invariant (or nearly so), they are said to be static (quasi-static). Since static loading's are observed in many applications it is important to characterize the behavior of materials under these conditions. For design and reliability engineers, the strength of the material may be of primary concern, along with the amount of elastic stretching or deflexion that may be experienced under load. Design criterion in the form of yield and fracture theories are available in the literature and salient design codes [21, 22] will be utilized throughout the work that follows, with particular attention being directed to the Huber-Mises and Tresca yield criteria in order to ensure appropriate sizing of pressurized components used in some of the case studies discussed in Chapters 3 and 4. The Rankine (maximum principal stress) criterion is also used extensively throughout the work in order to evaluate likelihood of brittle failure. Such analyses have been performed in some of the work described in this thesis, in order to partially fulfill the project objectives 4 & 5 (§§1.5.2).

1.6.3 Dynamic failure

Rapid technological developments of the present day require increasing application of the principles of mechanics, particularly dynamics. These principles are basic to the analysis and design of moving structures, to fixed structures subject to shock loads, to robotic devices and to machinery of all types such as turbines, pumps and valves etc [23]. Real products or components are subject to a variety of dynamic loads. These may include sudden impacts or loads that vary rapidly in magnitude, repeated cycles of loading and unloading, or frequent changes in the mode of loading, such as from tension to compression [19].

Whilst a system or component(s) may have been designed to cope with impacts as part of its design function for normal operations, fault sequences can introduce impact loading's from different locations with greater force, outside of a system's normal operating parameters. An example in the nuclear industry would be that of an in-cell bogie that has been designed to take impact loads from above. However, as the bogie is traveling from point A to point B laterally, where a proximity switch fails the bogie will collide with a stationary object, thus, introducing new dynamic loads outside of its normal operating parameters. It is for this reason when a component is designed that a very conservative design factor is applied, especially in nuclear design [24].

The properties and wear methods discussed above contribute to the failure of a component. These need to be fully understood, and how these failures can be accelerated or induced by the way they are manufactured or used within a system also need to be understood. Whence, such techniques will provide data in order to achieve objective 6 (§§1.5.2).

Materials can also fail by fracture if they are subjected to repeated applications of stress, even though the peak stresses have magnitudes less than the Ultimate Tensile Strength (UTS) and usually less than the yield strength. This phenomenon is known as 'fatigue' [19], and is caused when repeated mechanical stress is applied, the stress being above a limiting value called the fatigue limit [15]. Fatigue damage is cumulative. It arises from the cyclic repetition of a particular loading cycle or from entirely random variations in stress (for example, a spring subjected to cyclic extension beyond the fatigue limit will ultimately fail in tension). Almost 90% of all metallic fractures are in some way attributed to fatigue [19, 15]. German railway engineer August Wohler is credited with commencing the modern day study of fatigue. Between 1852 and 1870, Wohler set up and conducted the first systematic fatigue investigation. Very recently, fatigue analyses has become quite routine for top-of-the-range simulation codes such as ANSYS; together with well established probabilistic design for six-sigma techniques, such methodologies have been used to provide evidence of objective 5 (§§1.5.2).

Chapter 2

Literature Review

Simulation plays a crucial role in today's design engineering processes, from analysing the structural damage caused by earthquakes on bridges, to examining the effects on vehicle integrity during a collision. Before the advent of the Finite Element Method (FEM) these tasks would have been very complex and time consuming, if not impossible. Finite Element Analysis (FEA) or Finite Element Modelling (FEM) is a numerical method for solving problems of engineering and mathematical physics, using a computational technique to obtain *approximate* solutions. With the advancement of simulation, modelling, safety and costing tools over the past decade, it is now possible to use simulation software such as (ANSYS [25]) to construct a more robust argument/decision in the design stage, in order to improve the reliability, performance safety and life-cycle of equipment and components; as models can be changed and the design verified and validated before going to manufacture.

By using powerful analysis tools (such as ANSYS) to model the proposed designs, one can see how the designed SSC reacts under loading conditions, and determine the weak points in the design. As per objective number three (§§1.5.2), any ANSYS modelling will be subjected to a rigorous benchmarking regime implying verification with analytical modelling and where possible, validation with empirical data evident in the literature or otherwise. The simulations can be run over a few hours to provide data that could take years of expensive prototype testing to achieve.

The Design for Six–Sigma (DFSS) ANSYS workbench module will be employed for the application of probabilistic design. This analysis technique determines the probability that a design criterion is no longer met, and identifies which product variables contribute most to the scatter of a product parameter, enabling design decisions to evaluate the required mechanical reliability [26]. Baril *et al* [27], aims to reduce the number of defects in a manufacturing process. Design for Six Sigma (DFSS) is a powerful approach for designing products in a cost effective and simple manner to meet customer goals and expectations [28, 27]. Koch *et al* [9] define the two goals in DFSS as:

1. Striving to maintain performance within acceptable limits, consistently (reliability).

2. Striving to reduce performance variation and thus increase robustness.

Such state-of-the-art techniques will be employed in this project via the use of the open-source MAXIMA Computer Algebraic System [29] and SciLab [30] numerical analysis code; the details of these being presented in the proceeding chapter.

2.1 Failure criteria

In this section a brief description of the failure criterion is presented. With particular attention being directed toward the prediction of the yielding and fracture of engineering materials.

2.1.1 Brittle failure criteria

The so called maximum principal stress failure criterion is normally attributed to W. J. M Rankine (1820 -1872) [31]. The theory states that a brittle material will fail when the maximum principal stress exceeds some value, independent of whether other components of the stress tensor are present. The Rankine criterion assumes that the material undergoing deformation exhibits identical strengths in both compression and tension. A more realistic criterion is that of Mohr's which neglects this restriction allowing for different failure stresses in compression and tension:

$$\max\left(|\sigma_1|, |\sigma_2|, |\sigma_3|\right) > |\sigma_f| \tag{2.1}$$

hence the Coulomb-Mohr factor of safety is given by:

$$\frac{|\sigma_f|}{\max\left(|\sigma_1|, |\sigma_2|, |\sigma_3|\right)}$$

where $\sigma_{i=\{1,2,3\}}$are the principal stresses [32, 18] and σ_f the failure in tension or compression, depending on mode of failure under consideration. In the work that follows only the tension failure need be considered as this corresponds to crack opening mode-I failure as per the salient ASME code [21].

2.1.2 Ductile failure criteria

Metals are more renowned for ductility due in the main to their primary metallic bonding that act over larger distances than their ionic and covalent cousins since these materials generally sit in a *sea* of electrons which act like a glue at the molecular level; known to be polycrystalline at the micro-structural level. As a general rule the larger the crystals in the microstructures the more ductile and malleability of metals. Moreover dislocation theory has shown that this plastic deformation or plastic flow leads to large and permanent deformation. The objective being to obtain a relationship between stress components and the value of material yield stress, to determine critical loads or load combinations that cause initial yielding (*failure*) at a point in a material. Axially loaded bar will have initial yielding or plastic deformation once it is loaded until the axial stress reaches the yield stress. Since the so called dislocations effectively *move* through the solid ductile failure is generally associated with large absorption of energy, hence plastic deformation.

Depending on the nature of the metal different types of heat treatments can change the microstructure of the metals increasing (or decreasing) the deformation properties as desired. Heat treatments such as annealing and tempering increase ductility while quenching reduces the crystal sizes in order to harden but embrittle metallic materials. Unlike polymers the crystallinity is not just related to temperature but also the time exposed to a particular temperature. i.e. reducing the strength of the metallic bonds holding together the individual crystals.

Tresca criterion

The first of the theories for ductile failure was postulated by the French Engineer Henri Tresca. He is most renowned for a series of quite brilliant experiments investigating non-recoverable deformation while a professor at the world famous *Conservatoire National des Arts et Mėtiers* in Paris from about 1864. The yield criterion that bears his name is based on dislocation theory [16]. Tresca showed that many metals have a yield point and the onset of plastic deformation at this point. It is therefore assumed that yielding occurs when the absolute maximum shear stress at a point reaches the value of the maximum shear stress to cause yielding in a tensile test, whence:

$$\frac{\sigma_y}{2} = \frac{\sigma_1 - \sigma_2}{2}$$

therefore the material is deemed *failed* when:

$$\frac{\sigma_1 - \sigma_2}{\sigma_y} > 1 \tag{2.2}$$

therefore the Tresca safety factor is given by:

$$\frac{\sigma_y}{\sigma_1 - \sigma_2}$$

Huber-Mises criterion

Huber in 1904 suggested that yielding ductile material should not only be related to the stresses imposed upon them but also the elastic strain-energy and the shear strain energy. This was also independently postulated by Maxwell[1], Hencky and Von Mises.

$$2\sigma_y^2 = (\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_2 - \sigma_3)^2$$

The Von-Mises stress is used by design engineers to determine when a ductile material will fail i.e. will it withstand a given load condition. A material is said to start yielding when its Huber-Mises stress reaches a critical value known as the yield strength. Engineers can say that their design will fail if the maximum Von-Mises stress induced into the material is more than the yield strength of the material. In engineering and materials science Von-Mises stress is also known as Von-Mises yield criterion and equivalent tensile stress [21]. Using this criterion the material is deemed

[1]The genius Scottish mathematician and theoretical physicist, conducted many experiments of electricity, magnetism and even optics into a consistent theory. His set of equations—Maxwell's equations—demonstrated that electricity, magnetism and even light are all manifestations the same way: electromagnetic field. Maxwell's work in electromagnetism has been called the "second great unification in physics" after the first one carried out by Isaac Newton.

failed:

$$\frac{1}{\sigma_y\sqrt{2}}\sqrt{(\sigma_1-\sigma_2)^2+(\sigma_1-\sigma_3)^2+(\sigma_2-\sigma_3)^2}<1 \tag{2.3}$$

therefore the Huber-mises safety factor is given by:

$$\sigma_y\sqrt{2}\left\{(\sigma_1-\sigma_2)^2+(\sigma_1-\sigma_3)^2+(\sigma_2-\sigma_3)^2\right\}^{-\frac{1}{2}}$$

2.1.3 Fracture & fatigue

A common method at the design stage for predicting the fatigue life of steel structures is the use of the S-N curve using design data from a standard, such as BS 7910: 2005. However, a fatigue life assessment for existing structures with defects of a known or postulated size is largely carried out using fracture mechanics [33, 34]. This type of analysis requires the use of an appropriate crack growth model such as the Paris law, or a two stage crack growth relationship (where the relevant parameter values are known for the type of material being used and the operating environment [33]. Good engineering practice normally calculates a conservative safety factor [35]).

More advanced reliability based methods [36] can be used to determine the remaining life corresponding to one or more probabilities of failure e.g. 10^{-4}, 10^{-6}; an alternative way is to estimate the probability of failure for one or more specified time periods (e.g. $1, 10$ or 20) years of continued service life. Research carried out by Wang et al *[37]* looking at analysis and fatigue life prediction of cylinders postulated that fatigue damage caused by high working stresses and corrosion is the main reason of cracking.

Measures to extend fatigue life Engineering disasters can result from fatigue, such as that of the Comet airliner in the 1950's where investigations found the fuselage failure was caused by fatigue cracking originating from the rear ADF window. The corner of the square window design acted as a stress raiser and after repeated fuselage pressurisation and depressurisation, cyclic stress, the fatigue crack caused catastrophic failure of the airliner. However, this failure has provided the engineer with invaluable knowledge to try and limit and extend fatigue life (i.e. the introduction of round windows into aircraft design to eliminate stress raisers). Some even think the failure of the comet airliner has contributed more to airline safety than any other aircraft [38, 39].

Additional measures that may be taken to extend fatigue life include:

- Reducing mean stress levels.
- Eliminating sharp surfaces discontinuities, putting radii on where materials join.
- Improving the surface finish by polishing and shot peening or case hardening.

Evidence that fatigue crack initiation can be attributed to machining grooves coupled with a high mean stress level due to bolt torqueing, was observed by Byrnes & Lynch [40] when investigating unusual failures of hydraulic valves; whereas fatigue life enhancement research has been conducted by Duncheva et al *[34, 41]* by identifying potential places for initiation and growth of first–mode fatigue cracks and carrying out FEM analysis in order to identify key optimal values of the stiffened supports.

Fatigue Modelling Extensive fatigue modelling research has been undertaken over the years [42, 40, 43, 44, 41] (HSE paper) using various methods to understand the critical factors that influence the integrity of materials in service and to provide accurate predictions of service life [42]. Low cycle fatigue failure criteria are based on the stored energy which accumulates in the material microstructure during cyclic loading. Fekete's [44] experimental work was conducted on two types of reactor pressure vessel material. The result is higher prediction accuracy than by classical strain amplitude and strain energy based approaches [45]. Here, in some cases, high levels of plastic strain are established across the flawed industry standard power and pipeline sections. Validation of these approaches based on elastic-plastic cracked-body FEA of plates, and importantly to the work herein cylinders are presented.

Safety factor All engineering materials exhibit variability in their mechanical properties due to the presence of random defects in the microstructure. Additionally, uncertainties will also exist in the magnitude of the applied loads for in service applications. Ordinarily stress calculations are only approximate though for safety purposes in general conservative. Therefore, allowances in design must be made to protect against unanticipated failures [35]. This is achieved by calculating, for the particular material used, a safe stress, denoted as σ_w. For static situations and ductile materials, σ_w is taken as the yield strength divided by a safety factor, say N. It is important to specify an appropriate value for N, as with too large a value then component over-design will result: either too much material will be used (making the design heavy) or an expensive alloy will be used (having more than the necessary strength). Values of safety factor normally range between 1.2 and 4.0; a good average is 2.0 [18]. The approximate value of N will depend on numerous factors (i.e. economics, previous experience, the accuracy with which mechanical forces and material properties may be determined, etc).

2.2 Pressure vessel design

A pressure vessel is a container designed to store gases or liquids at pressure substantially different from the ambient pressure. Pressure engineering technology is of importance in many branches of industry, including nuclear, oil & gas and the chemical industry. Pressure vessels can range from simple mass produced vessels to large custom built vessels and tanks. However, it is important that an understanding of the structural integrity of these pressure vessels is understood at the design stage of a project and through the operational life phase (using maintenance and inspection history data), so that life extensions can be substantiated if the business requires on going operation. The field of pressure vessel technology development includes contributions on the following subjects: Pressure vessel engineering, structural integrity assessments, design methods, codes and standards, fabrication, welding, material properties, maintenance & inspection history, ageing and life management including environmental effects. Continuing research and development and case studies in the practical application in the above disciplines could lead to improvements in economy, reliability and the useful life of pressure vessels. The manufacture, construction and operation of pressure vessels is governed by codes, standards and regulations developed by different governing bodies such as the American Society of Mechanical Engineers (ASME), British Standards Institute (BSI) ensuring that such vessels are engineered to provide the required design function. Research in the field of pressure vessel and piping equipment failures and the causes is extensive and covers the oil and gas and nuclear industry to name a few. Fatigue evaluation performed by Rondan and Guzey following ASME BPVC Section VIII, Division 2 [21] , design-by-analysis rules, was used to investigate and establish service life due to cyclic loading using finite element analysis to determine the location of the most critical joints [46]. The evaluation of fatigue crack growth has been conducted to evaluate the crack growth rate in nozzle corners using Finite Element Modelling (FEM) [47]. Chapuliot used FEM in determining the stress intensity factors along the crack front for pressure loading in order to cover geometric defect sizes and loading situations encountered by large nuclear components [48]. Wu et al [49] studied better candidate steel for the next generation of Reactor Pressure Vessel (RPV) through a combination of experiments and the Finite Element Method (FEM).

2.2.1 Design codes (ASME)

Prominent engineers of the day Alexander Lyman Holley, Henry Rossiter Worthington and John Edison Sweet recognized that safety, reliability and operational efficiency would be ensured by providing engineers and designers with a set of engineering standards, establishing the Boiler Testing Code in 1884 [50].

The late 19th century witnessed the development of steam powered technology; however, despite their power, boilers and pressure vessels were temperamental, requiring constant maintenance and attention. There were no legal codes for boilers during this period even though numerous boiler explosions had occurred. However, this thinking undoubtedly changed in 1905 when a boiler explosion occurred at the Grover Shoe factory in Brockton, Massachusetts, resulting in 58 deaths and 117 injured and completely levelling the building. This disaster brought attention to the need to protect the public against such accidents from pressure equipment [22].

The ASME Boiler and Pressure vessel Code (BPVC) was conceived in 1911 out of the need to protect the safety of the public from disasters such as the one mentioned previously. The first BPVC was published in 1915 (1914 Edition), and consisted of one book, today there are 28 books, including 12 books dedicated to the construction and inspection of Nuclear Power Plant Components and two code case books [22].

The early 20th centuries drive to use nuclear energy for commercial power generation made engineers aware of the need for a set of design and fabrication rules to facilitate the development of safe economically competitive nuclear reactors. The number of similarities between a thermal-neutron reactor and steam powered pressure vessels meant that the nuclear industry relied on the ASME BPVC section I *Rules for Construction of Power Boilers* and VIII *Rules for Construction of unfired pressure Vessels* to help standardize their practices. ASME published the BPVC section III, *Nuclear Vessel* in 1963. This has now been expanded to cover practically all pressure and liquid storage components involved at a nuclear power site [50, 22].

The ASME code is based around avoiding excessive plastic deformation, by keeping maximum shear stresses (a close equivalent to the Huber-Mises stress) below chosen limits,

- Terms relating stress analysis
- Elastic-plastic analysis method
- Elastic plastic

The ASME code details a five stage assessment procedure to determine the acceptability of components using elastic -plastic stress analysis [21].

Step-1 is about developing a numerical model of the component detailing all the relevant geometry characteristics. The model selected for use in the analysis shall accurately represent the component geometry, boundary conditions and applied loads. Additionally, refinements of the model around areas of stress and strain concentrations shall be provided. To provide a more accurate description of the stresses and strains achieved in the component more than one numerical model may be required.

Step-2: is about defining all the relevant loads and applicable load cases.

Step-3 An elastic-plastic model shall be used in the analysis. If plasticity is anticipated the Von-Mises yield function and associated flow rule should be utilized.

Step-4 Using the information defined in step 2 the load combinations need to be determined in conjunction with Table KD-230.4, in the ASME code [21]. Each of the indicated load cases shall be evaluated.

Step-5 An elastic-plastic analysis is performed for each of the load cases defined in step 4. The component is stable under the applied loads for this load case, if convergence is achieved. If not, the component configuration (i.e. thickness) shall be adjusted or the applied loads reduced and the analysis repeated.

2.2.2 Plastic collapse

Work hardening is a mechanism that occurs in crystalline metals, manifests as a rise in the stress required for continued plastic deformation. In light of section 2.1.2 it is obvious that the modelling of the underlying mechanisms relating to work hardening are very difficult to formulate mathematically, hence the exact nature of the energy absorption required to cause such deformations. This is evidenced by Cottrell [51], who states that:

> *it is sometimes said that the turbulent flow of fluids is the most difficult remaining problem in classical physics. Not so. Work hardening is worse.*
>
> Cottrell [51] 2002

This said in the work that follows sensible decisions needed to be taken with regard to sizing of pressure vessel wall thickness, the evaluation of the on set of yielding and the so called plastic collapse of the vessel is of utmost importance. The aforementioned five stage process maintains that the equilibrium equations from classical elasticity theory [32] should be modified to:

$$\sigma_r(r) + \sigma_\theta(r) + r\left(1 - \frac{1}{m}\right)\frac{d\sigma_\theta(r)}{dr} + \left(\frac{r}{m}\right)\frac{d\sigma_r(r)}{dr} = 0$$

where σ_r is the radial stress, σ_θ the hoop stress, r is the spacial radial coordinate and m is the gasket factor as defined in the ASME code [21]; this being analogous to a safety factor. Assuming the yielding takes place when the stresses in accordance with Tresca's criterion; this being in fair agreement with experimental results for ductile materials [18], then a gasket factor of unity is apparent, whence:

$$\sigma_r\left(r\right) + \sigma_\theta(r) - r\frac{d\sigma_r\left(r\right)}{dr} = 0$$

when an internal pressure is applied the first two principal stresses have opposite signs, assuming a perfect elastic plastic material response then Tresca's criteria implies [18]:

$$r\frac{d\sigma_r\left(r\right)}{dr} = \sigma_Y$$

where σ_Y is the material's yield stress, the solution being:

$$\int_{\sigma_r(r)}^{-p_{ep}} d\chi = \sigma_Y \int_r^{R_P} \frac{d\xi}{\xi}$$

where p_{ep} is the inter-facial elastic-plastic pressure and R_p is the distance to the edge of the plastic region. Carrying out the integration gives:

$$\sigma_r(r) = \sigma_Y \ln\left(\frac{r}{R_p}\right) - p_{ep}, \tag{2.4}$$

and applying Tresca's criteria renders:

$$\sigma_\theta(r) - \sigma_Y = \sigma_Y \ln\left(\frac{r}{R_p}\right) - p_{ep};$$

that is:

$$\sigma_\theta(r) = \sigma_Y \left\{1 + \ln\left(\frac{r}{R_p}\right)\right\} - p_{ep}$$

Application of these two expressions together with the aforementioned Tresca yield criterion, are used to determine the pressure required to initialize yielding:

$$p_y = \frac{\sigma_y}{2R_o^2}\left\{R_o^2 - R_p^2\right\}. \tag{2.5}$$

These were therefore used in the work described throughout in order to evaluate the onset of yielding and the plastic collapse loads as detailed in section (5.2).

2.3 Protective device modelling

A Protective Device (PD) is one of the most critical parts of any pressure system. The designation of a PD covers a variety of specific equipment types the main ones being pressure relief valves, pressure-vacuum valves and rupture discs. The general engineering interpretation of what constitutes a PD is [52]:

- A device designed to protect against pressure system failure.
- Which in doing so, prevents a dangerous situation from occurring.

To this end PD are classed as safety – critical items of engineering equipment and are governed by codes, standards and regulatory requirements covering pressure systems worldwide.

A point to note is that there is no single agreed set of terminology governing PDs, particularly between US and European PD codes. The main area of confusion centring on the use of the following terms:

- Relief valve.
- Safety relief valve
- Pressure relief valve.
- Pressure safety valve

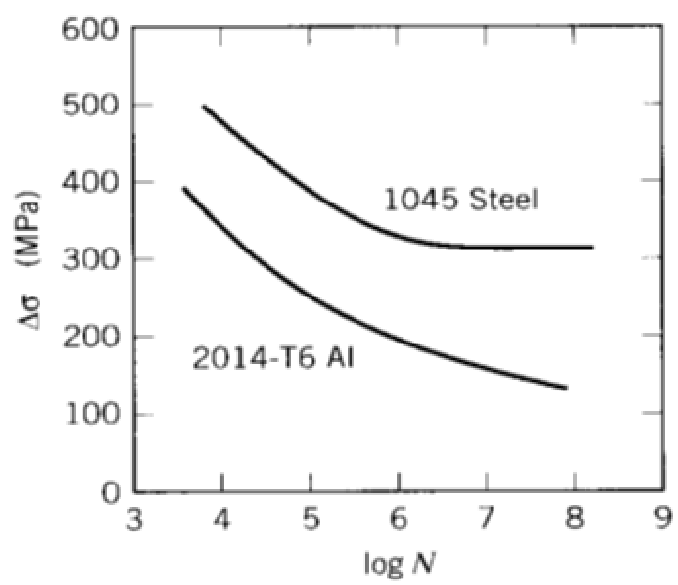


Figure 2.1: S-N curve

This thesis uses the terminology Pressure Relief Valve (PRV) to keep it simple. There are minor design detail differences between the four mentioned types but they are of secondary importance [52].

A common method at the design stage for predicting the fatigue life of steel structures is the use of the *S-N* curve, example shown in Figure 2.1 using design data such as BS 7910:2005. However, a fatigue life assessment is largely carried out using fracture mechanics on existing structures containing defects of a known or postulated size. To perform this type of analysis it is necessary to use an appropriate crack growth model (e.g. a two stage crack growth relationship or the simple Paris law), the loading environment and the values of the relevant parameters for the type of steel being used. It is usual to calculate conservative (safe) fatigue life estimates in line with normal engineering practice [34].

2.4 Reliability

Reliability can be defined as the probability that an item will perform a required function, under stated conditions, for a stated period of time [53]. However, as mentioned in the previous chapter (§§1.1.2), estimating or predicting the reliability of mechanical components is a difficult but important task, as Reliability and Maintainability (R&M) are vital factors in today's complex systems [54]. A reliability prediction is performed in the early stages of a development programme to support the design process. The reliability prediction provides the designer with the reliability requirements. The reliability prediction is a key part of the design process as this provides an awareness of the potential equipment degradation during its life-cycle [55]. Thus, equipment designs can be improved, costly re-designs prevented and Research and Development (R&D) work optimized as a result of a well done reliability prediction.

The focus of this work is mechanical reliability, as reliability prediction of electronic equipment is well established (e.g. the influential MIL-HDBK-217 [56] first published in 1961, has been developed for predicting the reliability of electronic equipment and is still in use today). Standardization and mass production of electronic parts also made it possible for the creation of valid failure rate data banks for high population electronic equipment [55], although care around duty cycles and operating context is required when working with the data.

