International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

Finite Element Analysis & Modeling of Cylindrical Pressure Vessels

Blgasem Omar Issa Alsharef

h1975h1975[at]gmail.com

Abstract: ***The project is based on the development of finite element models for cylindrical pressure vessels; with the aim of using them for the design and analysis of such components in the future, pressure vessels constitute the core of in - numerous mechanical system in almost every industry ranging from chemical, automobile and fluid transmitting plants to military equipment. Considering their importance in the industry, they have been selected for this study.***

Keywords: Finite Element, Pressure Vessel

1. Introduction

1.1 Purpose

The work described in this report revolves around the finite element analysis of some of the most common mechanical components viz., cylindrical pressure vessels. These components find a wide range of application in industries as well as in domestic areas. Moreover, these components constitute the basic structure of numerous mechanical systems; thereby have been selected for this study.

A lot of work has been done and is currently being done pertaining to these subjects. The most relevant ones referenced in this report are Majzoobi and Ghoomi (2006), Majzoobiet at (2004), Kaplan (1954), Kaptan (2007), sanal (2000) etc. Most of these works are based on optimizing the design or investigating the failure of components.

1.2 Scope

To avoid ambiguity, it should be noted that the scope of this project is limited to the development of valid Finite Element models of cylinder.

1.3 Objectives

The main objectives of the project are as follows:
- To develop a finite element model for a Thick Cylinder and rig available at through comparison with theoretical analysis and experiments on the TQSM1011 THICK cylinder rig available at the university.
- To investigate the functionality of the compound cylinder finite element model, by attempting to redesign and convert the single walled thick cylinder in the test rig to a compound cylinder.

2. Materials & Methods

ANSYS Workbench software will be used throughout the project for the finite element analysis of cylinders and the beam. The models developed therein will then be validated through comparison with the results of theoretical calculations and experiments in the lab.

3. Literature Review

3.1 Finite Element Analysis

Finite element method is an advanced and sophisticated technique used to analyses a design by splitting the whole structure into smaller pieces or shapes called elements

These elements are connected together at points called nodes to maintain continuity of the structure. The whole idea of the finite element method is to form a mathematical model by splitting the body into smaller subdivisions, so as to have better control and accuracy over the end results, Also, the accuracy of the solution depends upon the number of divisions/elements (Wardle, 199?)

3.2 ANSYS Workbench

Ansys Workbench is powerful and sophisticated software which is capable of performing advanced finite element calculation in a fraction of a second. The Ansys Workbench Tutorial 10.0 manual (2006), describes this product;

Ansys Workbench in and of itself is not a product, rather it is a product development platform and user GTI built for analysis needs with the objective of providing elegant next generation functionality and intelligent automation to the engineering community

3.2.1 Pre - processing in Ansys workbench

To start off with, workbench uses a CAD representation to the solid model which can be imported from any 3D modeling software or it can be created in the Design Modeler component of Ansys workbench. The program than breaks down the CAD model into finite elements through the process called meshing. A good representation of the physical model needs a high quality of meshing,

4. Analysis of Thick Cylindrical Pressure Vessels 3

Considerable attention is focused for the experimental and analytical methods for determining the operating stresses in pressure vessels, especially those working under high pressures. Economic considerations

4.1 Analysis of Compound Cylinder

It is observed from the analysis of thick pressure vessel (chapter3), that the maximum principal stresses in an internally pressurized thick walled cylinder decreases appreciably with the increase in radius. Thereby, the material towards the outer end is relatively lightly stressed and proves to be an uneconomic design (Wardle 199?). As discussed in the literature review, there are basically two methods to improve the efficiency of a cylindrical pressure vessel; auto - frettage and compound cylinder.

