Fitness-for-Service Assessment for Pressurized Components with Geometric Distortion

by

Firdaus bin Harun

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JULY 2008

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

(FIRDAUS BIN HARUN)

ABSTRACT

Fitness-for-Service (FFS) is a set of quantitative methods used to determine the integrity and remaining life of degraded components. Structural integrity of a pressurized component can be put at risk with the presence of geometrical discontinuities. FFS assessment is carried out based on API 579 code of practice to determine if equipment is safe and fit for continued service until the end of some desired period of operation. The objectives are to study and understand the FFS procedures covered in Section 8 of API Recommended Practice 579, as well as performing Level 1, Level 2, and Level 3 assessment procedures for cylindrical shell with out-of-roundness as per API 579. The project was further expanded by comparing data from the FFS assessment procedures with ANSYS simulation, a Finite Element Analysis (FEA) software to evaluate the acceptance criteria. The project demanded multi disciplinary engineering practice, ranging from applying codes and standards, solving case study, as well as computer simulation. The scope of work was targeted on out-of-roundness in terms of dent. The final step of this project would be comparing all analysis done on the model with FFS assessment for verification. Based on the results obtained, reference stress used for failure criteria for FFS Level 2 Assessment is based on the yield strength of the material of the structure.

ACKNOWLEDGEMENTS

First and foremost, I would like to praise my Lord God for His grace and mercy throughout the progress of my Final Year Project. I also would like to acknowledge both my parents and also my family for always being supportive to provide financial and moral support.

I would also like to take this special opportunity to honourably thank to my Final Year Project supervisor, Mr. Kee Kok Eng, for spending in personal time every week to further educate and assist me on the project undertaken while endlessly contributing reading materials which helped me a lot as my reference. He also taught me the aspects needed to complete my Final Year Project and the methodology to be used to analyze the project so that the needed result can be obtained.

In addition to that, I would also like to express my special gratitude to the Steel Recon Industries (SRI) for allowing me to obtain the much needed information. Such cooperation is truly appreciated, which will take one step forward for both our educational institution and industrial development.

I would also like to express my special thanks to all lecturers and others who have either directly or indirectly contributed to the project by giving priceless information on how to conduct and improve the project. Without such assistance from these people, this Final Year Project might not be as successful as it is.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS AND NOMENCLATURES

ASME	–	American Society of Mechanical Engineers
API	–	American Petroleum Institute
DOSH	–	Department of Occupational Safety and Health
FEA	–	Finite Element Analysis
FFS	–	Fitness-for-Service
LTA	–	Locally Thin Area
MPC	–	Material Properties Council, Inc.
MS	–	Malaysian Standards
PVP	–	Pressure Vessels and Piping
RP	–	Recommended Practice
SIRIM	–	Standards and Industrial Research Institute of Malaysia
SRI	–	Steel Recon Industries
UTP	–	Universiti Teknologi Petronas
UTS	–	Ultimate Tensile Strength
YS	–	Yield Strength

LIST OF APPENDICES

CHAPTER 1
INTRODUCTION

1.1 Project Background

Fitness-for-Service (FFS) assessment is performed to make sure that static or fixed equipment, such as pressure vessels, piping, and tanks, will operate safely and reliably for some desired future period.

Nowadays, there is already a standard code that provides guidance to practice periodic inspection. In January 2000, The American Petroleum Institute (API) developed a document, namely API Recommended Practice 579 (API 579) [1] with means to provide guidance in fitness-for-service (FFS) assessment. This standard code will be used as main resource and reference to complete the project.

This FFS assessment will need the person or group involved to master the knowledge covered in five disciplines which are materials; stress analysis and codes and standards; fabrication and welding; inspection; and operation of the system under evaluation [2].

For this final year project, a study is done to analyze and evaluate the acceptance criteria of FFS assessment procedures contained in API 579. An FFS assessment is done on a fire extinguisher, which is a pressurized vessel. The computational procedure was covered in the first semester. For the second semester, analysis on the pressure vessel was conducted using Finite Element Analysis (FEA) to compare the acceptance criteria in FFS assessment with theoretical calculation. The analysis is conducted by using ANSYS software.

1.2 Problem Statement

Structural integrity of a pressurized component can be put at risk with the presence of geometrical discontinuities such as shell distortion (characterized as out-of-roundness, bulge or dent). An equipment with out-of-roundness or dent may cause failure on the equipment and accident may happen if it is been neglected. FFS assessment can be done to evaluate the component if it is fit for service as per API 579. However for complex geometries or loadings, detailed numerical stress analysis is required to assess the integrity. This problem can be solved by performing FFS assessment, ranging from Level 1, 2, and 3. A computer simulation then can be done using ANSYS to evaluate and verify the results obtained.

1.3 Objectives

There are three main objectives in executing this project:

- To study and understand the FFS procedures covered in API 579, and any codes and standards that related to them. The study will be emphasized on Section 8, with only considering shell distortion. The scope is also narrowed down to pressurized cylindrical vessel with dent.
- To perform Level 1, Level 2 and Level 3 assessment procedures for cases involving out-of-roundness in terms of dent, as per API 579.
- To evaluate the acceptance criteria by comparing with FEA, with executing computer simulation using ANSYS.

1.4 Scope of Work

The scopes of work for this project focused on computational analysis and computer simulation. Scope of study for this project also would be ranging from preliminary study of the API Recommended Practice 579, focusing in Section 8 – Weld misalignment and shell distortion; and any other standards and codes related to it. Weld misalignment is not included in this project.

The pressurized equipment for case study which is fire extinguisher is selected and then be studied to gain detail information. The information obtained will be used for the Level 1, Level 2, and Level 3 assessment procedures as required by API 579.

The project will further be continued by evaluating the acceptance criteria by comparing with FEA results. FEA scope includes structural stress analysis of the cylindrical shell of the selected fire extinguisher model. The results then will be compared with FFS assessment results and further discussed.

CHAPTER 2
LITERATURE REVIEW

2.1 Fitness-for-Service (FFS)

George Antaki defined Fitness-for-Service (FFS) as a set of quantitative methods used to determine the integrity and remaining life of degraded components, and to make run-or-repair decisions [2]. It is also termed as fitness-for-purpose or mechanical integrity.

FFS also has its own website [3], defining FFS as a multi-disciplinary engineering analysis of equipment to determine if it is safe and fit for continued service until the end of some desired period of operation, for instance, until the next shutdown, until some specific future date, or until the end of its useful life.

The common reasons for assessing the fitness for service of equipment include the discovery of a flaw such as a locally thin area (LTA) or crack, failure to meet current design standards, and plans for operating under more severe conditions than originally expected. The main products of fitness for service assessment are (1) a decision to run, alter, repair, monitor, or replace the equipment and (2) guidance on inspection interval for the equipment. Fitness for service assessment employs analytical methods, mainly stress analysis, to evaluate flaws such as locally thin areas and cracks as well as damage such as dents, bulges, and distortions.

A joint industry program was initiated by the Materials Properties Council, Inc. (MPC) in 1990 to develop a consistent engineering approach to FFS assessment. The MPC program concentrated on the development of technology, and its results have been circulated through publications and symposium, especially ASME PVP Volumes. The peak of this program was the development and publication of the API Recommended Practice (RP) 579 on Fitness-For-Service.

2.2 API Recommended Practice 579 [1]

The draft of API 579 was started in 1994, and the first edition was published in January 2000. API 579 is organized in uniformed fashion based on type of material damage or flaw to facilitate its use and updating.

API RP 579 defined FFS as the ability to demonstrate the structural integrity of an in-service component containing a flaw or damage [1]. It provides methods and procedures for evaluating the suitability for continued service of pressurized components of fixed equipment that contain flaws or damage.

The procedures in API RP 579 utilize the design and construction rules and methods in the ASME Boiler and Pressure Vessel Code, Section I and Section VIII, Divisions 1 and 2, the ASME B31.1 and B31.3 piping codes, and the API 650 and 620 storage tank standards [4].

API RP 579 is a well-arranged document designed to assist practice by users and to ease future improvement and adjustment. Section 1 of the document covers: introduction and scope; responsibilities of the owner-user, inspector, and engineer; qualification requirements for the inspector and engineer; and references to other codes and standards. An overview of the overall FFS assessment methodology that is general to all assessment procedures contained in API 579 is presented in Section 2 of the document.

Table 2.1 shows general arrangement that is used for each different section [5]. This similar arrangement is used in all following sections that contain FFS assessment procedures, starting from Section 3 until Section 11, which are differentiated by flaw type or conditions. A brief description of the flaw and damage assessment procedures in this document is shown in Table 2.2 [5].

A series of appendices are provided which contain technical information that can be used with all sections of API RP 579, which cover FFS assessment procedures. An outline of the appendices is provided in Table 2.3 [5].

Table 2.1: Organization of each section in API 579

Section Paragraph Number	Title	Overview
1	General	The scope and overall requirements for an FFS assessment are provided
2	Applicability and limitations of the FFS assessment procedures	The applicability and limitations for each FFS assessment procedure are clearly indicated; these limitations are stated in the front of each section for quick reference
3	Data requirements	The data requirements required for the FFS assessment are clearly outlined; these data requirements include: - Original equipment design data - Maintenance and operational history - Required data/measurements for a FFS assessment - Recommendations for inspection technique and sizing requirements
4	Assessment techniques and acceptance criteria	Detailed assessment rules are provided for three levels of assessment: Level 1, Level 2, and Level 3. A discussion of these assessment levels is covered in the body of this paper
5	Remaining life evaluation	Guidelines for performing a remaining life estimate are provided for the purpose of establishing an inspection interval in conduction with the governing inspection code
6	Remediation	Guidelines are presented on methods to mitigate and/or control future damage. In many cases, changes can be made to the component or to the operating conditions to mitigate the progression of damage
7	In-service monitoring	Guidelines for monitoring damage while the component is in-service are provided, these guidelines are useful if a future damage rate cannot be estimated easily or the estimated remaining life is short. In-service monitoring is one method whereby future damage or conditions leading to future damage can be assessed or confidence in the remaining life estimate can be increased.
8	Documentation	Guidelines for documentation for an assessment are provided; the general rule is – *A practitioner should be able to repeat the analysis from the documentation without consulting an individual originally involved in the FFS assessment*
9	References	A comprehensive list of technical references used in the development of the FFS assessment procedures is provided; references to codes and standards are provided in this section
10	Tables and figures	Tables and figures including logic diagrams are used extensively in each section to clarify assessment rules and procedures
11	Example problems	A number of example problems are provided, which demonstrate the application of the FFS assessment procedures

Table 2.2: Overview of flaw and damage assessment procedures

Section in API 579	Flaw or damage mechanism	Overview
3	Brittle fracture	Assessment procedures are provided to evaluate the resistance to brittle fracture of in-service carbon and low alloy steel pressure vessels, piping, and storage tanks. Criteria are provided to evaluate normal operating, start-up, upset, and shutdown conditions.
4	General metal loss	Assessment procedures are provided to evaluate general corrosion. Thickness data used for the assessment can be either thickness readings or detailed thickness profiles. A methodology is provided to guide the practitioner to the local metal loss assessment procedures based on the type and variability of thickness data recorded during an inspection
5	Local metal loss	Assessment techniques are provided to evaluate single and networks of Local Thin Areas (LTAs), and groove-like flaws in pressurized components. Detailed thickness profiles are required for the assessment. The assessment procedures can also be utilized to evaluate blisters
6	Pitting corrosion	Assessment procedures are provided to evaluate widely scattered pitting, localized pitting, pitting which occurs within a region of local metal loss, and a region of localized metal loss located within a region of widely scattered pitting. The assessment procedures can also be utilized to evaluate a network of closely spaced blisters. The assessment procedures utilize the methodology developed for a local metal loss
7	Blisters and laminations	Assessment procedures are provided to evaluate either isolated, or networks of blisters and laminations. The assessment guidelines include provisions for blisters located at weld joints and structural discontinuities such as shell transitions, stiffening rings, and nozzles
8	Weld misalignment and shell distortions	Assessment procedures are provided to evaluate stresses resulting from geometric discontinuities in shell type structures including weld misalignment and shell distortions (e.g. out-of-roundness, bulges, and dents)
9	Crack-like flaws	Assessment procedures are provided to evaluate crack-like flaws. Recommendations for evaluating crack growth including environmental concerns are also covered
10	High temperature operation and creep	Assessment procedures are provided to determine the remaining life of a component operating in the creep regime. The remaining life procedures are limited to the initiation of a crack
11	Fire damage	Assessment procedures are provided to evaluate equipment subject to fire damage. A methodology is provided to rank and screen components for evaluation based on the heat exposure experienced during the fire. The assessment procedures of the other sections of this publication are utilized to evaluate component damage

Table 2.3: API 579 appendices

Appendix	Title	Overview
A	Thickness, MAWP and membrane stress equations for a FFS assessment	Equations for the thickness, MAWP, and membrane stress are given for most of the common pressurized components. These equations are provided to assist international practitioners who may not have access to the ASME codes and who need to determine if the local design code is similar to the ASME code for which the assessment procedures were primarily designed for
B	Stress analysis overview for a FFS assessment	Recommendations for stress analysis techniques that can be used to perform an FFS assessment are provided including guidelines for finite element analysis
C	Compendium of stress intensity factor solutions	A compendium of stress intensity factor solutions for common pressurized components (i.e. cylinders, spheres, nozzle, etc.) is given. These solutions are used for the assessment of crack-like flaws. The solutions presented represent the latest technology and have been re-derived using the finite element method in conjunction with weight functions.
D	Compendium of reference stress solutions	A compendium of reference stress solutions for common pressurized components (i.e. cylinders, spheres, nozzles, etc.) is given. These solutions are used for the assessment of crack-like flaws
E	Residual stress solutions in a FFS evaluation	Procedures to estimate the through-wall residual stress for different weld geometries are provided; this information is required for the assessment of crack-like flaws
F	Material properties for a FFS assessment	Material properties required for all FFS assessments are provided including: - Strength parameters (yield and tensile stress) - Physical properties (i.e. Young's Modulus, etc.) - Fracture toughness - Data for fatigue crack growth calculations - Fatigue curves (Initiation) - Material data for creep analysis including remaining life and creep crack growth
G	Deterioration and failure modes	An overview of the types of flaws and damage mechanisms that can occur is provided, concentrating on service-induced degradation mechanisms; API 571 is currently being developed to provide a definitive reference for damage mechanisms that can be used with API 579 and API 580
H	Validation	An overview of the studies used to validate the general and local metal loss, and the crack-like flaw assessment procedures are provided
I	Glossary of terms and definitions	Definition for common terms used throughout the sections and appendices of API 579 are given
J	Technical inquiries	Guidelines for submitting a technical inquiry to API are provided. Technical inquiries will be forwarded to the API CRE FFS task group for resolution

API RP 579 is largely self-contained so users do not have to refer to many other documents. One exception to this principle is that materials data need to be obtained from Section II of the ASME Boiler and Pressure Vessel Code. The developers of API RP 570 plan to compile data for service-exposed materials and add them to Appendix F in the future. All portions in API 579 except Section 10 and Appendix H are completed. Those two portions are being developed for inclusion in future editions [6].