Reliability engineering is essentially an analytic application of probability and statistical theory. Such was initially used to satisfy the answers to gaming and gambling questions by Blaise Pascal and Pierre de Femat in the 1600s and it was later expanded by Laplace in the 1800s [54, 3]. Today reliability engineering is a well established, disciplined engineering practice focusing on methods to investigate the uncertain boundaries between system operation and failures. System failures and the subsequent failure investigations have become increasingly important in today's society. There are liability issues, as exemplified by British Petroleum (BP) paying out $11.6 billion through various claims from May 2010 to the end of 2014 for the Gulf of Mexico oil spill [57]. An important reason for conducting a failure investigation like the April 1998 Aloha airlines incident (when approximately 18ft of cabin skin seperated from the airplane during flight), is to identify the mechanisms and cause of failure to prevent its re-occurrence [58]. Neglecting to identify the underlying cause of the failure (which may be due to fatigue, wear, cavitation, or corrosion) and neglecting to take corrective action to rectify the problem can expose an organization to litigation, liability and loss of customer and/ or public confidence [3]. A pipeline rupture due to fatigue cracking led to the release of 11,644 barrels of crude oil onto a golf course and into Two mile creek in the U.S in 2000. Although there where no fatalities or injuries, over $7 million dollars in compensation was spent [59]. These risks are unacceptable to companies in the modern global competitive business environment [6].

2.4.1 Weibull analysis

Swedish mechanical engineer, Professor Waloddi Weibull invented the concept of the Weibull distribution in 1937. His hallmark American paper was published in 1951 [60]. In it, he claimed that his distribution applied to a wide range of problems, which he illustrated using seven examples (one of which looked at the strength of steels and another the height of adult males in the British Isles. Weibull postulated that function may sometimes render good service; however, he did not claim that it always worked. Time has shown that he was correct on both of these counts [61].

The initial reaction to his paper was negative. Weibull's claim that data could be used to select the distribution fit of the parameters seemed too good to be true.

Over time, it became widely appreciated and was shown to have some application to the analysis of defect data by Pratt and Whitney in 1967. The U.S. Air Force recognized Weibull analysis's merit and funded research until 1975 [62, 53].

Weibull analysis is a well-known reliability assessment method that is applied to the time to failure data to assess the mean life of components. A paper presented by Xie *etal* [63] in Reliability Engineering & System Safety presented a simple Weibull model that demonstrated the usefulness of applying it to lifetime distributions for components with a bathtub-shaped failure rate function. More recently Mazhaz *et al* [64] have been applying Weibulll analysis (time to failure data) to assess the mean life of components; and combining it with degradation and condition monitoring data to develop an Artificial Neural Network (ANN), then using this to assess the remaining life of components with further recommendations to reduce downtime of SSC's, using this technique to better manage preventative maintenance [64].

2.4.2 Maintainability & Risk

An important point to remember is that there is a difference between maintainability and maintenance. Maintainability is part of the designer's function ensuring that maintenance can be carried out on the correct SSC. Maintenance is the actual (hands on) task(s) carried out by the maintenance teams to keep the SSC operating to design specifications [65]. Maintenance is often the most efficient way to keep a system operating through its life-cycle and is a crucial component in any company's strategy [66, 67]; as it sustains the company's reputation in the eyes of the public [68, 69], regulators and potential customers [70, 71]. Effective maintenance therefore allows for an organization to deliver safe and reliable operation of its assets, whilst minimizing the risk of unplanned and undesired events [58]. Beaurepaire *et al* and Nader *et al* [72, 73] support this argument, stating that by scheduling in maintenance activities performance of SSC's will be improved and kept within acceptable design limits; and that the harmful effects of fatigue cracks can be avoided including:

- injury to both employees and general public.
- environmental damage.
- increased operating costs.
- loss of reputation and adverse publicity.

Additionally, by having an effective maintenance strategy in place that demonstrates how a company can deliver the required reliability (both now and in the future) one can help reduce maintenance costs [73, 71], which can be a big driver in the world today. This can be achieved by:

- designing out recurring problems, making an SSC more reliable.
- minimizing the secondary damage as a result of allowing equipment to fail.
- avoiding unnecessary maintenance.

A major concern for companies is the desire to avoid simply carrying out maintenance just for the sake of it, or because it has always been done like that, so a more objective basis for assessing the maintenance requirements is desirable when planning for maintenance: using Fault Tree Analysis (FTA), Failure Mode Effect Analysis (FMEA), Reliability Centered Maintenance (RCM) etc.

2.4.3 FMECA

Failure Mode Effect and Criticality Analysis (FMECA) is a set of methodologies designed to identify potential failure modes for a product or process. It is used to assess the risk associated with these potential failures, and to rank the issues in order of importance [74, 75]. FMECA requires the identification of some basic information:

Item. The SSC to be analysed.

Function. This is the purpose of the item (e.g. transfer a quantity of water from tank 1 to tank 2 in a certain time)

Failure. This is any event which will cause none performance of the SSC.

Effects of failure. These are the descriptions that should state whether the failure causes warning or raises alarms to sound, and what happens when the failure mode occurs.

Causes of failure. These are the potential root causes of the failure (e.g. a blocked fuel line).

Current controls. These are the tests, procedures or mechanisms that are already in place, to reduce the likelihood of the failure occurring or to detect the failure.

Recommended actions. These identify what must be done to repair the failure and reduce the risk of repeat failure.

Once the issue(s) have been identified by the analysis, a method to identify the risk associated with the issue needs to be carried out, to prioritize the corrective action required to remove the risk. Detailed below are some methods used to achieve this:

Risk Priority Number (RPN). That is, a parameter to provide guidance for ranking potential failures in the order that they should be addressed.

Criticality Analysis. This is the procedure by which each potential failure mode is ranked according to the combined influence of severity and probability of occurrence.

	Calamity	Catastrophic	Critical	Marginal	Minor
Frequent					
Probable					
Occasional					
Remote					
Improbable					

Figure 2.2: Criticality matrix example

The RPN is calculated by rating the severity of each failure, the likelihood of occurrence of each failure and the likelihood of detecting each failure.

$$RPN = SOD$$

where S is the severity, O the occurrence and D is the detection.

Criticality analysis as described in MIL-STD-1629A [76] involves both qualitative and quantitative analysis. The qualitative analysis is carried out using a criticality matrix (Figure 2.2), with the severity identified on the horizontal axis and chance of occurrence identified on the vertical axis. Note: 'criticality' in this context is looking at the consequences of failure (i.e. it does not refer to a chain reaction).

The quantitative analysis considers the reliability/unreliability of each identified item at a given operating time and identifies the portion of the item's unreliability that can be attributed to each potential failure mode.

The FMECA process is in widespread use throughout industry for numerous purposes. It can be used to contribute to improving designs and it can result in higher SSC reliability, improved quality, reduced costs and increased safety; leading to better public perception, which is vitally important for industries such as nuclear (as public confidence remains fragile, post-Fukushima [77, 78]). However, public perception of the nuclear industry in the UK remained stable in the wake of Fukishima [79].

2.5 Design for Six Sigma

The Six Sigma concept is to eliminate defects through use of statistical tools within a structured methodology [26, 80] . The term *sigma* refers to the standard deviation σ or variance σ^2, and is a measure of the dispersion of a set of data around a mean value X . These are the parameters of the normal distribution.

The Six Sigma approach aims to reduce the number of defects in a manufacturing process, Design For Six Sigma (DFSS) is an approach for designing products, processes and services to meet or exceed customer requirements. The main goals of DFSS are [27, 81, 65]:

1. Endeavor to maintain performance within acceptable limits, consistently (reliability).

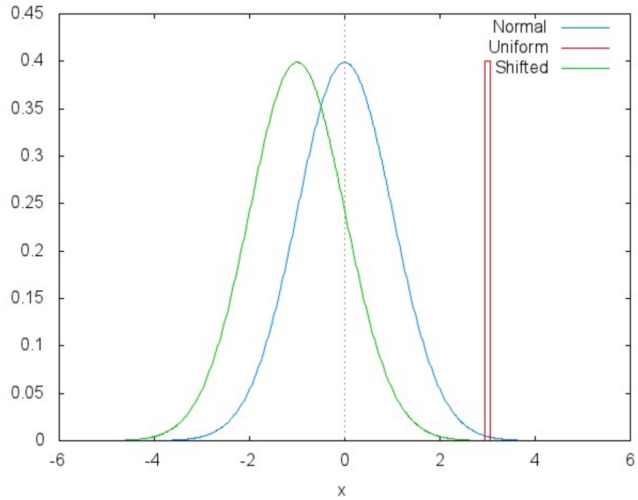
2. Striving to reduce performance variation and therefore increase robustness.

The graphs illustrated in Figure 2.3 illustrates this concept. The graph in Figure 2.3a shows by shifting the performance distribution relative to the constraint or specification limit, reliability is improved. The graph in Figure 2.3b illustrates that by shrinking the performance distribution to reduce the variability and sensitivity the design robustness is improved. The graph in Figure 2.3c demonstrates the goal to achieving improved design quality in DFSS to both shift and shrink a performance distribution. [27, 81]. Details of the 4 stages, identify, design, optimize and validate are detailed in some depth in [28].

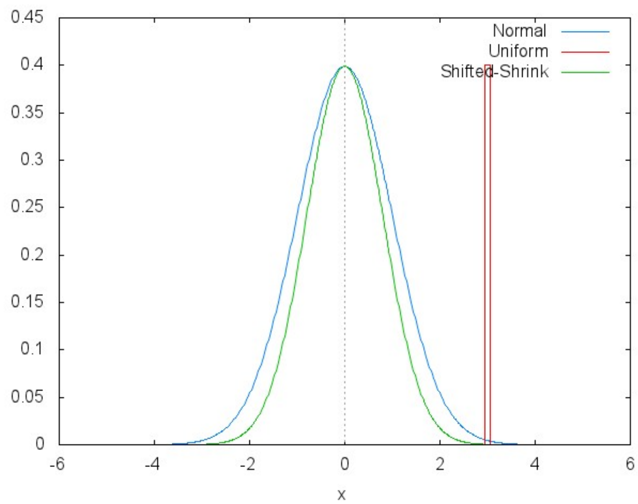
However, it is important to understand that there are differences between Six Sigma and DFSS:

- Design, Measure, Analyse, Improve, Control (DMAIC) is focused on reacting to and resolving problems, while DFSS is used more proactively as a means to prevent problems.

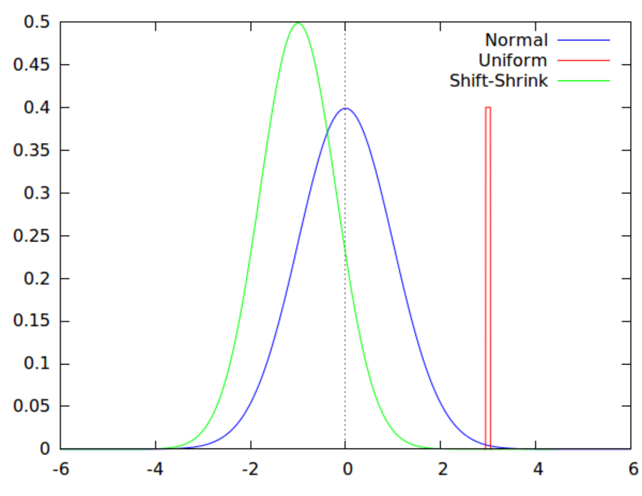
- DMAIC is for products or services that are in use currently; DFSS is used to



(a) Shift



(b) Shrink



(c) Shift & Shrink

Figure 2.3: Graphs illustrating (a) shift “reliability”, (b) shrink “robustness” and (c) shrink and shift together to achieve the desired goal of DFSS.

design new products or services.

- DMAIC is based on manufacturing and DFSS is more focused on design and Research and Development (R&D).

2.6 Reliability based design optimization

Reliability Based Design Optimization (RBDO) is a development of deterministic multiobjective optimization techniques [27]. The RBDO aim is to assess the probability of failure of a design with respect to the specification performance constraints and the evaluated variation of the constraint function [81]. The end goals of RBDO are to drive the mean performance toward the target and to minimize variance of performance. Whereas, the deterministic approach does not consider the impact of uncertainties due to variation in design and conditions, making the designed solution very sensitive to these variations. There are numerous methods for calculating the probability of failure, which were divided into 4 levels by Madsen & Egeland [36].

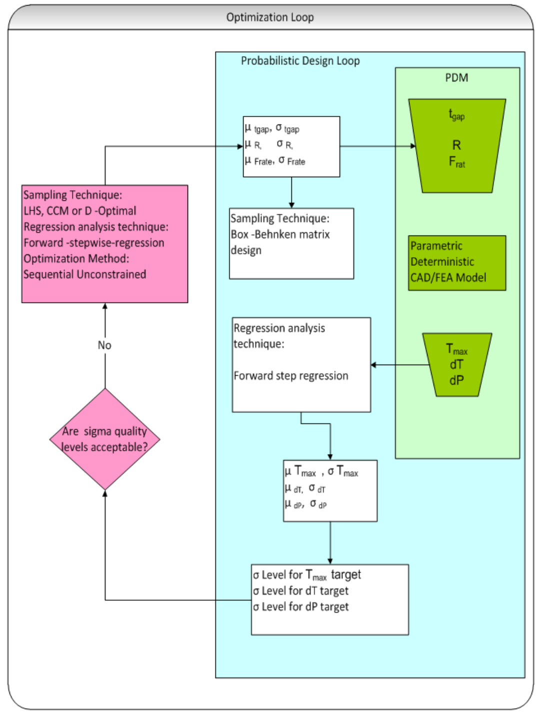
Level 1 methods use a deterministic result to get a reliability estimate.

Level 2 methods use two values of response to determine the reliability of a system.

Level 3 methods need an understanding of the joint distributions of inputs and responses; these are needed to calculate a probability of failure.

Level 4 (being the highest level) considers non-typical metrics when formulating a design, (e.g. cost of production, maintenance and repair).

Vlahinos [82] uses an optimization loop that shows a DFSS technique that incorporates FEA probabilistic and robust design tools within a Computer Aided Design (CAD) environment, illustrated in Figure 2.4. The assumption is that all the design



Source: [82]

Figure 2.4: Example of optimization loop.

variables have variation, and exhibit normal distribution and standard deviation values. In practice this means that a great deal of effort is required in obtaining realistic data using salient statistical sampling to characterize stochastic failures of components in service which is then fed in to the probabilistic design loop. The results of which are fed into the parametric FEA model for further stochastic analysis to be performed in order to establish realistic failure bounds.

As discussed earlier DFSS is a statistical method for reducing quality defects by developing designs that deliver a given target performance despite these variations [81]. To achieve this level of quality requires a determined effort in the development stage, design optimization driven by integration of DFSS into the process and rigorous use of simulation [83]. ANSYS DesignXplorer software is a powerful simulation tool that can be utilized to carryout these simulations. The advantage of using such simulation software is that all design parameters can be set to analyse all possibilities, including all those that may push the design beyond its limits, one may identify constraints showing the weakness in a design, without having to spend large sums of money to actually build and test a prototype.

2.7 Finite Element Analysis (FEA)

This has become the prevalent technique used for analysing physical phenomena in the field of structural and solid mechanics, and is also a popular choice for analysis of fluid mechanics. In order to be able to accurately predict the performance of systems and components, mathematical models need to be constructed and evaluated. The term Finite Element Method (FEM) was first used in 1960 by Clough [84], where the elastic response of a complex structure was evaluated via the assemblage of fundamental bar and beam elements.

However, an earlier paper presented by Turner *et al* [85, 86] in January 1954 at a meeting of the Institution of Aeronautical Sciences in New York is to many the start of the engineering Finite Element Method. With Finite Element Analysis essentially being referred to as the application of a suite of numerical analysis techniques to analyses a wide variety of forcing functions visited on a structure. The rapid development of the methodology from those early days can be attributed to the meteoric rise in computing power that permitted realistic calculations on a previously unparalleled scale [86, 87].

2.7.1 Finite Element Method (FEM)

The fundamental principles of this method are essentially an extension of the Ritz method as introduced by Courant [88], which involves the minimization of functionals formulated though variational calculus. The term Finite Element was probably penned by Turner *et al.* [89] whom generalized the previous method described to a continuum geometrical domain so that stiffness and deflexion of different shaped structures could be effectively analysed. In essence, a geometric domain is discretized in to a number of finite elements (referred to as mesh) each being in adherence to the world famous equation [90]:

$$k_i = \iiint \overrightarrow{B}[\mathbf{D}]\overrightarrow{B}^T dV \tag{2.6}$$

Where k_i is the stiffness matrix of the individual element within the Finite Element (FE) mesh, $[\mathbf{D}]$ is the elastic coupling, usually taking the form of the compliance matrix, though may be adjusted depending on the continuum mechanics to be simulated and $\overrightarrow{B}$ is a vector of shape-functions. The shape functions can be envisaged as salient interpolation functions used to effectively approximate the primary (or in some cases the secondary) solution field. In this thesis, the primary solution field is the displacement at each of the nodes and the secondary the strain field, (i.e. the numerical differential of the displacement field). The matrix $[\mathbf{D}]$ takes the form of the isotropic classical elastic compliance matrix:

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \end{bmatrix} = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & 0 \\ & \frac{1}{E} & 0 \\ & & \frac{1}{G} \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \end{bmatrix} = [\mathbf{D}]^{-1} \sigma \tag{2.7}$$

Where, $\boldsymbol{\varepsilon}$ is the three-dimensional strain vector, E is the elastic modulus of the material, G is the rigidity (or shear) modulus, ν is the Poisson's ratio and σ is the stress in the element.

2.7.2 Software

There is a vast array of both commercial and open source software available to carry out simulation and mathematical modelling. This next section introduces some examples available.

Commercial

Solidworks has been around for a number of years and is mostly used by Small, Medium sized Enterprises (SME), as it is a low cost option. It can be a bit unreliable [91]. COMSOL [92], founded in 1986, in Sweden is a *multiphysics* platform that can be used to model fluid flow, heat transfer and structural mechanics but at present, it has no fatigue modeling options. Abaqus has a convoluted modelling process and user interface and is used mainly in academia (not industry). ANSYS-Workbench, founded in 1970 [93], is regarded as an industry standard used by designers from a number of industries (i.e. power generation, oil and gas and the nuclear industry). It is used for engineering substantiation especially for Computational Fluid Dynamics (CFD), but also for Finite Element Analysis (FEA) of structures, ergo Fluid Solid Interaction (FSI). According to FORTUNE magazine 96 of the top 100 companies use ANSYS [94]. Moreover it has recently been reported that "*modern simulation can empower all engineers to make design decisions based on simulation through out the whole product developed*" [95]. The work detailed in this thesis aims to provide methodologies of how ANSYS can be employed to inform operational maintenance decisions outside the scope of the nominal design process.

Open source

MAXIMA is a system developed from Macsyma. It is a computer algebra system developed in the 1960s at Massachusetts Institute of Technology (MIT). It can be used for the manipulation of symbolic and numerical expressions, differential equations and to plot functions in two and three dimensions; and these are just a few of the functions that MAXIMA can be used for. William Schelter [29] maintained MAXIMA until his death in 2001. Since his death, a group of users and developers has formed to bring Maxima to a wider audience, it is updated frequently, to fix bugs and improve the code and the documentation [96].

Scilab is another open source software package for carrying out numerical computation for engineering and scientific applications, it is widely used to teach mathematics and engineering sciences in higher education institutions . Scilab was developed in the 1990s and was inspired by *MATLAB*, *ForTran* software developed by Cleve Moler [97]. As with *MAXIMA, Scilab* has numerous mathematical functions.

CalculiX is a FEM package designed to solve field problems. The solver is able to do linear and non-linear calculations. FEM can be built, calculated and post -processed. As the solver makes use of the abaqua input format it is possible to use commercial processors also. The CalculiX package was developed by employees of *MTU Aero Engines* in Munich, Germany [98].

The software packages discussed above are a small sample of the packages available to use both commercially and via open source. Due to its availability, credibility and the author's familiarity with the software [2], ANSYS workbench was selected for this research.

2.8 Closure

This literature review has revealed that a vast amount of research has been carried out in the various engineering fields to better improve the understanding of how SSC's fail in most industries from nuclear, oil & gas and military to aviation. The aim being to increase reliability and availability, of the SSC, thus reducing the risk and costs to the business. The importance of the positive influences that these processes have given to the engineering fraternity cannot be underestimated, as they have provided some powerful tools and processes for delivering complex engineering solutions. The remainder of the work described in this thesis will effectively build on these foundations by using modern simulation software to provide engineers and managers with a process to make more informed decisions, by linking the simulation and reliability evaluation methods together, so that the critical characteristics that can affect SSCs are identified and a correct level of rigor be applied to assess the aforementioned predefined research hypothesis [27, 81, 9].

Chapter 3

Methods

As noted in the opening chapter the models used throughout the work described in this thesis, ergo those detailed in this chapter are in essence a proof of the concept. That is, fatigue modelling methods resident in the ANSYS-Workbench simulation code are appropriate in the informing of nuclear safety cases of reliability values. The selection of the SSCs herein therefore was influenced by the availability of analytical solutions and empirical data evident in the literature, though care was taken to identify so called safety critical components. Models of pressure vessels and the particular Pressure Relief Valve (PRV) were also of interest from a strategic research viewpoint from within the John Tyndall Institute for Nuclear Research.

The sizing, materials, mechanical design and operating characteristics of PRVs are chosen with the prime objective of protecting against system over-pressure. The causes and effects of system over-pressure are well documented, e.g.: mechanical failure of the valve, fluid hammer occurs in water, steam and other process systems, closure of outlet valve, instrument failure and external fire. The effects of over-pressure on pressure system components in general are cyclic fatigue effects, decreasing their resistance to rupture. Reduction in overall component strength as existing defects in load bearing materials will grow. Gross deformation, bulging in vessels and fracture of the shell in storage tanks.

This thesis describes the use of finite element software to simulate a number of mechanical properties. To confirm the finite element codes generated accurate results a set of benchmarking procedures were implemented, before the stress, strain and total deformation simulations were carried out to ensure modelling protocols were accurate. Benchmarking used theoretical methods to calculate the stress applied to thin and thick walled cylinders. The models were constructed using $ANSYSv14.5$ and $ANSYSv16.2$. the models then undergo FEA in $ANSYSv14.5$ and $ANSYSv16.2$, the results are then compared to the theoretical values.

3.1 Analytic methods

In this section theoretical models are presented for the evaluation of stresses in the so called thin and thick-wall. In-line with the literature, e.g. [18], the formulae included in the following sub-section refer to thin components as those with aspect ratios $R/t > 20$. Conversely components will be referred to as thick cylinders; more general formula for evaluation of radial as well as hoop and longitudinal stresses in cylindrical components is then treated in section 3.1.2.

3.1.1 Thin-wall cylinders

Since there the wall of the cylinder is thin, form an engineering viewpoint, any radial stresses can be considered negligible. Ergo, the principal stresses (Figure 3.1) can be found by observing that the force is a product of the projected area and the applied internal stress, thus:

$$2RLp = 2tL\sigma_1$$

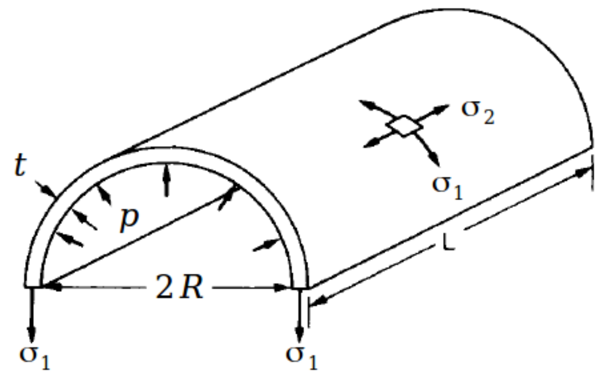


Figure 3.1: Thin-wall cylinder principal stresses

Rearranging renders, the required first principal (hoop) stress:

$$\sigma_1 = \left(\frac{R}{t}\right) p \tag{3.1}$$

Furthermore, using an identical principle at right angles to the this first principal stress, $p\pi R^2 = 2t\pi R\sigma_2$:

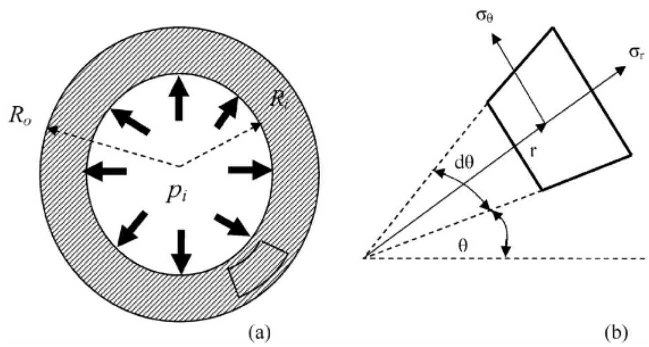
$$\sigma_2 = \frac{pR}{2t} \tag{3.2}$$

It is worth noting that since the radial stress can be assumed negligible then the Tresca yield criterion dictates that yielding, hence plastic-collapse occurs at a pressure of:

$$p_y = \frac{\sigma_y}{R} t \tag{3.3}$$

3.1.2 Thick-wall cylinders

As briefly mentioned in subsection 2.7.1 the degree to which a structure deforms or the magnitude of strains it undergoes depends on the magnitude of the imposed stress (i.e. 2.7). Most metals are stressed in tension and at relatively low levels,



Source [99]

Figure 3.2: Thick-wall cylinder theory stresses

stress and strain are proportional to each other through the relationship:

$$\sigma = E\epsilon$$

This is known as Hooke's law, and the constant of proportionality E is the modulus of elasticity, or Young's modulus.