The analysis of compound cylinder in chapter 4 demonstrated that the maximum stress in a compound cylinder is considerably less than that in a single cylinder of equivalent thickness and under the same internal pressure, In other words, for the same maximum stress, the total thickness of a compound cylinder would be appreciably less than the thickness of a single cylinder. This concept can be used to convert a single cylinder to a compound cylinder for any known maximum /design pressur**e**

4.2 Design Stress

As per the technical manual of the test rig (TQ Education and training ltd., 2004) the maximum Working pressure of the cylinder was found to be 7MPa. It could be safely assumed that the manufacturer has incorporated substantial factor of safety in the design before proposing this value of maximum working pressure therefore, the maximum stress developed could be used as the design stress with confidence. To determine the design stress for the compound cylinder, the value of 7MPa was then used as the pressurization pressure in the single cylinder FEA model developed in chapter 3. The graph below shows the radial and hoop stress distribution in the single cylinder with internal pressure P=7MPa

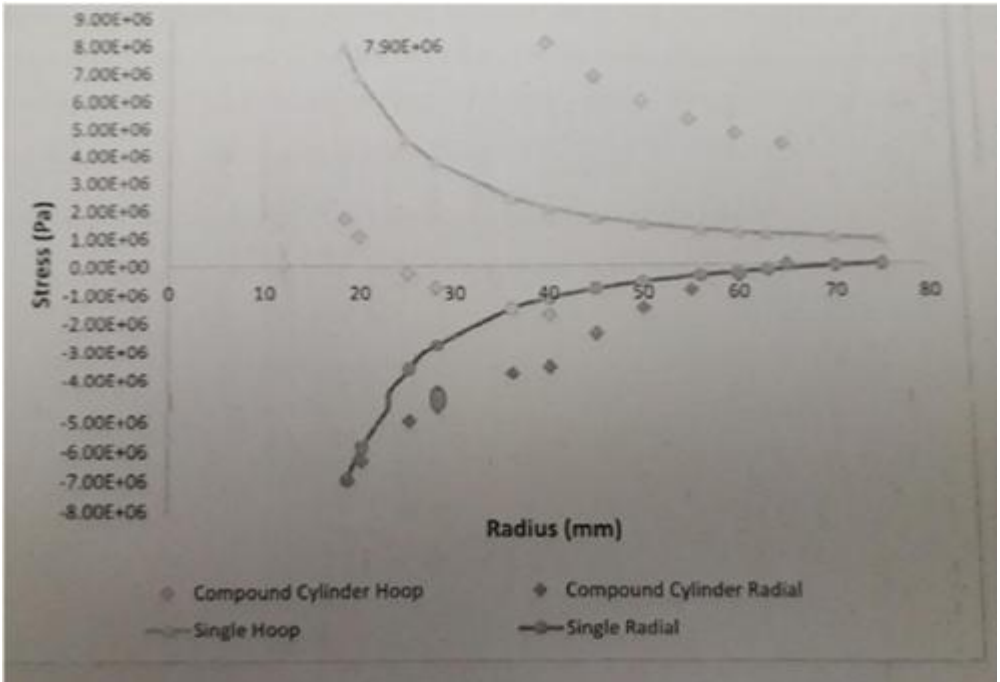

Fig. 5.6.: Stress distribution of redesigned cylinder and single cylinder

5.2 Modelling the Re - Designed Compound Cylinder

The compound cylinder (similar materials) model developed in chapter 4 was used as prototype for this process. The settings for the geometry and other boundary conditions were altered as required to reach upon the solution. Honouring the work of Majzoobi et at (2004), the interface radius was taken as 41 mm. The internal diameter was kept unchanged at 18.5mm.

The interface pressure of 1MPa would required the diameteral interference to be as 0.004 mm, which is practically very difficult to machine, hence not feasible. T he lowest possible diameteral interference could be 0.01 mm. (as discussed with professor Wardle) which generates an interface pressure of 2.5 MPa this the interface pressure was selected as 2.5MPa which was achieved by entering the offset value of 0.005mm in the contact settings.