FFS assessment procedures in API 579 incorporate a three-level assessment approach. The level of conservatism decreases with increasing level of assessment, but detail of analysis and data increase with increasing level of assessment. Level 1 assessment may be performed by an inspector or a plant engineer. Level 2 assessments requires at least a plant engineer, whereas Level 3 assessment must be performed by an expert engineers or by a team of engineers that includes at least one expert engineer [6]. Application of the higher levels of assessment is often limited by a lack of materials properties data and accuracy of operating data.

As for this project, the scope is narrowed down to geometric distortion. So, this project shall be closely concentrating on Section 8 – Well misalignment and shell distortion, ignoring the weld misalignment part.

2.2.1 Section 8 – Assessment of Shell Distortions

The procedures in this section can be used to assess geometric irregularities associated with shell distortions in components made up of flat plates; cylindrical, conical, and spherical shells; and formed heads. As per the document, four types of shell distortions are considered, which are general shell distortion; out-of-roundness; bulge; and dent. This project will be focusing on out-of-roundness and dent.

2.2.2 Dent

Dent is an inward or outward deviation of a cross-section of a shell member from an ideally circular geometry which is characterized by a small local radius or notch [1]. Fig. 2.1 and 2.2 shows graphical representation of a dent.

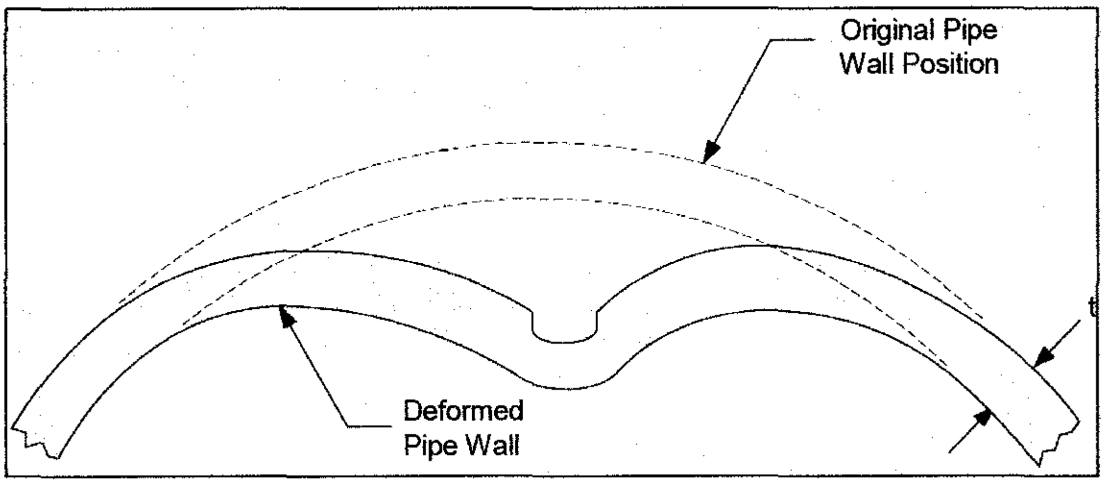


Fig. 2.1: Cross-sectional view of a pressure vessel with dent [1]

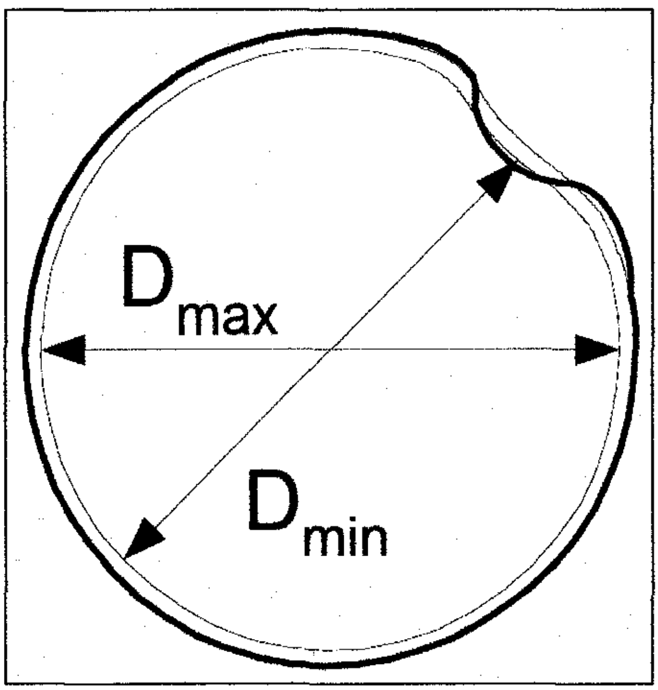


Fig. 2.2: Diameter difference of a pressure vessel with dent [1]

2.2.3 Out-of-Roundness

Out-of-roundness is interpreted as a deviation of the cross-section of a cylindrical shell and pipe bend from an ideally circular geometry. The out-of-roundness for a cylinder is assumed to be constant in the longitudinal direction and either global (oval shape) or arbitrarily shaped in the circumferential direction. Fig. 2.3 shows graphical representation of a shell with out-of-roundness.

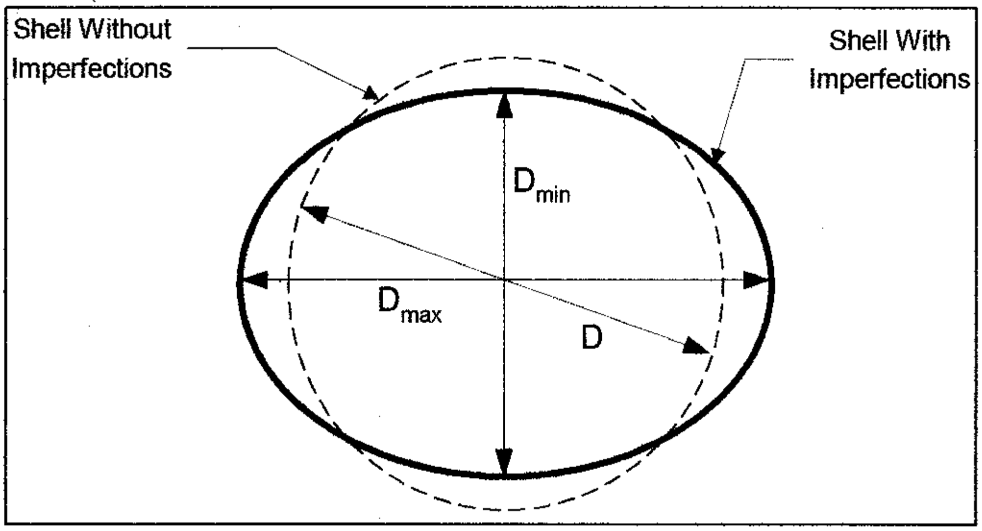


Fig. 2.3: Cross-sectional view of a cylinder with global out-of-roundness [1]

2.2.4 Overview of FFS evaluation techniques and acceptance criteria

Level 1

The Level 1 assessment procedures are anticipated to present conservative screening criteria that can be used with a minimum amount of inspection or component information. These procedures may be used by either plant inspection or engineering personnel. The Level 1 assessment is based on the fabrication tolerances of the original construction code. If the current geometry of the component is such that the original fabrication tolerances are satisfied, the Level 1 assessment criteria are satisfied, and additional analysis are not required unless the component is in cyclic service or has a dent. In this case, a Level 2 or Level 3 assessment is required.

Level 2

Level 2 assessments provide a means to estimate the structural integrity of a component with shell distortion characterized as out-of-roundness, a bulge or dent. Pressure as well as supplemental loads is considered as well as more general geometries (e.g. pipes of differing thickness). In a Level 2 assessment, inspection information similar to that required for a Level 1 assessment is required; however, more detailed calculations are used in the evaluation. Level 2 assessments are usually carried out by plant engineers or engineering specialists experienced and knowledgeable in performing FFS assessments.

Level 3

Level 3 assessments are intended for the evaluation of components with general shell distortions, complex component geometries and/or loadings. Detailed stress analysis techniques including fracture, fatigue, and numerical stress analysis are normally used in a Level 3 assessment. Significant field measurements are typically required in a Level 3 assessment to characterize the geometric irregularity. In a Level 3 assessment the most detailed inspection and component information is typically required, and the recommended analysis is based on numerical techniques such as finite element method. The Level 3 assessment procedures are primarily intended to be used by engineering specialists experienced and knowledgeable in performing FFS evaluations.

2.3 Stresses In Pressure Vessel

The study of stresses in pressure vessel is the determination of relationship between internal and external forces applied to a vessel and the corresponding stresses.

In pressure vessel, "thin wall" generally refers to a vessel having an inner-radius-wall-thickness ratio of 10 or more ($r_{in}/t \geq 10$). Vessels are referred to as membranes and the associated from the contained pressure are called membrane stresses [2].

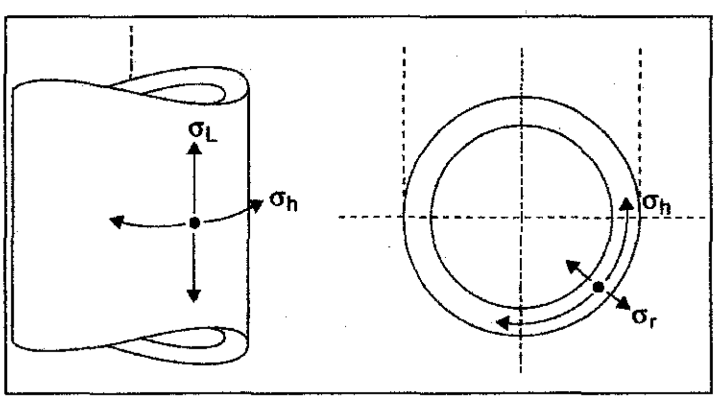


Fig. 2.4 Stresses due to internal pressure

The membrane stresses in a thin wall cylindrical shell subject to internal pressure are illustrated in Fig. 2.4, where:

σ_h = hoop (circumferential) stress

σ_l = longitudinal stress

$\sigma_{r,ID}$ = radial stress at inner diameter

$\sigma_{r,OD}$ = radial stress at outer diameter

P = internal pressure

P_e = external pressure

t = wall thickness

D = diameter

Membrane stress analysis is not completely accurate but allows certain assumptions to be made while maintaining a fair degree of accuracy. The main simplifying assumptions are that the stress is biaxial and that the stresses are uniform across the shell wall. For thin-walled vessels, these assumptions have proven themselves to be reliable. No vessel meets the criteria of being a true membrane, but it can be used within a reasonable degree of accuracy [7].

2.4 Finite Element Analysis (FEA)

During those days before computer improvement is achieved, engineers only capable of analyzing geometries that could be analyzed only by simplified methods. However, the availability of finite element analysis (FEA) software has enabled engineers to compute inelastic and elastic behavior of the vessel.

The FEA is used to determine the elastic stress distribution used in the evaluation. It involves the solution of simultaneous, algebraic equations. The algebraic equation resulted from subdividing a complex shape into many discrete, interconnected, simple shapes hence the phrase "finite elements" [8].

2.4.1 Definition of Static Analysis

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. The kinds of loading that can be applied in a static analysis include [9]:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)
- Fluences (for nuclear swelling)

ASME Boiler and Pressure Vessel Code, Section VIII, Division 1 classified stress involved in analyzing vessels components as one of three category as mentioned below [10]:

i) Primary stress

Stress that developed by imposing loading and is necessary to satisfy the law of equilibrium, for example hoop stress in a cylinder due to internal pressure. Primary stress can be divided into membrane stress and bending stress

ii) Secondary stress

Secondary stress is the stress that developed when the deformation of a component due to applied load is restrained by other components. It is also known as gross structural discontinuity. This type of stresses is a source of stress or strain intensification.

iii) Peak stress

Peak stress is a localized stress that does not cause any noticeable distortion in a component, but it may cause fatigue crack or brittle fracture.

2.5 Reviews On Past Research Works

Ted L. Anderson [5] developed an article as an overview of the published API RP 579, which covers FFS assessment procedures. The overall organization and assessment procedures in API 579 are reviewed in the article. This is followed by a more detailed discussion of the API 579 of cracks, which is covered in section 9 of the document.

R. Sheshari had written a paper titled Fitness-for-service assessment for spherical pressure vessels with local hot spots [11]. The paper develops a method for Level 2 FFS estimation of spherical shapes subjected to local hot spots where the temperatures are elevated due to local damage. In the findings, the decay length for spherical shells is determined, and the size of hot spot to be identified as local is proposed. Furthermore, a lower bound RSF for the spherical pressure vessels containing hot spots is formulated by the application of Mura's variational formulation and the m_α-multiplier method. The effectiveness of the proposed Level 2 method is evaluated and demonstrated through an example.

Tantichattanont, Adluri, and R. Seshadri [12], developed a paper that demonstrates a method for Level 2 FFS evaluation of spherical pressure vessels with localized corrosion. In this paper, they formulated lower bound RSF of spherical vessels containing corrosion damage. Three alternative design recommendations are also given. The effectiveness of the proposed methods is evaluated and demonstrated through illustrative examples and comparison with Level 3 inelastic FEA. Although theoretically a sphere would be the optimal shape of a pressure vessel, unfortunately the sphere shape is difficult to manufacture as cost is the main concern. This fact can be interpreted in the real life as most of the pressure vessel designed and manufactured in shape of cylindrical shell with either torispherical or elliptical in each end.

Shunqing Cai and Andrew J. Deeks [13], developed a paper to demonstrate an axi-symmetric approach to dynamic FEA of thin-walled cylinders that permits the inclusion of initial bi-symmetric radial distortion of the cylinder. A finite element model is developed in this paper, which uses a Fourier expansion of the displacements in the circumferential direction, while permitting an initial out-of-roundness. The performance of the simplified model is compared with full 3-D FEA, and is shown to represent the dynamic behavior of the distorted cylindrical shell.

Y. J. Kim developed a paper entitled 'Development of limit load solutions for corroded gas pipelines' [14]. In this paper, finite element simulations are carried out to derive an appropriate failure criterion by following a systematic approach.

David Heckman wrote in his paper [15], 'Finite Element Analysis of Pressure Vessels' had explored applicable methods using FEA in pressure vessel. Three models were tested, three dimensional, symmetric, and axisymmetric models with different elements used. The axisymmetric model had by far the shortest run time with relatively small computational error.

A number of research papers had been developed to do FFS based on API RP 579. However, none of them concentrate on the shell distortions in API RP 579. This is where the originality of the project took place, which primarily concerns with shell distortions type of flaw on the selected pressure vessel.

CHAPTER 3

METHODOLOGY

3.1 Project Methodology

Fig. 3.1 below shows a summary of the project work flow. Details on each level are discussed in the next subsection.