The equilibrium equation [32] for thick wall pressure vessel contains two unknowns radial and hoop stresses (σ_r and σ_θ) shown in Figure 3.2, [99].

$$\frac{r(d\sigma_r)}{dr} + (\sigma_r - \sigma_\theta) = 0$$

The radial displacement of a cylindrical surface of radius r is represented by u_r. Then, at radius $(r+dr)$ the displacement of a cylindrical surface is$(u + \frac{du}{dr}dr)$. Since, for an infinitesimal element dr undergoes total elongation in the radial direction of $(\frac{du_r}{dr}dr)$, with associated radial strain of[1] :

$$\varepsilon_r = \frac{du_r}{dr}.$$

Thus, the unit elongation of an element in the circumferential direction is equal to the unit elongation of the corresponding radius. The hoop (circumferential) strain can be shown to be:

[1] $\varepsilon_r = \frac{u_r + du_r - u_r}{dr}$

$$\epsilon_\theta = \frac{u_r}{r}$$

Since, change in length of the infinitesimal element in the hoop direction is $(u_r + r)\, d\theta - r d\theta$

Then using Hooke's law, i.e. equation (2.7) with $r = x$ and $\theta = y$:

$$\varepsilon_r = \tfrac{1}{E}(\sigma_r - v\sigma_\theta) \text{ and } \varepsilon_\theta = \tfrac{1}{E}(\sigma_\theta - v\sigma_r)$$

The stress equation can be derived by solving these simultaneously:

$$\sigma_r = \frac{E}{(1 - v^2)}(\varepsilon_r + v\varepsilon_\theta) \tag{3.4}$$

and

$$\sigma_\theta = \frac{E}{(1 - v^2)}(\varepsilon_\theta + v\varepsilon_r) \tag{3.5}$$

The stress equations can be obtained by substitution in to the classical elasticity equilibrium equation [99, 32]; integrating, and then multiplying throughout by radius r. Followed by a further integration thereafter solving for the required displacement field and two constants of integration, C_1 and C_2:

$$u = C_1 r + C_2/r$$

Differentiation followed by substitution into equation in to equation (3.4) and (3.5) gives:

$$\sigma_r = \tfrac{E}{(1-v^2)}[(1 + v)C_1 - (1 - v)\tfrac{C_2}{r^2}] \text{ and } \sigma_\theta = \tfrac{E}{(1-v^2)}[(1 + v)C_1 + (1 - v)\tfrac{C_2}{r^2}]$$

Substituting these equations for constants C_1 and C_2 into the equations produces the radial stress, σ_r and hoop stress σ_θ distribution equations:

$$\sigma_r = \frac{a^2P_i - b^2P_o}{b^2 - a^2} - \frac{(P_i - P_o)}{r^2}\frac{(a^2b^2)}{b^2 - a^2}$$

$$\sigma_\theta = \frac{a^2P_i - b^2P_o}{b^2 - a^2} + \frac{(P_i - P_o)}{r^2}\frac{a^2b^2}{b^2 - a^2}$$

Application of equation (2.5) provides the pressure at which yielding of the inner surface of the vessel:

$$p_y = \frac{\sigma_y}{R_o^2}\left(\frac{R_o + R_i}{2}\right)(R_o - R_i)$$

$$p_y = \frac{\sigma_y}{R_o^2}\bar{R}t \tag{3.6}$$

noting here that in the limit as $R_o \to \bar{R}$ this equation becomes identical to that of the thin-wall model, i.e. equation (3.3). Whilst application of equation (2.4) gives pressure required to render complete plastic collapse of the vessel:

$$p_p = \sigma_y \ln\left\{\frac{R_o}{R_i}\right\} \tag{3.7}$$

3.2 Pressure vessel construction

To verify the modelling protocols used in this thesis a thin and thick walled cylinder have been constructed using ANSYS v 14.5. The following is a brief description of how the models were constructed. The dimensions (in millimetres) are detailed in Table 3.1, it is pointed out that the onset of yielding as evaluated from application of equation (3.6) was 54.69 MPa and 93.75 MPa for the thin and thick wall pressure vessels respectively, *ipso facto* below the plastic collapse load (Table (5.1)).

In $ANSYSv14.5$, first select the static structural icon and open the geometry function to bring up the model construction workbench. The $xy-$plane was chosen to construct the model around. In the sketching platform a *polyline* was used to render a nominal L shaped design as shown in Figure 3.3. The model was dimensioned with the values displayed in Table 3.1. Using the revolve function the L shape was

	Thin walled cylinder (mm)	Thick walled cylinder (mm)
Horizontal	100	100
Vertical	200	200
Thickness	25	50
Radii	25	25

Table 3.1: Thin &Thick walled cylinder dimensions.

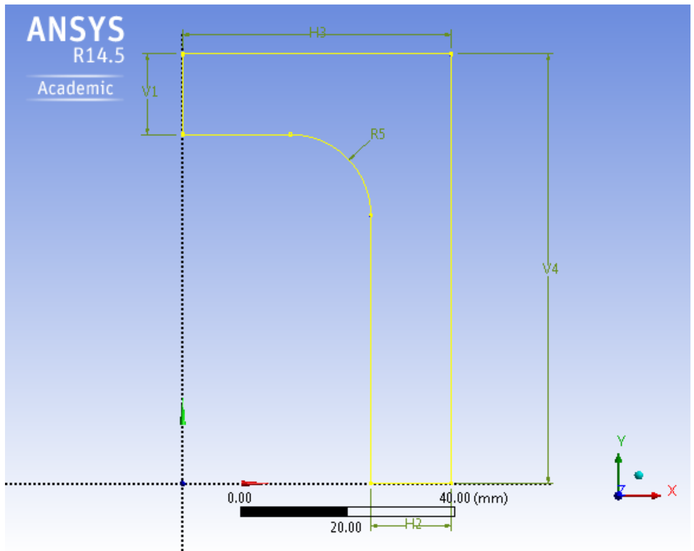


Figure 3.3: Pressure vessel construction daigram.

rotated round the y axis by 90^0 to produce the diagram in Figure 3.4.

A mesh was applied to the cylinder with a refinement added to the bend radius to produce a better analysis. Due in the main to the shape of the ANSYS mesher employed by default three-dimensional solid tetrahedral solid structural elements with mid-side nodes were employed throughout the solid models. On the other hand the two-dimensional models employed axisymmetric eight noded isoparametric (serendipity) elements. Each of the these elements employed quadratic interpolation functions with the displacement at each of the nodes being evaluated as the primary solution field. Initially global convergence was ensured by solving each of the models refining the mesh between each solution until the change in the maximum displacement was within 5%. To ensure local convergence of the structural error of the ith

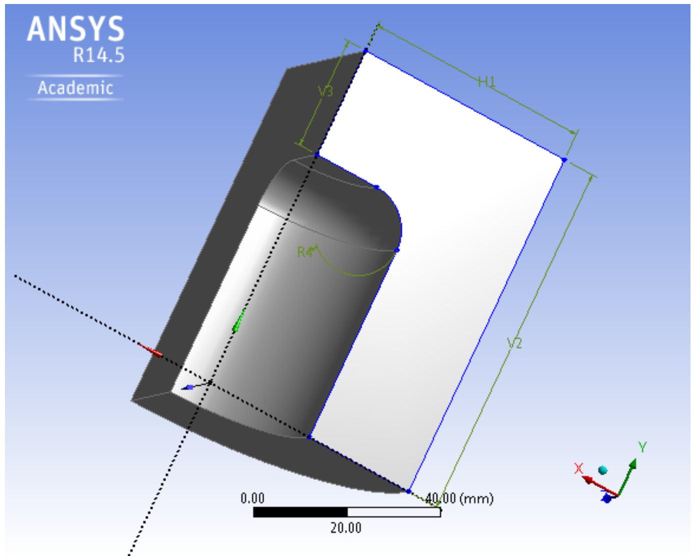


Figure 3.4: Thick walled cylinder showing solid body.

element (e_i) was evaluated using the ANSYS software, from:

$$e_i = \frac{1}{2}\int\int\int_v \left[\bar{\sigma}_n - \sigma_n^{(i)}\right]^T [\mathbf{D}]^{-1} \left[\bar{\sigma}_n - \sigma_n^{(i)}\right] dV$$

where $\bar{\sigma}_n$ is the average stress vector at node n, $\sigma_n^{(i)}$ is the stress vector of node n of element i, $[\mathbf{D}]$ elastic compliance tensor. In each of the models that follow local convergence was assumed once this value fell below a threshold of 1 mJ. This can be viewed in Figure 3.5. Boundary conditions were applied, *frictionless* supports were applied to the side and bottom faces of the cylinder; a pressure of 35 MPa was applied to the internal faces as seen in Figure 3.6.

The model was parameterized to facilitate the construction of one base model. The parameter set was produced by naming each dimension line. The purpose of this procedure was to negate the construction of two separate models, only one dimension was required to be changed to change the thin walled cylinder into a thick walled cylinder.

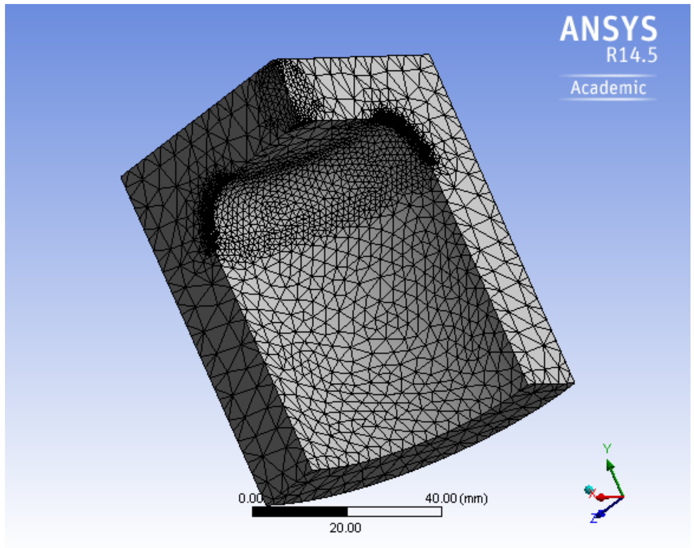


Figure 3.5: Model detailing mesh and added refinement.

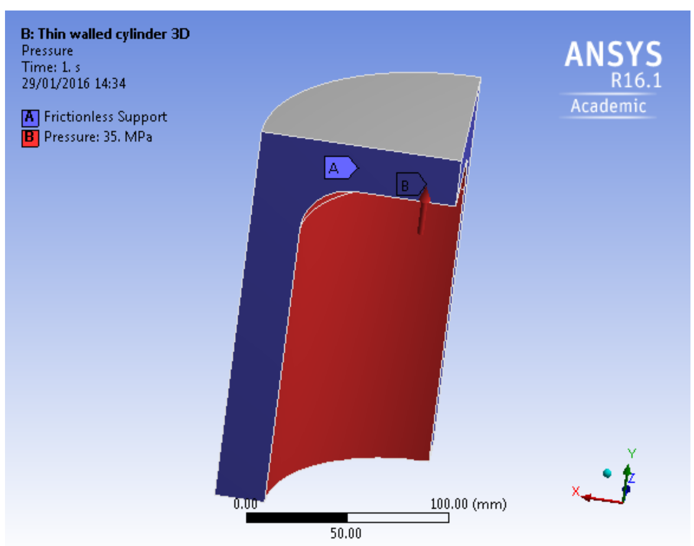


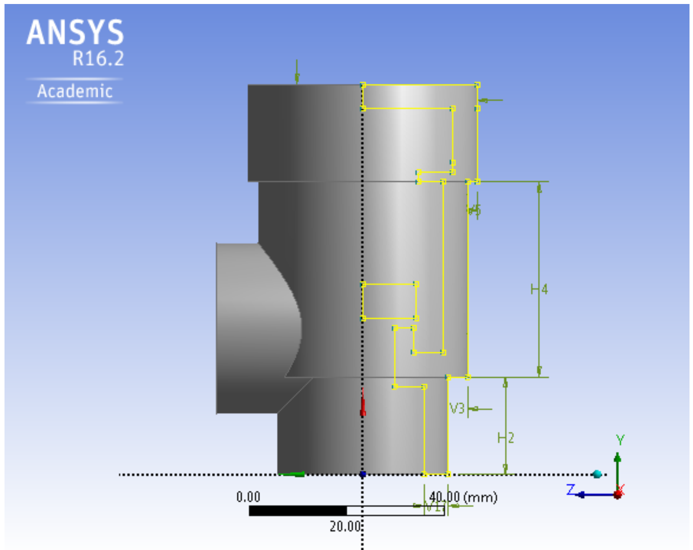
Figure 3.6: Thin walled cylinder showing frictionless supports & pressure.

3.3 Pressure relief valve construction

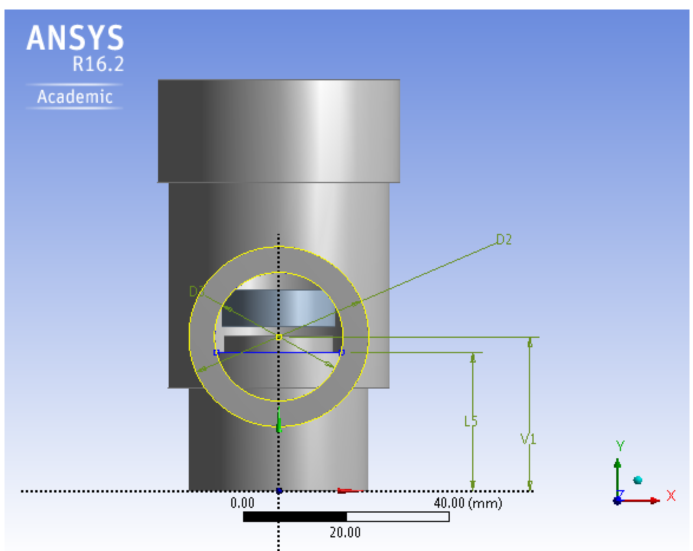
The initial starting point is the same as that detailed in section §3.2. Picking the polyline in the drawing function the shape displayed in Figure 3.7. The 'revolve' function was used to rotate the drawing round the $y-$axis by 90^0 to produce the three-dimensional model in Figure 3.7.

To create the bottom half of the valve body a new sketch was added to the xy plane. Again, picking the polyline function the illustration shown in Figure 3.7 was produced. The dimensions in Table 3.2 were added to the diagram; it is pointed out that the onset of yielding as evaluated from application of equation (3.6) was 23.75 MPa and 61.22 MPa for the thin and thick wall pressure vessels respectively, *ipso facto* below the plastic collapse load Table (5.1). The x axis was made coincident with the top line of the valve. Once more the revolve function was used to spin the outline round the y axis by 180^0, to develop the illustration as per Figure 3.7a.

The 3rd phase of design was to add the outlet to the bottom of the PRV body. A new sketch was added on the $zy-$plane of the bottom part of the PRV body; the new construction plane had a 15mm offset added. Two circles were then generated: circle 1 with a diameter of 35mm and circle 2 with a diameter of 25mm, as seen in Figure 3.7b. The extrude function was used to extend the circles up to the internal face of the PRV (as observed in Figure 3.7a), and the topology was merged at this point. Picking up the line tool in the drawing menu a horizontal line was constructed on the inner diameter of circle 2, in the modify function the trim tool was used to trim the selected section of the circle to provide the outlet as illustrated in Figure 3.8.



(a) Pressure Relief Valve body construction.



(b) Pressure Relief Valve outlet construction.

Figure 3.7: Pressure valve construction

Co-ordinate	Dimension (mm)
H2	20
H25	7
H29	5
H31	2
H32	12
H4	40
V11	5
V17	5
V20	6
V21	4
V24	11
V3	4
V5	2
V7	5
V9	7
V1	30
D2	35
D3	25

Table 3.2: Pressure relief valve construction dimensions.

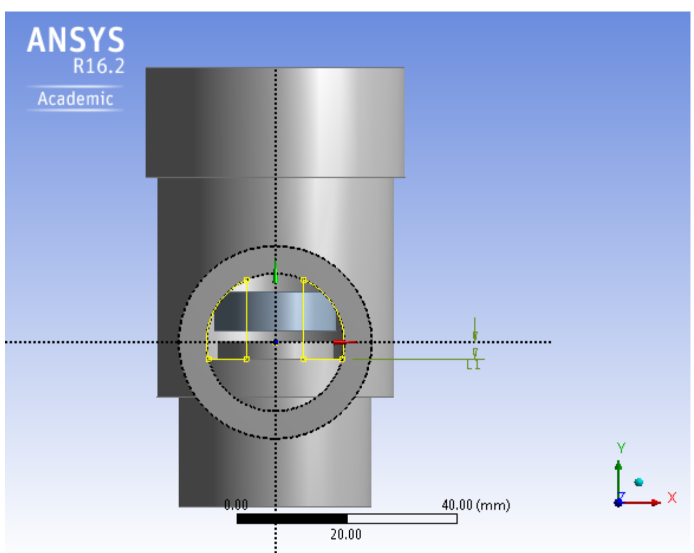


Figure 3.8: Pressure Relief Valve outlet construction.

3.4 Simulation methods

The purpose of the PRVs is to prevent overpressure by providing pressure relief. Different types of PRV can be used to achieve the pressure relief requirements of a system. Divided into two categories: 1. Recloseable PRVs, spring loaded or pilot operated. 2. Non-recloseable, bursting or rupture disc. These count for 99% of all PDs used in a pressure system.

Spring loaded PRVs (used to release pressure rapidly), work on the high-lift principle. The valve is designed to *pop*-open when a specific pressure is reached. Without this feature the PRV would lift gradually, and probably with continual oscillation (an undesirable condition termed simmering). The stages of operation being:

Stage-1 closed – Inlet pressure is below the PRV set pressure.

Stage-2 simmering – this is the point at which the inlet system pressure exactly matches the closing force of exerted by the PRV spring. As the pressure continues to raise the valve disc starts to lift, releasing that a small amount of fluid into the huddling chamber (i.e. to be deflected by the disc's skirt).

Stage-3 opening – the high lift principle comes into operation at this stage. The accumulating fluid in the huddling chamber is now acting on a larger surface area, producing a greater force than the PRV spring; the force is further enhanced by the reaction force of the kinetic energy of the fluid turning through the huddling chamber. The result is that the valve pops open with a positive movement, allowing a large escape area for the fluid and providing a quick but controlled release of over pressure.

Stage-4 *reseating* – As the over-pressure is released and the pressure begins to drop, the valve closes under the effect of the spring pressure. A blow down ring controls the pressure at which it re-seats. However, it is necessary to avoid *feathering*, which is a condition where re-seating does not fully take place and although the seat and seal maybe in contact, the fluid in the PRV still passes and the valve disc can move up/down periodically. This can also occur as a result of a damaged seal. even if the seat and seal are clamped together.

What follows throughout this section therefore is the development of a ANSYS static analysis simulation of essentially Stage-3 of the operation of this particular Protective Device (PD). The diagrams and graph illustrate the stages discussed above shown in Figure 3.9.

3.4.1 Meshing

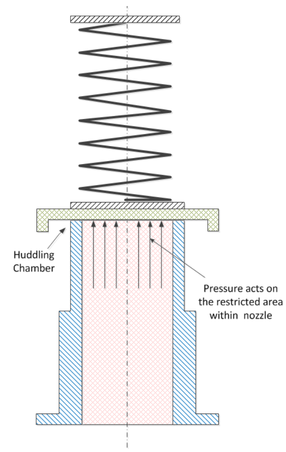
The mesh was generated using the same procedure as laid down in section §3.2, with Figure 3.10 showing the generated mesh.

3.4.2 Boundary conditions

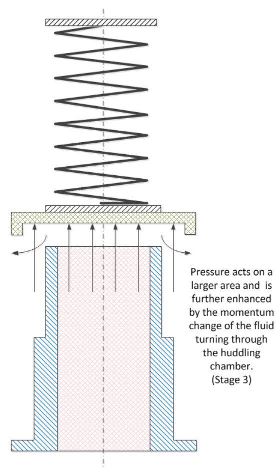
Figure 3.11 details the placement of the supports and the load applied to the PRV body. Two different types of support were used on the model; frictionless on the cross-sectional faces and a fixed support on the top face. A pressure of 20 MPa was applied to the internal faces of the PRV.

3.4.3 Verification

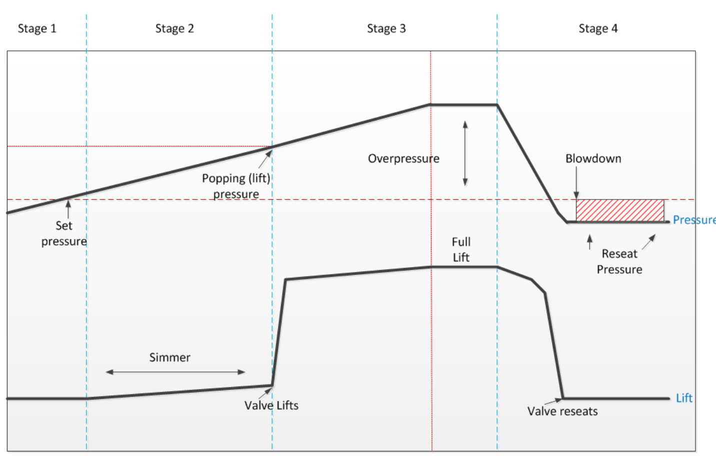
The results for the benchmark thick walled cylinder, with a smaller radius applied, are shown in Figure 3.13. Here as expected, the maximum stress is shown at the inside radius of the vessel. The maximum principal stress shown is 119 MPa reducing to −23 MPa towards the centre of the vessel. The minimum principal stress Figure 3.14a is 18.9 MPa at the centre of blend radius on the inside of the vessel with a small radius, dropping to a minimum level of −43.9 MPa in two small areas of vessel (one



(a) PRV in normal position.



(b) PRV at stage 3.



(c) Graph showing the PRV cycle.

Figure 3.9: Diagrams and graph showing the various stages of PRV operation.

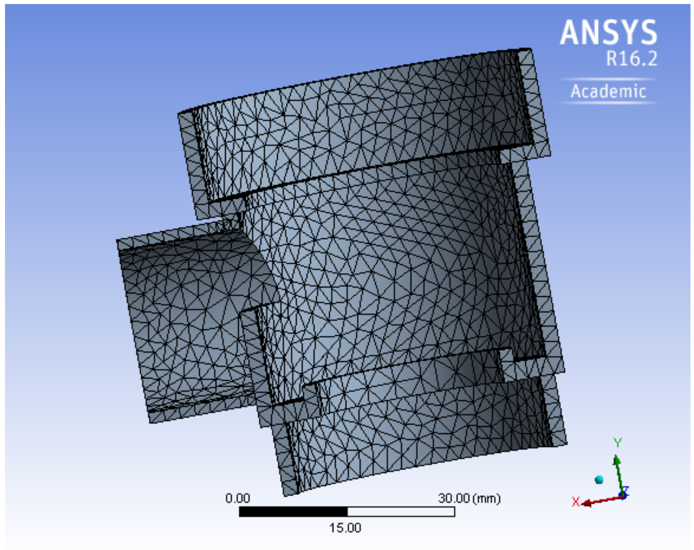


Figure 3.10: Pressure Relief Valve mesh.

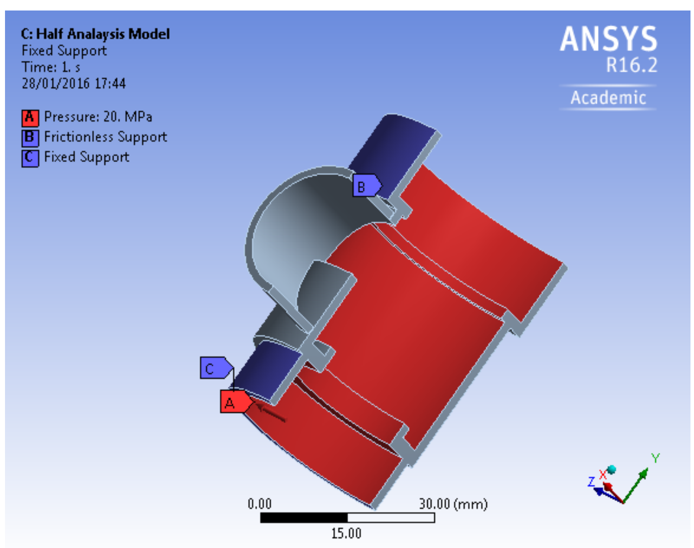
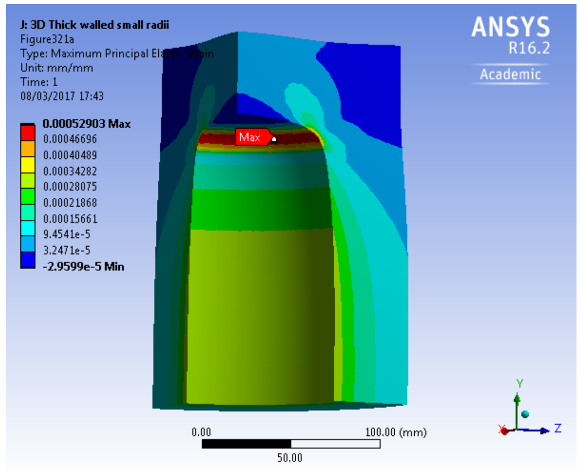
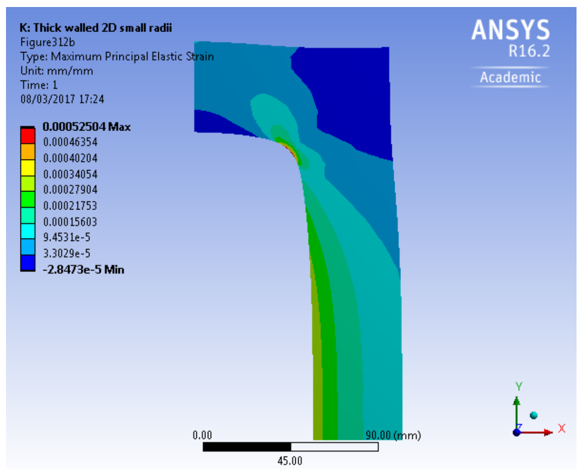


Figure 3.11: Pressure Relief Valve supports and load.



(a) Maximum principal elastic strain (small radius).

