Finite Element Simulation of Cylinder using Composite Material

Shivam Singh¹ Dr. K. K. Jain²
¹Student ²Professor
¹,²Department of Mechanical Engineering
¹,²NITTTR Bhopal India

Abstract— Hydraulic systems are generally encountered in various engineering applications. Hydraulic cylinders are very common components of the hydraulic systems used in many engineering applications like; power transmitting device, hydraulic breaking systems etc. The vibrations produced under dynamic conditions affect many important design parameters such as strength, production costs, productivity. Under this title we study, the modal and structural analysis of a double acting hydraulic cylinder subjected to dynamic loads. Computer aided engineering (CAE) procedures are used to analyse the dynamic response of the cylinder walls. The finite element methods used in the analysis are applied by a computer aided design and analysis software ANSYS. The studies on the moving load are extended to the hydraulic cylinder. The finite element model of the cylinder is created. The numerical solution presents first five mode shapes and corresponding frequencies. The main objective of this research work is to study the effect of mechanical properties of materials on mode shape of double acting cylinder. Material mechanical properties play an important role for evaluation of frequency, deformation, and stress and strain. Free vibration study was performed for the vibration response study of double acting cylinder. The free vibration study of cylinder was performed using finite element simulation. The vibration response for all materials shows the variation of natural frequency in Hz and various vibrations mode five modes were studied. The used material is steel, Glass fiber reinforced plastic (GFRP), S glass epoxy (fiber glass composite) and E glass epoxy.

Key words: Static and Modal Analysis, Hydraulic Cylinder, Finite Element Method, ANSYS

I. INTRODUCTION

As we know there are a lot of uses of the finite element method, still the simulation of modal and structural analysis provides great challenges to overcome because of the complex physical phenomena. Hydraulic systems are widely used systems in industry. While they are used widely in the industry, the system components like pumps, valves, cylinders are always became investigated topics in the history. Hydraulic cylinders are one of the most common components of the hydraulic systems used in many engineering applications.

A. Finite Element Method

The finite element method is a numerical technique which is commercially used for the finding of an approximate solution of partial differential equation as well as integral equation. In some solving partial differential equations the first problem is to create an equation that approximate the equation which is to be studied. It means that during calculations the error should not accumulate, thereby causing the output as to be meaningless. The concept of popular method finite element analysis is to simplify and solve the solution of complicated problem in relatively easy way. The finite element analysis is a powerful and useful tool for the numerical solution of a wide range of engineering problem. Applications ranges from deformations to stress analysis of automotive, aircraft, building, defence, missile, and bridge structures to the field of stability, fracture mechanics, dynamic analysis, heat flux, fluid flow, seepage, and other flow problems. With the advances in computer technology and CAD systems, complex problems can be modeled with some other relative case. Several alternate configurations can be tried out on a computer before the first prototype is built.

ANSYS software is used for creating the model and performing the analysis of this study. APDL stands for ANSYS Parametric Design Language, a scripting language that one can use to automate the common tasks or even building the model in term of parameters. In this paper a double-acting hydraulic cylinder is used in the analysis.

B. Boundary Condition

In the analysis, two different boundary conditions are considered for the hydraulic cylinder, both end fixed and single end fixed keeping other end free. It is assumed that, the cylinder is directly installed to the other components or to the body in two different mounting types and the dynamic analyses is made for these two mounting types.

II. LITERATURE REVIEW

Various related literature such as transactions, proceeding of various national and international conferences and other journals which available on Google scholar were reviewed.

[Kumar Akkimaradi, 2016] In this paper the structure analysis of steel and FRP composite pressure vessels is to carry out using ANSYS software and analytical
results are calculated for steel pressure vessel and compared with ANSYS results. FE model of a steel and filament wound FRP pressure vessel is established using ANSYS 11 software. Finally weight and structural efficiency of composite pressure vessel is compared with steel pressure vessel. The structural efficiency of the composite pressure vessel is 76 % more than steel pressure vessel.

[Prithish Tapare, 2015] The study of Vibration analysis of the hydraulic cylinder components for the different size, speed and the loading conditions can be performed by using the finite element analysis if the dynamic loading of the structure is properly defined. The finite element methods used in the analysis are applied by computer aided design and analysis software ANSYS. ANSYS is used to find the codes for defining both the dynamic loading of the hydraulic cylinder and structural parameters.

[Kadhim Hussein Mukhirmesh, 2015] The function of this project work is to analyze the material strength of the pressure vessels with help of Ansys software. The Model of composite Pressure vessels is done by Pro/Engineer then after structure analysis are to be done by ANSYS on the welded joint of pressure vessel for different weld efficiencies.