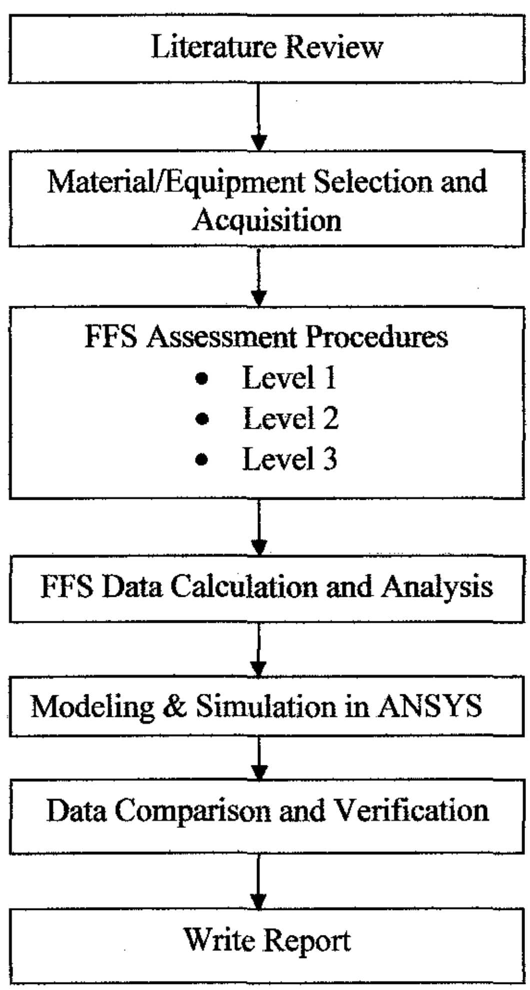


Fig. 3.1: Work Process Flow Chart

3.2 Project Activities

3.2.1 Literature Review

Literature review or preliminary research work on the topic is carried out at the beginning of the project. The study is conducted to understand the FFS procedures covered in API Recommended Practice 579 primarily in section 8.

The content of API 579 was studied generally at first hand. Then, the study is focused on Section 8 of the document. It is then decided that the investigation will be focused on pressurized vessel equipment with out-of-roundness and/or dent. The FFS assessment procedures will be conducted based on study case style.

3.2.2 Material/Equipment Selection and Acquisition

The FFS assessment procedures contained in API RP 579 are only used to evaluate pressure vessels, piping systems, and storage tanks. Thus, an object/equipment from any of those categories can be selected. As this topic concerns with the pressurized equipment, only object/equipment from pressure vessel type are being considered. The object/equipment has been selected, which is fire extinguisher.

Fig. 3.2: Fire Extinguisher

Fire Extinguisher

It had been decided that the object that will be assessed would be fire extinguisher. This decision is chosen because it is one kind of pressure vessel. Furthermore, this equipment can be found extensively in the residential college and under supervision of Maintenance Department of UTP.

To obtain the information needed regarding the fire extinguisher, the manufacturer of the equipment, Steel Recon Industries (SRI) Sdn. Bhd. is contacted to obtain permission of using their product data, and the manufacturer of the equipment had agreed and granted the technical drawing of the specified fire extinguisher. The technical drawing of the fire extinguisher is appended in **Appendix A.**

After the technical drawing is obtained, it is inspected to understand the geometric of the fire extinguisher. This will be useful in order to do the Fitness-for-Service (FFS) assessment procedures and the structural Finite Element Analysis (FEA) simulation later. The data of the fire extinguisher is collected from the specification written on the body of the equipment and also from the technical drawing of the equipment. The following table represented the specification of the fire extinguisher (See Table 4.2).

Table 3.1: Fire extinguisher specifications

No.	Item	Description/Value
1	Part No	FEC004-MS-090-NA
2	Description	9 kg
3	Material Thickness	1.5 mm
4	Finishing	Epoxy Red Powder Coating
5	Overall height (mm)	560.0
6	Cylinder diameter (mm)	176.0
7	Material Specification	Cold Rolled Steel JIS G3141 SPCD, SPCE
8	Working Pressure	1.4 MPa (362.5 psi)
9	Mininum Test Pressure	2.5 MPa (942.5 psi)
10	Type of extinguishant	ABC Powder (Ammonium Phosphate)

An interview had been conducted with Mr. Nazri [17], the owner of Hydrant Water Services Sdn. Bhd., the company responsible for supplying, installing, servicing, or refilling fire extinguishers and other fire fighting equipments in UTP.

A fire extinguisher in Malaysia normally has a service life of maximum ten years, provided that those equipments do not have any damage during its service life. It is also expected to undergo periodic inspection about once a month to make sure that equipment is in good condition and can be used. This inspection also meant to check the condition of the powder and also the internal pressure of that thin-walled shell. It is expected that the powder inside the equipment will be precipitate and compacted and thus, caused a significant decrease in internal pressure.

After it reached a life span of ten years, it will be required to undergo hydrostatic pressure test, where the equipment is tested to endure a pressure five times higher than working pressure to check if the structural integrity is still in satisfying condition. However, normally all fire extinguishers will be disposed after ten years due to many reasons such as safety, incremental service cost, etc.

There are three authorized bodies that are related with fire extinguisher. The first authorized body is the Department of Occupational Safety and Health (DOSH) of Malaysia, a department under Malaysian Ministry of Human Resources. The department is a government agency responsible for the administration and enforcement of legislations provided for any party that involved with designing, manufacturing, testing, and inspecting unfired pressure vessels, fire extinguishers in particular. The design, fabrication, testing, installation, operation, inspection and maintenance of Unfired Pressure Vessels are mainly governed by the Factories and Machinery Act 1967 and applicable regulations made under the act. One such regulation is the Factories and Machinery (Steam Boiler and Unfired Pressure Vessel) Regulation, 1970 [18].

Standards and Industrial Research Institute of Malaysia (SIRIM) is another statutory-become-incorporated authorized body that has direct relation with the studied equipment. They are responsible to set standards for any industrial equipment. Particularly for fire extinguisher, there are three standards that provide guidance for constructing, testing, installing, and maintenance for the equipment. Those standards are MS 1539: Part 1: 2002; MS 1539: Part 3: 2003; and MS 1539: Part 4: 2004 (See Table 3.2).

Table 3.2: MS for fire extinguisher

No.	MS Title	Description
1	MS 1539 : PART 1 : 2002 Construction and Testing Methodology	This Malaysian Standard specifies requirements for rechargeable and non-rechargeable metal-bodied portable fire extinguishers containing an extinguishing medium which can be expelled by the action of internal pressure.
2	MS 1539 : PART 3 : 2003 Selection and Installation	This Malaysian Standard gives requirements on the suitability and sitting of portable fire extinguishers, primarily those conforming to MS 1539: Part 1, that can be carried by one person and that are used for the protection of buildings and other premises and their contents.
3	MS 1539 : PART 4: 2004 Maintenance of Portable Fire Extinguishers	This Malaysian Standard specifies schedules for the maintenance of portable fire extinguishers, installed in all applications, to be followed by the user and the maintenance

The most important authorized body for fire extinguisher is the Fire and Rescue Department of Malaysia. This department is the main body that has the authority of the equipment as any fire fighting equipment must be approved by them. They are also responsible with the execution and synchronization of the fire extinguisher throughout the nation. For instance, any fire extinguisher that to be installed in any common building must be approved and licensed by the department. If they ever found that any party disobeys the guideline provided, they have the authority to take back the license they issued to that party.

3.2.3 FFS Assessment Procedures

The selected object/equipment then will be evaluated using FFS Assessment Procedures contained in API RP 579. These procedures are divided into three stages; Level 1; Level 2; and Level 3.

Level 1

Level 1 assessment procedures are based on the fabrication tolerances provided in the original construction code. An overview of these tolerances is provided for those construction codes in Tables 8.3 through Table 8.7 in API 579.

In some cases, these criteria are not completely defined by the original construction code and are dependent on the original design specification of the owner-user. In addition, the Level 1 assessment procedures should not be used if the component is in cyclic service.

As for this project, the scope is narrowed down to pressure vessel equipment with out-of-roundness in terms of dent. The data needed for Level 1 assessment is provided in Table 3.3.

Table 3.3: Data required for Level 1 assessment

No.	Item	Description
1	D_{max}	Maximum measured diameter
2	D_{min}	Minimum measured diameter
3	D	Nominal diameter

FFS Level 1 assessment for this project is based Table 8.3 in Section 8 of API 579, as attached in **Appendix B**. Calculation is done to find the correspond diameter difference that will be the limit of the fire extinguisher to fail FFS Level 1 assessment.

Level 2

For Level 2 assessment, the calculation is done to find the corresponding diameter difference that is considered the limit criteria for the equipment to pass FFS Level 2 assessment.

A few steps need to be taken before the computational procedure can be executed. The specification data regarding the fire extinguisher must be obtained first. All the parameter used in Level 1 assessment as in Table 3.3 will be used again.

Level 2 assessment involved several steps. For Step 1, a set of parameter is determined first before calculation can be done. Table 3.4 shows the list of parameter settings for FFS Level 2 assessment.

Table 3.4: Parameters settings for FFS Level 2 assessment

No.	Symbol	Description	Value	Unit
1	P	Internal pressure	1.4	MPa
2	E	Weld joint efficiency	1	dimensionless
3	R	Internal radius of the cylinder	86.5	mm
4	FCA	Future Corrosion Allowance	0.1	mm
5	t	Wall thickness	1.5	mm
6	C_s	Factor to account for the severity of the out-of-roundness	0.1	dimensionless
7	v	Poisson's Ratio	0.3	dimensionless
8	E_y	Young's modulus	200,000	MPa
9	D_m	Mean diameter	174.5	mm
10	H_f	Factor dependent on whether the induced stress from the shape deviation is categorized as a primary or secondary stress	3	dimensionless
11	S_a	Allowable Stress	102	MPa

In Step 2, the membrane stress of the equipment is determined based on the equation from Appendix A of API 579, as in equation (1) below (See **Appendix C**).

$$\sigma_m = \frac{P}{E}\left(\frac{R_c}{t - FCA} + 0.6\right) \text{---} (1)$$

In Step 3, the quantity of R_b, ratio of the induced bending stress to the applied membrane stress resulting from pressure loads is calculated using equation (2) below.

$$R_b^{or} = abs\left[\frac{1.5(D_{max} - D_{min})\cos 2\theta}{(t-FCA)\left(1+C_s\frac{P(1-v^2)}{E_y}\left(\frac{D_m}{t-FCA}\right)^3\right)}\right] --- (2)$$

In Step 4, the value of Remaining Strength Factor (RSF) is calculated. RSF is defined as the ratio of the limit or plastic collapse load of the damaged component to the undamaged component. The equation used as in equation (3).

$$RSF = \min\left[\frac{H_f S_a}{\sigma_m(1+R_b)}, 1.0\right] --- (3)$$

The final step in FFS Level 2 assessment is to compare the RSF of the equipment, obtained from equation (3), to the allowable remaining strength factor, RSF_a.

$$RSF \geq RSF_a --- (4)$$

Value of 0.90 is selected for RSF_a, as it is the recommended value for equipment in process services. This value has been shown to be conservative [1]. The extracted steps of this Level 2 assessment are included in **Appendix D.**

Level 3

Level 3 assessment can be performed where Level 1 and 2 methods do not apply, such as for the following conditions:

a. The components normal operating or design temperature exceeds the limitations in paragraph 3.3.2.c in API 579.
b. The geometric irregularity is classified as general shell distortion.
c. The geometric irregularity occurs in a component with a complicated geometry or at a major structural discontinuity (e.g. knuckle region of torispherical heads, toriconical heads and conical transitions, or stiffening rings on a cylindrical shell).
d. More complicated loading conditions are involved which result in significant stress gradients at the location of the geometric irregularity.
e. The region of the component containing the geometric irregularity contains a flaw, see paragraph 3.3.2.f in API 579.
f. The component is subject to a loading condition that results in compressive stresses where structural stability is a concern; note that Level 2 Assessment procedures are provided for cylindrical and conical shell subject to external pressure. However, the Level 2 assessment rules are not applicable to cylinders subject to external pressure in combination with supplemental loads which result in significant longitudinal compressive stresses.

According to API 579, Level 3 assessment can be done with stress analysis techniques. So, this project is further expanded the evaluation of the equipment for FFS Level 3 assessment by running simulation using ANSYS.

3.2.4 Computer Simulation using ANSYS

All data calculated and analyzed from the FFS Assessment procedures are applied into computer simulation. The computer software used is ANSYS Version 9.0. FFS assessment procedures done on the fire extinguisher that calculated using Microsoft Excel before then is compared and evaluated with structural analysis simulation of Finite Element Analysis (FEA) methodology. The expected results will be the difference in terms of stress profile.

Under the effect of internal pressure, the variation in stress profile especially at the flawed diameter due to dent is determined by using computer software. Finite Element Analysis (FEA) performed using ANSYS software will provide acceptable accuracy of stress profile at the area of the pressure vessel with dent.

The simplified solid model of the fire extinguisher is constructed in ANSYS and further simulated and analyzed in ANSYS software to be further compared and evaluated using Finite Element Analysis method.

Stages in Finite Element Analysis (FEA)

Finite-element solution in ANSYS is divided into the following three stages.

i) Preprocessing: defining the problem

The major steps in preprocessing are (i) define keypoints/lines/areas/volumes; (ii) define element type and material/geometric properties; and (iii) mesh lines/areas/volumes as required.

The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, axisymmetric, and 3D.

ii) Solution: assigning loads, constraints, and solving

Here, it is necessary to specify the loads (point or pressure), constraints (translational and rotational), and finally the resulting set of equations is solved.

iii) Post-processing: further processing and viewing of the results

In this stage results can be viewed and plotted including (i) lists of nodal displacements; (ii) element forces and moments; (iii) deflection plots; and (iv) stress contour diagrams or temperature maps.

3.2.5 Data Analysis and Verification

After all work had been completed, the data obtained from FFS assessment procedures and the simulated is analyzed by comparing the simulation result with the FFS assessment results. This is to evaluate the acceptance criteria of FFS assessment by comparing it with Finite Element Analysis results.

3.3 Detail Approach of ANSYS Simulation

The FEA will be performed using the ANSYS software version 9.0. The specific of the finite element model developed is discussed as below:

3.3.1 Problem Description

ANSYS simulation was run to evaluate criteria used as failure limit for Fitness-for-Service (FFS) Level 2 assessment. This also serves as Level 3 assessment for the equipment. Table 3.5 shows the parameters used in the simulation.

The fire extinguisher is modeled first according to standard specification obtained from the manufacturer. Fig. 3.3 and 3.4 shows the model representation in two-dimensional and three-dimensional.