The properties of the material Duralumin was taken from the Matweb database and used for both the cylinder in the simulation. Keeping all the other settings intact, the outer radius of the cylinder was varied. After a number of trials the radius R at which the design stress of 7.9 MPa was attained, was found to be 65mm. The hoop and radial strain distribution for this model was as shown in the figure below:

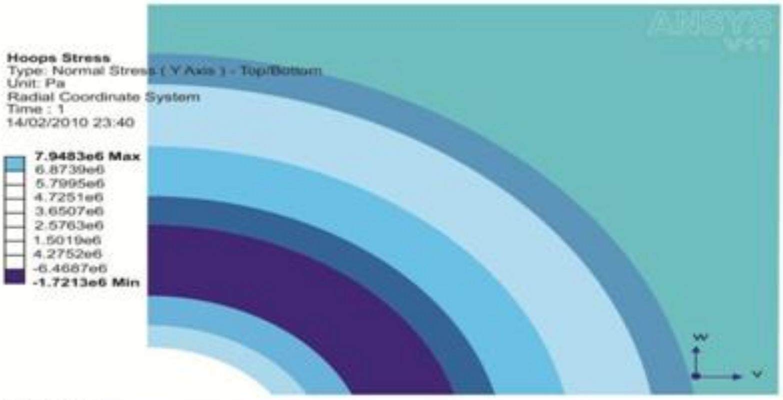

Fig. 5.4.:Hoop stress distriution on redesigned cylinder

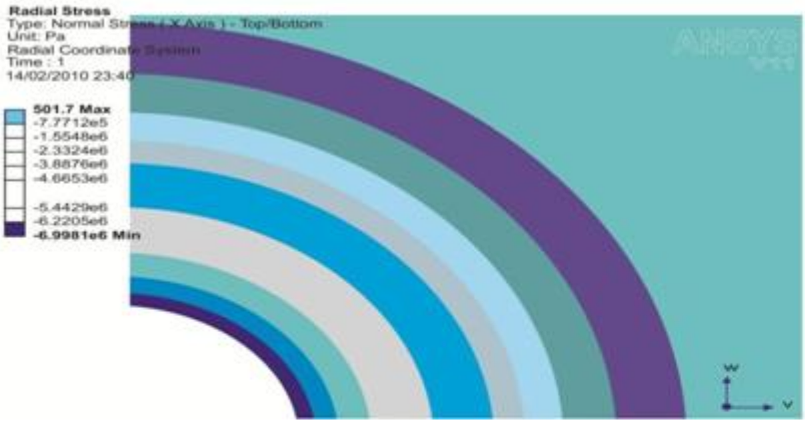

Fig. 5.5.:Radial stress distriution on redesigned cylinder

5. Conclusion

Finite element models were successfully developed for thick cylinders, compound cylinders as well as curved beams using Ansys Workbench, these models can be used for designing or analyzing similar components and predict their behavior in the real world .

The redesigning of the cylinder in TQSM rig through FEA proved that, compound cylinders are considerably efficient as compared to single walled cylinders and can lead to a material savings of over 35% The FEA also showed that they might be applicable /effective only for higher working pressures. To verify this point, further investigation is required in this area. However, it was beyond the scope of this project as there are no facilities available to verify the results through experimentations.

References

[1] Ansys Inc, (2006), Ansys Workbench 10.0 Totorial, version 1.0, Philadelphia: ANSYS, Inc. Proprietary
[2] Ansys Inc. (2001), Fundamental FEA concepts and applications, Ansys Inc., Available: <http: //www.wildefea. co. uk/>
[3] Ansys Theory Manual version 11.0 [logical], 2007, Ansys Inc.
[4] Automation creations Inc. (1996), Matweb materials database, Automation Creations Inc., Blacksburg, <http: //www.matweb. com>
[5] Bengt A. steen (2005), Abiotic Resource Depletion Different perceptions of the problem with mineral deposits, The International
[6] Journal of Life cycle Assessment Volume 11, pp49 - 54.
[7] Bruce Williams, craig Gygi and Nell DeCarlo (2005), Six Sigma for Dummies, NJ: Wiley Publishing Inc.