[Mohd. Azharuddin, 2015] In this paper, the composite materials considered are HM Carbon Epoxy and HS Carbon Epoxy. The composite materials are considered due to their high strength to weight ratio. The material for damping is rubber. The structural analysis is done to verify the strength of the shaft and compare the results for three materials. By observing the analysis results, the stress values are less when composite material Carbon Epoxy is used when compared with that of Steel. By observing the modal analysis results for shell element, when comparing the results without damping material and with damping material rubber, the frequencies are less when damping material is used, so vibrations will be less.

[Mayank Nirbhay, 2014] A comprehensive modelling approach in order to predict the behavior of CNG cylinders under various loading conditions has been proposed using Finite Element (FE) Software-ANSYS. In the present investigation two different materials viz. glass/epoxy and carbon/epoxy are used separately and in combination with different patterns of helical and hoop windings, for outer reinforced layers of all composite gas cylinder.

[Mr. Yashraj Jaywant Salunke, 2014] The main aim of this research work is reduced the LPG cylinder weight by replacing composite material at low density GFRP material for which ANSYS to be used. The internal pressure composite cylinder is to be analyzed by finite element analysis through ANSYS. The weight of LPG cylinder can be saved by using FRP composites and the stress values are also well within the specific limit of capability of materials.

[N. Upendra, 2014] The vibrations produced under static and dynamic conditions affect many important design parameters such as strength, production costs, productivity. In this paper, the dynamic and static analysis of a composite hydraulic cylinder subjected to pressure by varying fibre orientation and different boundary conditions is determined. The finite element methods used in the analysis are applied by using analysis software ANSYS.

[R. Indu Shekar, 2014] To design high pressure vessel composite cylinders there should be highest possible safety, reliability and minimum weight considerations, the behavior of composite under various thermal loading and mechanical need to be well understood. This paper deals with the system design and system engineering considerations for the Type-3 and Type-4 high-pressure vessel composite cylinders required for aerospace applications. The result of this paper is the carbon fiber based composite cylinders type-4 and type-3 with pressure of more than 700 bar but High-pressure composite cylinders have been designed for maximum operating pressures of 200 bar.

[Anand Kumar Agrawal, 2014] This paper calculates the fatigue life till crack nucleation for a steel-lined hoop wrapped composite pressure vessel containing a surface flaw (notch) using strain-life approach. Firstly, static stress analysis taking into account the non-linear material behaviour of steel liner was performed using finite element method to provide input for the fatigue analysis. Then fatigue analysis was carried out by applying a cyclic pressure inside the pre-stressed cylinder using strain-life approach, and the results obtained by the numerical simulations are discussed. The approach associated with these calculations can be used to study the effect of various material and geometry parameters on the fatigue life of the pressure vessel.

[Subhash N. Khetre, 2014] In the present work, structure of the composite pressure vessel and different orientations of symmetric shells designed. For pressure were investigated and 3-D finite element analyses using APDL Programming. FEA software is used for failure analysis on the composite shell of continuous angle ply laminas. The Tsai-Wu failure criterion is applied for the checking the first-ply failure of layers in a simple form. Some analytical and experimental solutions are compared with the finite element solutions, in which commercial software ANSYS 15.0 was utilized and close results are obtained between them.

[Sagar R Dharmadhikari, 2013] This paper mainly discusses the work done on composite material drive shafts using ANSYS and Genetic Algorithm. Composite material instead of conventional steel material for drive shaft has increasing the many advantages of design due to its high strength and specific stiffness. There are many methods are available at present time for the design optimization of structural systems and these methods based on mathematical programming techniques. The substitution of conventional drive shaft results in reduction in weight of automobile.

[C. Sasi Rekha, 2013] The present work deals with the Stress analysis of the FRP composite cylinder with closed ends. A four layered composite cylinder with semicircular ends is considered. A metal cap is provided in the ends for openings. The finite element models created in ANSYS software are validated and extended to evaluate the stresses at the top end, middle and bottom end portions of composite cylinder.

[A. Hocine, 2013] The present study deals with the analysis of the cylindrical part of a CNG storage vessel, combining a plastic liner and an over wrapped filament wound composite. Three kind of polymer are used in the present analysis: High density Polyethylene HDPE, Light low density Polyethylene LLDPE and finally blend of LLDPE/HDPE. The effect of the mechanical properties on the behaviour of type IV vessel may be then investigated.
[Valeriu, 2012] This paper confirms the accuracy of composite materials data input into ANSYS Parametric Design Language for the numerical analysis. For this purpose, some specimens of the laminated composite were subjected to a bending moment and the deformations were measured. At the same time, the data obtained by simulating the specimens with the help of ANSYS APDL, were analyzed and compared to the experimental data in order to establish the degree of the accuracy.