Table 3.5: Parameters used in ANSYS Simulation

No.	Parameter	Value
1	Element type	SOLID95
2	Young's modulus	200,000 MPa
3	Internal pressure	1.4 MPa
4	Poisson's ratio	0.3
5	Inner radius	86.5 mm
6	Outer radius	88.0 mm
7	Wall thickness	1.5 mm
8	Shell length	75.0 mm

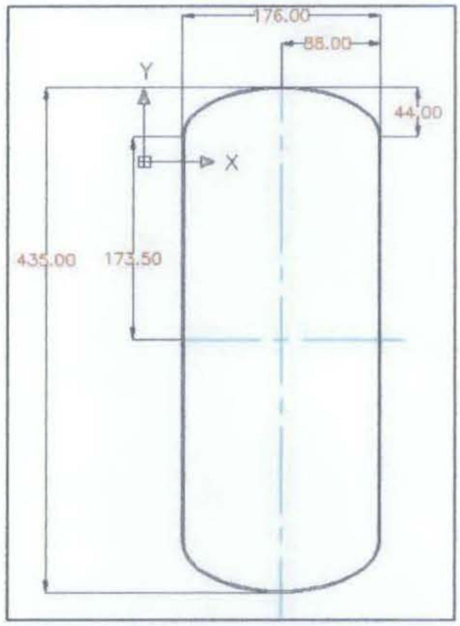


Fig. 3.3: 2-D model of fire extinguisher with dimension in mm

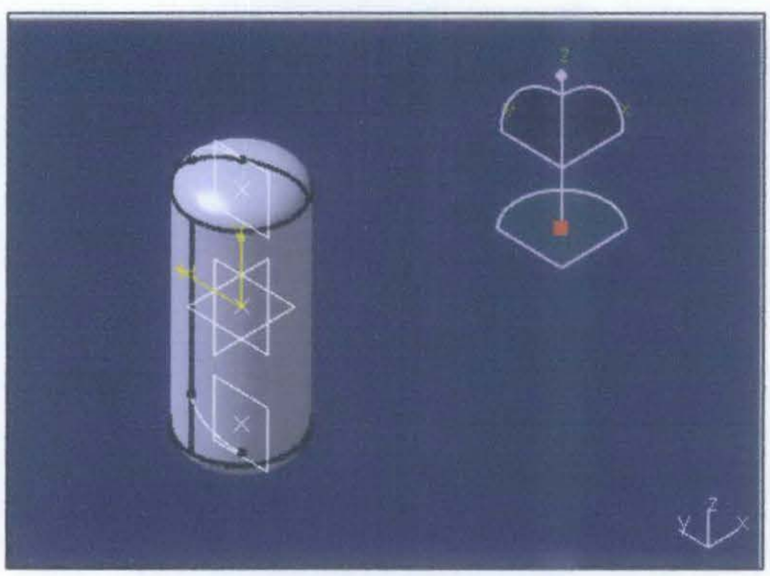


Fig. 3.4: 3-D model of fire extinguisher

3.3.2 Model Generation

The finite element model was generated in ANSYS itself. By considering symmetry, the model generated is only a quarter of a full fire extinguisher shell. In the end, the shell generated is less than quarter in longitudinal direction, which is supposed to be 173.5 mm, where 75 mm in length is used instead. This is a simplified approach, but will still gives accurate representation of the local membrane stresses.

Four models were generated. The first model is a quarter cylindrical shells with no dent effect. This model acted as standard control for the simulation. The other three models were also a quarter cylindrical shell, but with added spherical dent effect with different dimensions. The table below indicates the models generated and its descriptions.

The finite element model consists of three volumes, which are the main quarter of fire extinguisher, the quarter spherical representing the dent are, and another volume that is an overlap between the dent volume and the main quarter of fire extinguisher cylinder.

Table 3.6: Models generated and its description

Model No.	Dent diameter (mm)
1 (Control model)	-
2	5
3	6
4	7

To obtain the model as described above, two volumes are generated, which are the main quarter of fire extinguisher and the dent. For the outer corner where the main volume and dent volume is overlapping, the edge is filleted to reduce stress concentration. Those two volumes generated before are then overlapped and any unwanted, redundant volume is deleted. The total volume now became three.

These three volumes were then glued together to generate new volume that redefines the input volumes so that they share areas along their common boundaries. This new volume encompasses the same geometry as the original volumes. The following figures shows the control volume generated and also the model with dent effect.

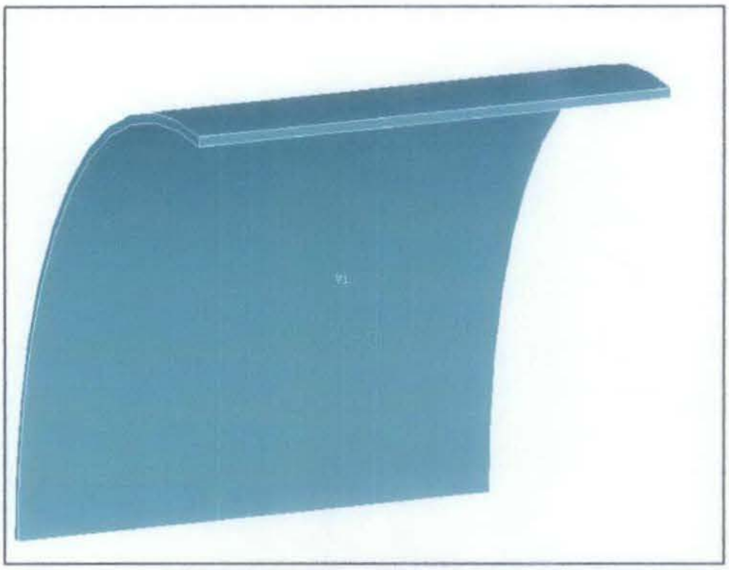


Fig. 3.5: Model generation of a quarter cylindrical shell

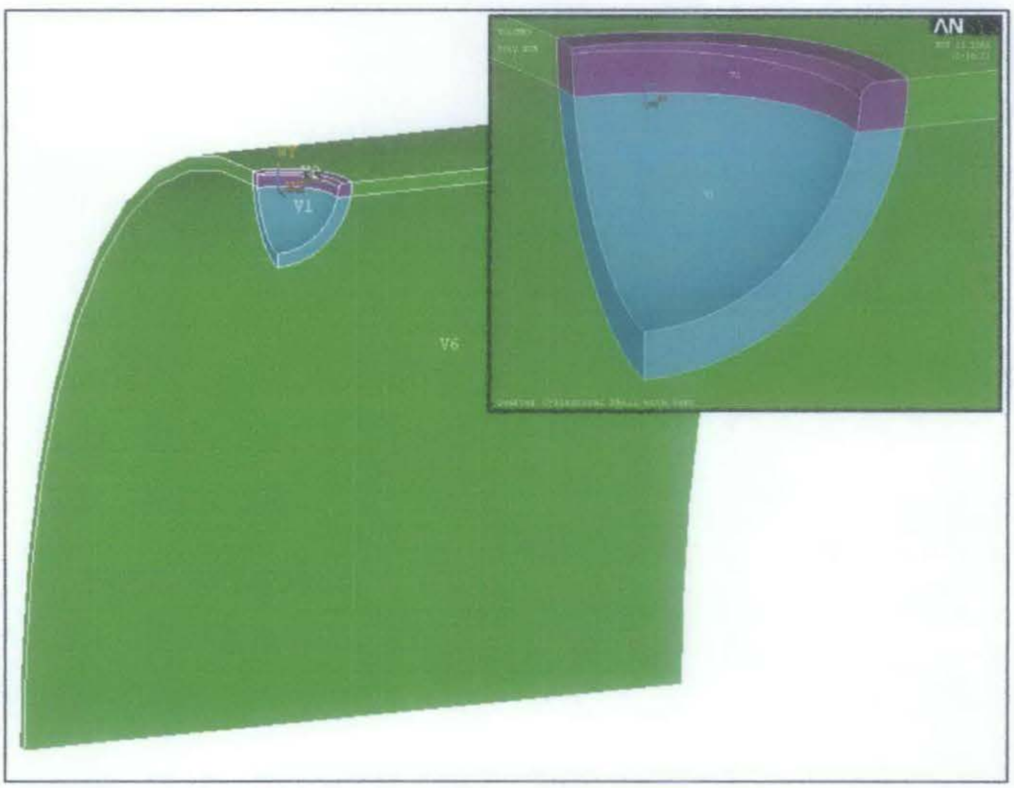


Fig. 3.6: Model generation of a quarter cylindrical shell with dent effect

Element Type

The model that generated before is then defined with quadrilateral elements and assumed to be linearly elastic. 20-node brick shaped element, which is SOLID95 in ANSYS designation, is preferable to be used throughout the entire model.

SOLID95 is a higher order version of the three-dimensional, eight-node, solid element SOLID45. It can tolerate irregular shapes without as much loss of accuracy. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation. SOLID95 has plasticity, creep, stress stiffening, large deflection, and large strain capabilities [9].

Meshing

Fine mesh will certainly yield more accurate results. Regions where stresses or strains vary rapidly, which are the regions of pressure vessel wall with dent effect, require a relatively finer mesh than overall pressure vessel regions where stresses or strains are nearly constant.

Initial trial of the analysis is done by meshing the element at the maximum fine element possible with free meshing. However, it is found that it is better to use mapped meshing where element size at each line is defined earlier. In addition to that, the smart meshing command will be resulting in the model meshed in triangular facets. There is also a possibility of failure to mesh to the model. So then it is decided to mesh the model using brick mapped meshing and volume sweeping, where possible.

Any line that represents the thickness of the model is divided into four divisions. Any other line is set to default with size of five per element. The main volume of quarter cylinder of the fire extinguisher is then meshed using mapped brick element. This resulted with the volume having 2520 elements and 12777 nodes.

For the dent volume, all lines that were represented with quarter circle with partial annulus are divided into 15 divisions. The volume is the meshed using volume swapping technique. This resulted with the meshed volume having elements and nodes of 540 and 2951, respectively. It can be observed that the dent volume had larger number of elements and nodes compared to the main volume, relatively bigger in the size compared to the dent volume.

For the volume that is an overlap between main volume and the dent, the volume sweeping technique is applied again. Any line that represents the quarter circle with partial annulus is divided into 15 divisions. This resulted in with the meshed volume having elements and nodes of 195 and 772, respectively. The number of elements and nodes for each volume was summarized in the Table 3.7. The following figures indicate models that had been meshed.

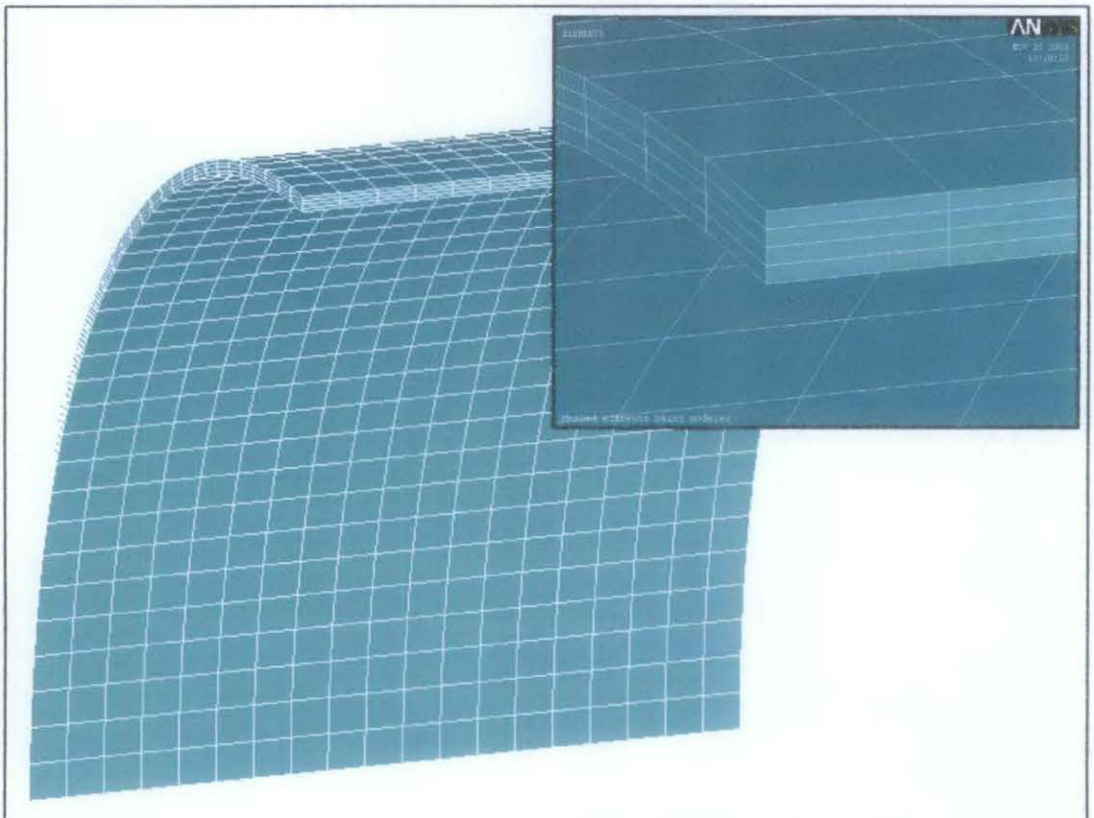


Fig. 3.7: Quarter cylindrical shell with meshed elements

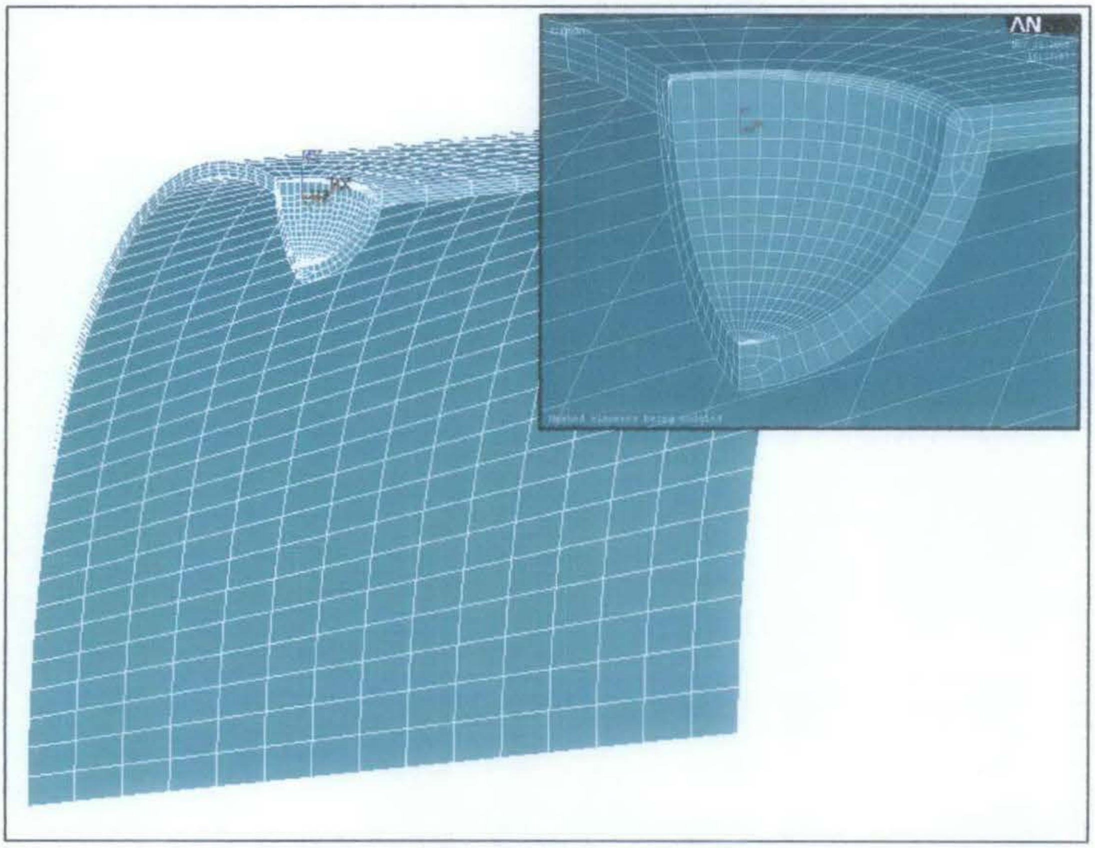


Fig. 3.8: Dented model with meshed elements

Table 3.7: Number of elements and nodes for each volume

No.	Volume	Elements	Nodes
1	Quarter Shell	2520	12777
2	Dent	540	2951
3	Shell and Dent Overlap	195	772
	TOTAL	3255	16500

Meshing is quite tricky when dealing with irregular surface in dented areas. Areas at the irregular corner of the model need to be concatenated before the mapped meshing can be done on the model.