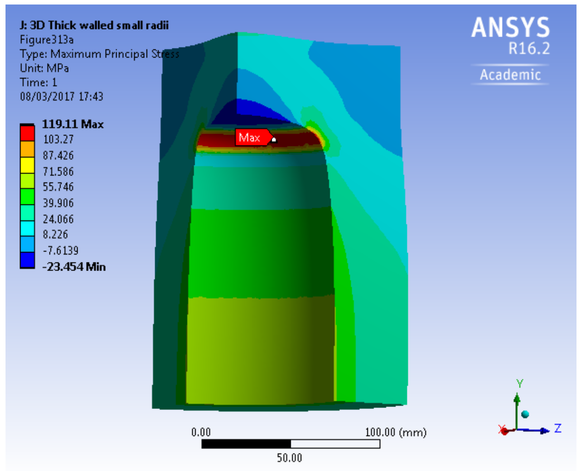


(b) Maximum Principal elastic strain field (2D small radius).

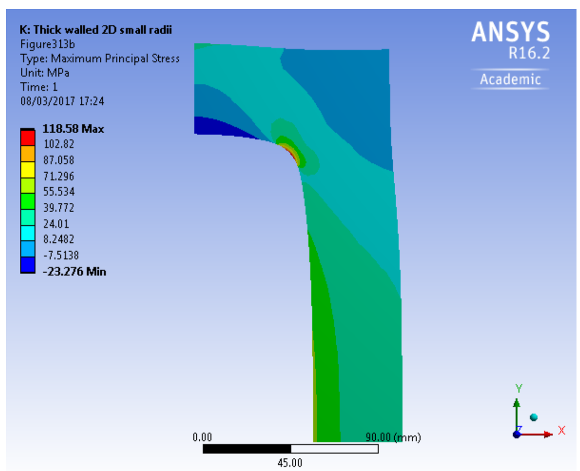
Figure 3.12: Maximum Principal elastic strain field (small radius) 3D vs 2D.

on either side of the of the radius). Figure 3.12a shows that the maximum principal elastic strain is 0.52 $\mathtt{mm/m}$ (i.e. 520 $[\mu - \mathtt{strains}]$) at the inside of the vessel on the bend. The Figure 3.15a details the safety factor, with the weak point being on the inside of the vessel along the inside edge of the radius; and the outside of the vessel shows the strongest area.

The results for the benchmark thick walled cylinder in two-dimensions (with small radius) are shown in Figure 3.12b through 3.15b. Figure 3.12b shows the maximum principal elastic strain with a maximum reading of 520 $[\mu m/m]$ located at the centre

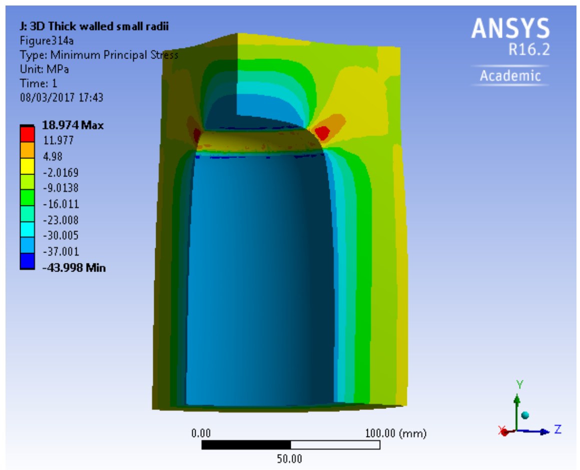


(a) Maximum Principal stress field (small radius).

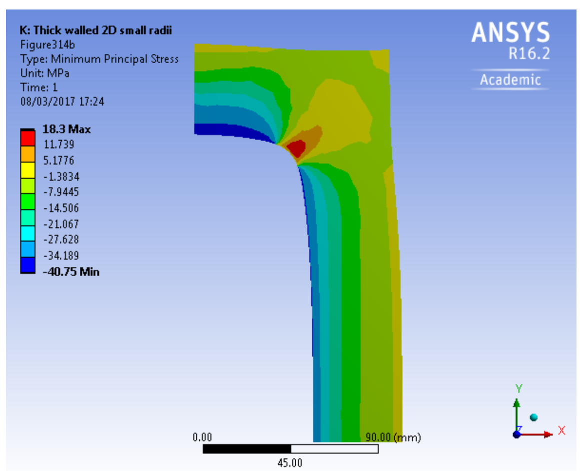


(b) Maximum Principal stress field (2D small radius).

Figure 3.13: Maximum Principal stress field (small radius)3D vs 2D.

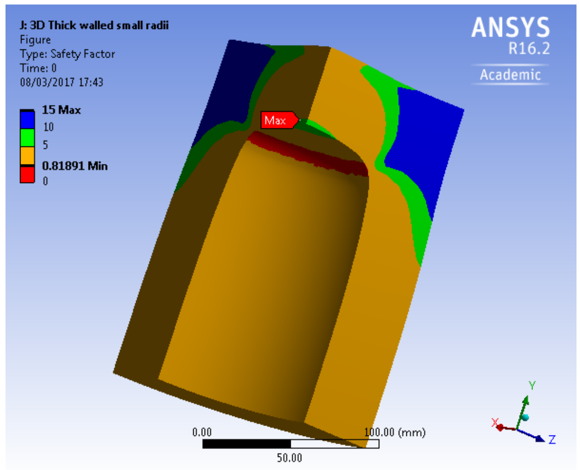


(a) Minimum principal stress field (small radius).

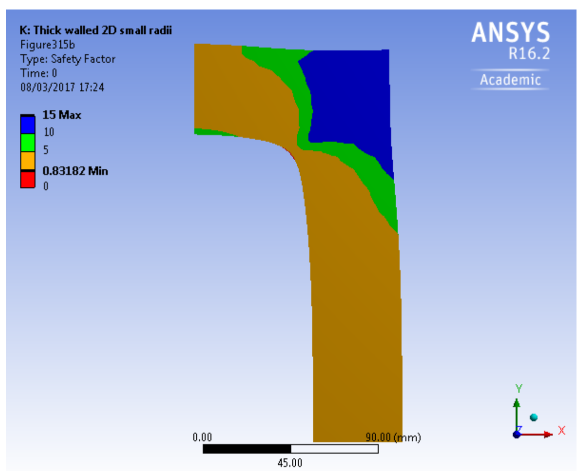


(b) Minimum Principal stress field (2D small radius).

Figure 3.14: Minimum Principal stress field (small radius)3D vs 2D.



(a) Safety factor (small radius).



(b) Safety factor (2D small radius).

Figure 3.15: Safety factor (small radius)3D vs 2D.

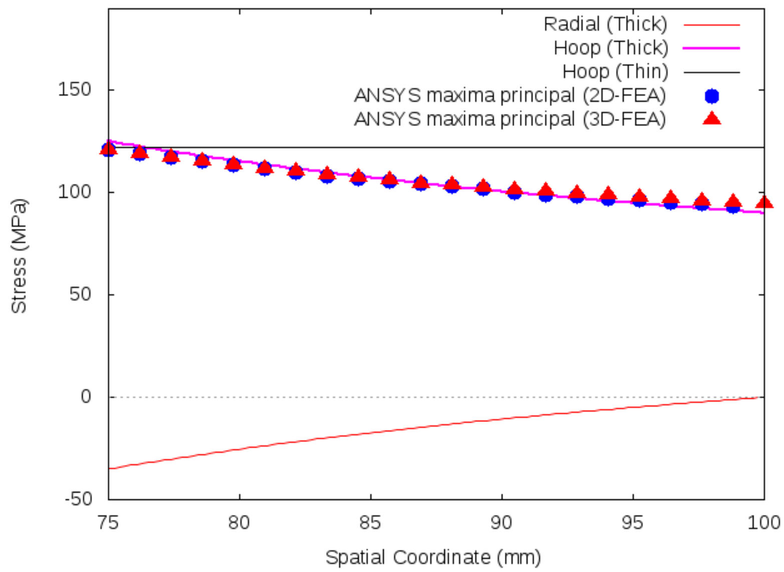
of the blend radius in the same location as the three-dimensional model Figure 3.12a. The minimum reading of -2.84 [mm/mm] is observed at the same location as that in the three-dimensional model, but with a lower value. The minimum principal stress visible in Figure 3.14b shows a maximum reading of 18.3 MPa in the bend area the same as in the three-dimensional model but with a lower value. The minimum value being -40.75 MPa at top and bottom inside edge, either side of the radius; the three-dimensional model shows this in the same areas. The safety factor in Figure 3.15b shows the same maximum of 15 in the same area as the three-dimensional model. The two-dimensional model shows the failure region around the inside edge of the bend with a minimum reading of 0.831.

Thus Figure 3.16 demonstrates appropriate verification of the methods detailed throughout this section; for the case study detailed in the proceeding chapter (§4.1) with predictions being akin to those of the reference [100]. This being evidenced by the radial stress (σ_r), as expected being calculated at the inner surface, as -35 MPa and reducing to zero at the outside, which can be observed in the graph detailed in Figure 3.16a. The overall minimum stress (i.e. maximum compressive stress) appears at the interface when $r = D_i/2$, i.e. equal and opposite to the applied pressure. The hoop stress (σ_θ) is 122 MPa at the inner surface with a modest reductions in the magnitude of the stress field being observed until a value of 100 MPa is reached at the external boundary of the model; this means the maximum stress appears at the interface when $r = D_i/2$ as expected.

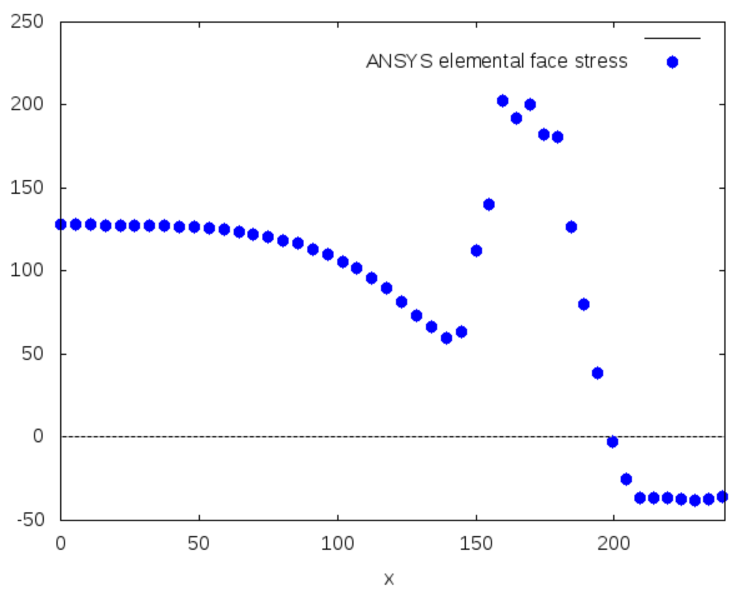
The thick and thin wall cylinder are showing almost equal stresses at the inner radius of ~ 122 MPa supporting the use of thick and thin wall cylinder theory for this geometry. Note that as expected, and true to the theory, being a point estimate the thin wall does not predict the stress reduction through the thickness of the section, as there is no material present for the stress to dissipate. An interesting point to note is that a small area of high stress can be observed on the outside centre surface of the cylinder. This is analogous to beam theory i.e. a beam supported at both ends with a force applied to it, the beam would fail at the centre.

The maximum tensile stress, (ergo the on set of fatigue), occurs at the pressure application and is some 4 times greater than the applied pressure. Examples of this can be observed in the model of Figure 4.5b. Both the three-dimensional and two-dimensional FE models being in close agreement, for both the thick and thin walled cylinders. The maximum stress at the outer surface was shown as $\sim 98\,\mathtt{MPa}$ which equates to approximately $2.5-2.6$ times greater than the applied pressure. There is obvious excellent agreement between the thick walled cylinder theory and the FEA, including both the two and three-dimensional models.

The graph in Figure 3.16b shows the results provided from a trace of the internal surface of the cylinder *with* large radius. Its shows a starting of $125\,\mathtt{MPa}$ with a slow reduction in stress to $50\,\mathtt{MPa}$ and then a rapid rise up to $200\,\mathtt{MPa}$. This corresponds to the inner radius of the vessel. A rapid reduction in stress values is observed until a final value of $-35\,\mathtt{MPa}$ is reached at the internal boundary of the model.



(a) Analytically and FE model predictions



(b) ANSYS inner face predictions

Figure 3.16: Maximum principal stress model verifications

Chapter 4

Results

This chapter is divided into two sections. The first set of results is obtained from the benchmarking carried out on a simple pressure vessel. The second shows the generated results from the Pressure Relief Valve (PRV) model, as both a thick body and a thin body.

4.1 Pressure vessel

Tables 4.1 (a) and (b) provide critical details and observations made with respect to the ANSYS modelling results described throughout this section.

The Von Mises stress is shown in Figure 4.2a showing maximum value of 84.3 MPa as per the ASME design code indicates no local yielding as it is below the yield stress of the structural steel, albeit 250 MPa. This is further evidence in Figures (4.3). On the other hand an indication of mode one crack propagation the maximum principal stress is shown for comparative purposes and further verification i.e. compared with the analytic models of section (3.1) in Figure 4.2b. Where a maximum value of 59.5 MPa was calculated by the software and is observed in two areas: the first (as in the vessel with a small radius) is on the inside, where the vessel bends and secondly

ANSYS Model (Figure number)	Title	Observations
Figure 4.2 (a-d)	Stress field (radius) 3D vs 2D	The Von-mises stress is shown., as per the ASME design code this indicates no local yielding as it below the yield stress of structural steel. Furthermore an indication of mode one, crack propagation the maximum principal stress is shown for comparative purposes and further verification (i.e. compared to with the analytic models of section (3.1) is also provided.
Figure 4.3 (a-d)	Von-Mises stress safety factor	The two-dimensional and three-dimensional models Von-Mises safety factors for the vessel with radius applied and a vessel with a small radius applied are shown. It can be observed in all four models that failure would not occur as the onset yielding would not happen as all the safety factors are showing values above 1.
Figure 4.4 (a-d)	Fatigue safety factor	The failure index **due to fatigue** as indicated by the software,suggests that the vessel with the radius would never fail as it does not show any areas of red that would indicate points of potential failure. The lowest figure observed at the inside area of the vessel moving to the outside. The maximum is seen the top corner of the vessel. However, the minimum values of the fatigue safety factors are of more importance as these indicate failure of the component due to fatigue at nominal ANSYS Workbench design life of 10^9[`cycles`]. In each of the cases shown failure is deemed imminent as evidenced by the failure index being near to unity for the model with the radius less than unity.

Table 4.1: (a) ANSYS generated models and observations.

Figure 4.5 (a-b)	Thin wall stress fields (radius) Von-Mises and maximum principal stress	In line with the ASME [21] standard the plastic collapse of the thin wall pressure vessel can be predicted from the Von-Mises stress calculations. Furthermore, details of the maximum principal stress results are also provided.
Figure 4.6 (a-b)	Safety factor Von-Mises stress & fatigue safety factors (thin)	The fatigue safety factor revealing a large area of failure over most of the cylinder due to fatigue. The failure region covers most of the internal surface of the cylinder.
Figure 4.7 (a-b)	Max principal elastic strain (Thick)	The benchmark thick walled cylinder in two-dimensions with *large* radius. The maximum principal elastic strain in the two-dimensional model is shown in the same regions are shown for the three dimensional model
Figure 4.8 (a-b)	Min principal stress (Thick)	The minimum principal stress shows a maximum value around the external surface of the vessel, with a minimum stress covering the internal surface of the vessel. The two-dimensional models stress fields mirrors that of the three-dimensional model; however, the values are slightly different.
Figure 4.9	Min principal stress (Thin)	The minimum principal stress field is displayed, with the most visible being either side of the the radius. Another can be seen at the top outside centre surface of the cylinder. The minimum value is indicated at the inside centre tip of the cylinder.
Figure 4.10	Max principal elastic strain (Thin)	The maximum principal elastic strain results can be observed covering the area of the bend radius from one side to the other, in three separate regions: the first area is around the inside centre tip, with the other two being either side of the radius on the outside edge of the cylinder.

Table 4.2: (b) Further ANSYS generated models and observations

on the inside from a third of the way down to the end of the vessel, but still with a much lower maximum stress value. The minimum value of $-5.7\,\texttt{MPa}$ is seen at the inside centre of the vessel. Examination of Figure 4.4a, showing the failure index due to fatigue as indicated by the software, would suggest that the vessel with the radius would never fail as it does not not show any areas of red that would indicate points of obvious potential failure. The lowest figure observed is 1.0413 which covers the inside area of the vessel. In the bottom third a small point can be seen at the outside tip centre of the vessel. The maximum level of 15 is seen on both the left and right hand top corners of the vessel. However, the minimum values of the safety factors are of more importance as these indicate failure of the component due to fatigue at a nominal ANSYS Workbench design life of $10^9\,[\texttt{cycles}]$. In each of the cases shown in Figure 4.4 failure is deemed imminent as evidenced by the safety factor being near to unity for the model with the large radius and less than unity (i.e. 0.82) for the small one. It is pointed out that given that nominally the yield stress of steel is $250\,[\texttt{MPa}] - 320\,[\texttt{MPa}]$ that the software predicts reduction in strength of orders 8.5 and 16.2 depending on the presence of stress relief. It is also worth pointing out there are discrepancies in these predictions when comparing them with the dimensional analogues, these being attributed to the relaxation of the frictionless constraint in the latter model.

The results for the benchmark thick walled cylinder with the large radius are shown in Figure 4.7b through 4.4c. The observed principal elastic strain in the two-dimensional model is shown in the same regions as that of the three-dimensional model but with a slightly higher values. Figure 4.2d shows a maximum principal stress of $59.2\,\texttt{MPa}$ at the centre of the curve, with the two-dimensional model is showing an almost identical value. The minimum value of the maximum principal stress is evident in two separate regions: the outside top right hand corner and the top inside left tip of the vessel, at $-5.5\,\texttt{MPa}$ (only $0.2\,\texttt{MPa}$ difference to that of the three-dimensional model), which can be attributed to the aforementioned different element formulations. An observation of note is that the maximum and minimum areas of stress between the three-dimensional & two-dimensional model are reversed. Two areas of maximum principal stress in the three-dimensional model and just one area in the two-dimensional model.

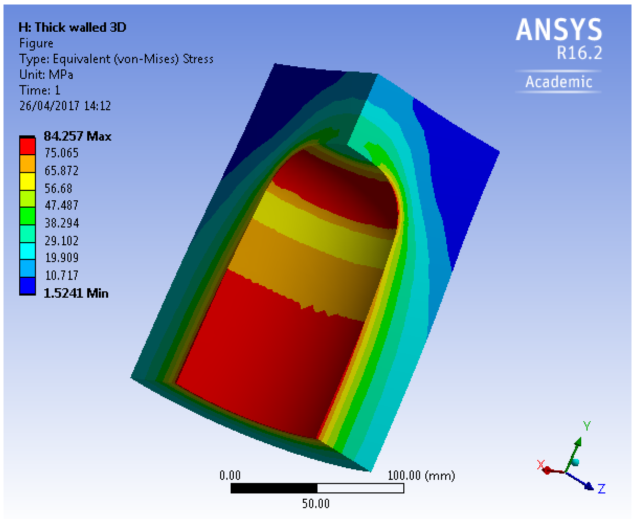
In line with the ASME standard [21] the plastic collapses of the thin wall pressure vessel can be predicted from the Von mises stress calculations shown in Figure 4.5a. Furthermore, Figure 4.5b details the maximum principal stress results, the highest value being 201 MPa (seen again along the radius), seen also on the outside centre tip area. This is expected, as this would be a weak spot in the cylinder's construction. The minimum value is -36.7 MPa at the internal centre area. Figure 4.6 displays the fatigue safety factor revealing a large area of failure over most of the cylinder. The region covers most of the internal surface of the cylinder with two areas of red continuing out to the outside surface at the top of the cylinder. The minimum value of 15 is observed at three small locations one each side of the radius, with the other in the centre at the top of the vessel.

Further verification of the benchmark thick walled cylinder are provided in Figures 4.7a 4.2b and 4.8a, and the fatigue safety factor is shown is Figure 4.4a. The minimum principal stress value seen in Figure 4.8a is -36MPa, equal and opposite to applied pressure boundary condition, that is around the whole of the external area of the vessel. This is still much lower than in the vessel with the small internal radius, however, the stress areas are different. The minimum principal stress observed in Figure 4.8b shows the two-dimensional model mirrors that of the three-dimensional model, Figure 4.8a, with almost identical values and a minimum principal stress of -35.8 MPa. Figure 4.4c shows the fatigue safety factor of the two-dimensional model, with an average minimum value of 0.82 is observed at the same position in both the two-dimensional and three-dimensional model, indicating fatigue failure of the vessel. It should be noted that under the static loading conditions the Von-mises safety factor was greater than unity (no failure). The minimum principal stress field is displayed in Figure 4.9 with the maximum reading of 12.6 MPa witnessed in a number of areas. The most noticeable two are visible either side of the radius. Another can be seen at the top centre outside surface of the cylinder. Further areas of attention are down the outside edge of the cylinder. A minimum value of −111.1 MPa is indicated at the inside centre tip of the cylinder. The maximum principal elastic strain results can be observed in Figure 4.10. The maximum reading is 0.00097mm/mm and is shown covering the area of the blend radius from one side to the other.

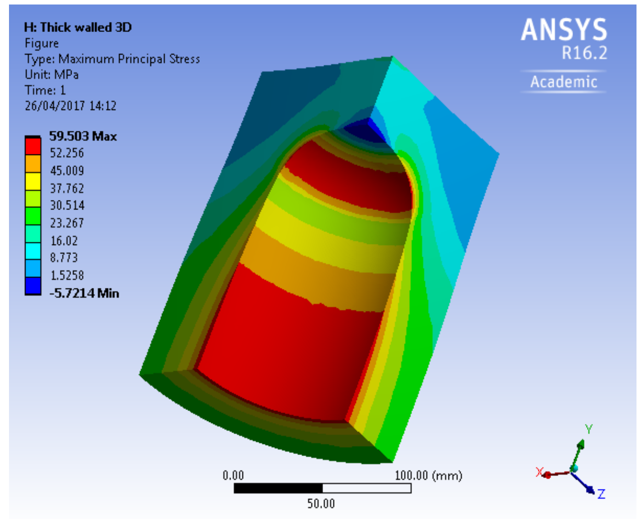
4.2 PRV results

The models shown in the following sections display the results from the Pressure Relief Valve (PRV) modeling, given the boundary conditions detailed in §§3.4.2. The results shown in Figure 4.11 show the total deformation of the PRV in both thick (Figure 4.11a) and thin (Figure 4.11b) variants. The thick PRV model in Figure 4.11a shows the model with a maximum value of 0.160 mm covering all of the PRV body. The exception to this is the PRV top hat displayed in blue, achieving a minimum value of 0 mm. In Figure 4.11b a maximum value of 0.263 mm can be observed in two areas of the PRV body: one area at the centre of the body and the second area shown at the centre of the restriction on the outlet side of the PRV. A minimum value of 0 mm is displayed in three locations, one at the inlet of the PRV and a further two areas that can be observed at the top centre and top right corner of the PRV.

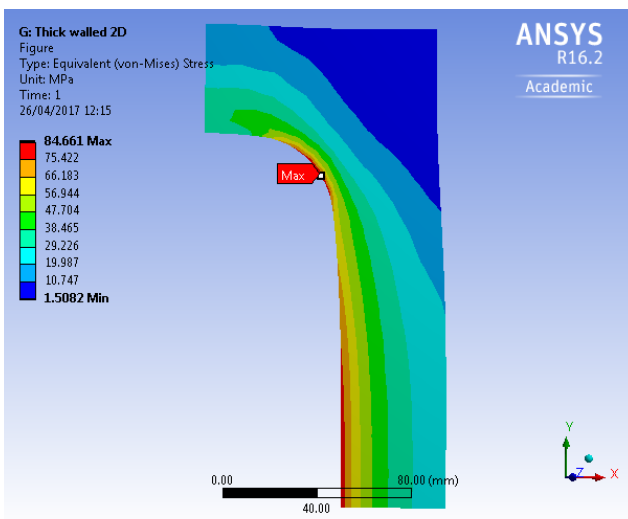
The models displayed in Figure 4.12 shows the PRV displayed with both a thick body, (Figure 4.12a) and thin body (Figure 4.12b) detailing the equivalent elastic strain (Von-Mises), as the material is ductile. It is interesting to note the



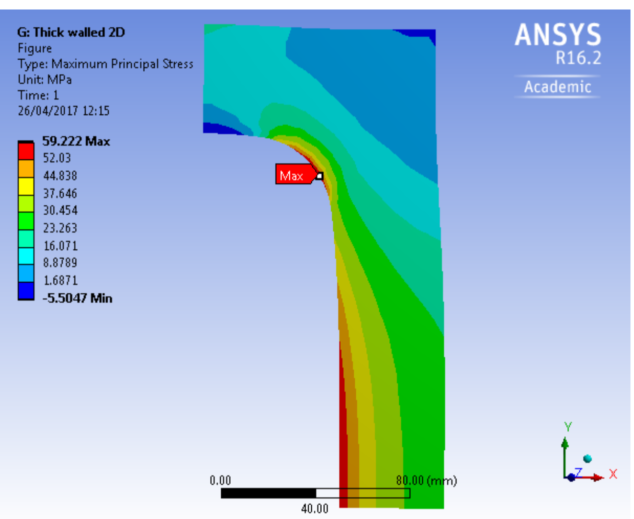
(a) Von-mises stress field (radius)



(b) Maximum principal stress field (radius).