[D Gopichand, 2012] The present work deals with the analysis of FRP composite cylinders. Two types of laminates, cross-ply and angle-ply laminates are considered for the present analysis. Radial deflection and stresses are obtained by varying the diameter to thickness ratio and fiber orientation in both the laminates. The problem is modeled based on layered element of ANSYS software which is designed based on 3D-elasticity theory which can be successfully applicable for the analysis of thick FRP composite cylinders.

[S. Bhavya, 2012] The present work aims to determine the effect of diameter-to-thickness ratio with respect to failure pressure of a four layers, introduction of hoop layers at ends on four layered cylinder and introduction of hoop layers at middle of six layered angle-ply laminated cylinder which is analyzed by using Finite Element software ANSYS. The variation of failure pressure with respect to fiber angles was also presented in this work.

### III. METHODOLOGY

In this section framework is to optimize the strength to weight ratio and to analyze different composite cylinder subjected to dynamic conditions. Various parameters which are used for the process are explained in detail.

List of activities that has been performed in order to achieve our goal is enlisted below:

1. Problem identification
2. Literature survey
3. Study of parameters
4. Study of hydraulic system
5. Selection of material
6. Selection of simulation process
7. Application of FEA on modal analysis
8. Finally conclusion will be discussed.

### IV. MATERIAL USED

#### A. Selection of Material

Materials of cylinder subjected to different dynamic conditions have significant contribution to the quality and the performance of the various hydraulic systems. Even a small amount in weight reduction may have a wider economic impact. Composite materials are proved as suitable substitutes for steel in connection with weight reduction of the cylinder. Hence, the composite materials have been selected for mode shape analysis of cylinder.

- GFRP
- S-glass/epoxy
- E-glass/epoxy
- Steel

#### B. Material Properties

<table>
<thead>
<tr>
<th>S. No.</th>
<th>Name of Material</th>
<th>Elastic Modulus</th>
<th>Density</th>
<th>Poisson’s Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Steel</td>
<td>203 GPa</td>
<td>7.86 g/cm³</td>
<td>0.3</td>
</tr>
<tr>
<td>2</td>
<td>GFRP</td>
<td>185 GPa</td>
<td>1.9 g/cm³</td>
<td>0.25</td>
</tr>
<tr>
<td>3</td>
<td>S Glass</td>
<td>89 GPa</td>
<td>2.46 g/cm³</td>
<td>0.2</td>
</tr>
<tr>
<td>4</td>
<td>E Glass</td>
<td>72 GPa</td>
<td>2.58 g/cm³</td>
<td>0.21</td>
</tr>
</tbody>
</table>

Table 1: Material properties

### C. Result

#### Material | Boundary Condition | Mod e 1 | Mod e 2 | Mod e 3 | Mod e 4 | Mod e 5
---|-------------------|---------|---------|---------|---------|---------|
Steel | Bc1 | 82.42 | 143.4 | 143.4 | 666.9 | 823.6 |
Steel | Bc2 | 82.42 | 981.3 | 981.3 | 1326 | 1326 |
GFRP | Bc1 | 18.85 | 107.2 | 107.2 | 502.9 | 617.0 |
GFRP | Bc2 | 18.85 | 735.9 | 735.9 | 999.5 | 999.5 |
S Glass | Bc1 | 25.97 | 165.5 | 165.5 | 795.8 | 957.2 |
S Glass | Bc2 | 25.97 | 1144. | 1144. | 1578 | 1578 |
E Glass | Bc1 | 26.60 | 150.4 | 150.4 | 729.6 | 871.6 |
E Glass | Bc2 | 26.60 | 1043. | 1043. | 1446 | 1446 |

Table 2: Natural frequencies at five different modes of vibration for boundary condition

---

Fig. 3: Natural frequencies at five different modes of vibration for boundary condition 1

Fig. 4: Natural frequencies at five different modes of vibration for boundary condition 2
V. CONCLUSION
Finally a useful conclusion is obtained regarding the behavior of composite double acting hydraulic cylinder and conventional steel cylinder. E-glass/epoxy has higher natural frequency at boundary condition one, but at boundary condition two s-glass/epoxy has higher natural frequency. Composite cylinder can perform efficiently when subject to dynamic conditions.

On the basis of the work reported in this paper, the following areas can be identified for further study:

- Harmonic analysis and random vibration analysis can also be performed by using this composite material.
- In future GFRP, s-glass and e-glass together can be fabricated by hand-lay-up method for better result.
- Modal and stress analysis can also be done by using different composite for another system subjected to different static and dynamic conditions with varying diameter and thickness.

REFERENCES