Loading and Boundary Condition

Only one loading is considered in this analysis which is the 'live' load, the internal pressure or distributed loading. The internal pressure is applied at the inner surface of the finite element model. This involves two surfaces in the inner cylinder, which are the inner surface of the main volume and the dent.

Symmetric boundary condition is set at three surfaces of the pressure vessel model. Concatenated area that is created before to ease the process of meshing the main volume is desirably to be deleted first to avoid confusion during setting the symmetry boundary condition of the model. In addition to that, the concatenated areas only useful during meshing process only.

Since the pressure vessel had elliptical head at both ends prior to the analysis, the corresponding axial stress was applied at one end of finite element model. This represents the longitudinal or axial stress experienced by the model. The value used is the theoretical value that is hand-calculated before.

Fig. 3.9 indicates the setting of the loading and boundary conditions applied on the model. Pink arrow represents the symmetry boundary conditions applied at three surface of the model. Blue arrow represents axial stress applied at one surface of the model.

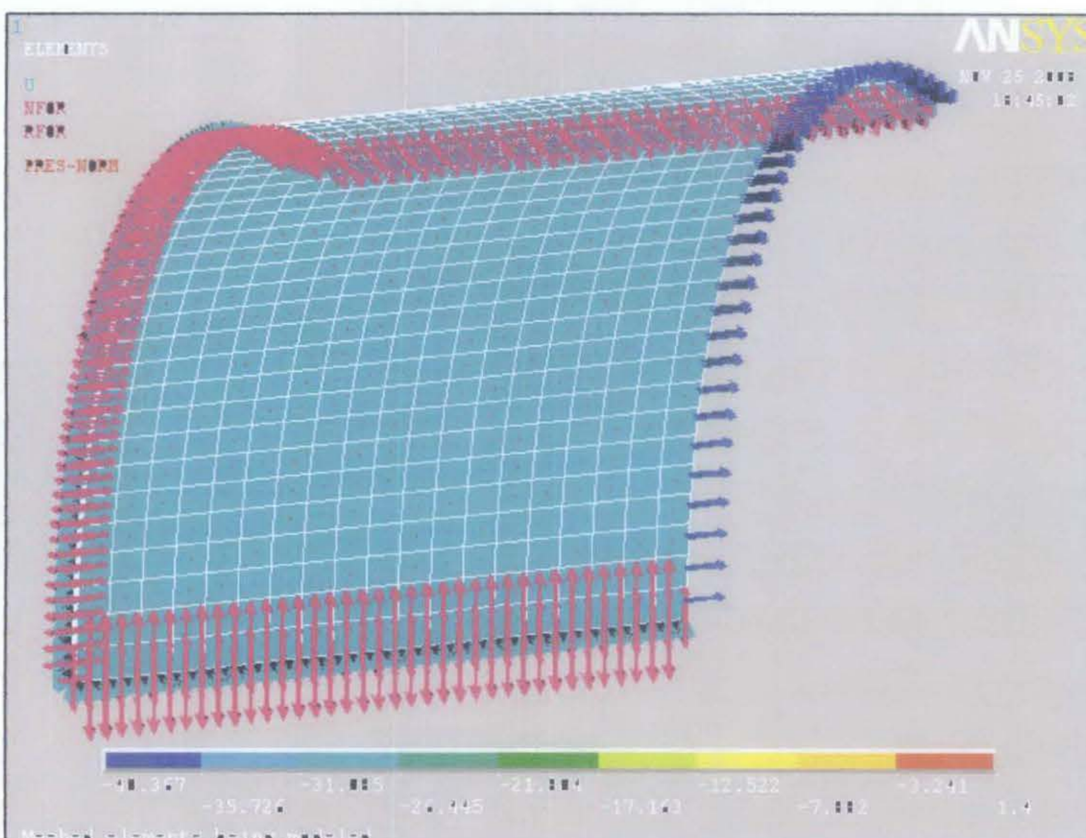


Fig. 3.9: Loadings & Boundary Conditions Setting

3.4 Gantt Chart

Gantt chart for first and second semester of this project is attached in the **Appendix E** and **Appendix F**, respectively.

3.5 Tools/Equipment Required

3.5.1 ANSYS Software

3.5.2 Microsoft Office Excel

CHAPTER 4
RESULTS & DISCUSSION

This section presents the findings and outcome of what had been investigated and explored throughout the project since the development of the project until its completion.

4.1 Fitness-for-Service (FFS) Assessment Results

In this section, results of FFS assessment procedures for a fire extinguisher with dent are presented based on a case study. The assessment consists of two levels, Level 1 and Level 2, respectively.

4.1.1 Level 1

Based on **Appendix B** (Table 8.3 of API 579), the difference between maximum and minimum measured internal diameter must not exceed 1% of nominal internal diameter. This requirement is taken from ASME Boiler & Pressure Vessel (B&PV) Code, Section VIII, Division 1 and Division 2. The code reference is UG-80(a) {AF-130.1}.

Based on this requirement, the limit of the equipment which it starts to failure is calculated. In the case of the selected equipment which has a diameter of 176.0 mm, the difference between maximum and minimum measured diameter must not exceed 1.76 mm, such as expressed in equation (5).

$$(D_{max} - D_{min}) \leq 1.76mm --- (5)$$

4.1.2 Level 2

As discussed in Chapter 3, the FFS assessment done on the fire extinguisher by doing calculations using Microsoft Office Excel 2003. A sample of calculation worksheet is enclosed in **Appendix G**. Table 4.1 summarized the results of the Level 1 and Level 2 FFS assessment calculation procedures.

Table 4.1: Fitness-for-Service Calculation Results

Diameter Difference (mm)	Diameter Difference (%)	RSF	Remarks
0.00	0.00	3.50	Passed Level 1 assessment
1.00	0.58	2.37	Passed Level 1 assessment
1.73	**1.00**	**1.91**	**Level 1 Limit**
2.00	1.16	1.79	Passed Level 2 assessment
3.00	1.73	1.43	Passed Level 2 assessment
4.00	2.31	1.20	Passed Level 2 assessment
5.00	2.89	1.03	Passed Level 2 assessment
6.00	**3.47**	**0.90**	**Level 2 Limit**
7.00	4.05	0.80	Failed Level 2 assessment

The basis used in this Level 2 FFS assessment calculation is that to find the diameter difference of the shell where the corresponding Remaining Strength Factor (RSF) of the fire extinguisher is equal to RSF_a, which is 0.9.

From Table 4.1 above, it can be observed that the limit for Level 2 assessment is when the diameter difference of the fire extinguisher reached 6.00 mm. Based on the summarized calculation results above, a graph was plotted to find the relationship between those parameters.

Material properties used for calculation was obtained from MatWeb [21], a website that provides material properties for engineering calculation. The sample of particular material properties for this calculation is included in **Appendix H.**

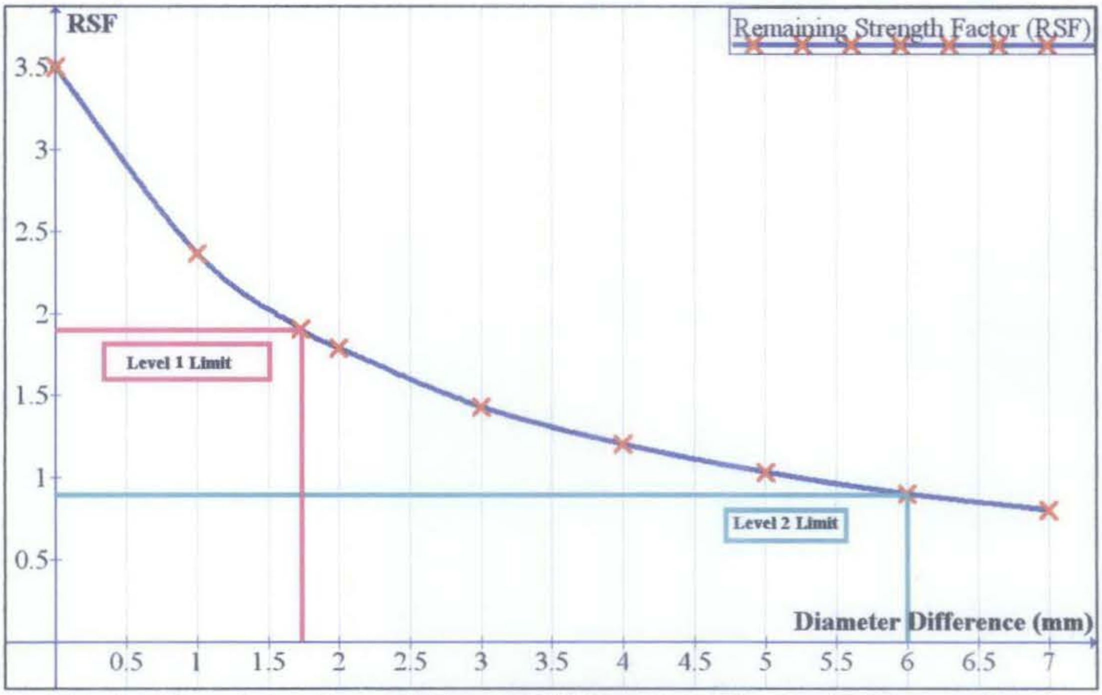


Fig. 4.1: Remaining Strength Factor (RSF) against Diameter Difference (mm)

Fig. 4.1 above shows the relationship between the Remaining Strength Factor (RSF) and the diameter difference. From Fig. 4.1, it can be observed that RSF of the equipment decreased exponentially as the difference of diameter increased. The pink and green lines in the figure show the Level 1 and Level 2 Limit, respectively. Obviously, the percentage difference of the diameter is proportional to the diameter difference.

The results obtained from FFS Level 2 assessment were then applied to run the structural analysis simulation of the using ANSYS. Three diameter differences were set to be analyzed during ANSYS simulation, which are 5, 6, and 7 millimeters. These variations of diameter differences are the located at lower, middle, and upper of the limit of the FFS Level 2 assessment, respectively.

4.2 ANSYS Simulation

As mentioned in previous section, ANSYS simulation was run to evaluate and compare the results obtained from calculation in Fitness-for-Service (FFS) Level 2 assessment. This also represented as FFS Level 3 assessment for the equipment.

4.2.1 Assumptions

A few assumptions were determined before conducting the analysis. Firstly, the wall thickness of the fire extinguisher was assumed to be uniform all the across the shell. By this simplification, the fire extinguisher can be modeled without encountering any difficulty.

The internal pressure of the fire extinguisher is then assumed to act uniformly onto the inner wall of the fire extinguisher. Also, the shell elements are assumed to be linearly isotropic.

4.2.2 Results

Four different models developed in the simulation process. The first cylinder was modeled without any deformation. This model served as the standard condition for the cylinder and was used to verify other model later. The other three models were built with a spherical-shaped dent effect placed at the centre of the cylindrical shell with dimensions 5, 6, and 7 millimeters, respectively. The following table and figures shows the results obtained from the simulation

Table 4.2: Results from ANSYS Simulation

No.	Model	Maximum Von Mises Stress, σ_{vm}
1	no dent	71.77 MPa
2	5 mm dent	230.83 MPa
3	6 mm dent	241.01 MPa
4	7 mm dent	250.46 MPa