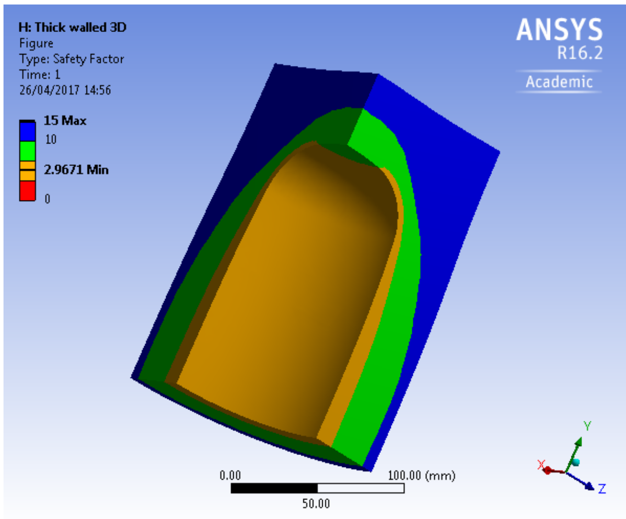


(c) Von-mises stress field (2D radius)

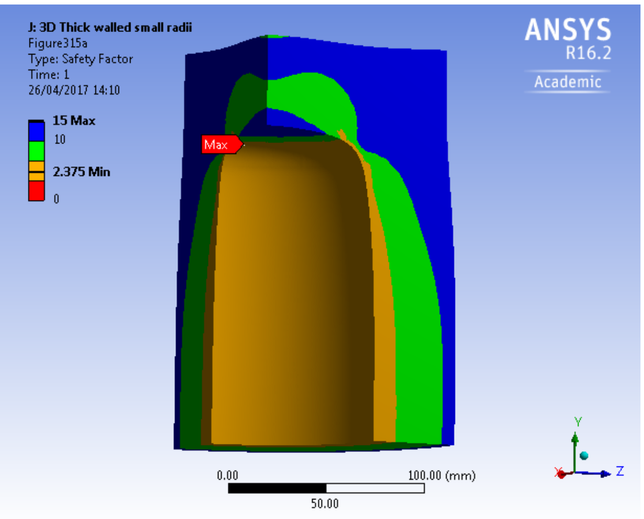


(d) Maximum Principal stress field (2D radius).

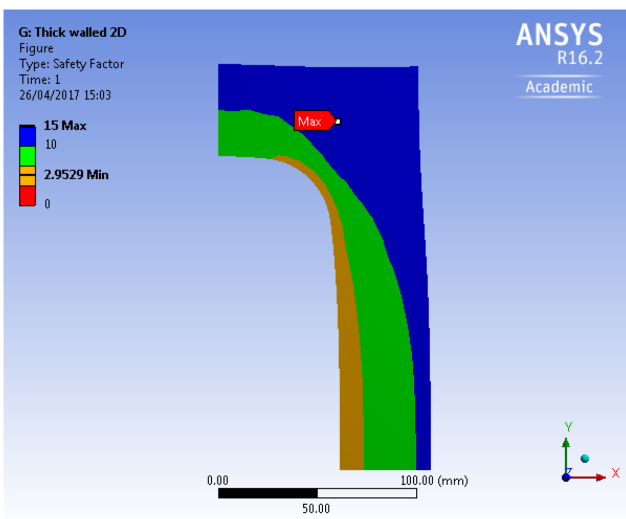
Figure 4.2: Stress fields (radius) 3D vs 2D.



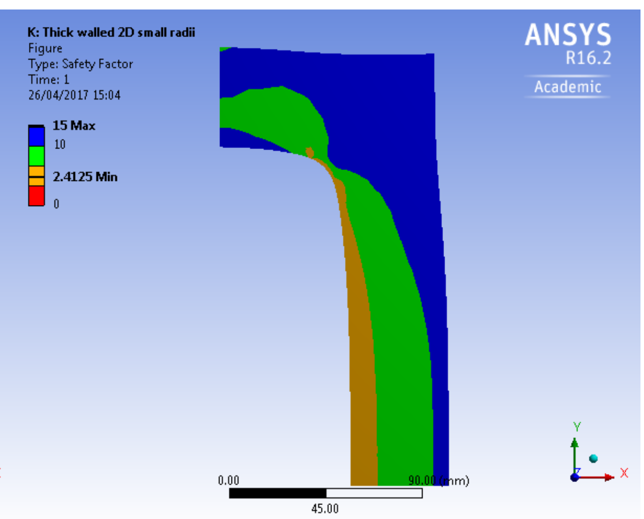
(a) Stress safety factor 3D



(b) Stress safety factor 3D

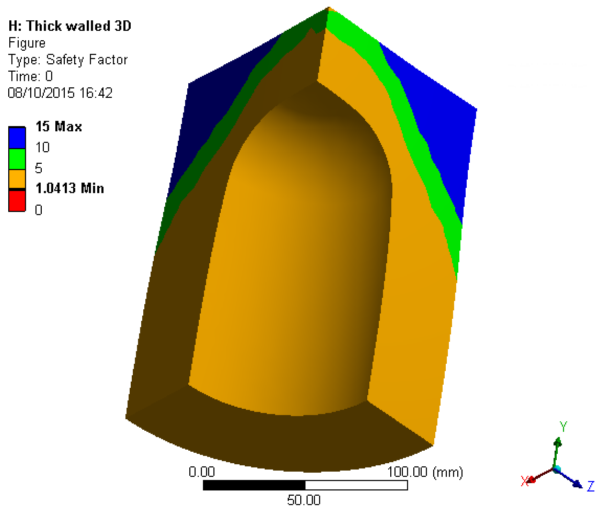


(c) Stress safety factor 2D

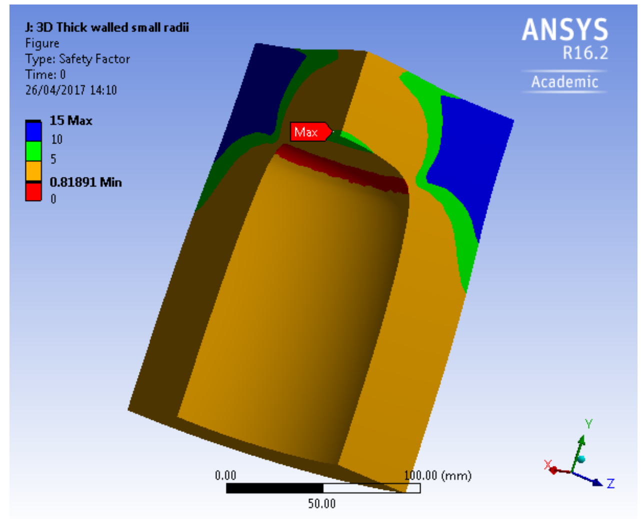


(d) Stress safety factor 2D

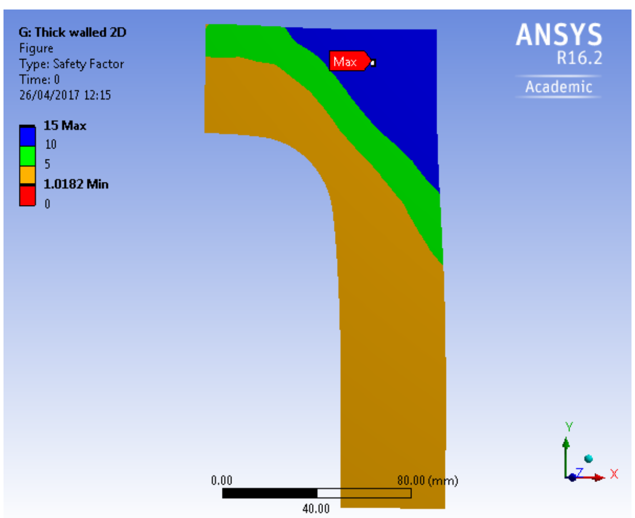
Figure 4.3: Von mises criteria factors of safety



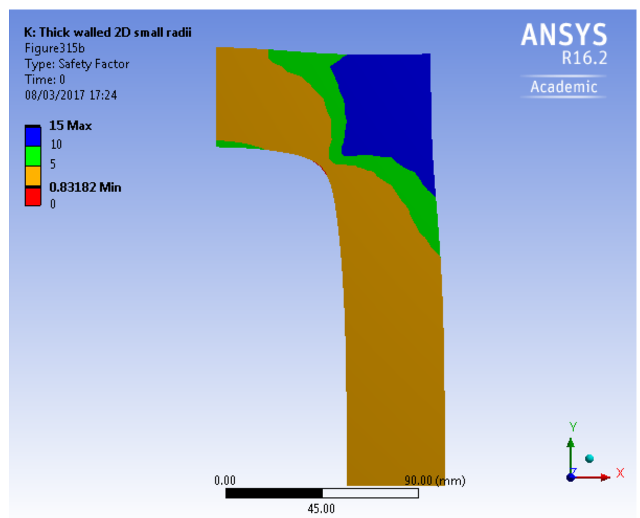
(a) Fatigue safety factor (radius).



(b) Fatigue safety Factor 3D small radius.

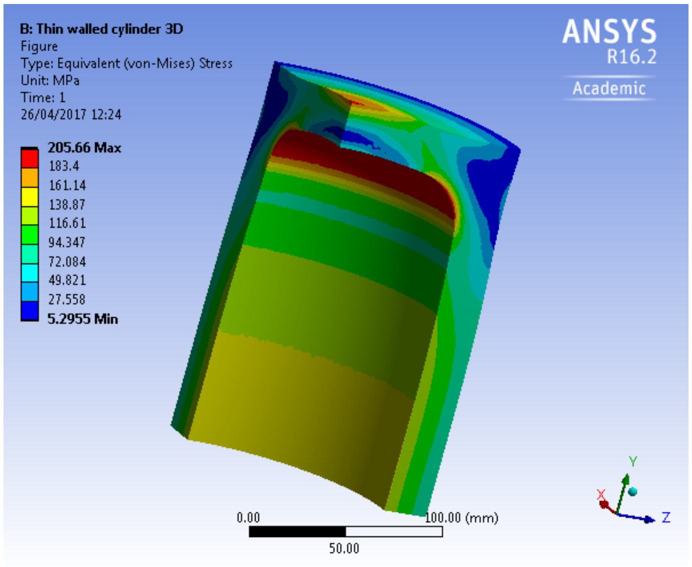


(c) Fatigue safety factor (2D radius).

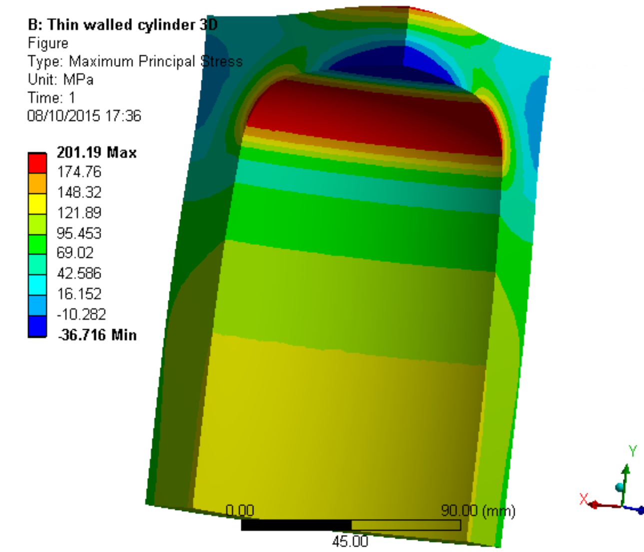


(d) Fatigue safety Factor 2D Small radius.

Figure 4.4: Fatigue safety factor (radius)3D vs 2D.

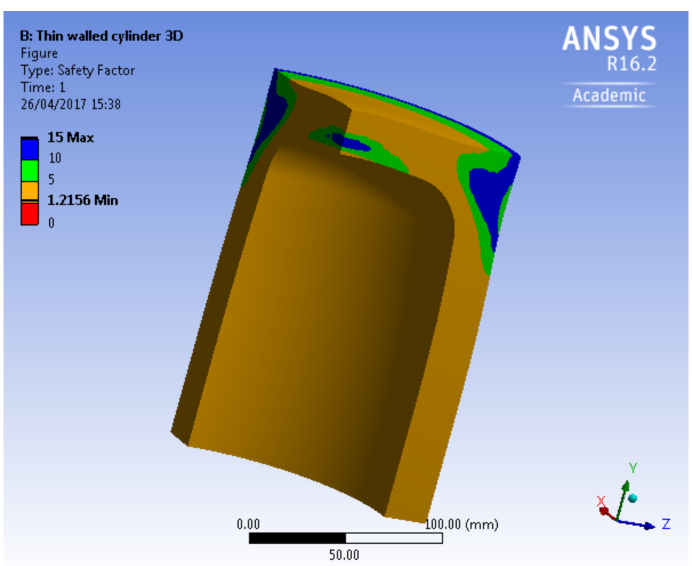


(a) Von mises stress field (radius)

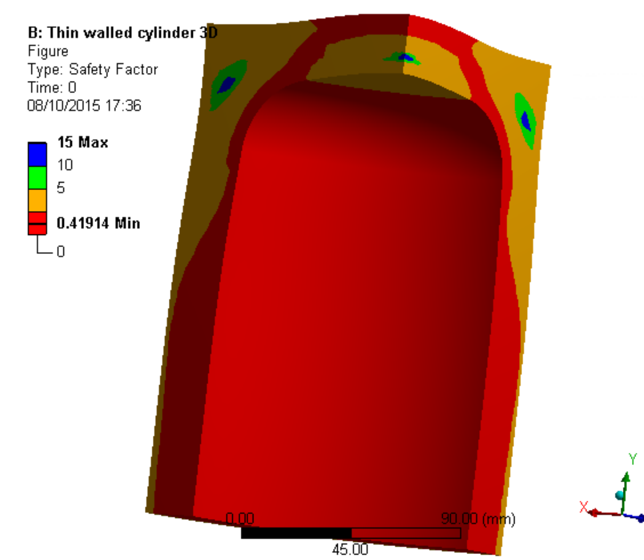


(b) Maximum Principal stress field (radius).

Figure 4.5: Thin wall stress fields (radius).

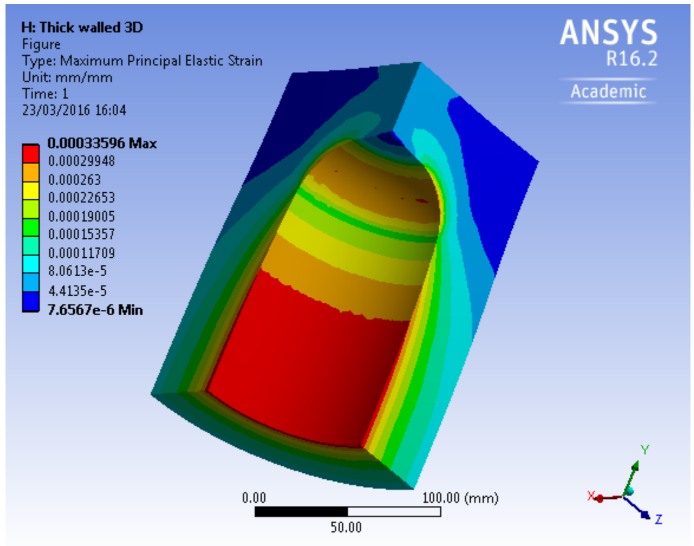


(a) Von-mises stress safety factor

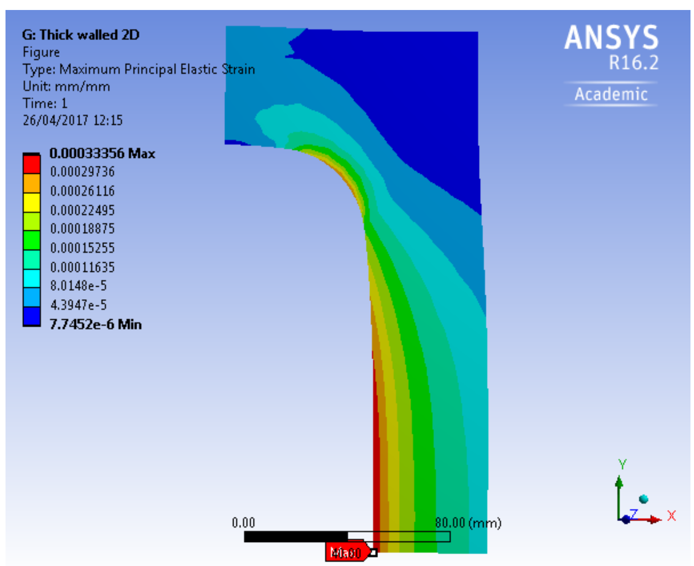


(b) Fatigue safety factor

Figure 4.6: Safety factor .

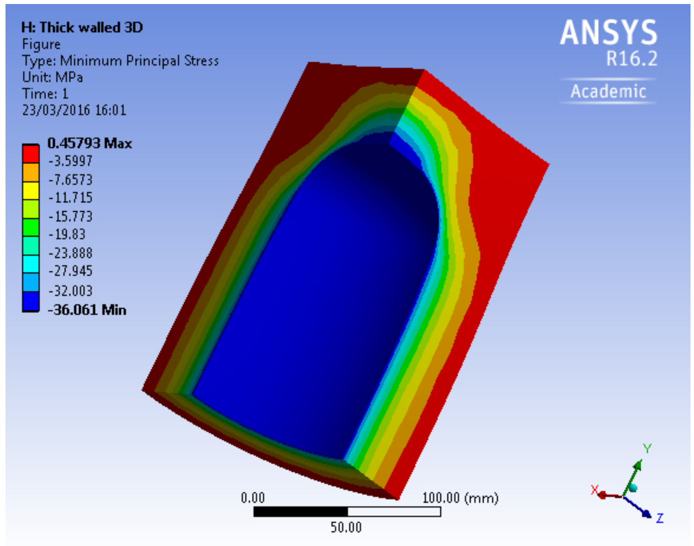


(a) Maximum Principal elastic strain field (Radius).

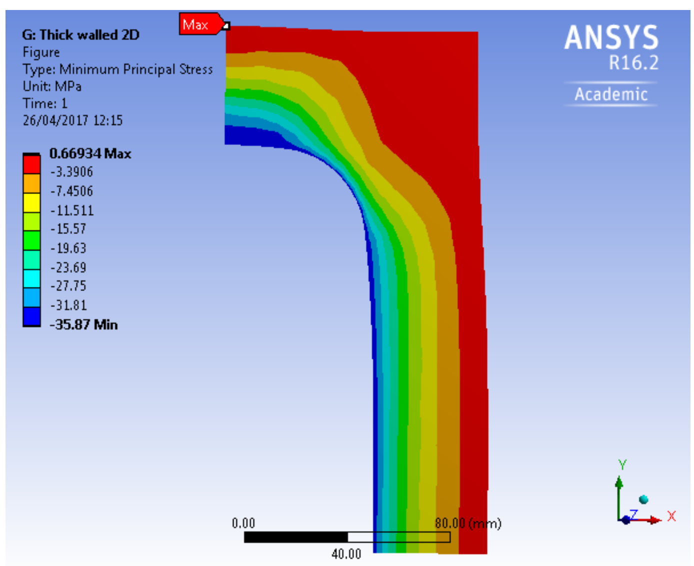


(b) Maximum Principal elastic strain .

Figure 4.7: Maximum Principal elastic strain field (Radius) 3D vs 2D.



(a) Minimum principal stress field (radius).



(b) Minimum Principal stress field (2D radius).

Figure 4.8: Minimum principal stress field (radius)3D vs 2D.

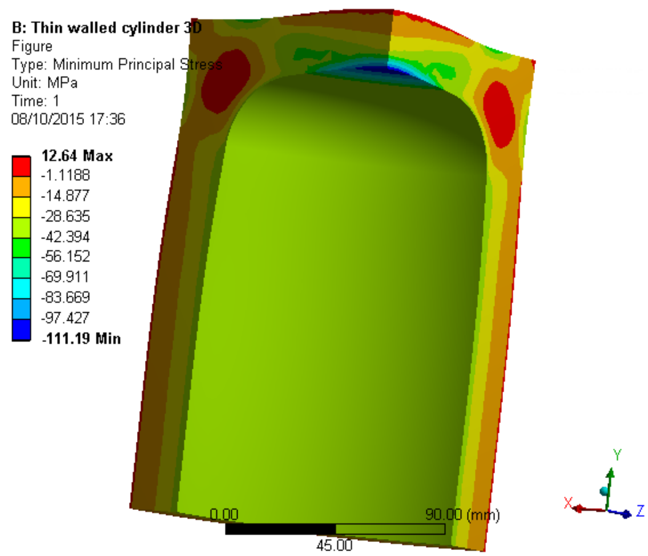


Figure 4.9: Minimum Principal stress field (3D radius).

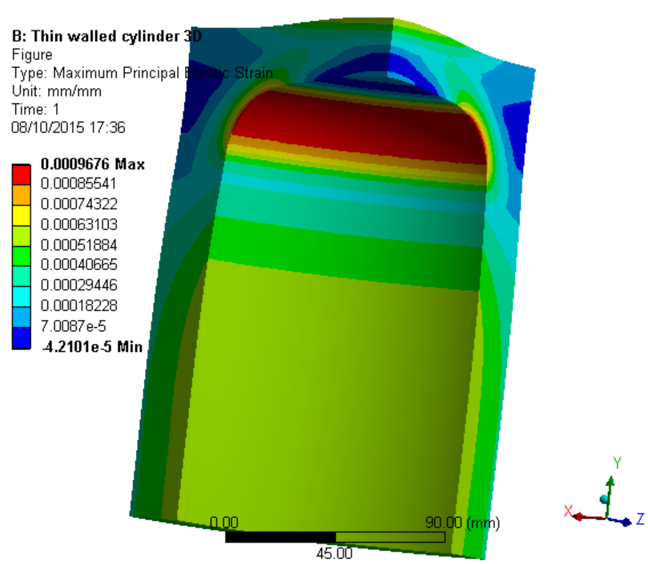


Figure 4.10: Maximum Principal elastic strain field (radius).

areas in which the maximum elastic strain is observed. The maximum figure of 0.001143 mm/mm occurs at the top edge of the outlet on the thick bodied valve. However, the maximum value of 0.0060253 mm/mm is seen at two areas: the middle of the valve body and the underside area of the inlet on the thin bodied value. This can be attributed to the models reacting in accordace to thick or thin wall cylinder theory. In the case of the former the outlet is providing more material in resistance to the stress induced by the internal pressure rendering higher hoop stress build up opposite to this; in the latter case higher stress is evident at a stress concentration point i.e. at the sharp corner.

Figure 4.13a showing the thick bodied PRV and Figure 4.13b the thin bodied PRV. The maximum equivalent (Von-Mises) stress for the thick bodied PRV is 228 MPa and is located in the same position as previous Figure 4.12a. A maximum value of 1204 MPa is displayed for the thin bodied PRV, again observed in the same locations as the previous model (Figure 4.12b). Since a linear static analysis was initially performed, prior to fatigue modelling, then as expected the position of the maximum stresses coincide with those previously calculated for maximum strain. It should be noted in the case of the thin wall component failure is of course indicated at this particularly high stress of just over 1.2 GPa. Based on previous calculations this produces a safety factor of the order of 0.25 which indicating significant plastic deformation, whence, rendering any subsequent fatigue analysis redundant in this case.

Figure 4.14a shows a maximum principal stress of 331 MPa displayed at the same location as the preceding thick bodied PRV models. The contour plot of the same PRV illustrated in Figure 4.14b with the addition of a number of radii shows a reduction in the maximum principal stress by approximately one third, to a value of 235 MPa .

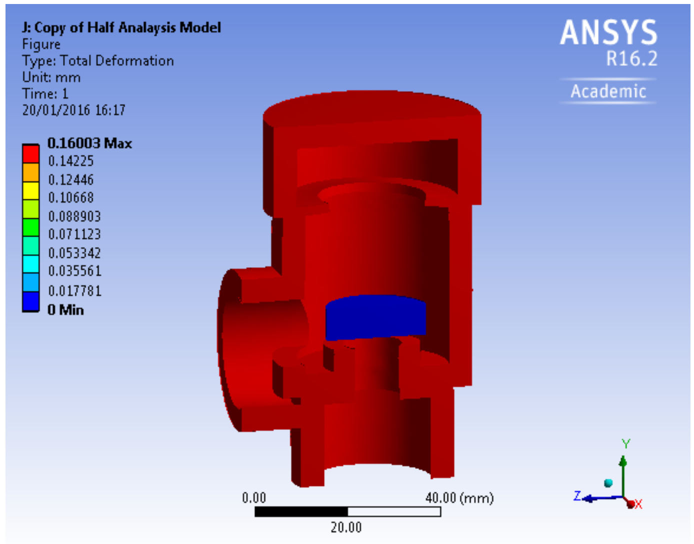
The thin bodied PRV model in Figure 4.14 has a maximum principal stress of

943 MPa shown around the underside of the inlet and decreasing in size as it moves through the material to the outside of the inlet. Additionally, a further area is observed in the top right corner of the PRV body.