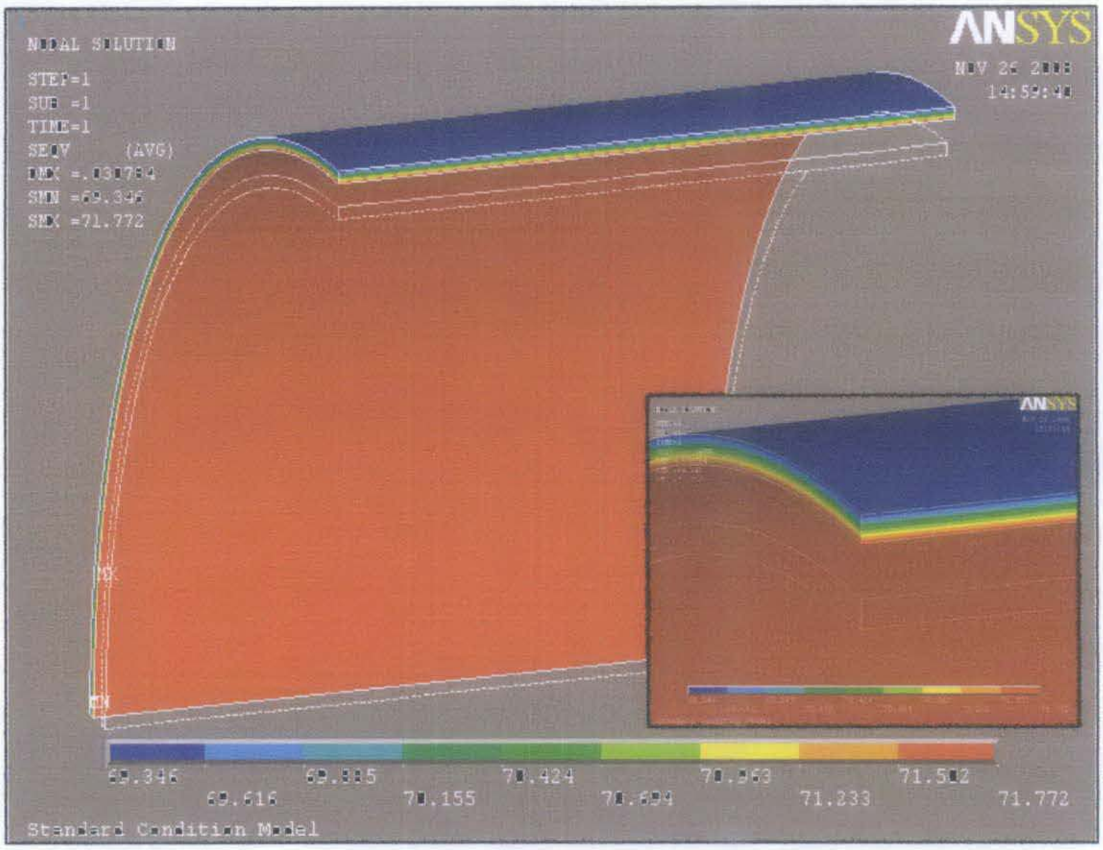


Fig. 4.2: Von Mises stress contour in standard condition model

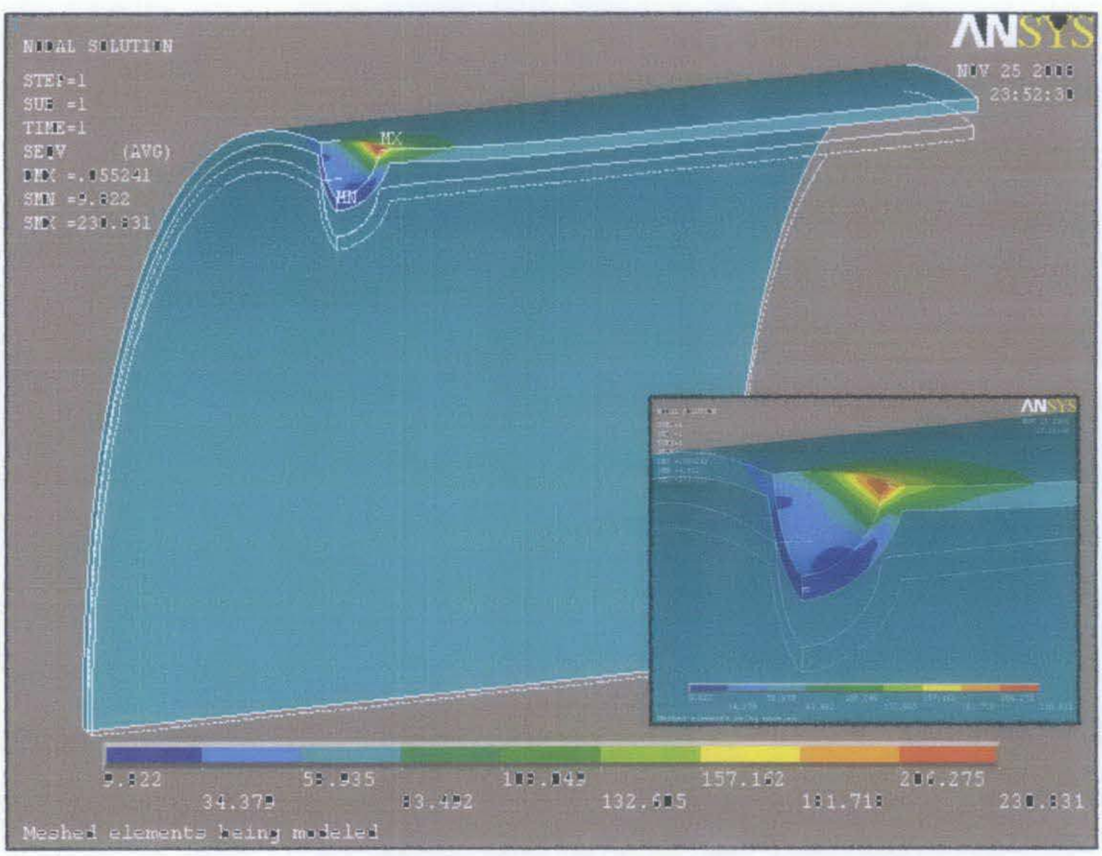


Fig. 4.3: Von Mises stress contour in 5-mm-dent model

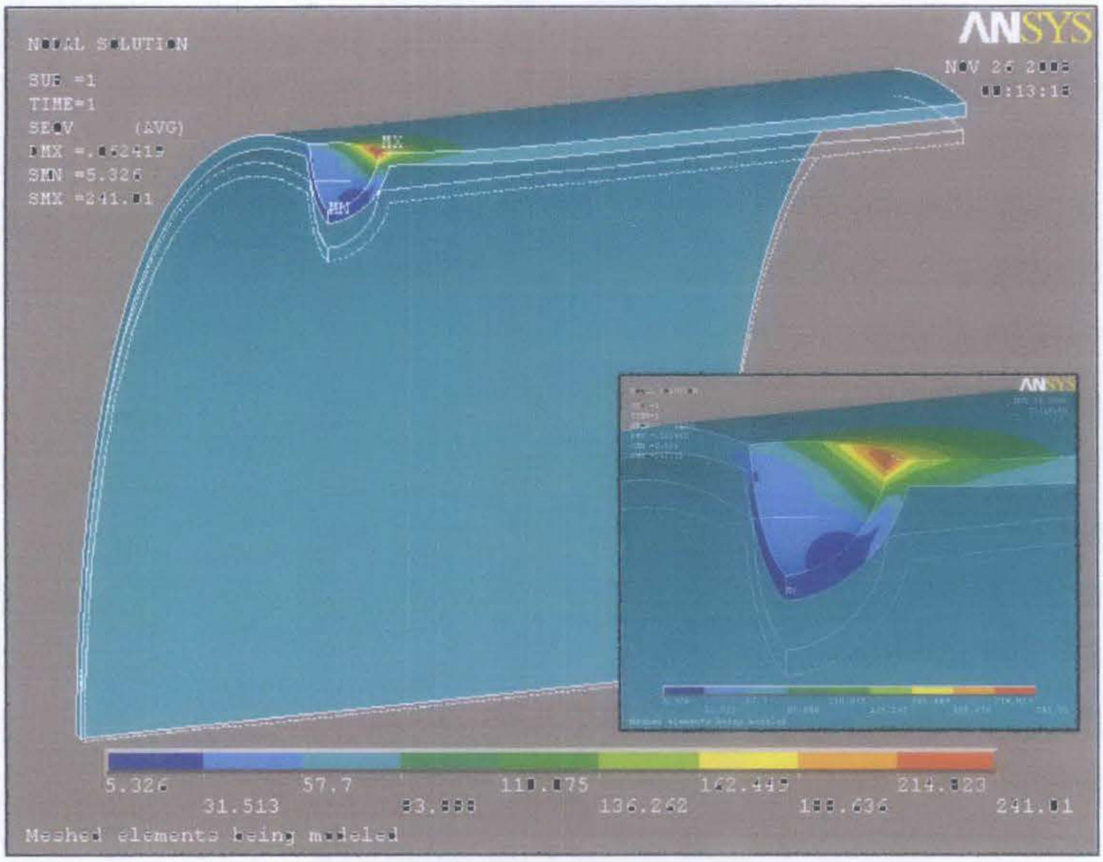


Fig. 4.4: Von Mises stress contour in 6-mm-dent model

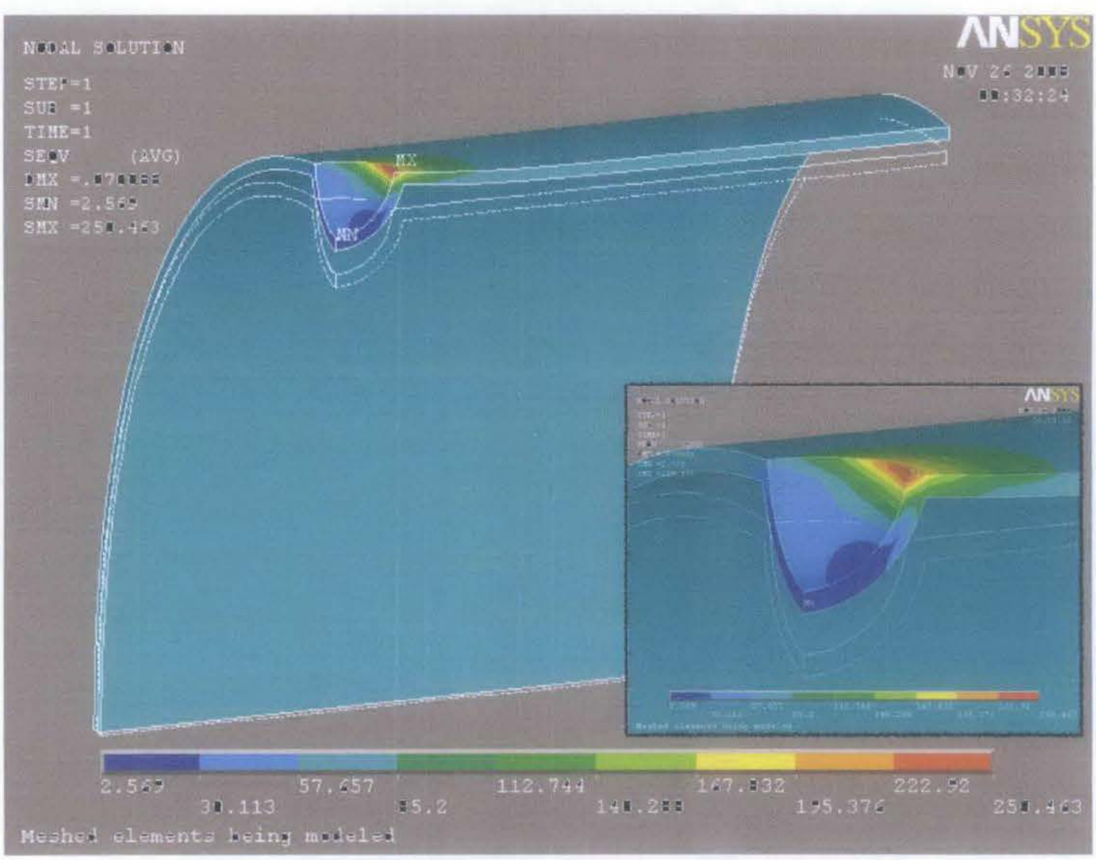


Fig. 4.5: Von Mises stress contour in 7-mm-dent model

Fig. 4.2 – 4.5 indicates Von Mises stress (σ_{vm}) contour profile for each model, respectively. Von Mises stress profile in standard condition model shows uniform distribution, with inner wall experience larger magnitude compared to outer surface. This is because inner wall experience higher stress exerted by internal pressure of the vessel compared to outer surface that only experience atmospheric pressure which is negligible in this case.

For quarter cylindrical models with dent effect, it can be observed that the contour profile of each model shows almost the same pattern, with no significant difference with each other. However, the magnitude of maximum von Mises stress increases as the diameter difference increases.

As the model applied with dent effect, the location of maximum von Mises stress also changed. For standard condition model, the maximum von Mises stress was observed to be in uniform at the inner surface of the shell. As for models with dent effect, the maximum von Mises were observed at the outer surface in longitudinal direction. This is due to the hoop stress that is generally higher than longitudinal stress. So, this region is most probably the region where crack will initiate and propagate in longitudinal direction.

The von Mises stress values at the critical area were reviewed in comparison with reference stresses. Reference stresses were set to yield strength, σ_y, and ultimate tensile strength, σ_{uts}. Failure was then assumed to occur when the von Mises stress distribution across the critical area reached the reference stress. Table below shows the ratio of maximum von Mises stress over yield strength and ultimate tensile strength, respectively

Table 4.3: Ratio of maximum σ_{vm} with σ_y and σ_{uts}

Model	**Max σ_{vm}**	**σ_y**	**σ_{uts}**	**σ_{vm}/σ_y**	**σ_{vm}/σ_{uts}**
No dent (control model)	71.77 MPa	240 MPa	358 MPa	0.30	0.20
5-mm-radius dent	230.83 MPa			0.96	0.64
6-mm-radius dent	241.01 MPa			1.00	0.67
7-mm-radius dent	250.46 MPa			1.04	0.70

From Table 4.3, it can be observed that model with effect of 6mm dent is equal to yield strength of the material. This means that the equipment starts to enter plastic deformation region if the fire extinguisher is subjected to have a change of diameter more than 6mm from its nominal diameter.

The FFS assessment calculation and simulation was then further expanded by applying test pressure of 2.5 MPa on the fire extinguisher, with other parameters were maintained. The results are presented in Table 4.4.

Table 4.4: Results from simulation of the same model with test pressure

Model	**Max σ_{vm}**	**σ_{vm}/σ_y**	**σ_{vm}/σ_{uts}**
No dent (control model)	131.88 Mpa	0.55	0.37
5-mm-radius dent	433.80 MPa	1.80	1.21
6-mm-radius dent	453.45 MPa	1.89	1.27
7-mm-radius dent	471.37 MPa	1.96	1.32

Results from further simulation reveals that the maximum von Mises stress for fire extinguishers with dent effect exceeded the ultimate tensile strength of the material. This means that the fire extinguisher would be rupture and fail when exerted with internal pressure equals test pressure. This is dangerous especially when the fire extinguisher is subjected to structural integrity tests such as hydrostatic test.

There are two possible yield criterions which can be used in discussing the load acting on the fire extinguisher. The first one is Tresca yield criterion, which stated that plastic flow starts when the maximum shear reaches a critical value k (the shear flow stress of the material). The second criterion is Von Mises which state that plastic flow depends on a combination of shear stresses (independent of the coordinate system) [19].

Throughout the analysis, Von Mises criterion is considered rather than Tresca for some reasons. For some ductile materials, it has been shown experimentally that Von Mises criterion predict failure better than Tresca. Furthermore, Von Mises gives better prediction of failure than Tresca [19].

The next following tables and figure summarized all the results obtained from ANSYS simulation.

Table 4.5: Summarized results for working pressure condition, P = 1.4 MPa

Model	σ_{vm}/σ_y	σ_{vm}/σ_{uts}	RSF	Remarks
No dent (Control model)	0.30	0.20	3.5	Passed Level 2 assessment
5-mm-radius dent	0.96	0.64	1.03	Passed Level 2 Assessment
6-mm-radius dent	1.00	0.67	0.90	Level 2 Limit
7-mm-radius dent	1.04	0.70	0.80	Failed Level 2 Assessment

Table 4.6: Summarized results for test pressure condition, P = 2.5 MPa

Model	σ_{vm}/σ_y	σ_{vm}/σ_{uts}	RSF	Remarks
No dent (Control model)	0.55	0.37	1.96	Passed Level 2 Assessment
5-mm-radius dent	1.80	1.21	0.73	Failed Level 2 Assessment
6-mm-radius dent	1.89	1.27	0.65	Failed Level 2 Assessment
7-mm-radius dent	1.96	1.32	0.59	Failed Level 2 Assessment

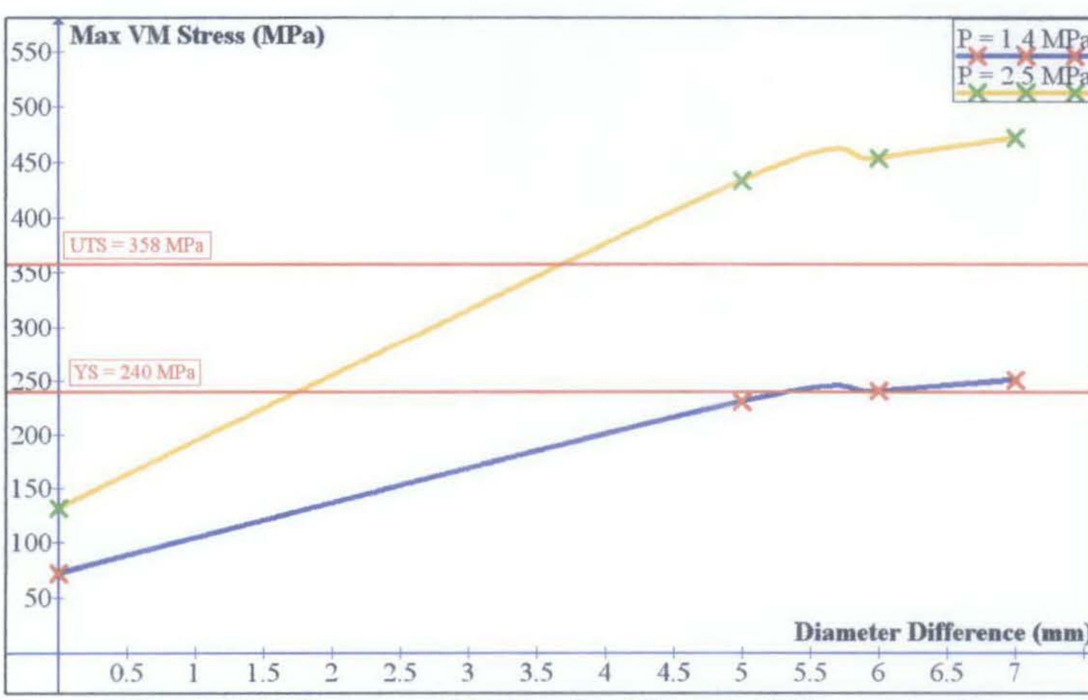


Fig. 4.6: Maximum σ_{vm} (MPa) against Diameter Difference (mm)

From the table and figure above, it can be observed that diameter difference of 6mm, which appears to be the limit criteria for FFS Level 2 assessment, also aligned with reference stress of Yield Strength (YS), when applied with working pressure value. When the model is applied with pressure equals to test pressure, the resulting von Mises stress exceeded Ultimate Tensile Strength (UTS), which means the shells probably cracked or burst due to high internal pressure applied.

For industrial application, FFS assessments procedures shall be used to determine the rerate, repair, or replace decision. The results obtained from this project can be used as a reference source on how to conduct FFS assessment for pressure vessel with geometric distortion.

CHAPTER 5
CONCLUSION & RECOMMENDATION

5.1 Conclusion

In this project, a systematic approach was followed to conduct a Fitness-for-Service (FFS) assessment and evaluate the acceptance criteria by running a simulation in ANSYS, and the resulting conclusions are as follows:

- Level 1 FFS assessment for a fire extinguisher with out-of-roundness in terms of dent is based on the fabrication tolerances of ASME B&PV Code, Section VIII, Division 1 and 2. The limit for Level 1 assessment for the fire extinguisher is that the diameter difference of the diameter must not exceed 1.73mm.
- Level 2 FFS assessment includes a more detailed computational procedure, where Remaining Strength Factor (RSF) should not be less than 0.9. Based on the calculation, the diameter difference of 6mm equals to the RSF of 0.9.
- Finite Element Analysis (FEA) was run using ANSYS simulation where the diameter difference of 6mm had the maximum von Mises stress equals to yield strength of the material.
- The FFS assessment procedures became more sophisticated and more data are required as the level of assessment increases. This confirmed that the FFS assessment become less conservative as the level of assessment is higher.

5.2 Recommendation

For further study and investigation, it is recommended to further conduct Fitness-for-Service (FFS) assessment by conducting experimental procedure to further evaluate the criteria used in FFS assessment.

It is also recommended to modify the geometry of the dent modeled in ANSYS in this project before. Sharp corners should be avoided and further modified to avoid stress concentration. The effect of having different geometries for dent such as spherical, rectangular, elliptical, etc should also be further investigated. The input listing for ANSYS application is provided in **Appendix J** to assist future researcher.

For a wider perspective of view, a further investigation can be done to conduct FFS assessment with different equipments such as storage tanks and pipelines, or different flaw of mechanisms such as corrosion, blister, weld misalignment, etc.

REFERENCES

1. API. Recommended practice for fitness-for-service. API 579. Washington, DC: American Petroleum Institute, 2000.
2. George Antaki 2005, *Fitness-for-Service and Integrity of Piping, Vessels, and Tanks,* New York, McGraw-Hill.
3. Fitness4Service.com < http://www.fitness4service.com/#>
4. David R. Thornton, PE <http://www.carmagen.com/newsletter/news29.htm>
5. Ted L. Anderson, David A. Osage, 2000, "API 579: a comprehensive fitness-for-service guide", *International Journal of Pressure Vessels and Piping 77 (2000)*: 953-963.
6. Carl E. Jaske, 2001, "*Process Equipment Fitness-for-Service Assessments Using API RP 579*", *Process & Power Plant Reliability Conference (Nov 2001)*: 43-56.
7. Moss, Dennis R, *Pressure Vessel Design Manual*, Gulf Publishing Co., Texas, 1997.
8. <www.ite.com>, Integrated Technologies Engineering, *White Paper: Practical Approaches to Engineering Analysis.*
9. ANSYS 9.0 Help File, *Structural Analysis.*
10. ASME Boiler and Pressure Vessel Committee, *Section VIII – Rules for Construction of Pressure Vessels – Division 1,* 1995 ed., United Engineering Center, New York, 1995.
11. P. Tantichattanont, S.M.R. Adluri, R. Seshadri, 2006, "Fitness-for-service assessment of spherical pressure vessels with hot spots", *International Journal of Pressure Vessels and Piping 84 (2007)*: 762-772.
12. P. Tantichattanont, S.M.R. Adluri, R. Seshadri, 2006, "Structural integrity evaluation for corrosion in spherical pressure vessels", *International Journal of Pressure Vessels and Piping 84 (2007)*: 749-761.
13. Shunqing Cai, Andrew J. Deeks, 2005, "Axi-symmetric dynamic finite element analysis of cylindrical shells with initial distortion", *Computers and Structures 83 (2005)*: 1834 – 1848.

14. Y.J. Kim, 2003, "Development of limit load solutions for corroded gas pipelines", *International Journal of Pressure Vessels and Piping (2003)*: 121 – 128.
15. David Heckman, 1998, "Finite Element Analysis of Pressure Vessels", Monterey Bay Aquarium Research Institute (MBARI) <http://www.mbari.org/education/internship/98interns/98internpapers/98heckman.html>
16. Motram, J. Tobby, *Using Finite Element in Mechanical Design*, Mc. Graw Hill, England, 1996.
17. Nazri, Owner of Hydrant Water Services Sdn. Bhd., Perak. Personal interview. Apr. 14. 2008
18. Department of Occupational Safety and Health Malaysia, <http://www.dosh.gov.my/wps/portal/>
19. <www.webd.etsii.upm.es>, *Solid Mechanics – Yield Criterion.*
20. William J, Koves, *Evaluation of Pressure Vessel Design Criteria for Nozzles (II)*, Pressure Vessel and Piping Codes (PVP-Vol. 383), ASME 1999.
21. <http://www.matweb.com/>

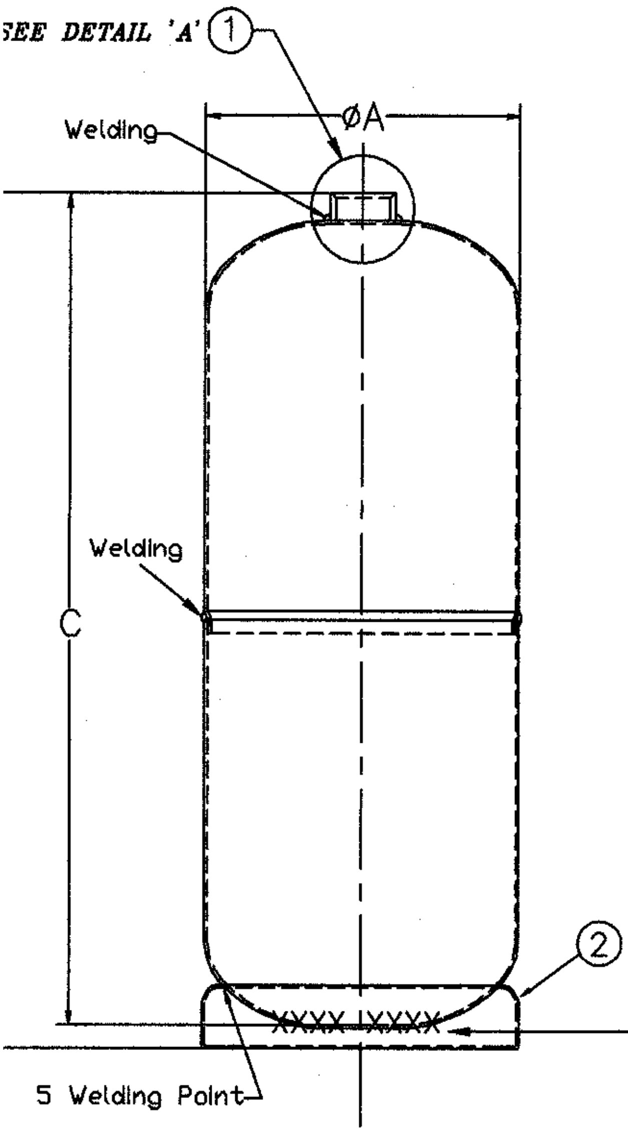
APPENDICES

APPENDIX A

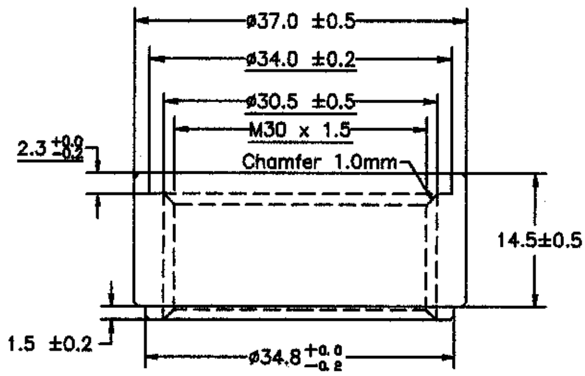
Technical Drawing of Fire Extinguisher from SRI Sdn. Bhd.

TECHNICAL SPECIFICATION

PART NO	FEC004-MS-090-NA
DESCRIPTION	9KG
MATERIAL THICKNESS	1.5mm
FINISHING	Epoxy Red Powder Coating
MATERIAL SPECIFICATION	Cold Rolled Steel JIS G3141 SPCD,SPCE
MIN.TENSILE STRENGTH	280N/mm² (280mpa)
GAS WELDING TYPE	Coogar(40% Of Co2 & 60% Argon Mixtures)
WIRE WELDING TYPE	Copper Coated Co2 Welding Wire(1.0mm)
MIN.TEST PRESSURE	25bar(362.5psi) min 30s
MIN.BURST PRESSURE	65bar(942.5psi)
VOLUME	9.5L
ØA	Ø176.0 ±1.0mm
B	465.0 ±5.0mm
C	451.0 ±2.0mm

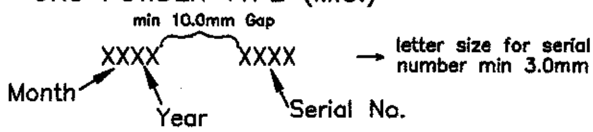


Note: Neck ring –no burr at all edges.
–smooth finishing.



DETAIL A

9KG POWDER TYPE (M.S.)



DETAIL B

ITEM	PART NO	DESCRIPTION	MATERIAL	QTY
2	FEC096-MS	BASE RING	MILD STEEL 1.0mm	1
1	FEC093-MS-M30	M30 NECK RING	MILD STEEL	1

UNLESS OTHERWISE STATED DIMENSIONS IN MILLIMETERS		DRN.	QA ENG.	STEEL RECON IND. SDN. BHD.
TOLERANCES: LINEAR: ±0.25 ANGULAR:	MATERIAL	APPD. 1	QA MGR.	PRODUCT : FIRE EXTINGUISHER
		APPD. 2	FAC. MGR.	
		ISSUED	3 FEB 2007	TITLE: 9KG CYLINDER
REV NO 08	FINISH	DRW NO.	FECB	
		PART NO.		

SRI

APPENDIX B

Level 1 FFS Assessment (Table 8.3 of API 579)

Table 8.3
Overview Of Fabrication Tolerances – ASME B&PV Code, Section VIII, Division 1 And Division 2

Fabrication Tolerance	Requirement		Code Reference
Out-Of-Roundness In Cylindrical Shells Under Internal Pressure	$(D_{max} - D_{min})$ must not exceed 1% of D where: D_{max} = Maximum measured internal diameter D_{min} = Minimum measured internal diameter D = Nominal internal diameter At nozzle openings, this tolerance is increased by 2% of the inside diameter of the opening.		UG-80(a) {AF-130.1}
Out-Of-Roundness In Cylindrical Shells Under External Pressure	The diameter tolerance for internal pressure should be satisfied. Using a chord length equal to twice the arc length determined from Figure 8.14, the maximum deviation from true circle shall not exceed the value e determined from Figure 8.15. Take measurements on the unwelded plate surface. For shells with a lap joint, increase tolerance by t. Do not include future corrosion allowance in t.		UG-80(b) {AF-130.2}
Shape Of Formed Heads	The inside surface must not deviate outside the shape by more than 1.25% of the inside diameter nor inside the shape by more than 0.625% of the inside diameter.		UG-81 {AF-135}
Cylindrical Shell-To-Head Attachment Weld	The centerline (radial) misalignment between the shell and the head shall be less than one-half the difference between the actual shell and head thicknesses.		UW-13(b)(3) {AD-420}
Centerline Offset Weld Misalignment – Longitudinal Joints (Category A)	For $t \le$ 12.7 mm (1/2 in)	e = t/4	UW=33 {AF-142}
	For 12.7 mm (1/2 in) < $t \le$ 19.1 mm (3/4 in)	e = 3.2 mm (1/8 in)	
	For 19.1 mm (3/4 in) < $t \le$ 38.1 mm (1-1/2 in)	e = 3.2 mm (1/8 in)	
	For 38.1 mm (1-1/2 in) < $t \le$ 50.8 mm (2 in)	e = 3.2 mm (1/8 in)	
	For t > 50.8 mm (2 in)	e = min(t/16, 9.5 mm) or e = min(t/16, 3/8 in)	
	Where t is the plate thickness and e is the allowable centerline offset.		
Centerline Offset Weld Misalignment - Circumferential Joints (Category B, C and D)	For $t \le$ 19.1 mm (3/4 in)	e = t/4	UW=33 {AF-142}
	For 19.1 mm (3/4 in) < $t \le$ 38.1 mm (1-1/2 in)	e = 4.8 mm (3/16 in)	
	For 38.1 mm (1-1/2 in) < $t \le$ 50.8 mm (2 in)	e = t/8	
	For t > 50.8 mm (2 in)	e = min(t/8, 19.1 mm) or e = min(t/8, 3/4 in.)	
	Where t is the plate thickness and e is the allowable centerline offset.		
Angular Weld Misalignment	None stated		---

APPENDIX C

Appendix A of API 579

t_c	=	$t - LOSS - FCA$ (mm:in),
t_k	=	Nominal or furnished thickness of the knuckle (mm:in),
t_{kc}	=	$t_k - LOSS - FCA$ (mm:in),
t_f	=	Nominal or furnished thickness of the flare at a conical transition (mm:in),
t_{fc}	=	$t_f - LOSS - FCA$ (mm:in),
t_{min}	=	Minimum required thickness (mm:in),
t_{sl}	=	Thickness required for supplemental load based on the longitudinal stress (see paragraph A.7), (mm:in),
t^s	=	Nominal or furnished small end cylinder thickness in a conical transition (mm:in),
t_c^s	=	$t^s - LOSS - FCA$ (mm:in).
t^c	=	Nominal or furnished cone thickness in a conical transition (mm:in),
t_c^c	=	$t^c - LOSS - FCA$ (mm:in).
σ_m	=	Nominal membrane stress (MPa:psi), and
α	=	One-half apex angle of the cone in a conical shell or toriconical head (degrees).

A.3.4 *Cylindrical Shells* – The minimum thickness, $MAWP$ and membrane stress equations are as follows (see ASME B&PV Code, Section VIII, Division 1, paragraph UG-27):

A.3.4.1 Circumferential Stress (Longitudinal Joints):

$$t_{\min}^C = \frac{PR_c}{SE - 0.6P} \tag{A.2}$$

$$MAWP^C = \frac{SEt_c}{R_c + 0.6t_c} \tag{A.3}$$

$$\sigma_m^C = \frac{P}{E}\left(\frac{R_c}{t_c} + 0.6\right) \tag{A.4}$$

A.3.4.2 Longitudinal Stress (circumferential Joints):

$$t_{\min}^L = \frac{PR_c}{2SE + 0.4P} + t_{sl} \tag{A.5}$$

$$MAWP^L = \frac{2SE(t_c - t_{sl})}{R_c - 0.4(t_c - t_{sl})} \tag{A.6}$$

$$\sigma_m^L = \frac{P}{2E}\left(\frac{R_c}{t_c - t_{sl}} - 0.4\right) \tag{A.7}$$

A.3.4.3 Final Values:

$$t_{\min} = \max(t_{\min}^C, t_{\min}^L) \tag{A.8}$$

$$MAWP = \min(MAWP^C, MAWP^L) \tag{A.9}$$

$$\sigma_{\max} = \max(\sigma_m^C, \sigma_m^L) \tag{A.10}$$

APPENDIX D

EXTRACTED STEPS FOR LEVEL 2 ASSESSMENT.