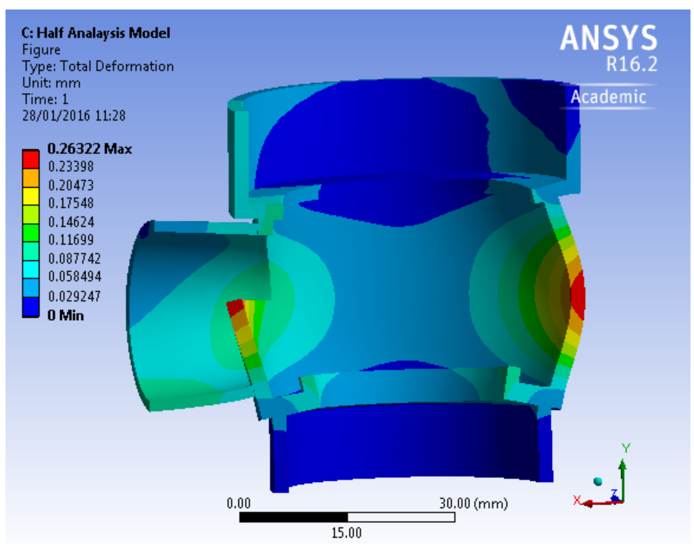
The PRV models in Figure 4.16 (a & b) display the fatigue life value for each of the valve bodies. In Figure 4.16a the thick walled PRV displays a minimum value of 15992 cycles to failure compared to 148 cycles to failure in the thin walled PRV illustrated in Figure 4.16c. The areas of concern (highlighted in red), can be observed in approximately the same regions in all the models. There is good correlation between all of the models displayed in regard to the area(s) of interest/concern.

The contour plots illustrated in Figure 4.16 show the available life of the models for a given fatigue analysis. The loading applied to both the thick and thin models detailed in Figure 4.16 is of constant amplitude: 20 MPa. This represents the number of cycles until the part will fail under fatigue. Figure 4.16a shows that the weak point in the valve represented by the red area would potentially fail after 15992 cycles. Figure 4.16b contour plot shows that the addition of a radius increases the number of cycles to failure to 19059 cycles (an increase in performance of roughly 25). This is a minimum life value, however, the majority of the valve would last for the maximum life value (i.e. a billion cycles). A number of red areas can be observed in the contour plot showing the thin PRV body in Figure 4.16c this would indicate that the thin PRV body would fail after 148 cycles. This of course is a consequence of the prediction of significant plastic deformation (as evidenced in Figures 4.12band 4.13b). *Prima facie* it appears that the software predicts a certain amount of strain hardening in the regions indicated; or more probable is in error, since as mentioned previously fatigue analysis here is redundant and is only included here for completeness. Clearly further analysis could be performed to provide greater in sight in to this particular result.

Figure 4.17 details the contour plots of the Factor of Safety (FS) with respect

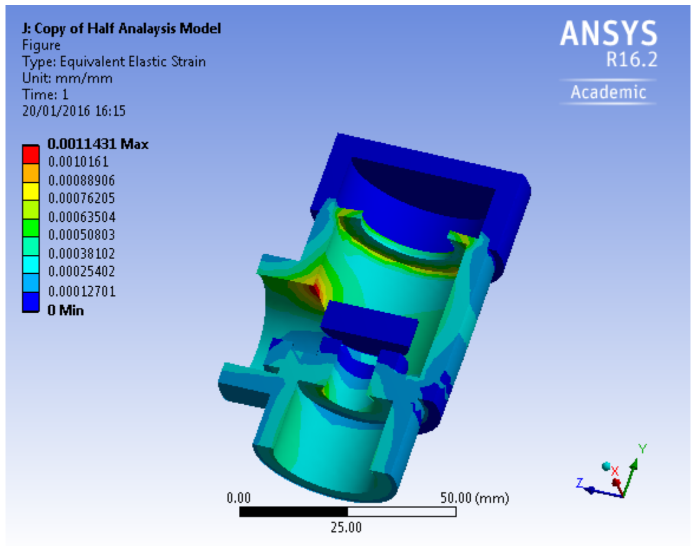


(a) PRV Total Deformation thick.

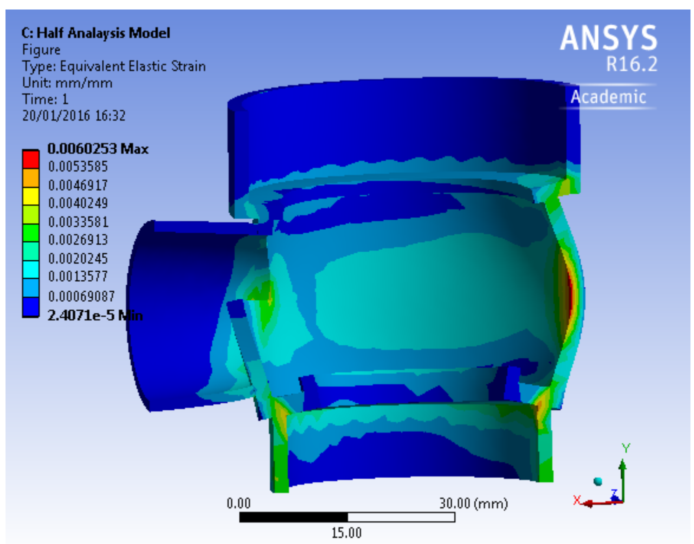


(b) PRV Total deformation thin.

Figure 4.11: Pressure relief valve Total Deformation.

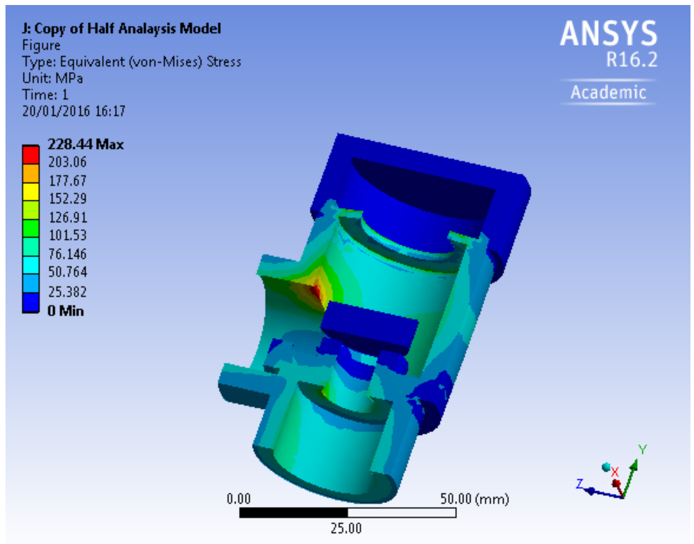


(a) Pressure relief valve Von-Mises (Equivalent) Strain thick.

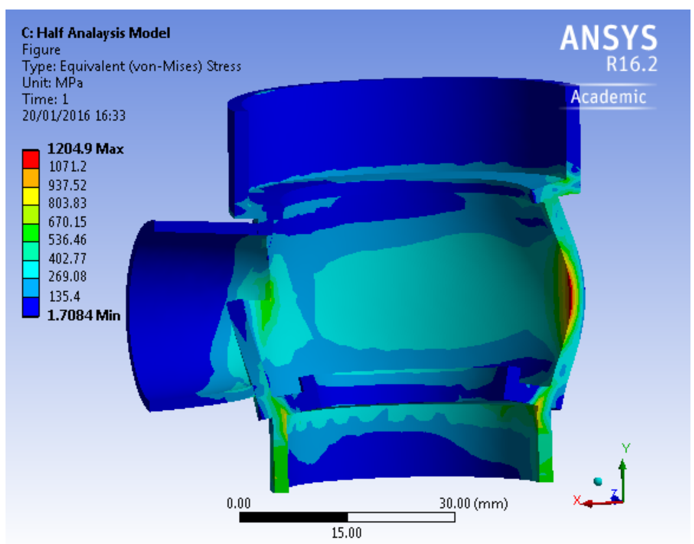


(b) Pressure relief valve Von-Mises (Equivalent) Strain thin.

Figure 4.12: Pressure relief valve Von-Mises (Equivalent) Strain.

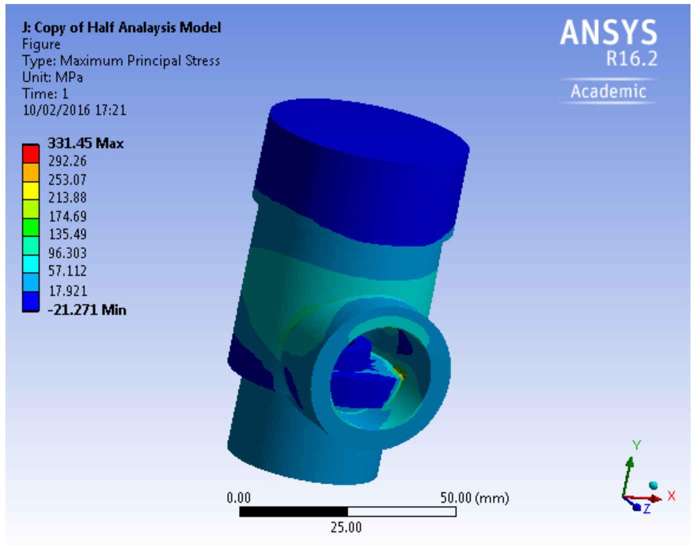


(a) Pressure relief valve Von-Mises stress thick.

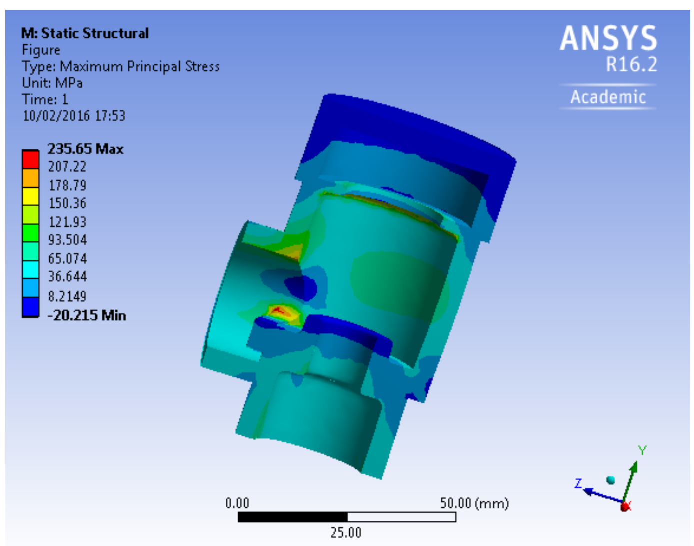


(b) Pressure relief valve Von-Mises stress thin.

Figure 4.13: Pressure relief valve Von-Mises stress.



(a) Pressure relief valve maximum principal stress no radius thick. .



(b) Pressure relief valve maximum principal stress with radius thick.

Figure 4.14: Pressure relief valve maximum principal stress thick.

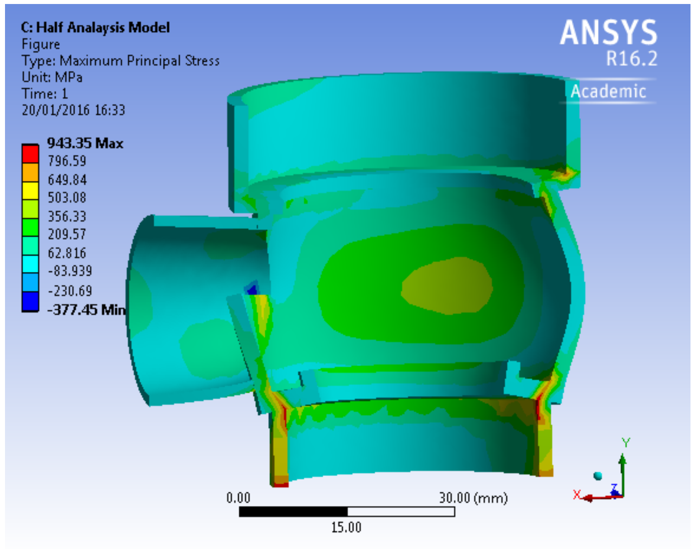


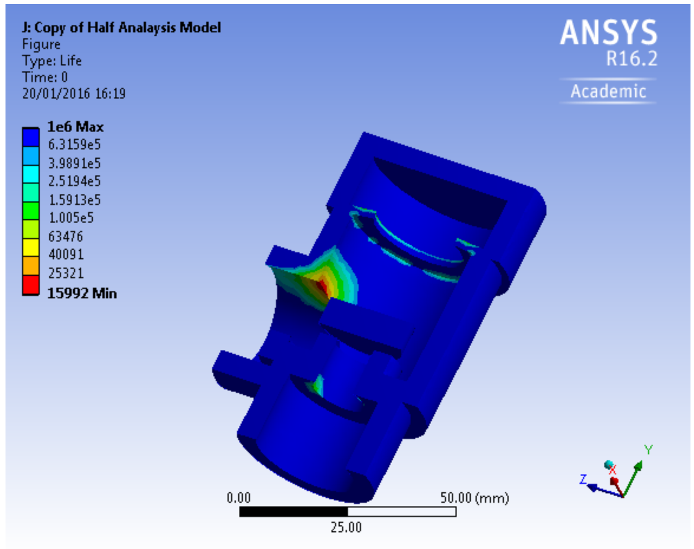
Figure 4.15: Pressure relief valve maximum principal stress thin.

to fatigue failure at a given design life. The maximum FS reported in both Figures 4.17a 4.17b is 15. However, in Figure 4.17b only a few areas are displaying the maximum FS value of 15. The majority of the thin walled PRV is shown as red and this equates to a FS of 0.07. The minimum FS of 0.37 can be be observed in a number of locations in Figure 4.17a for the *thick* component.

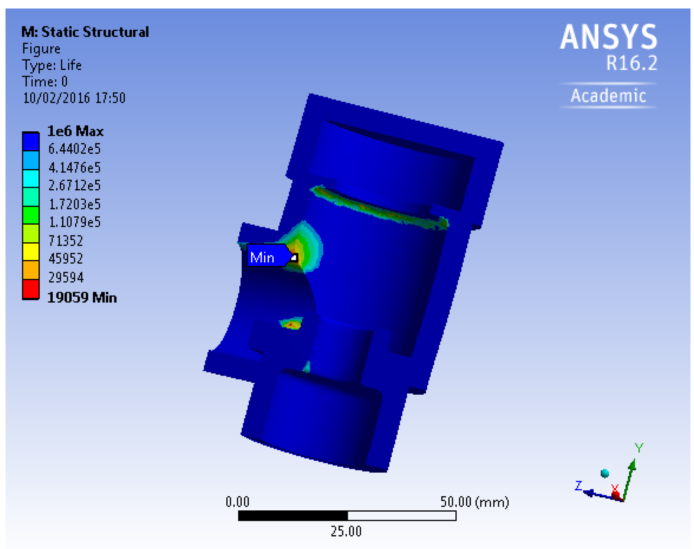
Figure 4.18 shows the contour plots of the fatigue damage, which is defined in the ANSYS mechanical applications user guide as:

> *the design life divided by the available life the default design life may be set through the Options dialogue box* (in this case a default value of 1 million cycles was used). *A damage of greater than 1 will indicate the the part will fail before the design life reached.*
>
> ANSYS *Inc.[25] 2018*

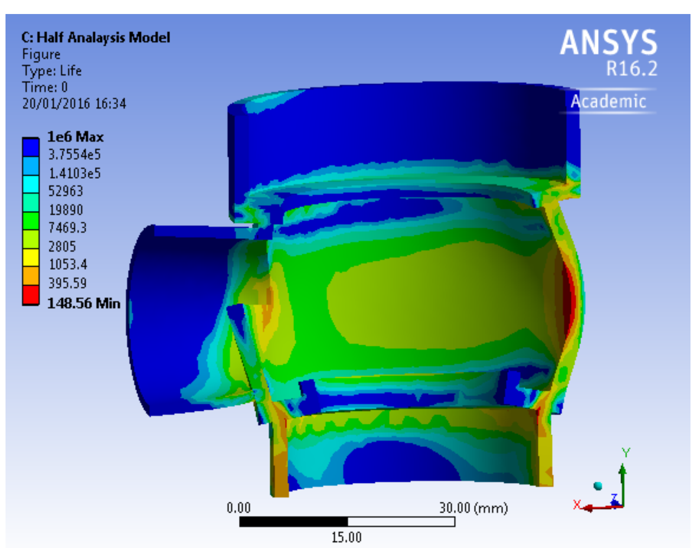
One small area of red can be observed in the thick bodied PRV, shown in Figure 4.18a, with a maximum value of 62530, with almost all of the other areas of the PRV body showing a minimum value of 1000. A maximum damage value of 6.7 million



(a) PRV fatigue life thick.

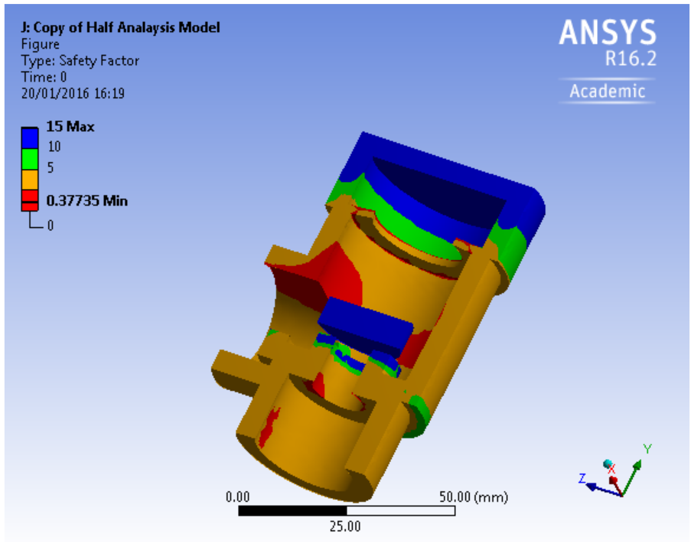


(b) Pressure relief valve fatigue life radius applied.

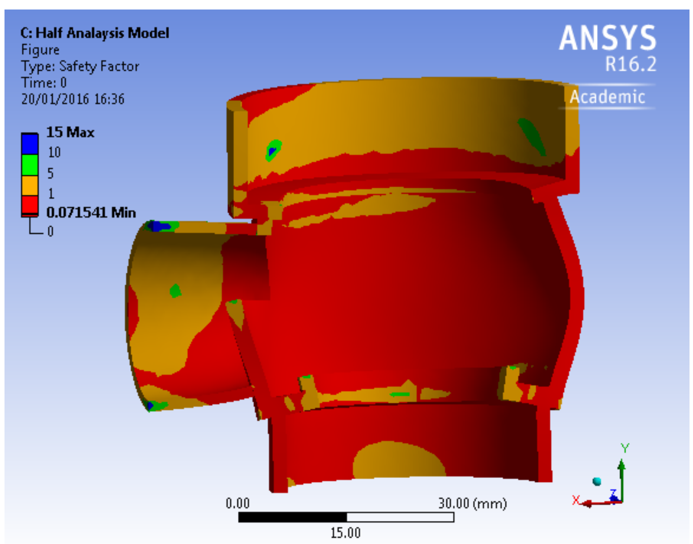


(c) PRV fatigue life thin.

Figure 4.16: Pressure relief valve fatigue life, Thick and Thin



(a) Pressure relief valve safety factor thick.

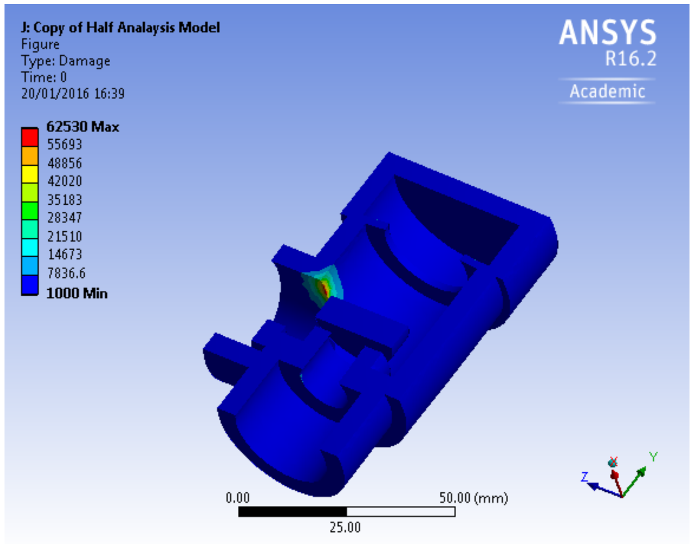


(b) Pressure relief valve safety factor thin.

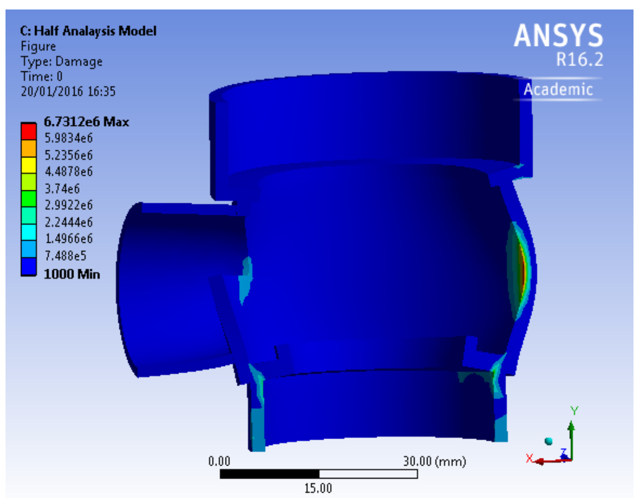
Figure 4.17: Pressure relief valve safety factor, Thick and Thin.

can be observed in the main body of the thin PRV as shown in (Figure 4.18b). Again most of the PRV is at a minimum damage value of 1000. Fatigue damage is defined as design life divided by the available life [101]. Here values greater than unity indicate failure before the design life is reached.

The contour plots illustrated in Figure 4.19 show that by defining a number of cycles to failure a FS of over two can be achieved.

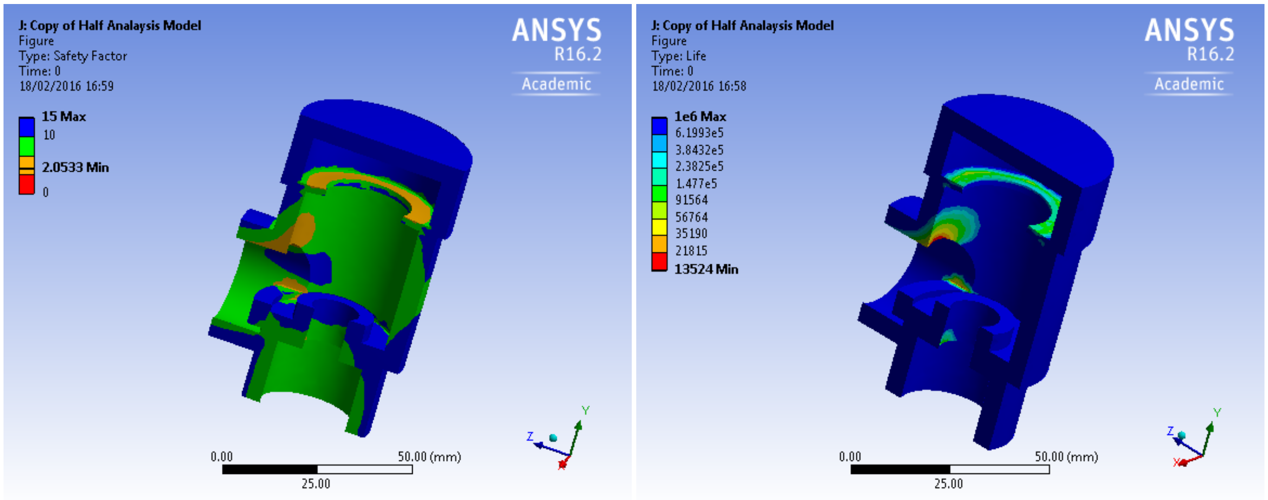


(a) PRV damage Thick.

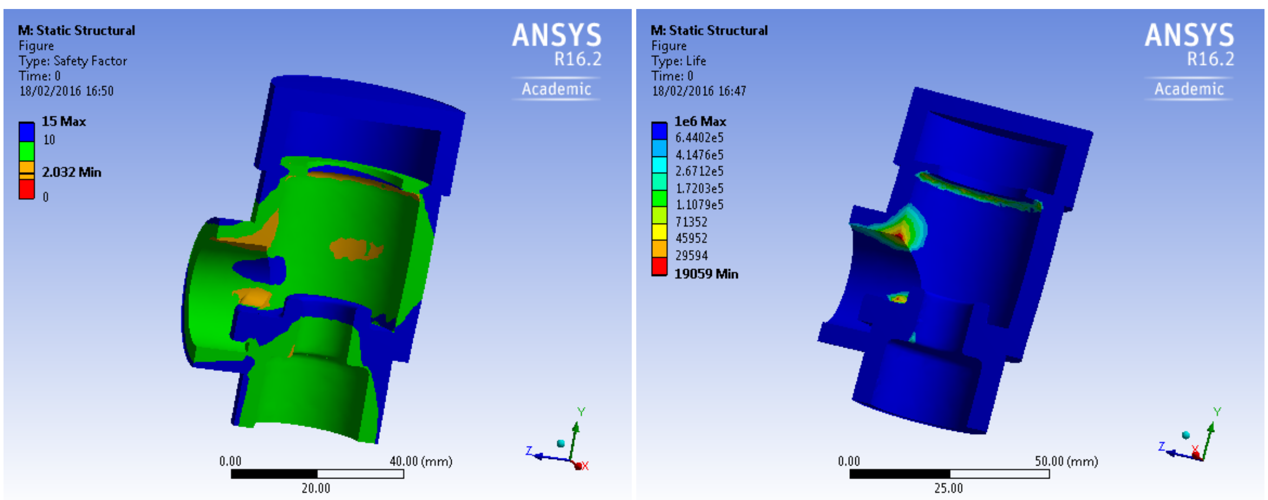


(b) PRV damage thin.

Figure 4.18: Pressure relief valve damage showing thick and thin.



(a) Saftey factor & fatigue life no radius.



(b) Safety factor and fatigue life radius applied.

Figure 4.19: Safety factor and fatigue life, with and without radius applied.