Step 1 – The following variables is to be determined based on the type of out-of-roundness.

E = Weld joint efficiency from the original construction code, if unknown use 0.7,

E_y = Young's modulus (MPa:psi)

FCA = Future corrosion allowance (mm:in)

H_f = Factor dependent on whether the induced stress from the shape deviation is categorized as a primary or secondary stress (see Appendix B); $H_f = 3.0$ if the stress is secondary and $H_f = 1.5$ if the stress is primary (for most applications the induced bending stress can be considered as secondary)

P = internal pressure (MPa:psi)

R = Mean radius of the cylinder or sphere (mm:in)

S_a = Allowable stress per the governing code (MPa:psi)

t = Current wall thickness of the component (mm:in)

v = Poisson's Ratio

θ = Angle to define the location where the stress will be computed (0° is chosen because this is the location of a longitudinal weld seam

C_s = Factor to account for the severity of the out-of-roundness, for a purely oval shape, $C_s = 0.5$; for shapes which significantly deviate from an oval shape, use $C_s = 0.1$

D = Nominal internal diameter (mm:in)

D_{max} = Maximum outside diameter (mm:in)

D_{min} = Minimum outside diameter (mm:in)

Step 2 – The membrane stress, σ_m, is to be determined based on the current design pressure (see Appendix A).

$$\sigma_m = \frac{P}{E}\left(\frac{R_c}{t - FCA} + 0.6\right)$$

Step 3 – Determine the ratio of the induced circumferential position stress to the circumferential membrane stress

$$R_b^{or} = abs\left[\frac{1.5(D_{max} - D_{min})\cos 2\theta}{(t-FCA)\left(1 + C_s \dfrac{P(1-v^2)}{E_y}\left(\dfrac{D_m}{t-FCA}\right)^3\right)}\right]$$

Step 4 – Determine the remaining strength factor, RSF, where

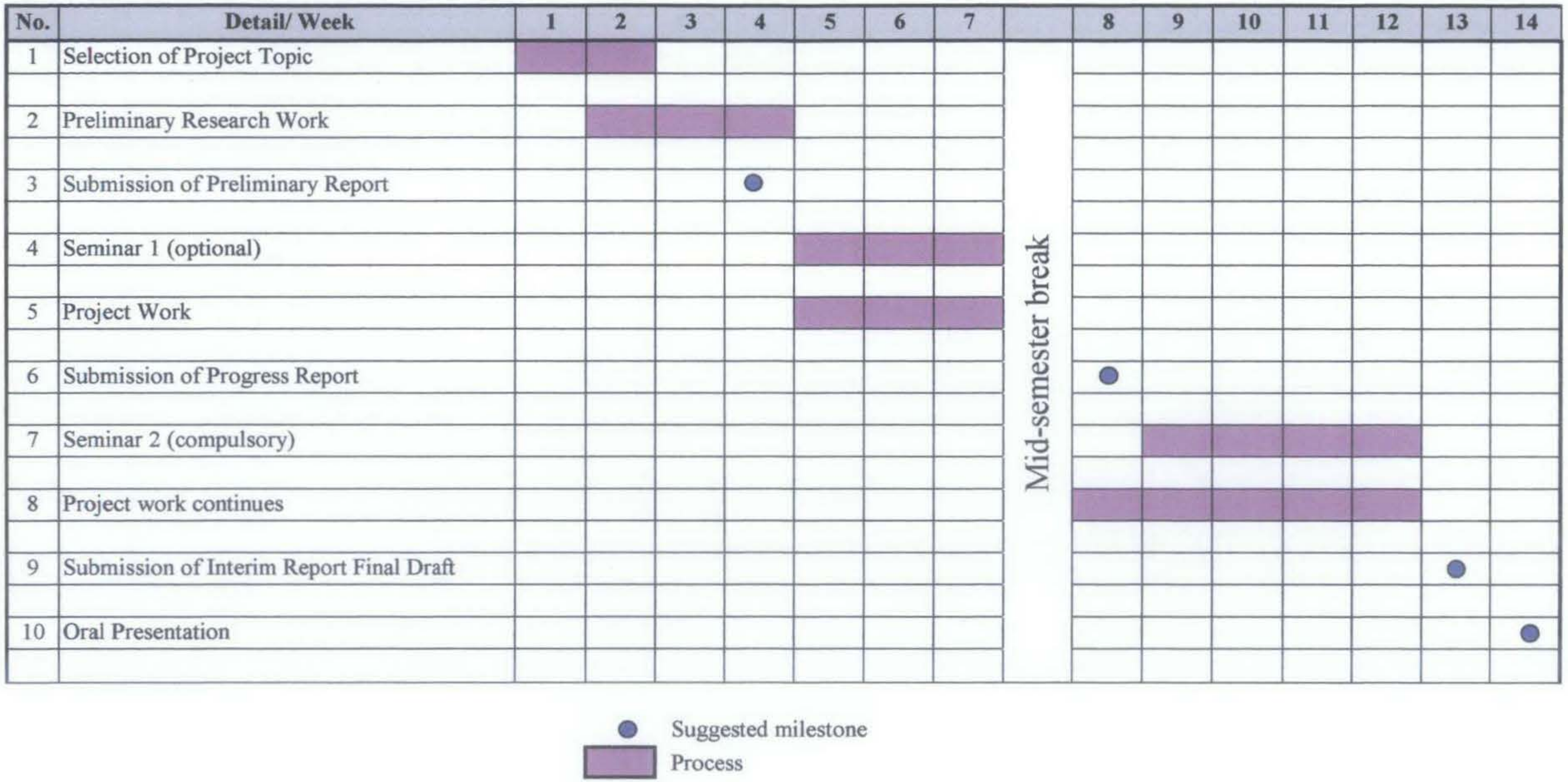
$$RSF = \min\left[\frac{H_f S_a}{\sigma_m(1+R_b)}, 1.0\right]$$

Step 5 – The results is evaluated by comparing RSF with RSF_a. The value of RSF of the object must be higher or equal to allowable value, RSF_a.

$$RSF \geq RSF_a$$

If it is found that the condition obtained as above, then the out-of-roundness is acceptable per Level 2. Otherwise, a procedure for rerating the fire extinguisher should be conducted.

Gantt Chart for the First Semester of 2-Semester Final Year Project



APPENDIX E

APPENDIX F

Gantt Chart for the Second Semester of 2-Semester Final Year Project

No.	Detail/ Week	1	2	3	4	5	6	7	8	9	10	Mid-semester break	11	12	13	14
1	Familiarizing the software	■	■	■												
2	Submission of Progress Report I				●											
3	Modelling with ANSYS				■	■	■									
4	Simulation in ANSYS							■	■	■						
5	Submission of Progress Report II								●							
6	Seminar									■	■					
7	Data Analysis and Verification										■		■			
8	Poster Exhibition												■	■		
9	Submission of Dissertation (soft bound)														●	
10	Oral Presentation															●
11	Submission of Project Dissertation (Hardbound)															●

● Suggested milestone

■ Process

Symbol	Metric unit	Imperial unit	Metric value	Imperial value
E	-	-	1	1
Ey	MPa	psi	200000	29010733.97
FCA	mm	in	0.1	0.003937008
Hf	-	-	3	3
			1.5	1.5
P	MPa	psi	1.4	203.0751378
R	mm	in	86.5	3.405511811
Sa	MPa	psi	102	14795.47433
t	mm	in	1.5	0.059055118
v	-	-	0.3	0.3
Cs	-	-	0.5	0.5
			0.1	0.1
Di	mm	in	173	6.811023622
Dmax	mm	in	180	7.086614173
Dmin	mm	in	173	6.811023622
Do	mm	in	176	6.929133858
Dmean	mm	in	174.50	6.87007874

Level 1	Metric	Imperial	
%	**4.05**	**4.05**	Acceptable value is less than 1%
Level 2	Metric	Imperial	
Membrane Stress	**87.44**	**12683.49**	
	MPa	psi	
	Straight Answer Conversion		
	87.44	Mpa	from psi to Mpa
Ratio	Metric	Imperial	
Numerator	10.50	0.41	
Denominator	3.13	0.12	
Answer	**3.36**	**3.36**	
t-FCA	1.4	0.05511811	
Cs(P(1-v2)/Ey)	0.000000637	0.000000637	
(122/(t-FCA))^3	1936431.71	1936431.71	
D34*(1+D35*D36)	3.13	0.12	
RSF	Metric	Imperial	
	0.80	**0.80**	Acceptable value is more than 0.9
Answer	0.803025341	0.803025341	

APPENDIX G

MatWeb Data Sheet

AISI 1008 Steel, CQ, DQ, and DQSK sheet, 1.6-5.8 mm thick

KeyWords:
UNS G10080, JIS G3141(96) SPCC, ASTM A29, ASTM A108, ASTM A510, ASTM A519, ASTM A545, ASTM A577, ASTM A576, ASTM A787, ASTM A830, FED QQ-S-637 (C1008), FED QQ-S-698 (C1008), MIL SPEC MIL-S-11310 (CS1008), SAE J405, SAE J412, SAE J414, DIN 1.0204, UNI CB10 FU
SubCat: Carbon Steel, AISI 1000 Series Steel, Low Carbon Steel, Metal
Material Notes:
Usually produced as rimmed, capped, semikilled, and fully killed, Rimmed have exceptional cold fomability. Weldability (spot, projection, butt, and fusion) and brazeability are excellent. Applications include extruded, cold headed, cold upset, and cold pressed parts and forms.

Component	**Value**	**Min**	**Max**
Carbon, C			0.1
Iron, Fe		99.31	99.7
Manganese, Mn		0.3	0.5
Phosphorous, P			0.04
Sulfur, S			0.05

Properties	**Value**	**Min**	**Max**
Physical			
Density, g/cc	7.872	--	--
Mechanical			
Tensile Strength, Ultimate, MPa	--	303	358
Tensile Strength, Yield, MPa	--	180	240
Elongation at Break, %	--	42	48
Modulus of Elasticity, GPa	200	--	--
Bulk Modulus, GPa	140	--	--
Poissons Ratio	0.29	--	--

APPENDIX I

HAND CALCULATION FOR MEMBRANE STRESSES

1. Hoop Stress

$$\sigma_h = \frac{PD_i}{2t}$$

$$\sigma_h = \frac{(1.4MPa)(173mm)}{2(1.5mm)}$$

$$\sigma_h = 80.73Mpa$$

2. Longitudinal Stress

$$\sigma_l = \left(\frac{PD_i}{4t}\right)$$

$$\sigma_l = \left(\frac{\sigma_h}{2}\right)$$

$$\sigma_h = 40.367Mpa$$

3. Von Mises Stress

$$\sigma_{vm} = \frac{1}{\sqrt{2}}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]^{0.5}$$

$$\sigma_{vm} = \frac{1}{\sqrt{2}}\left[(\sigma_h - \sigma_l)^2 + (\sigma_l - \sigma_{r,ID})^2 + (\sigma_{r,ID} - \sigma_h)^2\right]^{0.5}$$

$$\sigma_{vm} = \frac{1}{\sqrt{2}}\left[(80.73MPa - 24.22MPa)^2 + (24.22 - (-1.4MPa))^2 + ((-1.4MPa) - (80.73MPa))^2\right]^{0.5}$$

$$\sigma_{vm} = 72.78MPa$$

APPENDIX J

Input Listings for ANSYS Simulation

```
FINISH
/CLEAR, NOSTART
!
! Name the file
jobname = 'dent-7mm-model'        !should be changed to 6 & 7
/FILENAME, jobname
!
/PREP7
/TITLE, fire extinguisher without pit
!
ET, 1, 95               ! Define 20-node, 3-D structural solid element
MP, EX, 1, 200000       ! Young's Modulus
MP, PRXY, 1, 0.3              ! Poisson's ratio
!
! Define parameters for model generation
Pv = 1.4                ! MPa
RI1=86.5                ! Inner radius
RO1 =88.0               ! Outer radius
th_k = RO1-RI1
Z1=75                   !173.5(original)       ! Tank length
!
/PNUM, VOLU, 1
/VIEW,, -3, 1,-1
!
CYLIND, RI1, RO1,, Z1,, 90 ! Tank Wall
!
/TITLE, Quarter Cylindrical Shell Modeled
!
SPH4, 0, 88, 7          !5 !6 !7          ! Create a solid sphere
VSBV, 1, 2, , ,         ! Subtract sphere from main volume
!
WPOFFS, 0, 88, 0                  ! To offset the working plane
WPROTA, 0, 90, 0                  ! To rotate working plane
WPROTA, 0, 0, 90
WPROTA, 90, 0, 0
WPROTA, 0, 180, 0
!
CYL4, 0, 0, 7, 270, 8.5, 360      ! 5 pairs 6.5   ! 6 pairs 7.5   ! 7 pairs 8.5
LFILLT, 4, 3, 0.5                 ! Fillet lines with 0.5 mm radius
AL, 9, 4, 11, 3, 2                ! Create an area from selected lines
VROTAT, 3, , , , , , 3, 10, 90,   ! Rotate area to create volume of dent
VOVLAP, 1, 3                      ! Overlap main volume with dent
VDELE, 2, , ,1                    ! To delete redundant volume
```

```
VDELE, 5, , ,1                     ! To delete redundant volume
VGLUE, 4, 6, 7
VPLOT
/VIEW,, -3, 1, -1
/TITLE, Quarter Cylindrical Shell with Dent
!
! Meshing Generation
! Part 1: Major Volume 1
ACCAT, 2, 30
ACCAT, 30, 31
LESIZE, 1, , , 4
LESIZE, 8, , , 4
LESIZE, 6, , , 4
LESIZE, 39, , , 15
LESIZE, 32, , , 15
LESIZE, 49, , , 4
LESIZE, 51, , , 4
ALLSEL
ESIZE, 5                   ! Set default element size
MSHAPE, 0, 3D              ! 0 = mapped brick mesh, 1 = tetra
MSHKEY, 1                  ! 0 = free, 1 = mapped, 2 = both
VMESH, 6
!
! Part 2: Dent Volume
LESIZE, 9, , , 4
LESIZE, 41, , , 4
LESIZE, 40, , , 4
LESIZE, 36, , , 15
LESIZE, 37, , , 15
LESIZE, 35, , , 15
LESIZE, 38, , , 15
LESIZE, 24, , , 15
LESIZE, 39, , , 15
VSWEEP, 1, 22, 21   !5 & 6mm, NVOL = 4, 7mm, NVOL = 1
!
! Part 3: Mesh Overlapped Volume (between Main and Dent)
LESIZE, 11, , , 4
LESIZE, 46, , , 4
VSWEEP, 2, 35, 36   !5 & 6mm, NVOL = 7, 7mm, NVOL = 2
!
/TITLE, Meshed elements being modeled
!
/com, **** Obtain solution ******
!
/SOLU
ANTYPE, 0
```