Chapter 5

Discussion

As the human race strives to build bigger mechanically efficient structures with increased performance and maneuverability, one is designing to the limit of failure envelopes. It is important that we investigate and understand failures through research, as this can affect us in many different ways. The early commercial jet airliner programme is a good example of this. Not only from the loss of life but from a commercial point of view. The de Haviland comet jet airliner project of the 1950's was dogged by failures [39]. Investigators had found that up to 70% of the aircraft's ultimate stress under pressure was concentrated on the corners of the aircraft's windows [39], which were of a square corner design; the investigation found that this was due to the constant pressurization and de-pressurization of the cabin (*load and cycles*). As a consequence of the learning from this airliner's failures, fuselage windows are now rounded at the corners to reduce stress. Additionally, there are often commercial issues. The de Haviland comet programme never recovered and Boeing aircraft dominated the commercial airliner market thereafter.

> *The postmortem study of the Comet's problems was one of the most extensive in engineering history. It required salvaging almost the entire aircraft from scattered wreckage on the ocean floor and also involved full-*

scale pressurization of an aircraft in a giant water tank. Although valuable lessons were learned, it is hard to overstate the damage done to the DeHavilland Company and to the British aircraft industry in general. It is sometimes argued that the long predominance of the United States in commercial aircraft is due at least in part to the Comet's misfortune

Prof D. Roylance [38].

The initial focus of this thesis is to present methodologies in order to approximate suitable reliability figures to mechanical components. However, early in the project it soon became apparent that this would be a large undertaking, as the sheer number of mechanical components available in the market place is huge. It would be almost an impossible task due to the many different environments, duty cycles and how hard the component could be used in service; to name a small sample of the factors that would have to be assessed for just one particular mechanical component, so that a reliability figure could be assigned to it (i.e. change one factor and all the models would have to be run and analyzed again); but nonetheless, a methodology has been proposed and demonstrated.

It was outlined in Chapter 2 that over 90% of all mechanical components fail *in-situ* due to fatigue. It therefore became necessary to direct the main focus of the work described in this thesis to the fatigue modelling of safety critical components, in the hope that a suitable method could be established in order to approximate failure times, ergo reliability figures, which could then be employed in Probabilistic Safety Assessments (PSA) of the aforementioned safety cases.

5.1 Fatigue

A number of areas of interest can be observed in both the thick and thin bodied PRV contour plots represented in section § 4.2. The area of maximum principal stress shown in Figure 4.15 is as expected in the stress concentrations at the corner or where the radius is applied with the higher being observed in the former. Data in Figures 4.16c and 4.17b along with the fatigue life, fatigue damage and factor of safety effectively form reliability estimates of the components. Fatigue life is the number of cycles the PRV can operate before it fails. However, if we use this information in comparison with a bath tub curve, one should be able to approximate the region in which the device should fail (in cycles) with the aim of being able to predict the initial onset of excessive wear due to fatigue.

Interestingly, one component shouldn't have a bathtub curve as its a combination of different failure distributions to give a *norm*. It tells you when and where to look first. It also sets defect criteria for establishing critical crack lengths ergo the minimum required life. Another aspect besides cycles/load is material/component quality. If the duty/cycle or load placed on the PRV decreased or increased, one then would either observe an increase in the time period before the onset of fatigue. This could allow for better planning of maintenance activities as will be discussed in Section § 5.5. The bathtub curve can be divided up into four distinct separate phases,viz.:

1. Infant mortality - due to manufacturing defects.
2. Flat line middle section - Random failures.
3. Start of upward curve - Initial onset of wear, fatigue.
4. Failure - Due to wear out fatigue failure.

The third of these is of particular interest with regard to the results presented in the previous chapter.

5.1.1 Mitigating action to reduce fatigue

As discussed in sections 1.6 and 2.1, fatigue of engineering materials is sensitive to a number of variables. Examples of these factors include mean stress level, geometry, surface finish and environmental conditions. Measures may be taken to minimize the possibility of fatigue failure; however, cracks and crack nucleation sites will always exist in structural components. This section discusses some steps that could be taken to improve SSC resistance to fatigue, ergo increase the reliability of mechanical components *in-situ.*

Surface effects – As mentioned in §1.6 & 2.1 most cracks leading to fatigue failure originate at the surface positions, particularly at stress amplification sites and especially relate to the condition and configuration of the component's surface. A number of factors influence fatigue resistance, the proper management of which will lead to an improvement in fatigue life. These include design criteria as well as various surface treatments. The six-sigma approach discussed in §2.5 would be one such management approach, to drive down manufacturing defects and surface faults that lead to the formation of cracks and crack initiation sites. Moreover, one should apply this approach to a capital project such as an offshore oil & gas platform or a nuclear reactor pressure vessel, as the costs of failure in these structures would greatly exceed application of the six-sigma philosophy, (ergo the DFSS methodology resident in the ANSYS software).

Design factors - The way a component has been designed can play a major part in its fatigue characteristics. A geometrical discontinuity *(a change of direction)* can act as a stress raiser and hence a site for fatigue initiation; these design features can be grooves, holes, bends and key-ways, etc. The sharper the discontinuity *(the smaller the radius)* the greater the stress concentration [102]. By identifying these potential areas either through the use of FMEA studies (§§2.4.3) one can propose design modifications i.e. design out sudden contour changes leading to sharp corners to eliminate these areas. The use of simulation modelling as shown in sections 3.4.3, 4.1 and 5.5.1 are good examples of this principle. The pressure vessel models shown in Figure 3.13a with no radius applied result in a maximum principal stress of 173 MPa compared to the same pressure vessel with a radius applied (as illustrated in Figure 4.2b) showing a value of 60 MPa. This equates to a reduction of almost three times the stress and an increase in fatigue life from 15992 cycles to 19059 cycles (contour plots Figures 4.16a & 4.16b). The contour models displayed throughout this thesis can be used by engineers to focus on the correct areas as the colour plots displayed indicate the area(s) that require investigation (i.e. red, amber and green, with red being the area of priority).

Surface Treatments - Machining operations carried out during component manufacture invariably introduce small scratches and grooves into the component's surface by the action of the cutting tool. It has been shown that improving the surface finish by methods such as *polishing* or *shot peening* will enhance fatigue life [102, 16, 19]. The areas that would benefit from such methods are identified in the simulation models by the colour code, as mentioned previously above. Any red or amber areas could benefit from such a process. Blunting a crack, typically by drilling a hole at the end of the crack, when it initial appears, is a good way to stop the fatigue crack from growing. The models described throughout this thesis have not been set up to do this and could well be a salient area of further work.

Environmental Effects - Corrosive environments can produce shortened fatigue lives, as small pits may form as a result of chemical attacks. By identifying the environment that a component will be used in, through the FMEA process, preventative methods can be specified at the manufacturing stage to reduce corrosion fatigue. For example, by applying a protective surface coating or select a more corrosive resistant material. It may also be advisable to apply one of the techniques mentioned above to reduce the probability of normal fatigue failure [16]. Since the work described throughout this thesis, was concerned with reliability prediction during the so-called normal operations [4, 3], it was deemed in appropriate to consider these particular effects. These are preliminary modelling studies intended to demonstrate a concept. However, the RCM studies could be used in the future to direct the maintenance requirements when such effects are deemed important *in-situ*.

5.2 Importance of boundary conditions

Boundary conditions are any prescribed quantities, loads and constraints applied to the boundary, that represent the surrounding environment [100]. Loads are forces, pressures, temperatures and constraints resisting the deformations induced by the loads. If a model is over constrained (i.e. more supports added than needed or excessive constraints applied) this tends to add a false stiffness to a model (stiffer than the real thing) and also prohibits *Poisson* contraction, making excessive stress. Under-constrained or insufficient stiffness in the model can lead to rigid-body motion, causing cracks or high stress areas at the boundary as observed in Figure 3.13, where a region of high stress (84 MPa) can be observed at the bottom inside edge of the two-dimensional model but not observed in the same area on the three-dimensional model, due to a vessel support being missed off the two-dimensional model, thus

Model	Model dimensions (mm)	Model yield onset (MPa)	Model plastic collapse (MPa)
Thin wall PRV	$R_i = 18, R_o = 20$	23.75	26.34
Thick wall PRV	$R_i = 25, R_o = 35$	61.22	84.12
Thin pressure vessel	$R_i = 75, R_o = 100$	54.69	71.92
Thick pressure vessel	$R_i = 50, R_o = 100$	93.75	173.28

Table 5.1: Plastic collapse pressure values of models presented.

highlighting the importance of getting the boundary conditions correct.

It was pointed out in the Chapter 3 that each of the static loads applied to each of the models were below the on-set of yielding as per the appropriate ASME code [21] as evidenced by the equivalent stress safety-factor being of the order of 2; hence each of the pressure loads applied were also below the on-set of yielding ergo overall plastic collapse, as evaluated by direct application of equations (3.6) and (3.7) respectively. Table 5.1 (with with the yield of structural steel $\sigma_y = 250$ MPa employed) clearly demonstrates this for each of the salient models detailed throughout the work in the preceding chapters of this thesis.

5.3 Design optimization

Traditionally, engineering problems, have been formulated to handle uncertainty through conservative or crude safety factor methods that often lead to over designed products and do not offer insight into the effects of uncertainties; such as material selection/characteristics, loading and usage, manufacturing precision, degradation of the product over its lifetime and the actual margin of safety required [9, 81, 83]. This can be observed in the thick bodied PRV shown in Figure 4.16a which shows values indicating that it will never fail. More recently, however, it has been shown that a product's performance and quality is determined by early design decisions [81, 103]. A crude optimization exercise was performed as part of this work to highlight the two extremes; Figure 4.16c clearly shows that the thin bodied PRV would fatigue and fail quickly. A more robust method as used by Vlahinos [82] and discussed in section 2.5 could be used to find the optimal thickness of the PRV body without compromising the function of the PRV.

5.4 Ageing management

Nuclear facilities and power plants from an economic point of view represent a large capital investment for any government or operating organization [104]. The associated capital costs are very high and the time periods to design, construct and commission are usually lengthy, thus the return on investment can be quite long. Therefore, once a facility is in operation, it is important for the operating organization to keep it in service as long as possible; and continually, so this makes it cost effective for an organization to have a robust systematic ageing management process in place during design and operation: to investigate, understand and effectively manage ageing of SSC's, as this is a key element of safe and reliable operation of nuclear facilities and power plants. The IAEA [105] say that to maintain plant safety it is very important to detect ageing[1] effects of SSCs, to address associated reductions in capability in safety and to take corrective actions before loss of integrity or functional capability occurs.

Ageing is a time and duty dependent change in the characteristics of a SSC. Ageing mechanisms are various, such as (but not limited to), fatigue, wear, curing, erosion, creep, corrosion, embrittlement, chemical or biological reactions and/or combinations of these processes (e.g. creep-fatigue, erosion-corrosion). It is worth

[1] Ageing effects are net changes in the characteristics of an SSC that occur with time or use which are due to ageing mechanisms. Ageing effects may be positive or negative. Examples of positive effects are increases in concrete strength from curing and reduced vibration from wear-in of rotating machinery. Examples of negative effects are reduction in diameter from wear of a rotating shaft, cracking, thinning or loss of material strength from fatigue or thermal ageing, and loss of dielectric strength or cracking of cable insulation.

mentioning at this point that obsolescence is also an ageing mechanism, due to the SSC becoming out of date with current technology, standards, regulations and knowledge [105]. Since ageing can have an impact on safety and performance of nuclear facilities, effective and proactive management of the ageing of SSC's is a key element of the safe and reliable operation of nuclear installations [104]. To effectively deploy an ageing management process a systematic approach should be taken at each stage of a facility's life-cycle; that is during design, construction, commissioning, operations including maintenance and decommissioning [106]. The lack of spares or technical support can lead to a decline in plant performance and/or availability and reduction in safety. This would be due to increasing failure rates, ergo decreasing reliability or availability. Knowledge is concerned with maintaining understanding of standards, regulations and technology relevant to the SSC's, but also knowing what you have and what it was designed to do; i.e. this knowledge is not maintained and updated, then the capability for long term operation is reduced. Deviations from design standards and regulations can lead to design weakness (i.e. poor equipment qualification). The potential consequences of this are plant safety levels below current standards and regulations, again leading to a reduced facility capability in the long term unless appropriate modernization and upgrading takes place [105, 107]. The management of obsolescence is part of the process of continuous improvement for enhancement of both SSC performance and nuclear plant safety.

It is well understood in engineering that mechanical and electrical SSC's can be kept operational or maintained over long periods of time, using refurbishment, partial /complete replacement and reconditioning. Examples of this are, early automobiles from the 1900's which are still in as good condition and working order as the day they were manufactured [104, 108]. The ability to maintain SSC's in an 'as new' condition is called effective ageing management. The foundation of ageing management is based on three basic principles [108].

The first principal is that there cannot be a reduction in safety margins over the useful life of the plant. This implies that the plant licensees must maintain the plants in the 'as new' condition. This is policed by the Health & Safety Executive (oil & gas), and the Office of Nuclear Regulation (Nuclear) through Site Licence Condition 28[2] (SLC 28) in the U.K., Examination, Inspection, Maintenance, and Testing (EIM&T). The United States Nuclear Regulatory Commission (USNRC) carries out the same function in the United States (U.S) [108].

The second principal is to avoid intolerable failures. The reliability of the plant will never be better than its worst performing system or component [104, 109, 105, 108, 53]. Thus, to avoid failures companies and licensees must have the skills, knowledge and experience in place to be able to recognize and predict the pending failures and take appropriate timely corrective actions for all Structures, Systems and Components (SSC) that are critical to the safe operation of the plant; and predict which are likely to fail and target them.

The third principal is to understand the behavior of the materials when exposed to such loading conditions i.e. understand the applicable ageing mechanisms, be it environmental or mechanical etc. This knowledge helps to focus attention to the right place at the right time; and furthermore, providing the information necessary to address the ageing/degradation mechanisms; and to do this with the correct tools for developing effective actions to mitigate or prevent the problem from occurring, thus allowing safe plant operations to continue [104, 105, 108].

Understanding the ageing of a system, structure or component, is key to its effective ageing management [105]. The underlying reason for significant ageing and degradation of many SSC's in existing nuclear facilities is that there was insufficient

[2]SLC 28 states that to ensure that all plant that may affect safety, as identified in the safety case, receives regular and systematic examination, inspection, maintenance, and testing (EIMT) by and under the control of SQEP's in accordance with the plant maintenance schedule. The licensee should have a general program covering all aspects of EIMT for all plant on site. The SLC cover the arrangements for updating or amending the maintenance schedules.

understanding and therefore a lack of predictability, of the plant at the design and construction stage [104].

This understanding is derived from knowledge of:

- The design basis (including applicable codes and standards);
- Safety functions; —The design and fabrication (including the material, material properties, specific service conditions, manufacturing inspection/examination and testing);
- Equipment qualification (where applicable);
- Operation and maintenance history (including commissioning, repair, modification and surveillance);
- Generic and plant specific operating experience;
- Relevant research and development results;
- Data and trends from condition monitoring, inspection and maintenance.

5.5 Reliability Centered Maintenance

Reliability Centered Maintenance (RCM) was born out of the airline industry's Maintenance Steering Group (MSG) in the early 1960's. Stanley Nowlan and Howard Heap of United Airlines formally introduced RCM to the commercial airline industry in 1978 with the paper *Reliability-centered Maintenance* [110]. The objective of the techniques outlined in the paper was to develop a scheduled-maintenance program that assured the maximum safety and reliability of which the equipment was capable at a reduced cost [111]. Additionally, in the early 1980's seperate but parallel work was being conducted in the United States (US) nuclear power industry, by the Electric Power Research Institute (EPRI); two pilot applications of RCM were being trialed. Their interest arose from the belief that the industry was achieving adequate levels of safety and reliability, but was massively over maintaining its equipment [107, 111]. Their main goal was to reduce maintenance costs. The aim of the work in this thesis would be to better inform this type of process with a better technical understanding of where the failure would likely occur; and at approximately what time period. RCM is a logical way of identifying what equipment in your facility is required to be maintained on a preventative basis, rather than a reactive basis. When striving to achieve good levels of reliability and safety of a facility it is unsustainable to use the reactive approach. As the reliability will be based on a best guess situation [24], it thereby falls short of modern day expectations.

Equipment can fail in a number of ways (i.e. nature, human error or equipment failure). Acts of nature such as earthquakes and hurricanes etc are unavoidable, but engineering standards are available to help protect against damage during these extreme events. Human error has some latitude for movement and is a key attribute in preventing an occurrence. This engineering proficiency is achieved through good training/education, experience, procedural guidance and rigid design codes and standards [77]. It is not the purpose of this thesis to explore the factors creating safety risk. Nothing and no-one is 100% reliable, but SSC's together with some of the work described throughout this thesis at least provide suitable mechanisms for the exploration of likely failure, to enable achievement of a minimum reliability target in a cost effective manner. Thus, prevention of failure will ultimately increase the level of control to maintain facilities, whether it is an aircraft or a nuclear facility.

Companies today are focusing their efforts on good asset management as a means to safeguard their assets and reputation. By achieving reliability levels that prevent or minimize unplanned production delays, maintain power generation, ensure personnel/public and plant safety and prevent regulatory or environmental issues that bring unwanted publicity[3]. Fundamentally, asset management is about identifying P&E functions that must be preserved to protect the company's assets and ultimately their reputation.

In chapter § 4.2 further work was conducted to increase the Factor of Safety (FS) in the PRV to the accepted standard level of 2. For this, one had to first define a design life i.e. the number of cycles required before the onset of failure; the PRV (without the radius applied at the bend) reached a design life of 1500 cycles before failure. However, when the radius was applied to the PRV the number of cycles to failure increased to 2000 cycles, an increase of ~25 percent. When we look at obsolescence as an ageing mechanism, this too is an important value to consider (as it could inform a company's investment plan). For example, if one assumes the life of a process plant to be 26 years and it has 10 units of this type of PRV installed, and the valve supplier/company changes the valve specifications every 10 years, (making the valve obsolete after a period of time); the receiving company can then purchase a required number of valves at the outset before they become obsolete. Very simply, with:

10 valves in the facility, and assuming

Each valve does 500 cycles per/annum, then its MTTF is 2000 cycles

Every 4 years all valves will need replacing.

Thus the company would need 65 valves to keep the facility operational. This could be factored in before the start of operations to provide a more informed understanding of the facilities life cycle costs; as much as 80% of the total lifecycle costs of an asset are determined in the design phase [71].

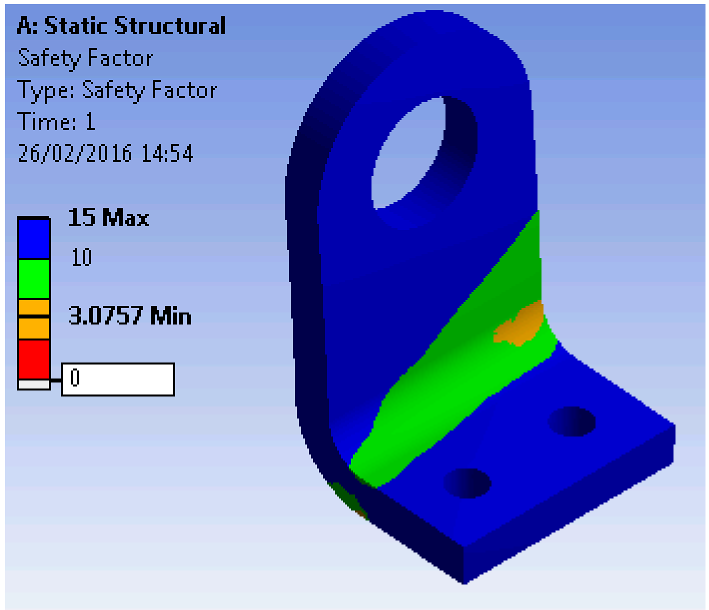
[3] http://www.ulrich-eppinger.net/

5.5.1 Predictive reliability model

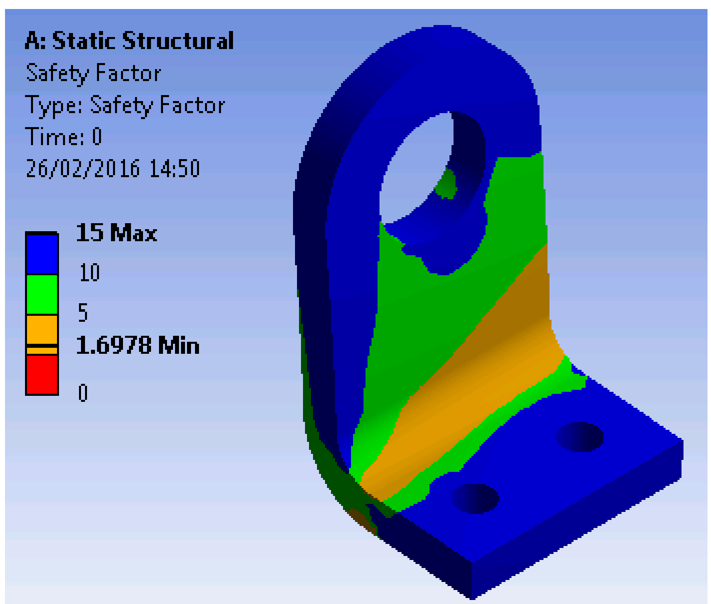
The simulated fatigue modelling from ANSYS workbench can therefore be employed to effectively affirm the research hypothesis, set out in §1.5.2, as the life of the component, given the acceptable conservative factor of safety, can be evaluated. This is achieved by iteratively finding the number of cycles which corresponds to an appropriate static failure safety factor, as dictated by a relevant design code [4]. To illustrate this consider the model shown in Figure 5.1.

The design life evaluated can essentially be taken as the estimate of the components Mean Cycles to Failure (MCTF). Suppose for convenience that the component is to be loaded fifty times per day. Given this condition the Mean Time To Failure (MTTF) of the component is 2000 [`days`], ergo a hazard rate of $\lambda = 5 \times 10^{-4}$. This means that the component would last for approximately five and half years. Noting that the component shown in Figure 5.1 is quite generic, the results of the preceding chapter therefore provide a method in order to ensure that the reliability values of *in-situ* components can be readily calculated. For instance the Weibull distribution, equation (1.1), with constant hazard rate (i.e. $\beta = 1$), as indicated in this discussion by Figure 5.1, with a statutory maintenance procedure of every 12 months in place. This particular component has a reliability of $\exp(-365/2000) = 0.833$, i.e. 83%. It should noted here that the reliability calculation is based on a fatigue safety factor of 1.6. That is about half of that under static conditions, and in this case being acceptable for the application. This implies that the component is still significantly over engineered. Reducing the fatigue safety factor by a quarter actually increases the design life of this particular component from 100000 cycles to 400000 cycles. Given the hypothetical scenario which was outlined here whereby the component loaded some fifty times per day, this corresponds to a different MTTF of 8000 [`days`]; with

[4] in this work a factor of two was used in line with many mechanical engineering industry standards



(a) conservative failure criterion



(b) Fatigue failure for 100,000 cycles

Figure 5.1: Bracket static and fatigue analysis

a reliability increasing at each of the aforementioned statutory maintenance time to$100\exp(-365/8000) = 96\%$.

5.6 Scale and validation

In order to validate the rudimentary concept presented throughout this thesis a more rigorous verification of the FEA fatigue modelling procedures used would be required, this would be achieved by underpinning using a variety of numerical and analytical techniques. Thereafter, the modelling methods will be validated using a combination of generated fatigue data as well as reliability figures from the nuclear industry. The scale of the work required to undertake such a detailed validation process would be large as it would require the generation of actual working SSC rigs, to enable real time data to be generated and analysed against that predicted by the simulation models generated by codes such as ANSYS.

It is envisaged at this rather conceptual stage that the benefits of carrying out such research extend beyond the nuclear industry, for the wider engineering community. It would allow industry to provide a more targeted risk based approach early in the design stage of new facilities.

5.7 Engineering substantiation

Engineering substantiation is the formal demonstration that a safety function and associated performance requirements (as appropriate) is met with sufficient confidence (§1.3.1, Table1.1). It is the safety function that should be demonstrated not the specific SSC. However, the concept put forward throughout this thesis is how confidence in the SSC can be demonstrated, to provide the safety case with the required degree of assurance in the SSC that forms part of the safety function. Additional Research and Development (R&D) would be required to provide the adequate verification and validation of the process put forward.

5.8 Closure

The diagram displayed in Figure 5.2 based on Deming's [5] Plan, Do, Check, Act (PDCA) model [112, 71] is used to illustrate how the author of this thesis would link together the different processes to help companies make a more informed decision of how their assets would initially perform; and also how they could better understand how they could age and degrade over a given time period, allowing for a more informed asset investment plan to be developed for the future. A reduction in maintenance costs may be achieved also as the correct maintenance tasks can be targeted to the correct component. The data recorded as part of the maintenance task can be fed forward to update the asset condition process and provide feedback to designers to facilitate improvement in future designs.

[5] W. Edwards Deming- Engineer & Statistician widely recognized as the main driving force behind the Japanese industrial success post WWII.

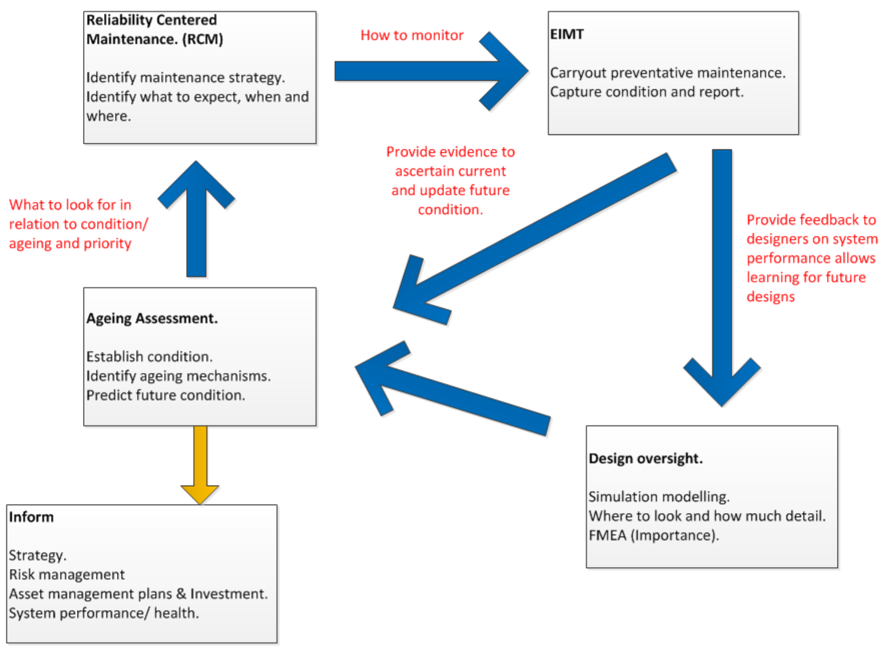


Figure 5.2: Process cycle.

Chapter 6

Conclusions and recommendations

This thesis has used finite element modelling to investigate potential areas of weakness within the body of a Pressure Relief Valve (PRV). In this final chapter the main conclusions which can be drawn from the results will be put forward, followed by some recommendations for further work in this field.

Before moving to the multiple technical conclusions that can be drawn from this work, it must be be pointed out that all of the objectives have been met, ergo the aims satisfied.

In the second chapter a thorough exposition of the literature critically evaluated a number of safety mechanisms, while a preliminary FMEA detailed critical components of a PRV worthy of study in the development of a predictive reliability model, which effectively formulated a modelling research hypotheses with regard to lifetime and hence reliability predictions; thereby provide suitable selection metrics for the evaluation of component lifetime predictions. A a number of case-studies were examined, especially in chapters three and four, with the analyses presented being underpinned using ANSYS simulation. Together with analytic methods evident in the literature; this thereby satisfies the fourth and fifth objectives of this thesis (§§1.5.2). The results represented in Chapter 4 clearly showed that the AN-

SYS software was able to predict the lifespan of PRVs during the front-end design stages. In the preceding chapter generated fatigue analysis data was used in order to approximate the reliability values (§5.5.1) of a generic mechanical component. Though no inference has been attempted with reference to the validity of these reliability figures, it is envisaged that they would be of use in a T-RAM system such as that used on Sellafield site in West Cumbria UK.

6.1 Conclusions

The main technical conclusions that can be drawn from the work presented in this thesis, are as follows:

- A predictive reliability method to evaluate the Mean Cycles To Failure (MCTF) has been presented (§5.5.1), using the fatigue analysis which approximates the reliability functions of a variety of mechanical components which may be present on a nuclear decommissioning site. Though no inference has been attempted with reference to the validity of the probabilistic assessment methods, the reliability values calculated were reasonable when a simplistic constant hazard rate reliability model was employed. It is envisaged that this particular method, as presented in this thesis, can inform maintenance protocol and/or the PSA data resident in the site nuclear safety cases.

- The aforementioned reliability prediction method using the ANSYS fatigue modelling capabilities therefore clearly indicated that *this particular nuclear Industry standard design simulation software code can be used to predict the life, ergo the reliability of mechanical components.*

- MTTF values obtained were reliant on Finite Element Modelling. Hence rudimentary analyses were carried out on a thick-wall pressure vessels using ANSYS finite element codes in order to effectively verify the use of the code. The benchmarking exercise was carried out to determine if the modelling protocols were set up correctly. The ANSYS code was used as this is the preferred industry standard for carrying out finite element modelling.

- The benchmarking study has compared a thick walled pressure vessel with an internal fillet applied, with a pressure vessel of the same dimensions but with no internal fillet applied. The observation from the results reveal that the vessel with no radius applied showed a Maximum principal stress at the internal bend of 173.66 MPa compared to 60.08 MPa for the vessel with a radius. Thus showing that the introduction of a radius reduces the applied stresses. Additionally, this observation can also be seen in the PRV. The PRV without a radius displays a maximum principal stress of 331.45 MPa and the PRV with the radius added shows a value of 235.65 MPa . This was evidenced by fully verified models between ANSYS simulation, modelled as thick and thin wall cylinders. Here particularly good agreement between stress-field calculated via the modelling protocols were demonstrated between thick-wall cylinder theory and ANSYS simulations.

- The benchmarking process therefore showed that both analytical and simulation modelling approaches estimated the maximum principal stress of the cylindrical portions of the components very well. This implies that for point estimates of the principal stress values thin wall cylinder theory are both appropriate.

- Both the two and three dimensional FE models predicted the stress concentrations at the corners of the benchmark models, where values of stress concentration factor of 2.7 was observed. Moreover material non-linearities were observed as expected at regions were these stress concentrations dominated. In the presence of a radius equivalent to approximately 40% of the inner radius rendered a reduction in the maximum principal stress value by some 20 MPa, thereby reducing the aforementioned stress concentration significantly to 1.7.

- Points in models where stress concentrations were observed effectively demonstrated where cracks were likely to form within the component. Simulations of the fatigue life and damage verified this clearly, indicating a rapid reduction in the life calculation at the stress concentrations. Additionally, as expected, a non-linear relationship was found between the component life and the maximum allowable stress, as evidenced by the reduction in safety factors.

- The static analysis of the PRV showed an initial safety factor of 1.5 apparent in the design based on the Von-Mises failure criterion. Since only one small part of the component experiences stresses in excess of 200 MPa it is entirely plausible that the designers were prepared to accept this. A just over three-fold yield of the material was observed when the thickness of the outer shell of the component was reduced by about half. Rather surprisingly the failure of the component moved to the opposite side. Clearly, further work could have been carried out to investigate this phenomenon further, however it was beyond the scope of what was required here for MSc, and also since the material had obviously failed under static conditions and therefore was not fit for purpose with these new dimensions assigned. As with the benchmark component the onset of fatigue in both thin and thick wall cases were clearly revealed from plots of the maximum principal stress.

- The number of cycles to failure for the nominal size PRV casing was some15992 cycles. The ANSYS software showed almost immediate failure of the component when the thickness of the outer shell was reduced almost by two-fold, this being evidenced by an almost zero factor of safety i.e. 0.07 being evident. This said, the fatigue damage was shown to result over quite small regions of the component which indicates possible areas for improvement of the design.

- The addition of a radius to the PRV showed an increase in the number of cycles 19059 to failure over the PRV without a radius, which displayed a value of 15992 cycles until failure. It is fair to report that adding a radius can increase the fatigue life of a component. For a design life of 10^9cycles this results in a factor of safety of approximately 0.4 and 0.6 respectively.

- The rudimentary analysis of a generic bracket which was to be loaded some fifty times per day showed that reducing the fatigue safety factor down by 22% would increase the design life of the component four-fold. When statutory maintenance times are employed this implies a reduction in failure rates of about about 9%.

6.2 Recommendations

There are a number of further areas of study which could further expand understanding in this field. Along with further studies into how the use of simulation modelling at the design stage can influence the maintenance strategies, through understanding of how the asset will age; Thus enabling further studies into how the reliability of components could be increased and optimized. There is certainly scope in extending the methodology proposed, with greater attention being directed toward the front-end Finite Element fatigue analysis using the ANSYS software to effectively evaluate the Mean Cycles To Failure. Three-dimensional generic mechanical nuclear safety systems/mechanisms constructed of materials other than steel as investigated throughout the work described herein. The expansion of these methodologies explored may have shown that the ANSYS-Workbench software can be used to predict the life, ergo the reliability of mechanical components. However the validity of said predictions are as yet unclear.

The author is currently working closely with the supervisor of this project to produce a PhD project proposal which will firstly aim to more rigorously verify the Finite Element Analysis (FEA) fatigue modelling procedures used through the development of a general theoretical framework. Thereafter, validation will be preformed using a combination of generated fatigue data as well as reliability figures, hopefully increasing the accuracy of the reliability prediction of mechanical equipment in order to further inform the living safety case documentation. The following is a brief treatise of some of the further avenues for possible research.

1. Validation of the model described in this thesis by the generation of actual fatigue data and comparing to the simulation modelling data generated.

2. Use a fatigue analysis based predictive reliability model to estimate specific failure rates of real components on-site. Hence, generate accurate predictions of the lifespan of components during the front-end design stages and *in-situ*.

3. An interesting area of study would be to find the optimal radius size to apply to a component to maximize stress reduction ergo the onset of fatigue; as it is already known and can be observed in Section § 4.2 of this thesis, that the application of a radius at a bend will reduce the maximum principal stress value and increase the life of the component.

4. A further avenue of research would be to use Fluid Solid Interaction (FSI) simulation modelling to determine and understand the weak point(s) in a system design i.e. is the problem caused by cavitation or stress.

5. Another direction of research could be to expand the method discussed above and apply it to three-dimensional printing. For example if an important

item/component has been found to be obsolete or the conventional manufacturing lead time is too long, would it be possible to use ANSYS simulation software to substantiate its use in the workplace and determine what the cost savings would be over the traditional design process.

Looking further into the future if the validation of this thesis is proven to be correct through one of the recommendations for further work; a useful expansion of this work could be to explore the possibility of linking up with other research engineering disciplines and combing them to establish system life predictions taking into account how the welding of materials affects the overall performance [113, 114].

References

[1] Office of Nuclear Regulation. The purpose, scope, and content of safety cases, ns-tast-gd-051 revision 3. Technical report, Office of Nuclear Regulation, 2013.

[2] S. D. Moore. Fatigue analysis of rotating components. University of Central Lancashire, UCLan-Nuclear BSc dissertation, School of Computing, Engineering and Physical Sciences, May 2014.

[3] Richard Maguire. *Safety Cases and Safety Reports. Meaning, Motivation and Management*. Ashgate, 2009.

[4] Anon. *A guide to the Control of Major Accident Hazards Regulations 1999*. HSE Books 1999, 1999.

[5] Nuclear installations act 1965.

[6] Pierre R. Roberge. *Handbook of Corrosion Engineering*. McGraw-Hill, 1999.

[7] International Nuclear Safety Advisory Group. Insag-19 a report by the international nuclear safety advisory group, maintaining the design integrity of nuclear installations throughout their operating life. Technical report, IAEA, 2003.

[8] T. Afzal. Private Cores., July 2014.

[9] P. Koch. Probabilistic design: optimizing for six sigma quality. In *43rd AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials*, 2002.

[10] ONR. A guide to nuclear regulation in the uk. ONR website, 2014.

[11] ONR. Safety assessment principles for nuclear facilities, office for nuclear regulation. ONR web site, 2014.

[12] Anon. Probabilistic safety analysis nuclear safety technical guide ns-tast-gd-030 rev 4 office for nuclear regulation. Technical report, ONR, 2013.

[13] Anon. Categorisation of safety functions and classification of structures, systems and components, nuclear safety technical guide, ns-tast-gd-094 rev 0, office for nuclear regulation. Technical report, ONR, 2015.

[14] Anon. Design safety assurance, nuclear safety technical assessment guide, ns-tast-gd-057 revision 3,. Technical report, ONR, November 2014.

[15] P. D. T. O'Connor with D.Newton and R.Bromley. *Practical reliability engineering*. Wiley, 2002.

[16] W. D. Callister. *Fundamentals of materials science and engineering: an interactive e.text*. Wiley, 2001.

[17] M. F. Ashby. *Materials Selection in Mechanical Design*. Butterworth Heineman, 2010.

[18] J. Hearn. *Mechanics of Materials*. Butterworth-Heinemann, 1996.

[19] B. J. Black. *Workshop processes, practices and materials (third edition)*. Oxford : Newnes, 2004.

[20] J. Vernon. *Introduction to Engineering Materials, (Fourth Edition).* John Wiley & Sons, 2003.

[21] Boiler and pressure vessel code of practice bpvc viii.3, 2015.

[22] (https://www.asme.org/about-asme/standards/history-of-asme-standards), accessed 22/05/2017 1130. accessed 22/05/2017 1130.

[23] J L Meriam. and L G Kraige. *Engineering Mechanics Volume 2 Dynamics (Sixth Edition).* John Wiley & Sons, 2010.

[24] D. J. Smith. *Reliability, maintainability and risk: practical methods for engineers including reliability centred maintenance and safety-related systems.* Butterworth-Heinemann, 2005.

[25] ANSYS Inc. Ansys workbench version 15.0. Software, Jan 2014.

[26] Breyfogle. Leveraging business process management and six sigma in process improvement initiatives. Technical report, Smarter Solutions, Inc., October 2004.

[27] Chantal Baril, Soumaya Yacout, and Bernard ClšŠment. Design for six sigma through collaborative multiobjective optimization. *Computers & Industrial Engineering*, 60(1):43 – 55, 2011.

[28] J. Antony. Design for six sigma: a breakthrough business improvement strategy for achieving competitive advantage. *Work Study*, 51(1):6–8, 2002.

[29] W. Schelter. Maxima, a computer algebra system,13.04.2. GNU General Public License (GPL)., August 2013.

[30] Scilab Enterprises. CeCILL license, November 2014.

[31] http://www.engineeringhalloffame.org/profile-rankine.html, May accessed 24/05/2017.

[32] S. Timoshenko and J. N. Goodier. *Theory of Elasticity*. McGraw-Hill Higher Education, 1970.

[33] Michael Baker Ian Stanley. Assessing and modelling the uncertainty in fatigue crack growth in structural steels rr643. Technical report, HSE, 2008.

[34] John Peters. Assessment of valve failures in the offshore oil & gas sector rr162. Technical report, HSE, 2003.

[35] M. C. Smith. Code based failure avoidance assessments of the nesc-1 large scale pressurised thermal shock experiment. In *Pressure Vessels and Piping*, 2002.

[36] Madsen H Egeland J. Structural reliability models and applications. *Int stat rev*, 57:185 203, 1989.

[37] GuoRong Wang LinYan Chen Min Zhao Rong Li BenSheng Huang. The research on failure analysis of fluid cylinder and fatigue life prediction. *Engineering Failure Analysis*, 40, 2014.

[38] D. Roylance. Introduction to fracture mechanics. MIT Lecture notes, September 2001.

[39] 1954: 'metal fatigue' caused comet crashes1954: 'metal fatigue' caused comet crashes. BBC News, October 1954.

[40] R.T. Byrnes S.P. Lynch. An unusual failure of a nickel-aluminium bronze (nab) hydraulic valve. *Engineering Failure Analysis*, Volume 49, 2015.

[41] Kun Zhang Jinyong Yao Tongmin Jiang. Degradation assessment and life prediction of electro-hydraulic servo valve under erosion wear. *Engineering Failure Analysis*, 36, 2014.

[42] M.K. Sapra S. Kundu A.K. Pal P.K. Vijayan K.K. Vaze R.K. Sinha. Design and development of innovative passive valves for nuclear power plant applications. *Nuclear Engineering and Design*, Volume 286, 2015.

[43] Gideon Praveen Kumara Fangsen Cuia Asawinee Danpinida Boyang Sub Jimmy Kim Fatt Honc Hwa Liang Leob. Design and finite element-based fatigue prediction of a new self-expandable percutaneous mitral valve stent. *Computer-Aided Design*, Volume 45, 2013.

[44] Balazs Fekete. New energy-based low cycle fatigue model for reactor steels. *Materials & Design*, 79:42–52, 2015.

[45] M. Smith P J Budden. Numerical validation of a strain-based failure assessment diagram approach to fracture. In *ASME Pressure Vessels and Piping*, 2009.

[46] A Rondon S Guzey. Fatigue evaluation of the api specification 12f shop welded flat bottom tanks. *International Journal of Pressure Vessels and Piping*, Volume 149:Pages 14, 23, January 2017.

[47] S. Chapuliot A. Jaubert S. Courtin S. Marie. An approach for the evaluation of fatigue crack growth in nozzle corners. *International Journal of Pressure Vessels and Piping*, Volume 146:Pages 22, 31, October 2016.

[48] S. Chapuliot. Stress intensity factor calculation in sharp and beveled edge nozzle corners. *International Journal of Pressure Vessels and Piping*, Volume 141:Pages 11, 18, May 2016.

[49] Sujun Wu Huijin Jin Yanbin Sun Luowei Cao. Critical cleavage fracture stress characterization of a508 nuclear pressure vessel steels. *International Journal of Pressure Vessels and Piping*, Volumes 123, 124:Pages 92, 98, November, December 2014.

[50] (https://www.asme.org/engineering-topics/articles/boilers/the-history-of-asmes-boiler-and-pressure), accessed 22/05/2017.

[51] A.H Cottrell. Commentary. a brief overview of work hardening, in: Dislocations in solids 11. *Elsevier B.V.*, 2002.

[52] Clifford Matthews. *Pressure Relief Valves*. Professional Engineering Publishing, 1998.

[53] O'Connor PDT. *Practical Reliability Engineering (Fourth Edition)*. John Wiley & Sons, 2006.

[54] Zio E. Reliability engineering: Old problems and new challenges. *Reliability Engineering & System Safety*, 94, 2009.

[55] Naval Surface Warfare Centre. Handbook of reliability prediction procedures for mechanical equipment. Technical report, Naval Surface Warfare Centre, 2011.

[56] Mil-hdbk-217f, reliability prediction of electronic equipment, 1991.

[57] Anon. Investigations and legal proceedings. British Peteroleum, external report, December 2014. Available at, http://www.bp.com/content/dam/bp/pdf/sustainability/group-reports/investigations-and-legal-proceedings.pdf.

[58] James B. Busey. National transportation safety board, safety recommendation. ntsb/aar-89/053. Technical report, National Transportation Safety Board., 1989.

[59] *NTSB 2001, National Transport Safety Board Pipeline Accident Brief PAB0102, NTSB, December.*, 2001.

[60] Weibull W. A statistical distribution function of wide application. *ASME paper 51-A-6*, 1951.

[61] Sherwin D J and Lees F P. An investigation of the application of failure data analysis to decision making in the maintenance of process plant. *Proc Instn Mech Engrs, Vol 194, No 29,*, 1980.

[62] Schafer R E Mann R N and Singpurwalla N D. *Methods for statistical analysis of reliability and life data.* Wiley, 1974.

[63] M. Xie C.D. Lai. Reliability analysis using an additive weibull model with bathtub-shaped failure rate function. *Reliability Engineering & System Safety Volume 52,*, 1996.

[64] M.I. Mazhar. S Kara. H Kaebernick. Supply chain management in a sustainable environment special issue on frontiers of empirical supply chain research. *Journal of Operations Management Volume 25, Issue 6*, 2007.

[65] Kai. Yang and Basem. El-Haik. *Design for Six Sigma.* McRaw Hill, 2003.

[66] Chris Atkinson Mike Briggs. *Strategies for Effective Maintenance.* IChemE, 2000.

[67] Takata S. Maintenance: changing role in lifecycle management. *Annals of the CIRP*, 53:643–655, 2004.

[68] Risk:improving government's capability to handle risk and uncertainity. the cabinet office. london 2002. crown copyright.

[69] Cotterill & Mazur. Risk and risk perception - a literature review. *Australian Rural Industries Research and Development Coporation*, 2004.

[70] Zhang Zaifang Chu Xuening. A new approach for conceptual design of product and maintenance. *International Journal of Computer Integrated Manufacturing*, 23, 2010.

[71] Asset management - an anatomy, December 2015.

[72] Nader M.Okasha Dan M. Frangopol. Redundancy of structural systems with and witwith maintenance:an approach based on lifetime functions. *Reliability Engineering & System Safety*, 95, 2010.

[73] P. Beaurepaire, M.A. Valdebenito, G.I. Schueller, and H.A. Jensen. Reliability-based optimization of maintenance scheduling of mechanical components under fatigue. *Computer Methods in Applied Mechanics and Engineering*, 221-222(0):24 – 40, 2012.

[74] Stamatis D H. *Failure Mode Effect Analysis FMEA from Theory to Execution (Second Edition)*. ASQ, 2003.

[75] Teoh P C Case K. An evaluation of failure modes and eeffect analysis generation method for conceptual design. *International Journal of Computer Integrated Manufacturing*, 18:279–293, 2005.

[76] Mil-hdbk-1629a, procedures for performing a failure mode effects and criticality analysis, 1980.

[77] Anon. Integrated risk and technical conscience. Limited distribution, Institute of Nuclear Power Operations (INPO), April 2014.

[78] D Bird K Hughes R van der Honert J McAneney W Poortinga. Nuclear power in australia: A comparative analysis of public opinion regarding climate change and the fukishima disaster. *Energy Policy, Volume 65,*, 2014.

[79] W Poortinga M Aoyagi. Public perceptions of climate change and energy before and after fukishima: A comparison between britain and japan,. In *WSA working paper 01-2013,*, 2013.

[80] Brue & Launsby. *Design for six sigma.* 2003.

[81] P N Koch, R J Yang, and L Gu. Design for six sigma therough robust optimization. *Structured Multidisciplinary Optimization*, 26:235–248, 2004.

[82] Andreas Vlahinos kenneth kelly Ahmad Pesaran and Terry Penney. Empowering engineers to generate six sigma quality designs. *American society for quality*, 2003.

[83] Andreas Vlahinos. Applying six sigma to drive down product defect. *ANSYS advantage*, 2, 2008.

[84] Clough R. W. The finite element method in plane stress analysis. *Proceedings of the second ASCE conference on Electronic Computation*, 1960.

[85] Turner M J Clough R W Martin H C Topp L.C Topp L. Stiffness and deflection analysis of complex structures. *Journal of Aeronautical Sciences.*, 1956.

[86] Zienkiewicz O.C. Origins, milestones and directions of finite element method - a personal view. *Handbook of Numerical Analysis Vol IV*, 1995.

[87] Edoardo Patelli, H. Murat Panayirci, Matteo Broggi, Barbara Goller, Pierre Beaurepaire, Helmut J. Pradlwarter, and Gerhart I. Schueller. General purpose software for efficient uncertainty management of large finite element models. *Finite Elements in Analysis and Design*, 51(0):31 – 48, 2012.

[88] R. Courant. Variational methods for the solution of problems of equilibrium and vibrations. *Bulletin American Mathematical Society*, 49(2):165–172, 1943. 187.

[89] M. J. Turner, R. W. Clough, H. C. Martin, and L. P. Topp. Stiffness and deflection analysis of complex structures. *Aerospace science and technology*, 23:805–823, 1956.

[90] O. C. Zienkiewicz and R. L. Taylor. *The Finite Element Method*. Butterworth-Heinemann, Oxford, sixth edition, 2005.

[91] S. Moaveni. *Finite Element Analysis Theory and Application with ANSYS*. Prentice Hall, 2007.

[92] https://uk.comsol.com/company2015.

[93] http://www.ansys.com/en-gb/2016.

[94] http://investors.ansys.com/ /media/files/a/ansys-ir/annual-reports/ansys-2013-annual-report.pdf.

[95] S. Scampoli. Introducing ansys aim: simulation for every engineer. *ANSYS advantage*, 2015.

[96] http://maxima.sourceforge.net/2016.

[97] http://www.scilab.org/2016.

[98] http://calculix.de/2015.

[99] J.P.M Whitty B. Henderson J. Francis N Lloyd. Optimized thick- wall cylinders by virtue of poisson's ratio selection. part1 isochoric pressure application. *Nuclear Engineering and Design*, 2010.

[100] Kent L. Lawrence. *ANSYS Workbench Tutorial Release 14, Structural & Thermal Analysis Using the ANSYS Workbench Release 14 Environment.* SDC Publications, 2012.

[101] Hancq D.A Walters A.J Beuth J.L. Development of an object-oriented fatigue tool. *Engineering with Computers Vol 16*, 2000.

[102] Jr. William D. Callister. *Materials Science and Engineering An Introduction, 3rd Edition.* Wiley, 1994.

[103] S. Nathan. Major contract win for sheffield forgemasters. *The Engineer*, 2016.

[104] INTERNATIONAL ATOMIC ENERGY AGENCY. Safety reports series no. 62, proactive management of ageing for nuclear power plants. Technical report, IAEA, 2009.

[105] INTERNATIONAL ATOMIC ENERGY AGENCY. Ageing management for nuclear power plants safety guide,iaea safety standards series no. ns-g-2.12. Technical report, IAEA, 2009.

[106] INPO. Ap-913, equipment reliability process description rev 4. Technical report, INPO, 2013.

[107] EPRI. Ageing identification and assessment checklist, mechanical components, technical report 1009743. Technical report, EPRI, 2004.

[108] CGI Report. Ageing management and life extension in the us nuclear power industry, the petroleum safety authority norway stavanger, norway psa project reference number: No 99b16. Technical report, 2006.

[109] Petroleum, petrochemical and natural gas industries, collection and exchange of reliability and maintenance data for equipment iso14224:2006,.

[110] Nowlan F S. Heap H. Reliability-centered maintenance. Technical report, National Technical Information Service, US Department of Commerce. Springfield, Virginia, 1978.

[111] Moubray John. *Reliability-centered Maintenance, Second Edition.* Butterworth-Heineman, 2007.

[112] Deming W E. Out of the crisis. In *MIT University Press (Originally published under the title Quality, Productivity and Competitive Position).*, 1986.

[113] C D Elcoate M. Smith P J Bouchard R J Dennis. 3-dimensional repair weld simulations - bead sequencing studies. In *ASME, ed. 3-Dimensional Repair Weld Simulations - Bead Sequencing Studies*, 2014.

[114] A. Warren, C. Watson, M. Smith, N. A. Leggatt, P. Hurrell, P. J Bouchard, R. J. Dennis, and S K Bate. Development of weld modelling guidelines in the uk. In *ASME, ed. Development of weld modelling guidelines in the UK*, 2014.